Matt Lombard

SolidWorks® 2010

Master all features, functions, and tools

Explore best practices and cautionary data

Learn specialized and advanced techniques

Bible

The book you need to succeed!

SolidWorks® 2010 Bible

SolidWorks® 2010 Bible

Matt Lombard

WILEY

Wiley Publishing, Inc.

About the Author

Matt Lombard is an independent mechanical design consultant. He spent time as the Engineering Services Manager at a high tech manufacturer where his job included the work of a Documentation Manager as well as a mixed environment CAD Manager with both direct and indirect staff. Matt also has worked as a consultant and has performed dozens of implementations covering the setup and optimization of SolidWorks, Toolbox, and PDMWorks.

Because of the popularity of his SolidWorks blog Dezignstuff (`http://Dezignstuff.com/blog`), and sessions at SolidWorks World, he has built a reputation as the go-to resource for everything SolidWorks.

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The ideas that go into a book of this size do not all originate with a single individual. Research for many of the topics was done at user group meetings, SolidWorks Corporation's discussion forums, and SolidWorks Customer Portal, as well as the blogs and Web sites of many individuals. I would like to thank all of the individuals who have posted to the public forums or blogs. Many of these people have knowingly or unknowingly contributed to my SolidWorks education, as well as that of many other SolidWorks users around the world.

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Introduction

Welcome to SolidWorks 2010 Bible. This book has been written as a desk reference for beginning and intermediate SolidWorks users. SolidWorks is such an immense software program that trying to cover all its functions is an extremely ambitious undertaking, which has resulted me leaving a few out. Because of the scope of the topic, I have limited the book to covering the basic SolidWorks package with Office, Office professional, or Office Premium add-ins, although I have devoted half of a chapter to toolbox.

About This Book

You will find enough information here that the book can grow with your SolidWorks needs. I have written tutorials for most of the chapters with newer users in mind, because for them, it is most helpful to see how things are done in SolidWorks step by step. The longer narrative examples give more in-depth information about features and functions, as well as the results of various settings and options.

This book includes many details that come from practical usage, and is focused on the needs of professional users, not on student learners. My approach is to teach concepts rather than button pushes.

How This Book Is Organized

This book is divided into eight parts.

Part I: SolidWorks Basics

This part explores basic concepts and terminology used in SolidWorks. You need to read this section if you are new to the software and especially if you are new to 3D modeling or parametric history-based design.

Part II: Building Intelligence into Your Parts

This part takes a deeper look at creating parametric relations to automate changes.

Part III: Working with Assemblies

Part III delves into assembly modeling in it various aspects, from efficiency to in-context modeling.

Part IV: Creating and Using Libraries

Part IV helps you understand how to build and manage libraries of various types with an eye toward upgrading versions, best practice, and reusing data when possible.

Part V: Creating Drawings

This part examines the functionality within the 2D drawing side of the software. Whether you are creating views, making tables, or customizing annotations, these chapters have something for everyone.

Part VI: Using Advanced Techniques

This part examines several types of advanced techniques, such as surface modeling and multi-body modeling. This is information you won't find in other SolidWorks books, explained here by someone who uses the functionality daily.

Part VII: Working with Specialized Functionality

Specialized functionality such as sheet metal, weldments, plastics, and animation requires detailed information. Part VII includes these topics because they are key to unlocking all the power available in SolidWorks.

Part VIII: Appendixes

The appendixes in this book contain information that was not appropriate in the main body of the text, such as the contents of the CD-ROM and other sources of help.

Icons Used in This Book

This book uses a set of icons to point out certain details in the text. While they are relatively self-explanatory, here is what each of these icons indicates:

Caution

Caution icons warn you of potential problems before you make a mistake.

Cross-Reference

Cross-Reference icons point out where you can find additional information about a topic elsewhere in the book.

New Feature

The New Feature icon highlights features and functions that are new to SolidWorks 2009.

Note

Notes highlight useful information that you should take into consideration, or an important point that requires special attention.

On the CD-ROM

This icon points you toward related material on the book's CD.

Tip

Tips provide you with additional advice that makes the software quicker or easier to use.

The *SolidWorks 2010 Bible* is unique in its use of these two icons:

Best Practice

Best practice icons point out recommended settings or techniques that are safe in most situations.

Performance

Performance icons elaborate on how certain settings, features, or techniques affect rebuild speed or file size.

These icons point out and describe techniques and settings that are either recommended or not recommended for specific reasons. Best Practice is usually considered to be very conservative usage, where the stability of the parametrics and performance (a euphemism for *rebuild speed*) are the ultimate goals. These two aspects of SolidWorks models are usually weighed against modeling speed (how long it takes you to create the model).

You should take Best Practice and Performance recommendations seriously, but you should treat them as guidelines rather than as rules. When it comes right down to it, the only hard and fast rule about SolidWorks is that there are no hard and fast rules. In fact, I believe that the only reason to have rules in the first place is so that you know when you can break them. Parametric stability and modeling speed are not always the ultimate goals, and are often overridden when work-around techniques are used simply to accomplish a geometric goal.

Because not everyone models with the same goals in mind, a single set of rules can never apply for everyone. You must take the best practice suggestions and apply them to your situation using your own judgment.

My point of view while writing this book has been that of someone who is actually using the software, not of someone trying to sell ideas, nor of someone trying to make the software look good, or even that of an academic trying to make a beautiful argument. I try to approach the software objectively as a tool, recognizing that complex tools are good at some things and not so good at others. Both kinds of information (good and not-so-good) are useful to the reader. Pointing out negatives in this context should not be construed as criticizing the SolidWorks software, but rather as preparing the reader for real-world use of the software. Any tool this complex is going to have imperfections. Hopefully some of my enthusiasm for the software also shows through and is to some extent contagious.

Terminology

An important concept referred to frequently in SolidWorks is *design intent*. As a practical matter, I use the phrase *design for change* to further distinguish design intent from other design goals.

The reader needs to be familiar with some special terminology before continuing. In many cases, I use a SolidWorks vernacular or slang when the official terminology is either not descriptive enough or, as is sometimes the case, has multiple meanings. For example, the word *shortcut* has multiple meanings in the SolidWorks interface. It is used to describe right mouse button menus as well as hotkeys. As a result, I have chosen not to use the word shortcut and instead substitute the words *RMB* and *hotkey*.

I frequently use RMB to refer to right mouse button menus, or other data that you access by clicking the right mouse button on an item. The word *tree* refers to the list of features in the FeatureManager.

Differences are frequently found between the names of features on toolbars and the names in the tooltips, menus or PropertyManager titles. In these cases, the differences are usually minor, and either name may be used.

Most functions in SolidWorks can work with either the object-action or the action-object scenarios. These are also called *pre-select* and *select*, respectively. The Fillet feature shows no difference between using pre-selection and selection, although for some fillet options such as face fillet, pre-select is not enabled. Most features allow pre-selection, and some functions, such as inserting a design table, require pre-selection. Although you cannot identify a single rule that covers all situations, *most* functions accept both.

Frequently in this book, I have suggested enhancement requests that the reader may want to make. This is because SolidWorks development is driven to a large extent by customer requests, and if a large number of users converge on a few issues, then those issues are more likely to be fixed or changed. Again, the enhancement request suggestions are not made to criticize the software, but to make it better. I hope that several of you will join me in submitting enhancement requests.

SolidWorks is an extremely powerful modeling tool, very likely with the best combination of power and accessibility on the MCAD market today. This book is meant to help you take advantage of its power in your work and even hobbyist applications. If I could impart only a single thought to all readers of this book, it would be that with a little curiosity and some imagination, you can begin to access the power of SolidWorks for geometry creation and virtual product prototyping. You should start with the assumption that there is a way to do what you are imagining, and that you should be open to using different techniques.

Whoever you are, I hope that you find insight deeper than simply "what does this button do?" in this book. I hope that you will find an intuition for thinking like the software. Jeff Ray, CEO of the SolidWorks Corporation, has said that the goal is to make the software as "intuitive as a light switch." While most people will agree that they have some work left to achieve that particular goal, I believe that approaching the interface intuitively, rather than attempting to remember it all by rote, is the best method. Good luck to you all.

Contacting the Author

You might want to contact me for some reason. Maybe you found an error in the book, or you have a suggestion about something that you think would improve it. It is always good to hear what real users think about the material, whether you like it or think it could be improved.

The best way to contact me is either through e-mail or through my blog. My e-mail address is `matt@dezignstuff.com`. You will find my blog at `http://dezignstuff.com/blog`. On the blog you can leave comments and read other things I have written about the SolidWorks software, CAD, and engineering or computer topics in general. If you want to contact me for commercial help with a modeling project, the previous e-mail address is the best place to start that type of conversation.

Thank you very much for buying and reading this book. I hope the ideas and information within its pages help you accomplish your professional goals.

Part I

SolidWorks Basics

I f you are a new SolidWorks user, the chapters of Part I are essential. I recommend reading these chapters in order because the material in each chapter builds on the material of the previous chapter.

If you are an existing SolidWorks user, I strongly recommend Chapters 2 and 5 because the interface has changed significantly since 2007. Even if you are a very experienced user, I recommend Chapter 5. The visualization techniques are things we all use on a constant basis in the software, and picking up even a small tip can add to your productivity dramatically.

Introducing SolidWorks

SolidWorks is a feature-based, history-based, associative, parametric 3D CAD program. That seems like many words, but they all have a specific meaning. In SolidWorks, you build 3D parts from a series of simple 2D sketches and features such as extrude, revolve, fillets, and holes, among others. You can then create 2D drawings from the 3D parts and assemblies.

This chapter will familiarize you with some of the tools available to make the transition to SolidWorks, and with some of the basic facts and concepts that you need to know to get the most out of the software.

If you want to start using the software without learning about how or why it works, you can skip directly to Chapter 4 for sketches or Chapter 5 to start making parts, assemblies, and drawings. Of course, I recommend you get a bit of background and some foundation first.

Installing SolidWorks for the First Time

I'm assuming you are installing SolidWorks for the first time on your computer. Detailed instructions for more involved types of installation and uninstallation are beyond the scope of this book.

Some of you will have SolidWorks installed for you by people in your company or by SolidWorks reseller experts, and some of you will do the installation on your own. Regardless, it is best to make sure that your

IN THIS CHAPTER

Installing SolidWorks

Getting started with SolidWorks

Identifying different types of SolidWorks documents

Grasping feature-based Modeling

Grasping history-based Modeling

Sketching in parametric software

Controlling changes with Design Intent

Modifying Design Intent

Working with links between documents

Creating a part template tutorial

Using parametrics in a sketch tutorial
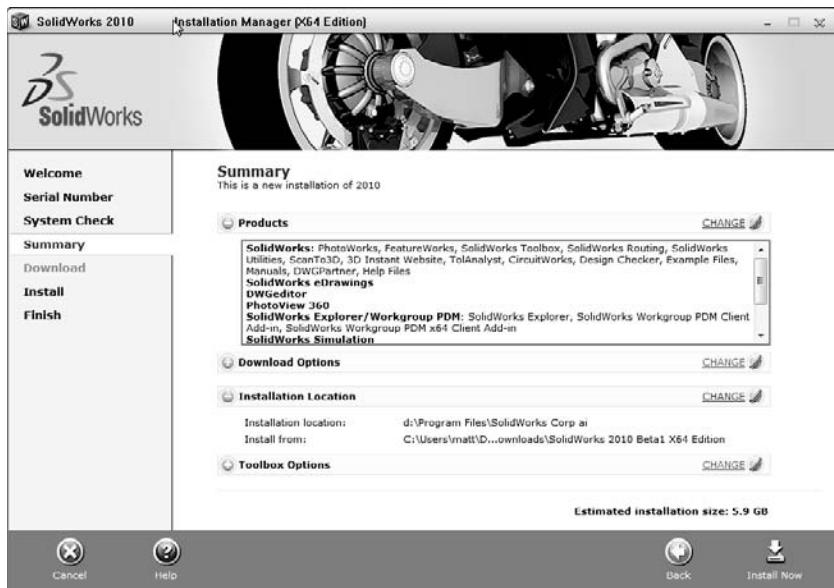
hardware and software are compatible with the SolidWorks system requirements, available on the SolidWorks Web site at `www.solidworks.com/sw/support/System Requirements.html`.

In addition, whether you are installing SolidWorks or someone else is installing it for you, you should consider purchasing a copy of the *SolidWorks Administration Bible* (Wiley, 2010), which contains all the information you need on installing, configuring, and troubleshooting SolidWorks.

Alternatively, you might not want to get that involved at first. You can install the software with all defaults just to get started, using it as a practice installation, especially if you intend to learn as much as you can about it, and then come back and do a more thorough job of implementing it. If this is the case, just put the DVD in the drive and accept all the defaults. In the Summary page, shown in Figure 1.1, you can choose to change the disk installation location and choose which products are installed.

FIGURE 1.1

Changing installation settings on the Summary install page

When installing any software, it is best practice to exit out of all other software first, make sure you have enough hard drive space and otherwise meet the system requirements outlined on the SolidWorks System Requirements Web page, and reboot when the installation is complete.

Starting SolidWorks for the First Time

SolidWorks has many tools for beginning users that are available when the software is installed. A default installation presents you with several options when the software is started the first time. Following is a description of these options, and how you can most benefit from them.

If you plan to go to formal SolidWorks reseller-based training classes, it is a very good idea to go through some of the tutorials mentioned in this section first; this way you are prepared to ask educated questions and have a leg up on the rest of the class. You will get more out of the training with the instructor if you have seen the material once before.

SolidWorks License Agreement

It is useful to be familiar with what this document says, but the agreement does not have any bearing on learning how to use the software other than the fact that it allows for a Home Use License. Many users find this part of the license agreement helpful. The primary user of the license at work is also allowed to use the license at home or on a portable computer. This is often a good option for learning techniques, doing additional practice, or completing the design of that deck or soapbox derby car. If your employer uses floating licenses, the rules are somewhat different. Contact your reseller for details.
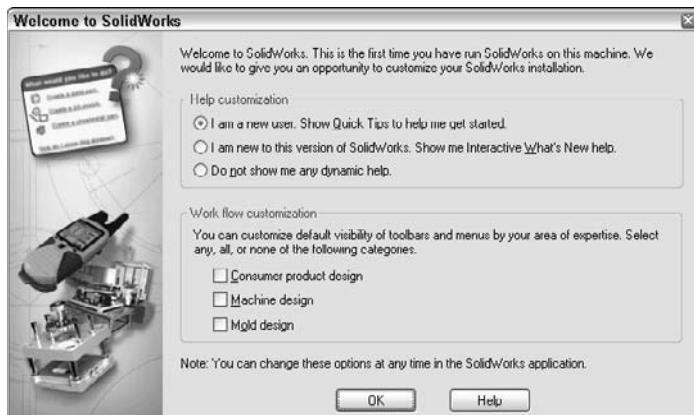
Welcome to SolidWorks

The Welcome to SolidWorks screen, shown in Figure 1.2, is the next thing to greet you. This helps you establish what type of tools you would like to see in the interface and gives you some help options. You may not get the chance to see this dialog box if someone else, such as an IT person, has installed and done an initial test on your software for you.

FIGURE 1.2

The Welcome to SolidWorks screen
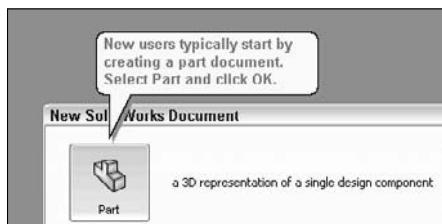
Quick Tips

Quick Tips enables balloons with tips to help you get started with several tasks. For example, the first Quick Tip you see may be the one shown in Figure 1.3. When you begin to create your first document in SolidWorks, a Quick Tip helps guide you on your way.

FIGURE 1.3
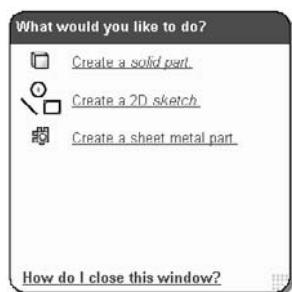
The SolidWorks Document Quick Tip

As you continue working, Quick Tips displays a box, shown in Figure 1.4, at the lower-right corner of the Graphics Window that offers context-sensitive help messages. As you work with the software, these messages change to remain relevant to what you are doing.

FIGURE 1.4
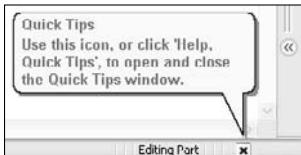
The main Quick Tip window

You can turn Quick Tips on or off by clicking the small square on the Status Bar in the lower-right corner, as shown in Figure 1.5. You can turn the Status Bar on or off in the View menu; however, the Status Bar serves many useful purposes for all users, so I recommend you leave it selected. You can also turn Quick Tips off in the Help menu by deselecting Quick Tips. The on/off setting is document-type sensitive, so if you deselected Quick Tips off in part mode, you will need to do it again for assemblies and drawings, as well. Quick Tips are a great way to get going or to get a little refresher if it has been a while since you last saw the software.

Turning Quick Tips on or off
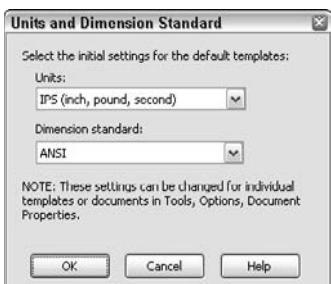
To start a new SolidWorks document, click the New icon in the title bar of the SolidWorks application. With standard functions such as creating a new document, SolidWorks works just like a Microsoft Office application, and the icons even look the same.

The first time you create a document, SolidWorks prompts you to select units for your default templates, as shown in Figure 1.6. This is an important step, although you can make changes later if needed. SolidWorks stores most of the document-specific settings in document templates, which you can set up with different settings for each type of document — parts, assemblies, and drawings. More information on part and assembly templates can be found later in this chapter. Drawing Templates are described in detail in Chapter 20.

The default template units selection

One of the most common questions new users ask is how they can change the default so that new documents come up with a certain type of units every time. Units in new documents are set by templates. To create a part with inch units, use a template with inch units. You can have as many templates as you want, and can have a different template for each type of units you might use.

ISO (International Organization for Standardization) and ANSI (American National Standards Institute) standards use different methods of projecting views, and these standards are controlled by templates. ISO is typically a European standard and uses First angle projection, while ANSI is an American standard and uses Third angle projection. The standard projection used throughout this book is Third angle.
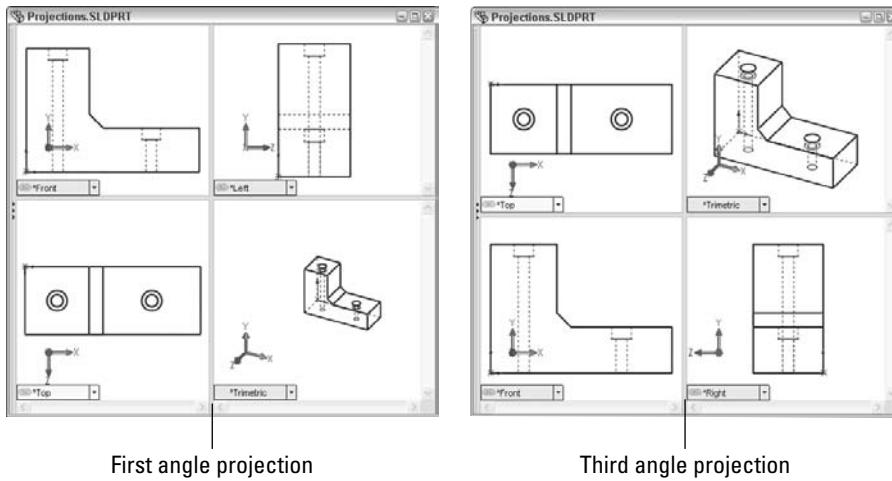
The difference between Third and First angle projections can cause parts to be manufactured incorrectly if those reading the prints (or making the prints) do not catch the difference or see that there is some discrepancy. Figure 1.7 demonstrates the difference between the two projection types. Make sure to get the option correct. If someone else, such a computer specialist who is not familiar with mechanical drafting standards, initially sets up SolidWorks on your computer, you will want to verify that the default templates are correct.

Differences between First (left) and Third (right) angle projections

First angle projection   Third angle projection

Another setting affecting projections that you will want to check can be accessed by choosing Tools ➪ Options ➪ Display/Selection ➪ Projection type for four view viewport. This does not follow the dimensioning standard selected for the default templates or the country in which the software is installed.

SolidWorks documentation

Several types of documentation are available to help SolidWorks learners along their path. A great place to start is the SolidWorks Resources tab in the Task Pane (on the right side of the screen). This is the first tab in the list and has the Home icon. The Getting Started option in the SolidWorks Resources tab is shown in Figure 1.8.

The Getting Started option on the SolidWorks Resources tab of the Task Manager

The terminology SolidWorks has used over the past several releases has changed and may seem confusing to some of you. In 2007, SolidWorks changed the name Online User's Guide to SolidWorks Help. This is a help file on the local computer; it is not online in that it is not on the Internet. For several years after software companies eliminated hard copy documentation, it was popular to call electronic documentation "online" even though the documentation was only on the local computer. In 2010, the SolidWorks Web Help option was added to the Help menu. This option is online in that it is on the Internet.

Tutorials

When you select the Online Tutorials option in the Help menu, you get a list of tutorials on subjects from sheet metal to macros in parts, assemblies, and drawings. These tutorials are certainly worth your time, and will build your skills and knowledge of basic functionality. This *SolidWorks Bible* distinguishes itself from the tutorials by going into far more detail and depth about each function, adding information such as best practices, performance considerations, and cautionary data, acting as a thorough desk reference. The purpose of this book is not to duplicate all the resources for beginners, but to take the information into far more depth and detail, and answer the "why" questions instead of just the "how" questions.

What's New

With every release, SolidWorks publishes a What's New document to help you keep up to speed with the changes. This is typically a PDF file with accompanying example files. If you have missed a version or two, reading through the What's New files can help get you back on track. (You can find every What's New document from SolidWorks 98+ through SolidWorks 2010 on Ricky Jordan's blog, at `http://rickyjordan.com`.) Again, don't expect many detail or interface screen shots; it introduces you to the basic changes.

Moving from AutoCAD

In the Help menu is a selection called Moving from AutoCAD. You may also see it referred to as Moving from 2D to 3D in various places in the SolidWorks interface. It is intended to help transitioning users acclimate to their new surroundings. Terminology is a big part of the equation when making this switch, and figures prominently in the Moving from AutoCAD help file.

Likely, the most helpful sections in Moving from AutoCAD are Approach to Modeling and Imported AutoCAD Data. The information in these categories is useful whether you are coming to SolidWorks from AutoCAD or from another CAD package.

SolidWorks Help

SolidWorks Help (formerly The Online User's Guide) is the new Help file. There are two formats: the traditional Help format and a Web-based help file. You can use Search capabilities to find what you are looking for. The SolidWorks Help has tutorials and a separate API (application programming interface) help file. Frankly, it lacks detail, and often skips over important facts such as what you might use a certain function for, what the interface looks like, or where you might find the command in the first place. This *SolidWorks Bible* fills in most of the gaps in information about the standard version of the software.

Tip of the Day

The SolidWorks Tip of the Day is displayed at the bottom of the SolidWorks Resources tab in the Task Pane. Cycling through a few of the tips or using them to quiz co-workers can be a useful skills-building exercise.

Hardcopy documentation

Unfortunately, hardcopy documentation has dwindled from all software companies. Software suppliers often claim that keeping up with the changes in print is too much work and inefficient. This is the same reason that SolidWorks gives for changing from help files that are on your local computer to help files that are only available across the Internet. Still many users prefer to have a physical book in their hands, one they can spread out on the desk next to them; earmark, highlight, and mark with Post-its; and take notes in, as evidenced by you holding this book at this moment. Hardcopy documentation has an important role in the dissemination of information.

Identifying SolidWorks Documents

SolidWorks has three main data type files: parts, assemblies, and drawings; however, there are additional supporting types that you may want to be familiar with if you are concerned with customization and creating implementation standards. Table 1.1 outlines the document types.

TABLE 1.1

Document Types

Design Documents	Description
.sldasm	SolidWorks assembly file type
.slddrw	SolidWorks drawing file type
.sldprt	SolidWorks part file type

Templates and Formats	Description
.asmdot	Assembly template
.asmprp	Assembly custom properties tab template
.drwdot	Drawing template
.drwprp	Drawing custom properties tab template
journal.doc	Design journal template
.prtdot	Part template
.prtprp	Part custom properties tab template
.sldbombt	BOM template (table-based)
.sldtbt	General table template
.slddrt	Drawing sheet format
.sldholtbt	Hole table template
.sldrevtbt	Revision table template
.sldwldtbt	Weldment cutlist template
.xls	BOM template (Excel-based)

Library Files	Description
.sldblk	Blocks
.sldlfp	Library part file

Styles	Description
.sldgtolfvt	Geometric tolerance style
.sldsffvt	Surface finish style
.sldweldfvt	Weld style

Symbol Files	Description
gtol.sym	A symbol file that enables you to create custom symbols
swlines.lin	A Line style definition file that enables you to create new line styles

Others	Description
.btl	Sheet metal bend table
calloutformat.txt	Hole callout format file
.sldclr	Color palette file
.sldreg	SolidWorks settings file
.sldmat	Material database
.sldstd	Drafting standard
.swb, swp	Macros, macro features
.txt	Custom property file, sheet metal bend line note file
.xls	Sheet metal gauge table

Saving your setup

If you have taken time to set up a computer and then need to reinstall SolidWorks, move to another computer, or duplicate the setup for another user, you need to copy out the files you have used or customized. By default, all these files are located in different folders within the SolidWorks installation directory. Chapter 2 deals with interface settings and creating a registry settings file to copy to other computers or use as a backup.

Best Practice

It is especially important to have copies of these files in a location other than the default installation folder when you are doing complex implementations that include templates for various types of tables or customized symbol files. Uninstalling SolidWorks or installing a new version will wipe out all your hard work. Choose Tools ➪ Options ➪ File Location to save these files in separate library folders on the local hard drive or on a network location. ■

Using templates

I have included some of my part and assembly templates on this book's CD-ROM for you. Copy these files to the location specified at Tools ➪ Options ➪ File Locations ➪ Document Templates.

When you begin to create a new document, and the New SolidWorks Document dialog box gives you the option to select one of several files to start from, those files are templates. Think of templates as "start parts" that contain all the document-specific settings for a part (Tools ➪ Options ➪ Document Properties). The same concept applies to assemblies and drawings. Templates generally do not have any geometry in them (although it is possible).

Tip

The Novice interface for the New SolidWorks Document dialog box (File ➪ New SolidWorks Document) only enables you to select default templates. The Advanced interface enables you to select any available template. ■

As shown in Figure 1.9, several tabs can be displayed on the advanced interface. Each of these tabs is created by creating a folder in the template directory specified in the Options dialog box (Tool ➪ Options).

Having multiple document templates available

Using multiple templates enables you to start working from multiple starting points, which is an advantage in many situations, such as:

- Standardization for a large number of users
- Working in various units
- Preset materials
- Preset custom properties
- Parts with special requirements, such as sheet metal or weldments
- Drawings of various sizes with formats (borders) already applied
- Drawings with special notes already on the sheet

FIGURE 1.9

The Novice and Advanced interfaces for the New SolidWorks Document dialog

Cross-Reference

Drawing templates and formats are complex enough that I cover them in a separate chapter. Chapter 20 discusses the differences between templates and formats, and how to use them to your advantage. This chapter addresses part and assembly templates. ∎

Depending on your needs, it might be reasonable to have templates for metric and inch part and assembly, templates for steel and aluminum, and templates for sheet metal parts and for weldments, if you design these types of parts. If your firm has different customers with different requirements, you might consider using separate templates for each customer. Over time, you will discover the types of templates you need, because you will find yourself making the same changes repeatedly.

To create a template, open a document of the appropriate type (part or assembly), and make the settings you want the template to have; for example, units are one of the most common reasons to make a separate template, though any Document Property setting is fair game for a template, from the dimensioning standard used to the image quality settings.

Some document specific settings do not appear in the Document Properties dialog box. Still, these settings are saved with the template. Settings that fall into this category are the View menu entity type visibility option and the Tools⇨Sketch Settings menu option.

Custom Properties are another piece of the template puzzle. If you use or plan to use BOMs (Bills of Materials), PDM (Product Data Management), or linked notes on drawings, you need to take advantage of the automation options available with custom properties. Setting up custom properties is covered in detail in Chapter 20.

In addition, the names of the standard planes are template specific. For example, the standard planes may be named Front, Top, and Side; or XY, XZ, and ZY; or Plane1, Plane2, and Plane3; or North, Plan, and East; or Elevation, Plan, and Side for different uses.

Locating templates

You can set the location of the templates folder at Tools⇨Options⇨File Locations⇨Document Templates. The folder location may be a local folder or a shared network folder. Multiple folders may be specified in the list box, each corresponding to a tab in the New Document's Advanced interface.
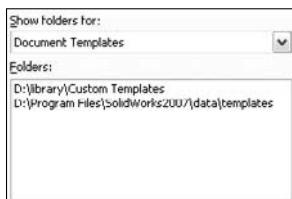
After all the Document Properties, custom properties, and other settings are set to your liking and you are ready to save the file as a template, choose File⇨Save As and select Part Templates in Files of Type. SolidWorks prompts you to save the template in the first folder listed in the File Locations list. You can create assembly templates the same way, except you change the settings for an assembly document.

You can also create additional tabs on the New SolidWorks Document dialog box by making subfolders in the main folder in the File Locations area. For example, if your File Locations list for Document Templates looks like Figure 1.10, your New SolidWorks Document dialog box will look like Figure 1.11.
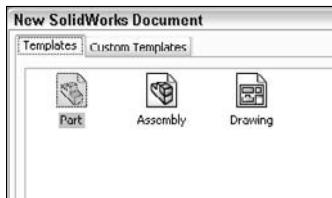
FIGURE 1.10

The Tools⇨Options⇨File Locations list
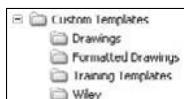
The New SolidWorks Document dialog box

Adding subfolders to either of the locations listed in File Locations results in additional tabs in the New SolidWorks Document dialog box, as shown in Figures 1.12 and 1.13.
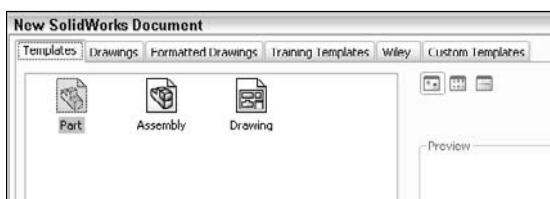
FIGURE 1.12

Additional subfolders added to a File Locations path

FIGURE 1.13

The tabs associated with the subfolders in New SolidWorks Document dialog box

Default templates
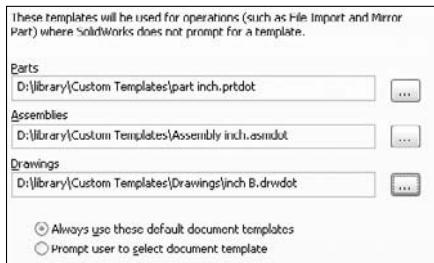
Default templates are established at Tools ➪ Options ➪ Default Templates. The default templates must be in one of the paths specified in File Locations. Figure 1.14 shows the Default Templates settings.

FIGURE 1.14

The Tools ⇨ Options ⇨ Default Templates settings

These templates will be used for operations (such as File Import and Mirror
Part) where SolidWorks does not prompt for a template.

Parts
D:\library\Custom Templates\part inch.prtdot   [...]

Assemblies
D:\library\Custom Templates\Assembly inch.asmdot   [...]

Drawings
D:\library\Custom Templates\Drawings\inch B.drwdot   [...]

⦿ Always use these default document templates
◯ Prompt user to select document template

There are two Default Template options: Always use these default document templates and Prompt user to select document template. The Default Template options apply to situations when a template is required by an automatic feature in the software such as an imported part or a mirrored part. In this situation, depending on the option selected, the system automatically uses the default template or the user is prompted to select a template.

Performance

Allowing the software to apply the default template automatically can have a great impact on speed. This is especially true in the case of imported assemblies, which would require you to select templates manually for each imported part in the assembly if the Prompt user to select document template option is selected. ∎

Sharing templates

If you are administering an installation of a large number of users, or even if there are just a couple of users working on similar designs, shared templates are necessary. If every user does what she thinks best, you may get an interesting combination of conflicting ideas, and the consistency of the company's documentation may suffer. Standardized templates cannot make users model, assemble, and detail in exactly the same way, but they do start off users on the same foot.

To share templates among several users, create a folder for templates on a commonly accessible network location, preferably with read-only access for users and read-write permissions for administrators. Then point each user's File Locations and Default templates to that location. Access problems due to multiple users accessing the same files do not arise in this situation because users copy templates to create new documents, and do not use them directly.

Caution

One of the downfalls of this arrangement is that if the network goes down, users no longer have access to their templates. This can be averted by also putting copies of the templates on the local computers; however, it has the tendency to undermine the goal of consistent documentation. Users may tend to use and customize the local templates rather than use the standardized network copies.

CAD administration and organizing any group of people on some level always comes down to trusting employees to do the right thing. There is no way to completely secure any system against all people trying to work around the system, so you must rely on hiring people you can train and trust. ∎

Understanding Feature-Based Modeling
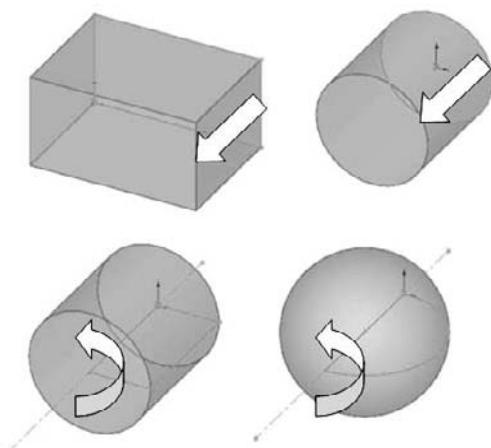
You need to be familiar with some terminology before diving into building models with SolidWorks. Notice that I talk about "modeling" rather than "drawing," or even "design." This is because SolidWorks is virtual prototyping software. Whether you are building an assembly line for automotive parts or designing decorative perfume bottles, SolidWorks can help you visualize your geometrical production data in the most realistic way possible without actually having it in your hand. This is more akin to making a physical model in the shop than drawing on paper.

"Feature-based" modeling means that you build the model by incrementally identifying functional shapes and applying processes to create the shapes. For example, you can create a simple box by using the Extrude process, and you can create a sphere using the Revolve process. However, you can make a cylinder using either process, by revolving a rectangle or extruding a circle. You start by visualizing the 3D shape, and then apply a 3D process to a 2D sketch to create that shape. This concept on its own is half of what you need to know to create models with SolidWorks.

Figure 1.15 shows images of simple feature types along with the 2D sketches from which they were created.

FIGURE 1.15

Simple extruded and revolved features

Many different feature types in SolidWorks enable you to create everything from the simplest geometry shown in Figure 1.15 to more complex artistic or organic shapes. In general, when I talk about modeling in this book, I am talking about *solid* modeling, although SolidWorks also has a complete complement of surfacing tools. I discuss the distinction between solid and surface modeling in Chapter 27.

Table 1.2 lists some of the most common features that you find in SolidWorks, and classifies them according to whether they always require a sketch, a sketch is optional, or they never require a sketch.

TABLE 1.2

Feature Types

Sketch Required	Sketch Optional	No Sketch (Applied Features)
Extrude	Loft	Fillet
Revolve	Sweep	Chamfer
Rib	Dome	Draft
Hole Wizard	Boundary	Shell
Wrap	Deform	Flex

In addition to these features, other types of features create reference geometry, such as curves, planes, axes, surface features (Chapter 27); specialty features for techniques like weldments (Chapter 31); plastics/mold tools (Chapter 32); and sheet metal (Chapter 29).

Understanding History-Based Modeling
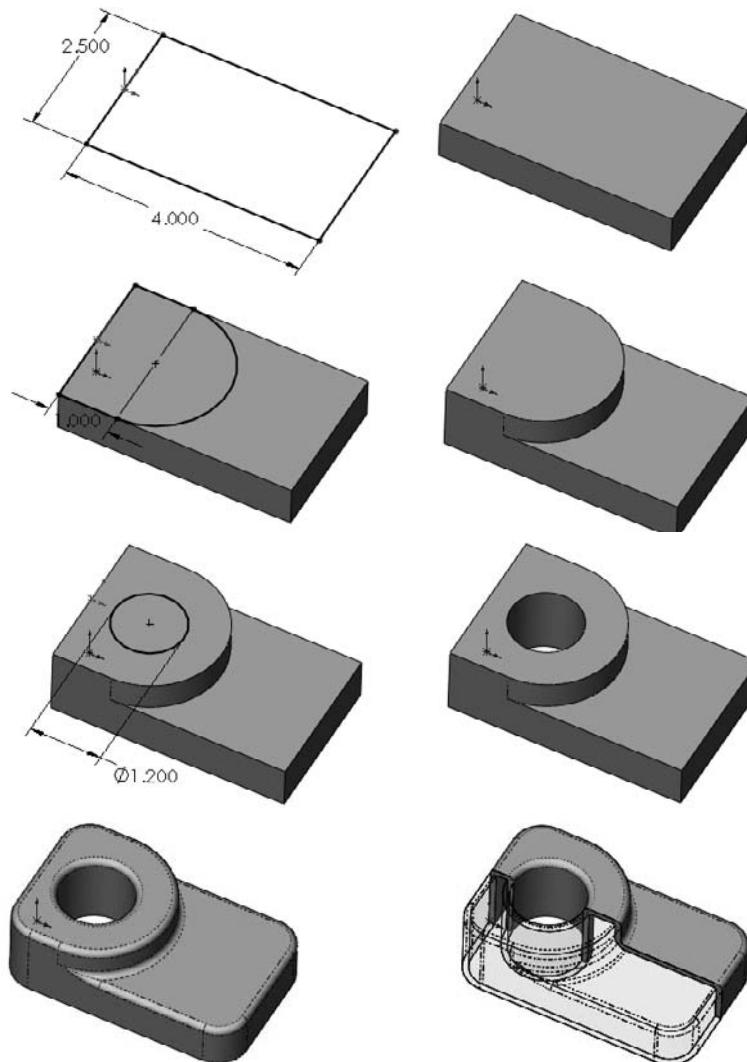
In addition to being feature-based, SolidWorks is also history based. To show the process history, there is a panel to the left side of the SolidWorks window called the *FeatureManager*. The FeatureManager keeps a list of the features in the order in which you have added them. It also enables you to reorder items in the tree (in effect, to change history). Because of this, the order in which you perform operations is important. For example, consider Figure 1.16. This model was created by the following process, left to right starting with the top row:

1. **Create a sketch.**
2. **Extrude the sketch.**
3. **Create a second sketch.**
4. **Extrude the second sketch.**
5. **Create a third sketch.**

6. **Extrude Cut the third sketch.**

7. **Apply fillets.**

8. **Shell the model.**

FIGURE 1.16

Features used to create a simple part

If the order of operations used in the previous part were slightly reordered (by putting the shell and fillet features before Step 6), the resulting part would also look slightly different, as shown in Figure 1.17.

Using a different order of features for the same part
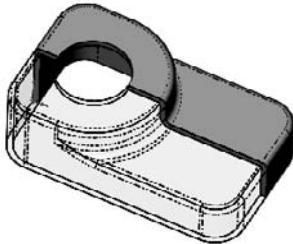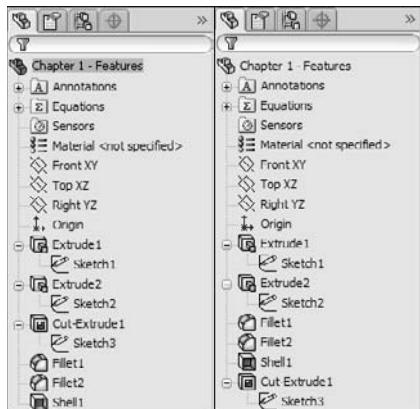
Figure 1.18 shows a comparison of the FeatureManager design trees for the two different feature orders. You can reorder features by dragging them up or down the tree. Relationships between features can prevent reordering; for example, the fillets are dependent on the second extruded feature and cannot be reordered before it. This is referred to as a *Parent/Child relationship*.

Cross-Reference

Reordering and Parent/Child relationships are discussed in more detail in Chapter 11. ■

Compare the FeatureManager design trees for the parts shown in Figure 1.16 and Figure 1.17.

On the CD-ROM

**The part used for this example is available in the material on the CD-ROM, named Chapter 1 — Features.
SLDPRT. ∎**

The order of operations, or *history*, is important to the final state of the part. For example, if you
change the order so that the shell comes before the extruded cut, the geometry of the model
changes, removing the sleeve inside instead of the hole on top. You can try this for yourself by
opening the part indicated previously, dragging the Shell1 feature in the FeatureManager, and
dropping it just above the Cut-Extrude1 feature.

Note

**You can only drag one item at a time in the FeatureManager. Therefore, you may drag the shell, and then drag
each of two fillets, or you could just drag the cut feature down the tree. Alternatively, you can put the shell
and fillets in a folder and drag the folder to a new location. Reordering is limited by parent-child relationships
between dependent features. ∎**

Cross-Reference

You can read more about reordering folders in Chapter 11. ∎

In some cases, reordering the features in the FeatureManager may result in geometry that might not
make any sense; for example, if the fillets are applied after the shell, they might break through to the
inside of the part. In these cases, SolidWorks gives an error that helps you to fix the problem.

In 2D CAD programs where you are just drawing lines, the order in which you draw the lines does
not matter. This is one of the fundamental differences between history-based modeling and drawing.

Features are really just like steps in building a part; the steps can either add material or remove it.
However, when you make a part on a mill or lathe, you are only removing material. The
FeatureManager is like an instruction sheet to build the part. When you reorder and revise history,
you change the order of operations and thus the final result.

Sketching with Parametrics

Sketching is the foundation that underlies the most common feature types. You will find that
sketching in parametric software is vastly different from drawing lines in 2D CAD.

Dictionary.com defines the word *parameter* as "one of a set of measurable factors . . . that define a
system and determine its behavior and [that] are varied in an experiment." SolidWorks sketches
are parametric. What this means is that you can create sketches that change according to certain
rules, and maintain relationships through those changes. This is the basis of parametric design. It
extends beyond sketching to all the types of geometry you can create in SolidWorks. Creating
sketches and features with intelligence is the basis of the concept of Design Intent, which I cover in
more detail later in this chapter.

In addition to 2D sketching, SolidWorks also makes 3D sketching possible. Of the two methods, 2D sketches are by far more widely used. You create 2D sketches on a selected plane, planar solid, or surface face and then use them to establish shapes for features such as Extrude, Revolve, and others. Relations in 2D sketches are often created between sketch entities and other entities that may or may not be in the sketch plane. In situations where other entities are not in the sketch plane, the out-of-plane entity is projected into the sketch plane in a direction that is normal to the sketch plane. This does not happen for 3D sketches.

You can use 3D sketches for the Hole Wizard, routing, weldments, and complex shape creation, among other applications.
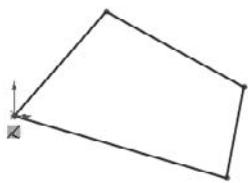
Cross-Reference

For more information on 3D sketching, please refer to Chapter 31. ■

For a simple example of working with sketch relations in a 2D sketch, consider the sketch shown in Figure 1.19. The only relationships between the four lines are that they form a closed loop that is touching end to end, and one of the corners is coincident to the part origin. The small square icon near the origin shows the symbol for a *coincident* sketch relation. These sketch relations are persistent through changes, and enable you to dynamically move sketch elements with the cursor on the screen. The setting to enable or disable displaying the sketch relation symbols is found at View ➪ Sketch Relations.

FIGURE 1.19

A sketch of four lines
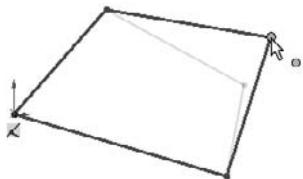
If you drag any of the unconstrained corners (except for the corner that is coincident to the origin), the two neighboring lines will follow the dragged endpoint, as shown in Figure 1.20. Notice the ghosted image left by the original position of the sketch. This is helpful when experimenting with changes to the sketch because you can see both the new and the old states of the sketch. The setting to enable or disable this ghosted position is found at Tools ➪ Options ➪ Sketch ➪ Ghost Image on Drag.

If you add a parallel relation between opposing lines, they now act differently, as shown in Figure 1.21. You add a parallel relation by selecting the two lines to make parallel and selecting Parallel from the PropertyManager panel. You can also select the Parallel relation from the context bar that pops up in the graphics window when you have both lines selected.

Dragging an endpoint
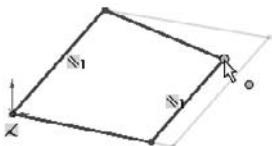
Cross-Reference

You can read more about the PropertyManager in Chapter 2. ■
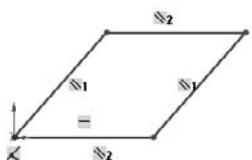
Dragging an endpoint where lines have relations

Next, add a second parallel and a horizontal relation, as shown in Figure 1.22. If you are following along by re-creating the sketch on your computer, you will notice that one line has turned from blue to black.

Horizontal and parallel relations are added.

The colors represent sketch states. It may be impossible to see this in the black and white printing of this book, but if you are following along on your own computer, you can now see one black line and three blue lines. Sketch states include Underdefined, Overdefined, Fully Defined, Unsolvable, Zero Length, and Dangling, and are described as follows:
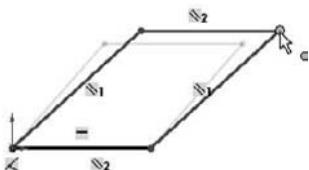
- **Blue: Underdefined.** The sketch entity is not completely defined. You can drag a portion of it to change size, position, or orientation.

- **Black: Fully Defined.** The sketch entity is fully defined by a combination of sketch relations and dimensions. A sketch cannot be fully defined without being connected in some way to something external to the sketch, such as the part origin or an edge. (The exception to this rule is the use of the Fix constraint, which, although effective, is not a recommended practice.)

- **Red: Overdefined – Not Solved.** When a sketch entity has two or more relations and one of them cannot be satisfied, the unsatisfied will be red. For example, if a line has both Horizontal and Vertical relations, and the line is actually vertical, the Vertical relation will be yellow (because it is conflicting but satisfied), and the Horizontal will be red (because it is conflicting and not satisfied).

- **Yellow: Overdefined - Conflicts.** Solving the sketch relations would result in a zero-length entity; for example, this can occur where an arc is tangent to a line, and the centerpoint of the arc is also coincident to the line.

- **Brown: Dangling.** The relation has lost track of the entity to which it was connected.

- **Pink.** The pink sketch status is no longer used.

There can be entities with different states within a single sketch. In addition, endpoints of lines can have a different state than the rest of the sketched entity. For example, a line that is sketched horizontally from the origin has a *coincident* at one endpoint to the origin, and the line itself is *horizontal*. As a result, the line and first endpoint are black, but the other endpoint is underdefined because the length of the line is not defined. Sketch states are indicated in the lower-right corner of the graphics window and in the status bar. You can see that dragging one corner allows only the lines to move in certain ways, as shown in Figure 1.23.

FIGURE 1.23
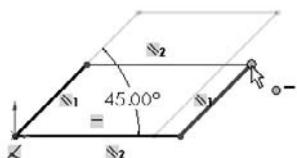
Sketch motion is becoming more constrained.

In addition to sketch relations, dimensions applied using the Smart Dimension tool are also part of the parametric scheme. If you apply an angle dimension (by clicking the two angled lines with the Smart Dimension tool) about the origin and try dragging again, as shown in Figure 1.24, you see that the only aspect that is not locked down is the length of the sides. Notice also that when the angle dimension is added, another line turns black.

FIGURE 1.24

Open degrees of freedom can be dragged.
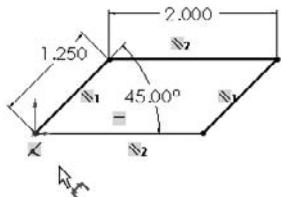
Finally, adding length dimensions for the unequal sides completes the definition of the sketch, as shown in Figure 1.25. At this point, all lines have turned black. This state is called "fully defined." Between the dimensions and sketch relations, there is enough information to re-create this sketch exactly.

Best Practice

It is considered best practice to fully define all sketches. However, there are times when this is not practical. When you create freeform shapes, generally by using splines, these shapes cannot easily be fully defined, and even if they are fully defined, the extra dimensions are usually meaningless, because it is impractical to dimension splines on manufacturing drawings. ■

FIGURE 1.25

The fully defined sketch cannot be dragged, and there are no degrees of freedom.

Parametric relations within a sketch control how the sketch reacts to changes from dimensions or relations within the sketch or by some other factor from outside the sketch. Other factors can also drive the sketch, such as equations, other model geometry that is external to the sketch, and even geometry from another part in an assembly, as you shall see later.

Understanding Design Intent

Design Intent is a phrase that you will hear often among SolidWorks users. I like to think of it as "design for change." Design Intent means that when you put the parametric sketch relations together with the feature intelligence, you can build models that react to change in predictable ways. This gives you a great deal of control over changes.

An example of Design Intent could be a written statement that describes general aspects that help define the design of a part, such as "This part is symmetrical, with holes that line up with Part A and thick enough to be flush with Part B." From this description, and the surrounding parts, it is possible to re-create the part in such a way that if Part A or Part B changes, the part being described updates to match.

Some types of changes can cause features to fail or sketch relations to conflict. In most situations, SolidWorks has ample tools for troubleshooting and editing that you can use to repair or change the model. In these situations, it is often the Design Intent itself that is changing.

Best Practice
When editing or repairing relations, it is considered best practice to edit rather than delete. Deleting often causes additional problems further down the tree. Many users find it tempting to delete anything that has an error on it. ■

Editing Design Intent

Design Intent is sometimes thought of as a static concept that controls changing geometry. However, this is not always the way things are. Design Intent itself often changes, thus requiring the way in which the model reacts to geometric changes to also change. Fortunately, SolidWorks has many tools to help you deal with situations like this.

Choosing Sketch Relations

Seeing the sketch relation symbols is the best tool for visualizing Design Intent. You can show or hide icons that represent the relations by choosing View ➪ Sketch Relations. When shown, these relations appear as an icon in a small colored box in the graphics area next to the sketch entity.

Clicking the icon highlights the sketch elements involved in that relation. Refer to Figures 1.19 through 1.25 for examples of these relations.

Tip

The Sketch Relations option is an excellent candidate for use with a hotkey, thus enabling you to toggle the display easily on and off. ■

Cross-Reference

For more information on creating and managing hotkeys, see Chapter 2. ■

You can use the sketch relation icons on the screen to delete relations by selecting the icon and pressing Delete on the keyboard. You can also use them to determine the status quickly of sketch relations by referring to the colors defined earlier.

Selecting Display/Delete Relations

You can find the Display/Delete Relations tool on the Sketch toolbar or by clicking a sketch entity in an open sketch. The sketch status colors defined earlier also apply here, with the relations appearing in the appropriate color. (Relations are not shown in blue or black, only the colors that show errors, such as red, yellow, and brown.) This tool also enables you to group relations by several categories:

- All in This Sketch
- Dangling
- Overdefining/Not Solved
- External
- Defined in Context
- Locked
- Broken
- Selected Entities

In the lower Entities panel of the Display/Delete Relations PropertyManager, shown in Figure 1.26, you can also replace one entity with another, or repair dangling relations.

Cross-Reference

You can read more about repairing dangling entities in Chapter 11. ■

FIGURE 1.26

The Display/Delete Relations PropertyManager enables you to repair broken relations

Using Suppressed sketch relations

Suppressing a sketch relation means that the relation is turned off and not used to compute the position of sketch entities. Suppressed relations are generally used in conjunction with configurations.

Cross-Reference

Configurations are discussed in detail in Chapter 10. ∎

Working with Associativity

In SolidWorks, associativity refers to links between documents, such as a part that has an associative link to a drawing. If the part changes, the drawing updates as well. Bi-directional associativity means that the part can be changed from the part or the drawing document window. One of the implications of this is that you do not edit a SolidWorks drawing by simply moving lines on the drawing; you must change the model, which causes all drawing views of the part or assembly to update correctly.

Other associative links include using *inserted* parts (also called *base* or *derived* parts), where one part is inserted as the first feature in another part. This might be the case when you build a casting. If the part is designed in its "as cast" state, it is then inserted into another part where machining operations are performed by cut features and the part is transformed into its "as machined" state. This technique is also used for plastic parts where a single shape spans multiple plastic pieces. A "master part" is created and split into multiple parts. An example would be a mouse cover and buttons.

One of the most important aspects of associativity is file management. Associated files stay connected by filenames. If a document name is changed, and one of the associated files is not updated appropriately, the association between the files can become broken. For this reason, you should use SolidWorks Explorer to change names of associated files. Other techniques will work, but there are some techniques you should avoid. I discuss these techniques in Chapter 28 with master model techniques.

Best Practice

It is considered poor practice to change filenames, locations, or the name of a folder in the path of documents that are referenced by other documents with Windows Explorer. Links between parts, assemblies, and drawings can be broken in this way. Using SolidWorks Explorer or a PDM application is the preferred method for changing filenames. ∎

Tutorial: Creating a Part Template

This simple tutorial steps you through making a few standard part templates for use with inch and millimeter parts, as well as making some templates for a couple of materials.

1. Choose Tools ➪ Options ➪ System Options ➪ File Locations and then select Document Templates from the Show folder for list.

2. Click the Add button to add a new path to a location outside of the SolidWorks installation directory; for example, D:\Library\Templates.

3. Click OK to dismiss the dialog box and accept the settings.

4. Choose File ➪ New from the menu.

5. Select any part template.

6. Choose Tools ➪ Options ➪ Document Properties ➪ Drafting Standard.

7. Make sure the ANSI standard is selected.

8. Click the Units page.

9. Change the unit system to IPS, inches with 3 decimal places, using millimeters as the dual units with 2 decimal places. Set angular units to Degrees with 1 decimal place.

10. Change to the Grid/Snap page.

11. Turn off Display grid.

12. Change to the Image Quality page.
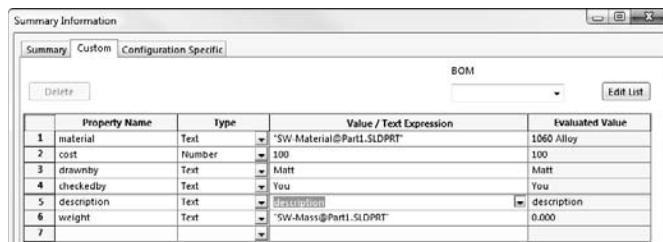
13. **Move two-thirds of the way to the right, so it is closer to High.** Make sure the Save tessellation with part document option is selected.

14. **Click OK to save the settings and exit the Tools ➪ Options dialog box.**

15. **Click the right mouse button (RMB) on the Materials entry in the FeatureManager, and select 1060 Alloy from the list**. If it is not there, click Edit Material and find it from the larger list.

16. **Choose File ➪ Properties, and click the Custom tab.**

17. **Add a property called Material of type Text.** In the Value/Text Expression column, click the down arrow and select Material from the list. Notice that the Evaluated Value shows 1060 Alloy.

18. **Add another property called description and give it a default value of Description.** At this point, the window should look like Figure 1.27.
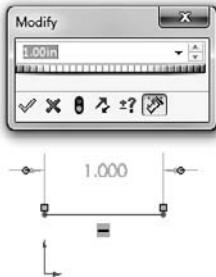
Setting up Custom Properties

19. **Click OK to close the Summary Information window.**

20. **Change the names of the standard planes by clicking them twice slowly or clicking once and pressing F2.** Rename them **Front**, **Top**, and **Side**, respectively.

21. **Ctrl+select the three planes from the FeatureManager, click the RMB, and select Show** (the eyeglasses icon).

22. **From the View menu, make sure that Planes is selected.**

23. **Click the RMB on the Front plane and select Sketch.**

24. **Select the Line tool and click and drag anywhere to draw a line.**

25. **Select the Smart Dimension tool and click the line, then click in the space in the Graphics Window to place the dimension. If you are prompted for a dimension value, press 1 and click the check mark, as shown in Figure 1.28.**

26. **Press Esc to exit the Dimension tool and click the RMB on the displayed dimension and select Link Value.**

27. **Type thickness in the Name box, and click OK.**

Drawing a line and applying a dimension

28. **Press Ctrl+B (rebuild) to exit the sketch, select the sketch from the FeatureManager, and press Delete on the keyboard.**

Note

You do the exercise of creating the sketch and deleting it only to get the link value "thickness" entered into the template. Once you've done it, every part made from this template that uses an Extrude feature will have an option box for Link to Thickness, which enables you to establish a thickness variable automatically for each part you create. This is typically a sheet metal part feature, but you can use it in all types of parts. ■

29. **Choose File ⇨ Save As and then select Part Template from the drop-down list.** Ensure it is going into your template folder by giving it an appropriate name reflecting the inch units and 1060 material, and then click Save.

30. **Edit the material applied to change it from 1060 Alloy to Plain Carbon Steel, and save it as another template with a different name.**

31. **Change the primary units to millimeters with 2 places, and save as a third template file.**

32. **Exit the file.**

Tutorial: Using Parametrics in Sketches

What separates parametric CAD tools from simple 2D drawing programs is the intelligence that you can build in to a parametric sketch. In this tutorial, you learn some of the power that parametrics can provide in both structured (using actual dimensions) and unstructured (just dragging the geometry with the mouse) changes.

1. **Open a new SolidWorks document by clicking the New toolbar button or by choosing File ⇨ New.**

2. **From the list of templates, select a new part template, either inch or millimeter.**
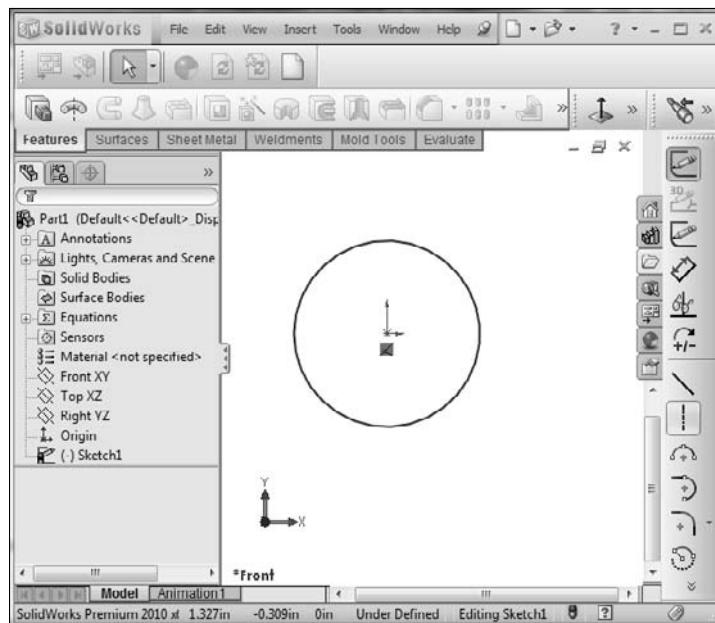
3. **Press the Spacebar on the keyboard to open the View Orientation dialog box, and double-click the Front view.**

4. **Right-click the Front plane in the FeatureManager, or whatever the first plane listed is, and select Sketch.**

5. **Click the View menu, and make sure the Sketch Relations item is depressed.**
   This shows small icons on the screen to indicate when parametric relations are created between sketch entities.

6. **Click the Circle from the Sketch toolbar (choose Tools ⇨ Sketch Entities ⇨ Circle).**

7. **Sketch a circle centered on the Origin.** With the Circle tool activated, click the cursor at the Origin in the graphics area. The Origin is the asterisk at the intersection of the long vertical red arrow and the short horizontal red arrow. After clicking the first point, which represents the center of the circle, move the cursor away from the Origin, and click again, which will establish the radius of the circle. (You can also click and drag between the circle center and the radius if you prefer). Figure 1.29 shows the result.

8. **Deactivate the circle by clicking its toolbar icon or pressing the Esc key on the keyboard.** Now click and hold the cursor on the circle, then drag it to change the size of the circle. The center of the circle is locked to the Origin as the Coincident icon near the Origin appears. The radius is undefined, so it can be dragged by the cursor. If the center-point were not defined, the location of the center of the circle could also be dragged.

FIGURE 1.29
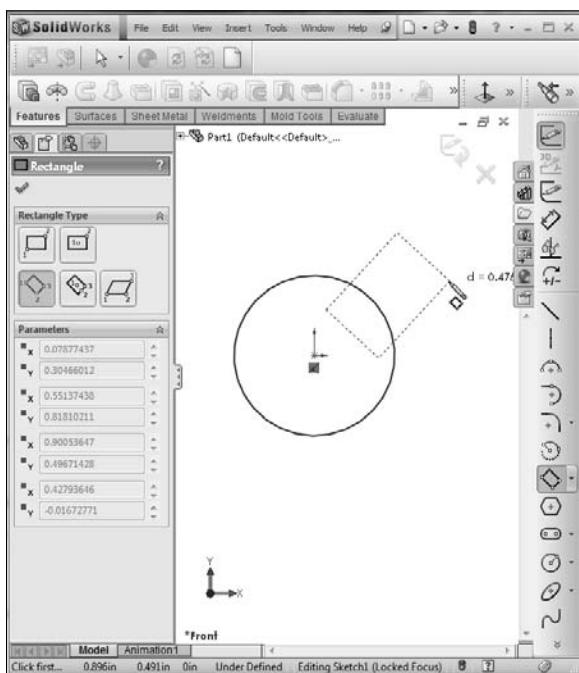
Sketching a circle

9. **Activate the 3 pt Corner Rectangle.** To find it through a menu, choose Tools ⇨ Sketch Entities ⇨ 3pt Corner Rectangle.

10. **Click the first point inside the circle, click the second point outside the circle, and click the third point such that one end of the rectangle is completely inside the circle.** Use Figure 1.30 as a reference. Avoid making any two points vertical or horizontal from one another. You will learn in Chapter 3 about how to control automatic relations in more detail.

FIGURE 1.30

Sketching a three-point rectangle
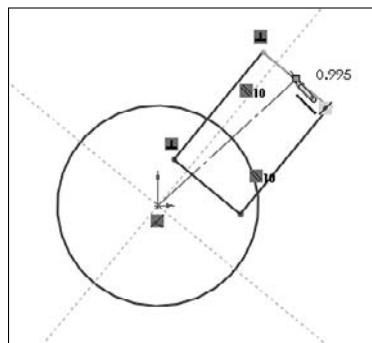
11. **Activate the Centerline tool by choosing Tools ⇨ Sketch Entities ⇨ Centerline.** Click and hold the left mouse button first at the Origin, then at the midpoint of the far end of the rectangle release the mouse button, as shown in Figure 1.31. When the cursor gets close to the midpoint of the end of the rectangle, it snaps into place. Press Esc to turn off the Centerline tool.

12. **Select the centerline and Ctrl+select the line with which it shares the midpoint relation.** After making the second selection, do not move the cursor, and you will see a set of options in a pop-up context toolbar. Click the Make Perpendicular icon, as shown in Figure 1.32, and the rectangle becomes symmetric about the centerline.
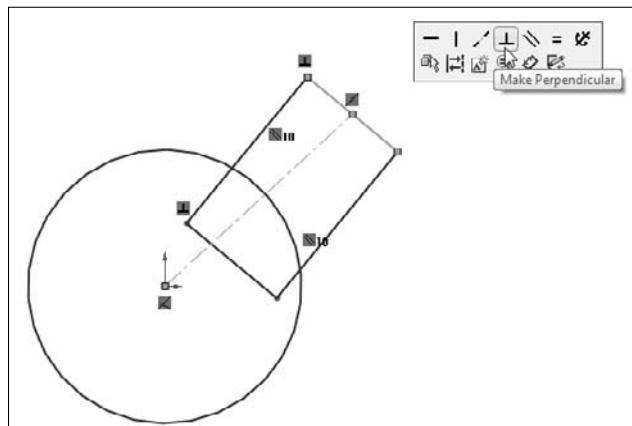
Creating a midpoint relation between a centerline and a line

Making the rectangle symmetric about the centerline
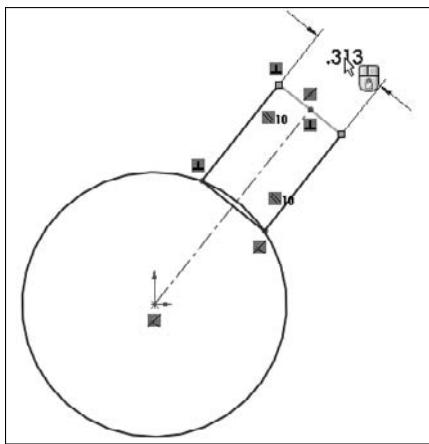
13. **Drag one of the corners of the rectangle from inside the circle and drop it on the circumference of the circle itself.** The point here is that you want the corner of the circle to be right on the circle always. A coincident relation is created by the drag-and-drop action. Now when you move either the circle or the rectangle, the other may change in size. Try dragging the circle itself, a corner point of the rectangle, or a line of the rectangle. Try dragging the centerline and the free endpoint of the centerline. Notice that the result of moving each different sketch entity is different from any of the others.

14. **Click the Smart Dimension tool in the Sketch toolbar or choose Tools⇨Dimensions⇨ Smart.** Click the outside end line of the rectangle, and move the cursor away from the line. You can align the dimension one of three ways: measuring the horizontal dimension of the line, the vertical dimension, or the dimension aligned with the angle of the line. When the dimension is aligned with the angle, as shown in Figure 1.33, click the RMB. This locks in that orientation, and enables you to select a location for the dimension without affecting its orientation. Click to place the dimension. If the dimension does not automatically give you the opportunity to change the dimension, double-click the dimension and change it to 0.5 inch or 12 mm. If this is larger than the diameter of the circle, notice that the circle changes to accommodate the new width.

FIGURE 1.33

Placing a dimension on an angled line

15. **Drag the endpoint of the centerline around the center of the circle to see how the sketch reacts.** Notice that the dimension added to the rectangle keeps it a constant width and makes it react more predictably.

16. **Put a dimension on the circle. Click the circumference of the circle with the Smart Dimension tool, and then place the dimension anywhere on the screen without clicking any other geometry.** Notice that diameter values of less than the dimension created in Step 14 cause errors. This makes geometrical sense. Change the diameter value to 1 inch or 25 mm.

17. **Drag the endpoint of the centerline again.** Notice that as you define more sizes and relationships between sketch entities, the motion of the sketch as you drag becomes more constrained and predictable.
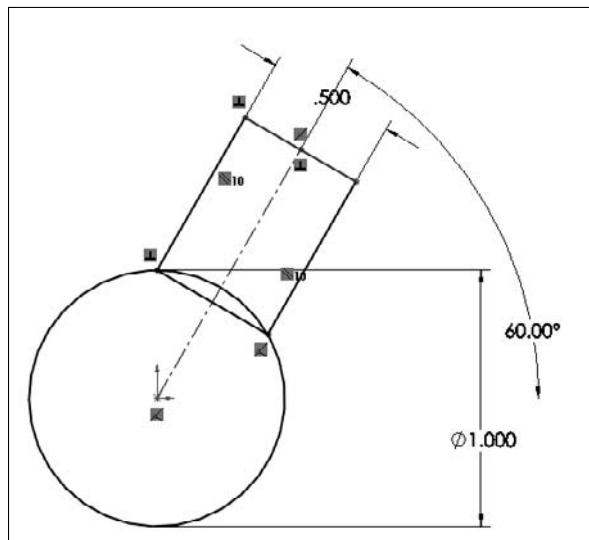
18. **Activate the Smart Dimension tool again.** Click the Top plane from the FeatureManager and then click the centerline. This creates an angle dimension. Position the angle dimension and change its value to 60 degrees, as shown in Figure 1.34.

FIGURE 1.34
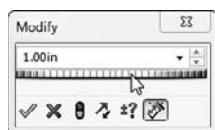
Creating an angle dimension

19. **Double-click any dimension and use either the up and down arrows to the right of it or the scroll wheel below it to change the value, as shown in Figure 1.35.** Notice that not all values of all dimensions produce valid results. Also, notice that the entire sketch is black except for the outer end of the rectangle, which is now blue. This is the only thing you can now drag.

FIGURE 1.35

Using arrows or scroll wheel to change dimension values

Summary

While product development is about design, it is even more about change. You design something once, but you may modify it endlessly (or it may seem that way sometimes). Similarly, SolidWorks is about design, but it really enables change. Think of SolidWorks as virtual prototyping software that enables you to change your prototype rather than having to make a new one. Virtual prototypes will never completely replace physical models, but they may reduce your dependence on them to some extent.

SolidWorks is also about reusing data. Associativity enables you to model a part once and use it for Finite Element Analysis (FEA), creating 2D drawings, building assemblies, creating photorealistic renderings, and so on. When you make changes to the model, your drawing is automatically updated, and you don't have to reapply FEA materials and conditions or redo the rendering setup. Associativity saves you time by reusing your data. Associativity and change driven by feature-based and history-based modeling can take some getting used to if you have had limited exposure to it, but with some practice, it becomes intuitive and you will see the many benefits for enabling change. Parametric sketching and feature creation help you to maintain Design Intent as well as adjust it as necessary.

Navigating the SolidWorks Interface

The SolidWorks interface offers a wide range of tools. You will often find more than one way to do almost everything. There is no one best way to use the interface; and each method has strengths and weaknesses depending on the task.

Once you have mastered the various interface elements and customized your SolidWorks installation, working with the software becomes much more efficient and satisfying. You may find your mastery of the interface comes with practice and experience. The interface is complex and can be daunting. Many existing users may discover features in this book of which they were not aware, even though they have been trained and have used the software for years.

I explain each interface element identified in Figure 2.1 in detail within this chapter.
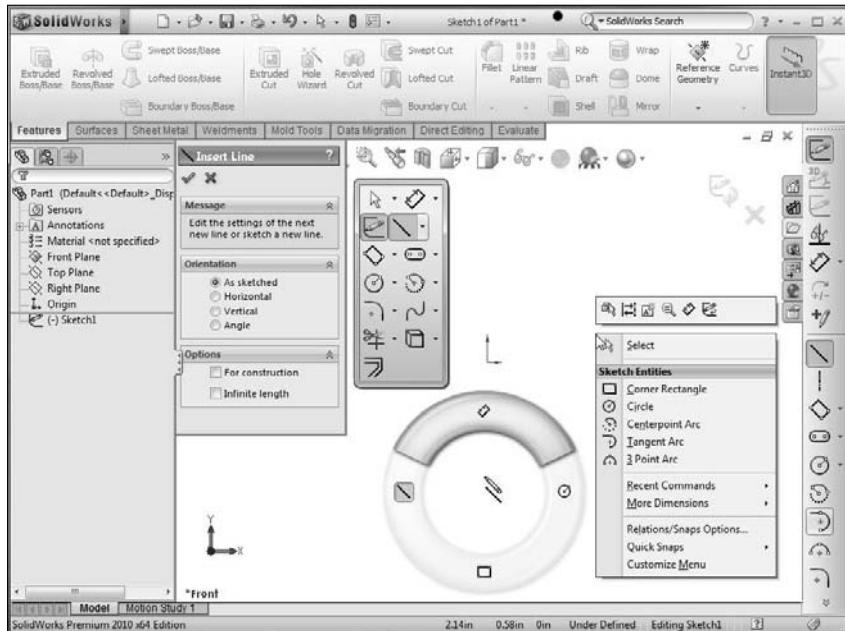
FIGURE 2.1

Elements of the SolidWorks interface

Identifying Elements of the SolidWorks Interface

The major elements of the SolidWorks interface are the graphics window, where all the action takes place, the FeatureManager, which is the list of all the features in the part, the PropertyManager, where most of the data input happens, and the CommandManager and toolbars, where you access most of the commands in the software. Of these, I cover the CommandManager and toolbars first.

Using the CommandManager and toolbars

In some respects, the CommandManager resembles the Microsoft (MS) Ribbon interface found in Office 2007 applications. SolidWorks did not do a strict implementation of the MS Ribbon, because it wanted to add more customizability. A far more complete array of interface configuration possibilities awaits you with SolidWorks 2010. In this section of the chapter, I show you how to make the 2010 CommandManager work for you, and how to use regular or flyout toolbars to replace it effectively.

CommandManager

The CommandManager is an area of the interface that you can use to save space traditionally used by toolbars. The main mission of the CommandManager is to enable you to have easy access to all toolbars, and even access to customized groups of icons not available on a single default toolbar, without cluttering the entire screen with toolbars.
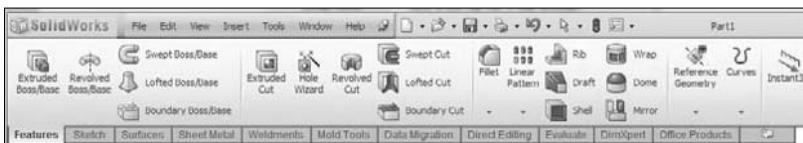
The CommandManager accomplishes this by providing small tabs under the left end of the toolbar area to enable you to switch the toolbar that appears. Figure 2.2 shows the CommandManager in customize mode, showing all the tabs available in a default setup. To get the CommandManager into customize mode, right-click one of the CommandManager tabs and select Customize CommandManager. Alternatively, you can choose Tools ⇨ Customize.

Note

To access the pull-down menus in a default setup, place the cursor over the SolidWorks logo or the small flyout triangle to the right of it in the upper-left corner of the SolidWorks window. Figure 2.2 shows the flyout for pull-down menus. To keep the menu in that position, click the pushpin on the right end of the flyout menu bar. ■

FIGURE 2.2

Customizing the CommandManager
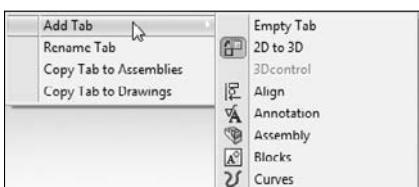
Customizing the CommandManager

Notice the last tab along the bottom of the CommandManager on the right. If you want to add a custom tab, you can right-click this tab and select the tab you want to add. You can also select to add a blank tab and populate it with individual buttons. Figure 2.3 shows a detail of the Add Tab menu options after you right-click the tab.

FIGURE 2.3

Adding tabs to the CommandManager
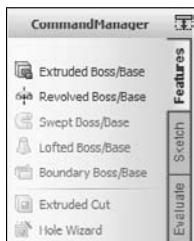
You can also left-click the special tab on the right and skip the Add Tab menu. To add individual buttons, first find the button you want to add in the Customize dialog box by choosing Tools ⇨ Customize. Click the Commands tab in the Customize dialog box, and then switch the CommandManager to the tab you want to add the button to, and drag the button from the Customize dialog box to the CommandManager. You can remove buttons from the Command-Manager by dragging them into the blank graphics window area.

Docking the CommandManager

In SolidWorks, you can undock the CommandManager and leave it undocked, pull it to a second monitor, or dock it vertically to the left or right. To undock it, click and drag on any non-toolbar button area of the CommandManager, such as around the border. To re-dock an undocked CommandManager or to change its docking location, drag it onto one of the docking stations around the screen. Figure 2.4 shows the CommandManager undocked.

FIGURE 2.4

The undocked CommandManager

Using Auto Collapse

The small box with the arrows in it in the upper right-hand corner of the undocked CommandManager and the undocked PropertyManager is the Auto Collapse option. Early versions of SolidWorks 2010 seem to have some inconsistencies in the way this option works with PropertyManagers. You may have to resize detached PropertyManagers manually for various features. When the option is active, the PropertyManager expands and collapses automatically, and should not require scroll arrows unless the PropertyManager is larger than your monitor's height. In the condition it is shown in Figure 2.4, the undocked CommandManager will not collapse, but if you click Auto Collapse, the arrows go away, and the entire CommandManager acts like a big fly-out toolbar. This can be very handy because it saves a lot of space on the screen, but at the same time, it requires additional mouse movement to open it up. This is the common trade-off in this interface —you can trade screen space for additional mouse movement or clicks.

Mixing CommandManager with toolbars

To put a toolbar inline with the CommandManager, drag the toolbar close to the CommandManager. A space on that row or column will open up. The amount of space that opens up depends on the tab with the longest set of icons, even if that tab is not showing. To increase the

amount of space available for a toolbar on the same row as a CommandManager, enter Customize CommandManager mode by right-clicking a tab and selecting Customize CommandManager. Then cycle through the available tabs, looking to see which one has the most icons. Remove icons from the tab with the most. This makes more room for toolbars to the right of the CommandManager.

Tabs and document types

SolidWorks remembers which tabs to show on a per document type basis. This means that when you are working on a part document, you will have one set of tabs. When you switch to an assembly document, you will see a different set of tabs. The same goes for drawings. Notice that in Figure 2.3, in the right mouse button (RMB) menu, the options Copy Tab to Assemblies and Copy Tab to Drawings appear. These options make it easier to set up customizations that apply for all document types.
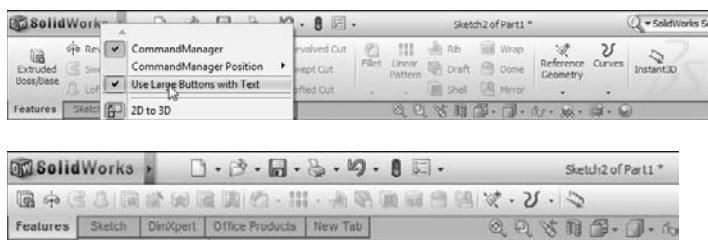
Changing the appearance of the CommandManager

For an element of the interface that is supposed to save space, the default settings for the CommandManager sure take up a lot of it! When many users see the CommandManager for the first time, they ask how to deselect the text.

You can deselect the text in one of two ways. The easiest way is to right-click in the CommandManager and deselect the Use Large Buttons with Text option, as shown in Figure 2.5.

FIGURE 2.5

Adding or removing text from the CommandManager buttons

Another way to remove text from the CommandManager is to remove it only from selected icons. To do this, first enter the customize mode by choosing Tools ⇨ Customize or right-click a CommandManager tab and select Customize CommandManager, and then right-click a button in the CommandManager and change the Show Text setting, as shown in Figure 2.6. The Show Text option is only available when Use large buttons with text is enabled.

Notice also that the text by default goes to the right side of the icon, but using the RMB menu, you can put the text beneath the icon. With these options and some patience to go through the entire interface, you can almost totally customize the appearance and function of your CommandManager.

Changing the text setting for individual buttons in the CommandManager
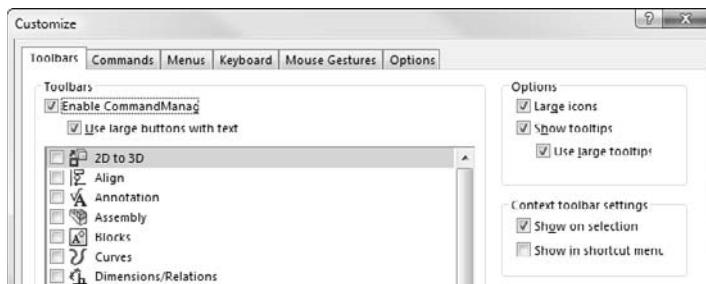
The most streamlined and space-efficient way to set up the CommandManager is to deselect the text. This arrangement is shown in Figure 2.5, in the lower image. Notice that the CommandManager without text takes up the same amount of height as a normal toolbar, with the added room for the tabs at the bottom. The text can be useful for new users or features that you do not commonly use. Also, notice that with the text turned off, you have room for more toolbar space at the end of the CommandManager.

The final setting for the CommandManager appearance is the size of the icons. You have control over the size of the icon images in the CommandManager. You can find this setting in the Customize dialog box (Tools ➪ Customize), and it is shown in the upper-right side of Figure 2.7. The difference between large and small icons is shown in the lower part of the figure.

Setting large icons

This setting applies to all the toolbar icons except the Menu Bar, RMB menu, and Context Bar icons. The setting does apply to the S shortcut toolbar and the Heads-up View toolbar, which I discuss later in this section. Large icons can be useful on displays with very high resolution, in particular on laptops where the screen itself may be small but the resolution is very high. All the screen shots in this book are taken with the Large Icons option selected for improved visibility. There is often a small difference between the appearance of the large and small icons.

Limitations

If you undock the CommandManager, you cannot reorient the tabs horizontally. They remain vertical. In addition, you cannot place multiple rows of toolbars on the same row as a Command-Manager using large buttons with text. In addition, you cannot dock the CommandManager to the bottom of the SolidWorks window. Another minor limitation is that although SolidWorks enables you to place toolbars at the right end of the CommandManager, it does not allow you to place them to the left of the CommandManager.

Using toolbars

The point of the CommandManager is to enable you have many toolbars available to you in a single click, with the main goal being space savings. SolidWorks is a complex program, with a sprawling interface. The CommandManager does a good job of making most of it available to you quickly without taking up a lot of space. Unfortunately, when you save one thing, you usually wind up giving up something else. Interface set up is frequently about compromise or balancing conflicting concerns. In the case of the CommandManager, the compromise is between mouse travels and clicks. You may find yourself clicking frequently back and forth between the Sketch and Features tabs. For this reason, in my interface setup, I put the Sketch toolbar vertically on the right, and remove it from the CommandManager. This enables me to see the Sketch and Features toolbars at the same time, and greatly reduces the times I have to click back and forth between the tabs.

You can enable and disable toolbars in several ways. To enable a toolbar, right-click in a toolbar area and you will be presented with a list of toolbars in SolidWorks. You can enable and disable this list from here. Another way to do this is to use the Customize dialog box by choosing Tools⇨Customize, or the Customize option near the bottom of the RMB toolbar list. Yet another way is to choose View⇨Toolbars.

Heads-up View toolbar

The Heads-up View toolbar appears along the middle of the top edge of the graphics window. Figure 2.8 shows the default arrangement of the Heads-up View toolbar, and it is shown in relation to the rest of the interface in Figure 2.1.

FIGURE 2.8

The Heads-up View toolbar

You can customize the Heads-up View toolbar by using the Toolbars dialog box (Tools ➪ Customize ➪ Toolbars).Customization includes turning the Heads-up View toolbar on or off and adding or removing buttons. This is new in SolidWorks 2010. Previously, you could not turn it off, and it had a special way of for adding and removing toolbar buttons. If you have multiple document windows or multiple view ports showing, the Heads-up View toolbar only shows in the active window or view port. This toolbar often overlaps with other interface elements when several windows are tiled or if the active window is not maximized. It can run into the PropertyManager if it is pulled out of the FeatureManager, as well as the ConfirmationCorner or the Task Pane.

Cross-Reference

Chapter 3 covers the Confirmation Corner in more detail. ∎

Menu Bar toolbar and menu

The Menu Bar toolbar is found just to the right of the SolidWorks logo on the title bar in the top left corner of the SolidWorks window. By default, it contains most of the elements of the Standard toolbar, and it is available even when no documents are open. It uses mostly flyout toolbar icons, so again it follows the trend of saving space at the expense of an extra click. This toolbar can be customized in the same way as normal toolbars in the Customize dialog box in the Commands tab. This toolbar cannot be turned off, but you can remove all the icons from it.
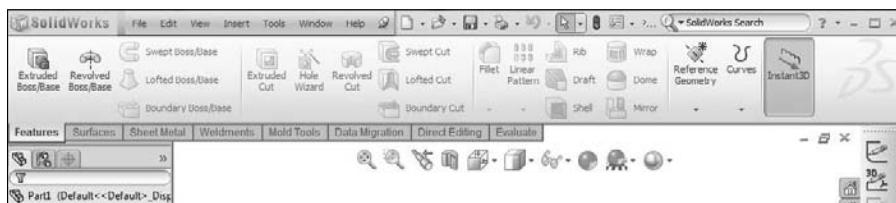
Note

You could run the SolidWorks interface from just the CommandManager without any additional toolbars; you could do the same with just the Menu Bar toolbar, customizing it with all flyout toolbars. The main advantages of this toolbar are that it is visible when no documents are open, and that it makes use of otherwise wasted space in the title bar. You might set up the interface for a 12-inch normal aspect display laptop very differently from that of a desktop unit with a 24-inch wide screen. ∎

There is also a Menu Bar Menu, which is hidden by default. The SolidWorks logo in the upper left of the SolidWorks window or the small triangle next to the logo serve as a flyout to expand the main SolidWorks menus. You can pin the menus in place using the pushpin shown at the right end of the menus in Figure 2.9. When the menu is pinned, the toolbar moves to the right to accommodate it.

FIGURE 2.9

The Menu Bar toolbar and menu

Note

The 3Dcontrol menu entry is not part of base SolidWorks; it is part of the 3Dconnexion (spaceball) driver. ∎

Notice that on low resolution or non-maximized SolidWorks windows, you can run into some space problems if the Menu Bar Menu is pinned open. The SolidWorks window in Figure 2.9 is the SolidWorks 2010 interface with all default settings displayed at 1024 × 768 resolution, which is a common resolution when using digital LCD projectors or small notebook computers. You need to examine the changes in the SolidWorks interface with display size in mind. You might consider having different sets of settings for using a laptop at a docking station with a large monitor, using the laptop with a small monitor, or using the computer with a low-resolution digital projector.

Flyout toolbar buttons

SolidWorks has implemented flyout toolbars that save space by putting several related icons on a flyout. For example, the Rectangle tool has several different ways to make a rectangle, with a tool for each, and they are all on the rectangle flyout.

You can see all the available flyouts in the Customize menu (Tools ➪ Customize), under the Commands tab, in the first listing in the window, Flyouts.

The purpose for flyouts is primarily to save toolbar space when several tools are closely related. SolidWorks has set up flyouts in two configurations: fly-outs that always maintain the same image for the front button (such as the Smart Dimension fly-out) and fly-outs that use the last used button.

Toolbar flyouts are listed in Tools ➪ Customize ➪ Commands and are listed from 2D to 3D through the Weldments toolbar. After Weldments in the list, the fly-outs are Similar Function fly-outs. You can change the order of the items in the flyouts by changing the order of the items in the toolbars. Just display the original toolbar and choose Tools ➪ Customize to reorder it to your liking. These toolbars will always have the same icon on the top. For example, if you use the Reference Geometry flyout to access the Axis command, the image for the Plane icon will remain on top. The image on top is considered the most commonly used function of that group of tools, and so remains on top.

Add-in flyouts, such as the FeatureWorks flyout, are controlled by that specific add-in and again keep the same icon always on top.

The flyouts used for tools of similar function are split between using the most recently used tool icon on top and keeping a consistent icon on top. The only tools that appear to follow the latest icon method are the Sketch Entities tools. Sketch tools and other flyouts use a hard-coded top image.

Context toolbars
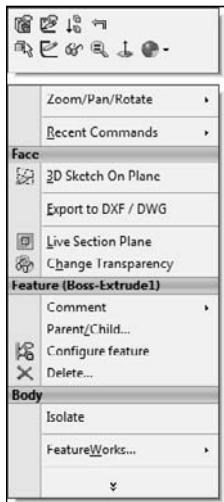
Context toolbars are toolbars that appear in the graphics window when you right-click or left-click something. When you right-click, a context toolbar appears at the top of the RMB menu, and shows the functions that SolidWorks deems the most commonly used functions. This is a static list, and does not change as you use the buttons. These functions are removed from the RMB menu and replaced with the Toolbar icon in a toolbar above the abbreviated RMB menu, as Figure 2.10 shows.

The right-click context toolbar

An identical toolbar appears when you click an item on screen. When this toolbar appears with a left-click, the rest of the RMB menu does not appear. Tool tips are available if you do not recognize the icons on the toolbar.

For reference, the icons in the context toolbar atop the RMB menu (the icons without text) shown in Figure 2.10 are in order from the upper-left: Edit Feature, Edit Sketch, Suppress, Rollback, Select Other, Sketch, Hide Body, Zoom To Selection, Normal To, and Appearance Callout. Notice that these selections do not reappear in the main RMB menu.

These context toolbars are not editable, but you can turn them off, and put the RMB menus back to a familiar configuration. To deselect, click the context toolbar and choose Tools ➪ Customize. Use the options on the right side of the main Toolbars tab, as shown in Figure 2.11.
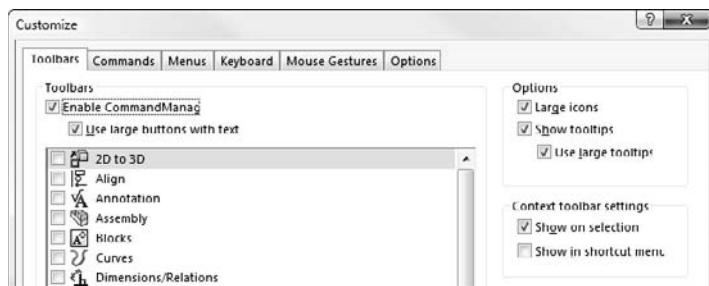
I find that the RMB context toolbars are distracting because they force you to read a two-dimensional list of icons and a one-dimensional list of text. To me, this is just too confusing. I turn these off so that the RMB menus look like they always did.

However, I do find left-click context toolbars useful for things like Hide Sketch, Edit Feature, Edit Sketch, Appearance Callout, sketch relations, and so on. When I use the RMB menus, I'm looking for a more general function. When using the left-click context bars, I'm looking for something specific that I know is there. In Figure 2.11, the Show on selection check box option simply refers to the left click toolbar, and Show in shortcut menu check box option refers to the RMB menu.

FIGURE 2.11

Context toolbar settings
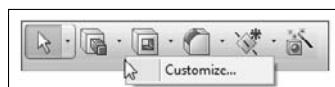
The purpose of the context toolbars is to save space by condensing some commands into a toolbar without text instead of a menu with icons and text. The left- and right-click toolbars are the same, but they work differently. The left-click context toolbar fades as you move the cursor away from it and becomes darker as you move the cursor toward it. Once it fades past a certain point, you cannot get it back, except if you have Ctrl+selected multiple entities. The context toolbar does not appear until you release the Ctrl key. To get a context menu to show up again after it has faded, you can just move the cursor back to approximately where the toolbar would have been and press Ctrl again. This works only for multiple selection menus where Ctrl was used to multi-select. The functionality is probably a bug, or unintentional in any case, or else it would also work somehow for single selections.

Shortcut "S" toolbar

The Shortcut toolbar is also known as the "S" toolbar because by default you access it by pressing the S key. You can customize this toolbar for each document type and another for sketches, so it can have different content for sketches, parts, assemblies, and drawings. To customize the "S" toolbar, right-click it when it is active and click Customize from the RMB menu, as shown in Figure 2.12.

FIGURE 2.12

Right-click on the Shortcut "S" toolbar to customize it

Many people claim to have customized the "S" toolbar to such an extent that they have been able to remove the CommandManager and all other toolbars from their interface. This is possibly true if you use a limited number of sketch entities, sketch relations, and feature types, or make extensive use of flyouts on the "S" toolbar. However, if you work with a wide range of tools (say, surfacing,

sheet metal, and plastic parts), you may need some additional toolbar space. It is completely believable to have access to most of the software's function with the "S" toolbar and either the Menu Bar toolbar or the CommandManager. CommandManager by far gives you the most flexibility, but it also requires the most space.

The S key shortcut may conflict with another keyboard customization you have done, depending on how your software was installed. To change the "S" toolbar key to another character or to reassign it, follow the directions for creating and maintaining hotkeys later in this chapter in the section on customization. It is referenced as the Shortcut Bar in the Keyboard list (Tools⇨Customize⇨Keyboard).

Tool tips

Tool tips come in two varieties: large and small. Large tool tips show the name of the tool, along with a brief description of what it does. Small tool tips show only the tool's name. To change the tool tip display from large to small, or to deselect the tool tip display altogether, choose Tools⇨Customize. The options for using large tool tips and showing tool tips appear in the upper-right corner, as shown in Figure 2.11. In addition to the tool tip balloons, tips also appear in the status bar at the bottom of the screen when the cursor is over an icon. Figure 2.13 shows a comparison between large and small tool tips.

FIGURE 2.13

SolidWorks uses large tool tips by default.

Note

The Customize option (Tools⇨Customize) is inactive unless a SolidWorks document is open. To access the Customize dialog box, first open a SolidWorks part, assembly, or drawing and then choose Tools⇨Customize from the menu. Customize is different from the Customize Menu option found in all SolidWorks menus. The Customize Menu option is discussed later in this chapter. ∎

Managing toolbars

After all that, if you still feel you need to work with standard toolbars, it is easy to move, select and deselect, and add icons to toolbars. It is important to remember that different document types retain different toolbar settings; for example, the toolbars that you see with a part open are different from the toolbars that you see for drawings.

When you are working on parts, it is important to have both the Sketch and the Features toolbars active. When you are working on a drawing, you will never use the Features toolbar, but you will

frequently use the Sketch toolbar. Likewise, for assemblies, you may want to display some additional toolbars and eliminate others. For this reason, when you change from a part document to a drawing document, you may see your display adjust because the changing toolbars increase or decrease the amount of space that is required.

Best Practice

It is best practice to set up the toolbars for each document type so that they take up the same amount of space — for example, two rows on top and one column to the right. This way, changing between document types is not so jarring, and the graphics area does not need to resize for each change. ■

Moving toolbars

To move a toolbar, you can click with the cursor at the dotted bar on the left end of the toolbar, as shown in Figure 2.14. The cursor changes to a four-way arrow and you can then drag the toolbar where you want it. Toolbars dock either vertically or horizontally. You can resize undocked toolbars so that they have rows and columns. This arrangement is typically used with the Selection Filter toolbar, which is often left undocked and compressed into a block that is three or four columns wide.

FIGURE 2.14

Dotted bars enable you to move toolbars.

If the SolidWorks window is not wide enough for the toolbar to fit entirely in the screen, double arrows like those shown in Figure 2.15 appear at the end of the truncated toolbar. When you click the double arrows, a flyout toolbar appears with the missing icons, as shown in Figure 2.16.

FIGURE 2.15

A truncated toolbar showing double arrows

FIGURE 2.16

You can display all of a truncated toolbar by clicking the double arrows.
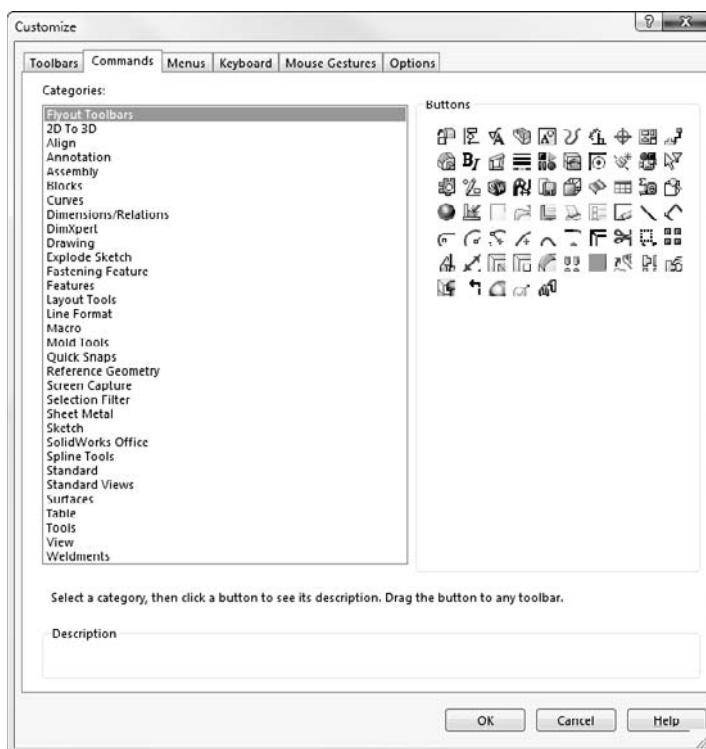
Flyout toolbars

You can use any toolbar as a flyout toolbar. Figure 2.17 shows the list of all flyout toolbars, which is the same as the list of *all* toolbars. Flyout toolbars are a nice space-saving feature for tools that you use infrequently, but frequently enough to want to avoid going through the menus. To use a toolbar as a flyout, select it from the Flyout toolbars list and drag it onto an existing toolbar. It displays with an arrow to the right. Clicking the arrow causes all the tools to scroll out temporarily until you click a toolbar icon or anything else.

FIGURE 2.17

The Flyout toolbars are on the Commands tab in the Customize dialog box.

To add icons to a flyout toolbar, temporarily show the regular toolbar that corresponds to the flyout toolbar and add icons to the regular toolbar. When you are done adding or removing icons, deselect the regular toolbar; the changes are applied to the flyout.

Tip

If you want to create a separate toolbar, you can commandeer an existing one for your own purposes. For example, because I do not use the Tools toolbar, I have removed all the regular icons from it and replaced them with relevant flyout toolbars, which I do use extensively. This enables me to consolidate space, and not have unused icons on my toolbars. Alternatively, creating a custom CommandManager tab and putting on it what you like is much easier. ■

Full Screen mode
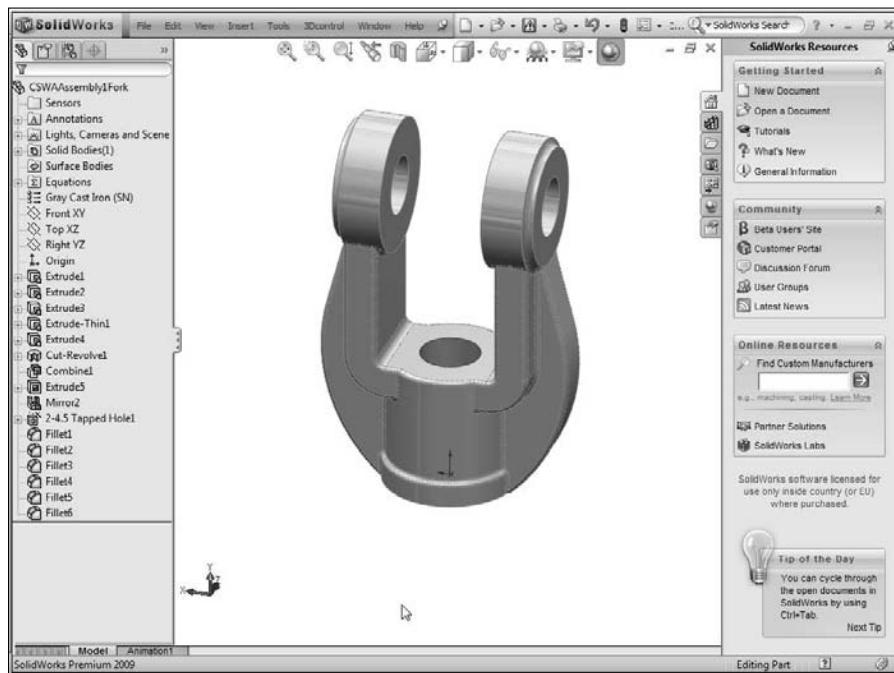
Full Screen mode enables you to toggle quickly to the display so that only the graphics window and the Task pane appear; the FeatureManager, menus, toolbars, and status bar are all hidden. Alternatively, you can hide just the FeatureManager or the toolbars.

In Full Screen mode, you can still access the menus by clicking the cursor along the top border of the window, as shown in Figure 2.18.

- To toggle to Full Screen mode, press the F11 key.
- To toggle the toolbar display, press the F10 key (see Figure 2.18).
- To toggle the FeatureManager display, press the F9 key.

FIGURE 2.18

The SolidWorks window with all toolbars hidden using the F10 key

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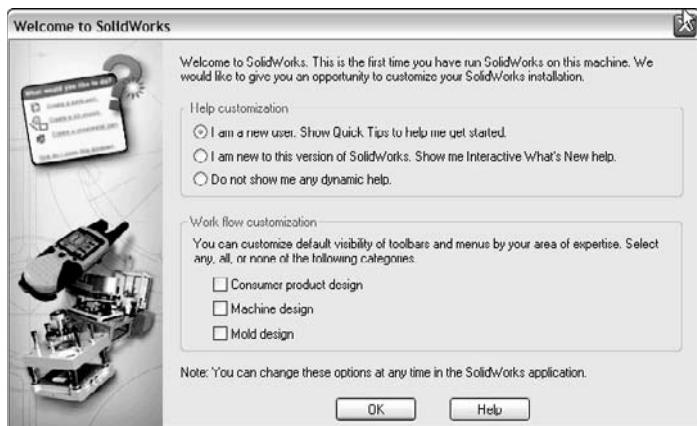
Workflow customization

When you first install and run the SolidWorks software, the SolidWorks Welcome screen shown in Figure 2.19 offers you the option to customize the interface using one of three preset option packages. Special menu and toolbar settings are made for Consumer product design, Machine design, or Mold design. After the software is initially installed, you only see this screen once, but you can change all the options in other places, including the Options tab in the Customize dialog box.

The Welcome to SolidWorks screen

The three-workflow customizations affect the interface as follows:

- Consumer product design adds the Surfaces toolbar to the CommandManager.

- Machine design adds Sheet Metal and Weldments toolbars to the CommandManager.

- Mold design adds Surfaces and Mold Tools toolbars to the CommandManager.
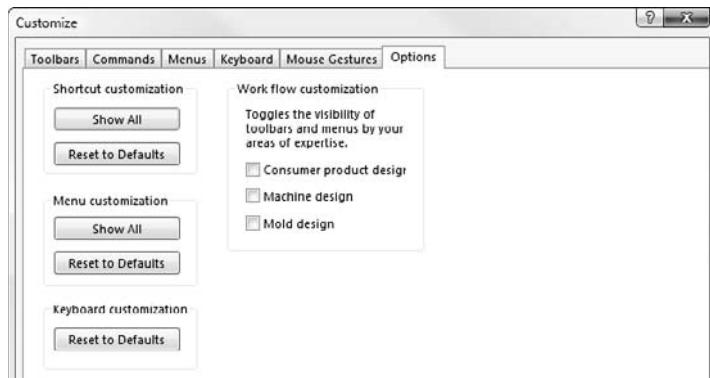
Similar changes are made to the menus to hide or show menu selections as appropriate. You can find more information about hiding and showing menu items later in this chapter.

If you want to select a different option after the initial setup, choose Tools➪Customize and click the Options tab, where you can specify a different choice. Figure 2.20 shows the Options tab of the Customize dialog box.

The Options tab of the Customize dialog box, where you can select a different workflow customization

Controlling menus

Everyone has his own style of working. For example, some people like to use menus and others do not. Some like to use hotkeys and others like the mouse. Modify Section View is an example of a tool that you cannot access via toolbars. It can only be accessed via the menus.

The most frequently used menu items are in the View, Insert, and Tools menus. All the menus shown in this section have all the possible options selected. As a result, the View menu in Figure 2.21 may contain options that are not available on your computer. Customizing menus is covered later in this chapter. Figure 2.21 also shows the Insert and Tools menus, along with an image of a menu with the Customize Menu mode activated.

You use the View menu primarily for turning on or off the visibility of entity types such as planes, sketches, or temporary axes. You can also do this by using hotkeys or by putting extra items on the View toolbar.
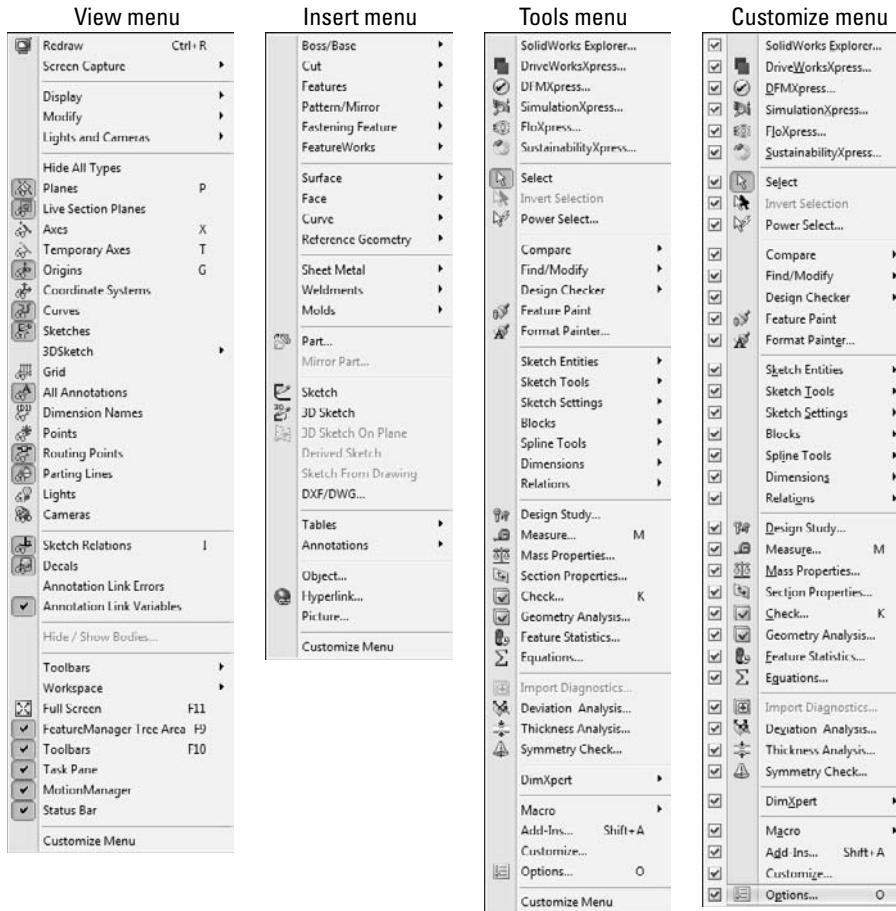
The Insert menu is used mostly for creating feature types for which you do not have a toolbar icon on the screen. For example, although the Move Face tool is on the Mold Tools toolbar, it has many uses aside from mold design. You can find the Move Face tool by choosing Insert ⇨ Face.

The Tools menu is used primarily for sketch entities or tools for which you have no icon on the screen. Several other commonly used tools, such as Measure, Equations, Customize, and Options, are also available in this menu.

FIGURE 2.21

Popular menus

View menu	Insert menu	Tools menu	Customize menu

You can customize menus by adding or omitting items. By using the Customize Menu option at the bottom of any menu — including shortcut (RMB) menus — you can remove items from any menu by clearing the check boxes next to tools that you do not use. To bring back the removed items, you can either go back to the Customize Menu or choose Tools ➪ Customize ➪ Options and click the Reset to Defaults buttons for menu and shortcut customization.

Interpreting SolidWorks use of the word "shortcut"

Between the SolidWorks and Microsoft interfaces, the word *shortcut* is used in several overlapping and confusing ways. Users replace most of the SolidWorks occurrences with words they use every day. The following list describes where SolidWorks and Microsoft users might encounter the word *shortcut* as a formal name for interface functionality, and how they might translate it.

- The Windows **Shortcut** link to another file or folder. Most users still refer to this link as a *shortcut* or *desktop shortcut*.

- **Shortcuts** (as identified in the Help Index under Shortcut⇨Keys) are either accelerator keys or keyboard shortcuts. Users refer to accelerator keys as *Alt-keys*, and to keyboard shortcuts as *hotkeys*.

- SolidWorks **shortcut menus** are commonly called the right mouse button (RMB) menus, and have detached toolbars called context bars for both right- and left-click options. These are commonly known as the *RMB bar* and the *LMB bar*.

- **Shortcut Tabs** (found as the "shortcut" entry in the SolidWorks Help Index). This presumably refers to DriveWorksXpress functionality, although there is no direct mention of that in the Help entry. DriveWorksXpress is beyond the scope of this book.

- **Shortcut Bars** are commonly known as the *S key toolbar*.

If you use the alternate terminology offered here, it will be clear to all users what you are talking about.

Note

Be careful not to confuse this Customize Menu selection with the Customize menu selection on the Tools menu. Figure 2.21 shows the Tools menu being customized. In addition, I do not recommend removing items from the menus. It doesn't take much for someone to need one of those items and no one remembers that it was supposed to be there or how to get it back.

The Options dialog box (Tools⇨Customize⇨Options), shown in Figure 2.20, contains the Shortcut (RMB) menu and Menu customization options. These options enable you to show all the menu items for both types of menus in a single stroke. By default, some items are hidden in various menus. Keyboard customization is discussed later in this chapter. Keyboard shortcuts are generally referred to as *hotkeys*.

Changing cursors

SolidWorks cursors are context sensitive, and change their appearance and function depending on the situation. Sketching cursors display a pencil and the type of sketch entity that you are presently sketching. Sketch cursors also display some dimensional information about the entity that you are sketching, such as its length or radius. Sketch cursor feedback is necessary for fast and accurate sketching.
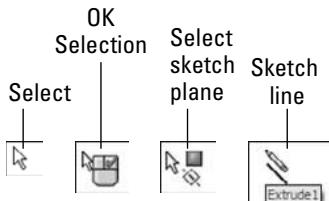
Cross-Reference

To learn more about sketch cursor feedback, see Chapter 3. ■

The Select cursor changes depending on the item over which you have positioned it. Cursor symbols also help remind you when selection filters are active. The cursor is frequently available as an OK button. For example, after selecting edges for a Fillet feature, the RMB functions as an OK button. Figure 2.22 shows various cursors and their significance.

FIGURE 2.22

Various SolidWorks cursors

Working with models in the FeatureManager and PropertyManager windows

The FeatureManager window is the panel to the left of the screen, which shows an ordered list of features describing how the part was built. SolidWorks users spend a fair amount of time using the FeatureManager to edit or inspect models. Figure 2.23 shows the FeatureManager for a simple model.

Using the FeatureManager

There is a splitter bar at the top of the FeatureManager that enables you to split the FeatureManager window into two windows, enabling you to display the FeatureManager and another window, such as the PropertyManager (see Figure 2.24). Small arrows in the middle of the right separator can collapse the FeatureManager to increase screen space. The F9 key also collapses or opens the FeatureManager (refer to Figure 2.23.)

Display pane

You can open the Display pane flyout from the FeatureManager by using the double arrows at the top-right corner of the FeatureManager. The Display pane helps you to visualize where appearances, display styles, or hidden bodies have been applied in a part document and additional functions in an assembly document. The display pane is helpful when you're looking for colors that are applied to the model at some level other than the part level.

Cross-Reference

Appearances are covered in more detail in Chapter 5. ∎

Rollback bar
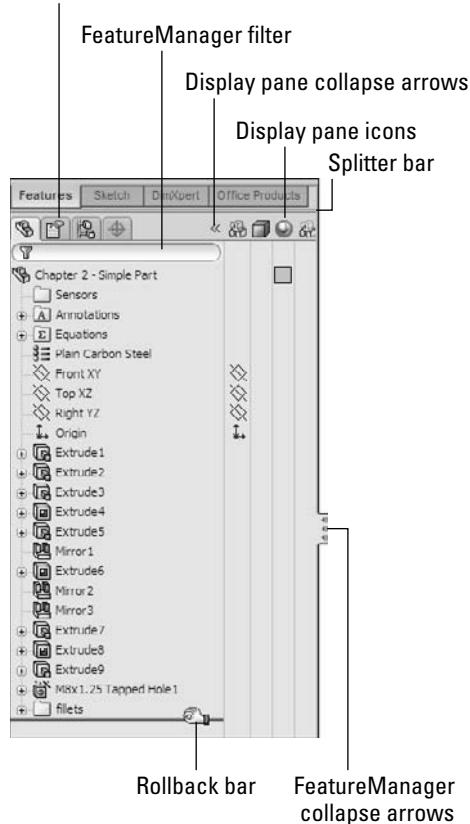
The Rollback bar at the bottom of the FeatureManager enables you to see the part in various states of history. Features can be added while the rollback bar is at any location. The model can also be saved while rolled back.

FIGURE 2.23

The FeatureManager for a simple model

PropertyManager tab

FeatureManager filter

Display pane collapse arrows

Display pane icons

Splitter bar

Rollback bar   FeatureManager
                      collapse arrows

FeatureManager filter

One of the most useful elements of the FeatureManager is the FeatureManager filter. The filter resides at the top of the FeatureManager. If you type text in the filter, SolidWorks searches feature names, descriptions, comments, tags, and dimension names for text matching the string, and only shows matching features in the window. This also works in assemblies, where you can filter for part names or document properties. The filter is very useful for quickly finding parts, features, mates, or anything else that shows up in the part or assembly FeatureManager.

Using the PropertyManager

The PropertyManager is where you go to set most of the feature parameters, and where you edit the properties of selected items such as sketch elements. You can manually switch to the Property-Manager using the tabs on the top of the Display panel, or allow it to pop up automatically when your input is needed. The leftmost tab in the row of icons is the FeatureManager tab, the second from the left is the PropertyManager tab, the second from the right is the ConfigurationManager tab, and the rightmost tab is the ToolAnalyst (available only in SolidWorks Premium). Other icons may also appear in this area for drawings, or if you have add-ins such as PhotoWorks or SolidWorks Simulation (formerly COSMOS) turned on. The ConfigurationManager tab appears with more detail in Chapter 10.
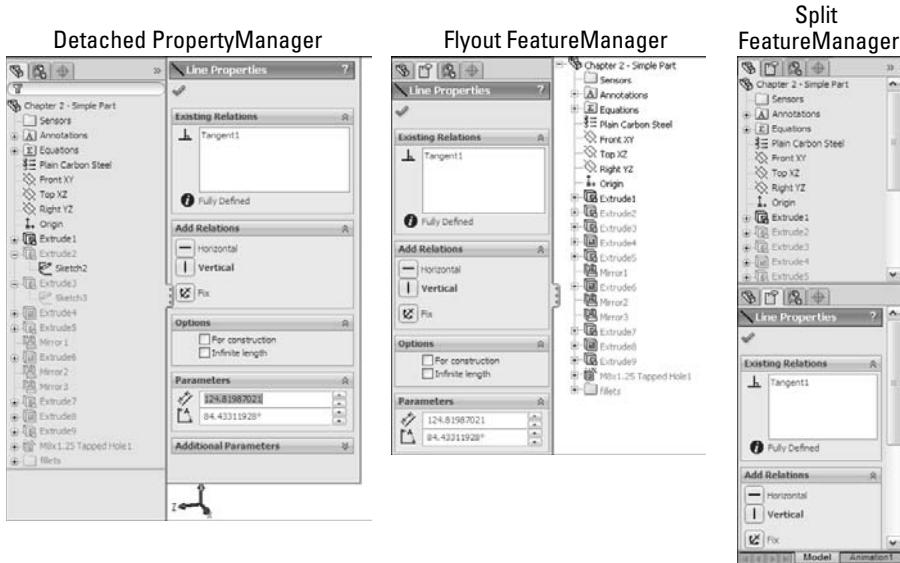
One of the benefits of putting dialog boxes in the PropertyManager is that it saves a lot of space on the screen. On the other hand, you will often need to make a selection from the FeatureManager at the same time that the PropertyManager pops up and takes its place. You can disable this automatic pop-up behavior by choosing Tools ➪ Options ➪ System Options ➪ General and selecting the Auto-show PropertyManager setting.

My favorite option for dealing with the PropertyManager is to detach it from the FeatureManager so that you can see the two side by side instead of one or the other. To detach the PropertyManager, drag its icon from the tabs out into the graphics area and release. Once the PropertyManager is detached, you can move it to a second monitor, float it within the SolidWorks window, or dock or reattach it. To put it back in its place under the FeatureManager, just drag it back on top of the FeatureManager using one of the docking station symbols on the screen, allow it to snap into place, and release it.

If you do not like the detachable PropertyManager, you can use the splitter bars either to put the FeatureManager on top and the PropertyManager beneath, or use the flyout FeatureManager. When creating or editing a feature, you can access the flyout FeatureManager by double-clicking the name of the feature at the top of the PropertyManager. The flyout FeatureManager is displayed just to the right of the regular FeatureManager, in the main graphics window, and is transparent to enable you to see the model through it. The various ways of combining the FeatureManager and PropertyManager are shown in Figure 2.24.

The detached PropertyManager, the split FeatureManager, and the flyout FeatureManager.

Getting around the Task pane

By default, the Task pane sits to the right of the SolidWorks screen, although you can undock it entirely. If you want to keep it open, click the pushpin in the upper-right corner of the pane. The Task pane is shown in Figure 2.25.
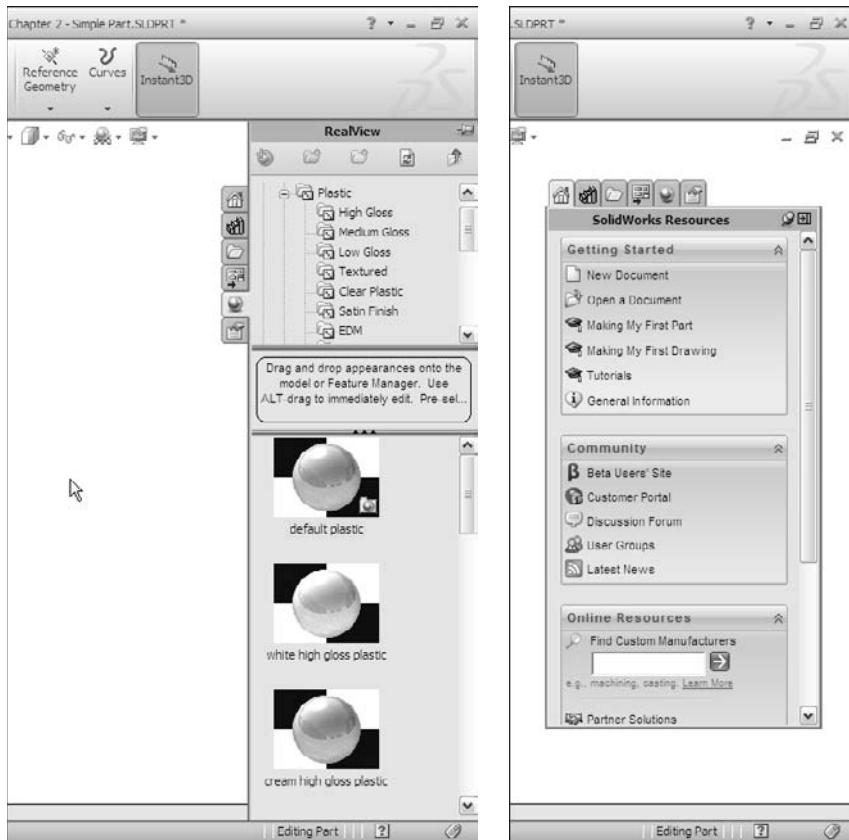
The Task pane is the home for several panels:

- **SolidWorks Resources.** These are useful links for templates, tutorials, tech support, news, GlobalSpec search, Tip-of-the-Day, and other resources.

- **Design Library.** This includes locally stored libraries, Toolbox, and 3D Content Central. This tab also contains "SolidWorks Content," which consists of additional library resources that can be downloaded directly from the Task Pane.

- **File Explorer.** This is a Windows Explorer interface that you can use to browse for files.

- **SolidWorks Search.** If you have installed the Windows Desktop Search with SolidWorks 2010 and indexed your files, you can perform searches that include filename and custom properties.

FIGURE 2.25

The Task pane

- **View Palette.** This palette enables you to visually select views and drag them onto a drawing sheet.

- **RealView.** This panel enables you to select appearances and scenes for your SolidWorks documents.

- **Custom Properties.** The custom property tab in Task pane enables you to create a custom interface that goes inside this Task Pane tab that will help you enter custom property data quickly, easily, and accurately.

- **Recovered Documents.** After a crash, auto recovered documents are listed in this special purpose Task Pane tab.

Getting familiar with the Status bar

The status bar is a non-intrusive way in which SolidWorks communicates information back to the user. It is located at the bottom of the screen, and you can enable it from the View menu. Figure 2.26 shows the status bar in action.

The status bar showing a tool tip for the Sketch Circle tool

Sketches a circle. Select the center of the circle, then drag to set its radius.   -1.998in   0.364in   0in Under Defined   Editing Sketch2   ⬚   ❔

The status bar can display the following information, indicators, and icons:

- Progress as parts, assemblies, or drawings load
- Tool tips for commands
- Measurements
- The sketch status for an active sketch
- In-context editing
- Suspend automatic rebuilds
- Icons that enable you to turn Quick Tips off or on
- The sheet scale for drawings
- The cursor position for drawings and sketches
- Whether you are editing the sheet, sheet format, or view of a drawing

Assigning tags

Tags work like document properties, except that they do not need a property name; they just use a value. A tag could be considered simply a keyword that you can associate with a part in an assembly or even a feature in a part. Tags can be searched by SolidWorks Explorer or by the FeatureManager Filter. You can assign tags by clicking the yellow tag icon on the status bar in the lower-right corner of the SolidWorks window. Figure 2.27 shows a tag being added to a feature.

Making use of Quick Tips
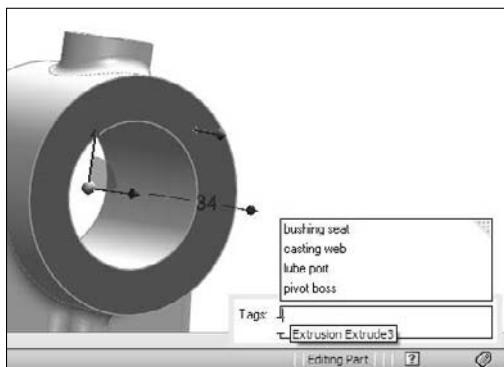
Quick Tips appear in the pop-up window in the lower-right corner of the graphics window. They can change as you work so that they are sensitive to the context in which you are working. They are a great way for new users or infrequent users to learn or be reminded of the next steps available to them. You can activate and deactivate Quick Tips by clicking the question mark icon in the lower-right corner of the SolidWorks window on the status bar. Figure 2.28 shows the Quick Tips window in action.
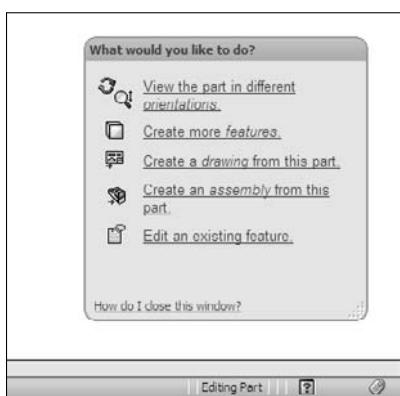
Adding a tag to a feature

Quick Tips in action
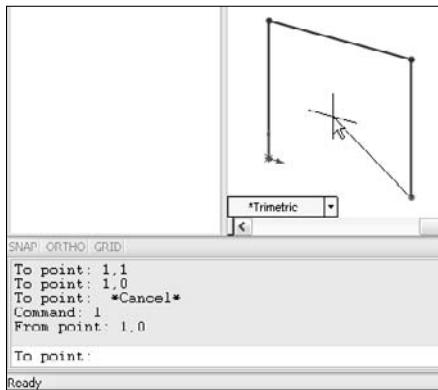
Using the 2D Command Line Emulator

This is a tool specifically for people who are coming to SolidWorks from AutoCAD. As the name suggests, it adds a command line to the bottom of the SolidWorks window that works like the AutoCAD command line in most respects. The available commands are somewhat limited compared to those that are available in AutoCAD. This tool only functions in the 2D sketch mode, on a drawing sheet, or in a drawing view; it does not work in a 3D sketch. The 2D Command Line Emulator is shown in Figure 2.29.

FIGURE 2.29

The 2D Command Line Emulator in action

Available sketch tools in the 2D Command Line Emulator include Align, Arc, Array, 'Cal, Chamfer, Chprop, Circle, 'Color, Copy, DDcolor, Dim, Dist, Ellipse, Erase, Exit, Extrude, Fillet, 'Grid, Line, List, Massprop, Mirror, Move, Offset, 'Ortho, 'Osnap, 'Pan, Plot, Point, Polygon, Qsave, Rectangle, 'Redraw, 'Redrawall, Revolve, Rotate, Save, Saveas, 'Snap, Spline, Trim, U (undo), 'Units, 'View, and 'Zoom. You can use commands preceded by an apostrophe (') as transparent commands without exiting an active command. Notice that even the cursor changes to crosshairs.

Best Practice

The best way to learn a new software package is to embrace the new way, not to cling to the old way. Although AutoCAD users may find the 2D Command Line Emulator more comfortable to work with, you will not achieve the same results, as you will with the SolidWorks default-sketching mode. For example, the resulting sketch entities created using the 2D Command Line Emulator are not constrained in any way, and the endpoints do not even merge. You can deselect the 2D Command Line Emulator by going to Tools ⇨ Add-ins. ∎

Note

The 2D Command Line Emulator is not available on 64-bit computers. ∎

Making the Interface Work for You

As engineers and designers, we all like to tinker with things to optimize efficiency and to apply our personal stamp. When the SolidWorks software is installed, the interface is functional, but not optimal. In the previous pages, I have discussed managing and customizing toolbars and menus. In the remainder of this chapter, I focus more on customizing the interface, and suggest some strategies that you might use to help customize your work environment.

Customizing colors
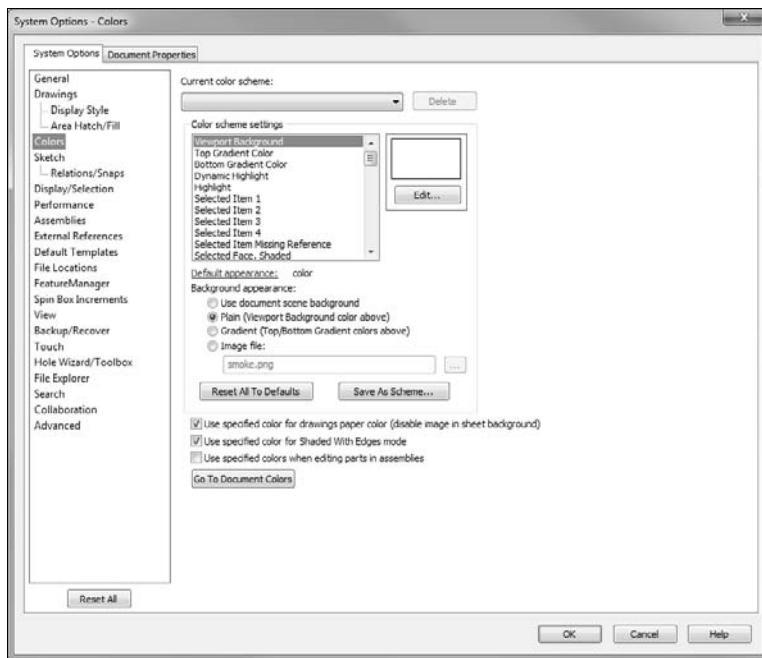
You need to be aware of a few things before you change the standard colors in the SolidWorks interface. The first is that SolidWorks does not automatically alter text color to contrast with your background. As a result, if you set the background to black, and the text is black, you won't be able to see the text. This may seem obvious to some people, but AutoCAD automatically changes text color to contrast with the viewport background, and so AutoCAD users may take this functionality for granted.

Exploring default selection colors

All the interface colors are controlled in the Systems Options Colors dialog box. You can access the dialog box by choosing Tools ➪ Options ➪ Colors. The selection color in particular is set at Selected Item 1, as shown in Figure 2.30.

Changing interface colors

Notice that you can set a color scheme. I recommend that if you want to change the colors used in the interface, you save the settings as a color scheme so that the scheme can be re-created easily later or handed off to another computer. Color schemes are stored in the Windows registry, not as separate files. To transfer color settings to another computer, you will need to either use the Copy Settings Wizard or manually copy data from the Windows registry.

Before making changes, you might consider saving your initial settings as a separate scheme so you can get back to them if you need to.

Caution

Making changes to the Windows registry can adversely affect software installation and hardware performance. You should not attempt changes to the registry unless you know exactly what you are doing. ■

Selecting Background options wisely

You should avoid some colors for the background, or you should make some changes if you choose these colors. Black is used with fully defined sketches, dimensions, FeatureManager text, and annotations. Blue can mask the underdefined sketch color. Bright green (or blue, depending on the version you are using) can cause problems with seeing selected items. Bright red, aside from being a terrible color to stare at all day, also does not contrast well with some of the red highlights and error colors.

You might think that no matter which color background you select, aside from the default, the items or features are difficult to see. For this reason, many users choose a gradient background, which enables them to pick colors where items are always visible on one-half of the screen or the other. Staring at a white screen all day can be uncomfortable for your eyes, so pick colors that enable you to see everything with "reasonable" contrast, yet are not glaringly bright. Very high contrast is hard on the eyes, and low contrast may make it difficult to distinguish items on the screen.

You have to consider what the purpose of the background is. Some people doing presentations may want the background to be attractive while otherwise staying out of the way. Others may only need the background to contrast with whatever is in front of it in a way that does not strain anyone's eyes. For writing a book, the background generally needs to be white to match the page. No one scheme will suit all needs.

In addition to colors and gradients, you can use an image as the graphics window background. This gives you a wider range of customization capabilities, and several sample images are already available in the default settings. Also, be aware that document scene backgrounds are document specific and override system options.

RealView also adds some capabilities with *scenes*. Scenes can be applied from the RealView tab on the Task Pane. RealView offers three different types of scenes: Basic, Studio, and Presentation. Of these, I find the Studio scenes to be the best when I need something with a reflective floor with shadows, otherwise I stick with the Basic scenes. I describe RealView, along with scenes, in more detail in Chapter 5.

Customizing strategies

You can easily customize many aspects of the SolidWorks interface, including:

- Toolbars
- Menus
- Background colors or images

- PropertyManager skins
- Task pane location
- Hotkeys
- Macros
- Custom application programming

Whether or not you should customize each of the previous items depends partially on how much time and energy you have to spend, as well as how much money you are ready to dedicate in the case of custom programming.

Considering hotkey approaches

Some of us old-timers prefer to use the keyboard instead of the mouse. If your hand-eye coordination is as bad as mine is, you may also choose this approach. I can type without looking at the keyboard, but when I use the mouse, it takes me a few seconds to aim at an icon and hit it accurately. This means that I customize SolidWorks to use as many hotkeys as possible, and remove icons from the interface if I have them on hotkeys. Unfortunately, my memory is as bad as my eyesight, and so remembering 75 hotkey commands is a bit of a problem. I admit to having a printed list of hotkeys taped to the side of my monitor. While I know that needing to read the list to find a particular hotkey defeats most of the purpose of using them in the first place, I just accept it as a learning aid. This is a self-solving problem, because the hotkeys that I use the most are the ones that I learn most quickly.

I generally do not advocate trying to standardize a hotkey scheme across multiple users, unless the users all agree to it. The underlying reason for writing this section is that everyone remembers things differently in the first place.

Any command that I use more than a few times an hour is worth assigning to a hotkey. I like to use alliteration when assigning keys to help with my faulty memory. The most frequently used commands are assigned single-letter hotkeys, and the less frequently used commands are assigned combinations. Thus, Tools Options is linked to O, Measure to M, Select Vertex to Shift+V, and Curve Projected to Ctrl+J (Ctrl+P is the Windows standard for the Print command). Other people like to group keys into easy-to-reach combinations; this is why the Q, W, A, S, Z, and X keys are often assigned first for right-handed mouse users.

Organizing hotkeys
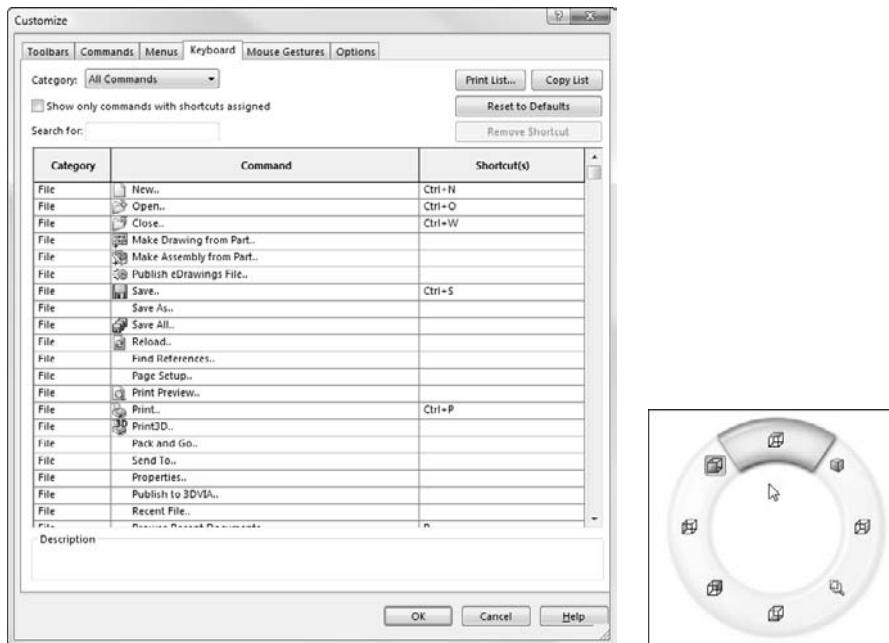
Hotkeys are assigned and organized in the Keyboard dialog box (Tools ➪ Customize ➪ Keyboard), as shown in Figure 2.31. This interface enables you to see all the hotkeys (called *shortcuts* in the list) easily. If you try to enter an existing hotkey, SolidWorks issues a prompt, telling you that the key is assigned to another command and its name. The prompt asks you whether you want to clear the other instance of the hotkey and make the new one active. You can also print out or copy to the Clipboard a list of only commands that use hotkeys.

Because the list of commands is so long, a Search function is available, and a drop-down arrow makes only the commands from a selected menu visible. The list of commands is organized by menu name, and the menus are listed as they occur in the interface. Fortunately, on the Keyboard

tab, SolidWorks enables you to sort, using the column headers to list the menus, commands, or hotkeys in alphabetical order, simply by clicking the column header. This is a highly usable interface, one of my favorite interface changes in the last several releases.

FIGURE 2.31

The Keyboard dialog box (Tools ⇨ Customize ⇨ Keyboard) and the mouse gesture donut

Using mouse gestures

Starting in SolidWorks 2010, you can now customize and use a mouse gesture interface, shown in the second image of Figure 2.31. To make the interface appear, just drag the RMB slightly, about 14 inch. Once you get used to the interface, a drag of about 34 inch in a single motion will activate the commands.

Note

I have found that this interface works best when you have memorized the commands available at various positions around the donut. ∎

You can establish the donut in four or eight segments; it comes set to four by default. You can also do the customization in the Customize dialog box (Tools ⇨ Customize) using the Mouse Gestures tab. The advantage of this interface is that it is very easy to invoke. I like the way you can use the default setup to control views. The mouse moving in a particular direction is easily associated with a view direction, so should be easy to remember.

Mouse gestures will probably not replace hotkeys or the "S" toolbar, but they do add effective quick access for a few functions.

Using the keyboard
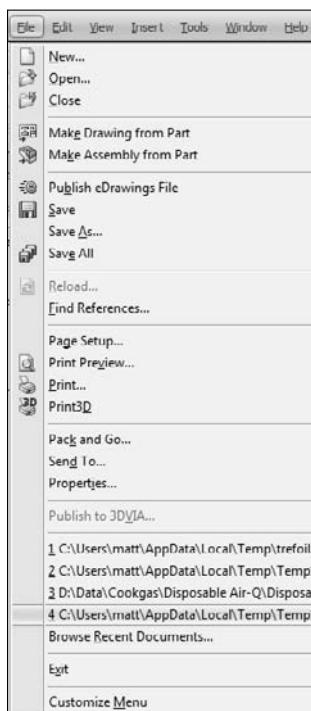
Moving between the mouse and the keyboard can be bothersome and time-consuming. In addition to the hotkey approach, you can use another keyboard method to save time. Many users become adept at using the Alt-key combinations to invoke menu items. Most menu items in Windows applications contain a single underlined letter.

To access a top-level menu, you can hold down the Alt key and press the underlined letter for that menu, and then just press an underlined letter in the menu to access specific commands. This technique enables you to navigate most of the interface without using the mouse. For example, to exit SolidWorks, instead of using the mouse to click the red X in the upper-right corner, you could press Alt+F, X. In Figure 2.32, you can see that the F in File is underlined, as is the X in Exit.

You may potentially run into conflicts when using Alt-keys. A combination of Alt + another keyboard key is a valid use of a hotkey combination. If you use any Alt hotkey combinations, it is likely that you have seen a conflict like this. In cases of conflict, the hotkey combination seems to gain priority over the Alt-key accelerator.

FIGURE 2.32

Alt-Keys in the File menu

Minimizing icons

In order to maximize valuable space on the monitor, many SolidWorks users strive to minimize the number of toolbar icons on the screen, or confine them to two rows of toolbars. You can do this by using the CommandManager, flyout toolbars, the "S" toolbar, right-click toolbars, and hotkeys, and removing unused icons, as well as the other techniques discussed here.

Having an uncluttered workspace is definitely a plus, but having easy access to commands is the real purpose of an interface in the first place. You need to strike a balance between too much and not enough. The more kinds of work you do in SolidWorks, the more tools you will need to have available. If you only create relatively simple machined parts and drawings, you will need fewer tools available than someone who creates complex plastic part assemblies with rendering and animation.

Contemplating device approaches

If you have never used a Spaceball or equivalent view-manipulation device, you should consider it. They are wonderful devices and do far more than just spin the view. Most of the devices also have several programmable buttons that you can link to menu items. They can move drawing views, parts within assemblies, and even manipulate selected objects in other Office applications and Web browsers.

Using touch and multi-touch support

I have written portions of this book on a Tablet PC. A tablet might not be ideal for long periods of SolidWorks usage. However, I use it regularly for presentations and even modeling when I really want to get the feel of drawing a line by hand. The stylus is not quite as intuitive as a pencil, but it is less of an impediment to the tactile feel of actual drawings than a clunky mouse. Tablets are a great option when used in combination with the new touch functions in SolidWorks, such as mouse gestures.

The mouse gestures functionality is considered a tool well suited to a tablet interface, where flicking the stylus is easier than mouse clicking. This is a single touch technique, since the stylus typically adds only a single point of contact with the screen.

Multi-touch devices are still rare, especially in the CAD-enabled office, but they are becoming more widely available. In preparation for this future functionality that seems ideally suited to visual applications such as CAD and 3D, SolidWorks has added functionality to take advantage of these tools. Multi-touch Action Mappings, as the SolidWorks Help refers to them, are intuitive two finger motions that enable you to control the view for actions like:

- Zoom in or out
- Rotate
- Pan
- Roll
- Zoom to fit
- Right click
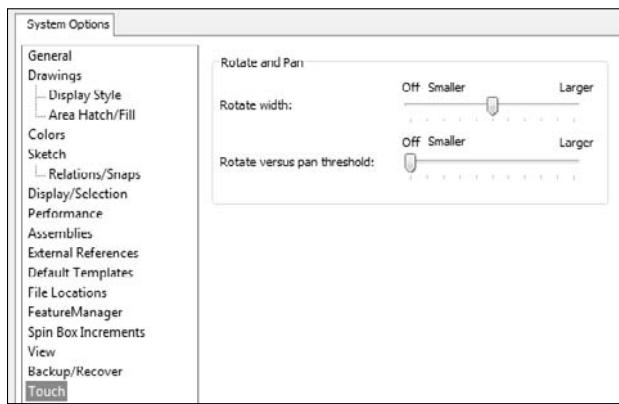
SolidWorks gave these special actions a new page in the Options dialog, shown in Figure 2.33.

SolidWorks special settings for multi-touch controls

Accessing macros

Macros are short snippets of programming code that have a particular function. Most macros are small and intended for simple tasks that are repeated many times, such as changing selected dimensions to four decimal places or zooming the screen so that it is sized 1:1 (actual size). Macros may be recorded, written from scratch, or a combination where you record a particular action for use as a starting point and then embellish it manually from there. Recorded macros may not always record the parts of the action that you want to make into a macro, but you can edit them manually to include anything that you can program with VBA (Visual Basic for Applications), which is included with the base SolidWorks package at no extra cost.

To access macros by using hotkeys, follow these steps:

1. **Make a folder in your SolidWorks installation directory called "macros."**
2. **Copy macros into this folder.**
3. **Start (or restart) SolidWorks.**
4. **Choose Tools ⇨ Customize ⇨ Keyboard.**
5. **Scroll to the bottom of the list under the Macros category and then assign hotkeys as you would for standard SolidWorks commands.**

Whether you are skilled at writing or recording macros, or you are just using macros collected from other people, they can be huge time-savers and offer functionality that you would not otherwise be able to access.

Saving custom interface settings

Once you have set up your menus and toolbars, worked out all of custom colors, figured out your hotkey usage, and connected your macros, you don't want to lose these settings when you reinstall the software or move to a different computer. Another user may want to share your settings, or you may want to transfer them to your home computer (for modeling the new deck or the doghouse, of course). Fortunately, these settings are very portable.

You can use the Copy Settings Wizard to save these settings out to a file. Access the wizard by choosing Start➪Programs➪SolidWorks 2010➪SolidWorks Tools➪Copy Settings Wizard. This creates a file with an *.sldreg file extension. You can restore settings by double-clicking this file on a computer that has SolidWorks installed on it.

Note

You may need to have administrator access to your computer to apply a SolidWorks registry file. ∎

The SolidWorks settings are actually Windows registry settings. The file that is saved by the wizard is just a registry file that has a different extension to prevent it from being applied too easily. Saved-out Windows registry files have a *.reg file extension, and you can integrate them into the registry by simply double-clicking them. If you are not familiar with the Windows registry, you should not make direct changes, because even small changes can cause serious problems with your operating system, installed software, or even hardware. The settings that are saved out by the Copy Settings Wizard are safe to transfer between computers. In order for the Copy Settings Wizard to work, you need to have Administrator-level access to your computer. The Copy Settings Wizard is shown in Figure 2.34.

FIGURE 2.34

The Copy Settings Wizard

Working with multiple document windows

You may sometimes have the luxury of working on a single part at a time, but more often, you will find yourself with several documents open at once. This is a common situation for most users. Fortunately, SolidWorks has several methods for dealing with "information overload," to help you sort through it all.

Managing windows

Like most Windows applications, SolidWorks can arrange the open document windows in one of several ways that are available through the Window menu (see Figure 2.35):

- **Cascade.** Most useful for accessing documents that are to be edited one by one.

- **Tile Horizontally.** Most useful for wide and short parts.

- **Tile Vertically.** Most useful for tall, narrow parts, or documents where you want to compare items in the FeatureManager.

- **Arrange Icons.** When windows are minimized to icons, this menu selection arranges the icons neatly, starting in the lower-left corner of the window.

FIGURE 2.35

The Window menu
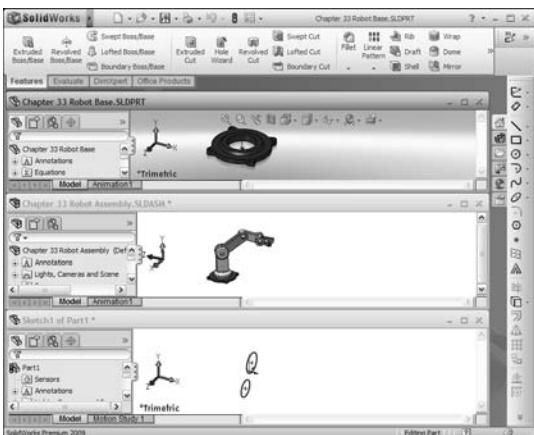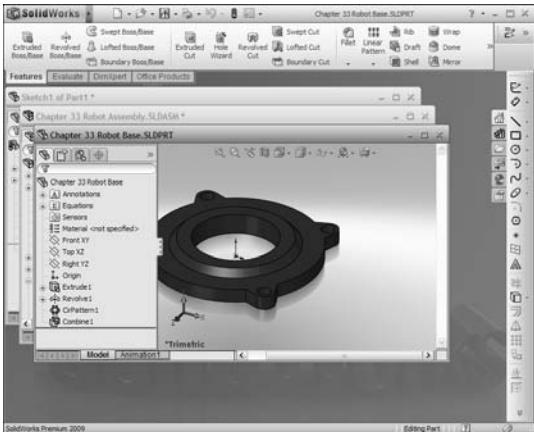
The images in Figure 2.36 are meant to show the arrangement of the windows, not the content of the windows. Also, remember that you can use the F9 key to close the FeatureManager, the F10 key to remove the toolbars to create extra interface space when arranging several windows in the graphics window, and the F11 key to remove portions of the interface and enable you to work full screen.

FIGURE 2.36

Window Arrangements: Cascade, Tile Horizontally, and Tile Vertically

Changing windows

You can use several techniques to change from one SolidWorks window to another. By clicking on the Window menu, you can view a list of open document windows (refer to Figure 2.35). You can then select the desired window directly from this menu. If more than a few windows are open, a More Windows option appears at the end of the list, as shown in Figure 2.37. Clicking this option brings up a separate window that enables you to select from the complete list. Press Ctrl+Tab to open the Open Documents dialog box (see Figure 2.37). This enables you to select the document visually that you want to open.

Additionally, by default the R hotkey opens the Recent Documents dialog box, similar to the Recent Documents list in the File menu. This can also be accessed via the File menu if necessary. The Recent Documents dialog box is shown in Figure 2.38.
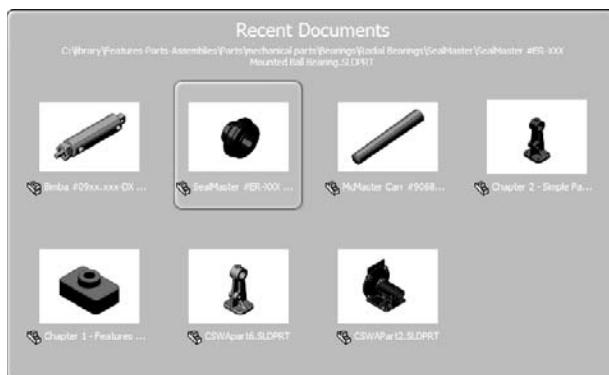
FIGURE 2.37

The Open Documents dialog box

FIGURE 2.38

The Recent Documents dialog box

Getting to Know the Interface

By this point, you have learned quite a bit about all the tools involved in using the SolidWorks interface. In this tutorial, you get some hands-on practice at manipulating the interface. This tutorial is intended to reinforce the following skills:

- Adding and removing toolbars
- Adding and removing toolbar buttons
- Adding and removing items from drop-down and RMB menus
- Setting up the CommandManager
- Setting up hotkeys
- Linking a hotkey to a macro
- Changing interface colors

Copying the existing settings

Regardless of what your initial settings are, you do not want to lose them. Before you start to make changes to your system, you should save out the existing settings to a file from which they can be recovered. You can do this using the Copy Settings Wizard, as shown in Figure 2.39.

FIGURE 2.39

The Copy Settings Wizard

To use the Copy Settings Wizard, follow these steps:

1. Close SolidWorks.

2. **Choose Start ⇨ Programs ⇨ SolidWorks 2010 ⇨ SolidWorks 2010 Tools ⇨ Copy Settings Wizard.**

3. **Select Save Settings, and click Next.**

4. **Enter a location and a name for the file.**

5. **Select the items that you would like to save.** For the purposes of this tutorial, make sure that the following options are selected: Keyboard Shortcuts, Menu Customization, Toolbar Layout, and All Toolbars.

6. **Click Finish.** Browse to the location where you saved the file and make sure that it is there.

Setting interface items to their default settings

You can set the interface back to the default settings using one of two methods. The first method, editing the Windows registry, may not be available to all users. It requires Administrator access to your computer and a good familiarity with Windows.

Caution

Editing the Windows registry can be dangerous if you make a mistake. Do not attempt this method is you have any doubts about what you are doing. ∎

To set SolidWorks back to its default settings by editing the Windows registry, follow these steps:

1. **Close SolidWorks.**

2. **Choose Start ⇨ Run.**

3. **Type** regedit, **and click OK.**

4. **Browse to** HKEY_CURRENT_USER\Software\SolidWorks\SolidWorks 2010 **or the appropriate folder for the version that you are using.**

5. **To return all settings in SolidWorks to default, rename the entire SolidWorks 2010 folder to include "(old)" at the end of the filename.**

6. **Close the Registry Editor.**

7. **The folder is re-created when SolidWorks starts up again, and is populated with default values.** If you need to get the previous folder back, you can delete the new one and rename the old one to remove the "(old)" from the name.

The second method, which is less risky but less complete, is to go to the main settings locations and use the tools provided to return settings to their defaults. Restart SolidWorks and create a new blank document (you cannot display the Customize dialog box without a document open). To access the resets for the interface, do the following:

1. **Choose Tools ⇨ Options ⇨ General ⇨ Reset, and go to the bottom-left area of the dialog box.**

2. **Choose Tools ⇨ Customize ⇨ Toolbars ⇨ Reset**, and go to the bottom-left area of the dialog box.

3. **Choose Tools ⇨ Customize ⇨ Menus ⇨ Reset All**, and go to the right side of the dialog box.

Note

The menu reset is known to work improperly in version 2009 and prior. It is not clear if this will be fixed in the 2010 version. Fortunately, customizing menus is not a task you will frequently do, so resetting them to default is something you will do even less frequently. ■
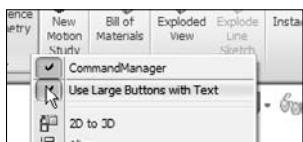
4. **Choose Tools ⇨ Customize ⇨ Keyboard ⇨ Reset to Defaults**, and go to the upper-right area of the dialog box.

5. **Choose Tools ⇨ Customize ⇨ Options**; there are three Reset to Defaults buttons along the left side of the dialog box.

Customizing the CommandManager

Now that you have restored the default settings, you can begin customizing the interface with the CommandManager. To do this, open a part document or create a new one, then click the RMB anywhere on the CommandManager, and deselect the Use Large Buttons with Text option, as shown in Figure 2.40. When you have done this, the check mark should no longer appear in front of the option.

FIGURE 2.40

Deselect the Use Large Buttons with Text option

Next, add some toolbars to CommandManager, as follows:
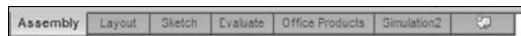
1. **RMB-click the CommandManager tabs and select Customize CommandManager.**

2. **Click (left-click) the New Tab icon at the right end of the CommandManager tabs and select Surfaces, Sheet Metal, and Annotations. Deselect the Sketch tab by right-clicking it and selecting Hide Tab.** The new tab icon is shown in Figure 2.41.
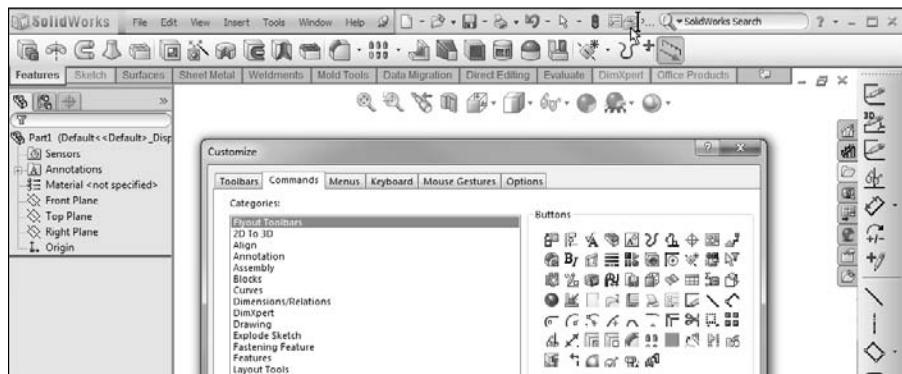
Adding tabs to the CommandManager

3. In the Customize dialog box, select the Large Icons option.

4. **Turn the Sketch toolbar on, but not inside the CommandManager; use the Toolbars tab of the Customize dialog box.** Dock the Sketch toolbar to the right side of the window.

5. **Select the Standard Views toolbar and drag it to the right end of the Command-Manager.** Now drag a couple of buttons off it, such as the bottom, or left, or back views.

6. **On the Commands tab of the Customize dialog box, select the Flyout toolbars entry, and drag any Flyout toolbars from the Customize dialog box to the Menu Bar toolbar.** Figure 2.42 shows this step in action.

FIGURE 2.42

Adding flyouts to the Menu Bar toolbar
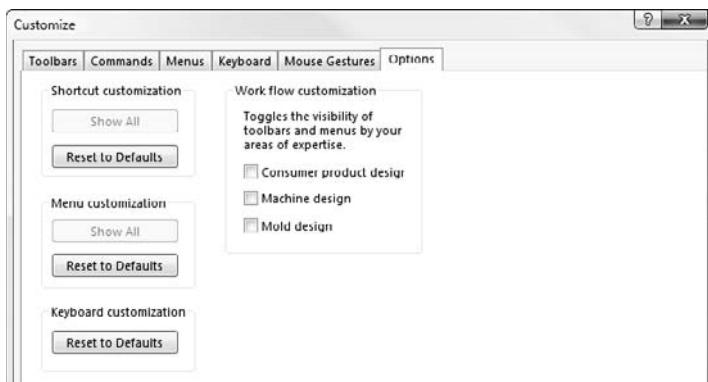
7. **With the Commands tab of the Customize dialog box still active, click on the View Category.**

8. **Add or remove tools until this toolbar has the tools you want to use on it. If you want to deselect the Heads-UP View toolbar, activate the Toolbars tab in Customize, and de-select the View (Heads-Up) option.**

9. **Run your cursor over the main menu flyout and use the pushpin to pin the menu open**.

10. **Choose Tools ⇨ Customize ⇨ Options and select both Show All buttons for Shortcut customization and Menu customizations.** This removes the double arrows at the bottoms of RMB menus. This setting is shown in Figure 2.43.

FIGURE 2.43

Removing truncated menus
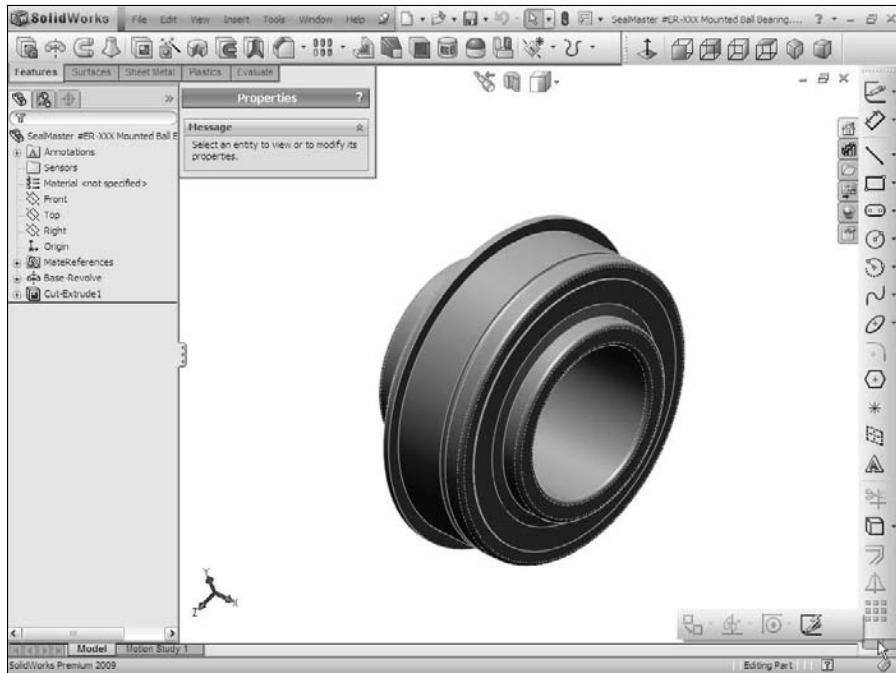
11. **While still in the Customize dialog box, change to the Toolbars tab and deselect the Context Toolbar Settings ⇨ Show In Shortcut Menu option to disable the right click context toolbar to put the RMB menus back to their pre-2008 state (all entries in menu use text).**

12. **Click and drag the PropertyManager tab (second tab from the left) from the FeatureManager, and dock it just to the right of the FeatureManager under the CommandManager tabs. The display should now look like Figure 2.44.**

The small flyout in the lower-right corner of Figure 2.44 is the remainder of the Sketch toolbar that does not fit on the screen. This screen shot was taken on a 12-inch monitor with large icons at 1024×768 resolution. At such a low resolution, long toolbars do not fit on the screen, and the remaining icons are activated by the two small arrows at the end of the toolbar.

FIGURE 2.44

The customized SolidWorks interface
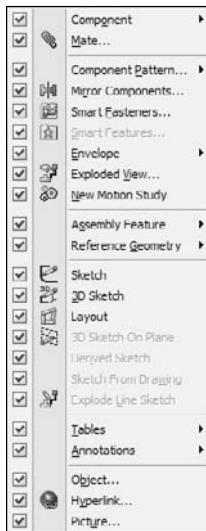
Customizing menus

If you always do the same types of work, or more importantly, *never* do certain types of work, you might consider customizing some menus to remove items that you never use. Customization applies to both the main drop-down menus and the context-sensitive RMB menus. To customize a menu, follow these steps:

1. **Choose Insert ➪ Customize Menu. Note that Customize Menu is different from Customize.**

2. **Deselect the menu items Sketch from Drawing, DXF/DWG, Object, Hyperlink, and Picture (see Figure 2.45).** Click anywhere outside the list to close it.

3. **Click on the Insert menu to ensure that the deselected items have been removed.**

4. **RMB-click the Right plane in the FeatureManager.**

FIGURE 2.45

Customizing the Insert menu

5. **Select Customize Menu.**

6. **Deselect Section View.** Click anywhere outside the list to close it.

7. **RMB-click the Right plane to verify that Section View has been removed.**

Changing interface colors

This tutorial does not depend on RealView capabilities, but later work will. To find out whether your computer is RealView capable, check your video card and driver version against the list on the SolidWorks Web site. On the main page of the site, follow the link for Video Cards.

Before starting this tutorial, make sure the RealView icon is disabled. You can deselect RealView by choosing View ➪ Display ➪ RealView Graphics.

Cross-Reference

For more information on RealView graphics, see Chapter 5. ∎

Edit the colors used in the interface:

1. **Choose Tools ➪ System Options ➪ Colors.** Make sure the "Plain (Viewport Background color above)" option is selected.
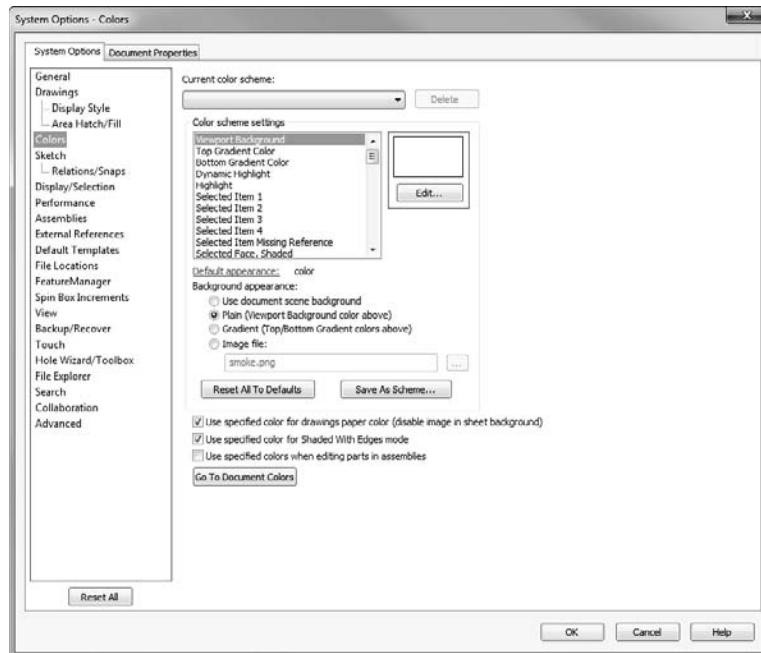
2. Change the Viewport Background color, first in the Color Scheme Settings list, to a light gray color, then click OK and make sure the setting was applied.

3. Return to the Colors dialog box by choosing Tools ⇨ System Options ⇨ Colors, click the Save As Scheme button, and save the color scheme as Plain Gray.

4. Now select the Gradient option, and select the top and bottom gradient colors (second and third in the top list). Click OK and see how you like the gradient display. Adjust the colors until you are happy with them.

5. Return to the Colors dialog box by choosing Tools ⇨ System Options ⇨ Colors, click the Save As Scheme button, and save the color scheme as Gradient. Figure 2.46 shows the Colors dialog box.

6. Use the Current color scheme drop-down list at the top to change the color schemes between the newly saved schemes.

FIGURE 2.46

The Colors settings

Adding hotkeys

For many users, hotkeys are an integral part of the everyday experience of using SolidWorks. You can easily assign hotkeys and manage the assignments by following these steps:

1. **Choose Tools ⇨ Customize ⇨ Keyboard.**

2. **In the Search For text box, type** Options.

3. **Click in the Shortcut column, and type** O.

4. **Click again in the Search For text box, and type** Customize.

5. **Click in the text box next to the Tools ⇨ Customize entry and press Ctrl+C.** Click No to the question of whether you want to reassign the hotkey. Press Ctrl+T instead.

6. **Click OK to exit the Customize dialog box.**

7. **Press the O key to open the Options tab. Click OK to exit.**

8. **Press Ctrl+T to open the Customize dialog box. Click OK to exit.**

9. **Press Alt+F, and then click the Close button to exit SolidWorks.**

Combining macros with hotkeys

The following steps show you how to link a macro to a hotkey:

On the CD-ROM

You can use the macro called rectangle.swp, located on the CD-ROM. ∎

1. **Find your SolidWorks installation directory.** By default, this directory is `C:\Program Files\SolidWorks`.

2. **Create a folder called Macros in the SolidWorks directory and put the rectangle. swp macro in it.**

3. **Open SolidWorks.**

4. **Create a new blank part document.**

5. **Press Ctrl+T to access the Customize dialog box.**

6. **Click the Keyboard tab.**

7. **In the Search For text box, type** rectangle.

8. **Click in the Shortcut column next to the listing with the Category of Macros, and press R.**

9. **Click OK to exit the Customize dialog box.**

10. **Press R.** The rectangle macro runs and draws a sketch rectangle on the Front plane, centered on the origin.

11. **Press Ctrl+S to access the Windows standard hotkey for the Save command.** Name the part **rectangle.sldprt** and save it to a workspace directory.

12. **Press Alt+F and then click Close to exit SolidWorks.**

The use of Alt-keys and hotkeys is somewhat exaggerated in this tutorial, but it is intended to get you used to working with them.

Summary

The SolidWorks interface has many elements because SolidWorks has so much functionality. You can access most elements multiple ways, which can be liberating because it offers options, but it also can add to the confusion because there is so much to know. You do not need to know every way to do everything; you only need to know the best way for you. After using this book to find the various ways of using the interface, you can develop the way that is most comfortable for you and stick with it.

Be aware that every couple of releases, SolidWorks changes the interface, and often, they use the most radical options available as the new defaults. Keep a copy of your settings file with you, so you can restore settings or take your settings to another computer quickly if you need to re–set up.

Working with Sketches

S o far, in this book, you have looked mainly at concepts, settings, and setup, which is necessary but mundane business. In this chapter, you begin to learn how to control parametric relationships in sketches. Then in later chapters, you begin to build models, simple at first, but gaining in complexity and always demonstrating new techniques and features that build your modeling vocabulary. Beyond this, you will move into putting the parts together into assemblies. Finally, you use the parts and assemblies to create drawings.

This chapter deals mainly with sketches in parts. However, you will be able to apply many of the topics I cover here to assemblies. Some related topics, such as layout sketches, have functionality that is exclusive to assemblies and are covered in the assemblies' chapter.

Cross-Reference

I discuss 3D sketches in detail in Chapter 31. ■

When you open a sketch, several tools become available, specifically all the sketch entities and tools. Conversely, you cannot do several things *until* you open a sketch. For example, you cannot apply a Fillet feature while a sketch is open. Open sketches and selection filters are two very common sources of frustration for new users. Several indicators exist to let you know when you are in Sketch mode:

- The title bar of the SolidWorks window displays the text Sketch X of Part Y.

- The lower-right corner of the status bar displays the text Editing Sketch X.

- The Confirmation Corner displays a sketch icon in the upper-right corner of the graphics window.

IN THIS CHAPTER

Beginning a sketch

Distinguishing sketch entities

Creating relationships in sketches

Examining sketch settings

Using sketch blocks in parts, assemblies, and drawings

Using reference geometry

Learning to use sketch relations tutorial

Using blocks and belts tutorial

Referencing geometry tutorial

- The Sketch toolbar button displays the text Exit Sketch.

- The red sketch Origin displays.

- If you are using the grid, it displays only in Sketch mode.

While most users find the sketch grid annoying or distracting, when teaching, I've always used the grid to remind students when they are in Sketch mode. If you forget or would like a visual cue, the sketch grid is a useful option.

Opening a Sketch

Sketches must be either open or closed, and you can only have one sketch open at a time. SolidWorks uses many indicators to show the state of a sketch, including the Confirmation Corner and the Task Bar.

Several methods exist to open new sketches in SolidWorks:

- Click a sketch entity toolbar button from the Sketch toolbar; SolidWorks prompts you to select a sketch plane. When you select the plane, the sketch opens.

- Preselect a plane or planar face and then click either a sketch entity button or the Sketch button.

- Use the left mouse button to click context toolbar — click a face or plane and select the Sketch icon.

- Use the right mouse button (RMB) to click a plane or planar face and select Insert Sketch. Planes can be selected from either the graphics window or the FeatureManager.

You can open existing sketches several ways:

- Right-click a sketch in the FeatureManager or graphics window, and select Edit Sketch.

- Select a sketch from the FeatureManager or graphics window, and click the Sketch button on the Sketch toolbar.

- Left-click a sketch or feature and click the Edit Sketch icon from the context toolbar.

- Double-click a sketch with the Instant 3D tool active.

Identifying Sketch Entities

The first step in creating most SolidWorks parts is a sketch. This will usually be a 2D sketch, although you can also use 3D sketches. A 2D sketch is simply a collection of 2D lines, arcs, and other elements that lie together on a plane; it usually also contains relations and/or dimensions between the entities so that the sketch can automatically adjust to changes because each sketch entity understands its function.

SolidWorks sketch entities include many types, some of which you will use all the time, and some of which you may not use, even if you spend years working with the software. Next, I identify each entity type. This enables you to see it at least once and know that it is available if you need it at some point.

The Sketch toolbar

In the following section, I first identify the default buttons on the Sketch toolbar, followed by the rest of the entities that you can access by choosing Tools ➪ Customize ➪ Commands ➪ Sketch.

Sketch opens and closes sketches. You may notice that the name of the button changes depending on if the sketch is open or closed. If you preselect a plane or planar face and then click the Sketch button, SolidWorks opens a new sketch on the plane or face. If you preselect a sketch before clicking the Sketch button, SolidWorks opens this sketch. If you preselect an edge or curve feature before clicking the Sketch button, SolidWorks automatically makes a plane perpendicular to the nearest end of the curve from the location you picked. If you do not use preselection, and only click the Sketch tool with nothing selected, SolidWorks prompts you to select a plane or planar face on which you want to put a new sketch, or an existing sketch to edit.

3D Sketch opens and closes 3D sketches with no preselection required. 3D sketch is covered in more detail in Chapter 31.

Smart Dimension can create all types of dimensions used in SolidWorks, such as horizontal, vertical, aligned, radial, diameter, angle, and arc length. You can create dimensions several ways, as shown in Figure 3.1:

- By selecting a line and placing the dimension
- By selecting the endpoints of the line and placing the dimension
- By selecting a line and a point and placing the dimension
- By selecting a pair of parallel lines and placing the dimension

Selecting the line is the easiest and fastest method. Selecting parallel lines on the ends is not recommended because if you delete either of the selected lines, the dimension is also deleted; however, sometimes this method is necessary.

You can use the first and second techniques for the angled line shown in Figure 3.1 to create any of the three dimensions shown. To accomplish this task, drag the cursor while placing the dimension until the witness lines snap to the orientation you want.

Tip
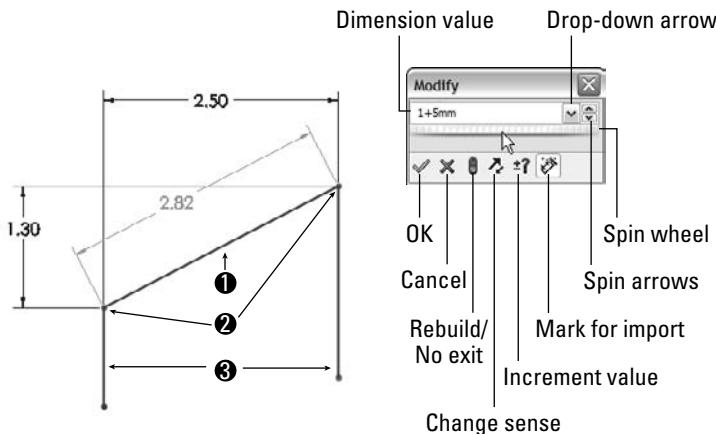
To lock the orientation of a dimension while moving the cursor to place the actual dimension value, click the RMB. To unlock it, click the RMB again. The RMB cursor appears as a lock when the functionality is unavailable and an unlock icon when it is. ∎

The third technique locks you into the horizontal orientation because of the orientation of the selected lines.

FIGURE 3.1

Selection options for linear Smart Dimension

Note

In some situations, you may run into lines that appear to be parallel but are not exactly so. This will result in an angle dimension instead of a linear dimension. Here, you can select one of the lines and one endpoint. SolidWorks requires parallelism be precise, and situations where lines aren't exactly parallel happen if the angular measurement is off in the second or third decimal place, measuring in degrees. Imported 2D drawings and reverse engineered 3D models can be particularly susceptible to this type of error. ∎

Caution

When you select lines to establish a dimension instead of endpoints, both of the lines gain an implied parallel relation that prevents them from moving as you might predict. In the example shown in Figure 3.1, neither of the end lines can be angled unless you remove the dimension.

Another issue with adding dimensions to lines is that if you delete either line, the dimension is also deleted. This is not true for the first and second techniques, where as long as the endpoints remain, the dimension remains. ∎

You can change Smart Dimension values several ways. The most direct way is to simply key in a value such as 4.052. The software assumes document units unless you key in something specific. You could also key in an expression, even with mixed units, such as 8.5 mm/2+.125 or 25.4+.625 in. You can also key in negative dimensions, which function the same as the Change Sense button in the Modify box.

Another way to put a value into the Modify box is to click the down arrow to the right of the value field, and select either to use an equation to calculate a value or a Link Value. A Link Value is like a variable name to which you can assign a value. You can link multiple dimension values to that Link Value. In sheet metal parts, the default Link Value of Thickness is used; if you change the thickness in one feature, it changes for all the sheet metal features.

To the right of the drop-down arrow is a pair of up and down "spin" arrows that enable you to change the value in the Modify box by a set increment amount. You set the increment in Tools ⇨ Options ⇨ System Options ⇨ Spin Box Increments. You can also store multiple increment values within the Increment Value icon on the Modify box.

The final way to change the value in the Modify box is by using the wheel underneath the value field. The wheel uses the default increment value. Pressing Ctrl while using the wheel multiplies the increment by ten, and pressing Alt while using the wheel divides the increment by ten.

Here is a look at the Dimension Properties interface.

- **Radial.** You create the dimension by selecting an arc and placing the dimension. If you want a radial dimension of a complete circle, you must right-click the dimension after you create it, select Display Options, and select the Display as Radius/Display as Diameter toggle, as shown in Figure 3.2. Alternatively, you could use the Radius or Diameter leader display options on the Leaders tab of the Dimension PropertyManager.

FIGURE 3.2

The Dimension Properties interface

- **Diameter.** You can create the dimension by selecting a complete circle and placing the dimension. If you want a diameter dimension for an arc, use the RMB menu or Dimension Properties dialog box and select the Diameter Dimension option.
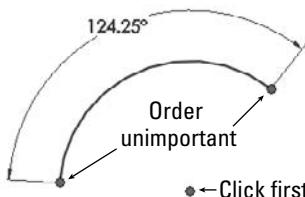
Note

Along with the Radial and Diameter dimensions, you may also want to dimension between arcs or circles, from tangent or nearest points. To do this, press the Shift key and select the Smart Dimension tool to select the arcs near the tangent points. Alternatively, to change a dimension from a center-to-center dimension to a max-to-max dimension, you can drag dimension attachment points to tangent points or use the dimension properties. ∎

- **Angle.** You can create the angle dimension one of two ways. If the angle to be driven is between two straight lines, simply select the two straight lines and place the dimension. If you are creating an included angle dimension for an arc where there are not necessarily any straight lines drawn, then with the Smart Dimension tool active, first select the vertex of the angle, and then the two outlying points, as shown in Figure 3.3.

FIGURE 3.3

Creating an included angle dimension

- **Arc Length.** You can create the dimension by selecting an arc and its endpoints with the Smart Dimension tool.

Line creates straight lines using one of two methods:

- **Click+Click.** Used for drawing multiple connected end-to-end lines. Click and release the left-mouse button to start the line; each click and release ends the previous line and starts a new one. Double-click, press Esc, or deselect the Line tool to end.
- **Click and Drag.** Used to draw individual or unconnected lines. Click, drag, and drop. The first click initiates the line, and the drop ends it.

Note
The click+click and click and drag methods have been treated equivalently by SolidWorks until recent releases. Some new functions like Enable on-screen numeric input seem to function better with click+click than click and drag. ∎

Alternate methods for drawing lines include horizontal, vertical, angle, and infinite lines. The interface for these options appears in the PropertyManager, as shown in Figure 3.4.

- **Horizontal, Vertical.** These settings require you to select a starting point and an ending vertical or horizontal position. There does not seem to be any compelling reason to use this instead of the regular line command.

- **Angle.** This setting enables you to specify an angle and drag a line at this angle. Again, I can find no compelling reason to use this tool.

- **Infinite length.** SolidWorks parts have a working space limited to 1000 meters on a side, centered on the Origin. Infinite lines extend well beyond this, although you cannot draw or dimension a regular line outside of this box. I have not come across a compelling use for this feature.

FIGURE 3.4

The Insert Line PropertyManager interface

Note
The Add Dimensions option exists in several sketch entity PropertyManagers and adds Smart Dimensions to newly sketched entities. The option only appears in the sketch entity PropertyManager if the setting at Tools⇨ Options⇨Sketch⇨Enable on-screen numeric input on entity creation is selected.

The on-screen numeric input is not the same as the Input Dimension Value function, and, in fact, it overrides that option. You cannot input dimension values when using the Add Dimensions in conjunction with click and drag sketching. It appears to be intended for click+click drawing only, so that you can enter values between clicks. ∎
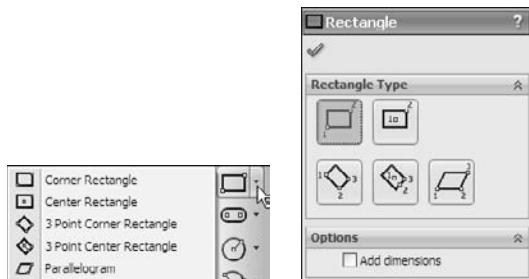
Corner Rectangle creates a rectangle by clicking one corner and dragging to the diagonal corner. This action creates four lines with Horizontal and Vertical sketch relations, as appropriate. The Corner Rectangle is also available as a flyout icon with a Corner Rectangle, Center Rectangle, 3 Point Corner Rectangle (rectangle at an angle), and 3 Point Center Rectangle, as well as a Parallelogram. Figure 3.5 shows the flyout and flyout icons, and the PropertyManager for the Rectangle, which also enables you to switch types of rectangle easily.

The Rectangle flyout with associated icons

Notice the Add dimensions check box in the PropertyManager. Selecting this box while creating a rectangle causes the software to add dimensions aligned with the sides of the rectangle. This option is also available for lines, arcs, and circles.

Note that if you use this option in conjunction with the Enable on-screen numeric input on entity creation setting, found at Tools ➪ Options ➪ Sketch, it makes creating sketch entities to the correct size immediately much easier.
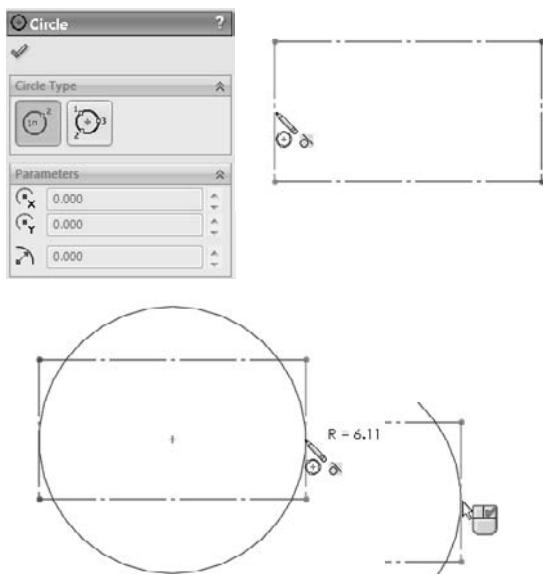
Circle creates a circle using one of two methods, which are available from either the flyout icon or the Circle PropertyManager:

- **Center Creation.** Click the center of the circle and drag the radius. The Circle PropertyManager calls this function *center creation*.

- **Perimeter Creation.** To create a circle using this technique, you must select the Perimeter Creation option from the Circle PropertyManager window after clicking the Circle tool. There is also a separate Perimeter Creation toolbar button and a menu selection for Tools ➪ Sketch Entities ➪ Perimeter Circle. This only creates tangent relations with other entities in the current sketch; if you are building a circle from model edges or entities in other sketches, you need to apply the relations manually. SolidWorks calls these functions *perimeter creation*.

- **Tangent to Two Entities.** Start the circle with the cursor near one line in the sketch. A Tangent symbol appears by the cursor with a yellow background. Click and drag the diameter to the second tangent entity, where a similar cursor symbol should appear. Release the mouse button and right-click the green check mark icon. This process is shown in Figure 3.6.

- **Tangent to Three Entities.** Use the same process for Tangent to two entities, but omit the right-click of the green check mark icon. After dropping on the second tangent, drag again to the third tangent entity.

FIGURE 3.6

Creating a perimeter creation circle

Centerpoint Arc creates an arc by clicking the center, dragging the radius, and then clicking and dragging the included angle of the arc. The first two steps are exactly like the Center-Radius circle.

Tangent Arc creates an arc tangent to an existing sketch entity. Depending on how you move the cursor away from the end of the existing sketch entity, the arc can be tangent, reverse tangent, or perpendicular, as shown in Figure 3.7.
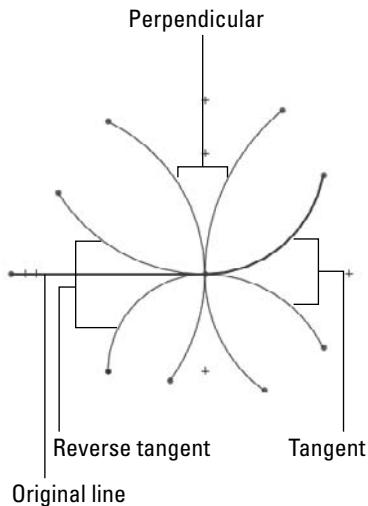
Another way to create a tangent arc (called autotransitioning) is to start drawing a line from the end of another sketch entity, and while holding the left mouse button, press the A key; or return the cursor to the starting point and drag it out again. This second method can be difficult to master, but it saves time compared to any of the techniques for switching sketch tools.

Using the Tangent Arc feature

Perpendicular

Reverse tangent   Tangent

Original line

3 Point Arc creates an arc by first establishing endpoints, and then establishing the included arc, as shown in Figure 3.8. Again, this tool also works using the click+click or click and drag methods.

Creating a three point arc

A = 90° R = 18.98

L = 37.28

Click-drag

Click-drag

Sketch Fillet creates a sketch fillet in one of two ways. Either you can select the endpoint where the sketch entities intersect, or you can select the entities themselves, selecting the portion of the entity that you want to keep. Figure 3.9 illustrates both techniques.

The Sketch Chamfer tool is on the same flyout as the Sketch Fillet by default. Sketch Chamfer does not have a list selection box the way that fillet does, and does not use a preview like the fillet.

Sketch fillets
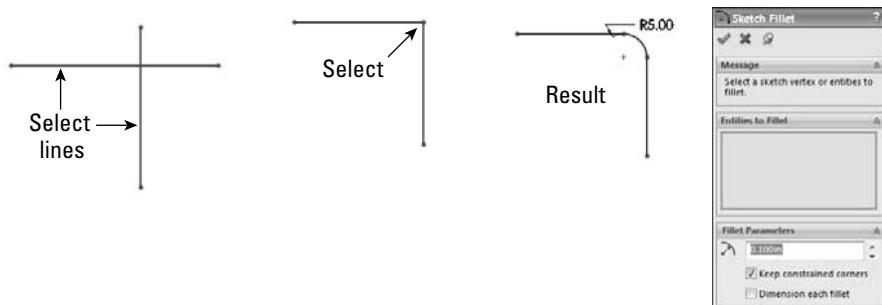
While the Sketch Fillet tool is easy to use and may align with your way of working in a 2D program, it is not considered best practice to use sketch fillets extensively. Some reasons for this include:

- Large changes in the size or shape of the rest of the sketch can make the fillets result in the feature built from the sketch to fail.

- SolidWorks (and other parametric programs as well) often has difficulty solving tangent arcs in some situations. You may see fillets flip tangency or go around 270° instead of just 90°. Using many fillets in a sketch can often cause trouble.

- If you want to remove the fillets temporarily, there is no good way to do this if you have used sketch fillets.

- Sometimes feature order requires that other features, such as draft, come before the fillet, which is difficult to do if they are part of the sketch.

- Sometimes a 2D fillet simply cannot create the required 3D geometry.

Fillet features are the preferred method for creating rounds and fillets. The same can be said for chamfers. Still, sometimes you need to use tangent arcs in sketches. You will have to decide which way works best for you.

FIGURE 3.9

Creating a sketch fillet
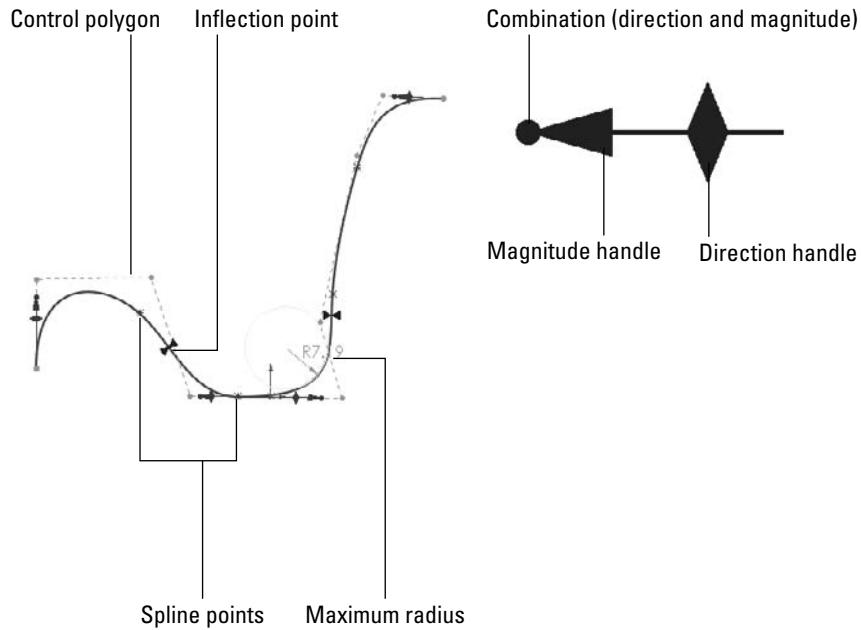
Centerline follows the same methods as regular lines and is called a *construction line* in some cases. Other construction entities, such as construction circles, are not available directly, but you can create them by selecting the For Construction option in the PropertyManager for any entity.

Spline draws a freeform curve. Splines may form either a single closed loop or an open loop. In either case, the spline is not allowed to cross itself. You can draw a spline by clicking each location where you want to add a control point. Figure 3.10 identifies the elements of a spline. The detail image shows the structure of a spline handle.

FIGURE 3.10

The structure of a spline and spline handles

Splines are used mainly for freeform complex shapes in 2D and 3D sketches, although you can also use them for anything in which you would use other sketch elements. If you need more information on splines and complex shape modeling, refer to the *SolidWorks Surfacing and Complex Shape Modeling Bible* (Wiley, 2008).

Point creates a sketch point. Aside from limited cases of lofting to a point or using a point as a constraint sketch in a Fill feature, sketch points are usually used for reference or for the location of the centerpoint of Hole Wizard features.

You can also use the sketch point as a virtual sharp. If two sketch entities do not actually intersect because of a fillet or chamfer, selecting the two entities and clicking the Point tool creates a point at the location where they would intersect if they were extended. This is useful for dimensioning to the sharp. Virtual sharp display is controlled by a Document Property setting.

3D Sketch Plane creates a plane in a 3D Sketch. I discuss 3D Sketches in more detail in Chapter 31. By sketching on planes within a 3D sketch, you get most of the benefits and usage of 2D sketches, and you do not have to deal with history between sketches. Before committing too much

work to this course, you should look into some of the shortcomings of 3D Planes. The planes are treated just like another entity in the 3D sketch, which means you can assign sketch relations to them, but it also means that they can move around within the sketch like sketch entities.

Add Relation displays a PropertyManager window that enables you to apply sketch relations. This interface appears to be obsolete because it is easier to simply select sketch items and apply relations via the context toolbar or in the PropertyManager window that appears automatically when you select them; however, there are some subtle workflow-related reasons for using this tool.

Two advantages exist to using the Add Relations dialog box over simply selecting sketch entities and adding relations. When the Add Relation PropertyManager is active, you do not need to use the Ctrl key to select multiple entities. You also do not need to clear a selection before making a new selection for the next relation. These two reasons sound minor, but if you have a large number of sketch relations to apply, the workflow goes much more smoothly using this tool than the default method.

Display/Delete Relations enables you to look through the relations in a sketch, and sort them according to several categories. From this window, you can delete or suppress relations and replace entities in relations.

Quick Snaps flyout enables you to quickly filter types of entities that sketch elements will snap to when you move or create them. To access the tools, click the drop-down arrow to the right of the toolbar button.

Mirror Entities mirrors selected sketch entities about a single selected centerline, and applies a Symmetric sketch relation. In addition, a Dynamic Mirror function is described later in this chapter.

Note

PropertyManagers for sketch tools such as Mirror, Convert, Fillet, and Intersection Curve now include a selection box for the entities to be used in the operation. The Offset PropertyManager is one that is conspicuously missing this functionality. ■

Convert Entities convert edges, curves, and sketch elements from other sketches into entities in the current sketch. When edges are not parallel to the sketch plane, the Convert Entities feature projects them into the sketch plane. Some elements may be impossible to convert, such as a helix, which would produce a projection that overlaps itself. Sketch entities created using Convert Entities get an On Edge sketch relation.
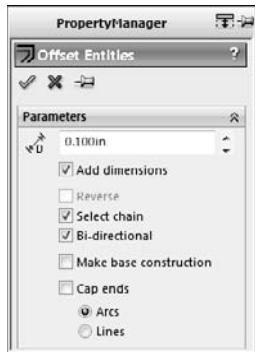
Offset Entities works like the Convert Entities feature, except that it offsets the sketch to one side or the other of the projection of the original edge, sketch, or curve. Figure 3.11 shows the interface for this command.

The Offset Entities interface

The options available in the Offset Entities interface are as follows:

- **Add dimensions.** Constrains offset sketch entities. Instead of the On Edge relations, Offset Entities creates an Offset sketch relation that cannot be re-created manually.

- **Reverse.** Changes the direction of the offset.

- **Select chain.** Selects continuous end-to-end sketch entities.

- **Bi-directional.** Offsets to both sides simultaneously.

- **Make base construction.** If you are offsetting sketch entities within the active sketch, this option converts the original sketch entities to construction sketch geometry.

- **Cap ends.** Is available only when you have selected the Bi-directional option. Capping the ends with arcs is an easy way to create a slot from a sketch of the centerline. This function works with all sketch entities; it is not limited to straight slots. Figure 3.12 shows examples of the Cap ends option.

Caution

The Offset Entities command may fail if the offset distance is greater than the smallest radius of curvature, and you are attempting to offset to the inside of the arc. ■
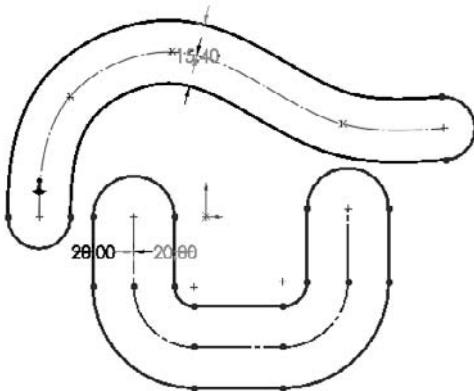
In addition to the bi-directional offset with capped ends, SolidWorks also has slot sketch entities for straight and curved slots, which are covered later in this chapter. Composite slots (made of a combination of straight and curved sections) still require the offset method.

Trim is actually several functions rolled into one, and it is an extremely powerful tool for editing sketches. Trim Entities allows several methods for trimming, as well as extending and deleting sketch entities. Figure 3.13 shows the PropertyManager interface for this function.

The results of using offset entities cap ends

The Trim interface

- **Power trim.** Trims by dragging a cursor trail over multiple entities. The entities that you drag the cursor over are trimmed back to the next intersecting sketch entity. Each time you trim an entity, a red box remains until you trim the next entity. If you backtrack with the cursor and touch the red box, this trim is undone. This option is best used when you need to trim a large number of entities that are easy to hit with a moving cursor. Figure 3.14 shows the Power Trim feature in action.
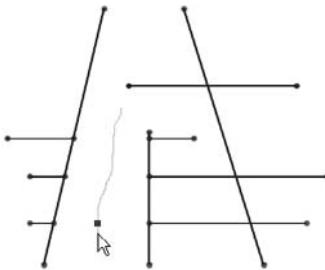
You can also use power trim to extend sketch entities along their paths by dragging the endpoints. Regular dragging can also change the position or orientation of the rest of the entity, but by using the Power trim feature, you affect only the length.

FIGURE 3.14
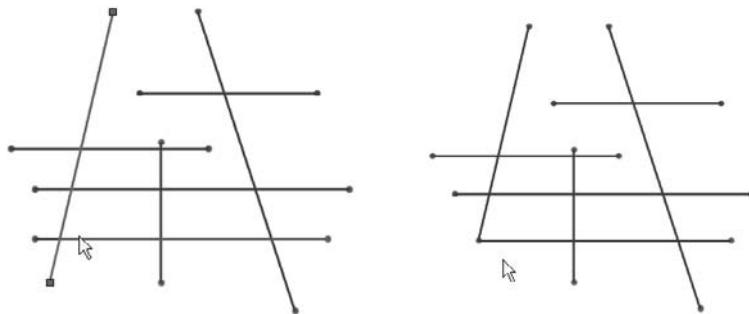
The Power trim feature in action

- **Corner.** Trims or extends two selected entities to their next intersection. When you use the Corner option to trim, the selected portion of the sketch entities is kept, and anything on the other side of the corner is discarded. Figure 3.15 shows two ways that the Corner option can work.

FIGURE 3.15

Using the Corner option
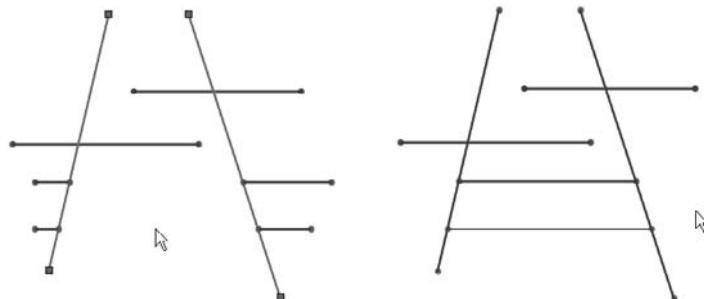
- **Trim away inside.** Trims away selected entities inside a selected boundary. The boundary may consist of a pair of sketch entities or a model face (edges of the face are used as the boundary). Only entities that cross both selected boundaries (or cross the closed loop of the face boundary twice) can be trimmed. This option does not trim a closed loop such as a circle, ellipse, or closed spline.

- **Trim away outside.** Functions exactly like the Trim away inside option, except that sketch entities outside of the boundary are discarded. The Trim away inside and Trim away outside options are illustrated in Figure 3.16.

Using the Trim away inside and Trim away outside options
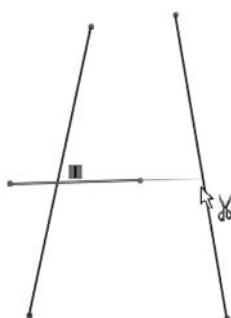
- **Trim to closest.** This is the default setting. Clicking a sketch entity will:
  - Trim it back to the next entity if there is only one crossing entity.
  - Trim between two crossing entities if there is more than one.
  - Delete the entity if there are no crossing entities.

In all cases, the selected section of the entity is removed. The Trim to closest option can also extend when you drag one entity to another; if an intersection is possible, the first entity is extended to the second entity. Figure 3.17 illustrates how the Trim to closest option functions.

Using Trim to closest to extend

Construction Geometry toggles between regular sketch entities and construction entities. Construction sketch entities are not used to create solid or surface faces directly; they are only used for reference —for example, revolve centerlines, extrude and pattern directions, and so forth. Be careful with the icon for this function, because it looks almost identical to the No Solve Move icon, especially as printed here in gray scale.
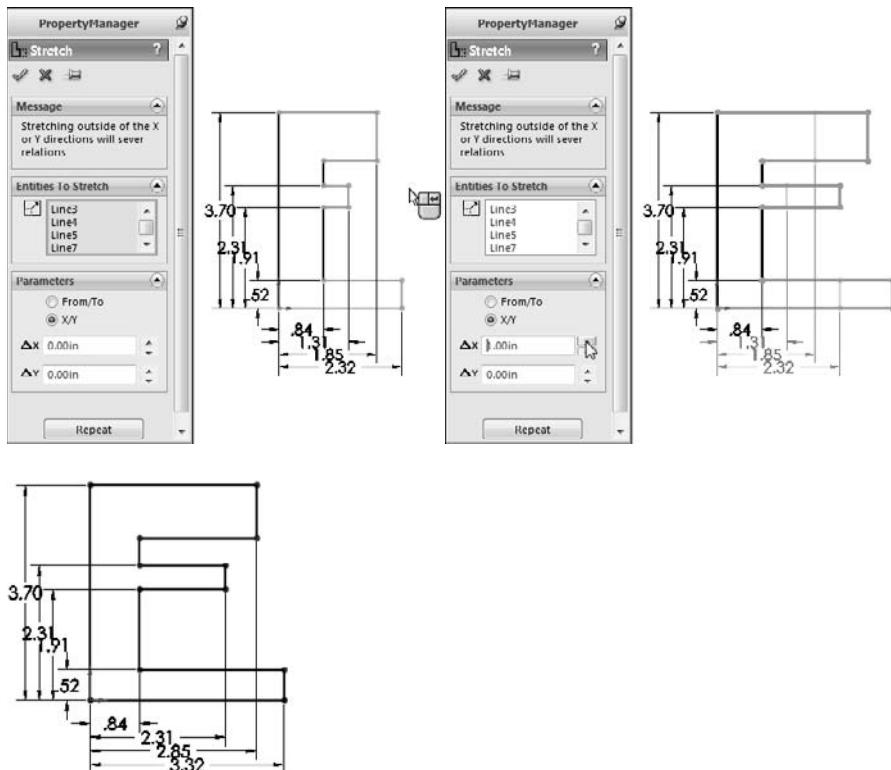
Note
The icons for Hide/Show Edges, No Solve Move, and Construction Geometry look substantially similar, and in this black and white book, they may be indistinguishable. ■

The Stretch sketch tool is intended for use in sketches where there are enough dimensions to make a particular change difficult by changing dimensions only. It is similar in purpose and use to the AutoCAD Stretch function because it was loosely modeled after the AutoCAD functionality. Stretch enables you to specify a change that will change several dimensions simultaneously. Figure 3.18 shows the initial, intermediate, and final states of the sketch being stretched.

FIGURE 3.18

Using the Stretch sketch tool

Tip

The main ideas to remember with the Stretch tool are that it is used to stretch dimensioned lines, and that you need to select the lines that will lengthen or shorten as well as the lines that will move. Because if this, selecting entities for Stretch is best done with the right-to-left window selection, which also selects any items that the selection box crosses. (Left-to-right window selection only selects items that are completely within the selection box.) ■

Caution

Figure 3.18 shows the X/Y option being used, but if you use the From/To option, be aware that it may delete some sketch relations along the way. ■

The Move, Rotate, Copy, and Scale sketch tools operate on selections within a sketch. You can use these tools with pre- or post-selection methods. These tools delete existing sketch relations when necessary to accomplish the task. For example, if you want to move a rectangle connected to the origin, the Move tool will delete the Coincident relation between the sketch endpoint and the origin. If you want to rotate a rectangle, the Rotate tool will delete all the horizontal and vertical relations on the entities being rotated. This operation may result in a completely underdefined sketch. SolidWorks does not warn you that sketch relations are being deleted.

If you use the Scale tool on a fully defined sketch, SolidWorks will scale the position of the selected entities, deleting sketch relations if necessary to do so, but no dimensions will be scaled or deleted.

Caution

Be careful when using these sketch tools. They can delete sketch relations without warning. ■

These sketch tools were originally put in the software to avoid some of the complexities and limitations of the Modify Sketch tool, which can also move, copy, rotate, and scale sketches. Figure 3.19 shows the simple interface for the Move Entities command. Select the entities to move in the upper box and the method to move them below.

Select is usually used to turn off the previous command.

Grid/Snap is used to open the Grid/Snap section of Tools ➪ Options ➪ Document Properties.

Parallelogram can be used in two different ways:
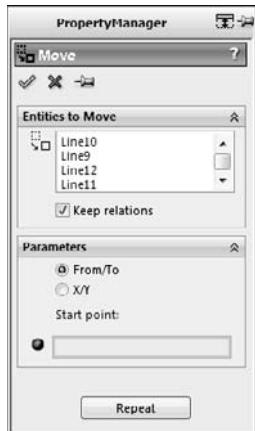
- To draw a rectangle at an angle, click one corner of the rectangle, and then drag a side connected to that corner. Then click and drag the length of the second direction.

- To draw a parallelogram (adjacent sides are not perpendicular, but opposite sides are parallel), draw the first side of the parallelogram in the same way as the first side of the rectangle, but hold down the Ctrl key when dragging the second side; you can set the angle as well as the length.

FIGURE 3.19

The Move Entities interface

Polygon creates a regular *n*-sided polygon in the same way as a circle. Click the center and drag the radius. You need to set the number of sides in the PropertyManager before clicking in the graphics window.

Ellipse is created by clicking the center, dragging one axis, and then dragging the other axis.

Partial Ellipse is created by clicking the center, dragging one axis, dragging the other axis, and then clicking and dragging the included angle of the partial ellipse. The Partial Ellipse feature works like the Centerpoint Arc command.

Parabola is created by clicking the location for the Focus, and then dragging the position of the Apex. You then click and drag the included angle of the parabola, as shown in Figure 3.20. This is a rarely used sketch entity and is often difficult to control with sketch relations or dimensions.
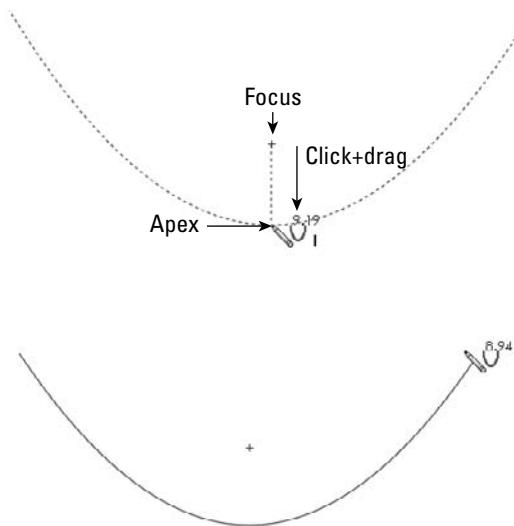
Note
The Parabola is the closest thing SolidWorks has to the conic sketch type found in other popular CAD packages. Few people use the Parabola in SolidWorks. The next closest to the conic is the Partial Ellipse, then the three-point spline. ■

Spline on Surface is used in 3D sketches to draw a freeform spline on any 3D surface. The Spline on Surface feature can cross face boundaries as long as the faces are at least tangent (ideally curvature continuous) across the edge. Spline on Surface can be used to trim surfaces or create split lines.

FIGURE 3.20

Drawing a parabola

Sketch Text creates editable text in sketches using TrueType fonts installed in your Fonts folder. Some fonts produce sketches that are unusable for solid features, due to violating sketch rules with overlapping or zero thickness. You need to be careful which fonts you select, but I have had success with a wide variety of fonts I have found on the Internet. Sketch Text may be dissolved into lines and arcs so that you can edit them manually. Dissolve is available on the RMB menu. Figure 3.21 points out the key elements of the Sketch Text interface.

Note

Note that the Link To Property icon enables you to link sketch text to a custom property, or a configuration specific custom property. Using configurations and properties to drive sketch text can be a valuable function. ■

Intersection Curve, in 2D sketches, creates sketch entities where the sketch plane intersects selected faces. In 3D sketches, the Intersection Curve sketch tool creates sketch entities where any types of selected faces intersect. This can be an extremely useful tool in many situations.
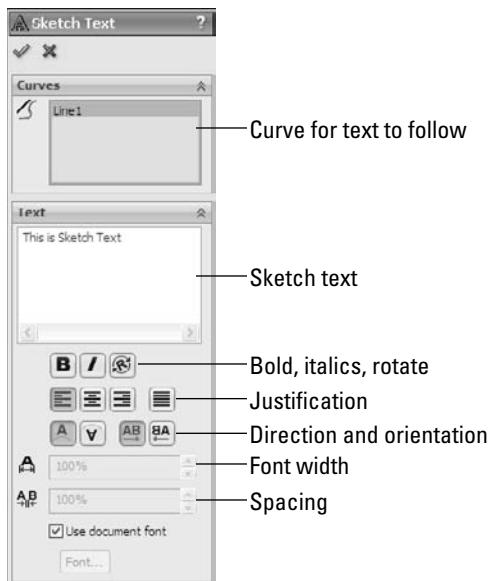
Face Curves applies the underlying U-V isoparameter mesh to a selected face. It is most commonly used as an evaluation tool for complex surfaces, but you can also use it to create curves to rebuild faces. Accepting the results by clicking OK creates a separate 3D sketch for each spline. Figure 3.22 shows the original surface and the results of using face curves on a complex lofted surface.
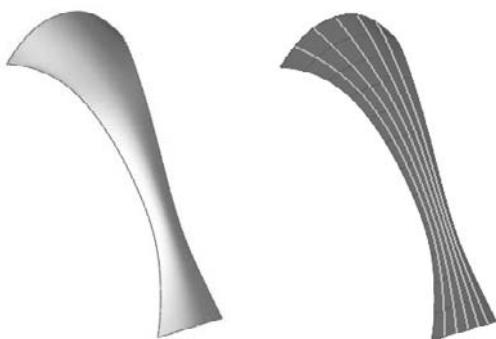
FIGURE 3.21

The Sketch Text interface

Curve for text to follow

Sketch text

Bold, italics, rotate

Justification

Direction and orientation

Font width

Spacing

FIGURE 3.22

Using face curves on a complex surface

Extend extends a sketch entity up to its next intersection with another sketch entity. This is not to be confused with the Extend for surface entities.

Split Entities splits a sketch entity into two segments. You can also delete it later to rejoin the entity back into a single segment. Closed loop entities require at least two split points.

Dynamic Mirror can be used when you pre-select a centerline and Dynamic Mirror is turned on. Any new sketch entity that you draw is automatically mirrored to the other side of the centerline. The ends of the mirror line have hatch marks on them to remind you that you have mirroring turned on.
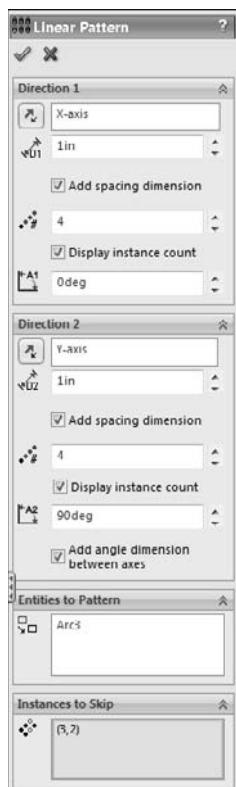
Linear Pattern creates a one- or two-directional pattern of sketch entities. You can define spacing and angles. Figure 3.23 shows the interface and the results of this function.

Circular Sketch Pattern creates a circular pattern of sketch entities.

FIGURE 3.23

The Linear Pattern interface

Best Practice

You should use sketch patterns as little as possible. For many of the same reasons that fillet features are preferred over sketch fillets, pattern features are preferred over sketch patterns. Sketch patterns are not as editable or as flexible as feature patterns. They solve slowly, especially when you pattern many entities. Best practice is to avoid sketch patterns unless there is no alternative. ■
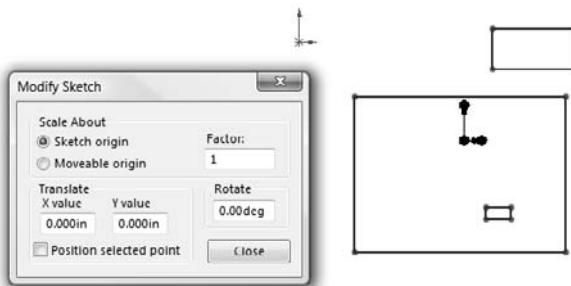
Make Path is intended to help create machine-design motion in sketches, in particular, cam type motion. Although it is helpful, you do not need to make a block of the cam first. You can then right-click the block and select Make Path. A tangent relation to a path enables a follower to roll around the entire perimeter.

Modify Sketch is one of my favorite sketch tools, but it has been falling out of favor in more recent versions of SolidWorks because of the improvements to tools such as Move Sketch. The Modify Sketch feature is flexible and powerful, and enables you to move, rotate, and scale the sketch, as well as mirror about a horizontal or vertical axis or about both axes simultaneously. Figure 3.24 shows the interface, which consists of a dialog box, a special Origin, and a context-sensitive cursor.

FIGURE 3.24

The Modify Sketch interface

Both the left and right mouse buttons have special functions, which change when the cursor is moved over the three knots on the special Modify Sketch Origin. The RMB enables you to mirror or rotate the sketch, and the left mouse button (LMB), enables you to move the Origin or move the sketch.

This function has some limitations when you use it with sketches that have external relations. Certain functions may be disabled or a warning message may appear, saying that you need to remove external relations to get a particular function to work correctly.

No Solve Move enables the moving of sketch entities without solving any relations in the sketch. If you select this option and you move an entity with relations that would otherwise not allow it to
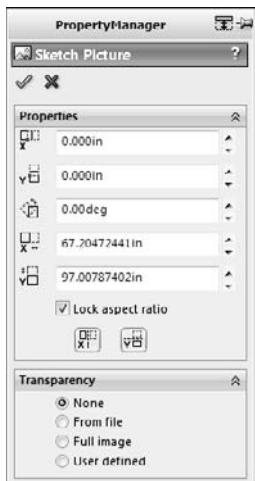
move (such as a collinear relation), you are prompted with a choice to delete the existing relation and continue or copy the entity without the relation. As mentioned earlier, be careful with the icon for this function because it looks almost identical to the Construction Geometry icon, especially as printed here in gray scale.

Sketch Picture is a picture that is placed in the sketch, lies on the sketch plane, and is listed in the FeatureManager indented under the sketch. The Sketch Picture may be suppressed independently from the rest of the sketch, and when the sketch is hidden, the picture is not visible. You can easily move, resize, and rotate Sketch Pictures, as well as apply a transparent background color to them. Sketch Pictures are usually used for tracing over or as a planar decal without the need for PhotoWorks. Figure 3.25 shows the controls for manipulating the Sketch Picture feature.

FIGURE 3.25

The Sketch Picture interface

Equation Driven Curve is a sketch spline driven by either an explicit or parametric equation, as shown in Figure 3.26. An explicit equation is in the form y = f(x), while a parametric equation uses multiple equations driven by a common parameter value of the form, such as

x = cos(t)

y = sin(t)

0>t>pi
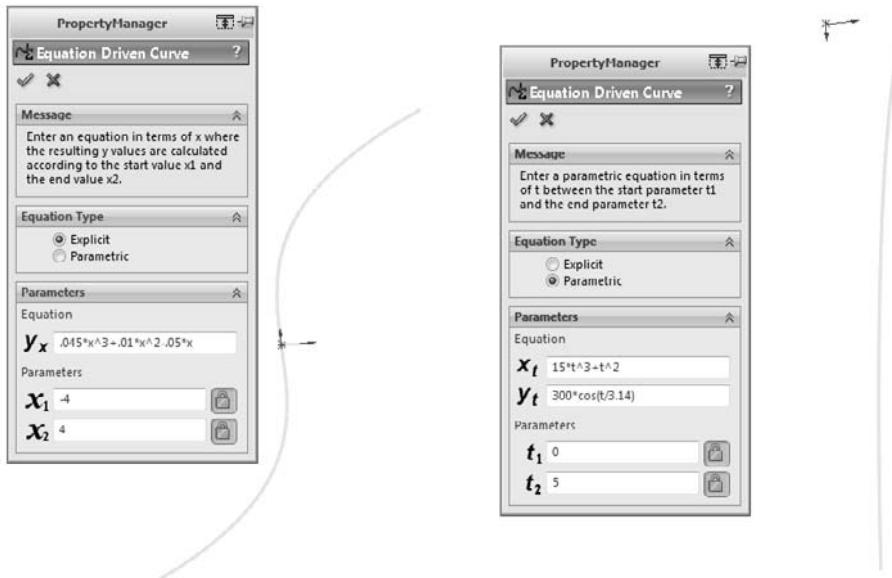
where *t* is a number in radians.

The result is a proportional spline in a sketch, not a curve feature as the name suggests. New in SolidWorks 2010 is the capability to drag the spline itself or its endpoints in 2D or 3D and SolidWorks calculates the new transformation. Previous versions of this function required you to key in deltas for positional and rotational transforms, but 2010 does not allow this, you can only drag, apply sketch relations and dimensions.

The Equation Driven Curve PropertyManager

If you start an Equation Driven Curve in a 2D sketch, you get the form for a 2D curve equation. If you start in a 3D sketch, you get the form for a 3D curve. Once these splines are created, you cannot remove the relation to the equation and manually edit the spline; they are tied to the equation until you delete the entire spline.
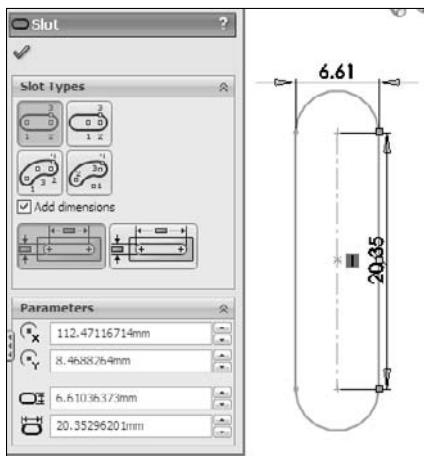
One way to get around this limitation would be to create an equation driven curve in one sketch, and then open another sketch and use convert entities to copy the spline, delete the On Edge relation, and use Simplify Spline to add control points to it. This is a technique commonly used with other types of curves; it does not enable you to update the overall size or shape of the spline through the equation, but you can manually adjust sections of a curve originally created from equations. Examples of where this might be useful would be a lead in or lead out on a cut thread, a special attachment loop in the middle of a spring, or a flare around the edge of a lens or reflector dish for mounting.

Straight Slot and Curved Slot draw slots of a given width and length with full rounds on the ends. All the slot sketch entities can be seen in the PropertyManager shown in Figure 3.27. If you need to draw a composite slot or a slot with multiple entities, you will need to use the bi-directional sketch offset with capped ends mentioned earlier.

FIGURE 3.27

PropertyManager for the slot sketch entities
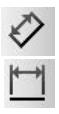
The Dimensions/Relations toolbar
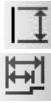
The Dimensions/Relations toolbar has a few tools that you have already seen, but as the name suggests, it also contains tools that will either help you to create or investigate dimensions and sketch relations. Figure 3.28 shows the default toolbar, but in the following pages, you look at all the available tools you can see at Tools ⇨ Customize ⇨ Commands ⇨ Dimensions/Relations.

FIGURE 3.28

The Dimensions/Relations toolbar
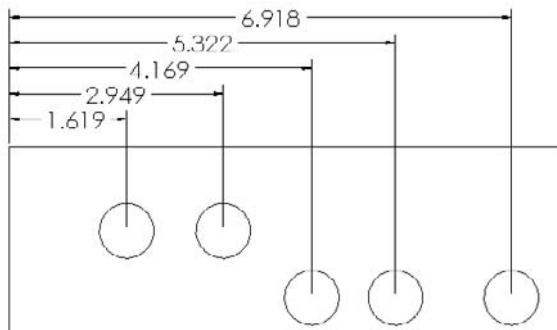
- **Smart Dimension.** Lets you dimension the sketch entity as you sketch.
- **Horizontal Dimension.** Applies a dimension to a sketch entity that drives the horizontal distance between the endpoints.

- **Vertical Dimension.** Works like a horizontal dimension but vertically.

- **Baseline Dimensions.** Creates dimensions only in *drawing* documents. Baseline Dimensions are different from most of the dimension tools that you find on the Dimensions/Relations toolbar in that they can create driven dimensions on view geometry or driving dimensions on sketch geometry in a drawing, but cannot be used on sketch geometry in parts. Baseline Dimensions start from a single reference; then as you select additional references, additional dimensions are stacked (see Figure 3.29).

FIGURE 3.29

Baseline Dimensions on a drawing

- **Ordinate Dimensions.** Drives dimensions where a set of ordinate dimensions originate from a common zero point. To use these dimensions, simply click a zero location, place the zero dimension, and then click additional points. The dimensions are placed and are automatically aligned to the rest of the dimensions.

Note
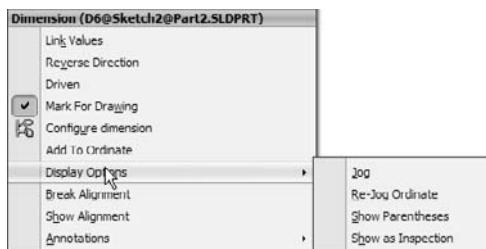
If a line is not selected as the zero reference entity, the Ordinate Dimension feature defaults to a Horizontal Ordinate. ■

You can remove Ordinate Dimensions from the common alignment by right-clicking the dimension and selecting Break Alignment. Ordinate Dimensions will jog automatically if SolidWorks senses the dimensions are getting too close to one another. You can also jog them manually. After you create the Ordinate Dimension set, you can add to it by accessing the Add to Ordinate command through the RMB menu. All the options for Ordinate Dimensions are shown in Figure 3.30.

Not all the listed options are available in the model sketch environment; some are available only in drawings.

FIGURE 3.30

Options for Ordinate Dimensions
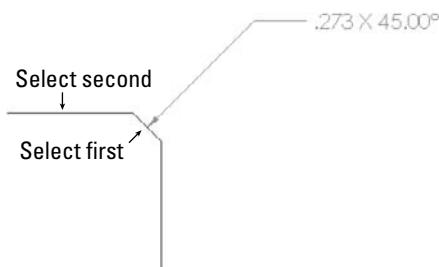
Horizontal and Vertical Ordinate Dimensions have the same function as the regular Ordinate Dimensions, except that they only drive horizontal and vertical dimensions, respectively.

Chamfer Dimension is another type of dimension that is only driven and only applied in drawing documents. It works by first selecting the chamfered edge and then selecting the angle reference edge. It produces dimensions like the one shown in Figure 3.31.

FIGURE 3.31

Applying a chamfer dimension

Automatic Relations toggles to enable or disable the automatic creation of sketch relations while sketching. This toggle is also available through Tools ➪ Sketch Settings ➪ Automatic Relations. Automatic relations help you to create intelligent sketches with less manual intervention. Although using them takes a little practice, it is well worth the effort.
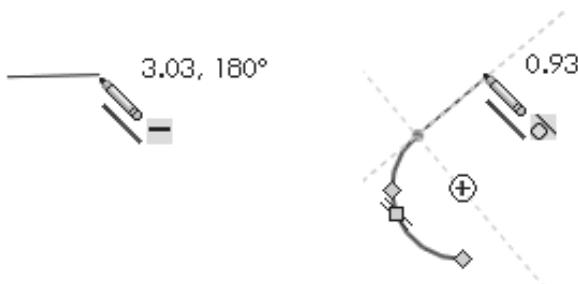
Caution

As with any automatic function, there are times when automatic relations will do things that you do not expect or want. While you are sketching, it is recommended that you watch the cursor and the relations that it automatically applies. ■

While sketching, symbols appear on the cursor to show that a relation will automatically be created. These symbols have a yellow background and will apply horizontal, vertical, coincident, tangent, parallel, and perpendicular relations. Figure 3.32 shows two situations where automatic relations are applied — a horizontal and a tangent relation.

Some automatic relations that appear on the cursor

Inferencing in Sketch

Although SolidWorks users (even experienced ones) often confuse Inferencing and Automatic Relations, these functions are not the same. *Inferencing* refers to the blue dotted lines that display in Sketch mode when the cursor aligns with endpoints, centerpoints, or the Origin. Inferencing does not create sketch relations, with one exception. If both ends of a line pick up the same inference to the same point, SolidWorks will add that inference as an automatic relation.

When the cursor displays a small sketch relation symbol with a yellow background, this means that an automatic relation is going to be applied. If the relation symbol has a white background, the relation is inferenced, but not applied as an actual sketch relation. The symbols with the blue (the color may also be green in SolidWorks 2008 or later) background are relations that have been applied to existing sketch entities. The symbols look the same, regardless of background color. Be aware that differences in versions and differences in color schemes can cause these colors to be different on your system.

Table 3.1 shows the symbols for the various inferences, automatic relation cursors, and applied sketch relations. The difference between the three types is simply the background colors: white, yellow, and blue, respectively.

TABLE 3.1

Symbols

	Along X		Along Y		Along Z
	At Intersection of Two Faces		Coincident		Collinear
	Concentric		Coradial		Equal
	Equal Curvature		Fix		Horizontal
	Intersection		Midpoint		Offset
	On Edge		On Surface		Parallel
	Perpendicular		Pierce		Symmetric
	Tangent		Vertical		Display/Delete Relations
	Fully Define Sketch				

The Fully Define Sketch interface uses sketch relations and dimensions to fully define the active sketch. It enables you to select which types of sketch relations and dimensions will be used to do this. Figure 3.33 shows the Fully Define Sketch interface. Be careful of this icon because it looks almost identical to the Sketch icon.

If you are familiar with older versions, the Fully Define Sketch function was formerly called Auto Dimension and has absorbed the functionality of Scan Equal and Add Relations. This function is very useful when used with imported sketch data. If you do not like the automatic dimensioning scheme, you can at least take advantage of the automatic sketch relations.
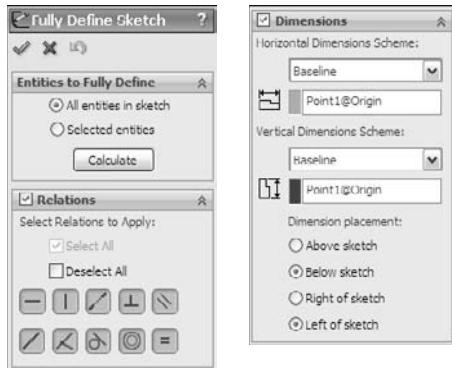
Best Practice
While the Fully Define Sketch function can clearly save you many time dimensioning parts on a drawing, it does not necessarily use the best dimensioning practice for manufacturing drawings or for design intent. This tool is best used in situations when baseline and ordinate dimensions are appropriate. ∎

FIGURE 3.33

The Fully Define Sketch interface

Exploring Sketch Settings

In addition to sketch tools, another important aspect of controlling sketches is sketch settings. Sketch settings are found in two different locations. The first location is at Tools ➪ Options ➪ Sketch. In this chapter, I cover the settings found at the second location, Tools ➪ Sketch Settings. These settings mainly affect sketch relations.

Automatic Solve is turned on by default. As you make changes to a sketch by adding relations or changing dimensions, SolidWorks automatically and immediately updates the sketch to reflect the changes. When the Automatic Solve setting is turned off, these changes are deferred until you exit the sketch or turn the Automatic Solve setting back on. The setting can be useful to prevent inter-mediate solutions (for example, when half of the changes are made) that may cause problems with the sketch, and when you are confident that the outcome will be correct. It is a rarely used option, and you could probably exist just fine without even knowing this option was there at all.

If you import a large drawing from the DXF or DWG format, these drawings import as sketch enti-ties into either a SolidWorks sketch or a drawing. SolidWorks may automatically turn off the Automatic Solve setting for performance (speed) reasons on files of this type.

Enable Snapping is turned on by default. It enables the cursor to snap to the endpoints of existing sketch entities to help you make cleaner sketches. When you turn this setting off, Automatic Relations is also disabled (although the icon for the setting remains depressed, Automatic Relations are not created). Holding down the Ctrl key while sketching disables snapping. Holding down the Ctrl key while dragging sketch entities functions such as copying sketch geometry. The No Solve Move is discussed in the Sketch toolbar section.

Detach Segment on Drag is turned off by default. When you turn this setting on, the Detach Segment on Drag feature enables you to pull a single sketch element away from a chain of elements. For example, if you had a rectangle and you wanted to detach one of the lines from the rest of the rectangle without using this setting, you would have to draw extra geometry and then trim and delete lines in order to release the endpoints.

Best Practice

It is recommended that you leave Detach Segment on Drag off. Turn it on only when you need it, and then immediately turn it off again. This setting can be hazardous for everyday use, because it enables you to simply drag sketch elements that may be otherwise fully defined.■

Override Dims on Drag is off by default. When you turn this setting on, it enables you to drag fully defined sketch geometry, and the dimensions will update to match the dragged size. This is another setting that you should use sparingly. It can be useful for doing concept work, but you should leave it off when working with production data for obvious reasons.

Note

Combining Override Dims on Drag with Instant3D (formerly Move/Size Features) can be very handy for concept work, enabling you to drag model faces and edge easily.■

Using Sketch Blocks

Sketch blocks are collections of sketch entities that can be treated as a single entity, and can be reused within a single document or shared between documents. You can use sketch blocks in parts, assemblies, and drawings. To create a sketch block, select a group of sketch entities and click the Make Block button on the Blocks toolbar, or select Tools ➪ Blocks ➪ Make. Pre-selection is not necessary, you can also select the entities after you invoke the command.

Blocks may be internal to a particular document, or they may be saved as an external file. The externally saved block may be linked to each document where it is used, so that if the block is changed, it updates in the documents where it is used.

You can use blocks in conjunction with the Make Path function mentioned earlier in this chapter to create functional layouts for mechanisms. You can also use blocks in an assembly to build parts in-context. Refer to Chapter 12 for a more in-depth examination of the assemblies aspects of blocks in SolidWorks.

The following is a description of the various tools that are available on the Blocks toolbar:

- **Make Block.** Creates a sketch block from selected sketch entities. You can position a manipulator to denote the insertion point for the block. Blocks may attach at any entity endpoint, but the insertion point follows the cursor.

- **Edit Block.** Enables you to edit an existing block as if it were a regular sketch.

- **Insert Block.** Enables you to select from a list of open blocks or browse to a location where blocks are stored. You can edit the insertion point by using the Edit Block function.

- **Add/Remove.** Enables you to add or remove sketch entities from the block without deleting them from the sketch while editing a block.

- **Rebuild Block.** Allows changes to a block to be reflected in any external relations without exiting the block. For example, if you have a block in a sketch and a sketch line is coincident to one of the endpoints in the block, you may edit the block such that the referenced endpoint moves. As a result, the line in the sketch will not move until you exit the block or use the Rebuild Block function.
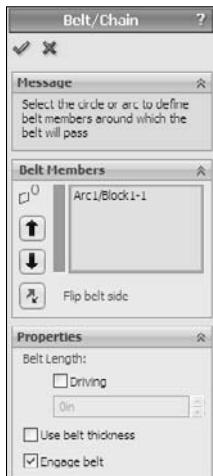
- **Save Block/Save Sketch As Block.** Saves a selected block to an external file (with the *.sldblk extension), or saves the selected sketch as a block.

- **Explode Block.** Removes all the sketch entities from a block and brings them into the current sketch.

- **Belt/Chain.** Enables you to make a belt or chain around a set of pulleys. Each pulley must be a block. After activating the command (by right-clicking on a sketch or block), you can select each pulley and use the arrow on the pulley to switch the side of the pulley to which the belt goes. You can also compensate for the thickness of the belt (this is important when both sides of the belt are in contact with pulleys) and drive the pulley arrangement using the length of the belt. Figure 3.34 shows the Belt/Chain dialog box.

FIGURE 3.34

The Belt/Chain dialog box

Working with Reference Geometry

Reference geometry in SolidWorks is used to help establish locations for geometry that you can't physically touch, such as planes, axes, coordinate systems, and points. You often use reference geometry to establish a characteristic of the finished solid model before the model is created or to include an item that you may want to mate another part to in an assembly later. Mate References are also classified as Reference Geometry, although I do not cover them until Chapter 19, in the course of discussing library parts.

The importance of working with reference geometry becomes obvious in situations where you need to create geometry that doesn't line up with the standard planes. You might use planes to represent faces and axes to represent the centers of holes. Axes are often used to establish a direction, such as in plastic parts where, because of draft, you never truly have any vertical edges; an axis is frequently used to establish the direction of pull for the mold.

Coordinate systems come in handy, especially when translating a part from one system to another for the purpose of machining or some type of analysis. SolidWorks users usually model in such a way that the modeling work is made simpler by the choice of how the part origin is positioned relative to features of the part, but rapid prototyping, machining, mold building, and sheet metal manufacturing applications may have different requirements. As a part modeler, you cannot account for the needs of all downstream applications with your initial choice of origin placement, but you can always create a reference coordinate system for those downstream applications to use.

Creating planes
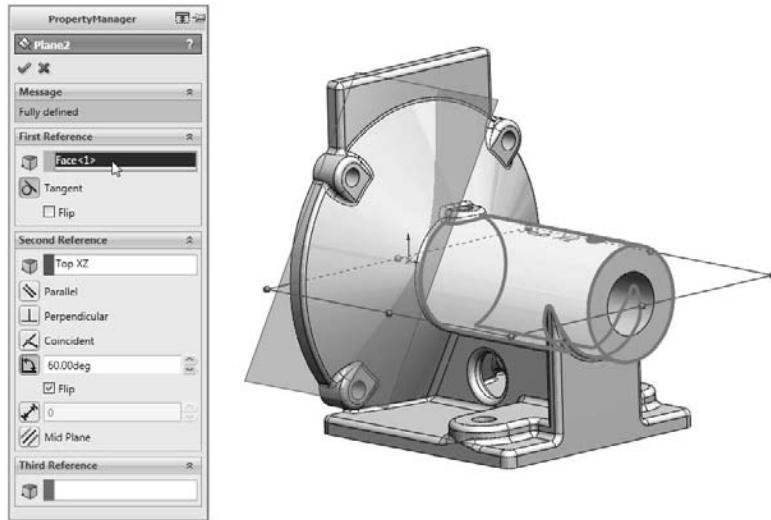
Planes are the most commonly used type of reference geometry because they are used for sketching, cutting, as extrude end conditions, and more. The Plane feature PropertyManager and functionality has changed significantly in SolidWorks 2010. With the new interface, shown in Figure 3.35, you start by selecting model items (faces, edges, points, vertices, or other sketch or reference geometry) that you want to use to create the plane. The new plane uses constraints like sketch relations from the selected references. For example, in Figure 3.35, the new plane is tangent to the selected First Reference cylindrical face and at an angle to the selected Second Reference of a plane.

The good news about this method is that there are far more options for creating planes than in the previous method, but the bad news is that going into the feature, you may not know all the available options. You have to make a selection before it shows you the available constraints. The older interface presented the available options right up front, but there were fewer to choose from. Hopefully this interface matures in the future. Meanwhile, you may need to experiment to see what works best for the type of modeling you do.

Creating a new plane from a set of selections and constraints

Note

When you start using the new Plane functionality, it may be best to try to ignore the new interface and simply make the same types of selections that you made in the past. The new Plane functionality will work perfectly that way. As of this writing, there is not a lot of common user experience with the tool and its available functionality is completely undocumented. ■

Working with axes

Axes are less popular than planes, but equally important. You can use axes to create pivot points in a part where you do not have any hole type geometry for mating with other parts. Axes are used to establish direction frequently. Figure 3.36 shows that the first three features in a plastic part are axes established from the standard planes.
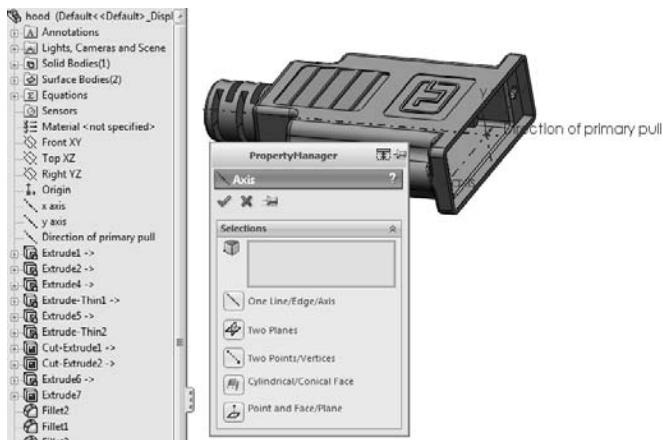
Consider using axes set up in this way as standard features in your template files. They can be effective in assemblies for moving parts in orthogonal directions and in parts for pattern or draft directions.

Note

The interface for the Axis feature has not been changed to match the Plane interface as of pre-release SolidWorks 2010. Notice that the methods for creating an axis are spelled out clearly in the interface itself. I expect to see a change to Axis to make it match the Plane interface. ■

Using axes to establish direction

Using coordinate systems

Coordinate Systems in SolidWorks are primarily used for import and export, but can also be used for mating, mass properties, and other purposes. Coordinate Systems are usually located by a point to determine location or a set of edges to determine direction. Figure 3.37 shows the PropertyManager for assigning a coordinate system, along with the Export Options dialog box, which you can access from the Save As dialog box when the Files of Type drop down is set to an export format.
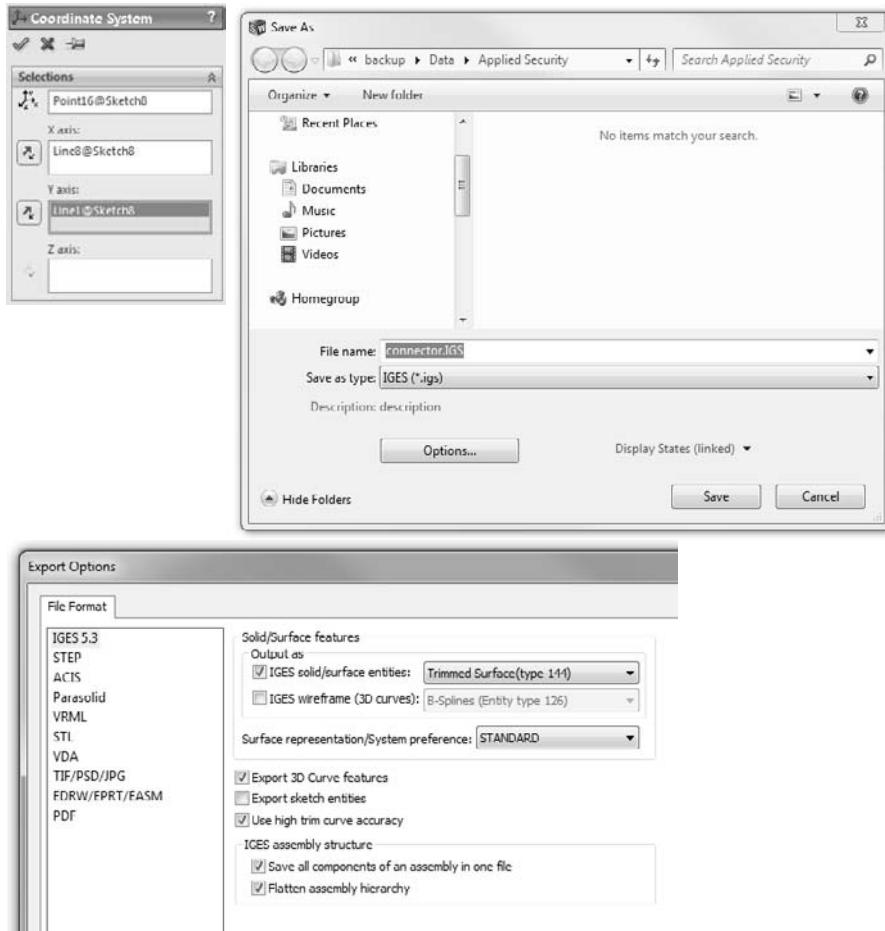
To use the coordinate system when saving as a translated file type such as IGES or Parasolid, click the Options button, as shown in Figure 3.37; at the bottom of the Export Options dialog box, a selection box for Output coordinate system appears.

To use the coordinate system with mass properties, choose Tools ⇨ Mass Properties from the menu, and then select the Output coordinate system, as shown in Figure 3.38.
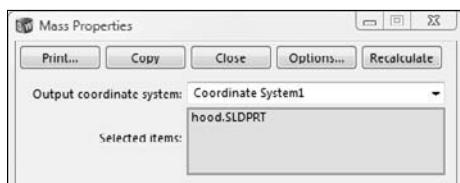
FIGURE 3.37

Use Coordinate Systems to export parts and assemblies with a new origin location.

FIGURE 3.38

Assigning the coordinate system with a well-hidden setting

Using points as reference geometry

✳ The Point reference geometry feature is infrequently used; however for some uses there is nothing else that does the job quite as well. Figure 3.39 shows the PropertyManager for reference geometry Point feature.

The Point reference geometry feature has several modes of point creation you won't find elsewhere.

My favorite Point placement method is the Along Curve Distance option (the bottom one in Figure 3.39). It enables you to create several regularly spaced points along a curve.

Tutorial: Learning to Use Sketch Relations

While it is useful to read through the definitions and functions of all the sketch entities, tools, and relations, using your mouse to create is what this is all about. This tutorial makes sure that you get to know all the major functions in SolidWorks sketches. Almost every part that you build will start with a sketch, so this is a skill worth mastering. Follow these steps to learn about sketch relations:

On the CD-ROM
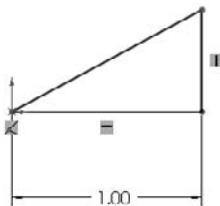
The `BibleInchTemplate.prtdot` template used in this tutorial is in the Chapter 3 folder on the CD-ROM. ∎

1. **Open a new part using a template that you set up in the Template tutorial from Chapter 1.** If you do not have this template, there is one provided for you on the CD-ROM named `BibleInchTemplate.prtdot`. Copy it to your templates folder and use it to create a new part. You may also use a SolidWorks default template.

2. **Select the Front plane in the FeatureManager, and click the Sketch button on the Sketch toolbar.** Click the Line tool from the Sketch toolbar.

3. **Move the cursor near the Origin; the yellow Coincident symbol appears.**

4. **Draw a line horizontal from the Origin.** Remember that there are two ways to sketch the line: Click+click or click and drag. Make sure that the line snaps to the horizontal and that there is a yellow Horizontal relation symbol. The PropertyManager for the line should show that the line has a Horizontal relation. Also notice that the line is black, but the free endpoint is blue (after you hit Esc twice to clear the tool, then clear the selection). This means that the line is fully defined except for its length. You can test this by dragging the blue endpoint.

5. **Click the Smart Dimension tool on the Sketch toolbar, use it to click the line that you just drew, and place the dimension.** If you are prompted for a dimension, type **1.000**. If not, then double-click the dimension; the Modify dialog box appears, enabling you to change the dimension. The setting to prompt for a dimension is found at Tools ⇨ Options ⇨ General, Input Dimension Value.

6. **Draw two more lines to create a right triangle to look like Figure 3.40.** If the sketch relations symbols do not show in the display, turn them on by clicking View ⇨ Sketch Relations. You may want to set up a hotkey for this, because having sketch relations is useful, but often gets in the way. Note that the sketch relation symbols may also be green, depending on how your software is installed.

FIGURE 3.40

Draw a right triangle.
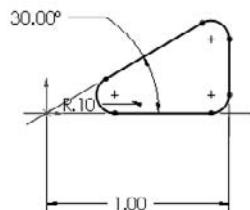
7. **Drag the blue endpoint of the triangle.** Dragging endpoints is the most direct way to change the geometry. Dragging the line directly may also work, but this sometimes produces odd results. The sketch leaves a ghost when dragging so that you can see where you started. Note that the setting for leaving a ghost when dragging a sketch is found at Tools ⇨ Options ⇨ Sketch, Ghost Image On Drag.

8. **Click the Smart Dimension tool, and then click the horizontal line and the angled line.** This produces an angle dimension. Place the angle dimension and give it a value of 30°.

9. **Click the Sketch Fillet tool, set the radius value to 0.10 inches, and click each of the three endpoints.** Where the 1.000-inch dimension connects to the sketch, SolidWorks

has created *virtual sharps*. Figure 3.41 shows the sketch at this point. You may now want to turn off the Sketch Relations display because the screen is getting pretty busy. You can find this setting at View ⇨ Sketch Relations.

FIGURE 3.41
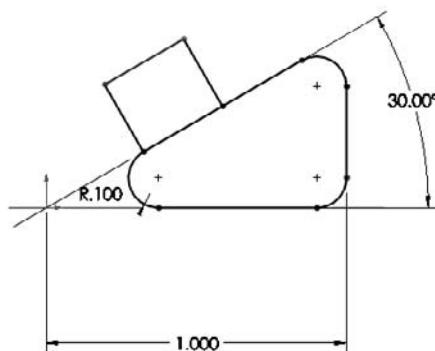
The resulting sketch after you perform Step 9

10. **Draw a line starting from the midpoint of the angled line.** The midpoint should highlight when you move the cursor close to it. Draw the line perpendicular to the angled line. A dotted yellow line appears, showing where the perpendicular lies. When you follow this line, the yellow perpendicular symbol appears on the cursor. Make this line approximately .25 inches long. Feedback on the cursor also shows the length of the line as you draw it.

11. **Draw two more lines ending at the endpoint of the sketch fillet, as shown in Figure 3.42.** Use the Inferencing lines to line up the second angled line with the end of the arc.

FIGURE 3.42
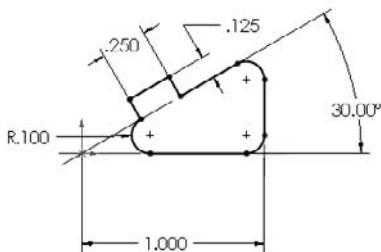
The resulting sketch after you perform Step 11

12. **Click the Trim tool from the Sketch toolbar.** Make sure that the Trim option is set to Closest. Click the angled line of the triangle between the two lines sketched in Step 11. This trims out that section and makes the sketch a single closed loop. A warning may appear because you have a Midpoint relation to the line being trimmed; you no longer want this relation, but you want the lines to intersect at their endpoints. Select Yes at the prompt.

13. **Click the Smart Dimension tool.** Use it to create aligned dimensions on the short line (.25 inches) and one of the long lines (.125 inches). You may now want to reorganize some of the dimensions if the display is becoming crowded.

14. **At this point, two of the lines should be blue, but it may not be clear why they are not defined.** Select one of the blue lines and drag it. Notice that what changes is the arc nearest the Origin. This changes in a way that is not useful for this part. To lock this line where it needs to be, select the blue line nearest the Origin and the centerpoint of the arc nearest the Origin, and give them a Coincident relation in the PropertyManager. The result is a fully defined sketch, as shown in Figure 3.43.

FIGURE 3.43

The resulting sketch after you perform Step 14

15. **Save the part with the name** `Sketch Relations Tutorial.sldprt`. Close the part.

Tutorial: Using Blocks and Belts
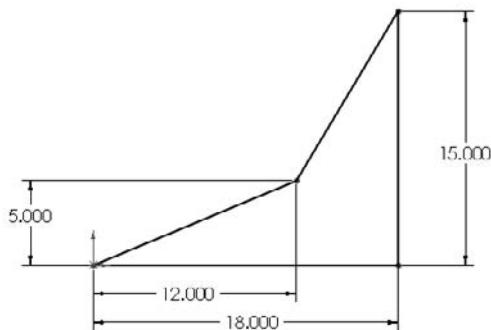
Sometimes I am amazed at the things that can be done in SolidWorks, even with fairly simple tools. This is one of those times. If you design machines, this tutorial will have some extra meaning for you. If you do not design machines, Blocks and Belts are still valuable tools to have in your toolbox ready to use in various situations. Follow these steps to learn about using Blocks and Belts.

1. **Open a new part with inches as the units.**

2. **Draw a sketch on the Front plane as shown in Figure 3.44, with four lines con-nected to the Origin.** Exit the sketch and rename it Layout Sketch, either by clicking twice on the name of the feature in the FeatureManager or by selecting it and pressing F2.

The layout sketch
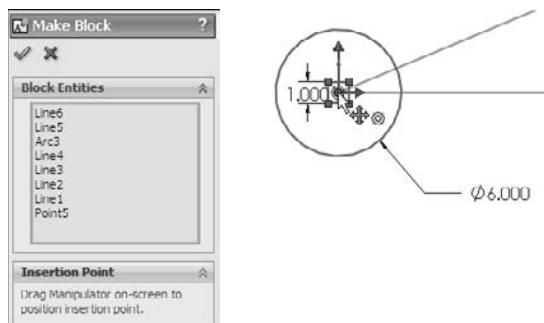
3. **Open a second sketch on the Front plane, and draw a circle centered on the Origin with a 6-inch diameter.**

4. **Inside the circle, draw a Centerpoint rectangle centered on the Origin.**

5. **Select two adjacent sides of the rectangle and make an Equal sketch relation between them.** (This makes the rectangle into a square.)

6. **Click the Smart Dimension tool, and apply a 1.000-inch dimension to one side of the square.** Turn off the Smart Dimension tool by clicking it again on the toolbar or pressing Esc.

7. **If the Blocks toolbar is not active, activate it and then select Make Block.** You can also access this command through Tools ⇨ Block ⇨ Make.

8. **Window select the circle and the square by clicking and dragging a box that includes all the items in the sketch.** The PropertyManager to the left displays a circle and five lines that are to be made into a block.

9. **Expand the Insertion Point panel in the PropertyManager.** This causes a blue manipu-lator Origin to appear in the graphics window. Click this Origin and drag it onto the cen-ter of the circle. Then click the green check mark icon to exit the Make Block dialog box. This is shown in Figure 3.45.
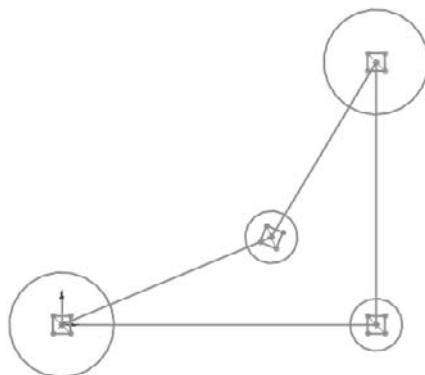
Creating a block

10. **The items in the block now turn gray.** Click anywhere on the block and drag it out of the way. Then drag the center of the circle and drop it on the part Origin.

11. **Click the Insert Block tool on the Blocks toolbar.** Place the block on the opposite sharp corner of the layout sketch.

12. **Create another block that is identical to the first one, except that it has a diameter of 3 inches instead of 6 inches.** You can do this by selecting the first block, clicking Edit Block from the toolbar, and copying (Window select and Ctrl+C). Then exit the Edit Block and paste (Ctrl+V) in the regular sketch. Make sure to also change the insertion point for this second block to the center of the circle.

13. **Insert a second instance of this second block, and make sure that both of them have the center of the circle at the two remaining intersection points of the four-sided shape of the layout sketch.** At this point, your sketch should look like Figure 3.46.

14. **Click the Belt/Chain tool on the Blocks toolbar.** Select the blocks in counterclockwise order, starting at the Origin. On the last pulley, you will have to click the arrow to get the belt to go the correct way around the pulley.

15. **Make sure that the Engage Belt option is selected.** This enables you to make the pulleys move in the same way that they would in a real belt-driven mechanism.

16. **Click the Use Belt Thickness option and assign .25 inches for the thickness.** The belt should be offset from the pulleys.

17. **Click the green check mark icon.**

18. **Click and drag one of the corners of the square in a pulley.** All the pulleys should turn as if this were a real mechanism. The ratios are also observed because the small pulleys rotate faster than the large ones.

19. **Save this part as** `Blocks and Belts Tutorial.sldprt.` Exit the part.

FIGURE 3.46

Block placement

Tutorial: Reference Geometry

This tutorial steps you through creating reference geometry on an existing part in preparation for locating 3D features.

On the CD-ROM

The `Chapter 3 Reference Geometry.SLDPRT` **file (or drawing) used in this tutorial is in the Chapter 3 folder on the CD-ROM.** ■

1. **Open the file from the CD-ROM in the Chapter 3 folder called** `Chapter 3 Reference Geometry.sldprt`.

2. **In the FeatureManager filter, type** plane1. Double-click Plane1 in the FeatureManager, then double-click the 3.25-inch dimension on the screen and change it to 3.35 inches. Click the rebuild symbol (traffic light) and watch the update. This plane locates the mounting base of the part.

   When you are done, click the X at the right end of the FeatureManager filter to clear the result. Pressing Esc does the same thing.

3. **Click the Axis toolbar button from the Reference Geometry flyout menu (on the Features tab of the CommandManager in a default install).**
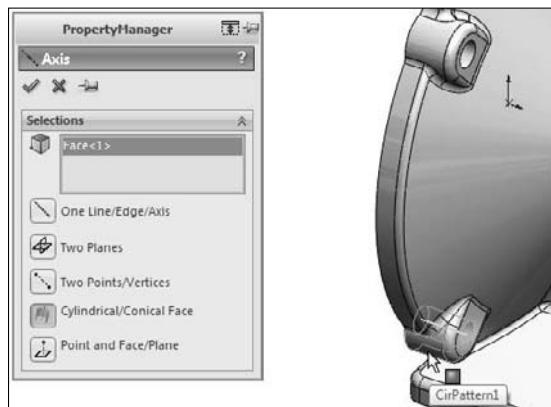
4. **Select the inside face of a hole on the part, as shown in Figure 3.47.** This creates an axis on the centerline of the hole. You should note that a temporary axis already exists for all cylindrical faces, but making a true axis feature helps this one stand out as different from the other holes on the part.

FIGURE 3.47

Making one hole stand out by creating an axis feature

The selection of the Cylindrical/Conical Face option should be automatically activated by your selection of the cylindrical face of the hole. Accept the result with the green check mark when the selections and settings are complete.
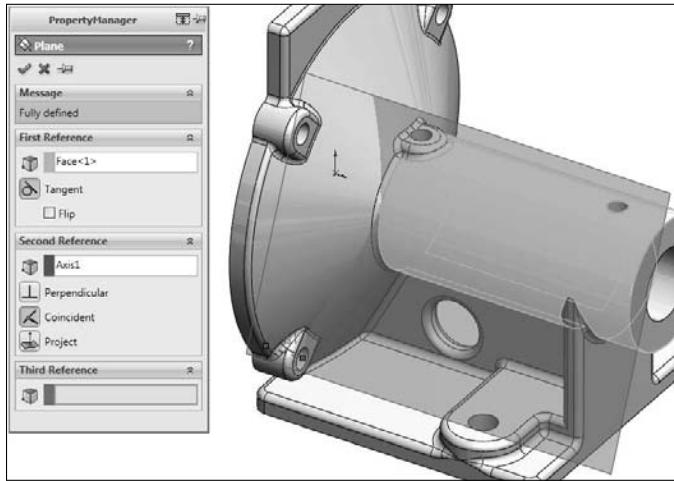
5. **Click the Plane toolbar button from the Reference Geometry flyout on the Features tab of the CommandManager.**

6. **Select the large cylindrical face of the part as the First Reference and the axis you just created as the Second Reference.** Make sure the First Reference uses the Tangent constraint and the Second Reference uses the Coincident constraint. This makes a plane tangent to the main cylinder in the part that goes through the patterned hole, as shown in Figure 3.48.

   Click the green check mark to accept the result.

7. **Open a new sketch on the new plane.**

8. **Click the View menu and activate the Temporary Axes option.** You should now see blue axes (without names) appear along the centerlines of every conical or cylindrical face on the model (except for faces created by fillet or chamfer features).

Creating a plane from tangent and coincident constraints

Note

If the view is already normal to the selected plane, and you double-click Normal To, the view switches to 180 degrees opposite. For this exercise, it doesn't matter which way you view the part, but from the top is better than from the bottom because the view is clearer. ∎

9. **Select the sketch plane either from the graphics window or the FeatureManager. Press the Spacebar on the keyboard and double-click Normal To.** There are other ways to access this command, but this is the method that will work regardless of your interface setup. You can use whatever method you prefer.

10. **Use the Centerpoint Rectangle to create a rectangle centered around the temporary axis of the large cylindrical face.** Make sure that the centerpoint, which is the first click you make, picks up a coincident automatic relation with the temporary axis. Make sure the second click to place the corner of the rectangle does not pick up any automatic relations.

    You can tell if any relations have been applied if you go to the View menu and activate Sketch Relations. Make sure the centerpoint has a coincident relation and that none of the corners has any relations.

Note

The temporary axis is not on the same plane as the sketch plane, so if the view is not normal to the sketch plane, picking up an automatic relation between the centerpoint of the rectangle and the temporary axis will be difficult. ∎
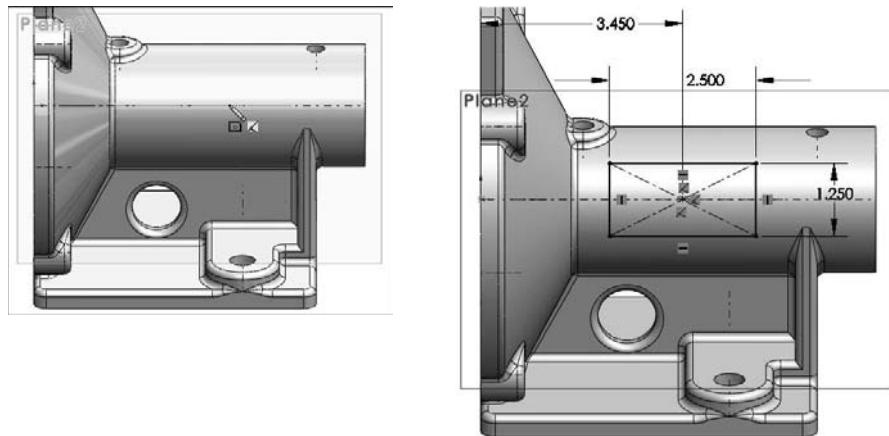
11. Use the Smart Dimension tool to apply dimensions, as shown in Figure 3.49. Note that the 3.450-inch dimension goes to the part origin on the left. You can select this from either the graphics window or the FeatureManager.

Dimensioning the new sketch

Best Practice
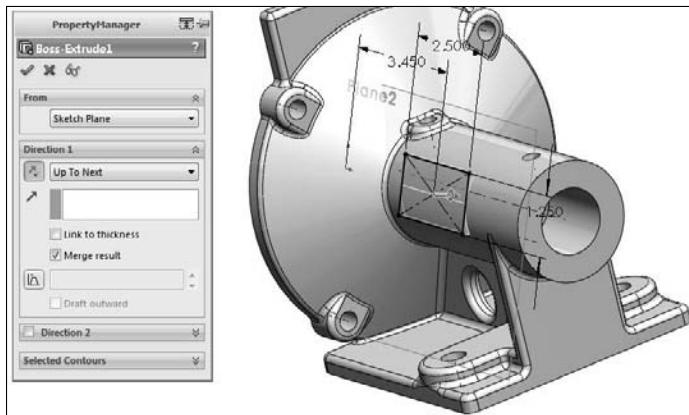
It is best practice to dimension or create sketch relations to items that have the fewest other relations. You should try to use the part origin and standard planes when possible. Dimensioning to reference geometry is better than dimensioning to model edges, although this is not always possible. Experienced users will immediately recognize the need for removing layers of references to prevent restrictive parent-child relations and broken or dangling relations. For beginning users, after you have some experience with making changes to models where relations have been applied carelessly, being more selective with sketch and dimension references will look more attractive to you. ∎

12. Click the Extrude toolbar button on the Features tab of the CommandManager. Rotate the model (by dragging with the mouse wheel depressed. Click the middle mouse button and drag) slightly so you can see the side of the extrusion preview, as shown in Figure 3.50.

You may have to adjust the direction of the extrude using the icon with the arrows just below the Direction1 heading in the Boss-Extrude PropertyManager.

13. Use the Up To Next end condition, which makes the sketch go up to the next solid that it encounters.

14. Use the View menu to turn off the display of Axes, Temporary Axes and Planes.

Creating the extrusion from the new sketch

Summary

Sketching in SolidWorks is something that you will do almost every time you open the software. A lot of automated functions are available that you can allow to do much of the work for you. You also have a lot of control to make changes manually. Remember that the best way to create most sketches is to use automatic relations when you can, sketch the approximate shape that you want to make, and then either drag it to pick up automatic relations, add dimensions, or add relations manually.

The options for creating intelligent relationships that establish your design intent, as well as SolidWorks's capabilities in laying out mechanisms, is only limited by your imagination. The more familiar you become with the tools in your toolbox, the more of a craftsman you can become with this software.

Reference geometry is an essential part of creating and controlling relationships in any parametric model. Reference geometry is usually more stable than solid geometry, so sketch relations and dimensions should use reference geometry as references when possible.

Creating Simple Parts, Assemblies, and Drawings

Good modeling practice is based on robust design intent. This just means that you should try to build parts that can adapt easily to changes. This section of the book begins with questions that you need to ask to model effectively.

Beginning to create simple parts will help you understand techniques used in more complex modeling projects. Learning on simple tools and then expanding your skills helps you to understand best practice issues, which makes you a better contributor to a team environment.

Discovering Design Intent

By asking questions about the part's function before you start modeling or designing, you can create a model that will be easier to edit, easier to properly place into an assembly, easier to detail in drawings, and easier for other SolidWorks users to understand when someone else has to work on your models. Whether you are doing the modeling for someone else or doing the design and modeling for yourself may make a difference in how you approach the modeling task.

The purpose of these questions is to help you establish design intent. The term *design intent* is a statement of how the part functions and how the model reacts to modeling changes.

It may help if you try to put the design intent into words to help you focus on what is important in the design. An example of a statement of design intent is "This part is symmetric about two planes, is used to support a 1.00" diameter shaft with a constant downward load of 150 pounds using

a bronze bushing, and is bolted to a plate below it." This does not give you enough information to design the part, but it does give you information about two surfaces that are important (a hole for the bushing and the bottom that touches the mounting plate), as well as some general size and load requirements. The following questions can help you develop the design intent for your own projects.

Is the part symmetrical?

Symmetry is an important aspect of design intent. Taking advantage of symmetry can significantly reduce the time needed to model the part. Symmetry can exist on several levels:

- Sketch symmetry
- Individual feature symmetry
- Whole-part symmetry
- Axial symmetry (a revolved part)
- "Almost" symmetry (the whole part is symmetrical, except for a few features)
- Left and right symmetrical versions of the part
- Assembly symmetry

What are the primary or functional features?

This is probably the most important question. Primary or functional features include how the part mounts or connects to other parts, motion that it needs to accommodate, and additional structure to support loads.

Often it is a good idea to create a special sketch as the first feature in the part that lays out the functional features. This could be as simple as a straight line to denote the bottom and a circle to represent the position and size of a mating part, or as complex as full outlines of parts and features from all three standard planes. This technique is called a *layout sketch*, and it is an important technique in both simple and complex parts. You can use layout sketches for anything from simply drawing a size-reference bounding box to creating the one point of reference for all sketched features in the part. You can use multiple layout sketches if a single sketch on one plane is not sufficient.

In what ways is the part likely to change?

When the marketing department gets out of their meeting at 4:45 pm, what changes do you need to be prepared for so that you can still be out the door by 5:00 pm? No one expects you to be able to tell the future, but you do need to model in such a way that your model easily adapts to future changes. As you gain experience with the software and keep this idea in mind, you will develop some instincts for the type of modeling that you do.

What is the manufacturing method?

Modeling for the casting process is very different from modeling for the machining process. When possible, the process should be evident in your modeling. There are times when you will not know which process will be used to create the part when you start to create a model. If you are simply making an initial concept model, you may not need to be concerned about the process. In these cases, it may or may not be possible to reuse your initial model data if you need to make a detailed cast part from your non-process-specific model. Decisions like this are usually based on available time, how many changes need to be made, and a determination of the risk of making the changes versus not making the changes as well as which decision will cost you the most time in the long run.

Sometimes it makes sense to allow someone else to add the manufacturing details. A decision like this depends on your role in the organization and your experience with the process compared with that of other people downstream in the manufacturing process. For example, if you are not familiar with the Nitrogen Gas Assist process for molding polypropylene, and you are modeling a part to be made in that process, you might consider soliciting the help of a tooling engineer or passing the work on to someone else to add engineering detail.

Best Practice

As engineers, we are typically perfectionists. However, there always needs to be a balance between perfection and economy. Achieving both simultaneously is truly a rare event. Still, you should be aware that problems left by the designer for other downstream applications to solve (such as machining, mold making, and assembly) also have an impact on the time and cost of the project.

The best practice in this case is a judgment call. When faced with assembling a model sloppily or remodeling it perfectly, I usually choose to remodel because doing it the second time is always faster. In addition, if additional changes are required, you do not need to struggle with the sloppily assembled model. You can easily copy sketches from one part to another, while keeping the old part open as you build the new part. As a result, you may be surprised how quickly things go. ∎

Will there be secondary operations?

When working with any manufacturing process, some secondary processes are generally required. For example, if you have a cast part, you may need to machine the rough surface to create a flat face in some areas. You may also need to ream or tap holes. In plastic parts, you may need to press in threaded inserts.

SolidWorks includes special tools you can use to document secondary operations:

- **Configurations.** This SolidWorks technique enables you to create different versions of a part. For example, one configuration may have the features for the secondary operations suppressed (turned off) and showing just the part as cast, while the other configuration shows the part as machined.

Cross-Reference

Chapter 10 discusses configurations and feature suppression in depth. ∎

- **Insert Part.** This SolidWorks technique enables you to use one SolidWorks part as the starting point for a second part. For example, the as-cast part has all the features to make the part, but it is inserted as the first feature in the as-machined part, which adds the cuts required by the machining process.

Will there be other versions?

Sometimes size-based versions of parts have to be created or versions based on additional features. If these are simple, they can also be handled with configurations, but you need to plan this flexibility in advance.
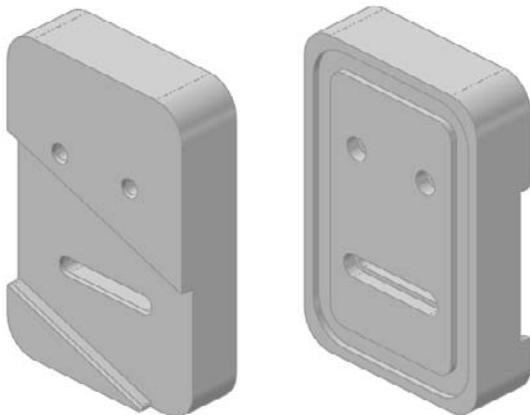
Creating a Simple Part

You need to practice some of the skills you will learn on simple parts. Chapter 2 introduced the tools and features you will use, and this chapter teaches you how to string the simple features together intelligently. In this section, I'll show you how to build the simple part shown in Figure 4.1. While the shape is simple, the techniques used and discussed here are applicable to a wide variety of real-world parts. The discussion on how to model the part contains information on some of the topics you need to understand in order to do the work.

FIGURE 4.1

A simple machined part

Deciding where to start

The first feature that you create should be positioned relative to the Origin. Whether there is a corner of a rectangle that is coincident to the Origin, the rectangle is centered on the Origin, or dimensions are used to stand the rectangle off from the Origin at some distance, you need to lock the first feature to the Origin with every part you build.

When working with a simple part, the entire part can usually be described as rectangular or cylindrical. In cases like these, it is easy to know where to start: you simply draw a rectangle or a circle, respectively. On complex parts, it may not be obvious where to start, and the overall part cannot be said to have any simple shape. In cases like these, it may be best to select *the* (or *a*) prominent feature, mounting location, functional shape, or focus of the mechanism. For example, if you were to design an automobile, what would you designate as the 0,0,0 Origin? The ground might be a reasonable location as would the plane of the centers of the wheels. As long as everyone working on the project agrees, many different reference points could work. With that in mind, it seems logical to start the rectangular part by sketching a rectangle. Select the Top plane and sketch a rectangle centered on the part Origin.

Building in symmetry

Your next decision is about part symmetry. This part is not completely symmetrical: modeling a quarter of it and mirroring the entire model twice is not going to be the most effective technique. Instead, you should build the complete part around the Origin and mirror individual features as appropriate. To start this type of symmetry, you need to sketch a rectangle centered on the Origin. The centered rectangle is something that you will create frequently.

Figure 4.2 shows a centerpoint rectangle that has been sketched with the centerpoint at the part Origin. This creates symmetry in both directions. You can use additional construction geometry and sketch relations to make the rectangle only symmetrical side to side.

Tip

To make a rectangle work like a square, use an Equal sketch relation on two adjacent sides. This only requires a single dimension to drive the size of the square. ∎

Beginning with the rectangle you sketched in the previous section, apply one horizontal dimension by clicking the Smart Dimension tool on a single horizontal line, placing the horizontal dimension (4.00 inches), by clicking a vertical line, placing the vertical dimension (6.00 inches). The sketch is fully defined at this point because both the size and position of the rectangle have been established.

Best Practice
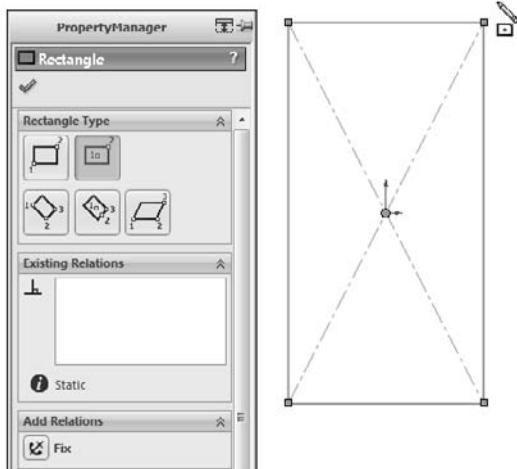
If you are dimensioning a horizontal line, the best way to do it is to simply select the line and place the dimension. Selecting the line endpoints can also work, but selecting the vertical lines on either side of the horizontal lines is not as robust. The problem is that if you use this third method, deleting either of the vertical lines causes the dimension to be deleted. In the first two dimensioning methods, dimensions are not deleted unless you remove one of the endpoints, which requires deleting two lines: the horizontal line and one of the vertical lines. ∎

FIGURE 4.2

Using a center point rectangle to build symmetry about the Origin
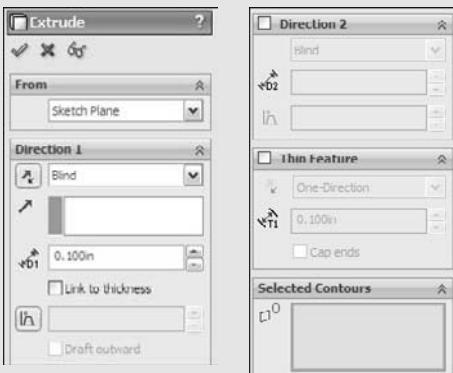
Making it solid

Next, click Extrude in the Features toolbar or choose Insert ⇨ Boss/Base ⇨ Extruded. In the Direction 1 panel, select Mid Plane as the end condition. SolidWorks takes the distance that you entered and extrudes it symmetrically about the sketch plane. Enter **1.00 inch** as the distance.

Extrude feature options

The Extrude feature is one of the staples of SolidWorks modeling. Depending on the type of modeling that you do, the Extrude feature may be one of your main tools.
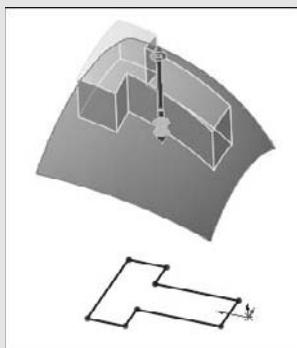
The Extrude interface

Extruding from a selection

The From panel establishes where the Extrude feature starts. By default, SolidWorks extrudes from the sketch plane. Other available options include:

- **Surface/Face/Plane.** The extrude begins from a surface body, a face of a solid, or a reference plane.

Extruding from a surface

Cross-Reference

Surface features are discussed in detail in Chapter 27. ∎

- **Vertex.** The distance from the sketch plane to the selected vertex is treated as an offset distance.
- **Offset.** You can enter an explicit offset distance, and you can change the direction of the offset.

Direction 1 and Direction 2

Direction 1 and Direction 2 are always opposite one another. Direction 2 becomes inactive if you select Mid Plane for the end condition of Direction 1. The arrows that display in the graphics window show a single arrow for Direction 1 and a double arrow for Direction 2. For the Blind end condition, which is described next, dragging the arrows determines the distance of the extrude.

Each of the end conditions is affected by the Reverse Direction toggle. This toggle simply changes the default direction by 180 degrees. You need to be careful when using this feature, particularly when using the Up to end conditions, because if the entity that you are extruding up to is not in the selected direction, an error results.

Following is a brief description of each of the available end conditions for the Extrude feature:
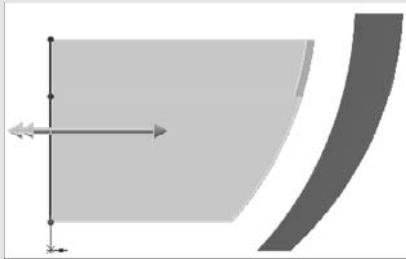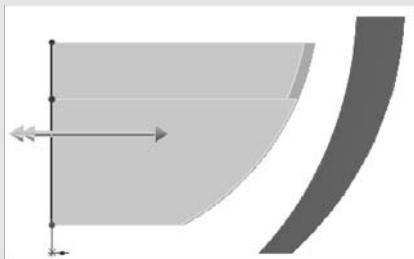
- **Blind.** Blind in this case means an explicit distance. The term is usually used in conjunction with holes of a specific depth, although here it is associated with a boss rather than a hole.
- **Up to Vertex.** In effect, Up to Vertex works just like the Blind end condition, except that the distance is parametrically controlled by a model vertex or sketch point.
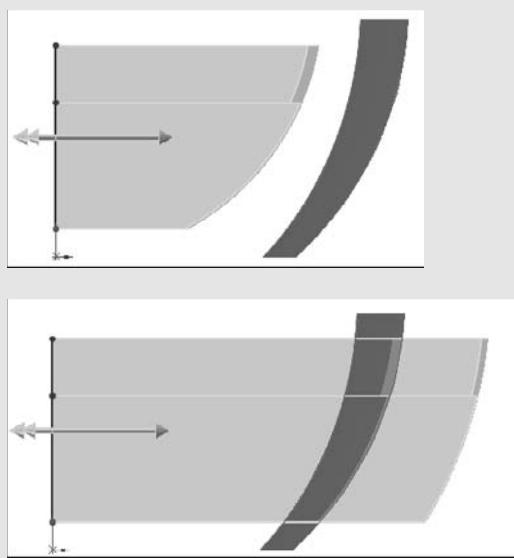
continued

- **Up to Surface.** Up to Surface could probably be better named *Up to Face*, because the end does not necessarily have to be an actual surface feature. This end condition may display a warning if the projection of the sketch onto the selected face extends beyond the boundary of the face. In that case, it is advisable to knit several faces together into a surface body and to use the Up to Body end condition.

- **Offset from Surface.** By default, Offset from Surface extrudes until it reaches a specified distance from a selected surface. There are two methods for determining the type of offset and one to determine direction.

  - The default offset method behaves as if the selected surface were offset radially, so that a surface with a 4-inch radius and a 1-inch offset would give a curvature on the end of the extrude of a 3-inch radius.

  - The second method, called Translate Surface, behaves as if the surface were *moved* by the offset distance.
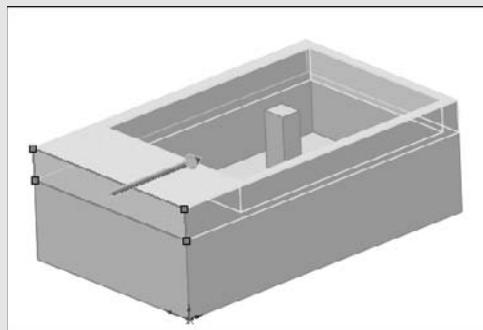
Offset from surface using the default and Translate Surface options

  - Reverse Offset refers to when the offset stops short of the selected face or when it goes past it.

- **Up to Body.** The Up to Body end condition is very useful in many situations, especially when receiving the error message, "The end face cannot terminate the extrusion," from the Up to Face end condition.

- **Mid Plane.** The Mid Plane end condition eliminates the Direction 2 options and divides the extrude distance equally in both directions; for example, if you specify a 1.00-inch Mid Plane, SolidWorks extrudes .50 inches in one direction and .50 inches in the other direction.

The Reverse Offset option

- **Through All.** The Through All end condition is available only when there is already solid geometry existing in the part. When used for an extruded boss (which adds material), it extrudes to the distance of the farthest point of the solid model in a direction perpendicular to the sketch plane. When used for a cut, it simply cuts through everything.

- **Up to Next.** Up to Next extrudes the feature until it runs into a solid face that completely intercepts the entire sketch profile. If a portion of the sketch hangs over the edge of the face, the extrude feature will keep going until it runs into a condition that matches that description, which may be the outer face of the part in the direction of the extrusion.

The Up to Next end condition used with a Cut extrude

continued

By default, the Direction of Extrusion is normal to the sketch plane, but you can also select a linear entity such as an edge or axis as the direction. All the end-condition options are still available when you manually define the Direction of Extrusion as something other than the default.
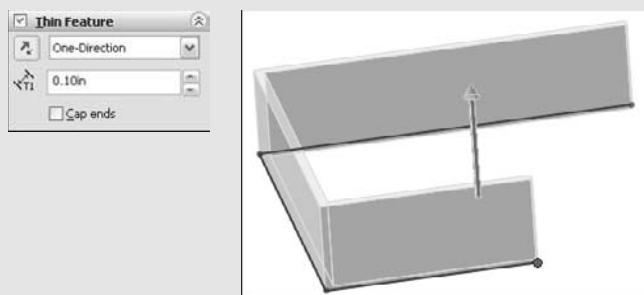
You can also assign a draft option to an extrusion as it is created, and you can control the draft separately for Direction 1 and Direction 2.

Best Practice

When dealing with molded or cast parts, certain types of features, such as draft, fillets, and shells, are often the targets of users trying to assign best practices. This is partially because using draft, fillets, and shells is very much like playing Rock, Paper, Scissors; the only way to win this game is by luck. Arranging the features in the correct order so that the model is efficient and achieves the desired results is challenging. It is usually best to apply draft as a separate feature rather than using it in the definition of the Extrude feature. It is also best to apply draft after most of the modeling is done, but before you apply the cosmetic fillets and before you use the shell feature. ■

Thin Feature panel

The Thin Feature panel is activated by default when you try to extrude an open loop sketch (a sketch that does not fully enclose an area). The end-condition options remain the same. What changes is the feature applies a thickness to the sketch elements in the manner of a sheet metal part, thin-walled plastic part, or a rib. The Thin Feature panel of the Extrude PropertyManager, along with a representative thin feature extrusion.

The Thin Feature panel and a thin feature extrusion

The Cap Ends option is available only when you specify a Thin Feature to be created from a closed loop sketch. This creates a hollow, solid body in a single step. You can also use Thin Features with cuts, and they are very useful for creating slots or grooves.

Contour selection

Contour selection enables you to select areas completely bounded by sketch entities for use with features such as Extrude. For example, you could use a sketch like the number sign (#). Although it breaks the rule and cannot be used with regular extrusion methods, you can use contour selection to select the box in the center, which is completely bound by sketch elements.

SolidWorks works best when the sketches are neat and clean, when nothing overlaps, and when no extra entities exist. However, when you need to use a sketch that does not meet these criteria, you can use this alternative method.

Best Practice

I believe this feature was introduced into SolidWorks only to keep up with other CAD packages, not because it is a great feature. I do not recommend using contour selection on production models. It is useful for creating quick models, but the selection is too unstable for any data that you may want to rely on in the future. The main problem is that if the sketch changes, the selected area may also change, or SolidWorks may lose track of it entirely. ∎

Using Instant 3D

Instant 3D enables you to pull handles to create extrusions and to drag model faces to change the size and location of features. Several feature types enable you to use arrows to adjust elements visually of parametric features and sketches. The function largely replaces and expands on the older functionality called Move/Size Features. Figure 4.3 shows the arrows added by Instant 3D, which are the handles that you pull on to create a solid from a sketch or edit an existing feature. Notice also that you can make cut features with Instant 3D. In fact, you can change a boss feature into a cut. I'm sure this is a neat sales demo trick, but I'm not aware of any practical application of changing a boss into a cut.

One of the attractive things about Instant 3D is that it allows you to make changes to parts quickly without any consideration for how the part was made. For example, the cylindrical part was made from a series of extrudes, with a hole cut through it with draft on the cut feature. The flat faces can be moved, and the cylindrical faces offset. Behind the scenes, SolidWorks figures out which sketches or feature parameters of which features have to be edited. This saves you time searching the FeatureManager to figure out which features or sketches control a given face. As you work through more complex parts, you will see how handy this can be at times. You can activate or deactivate Instant 3D using the icon on the Features toolbar.
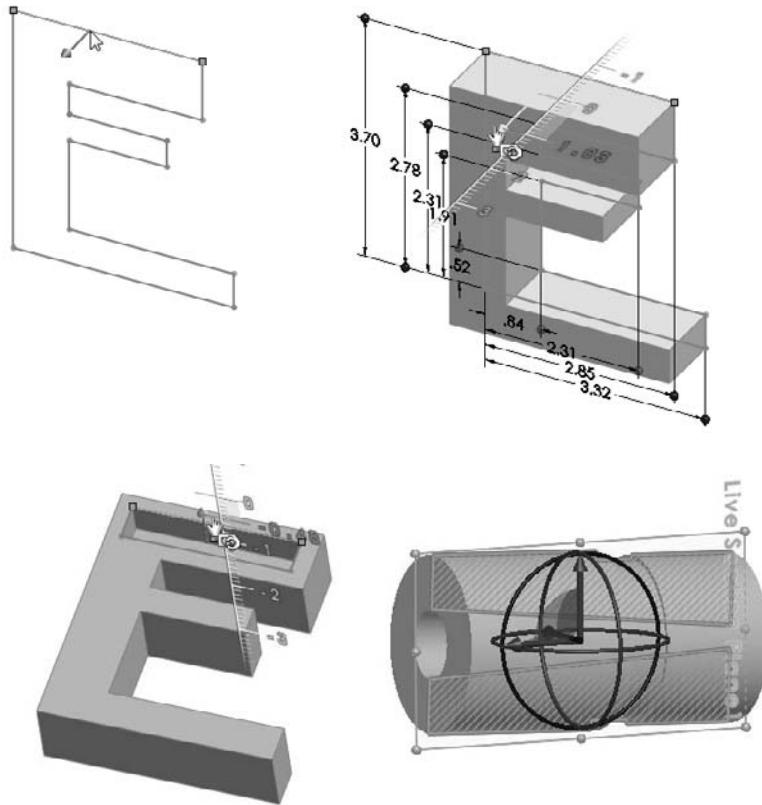
Note

When combined with the sketch setting Override Dims on Drag, Instant 3D can be a powerful concepting tool, even on fully dimensioned sketches. ∎

FIGURE 4.3

Using Instant 3D and Live Section

Instant 3D also offers a tool called Live Section. Live Section enables you to section a part with a plane, and you can drag the edges of the section regardless to which features the edges belong. To activate Live Section, right-click a plane that intersects the part and select Live Section Plane.

Instant 3D can also be an effective tool when used in conjunction with the direct editing type of tools such as Move Face. In fact, Instant 3D mimics some of the direct edit type of functionality found in applications such as Sketchup and Spaceclaim.

Chapter 30 discusses the direct edit theme in more detail, and revisits the Instant 3D manipulators in that light.
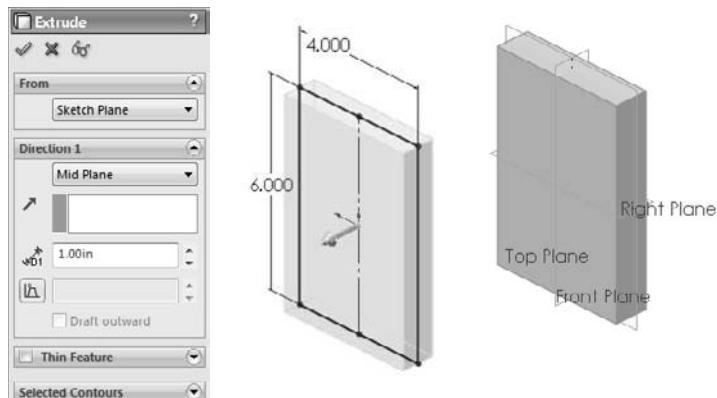
Making the first extrude feature

Going back to the sketch in Figure 4.2, I will show you how to continue building the part using the newly learned tools. By centering the sketch on the Origin and extruding by using a Mid Plane end condition, the initial block is built symmetrically about all three standard planes, with the part Origin at the center. In many parts, this is a desirable situation. It enables you to create mirrored features using the standard planes, and helps you to assemble parts together in an assembly later, when parts must be centered and do not have a hard face-to-face connection with other parts. Figure 4.4 shows the initial feature with the standard planes.

FIGURE 4.4

An initial extruded feature centered on the standard planes

Note

When you create a feature from a sketch, SolidWorks hides and absorbs (consumes) the sketch under the feature in the FeatureManager, so you need to click the plus sign next to the feature to see the sketch in the tree. You can right-click the sketch in the FeatureManager to show it in the graphics window. ■

The next modeling step is to create a groove on the back of the part. How is this feature going to be made? You can use several techniques to create this geometry. List as many techniques as you can think of, whether or not you know how to use them. Later, I will go through several techniques that work.

Tip

One of the secrets to success with SolidWorks, or indeed any tool-based process, is to know several ways to accomplish any given task. By working through this process, you gain problem-solving skills as well as the ability to improvise when the textbook method fails. ■
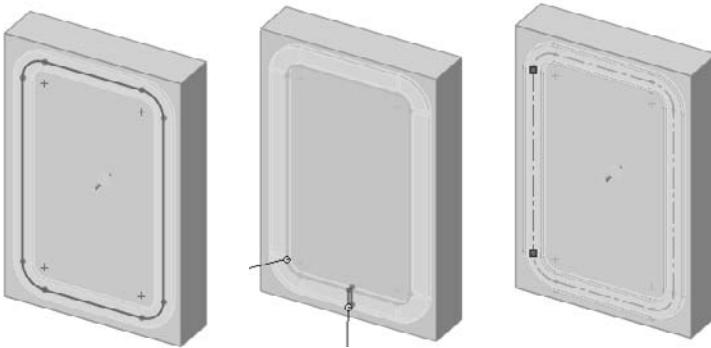
Figure 4.5 shows multiple methods for creating the groove. From the left to the right, the methods are a thin feature cut, a swept cut, and a nested loop sketch.

FIGURE 4.5

Methods for creating the groove

With a thin feature cut, you sketch the centerline, and in the Cut-Extrude feature, select the Thin Feature option and assign a width and depth. The option on the right is what is called a "nested loop," because it has a loop around the outside of the slot and another around the inside. Only the material between the loops is cut away. The method in the center is a sweep where the cross section of the slot is swept around a path to make the cut.

Another potential option could include a large pocket being cut out, with a boss adding material back in the middle. Each option is appropriate for a specific situation. The thin feature cut is probably the fastest to create, but also the least commonly used technique for a feature of this type. Most users tend to use the nested loop option (one loop inside another).

Relative size or direct dimensions

You can control the size of the groove as an offset from the edges of the existing part or you can drive them independently. Again, this depends on the type of changes you anticipate. If the groove will always depend on the size of the part, the decision is easy. If the groove changes independently from the part, you will need to re-create relations within the sketch to reflect different design intent. To create a groove, you can create a rectangle by offsetting the block shape and use sketch fillets to round the corners.

Creating the offset

You need to consider one more thing before you create the sketch. What should you use to create the offset — the actual block edges or the original sketch? The answer to this is a Best Practice issue.

Best Practice

When creating relations that need to adapt to the biggest range of changes to the model, it is best to go as far back in the model history as you can to pick up those relations. In most cases, this means creating relations to sketches or reference geometry rather than to edges of the model. Model edges can be fickle, especially with the use of fillets, chamfers, and drafts. The technique of relating features to driving layout sketches helps you create models that do not fail through the widest range of changes. ■

To create the offset for your part, follow these steps:

1. **Open a sketch on the face of the part.** To create the offset, expand the Extrude feature by clicking the plus icon next to it in the FeatureManager so that you can see the sketch. Regardless of how it displays here, this sketch appears before the extrude in the part history. Right-click the sketch and select Show.
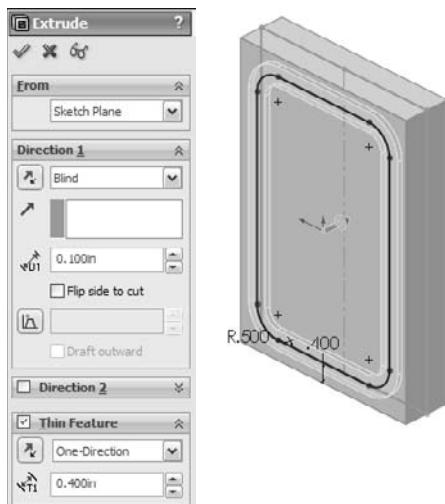
Tip

You can view individual sketches and reference geometry entities such as planes from the RMB menu. The global settings for the visibility of these items are found in the View menu. You can access these items faster by using the View toolbar, or by linking the commands to hotkeys. ■

2. **Right-click the sketch in the graphics window and click Select Chain.** This selects any non-construction, end-to-end sketch entities. Click Offset Entities on the Sketch toolbar. Offset to the inside by .400 inches. Apply .500-inch sketch fillets to each of the corners.

3. **Click Extruded Cut on the Feature toolbar.** By default, the extruded cut will cut away everything inside the closed profile of the sketch. Look down the FeatureManager window and select the check box on the top bar of the Thin Feature panel. Make the cut Blind, .100 inch. The Thin Feature type should be set to Mid Plane with a width of .400 inches. The PropertyManager and graphics window should look like Figure 4.6.

FIGURE 4.6

Creating the groove with a thin feature cut

Sketch techniques

Although you could create the next two features more easily and efficiently using a cut, I will show you how to create them as two extrudes. The intent here is to show some useful sketch techniques rather than optimum efficiency. Begin with the part from the previous section and follow these steps:

1. **Open a new sketch on the large face opposite from the groove.** Draw a corner rectangle picking up the automatic coincident relation to one corner and then dragging across the part and picking up another coincident to the edge on the opposite side. Figure 4.7 shows the rectangle before and after this edit.
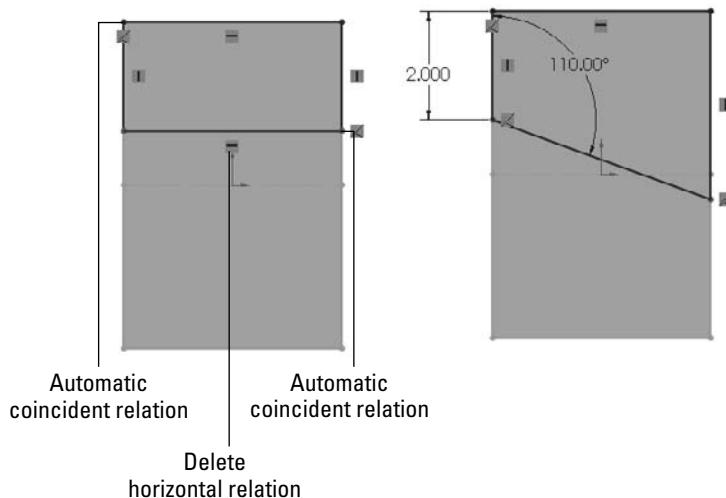
Tip

If you want to continue using the recommended best practice mentioned earlier of making relations to sketches rather than model edges, here are a few tips. In some situations (such as the current one), the sketch plane is offset from the sketch that you want to make relations to, and the best bet is to use the Normal To view. The next obstacle is making sure that automatic relations pick up the sketch rather than the edge, and so you can use the Selection Filter to select sketch entities only. ∎

2. **Delete the Horizontal relation on the line that is not lined up with an edge.** This enables you to drag it to an angle or apply the dimensions shown in Figure 4.7.

3. **Extrude sketch to a depth of 0.25 inch.**

FIGURE 4.7
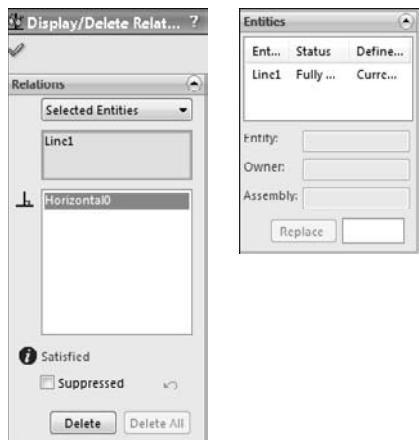
Edits to a rectangle

You can delete the Horizontal relation by selecting the icon on the screen and pressing Delete on the keyboard. As a reminder, you can show and hide the sketch relation icons from the View menu. You can check to ensure that the relations were created to the sketch rather than the model edges by clicking the Display/Delete Relations button on the Sketch toolbar, clicking the relation icon to check, and expanding the Entities panel in the PropertyManager. The Entities box shows where the relation is attached to, as shown in Figure 4.8. In this case, it is a point in Sketch1. Without custom programming, there is no way to identify items in a sketch by name, but you already know which point it is; you just needed to know whether it was in the sketch or on the model. The second sketch trick involves the use of a setting.

FIGURE 4.8

The Display/Delete Relations dialog box
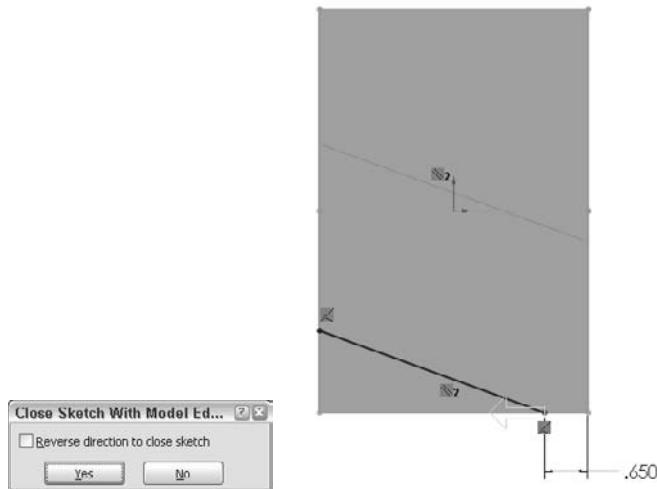
4. **Choose Tools ⇨ Options ⇨ Sketch and ensure that Prompt to Close Sketch is turned on; then click OK to close the dialog box.**

5. **Open another new sketch on the same face that was used by the last extrusion.** Draw an angled line across the left and bottom sides of the box with the dimensions shown in Figure 4.9. In this case, for this technique to work, the endpoints of the line have to be coincident with the model edges rather than the sketch entities.

   This line by itself constitutes an open sketch profile, meaning that it does not enclose an area and has unshared endpoints. Ordinarily, this results in a Thin Feature, as described earlier, but when the endpoints are coincident with model edges that form a closed loop and the setting mentioned previously is turned on, SolidWorks automatically gives you the option of using the model edges to close the sketch. This saves several steps compared to selecting, converting, and trimming manually.

FIGURE 4.9

Using the prompt to close a sketch setting

6. **Click the Extrude tool on the Features toolbar.** Answer yes to the prompt, and double-click the face of the previous extrusion. SolidWorks automatically uses the face that you double-clicked for an Up to Surface end condition. This is a simple way of linking the depths of the two extrusions automatically. Again, this entire operation could have been handled more quickly and efficiently with a cut, but these steps demonstrate an alternative method, which in some situations may be useful.

Hole Wizard
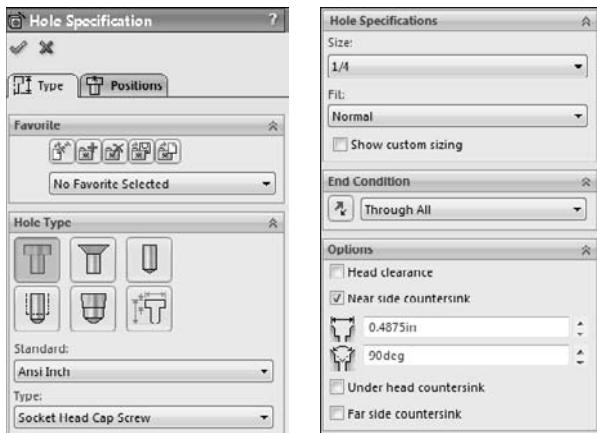
The next features that you will apply are a pair of counterbored holes. SolidWorks has a special tool that you can use to create common hole types, called the Hole Wizard. The Hole Wizard helps you to create standard hole types using standard or custom sizes. You can place holes on any face of a 3D model or constrain them to a single 2D plane or face. A single feature created by the Hole Wizard may create a single or multiple holes, and a feature that is not constrained to a single plane can create individual holes originating from multiple faces, non-parallel, and even non-planar faces (holes may go in different directions). All holes in a single feature that you create by using the Hole Wizard must be the same type and size. If you want multiple sizes or types, you must create multiple Hole Wizard features.

To apply counterbored holes to your part, follow these steps:

1. **Select the face that the groove feature was created on, and click the Hole Wizard tool on the Features toolbar.** Then set the hole to Counterbored, set the type to Socket Head Cap Screw, the size to one-quarter, and the end condition to Through All, as shown in Figure 4.10.

The Hole Wizard Hole Specification interface

2. **Next, click to select the Positions tab at the top of the PropertyManager.** This is where you place the centerpoints of the holes using sketch points. It is often useful to create construction geometry to help line up and place the sketch points. Make sure the plane or face that the holes originate from is selected in the Positions tab.

3. **Draw two construction lines, horizontally across the part, with Coincident relations to each side.** Select both lines and give them an Equal relation. The point of this step is to evenly space holes across the part without dimensions or equations.

Tip

Although several methods exist to make multiple selections, a box or window selection technique may be useful in this situation. If the box is dragged from left to right, only the items completely within the box are selected. If the box is dragged from right to left, any item that is at least partially in the box is selected. ■

Tip

SolidWorks displays an error if you try to place a sketch point where there is an existing sketch entity endpoint. If you build construction geometry in a sketch and want to place a sketch point at the end of a sketch entity, you have to create the sketch point to the side where it does not pick up other incompatible automatic sketch relations, and then drag it onto the endpoint. ■
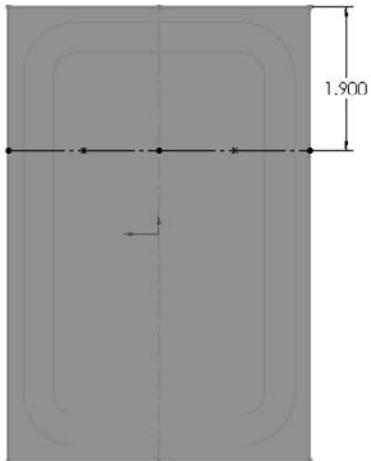
4. **Place sketch points at the midpoint of each of the construction lines.** If there is another sketch point other than the two that you want to make into actual holes, delete the extra points. Dimension one of the lines down from the top of the part, as shown in Figure 4.11. All the sketch relation icons display for reference. Click OK to accept the feature once you are happy with all the settings, locations, relations, and dimensions.

FIGURE 4.11

Placing the centerpoints of holes

Cutting a slot

The Hole Wizard does not specifically enable you to cut slots. However, SolidWorks has Slot sketch entities or you can use one of the following methods to cut a slot:

- **Use one of the Slot sketch tools.** SolidWorks has straight and arc slot options on the sketch toolbar.

- **Explicitly drawing the slot.** Draw a line, press A to switch to the Tangent Arc tool, draw the tangent arc, and press A to switch back to the Line tool, and so on. Although you can press the A key to toggle between the line and arc functions, you can also toggle between a line and a tangent arc by returning the cursor to the line/arc first point.

- **Rectangle and arcs.** Draw a rectangle, place a tangent arc on both ends, and then turn the ends of the rectangle into construction entities.

- **Thin feature cut.** As you did earlier with the groove, you can also create a Thin Feature slot, although you need to follow additional steps to create rounded ends on it.

- **Offset in Sketch.** By drawing a line, and using the Offset with Bi-directional, Make Base Construction, and Cap Ends settings, it is easy to create a slot from any shape by drawing only the centerline of the slot.

- **Library feature.** A library feature can be stored and can contain either simple sketches or more complex sets of combined features. The library feature is a good option for the counterbored slot used in this example. Library features are discussed in depth in Chapter 19.
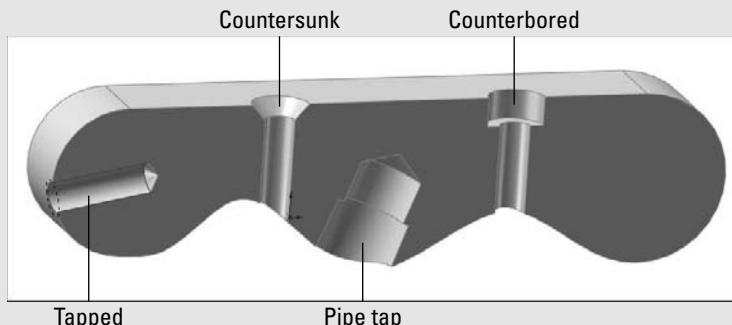
Hole Wizard: Using 2D versus 3D sketches

Hole Wizard holes use either a 2D or a 3D sketch for the placement of the hole centers. You can define the centers by simply placing and dimensioning sketch points. Starting with SolidWorks 2010, the 2D sketch type is used by default, with the 3D sketch type only being used when you specify it in the Positions tab of the Hole Wizard tool.

Cross-Reference

In Chapters 17 and 31, 3D sketches are discussed. Chapter 17 also gives a more detailed description of the Hole Wizard. Chapter 22 has additional information on the display of threads. ■

The following image shows a part with various types of holes created by the Hole Wizard, including counterbored, countersunk, drilled, tapped, and pipe-tapped holes. The part is shown in section view for clarity; however, the drilled hole is not shown in the figure.

Holes created by the Hole Wizard

To cut slots in your part, follow these steps:

1. **In this case, use the Centerpoint Straight Slot option.** Slots are easiest to create with the Click-click method rather than Click+drag. Click near where you want the center of the slot. Click again for the center of one end; then click a third time for the width/end radius. The Slot PropertyManager is shown in Figure 4.12.

   Create a horizontal sketch relation between the origin and the center point of the slot. Add dimensions as shown in Figure 4.12.
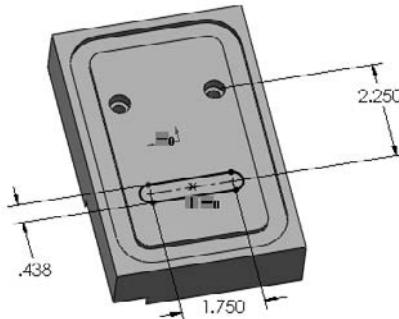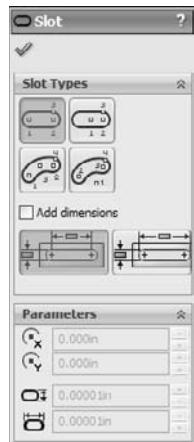
Note

Using the Add dimensions option in the Slot PropertyManager can help you size the slot more quickly. This does not require the Enable on screen numeric input option to be turned on. ■

2. **From this sketch, create an extruded cut that extrudes up to the surface of the counterbore in the holes.** The through hole for the counterbored slot is also a slot, and so you can use the same technique.

FIGURE 4.12

Creating a slot

3. **Open a sketch on the bottom of the previous slot, and draw a straight slot.** Make the new slot 0.05 inch smaller than the first slot. You can create a cut using the Through All end condition.

Tip

Picking up relations automatically may seem difficult at first, but with some practice, it becomes second nature. When trying to find the center of an arc, the centerpoint is usually displayed and is easy to select. However, when making a relation to an edge, the centerpoint does not display by default. To display it, hold the cursor over the arc edge for a few seconds; a marker that resembles a plus sign inside a circle will show you where the center is, thus enabling you to select it with a sketch tool and pick up the automatic relations. ■

In Figure 4.13, the first centerpoint has already been referenced, and the cursor is trying to find the centerpoint of the other end of the slot.

Fillets and chamfers
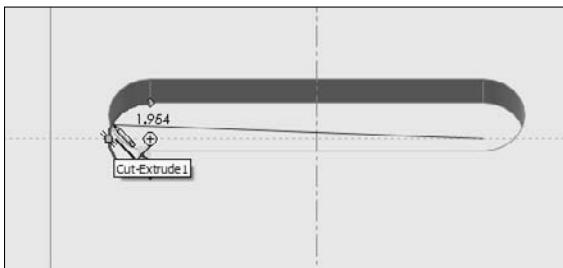
As mentioned earlier, it is considered a best practice to avoid using sketch fillets when possible, using feature fillets instead. Another best practice guideline is to put fillets at the bottom of the design tree, or at least after all the functional features. You should not dimension sketches to model edges created by fillets unless there are no better methods available. Several chapters could be written just about fillet types, techniques, and strategies in SolidWorks. Chapter 7 deals with more complex fillet types.

Best Practice

Do not dimension sketches to model edges that are created by fillets. While the previous best practice about relations to sketch entities instead of model edges was a mild warning, you must heed this one more carefully. ■

FIGURE 4.13

Applying automatic relations to a circular edge

To add fillets and chamfers to your part, follow these steps:

1. **Initiate a Fillet feature, and select the four short edges on the part.** Set the radius value to .600 inches. Click OK to accept the Fillet feature.

Tip

When selecting edges around a four-sided part, the first three edges are usually visible and the fourth edge is not. You can select invisible edges by expanding the Fillet Options panel of the Fillet PropertyManager, and selecting the Select through faces option. When you have a complex part with many hidden edges, this setting can be bothersome, but in simple cases like this, it is useful. Figure 4.14 shows this option in action. ■

2. **Apply chamfers to the edges of the angled slot through the part, as indicated in Figure 4.15.** Make the chamfers .050 inches by 45 degrees.

Chamfers observe many of the same best practices as fillets.

Tip
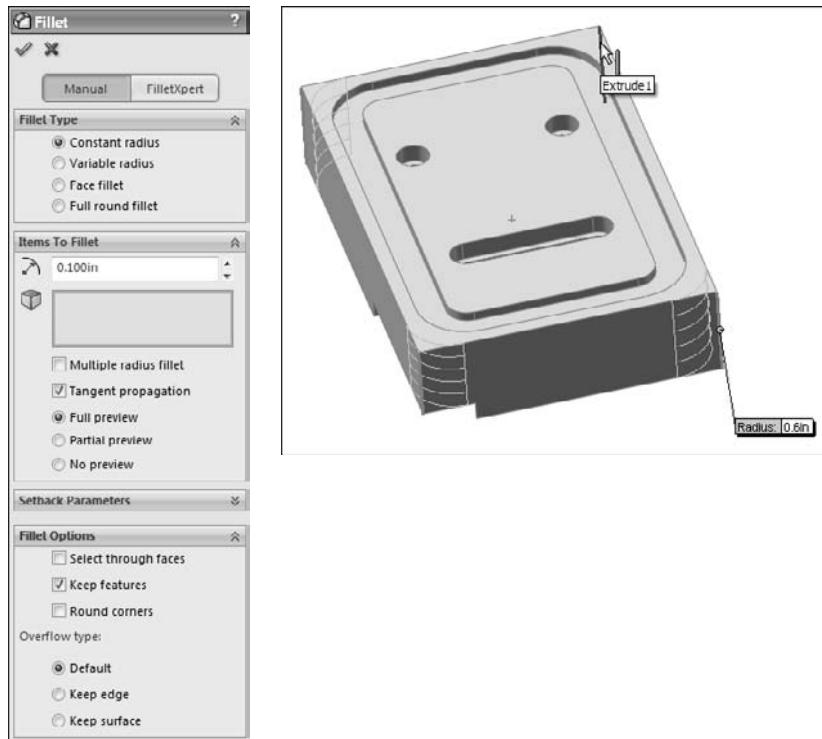
Feature order is important with features like chamfers and fillets because of how they both tend to propagate around tangent edges. Although you can turn this setting off for both types of feature, it is best to get the correct geometry by applying the features in order. ■

Cross-Reference

The Fillet Xpert, which helps you to manage large numbers of overlapping fillets by automatically sorting through feature order issues, is discussed in detail in Chapter 27. ■

Selecting an edge through model faces
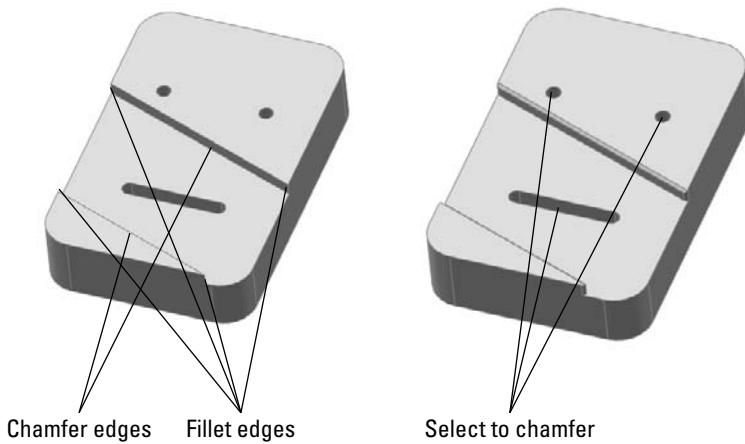
3. **Select the four edges that are indicated for fillets in Figure 4.15.** Apply .050-inch-radius fillets.

4. **Apply a last set of .050-inch chamfers to the backside of the counterbores and slot.**

The finished part is simple, but you have learned many useful techniques along the way. In the rest of this chapter, you will put the part together with other parts to form an assembly and then create a quick 2D drawing of the part and the assembly to document the design.

FIGURE 4.15

Edges for fillet and chamfer features

Chamfer edges   Fillet edges   Select to chamfer

Tutorial: Creating a Simple Assembly

Up to this point, you have been learning about how to create geometry, or parts. Assemblies involve organizing that geometrical data to represent real products or parts of products. Assemblies can be complex or simple. They can be structured in a single level or use many subassemblies. Assemblies can be static or allow Dynamic Assembly Motion. Relationships in the assembly can also drive part geometry.

This part of this chapter serves as an introduction to some of the basic functions and main features of assemblies. Because all the geometry creation is done in the part document, most of what goes on in the assembly document has to do with organizing that geometry in space.

Cross-Reference
Chapters 12 to 16 discuss various aspects of assemblies in detail. ■

The following pages describe common techniques that are used in assemblies. The part created earlier in this chapter is assembled with some additional parts that have already been created. The main point here is to give you a basic understanding of the assembly functions that exist, and how they work before exploring various aspects of the software in detail in Part II.

On the CD-ROM

This tutorial uses parts called `Chapter4Frame.sldprt` **and** `Chapter4Screw.sldprt` **from the CD-ROM, in the material for Chapter 4.** ∎
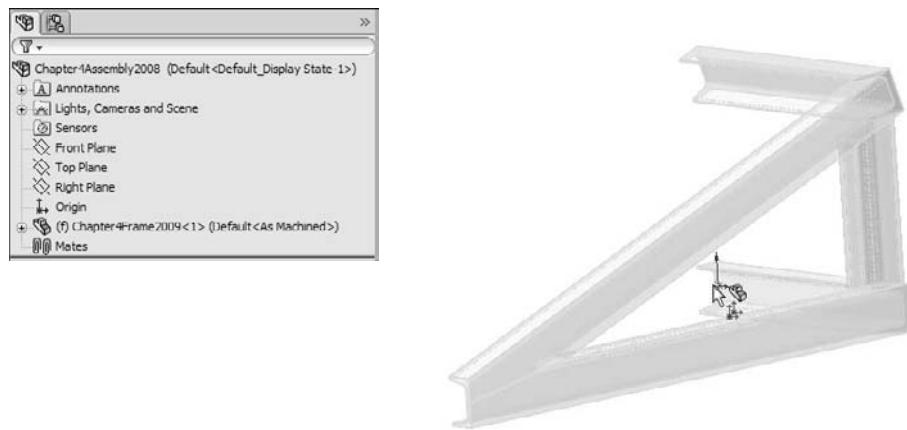
To create a simple assembly, follow these steps:

1. **From the CD-ROM, open the part named** `Chapter4Frame.sldprt.` With the part open, click the Make Assembly From Part/Assembly button on flyout toolbar under the New button in the title bar. (If you have not made a custom template for assemblies, use the default assembly template that installed with SolidWorks.) Move the cursor to the assembly Origin, where the cursor changes to indicate that the part Origin will be lined up with the assembly Origin. If the Origin is not visible on the screen, choose the View⇨ Origins menu selection to turn it on.

   The first part that you insert into an assembly has a Fixed constraint applied to it. This constraint is indicated by the (f) in front of the name of the part in the FeatureManager. Figure 4.16 shows the placement preview and cursor from Step 1, as well as the FeatureManager after the part has been added.

FIGURE 4.16

Placing a part in a new assembly

Cross-Reference

The Frame part is a weldment. Weldments are discussed in detail in Chapters 31 and 26. Weldments are multi-body parts. ∎

2. **Open the part that you created in the previous tutorial.** If you do not have it, then you can open a prebuilt copy from the CD-ROM materials for Chapter 4, named `Chapter4SimpleMachinedPart.sldprt`. Once you open the part, change to the assembly window.

Tip
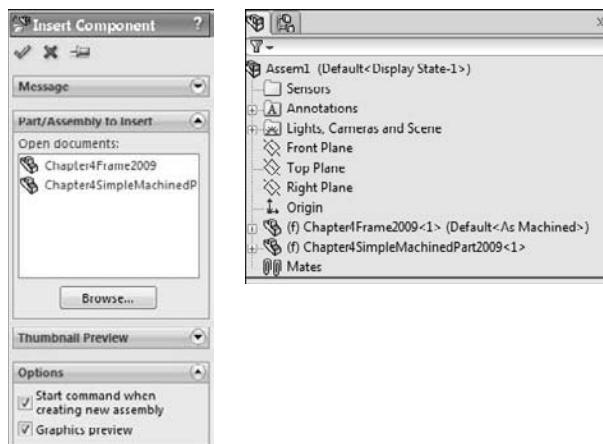
You can press Ctrl+Tab to change between open documents, and Alt+Tab to change between open applications. These Windows conventions are not exclusive to SolidWorks. ■

3. **From the assembly menus, choose Insert ➪ Component ➪ Existing Part/Assembly.** This displays the PropertyManager, as shown in Figure 4.17. Select the machined part from the selection box and click in an open space in the graphics window to place it.

   Newly placed parts in the assembly (except for the very first part) are completely undefined in terms of position or location. Instead of the (f) symbol, for Fixed, the newly placed part displays a (– symbol, which means Underdefined. You can change a Fixed part to underdefined by selecting Float in the RMB menu. Figure 4.17 also shows the FeatureManager with the new part in it. It is a little confusing that (f) stands for Fixed when the opposite condition, Float, also starts with an f.

FIGURE 4.17

The Insert Component interface

Note

Parts in an assembly are positioned relative to one another with mates. Mates are in many ways similar to sketch relations. ■
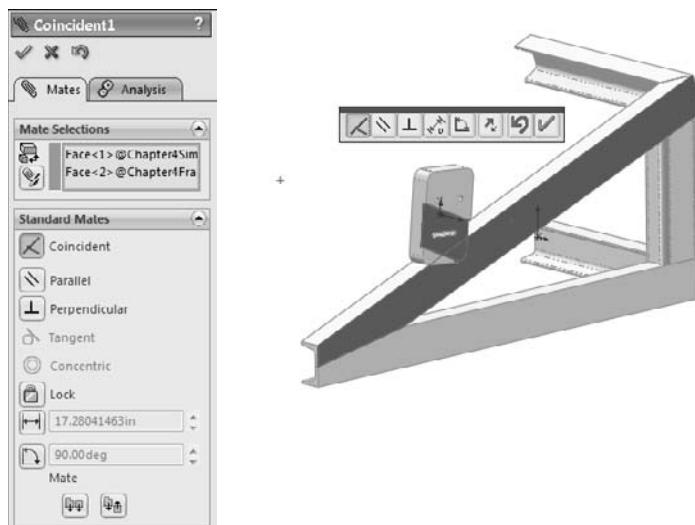
4. Click the Mate button from the Assembly toolbar. The mate options that are not grayed out are available with the current selection. For example, in Figure 4.18, the corresponding faces of the weldment and the machined part are selected, and these faces can be mated coincident, parallel, perpendicular, at a distance, or at an angle.

FIGURE 4.18

Mate options

Tip

You can move a part in an assembly by clicking the part and dragging it with the LMB (left mouse button). It follows whatever mates you have applied to it. To rotate a part that does not have any mates applied to it, drag the part with the RMB. The MMB (middle mouse button) still rotates the view. ∎

5. Select the Coincident mate, and then the faces as shown in the figure. If the machined part is turned as it appears in Figure 4.18 (so that it interferes with the welded frame if the selected surfaces touch), then click the Flip Mate Alignment button on the pop-up toolbar, or toggle the Mate Alignment buttons in the PropertyManager. Close the popup toolbar by clicking the green check mark icon.
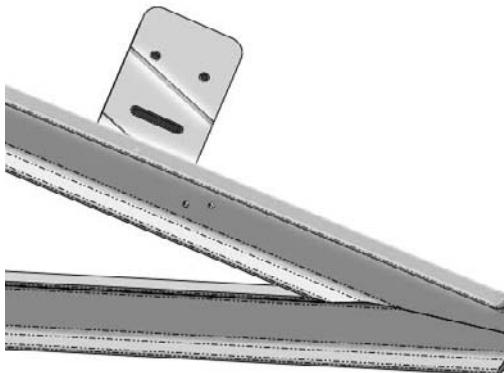
Best Practice

In contrast to sketch relations, most assembly mates have alignment orientation. Flat faces can be coincident in one of two orientations separated by 180 degrees. The same is true of concentric relations, as well as others. SolidWorks orients a part to the closest orientation that works. This means that it is often best to preposition parts to make it easier for the software. This usually involves some combination of rotating the view and rotating the part. ∎

6. **Select the top angled face of the angled frame member and the corresponding flat face of the machined part.** Figure 4.19 shows which faces to select. Make these faces coincident. In this case, the parts are already in the correct orientation, and so there is no need to pre-position them. Click the green check mark icon to accept the mate.

Selecting mating faces
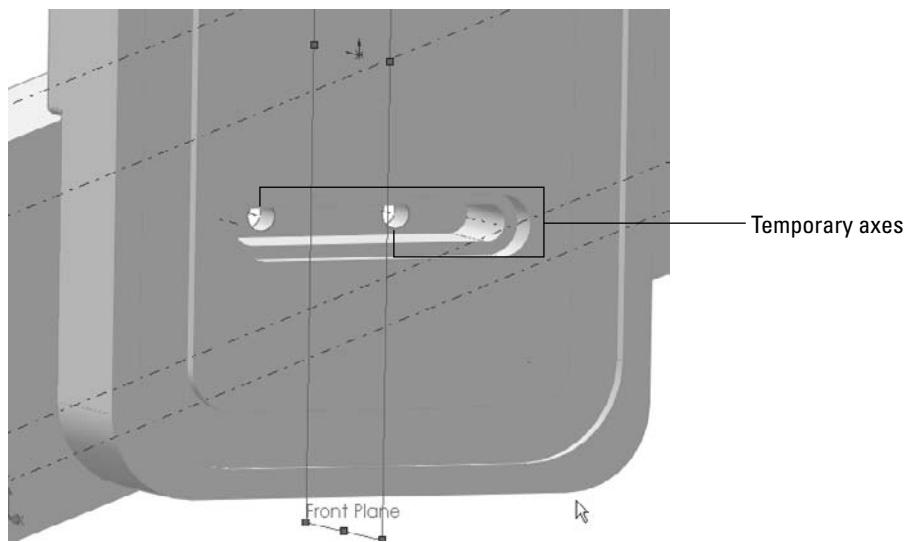
7. **One more mate is required to fully define the position of the machined part.** Drag the part and verify that it slides up and down the angled weldment member. Find the two tapped holes in the weldment and slide the machined part so that the holes appear in the counterbored slot. Ideally, the holes should be symmetrical with the part, but the slot was created to allow room for adjustment.

8. **Expand the machined part in the FeatureManager and select its Front plane.** From the View menu, turn on the display of temporary axes and Ctrl+select the temporary axes in the centers of the threaded holes in the frame, as shown in Figure 4.20. Select the Symmetric mate on the Advanced Mates panel. Turn off the display of temporary axes when the mate is complete.

9. **Through the menus or Assembly toolbar, click Insert Component and use the Browse button to find the existing** Chapter4Screw.sldprt **part on the CD-ROM, or on your hard drive if you have copied it there.**

10. **Notice that this part behaves differently in certain situations.** For example, when the cursor is over empty space, it is attached to the centroid of the part, but when the cursor is over a flat or cylindrical face, the part snaps to that face. This is because the part uses a Mate Reference, enabling planar and cylindrical faces to get Coincident and/or Concentric mates automatically when the part is dropped on them.

FIGURE 4.20

Creating a Symmetric mate

Temporary axes
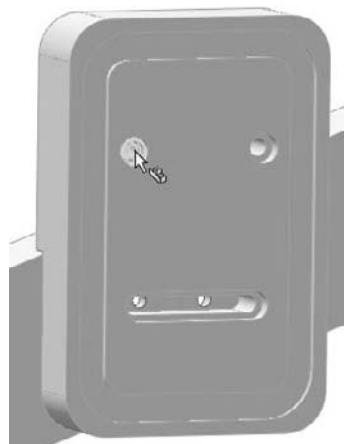
11. **Make sure that the Push Pin feature is activated in the Insert Component PropertyManager and then drop the part at the bottom of each counterbored hole.** The part automatically gets Concentric and Coincident mates. Figure 4.21 illustrates the location where you should drop the part. Click OK to accept the part placement.

FIGURE 4.21

Using a Mate Reference

12. **You need to place two more screws in the assembly, but these cannot be automatically mated; you need to do this manually.** Copy two instances of the screws. To copy a screw, Ctrl+drag the part either from the graphics window or from the FeatureManager and drop it into the graphics window.

13. **Position the part and the view so that you can see the cylindrical body of the screw and the cylindrical face of the threaded hole in the C-channel.** With the Mate function active, select both faces and click OK. Repeat the process for the other screw and hole.

14. **Now click the underside of the screw head and the counterbored surface of the slot, making sure they will be coincident, and click OK.** Repeat the process for the other screw.

15. **Save and close the assembly.**

This is a quick overview of the basic assemblies' functionality, which is expanded on in later chapters.

Tutorial: Making a Simple Drawing

If you are coming to SolidWorks from a dedicated 2D software, you will be creating drawings very differently from what you may be used to. In 2D software, you draw each view individually, and when a change occurs, you have to go back through the views and ensure that each view is updated appropriately. In 2D, views are sometimes created sparingly because they are difficult to create and to update. This includes view types such as Isometric views, complex sections, and views projected at non-orthogonal angles.

In SolidWorks, drawing views are almost free, because they are simply projected from the 3D model. Updates are made in the 3D model, and all views update automatically from there. You can handle dimensions in a couple of ways, either using the dimensions that you used to create the model, or placing new dimensions on the drawing (best practice for modeling is not necessarily the same as best practice for manufacturing drawings). To make a simple drawing, follow these steps:
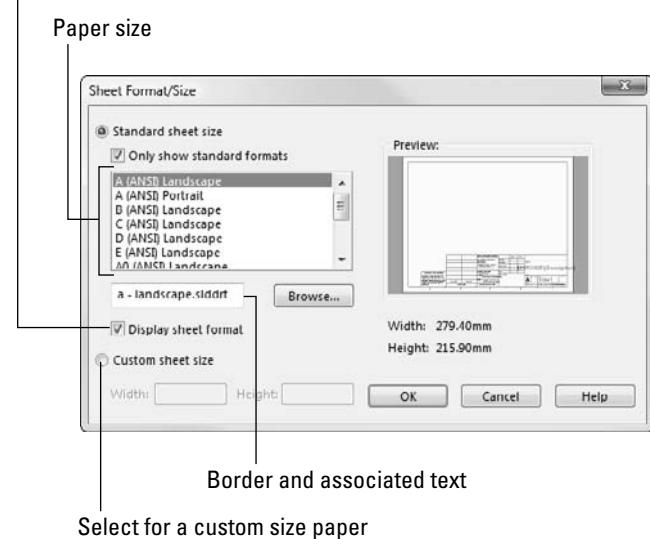
1. **Click the New button from the Standard toolbar or choose File ➪ New.** From the New SolidWorks Document window, select the Drawing template. The template contains all the document-specific settings.

2. **After selecting the drawing template, the Sheet Format/Size dialog box appears, as shown in Figure 4.22.** Select the D-Landscape sheet size, as well as the format that automatically associates with that sheet size, and click OK. If the Model View PropertyManager appears, click the red X icon to exit.

3. **Before creating any views on the drawing, set up some fields in the format to be filled out automatically when you bring the part into the drawing. Right-click anywhere on the drawing sheet (on the paper), and select Edit Sheet Format.**

FIGURE 4.22

The Sheet Format/Size dialog box

Deselect for a blank drawing sheet

Paper size

Border and associated text

Select for a custom size paper

4. **Zoom in to the lower-right corner of the drawing.** Notice that there are several variables with the format $PRPSHEET:{Description}. These annotations are linked to custom properties. Some of them have properties with values (such as the Scale note), and some of the properties do not have values (such as the Description).

5. **Add an annotation in the Drawn row, in the Date column.** You can add annotations by choosing Insert ⇨ Annotations ⇨ Note, or by activating the Annotations toolbar in the CommandManager and clicking the Note button. Type today's date as the text of the note.

Caution

If you are using a SolidWorks default template and a circle appears around your note, and then use the Text Format PropertyManager that appears when you are creating a note, expand the Border panel, and change the Circle option to None. ∎

6. **Add another note, this time to the Name column.** Do not type anything in the note, but click the Link to Properties button in the Note PropertyManager to create a link to a custom property. In the Link to Property dialog box, click the Model in View Specified option in Sheet Properties. Type **user** in the drop-down text box below the option. This now accesses a custom property in a part or assembly that is put onto this drawing and called "user," and will put the value where the note is placed.
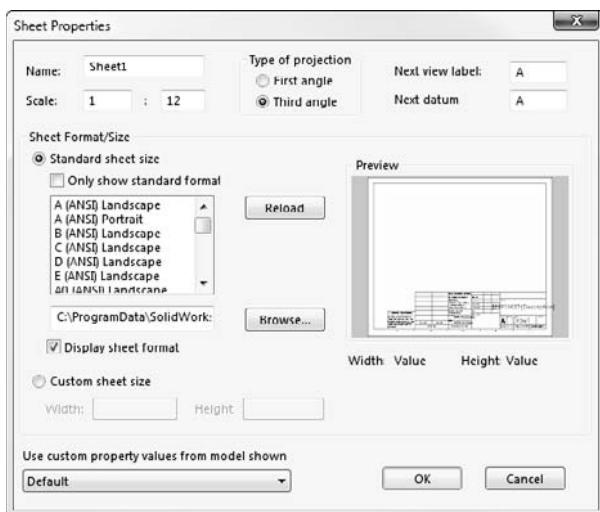
7. **To return to Edit Sheet mode (out of Edit Format mode), select Edit Sheet from the RMB menu.** A little text reminder message appears in the lower-right corner on the status bar to indicate whether you are editing the Sheet or the Format.

8. **From the Drawings toolbar, click the Standard 3 View button, or through the menus, choose Insert ➪ Drawing View ➪ Standard 3 View.** If the Chapter4SimpleMachinedPart document does not appear in the list box in the PropertyManager, then use the Browse button to select it. When you click the OK button, the three drawing views are created.

9. **Drawing views can be sized individually or for each sheet.** The Sheet Properties dialog box in Figure 4.23 shows the sheet scale. If this is changed, all the views on the sheet that use the sheet scale are updated. If you select a view and activate the Drawing View PropertyManager, you can use the Scale panel to toggle from Use Sheet Scale to Use Custom Scale.

Caution

In the United States, drawings are traditionally made and understood using the Third Angle Projection, which is the ANSI (American National Standards Institute) standard. In Europe, drawings typically use First Angle Projection, which is the ISO (International Organization for Standardization) standard. If you are not careful about making and reading your drawings, you could make a serious mistake. There are times when in the United States, the SolidWorks software will install with ISO standard templates, which will project views using First Angle Projection. When you're using a template that you are unfamiliar with, it is a good idea to check the projection method. To do this, right-click the drawing sheet and select Sheet Properties. The Type of projection setting appears in the top middle of the dialog box, as shown in Figure 4.23. This dialog box looks similar to the Sheet Format/Size dialog box, but it has some additional options, including the projection type. ■

FIGURE 4.23

First angle versus third angle projections

10. **To create an Isometric view, activate the Drawings toolbar in the CommandManager, and click the Projected View button.** Then select one of the existing views, and move the cursor at a 45-degree angle. If you cannot place the view where you would like it to go, press the Ctrl key to break the alignment and place the view where you want it.

11. **You can change the appearance of the drawing view in several ways.**

- **View ⇨ Display ⇨ Tangent Edges with Font** uses phantom line type for any edge between tangent faces.

- **View ⇨ Display ⇨ Tangent Edges Removed** completely removes any tangent edges. This is not recommended, especially for parts with many filleted edges, because it generally displays just the outline of the part.

- **Shaded** or **Wireframe** modes can be used on drawings, accessed from the View toolbar.

- **Perspective** views must be saved in the model as a named view and placed in the drawing using the view name.

- **RealView** drawing views are not available on a drawing except by capturing a screen shot from the model and placing this screen shot in a drawing. The same applies to PhotoWorks renderings.

12. **Look at the custom properties that you created in the title block.** The date is there because you entered a specific value for it, but the Name field is not filled in. This is because there is no User property in the part. Right-click the part in one of the views and select Open Part. In the part window, choose File ⇨ Properties, and in the Property Name column, type the property name **user**, with a value of your initials, or however your company identifies people on drawings. The Properties dialog box, also called Summary Information, is shown in part in Figure 4.24.

FIGURE 4.24

The Custom Properties entry table
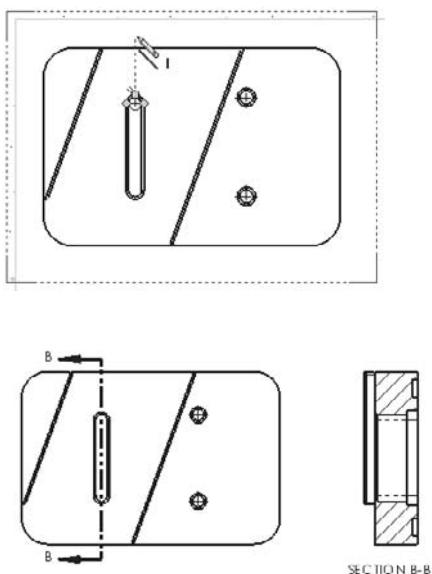
13. **When you flip back to the drawing (using Ctrl+Tab), the Name column now contains the value of your initials.**

14. **Click the Section View button on the Drawings toolbar.** This activates the Line command so that you can draw a section line in a view. When sketching, a line can go either on the Sheet or in a view. This is similar to the distinction between the Sheet and the Format. To make a section view, the section line sketch must be in the view. You will know that you are sketching in a view when a pink border appears around the view. You may also use Lock View Focus from the RMB menu to lock view focus manually.

15. **Bring the cursor down to the circular edge of the slot to activate the center point of the arc.** Once the center point is active, you can use the dotted inference lines to ensure that you are lined up with the center. Another option is to create manually sketch relations. Turning on temporary axes displays center marks in the centers of arcs and circles. Figure 4.25 shows the technique with the inference lines being used. Draw the section line through the slot and then place the section view.

FIGURE 4.25

Creating a section view

SECTION B-B

16. **As mentioned earlier, you can use two fundamentally different methods for dimensioning drawings:**

   - **Model Items** imports the dimensions used to build the SolidWorks model and uses them on the drawing. These dimensions are *bi-directionally associative*, meaning that changing them on the drawing updates the model, and changing them on the model

updates them in the drawing. On the surface of things, this sounds too good to be true, and it is. The potential problems are that you might not model things the way you would dimension them for the shop. You have to answer several questions for yourself, such as do the leader lines go to the right locations or can they be moved and the dimensions usually come in in such a way that they require quite a bit of moving them around.

- **Reference (driven) Dimensions** can be applied to the drawing view directly. These are only associative in one direction, meaning that they measure what is there, but they do not drive the size or position of the geometry. All changes must be made from the model. Again, on the face of things, this appears to be redundant and a waste of time, but in my personal estimation, by the time you finish rearranging dimensions, checking to ensure that you have everything you need and hiding the extraneous dimensions, you are usually far better off using reference dimensions.

Best Practice

Users have strong opinions on both sides of this issue. The best thing for you to do is to use both methods and decide for yourself. ∎

17. **If you choose to use the Model Items approach, you can do this by choosing Insert ⇨ Model Items.** Then specify whether the dimensions should come from the entire model or just a selected feature. You also need to ask whether the dimensions should come into all views or just the selected one, and whether you want just a certain type of dimension, annotation, or reference geometry.

18. **Once the dimensions are brought in, you need to move some of them from one view to another, which you can do by Shift+dragging the dimension from the old location to the new location.** Ctrl+dragging predictably copies the dimension. You can move views by dragging an edge in the view.

Sheet versus Sheet Format

With new and even experienced users, there is some confusion around the Sheet versus Sheet Format issue. Part of the confusion is due to SolidWorks terminology. SolidWorks names the two items Sheet and Sheet Format. In this book, I simply use the terms *Sheet* and *Format*, to avoid linking the two items with a common first name. It would be better yet if *Format* were changed to *Border* or *Title Block* so that the name more closely matched the function.

In a SolidWorks drawing, you are editing either the sheet or the format. When editing the sheet, you can perform actions such as view, move, and create views, but you cannot select, move, or edit the lines and text of the drawing border. When editing the format, you can edit the lines and text that make up the drawing border, but the drawing views disappear.

Often, users save a template that already contains a format, and save themselves some time every time they create a new drawing.

While you cannot change templates after you create a document, you can swap and update formats, and change sheet sizes.

Summary

This chapter helps to lay the foundation for the more detailed information that will follow. The chapters in Part I include recommendations and answers to questions that help you to develop an intuition for how SolidWorks software operates, which is the most crucial kind of knowledge when troubleshooting a modeling or editing problem.

This chapter has glossed over many of the important details in order to give you a quick overview of the basic functionality in SolidWorks for the three main data types: Parts, Assemblies, and Drawings. Later chapters expand on this information significantly.

Using Visualization Techniques

I n SolidWorks software, visualizing geometry is part of the overall mission of the software. Visualizing 3D CAD data is more than seeing shaded solids or shiny surfaces; it includes being able to see the interior and exterior at the same time and using sections, transparency, wireframe, and other tools. In AutoCAD, the visualization is entirely in your head. You have to imagine what the 3D looks like given the 2D views. SolidWorks takes it so much further than just being able to see things in 3D; you can look at some parts of an assembly in wireframe while others are transparent and others are opaque. You can see a part with a reflective appearance. You can create section views in parts and assemblies to visualize internal details.

My aim with this chapter is to show you important capabilities that will expand your SolidWorks knowledge, at the same time, provide some of the awe and wonder we sometimes experience while using incredible 3D tools to do actual work. I start with visualization tools and pass through to some more advanced visualizations tools and techniques. If I sound a little enthusiastic about this topic, it is because visualization is the part of this software that really brings your imagination to life. It can be the source of real inspiration and allows you to communicate geometrical ideas with other people.

Manipulating the View

One of the most important skills in SolidWorks is manipulating the view. This is something you'll do more frequently than any other function in SolidWorks; therefore, learning to do it efficiently and effectively is very important, whether you look at it as rotating the model or rotating the point of view around the model. The easiest way to rotate the part is to hold down

the middle mouse button (MMB) or the scroll wheel and move the mouse. If your mouse does not have a middle button or a scroll wheel that you can use as a MMB, then you can use the Rotate View icon on the View toolbar, or the icon on the Heads-up View toolbar. The Heads-up View toolbar is shown in its default state in Figure 5.1.

FIGURE 5.1

Use the Heads-up View toolbar to easily access most visualization tools.

The Heads-up View toolbar can be customized and disabled using the same method that you use for all other toolbars, through the Tools ⇨ Customize dialog.

Tip

Some mouse drivers change the middle-button or scroll-wheel settings to do other things. Often, you can disable the special settings for a particular application if you want SolidWorks to work correctly and still use the other functionality. For example, the most common problem with mouse drivers is that when the model gets close to the sides of the graphics window and the scroll bars engage, the middle mouse button suddenly changes its function. If this happens to you, you should change the function of the MMB to Middle Mouse Button from its present setting. ∎

Using arrow keys

You can use the arrow keys on the keyboard to manipulate the view in predictable and controllable ways. You can use the Shift, Ctrl, and Alt keys to add to the behavior.

The arrow keys enable you to rotate to the following views:

- **Arrow.** Rotate 15 degrees. To customize this setting, choose Tools ⇨ Options ⇨ View.
- **Shift+arrow.** Rotate 90 degrees.
- **Alt+arrow.** Rotate in a plane flat to the screen.
- **Ctrl+arrow.** Pan.

Using the middle mouse button

Most, if not all, mice sold today have middle mouse buttons (MMBs), usually in the form of a clickable scroll wheel.

The MMB or scroll wheel has several uses in view manipulation:

- **MMB alone.** Rotate.
- **Click or hover on edge, face, or vertex with MMB, and then drag MMB.** Rotate around selected entity.

- **Ctrl+MMB.** Pan.
- **Shift+MMB.** Zoom.
- **Double-click MMB.** Zoom to fit.
- **Scroll with wheel.** Zoom in or out. To reverse direction of the zoom setting, choose Tools ⇨ Options ⇨ View.
- **Alt+MMB.** Rotate in a plane flat to the screen.

Using mouse gestures

Mouse gestures are an interface method that you can customize to do anything a SolidWorks toolbar button can do, but by default, it controls view orientation. Figure 5.2 shows the default configuration of the mouse gesture donut.

FIGURE 5.2

Click+drag the right mouse button (RMB) to access the commands on the donut.

It may take a little time for you to get used the interface. It works best when you understand what the commands are before you use them, so that you can invoke the Top View command in a single motion. It does not work well if you have to initiate the interface with a very short RMB drag, then drag again to select the command. For this reason, it might be better to limit the donut to four commands rather than eight, and set it up intuitively such that the top view is a RMB stroke up, a right view is a RMB stroke to the right, and so on.

You can customize the mouse gesture donut in the Tools ⇨ Customize ⇨ Mouse Gestures. This works much like the Keyboard (hotkey) customization, where you can turn gestures on or off, set the mouse gesture donut to four or eight sections, and any gesture direction to any available command.

Using the View toolbar

The View toolbar, shown in its entirety in Figure 5.3, contains the tools that you need to manipulate the view in SolidWorks. Not all of the available tools are on the toolbar by default, but I have added them here for this image. To customize your own View toolbar, you must use choose Tools ⇨ Customize from the menu and select the Commands tab. Then click on the View toolbar,
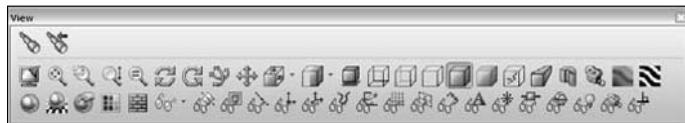
and either drag items from the Customize dialog box to the View toolbar to add them, or from the View toolbar into the empty graphics area to remove them. You can use these tools with part and assembly models but not drawings.

FIGURE 5.3

The View toolbar
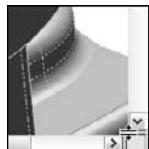
Adding scrollbars and splitters

An option exists to add scrollbars and view pane splitters to the graphics window. To use it, choose Tools➪Options➪Display/Selection, Display Scrollbars in graphics view. This selection will be grayed out if any SolidWorks documents are open (so you must close all SolidWorks documents to change it). When you zoom in such that the part/assembly/drawing is partially off the screen, the scrollbars will activate on the right side and bottom of the SolidWorks window, enabling you to scroll up and down as well as left and right to pan the view.

Figure 5.4 shows a detail of the bottom-right corner of the SolidWorks graphics window, where you find the scrollbars and splitters. Notice the cursor in the lower right over one of the splitters. The splitters can be easy to miss if you do not know what they look like.

FIGURE 5.4

Scrollbars and splitters controls can be turned on or off.

The splitters enable you to split the main graphics window into multiple view ports. The options are two ports horizontally, two ports vertically, or four view ports. The splitter bars are located at the intersection of the scrollbars in the lower-right corner of the graphics window. Of course, you can also use the icons on the Standard Views toolbar for splitting the view into two vertical ports, two horizontal ports, or four ports, the Heads-up View toolbar, or the View Orientation flyout.

Once a viewport has been split, you can remove the split with the toolbar icons, either by dragging the border back to the edge of the display window or by double-clicking the split border. If the view has been split into four, you can set it back to a single viewport by double-clicking the intersection of the horizontal and vertical port borders.
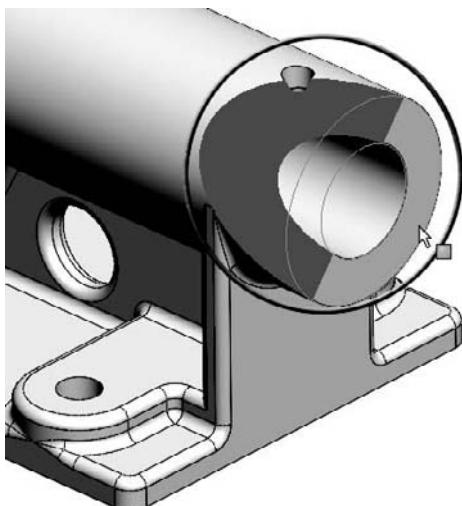
Using the Magnifying Glass

You can invoke the Magnifying Glass by pressing G, and dismiss it when you select something or when you press Esc. To change the hotkey it is associated with, choose Tools➪Customize➪ Keyboard. Magnifying Glass is listed in the Other category. The Magnifying Glass is intended to magnify a small area of the view to enable you to make a more precise selection.

The magnified area follows your cursor as it moves, and you can zoom in and out by scrolling the MMB. Ctrl and dragging keeps the Magnifying Glass centered on the cursor. Pressing Alt creates a section view parallel to the view, and scrolling with Alt pressed moves the section plane further away or closer. Figure 5.5 shows the Magnifying Glass in operation, cutting a section view through a part.

FIGURE 5.5

Using the Magnifying Glass with the section view

Note
The intended purpose of the Magnifying Glass is to select small items. You may use it to inspect things, but remember it will disappear as soon as you select something. ■

Clicking the Triad axes

The Triad is the multicolored coordinate axis in the lower-left corner of the SolidWorks graphics window. You generally use it passively to see how the view is oriented and to get X, Y, Z reference directions for features that need it.

To use the Triad to control the view orientation, try the following:

- **Click an axis.** This axis will point out of the screen.

- **Click an axis a second time.** This axis will point into the screen.

- **Shift-click an axis.** This view will spin 90 degrees about that axis (using the right-hand rule).

- **Alt-click an axis.** This view will spin 15 degrees (or the default view rotation angle) around the axis.

When you are in a named view, a little box in the lower-left corner shows the name of the view. This includes standard named views and custom named views. Anything that shows up in the View Orientation box (accessed by spacebar) displays a name in the corner. Figure 5.6 shows the Triad and the named view box in the lower-left corner.

The Triad and named view box

By Shift-clicking an axis of the triad, the view is rotated 90 degrees from the original orientation. Alt-clicking rotates the view around the clicked axis by the view rotation increment set in Tools ➪ Options ➪ View, which is 15 degrees by default. Pressing Ctrl in conjunction with any of these causes the view to rotate in the opposite direction. Therefore, if pressing Shift-click makes the view rotate against the right-hand rule about the clicked axis, pressing Ctrl+Shift-click makes the view rotate with the right-hand rule.

Using the View Tools

Manipulating the view is one of the most important and commonly used methods to help you visualize a CAD model to which users have access. SolidWorks has a very wide range of tools, mostly represented by the tools on the View toolbar.

The tools in this section will help you to view parts and assemblies. The following tools are mainly found in the View ➪ Display and View ➪ Modify menu areas.

- **Zoom to Fit.** Resizes the graphics window to include everything that is shown in the model. You can also access this command by pressing the F key, or double MMB-clicking.

- **Zoom to Area.** When you drag the diagonal of a rectangle in the display area, the display resizes to fit it. The border size around the fit area is fixed, and cannot be adjusted. This only zooms in, not out.

- **Zoom In/Out.** Drag the mouse up or down to zoom in or out, respectively. You can also access this command by holding down the Shift key and dragging up or down with the MMB. The hotkey Z and Shift+Z work for Zoom Out and Zoom In, respectively. The percentage of the zoom is a fixed amount and cannot be adjusted.

- **Zoom to Selection.** Resizes the screen to fit the selection. You can also access this command by right-clicking on a feature in the FeatureManager. For example, if you select a sketch from the FeatureManager and right-click and select Zoom to Selection, the view positions the sketch in the middle of the screen and resizes the sketch to match the display. The view does not rotate with Zoom to Selection.

Tip

A reciprocal function enables you to find an item in the tree from graphics window geometry. If you right-click a face of the model, then you can select Go to Feature in Tree, which highlights the parent feature. ∎

- **Zoom about Screen Center.** Enables you to zoom straight in and straight out. This tool is off by default. The default behavior is that zooming works around the cursor. If the cursor is off to one side, zooming in and out can cause the view to "walk" away from that side. This command is only found in the menus at View ⇨ Modify and does not have an icon.

- **Draft**, **Undercut, and Parting Line Analysis.** Evaluates the manufacturability of plastic and cast parts. These three types of geometric analysis are discussed in more detail in the discussion on model evaluation in Chapter 11.

- **Rotate View.** Enables you to orbit around the part or assembly using the left mouse button (LMB). You can also access this command by using the MMB without the Toolbar icon.

- **Roll View.** Spins the view on the plane of the screen.

- **Pan.** Scrolls the view flat to the screen by dragging the mouse. You can also access this command by holding down the Ctrl key and dragging the MMB without using the Toolbar icon, or with Ctrl+arrow.

- **3D Drawing View.** Enables you to rotate the model within a drawing view to make selections that would otherwise be difficult or impossible. This is of no use in part and assembly models.

- **Standard Views flyout toolbar.** The Standard Views toolbar is discussed later in this chapter. The flyout enables you to access all the Standard Views tools. This button is also called the View Orientation flyout, depending on where you see it.

- **Wireframe.** Displays the model edges without the shaded faces. No edges are hidden.

- **Hidden Lines Visible (HLV).** Displays the model edges without the shaded faces. Edges that would be hidden are displayed in a font.

- **Hidden Lines Removed (HLR).** Displays the model edges without the shaded faces. Edges that are hidden by the part are removed from the display.

- **Shaded with Edges.** The model is displayed with shading, and edges are shown using HLR. Edges can either be all a single color that you set in Tools ⇨ Options ⇨ Colors (typically black), or they can match the shaded color of the part. Tools ⇨ Options ⇨ Document Properties ⇨ Colors is where you find the document specific setting to use the same color for shaded and wireframe display, which becomes very useful in an assembly when all the parts shown in wireframe are the same color as they are when they are shaded, instead of all being black.

- **Shaded.** The model is displayed with shading, and edges are not shown.
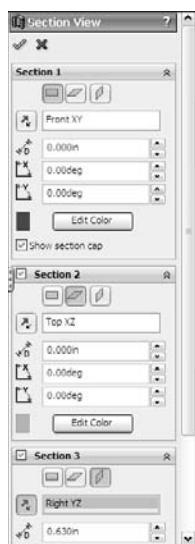
- **Shadows in Shaded Mode.** When the model is displayed shaded, a shadow displays "under" the part. Regardless of how you rotate the model, when Shadows are initially turned on, the shadow always starts out parallel to the standard plane that is closest to the bottom of the monitor. As you rotate the model, the shadow moves with it. If Shadows are turned off and then on again, they again display parallel to the standard plane that is closest to the bottom of the monitor.

- **Section View.** Sections the display of the model. Figure 5.7 shows the Section View command at work. You can use up to three section planes at once. Solid and surface models as well as assemblies can be sectioned. You can use the spin boxes, enter numbers manually, or drag the arrows that are attached to the section planes to move the section through the model. Section planes can also be rotated by dragging the border of the plane.

FIGURE 5.7

The Section View tool

Clicking the check mark icon in the Section View PropertyManager enables you to continue working with the sectioned model, although you may not be able to reference edges or faces that are created by the section view. It is only a displayed section; the actual geometry is not cut.

Section Views can be saved either to the View Orientation box or to the Annotation View folder, which enables section views to be reused on the drawing. Annotation Views are described in more detail in Chapter 22.
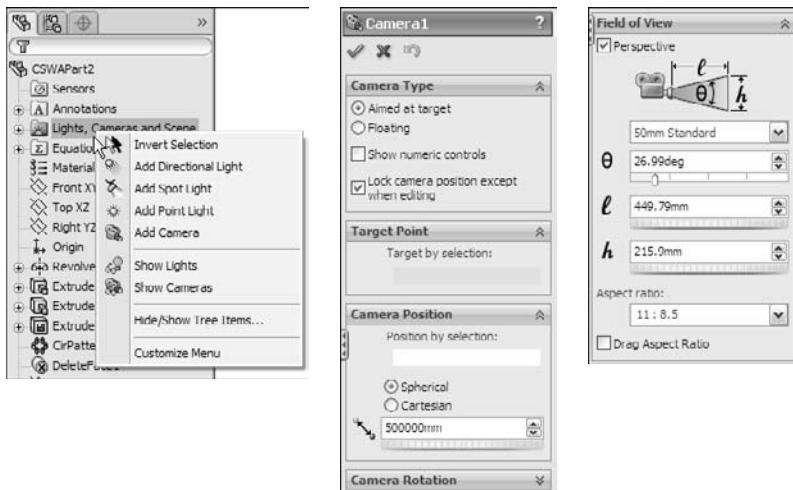
- **RealView.** Creates a more realistic reflective or textured display for advanced material selections. This feature does not work with all graphics hardware, so check the SolidWorks Web site to see if it supports your hardware. An entire section of this chapter is devoted to the various tools available with RealView graphics.

- **Edit Appearance.** Edit Appearance enables you to apply colors, textures, and materials to faces, bodies, features, parts, and components. This functionality replaces the old colors and textures interfaces. The following toolbar buttons are not on the View toolbar by default, but you can add them if you want:

Camera views

Cameras are created through the RMB menu on the Lights, Cameras and Scene folder in the FeatureManager, as shown in Figure 5.8. When you add a Camera, an interface displays in the PropertyManager, as shown in Figure 5.9.

FIGURE 5.8

Adding a new Camera with the Camera PropertyManager

183
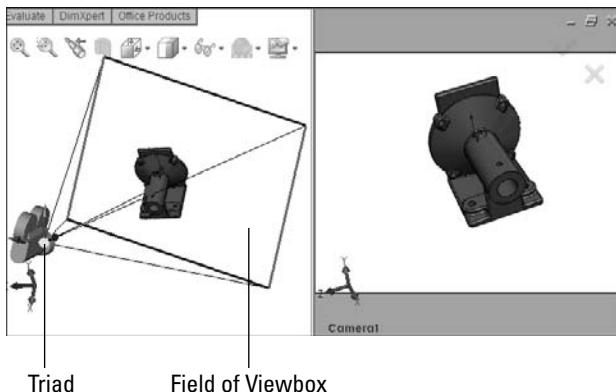
Positioning a camera with split windows

Triad   Field of Viewbox

In this interface, you can position the Camera object by dragging the triad, and you can resize the Field of View box by dragging the border. In the graphics window, you can use the left panel to target and position the Camera, while the right panel shows the view through the Camera.

The Depth of Field panel of the Camera PropertyManager is not shown, because it requires that PhotoWorks be added in. Depth of field can make objects outside of the focus area slightly out of focus, which can greatly add to the realism of renders.

There are three methods to switch the graphics window to the Camera view:

- Through the View Orientation dialog box (accessed through the spacebar)
- Through the View Orientation popup (in the lower-left area of the graphics window)
- Through the RMB menu on the Camera in the Lights, Cameras and Scene folder in the FeatureManager

When you switch the view to the Camera view, the regular Rotate View command does not function. Rotating the view means moving the Camera. You can move the Camera by editing the Camera properties, reposition the Camera by dragging the triad, or rotate the view while looking through the Camera using the Turn Camera tool.

- **Camera View.** Views the model through a Camera. You can use Cameras for:
  - Viewing the model from a particular point of view.
  - Creating renderings with perspective and depth-of-field (focus) blur; this feature is only available when PhotoWorks is added in.
  - Animating the position and target of the point of view in an animation. This feature is only available when a Motion Study is active.

- **Turn Camera.** Enables you to rotate the camera view when looking through the Camera without editing the Camera properties. You must be looking through the camera and it must be unlocked for this to work. Dragging with the MMB does the same thing if the camera is unlocked.

- **Draft Quality HLR/HLV.** Toggles between low-quality (draft) and high-quality edge Hidden Lines Removed (HLR) or Hidden Lines Visible (HLV) display. This affects display speed for complex parts or large assemblies. When in draft-quality mode, edge display may be inaccurate.

- **Perspective.** Displays the model in perspective view without using a Camera. If you want to create a perspective view on a drawing, you must create a custom view in the View Orientation dialog box with Perspective selected. You can adjust Perspective through View ⇨ Modify ⇨ Perspective by adjusting the relative distance from the model to the point of view. Relative distance is measured by the size of the bounding box of the model; therefore, if the model fits into a box roughly 12 inches on a side and the perspective is set to 1.1, the point of view is roughly 13 inches from the model. For more accurate perspective, you can use a Camera.

Caution

Perspective view and sketching do not work well together. Sketches and dimensions will look distorted and incorrect with perspective turned on. I recommend disabling perspective view when sketching. ∎

- **Curvature.** A geometrical analysis tool that applies a color gradient to the part based on the local curvature. You can also apply curvature display to individual surfaces through the RMB menu. With some hardware, curvature display can take more time to generate for complex models.

Performance

Settings in Tools ⇨ Options ⇨ Performance can greatly affect rebuild speed if curvature display data is regenerated for each part rebuild. You should leave this setting at the default setting, which is Only on Demand. ∎

Zebra Stripes

Another geometrical analysis tool that helps you visualize the quality of transitions between faces across edges. Zebra Stripes simulates putting a perfectly reflective part in a room that is either cubic or spherical and where the walls are painted with black-and-white stripes. In high-end shape design, surface quality is measured qualitatively using light reflections from the surface. Reflecting stripes makes it easier to visualize when an edge is not smooth.

The three cases that Zebra Stripes can help you identify are as follows (see Figure 5.10):

- **Contact.** Surfaces intersect at an edge, but are not tangent across the edge. This condition exists when stripes do not line up on either side of the edge.
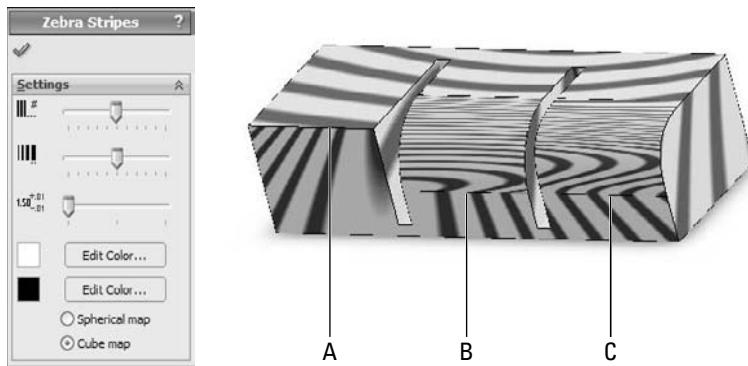
- **Tangency.** Surfaces are tangent across an edge, but have different radius of curvature on either side of the edge (non-curvature continuous). This condition exists when stripes line up across an edge, but the stripe is not tangent across the edge.

- **Curvature continuity.** Surfaces on either side of an edge are tangent and match in radius of curvature. Zebra Stripes are smooth and tangent across the edge.

FIGURE 5.10

Zebra Stripes

In Figure 5.10, the Zebra Stripes in example A do not match across the edge labeled A at all. This is clearly the non-tangent, contact-only case. Example B shows that the stripes match in position going across the indicated edge, but they change direction immediately. This is the tangent case. Example C shows the stripes flowing smoothly across the edge. This is the curvature continuous case.

You can use the remaining icons in the View toolbar to toggle the display of various types of entities from reference geometry to sketches.

Tip

Consider using hotkeys to toggle the display of your favorite items to hide and show. I use T for Temporary Axes, P for Planes, R for Origins, and so on. ■
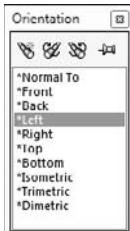
View Orientation

You can activate the View Orientation box by pressing the spacebar. View Orientation, shown in Figure 5.11, keeps all named views, saved section views, and standard views. Tools in the box also enable you to update standard views to the current view or to reset standard views to their defaults. Be aware that a toolbar button on the View toolbar is also called View Orientation.

The View Orientation dialog box

The Standard Views flyout is called either Standard Views or View Orientation, depending on where you see it. The View Orientation dialog box contains the following controls:

 ● **Push Pin.** Keeps the dialog box active.

 ● **New View.** Creates a new custom-named view.

 ● **Update Standard Views.** Sets the current view to be the new Front view; all other views update relative to this change. This also updates any associated drawing views, but does not move any geometry or change plane orientation.

 ● **Reset Standard Views.** Resets the standard views so that the Front view looks normal to the Front plane (Plane1, XY plane).

 ● **Previous View** (undo view change). You can access this tool by pressing the default hotkey Shift+Ctrl+Z.

The Standard Views toolbar

I have already mentioned the Standard Views flyout on the View toolbar, but here I describe the tools it contains in detail. Figure 5.12 shows the Standard Views toolbar in its default configuration.

The Standard Views toolbar

By default, the Standard Views toolbar contains the View Orientation button, a tool from the View toolbar. The View Orientation button is discussed in detail earlier in this section.
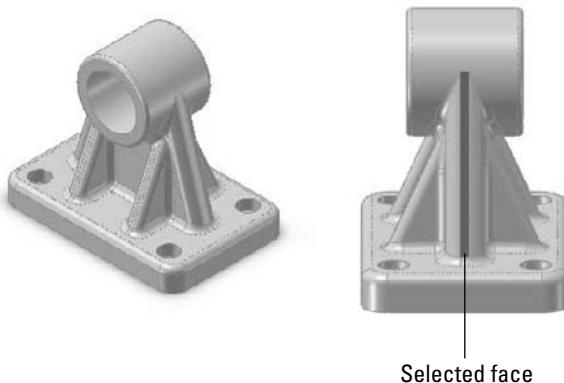
Normal To has three modes of operation:

- **First Mode.** Click a plane, planar face, or 2D sketch. When you click Normal To, the view reorients normal to the selected plane, face, or sketch, and zooms to fit the model in the view. This method is shown in Figure 5.13.

The result of using Normal To on the end rib angled face

Selected face
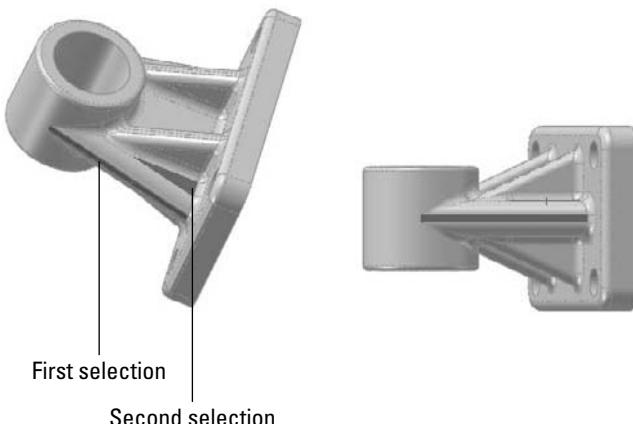
- **Second Mode.** Click Normal To a second time. The view rotates 180° to display the opposite direction.
- **Third Mode.** After making the first selection, Ctrl+select another planar entity. The view is normal to the first selection and the second selection is rotated to the top. This method is shown in Figure 5.14.

Annotation views

Annotation views enable you to group annotations that were made in the 3D model into views that will be used on the drawing. They are collected under the Annotations folder in the FeatureManager for parts and assemblies. Annotation views can be created either automatically, when 3D annotations are added, or manually. An Unassigned Items annotation view acts as a catchall for annotations that are not assigned to any particular views. In the 3D model, you can use the views to reorient the model and display annotations. As mentioned earlier, annotation views can also capture a model section view to be shown in a drawing view. The Annotation views are shown for the Chapter5SampleCasting part in Figure 5.15.
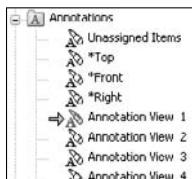
Using Normal To with Second Selection to define the top

First selection

Second selection

Annotations views for Chapter5SampleCasting.sldprt

Getting the Most from RealView

RealView is a hardware-driven set of visualization tools that help make the SolidWorks display look more realistic, adding reflections, reflected backgrounds, and shadows. RealView is intended to help the user apply advanced effects to 3D models. The fact that RealView is hardware-driven means not all video cards that are certified to work with SolidWorks also work with RealView. You need to check on the SolidWorks Web site to see what level of RealView your graphics card supports. A link to the video card testing site should be available from the SolidWorks home page at `www.solidworks.com`.

In some situations, you can use RealView instead of rendering. In these cases, RealView acts as a real-time renderer. The main advantages that rendering software such as PhotoWorks or PhotoView 360 holds over RealView are improved antialiasing control, improved shadow control, indirect illumination, global illumination, caustics, and effects such as depth of field from a camera.

You can even use RealView as a diagnostic tool for smooth transitions between surfaces because RealView appearances apply a reflective surface to a part and then apply a reflective background. This is essentially what the Zebra Stripes functionality is doing, but Zebra stripes applies a specific reflective background to make examining curvature continuity across edges more straightforward.

You can turn RealView on or off by using the golden sphere icon shown by default on the Heads-up View toolbar. If this icon is grayed out, then your system is not equipped with an appropriate RealView-capable graphics card. Generally, you need an nVidia 500 series or higher to get RealView capabilities, and an appropriate graphics driver must be installed with the hardware. NVS series cards are not 3D cards and will not enable RealView. Some ATI FireGL cards may also work.

Using RealView basic components

RealView consists of appearances and scenes. The Appearances section controls colors, textures, reflectivity, optical properties, and materials. The Scenes controls the background image and reflectivity image. Figure 5.16 shows the contents of the RealView tab in the Task Pane, as well as the callout that appears when you drop an appearance onto a part. The callout enables you to select whether you want the appearance applied to the face, body, part, or assembly level.

FIGURE 5.16

Using the Appearances tab on the Task Pane to apply appearances

Applying appearances
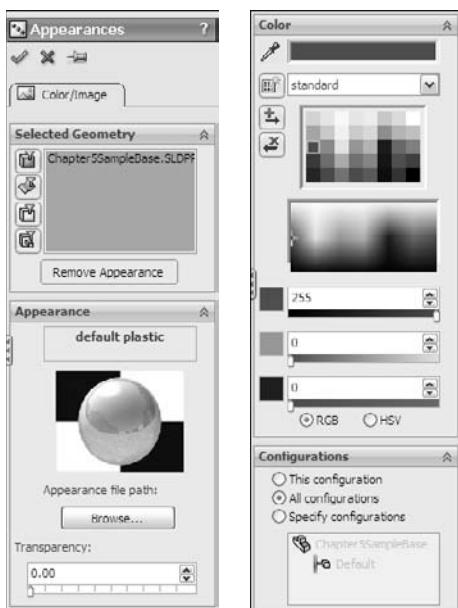
To apply an appearance, use the Task Pane at the right of the graphics window, and click the Appearances tab. Figure 5.17 shows this. Then select an appearance from the list and drag it onto the part where you would like to apply it. You can do this either in the part window or in the assembly window. A small popup will appear and present you with options for what to apply the appearance to, which is the face, feature, body, or part.

Appearances also enable you to change colors. To change the color of an applied appearance, use the Appearance Callout icon on the RMB menu, select the level you want to apply the color to (face, body, feature, part), and then select the color from the Appearance PropertyManager. Figure 5.17 shows the Appearance PropertyManager.

FIGURE 5.17

The Appearance PropertyManager

Appearance overrides

One of the things that confuses many users about applying an appearance is that it can be applied on many levels, and may override or be overridden by other appearances. This means that in a part, an appearance may be applied to a face, a feature, a body, or the entire part. There is a specific hierarchy to this system of overrides: part, body, feature, face, component, and automatic color changes.

When you apply an appearance at the level of the part (the name of the part shows in the Color and Optics PropertyManager), any other entity color will override it. You can assign Solid or Surface bodies an appearance that overrides the part appearance. Some color changes are automatic; for example, when you are editing parts in the context of an assembly, they can temporarily change color or become transparent, which overrides everything else.

Other entity colors

You can color other entities in addition to the 3D shaded model. Curve entities (such as a helix or projected curve) can be colored in addition to sketches with the Edit Sketch or Curve Color tool. You can only view sketch colors when the sketch is closed and shown, because when the sketch is open, the entity colors have special significance indicating the sketch status.

Note
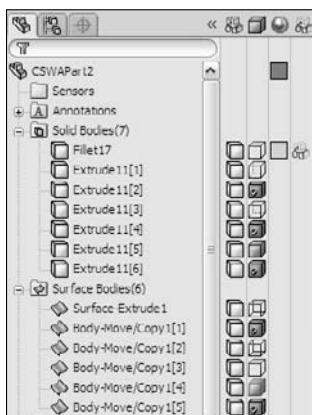
The Edit Sketch or Curve Color works for sketches and limited curve features. It works for all curve features except for projected curves. In addition, pre-selection does not work with this tool. The Edit Sketch or Curve Color tool is grouped with the commands in the View toolbar in the Tools⇨Customize⇨Commands menu area. ∎

The Display pane

The Display pane flies out from the right side of the FeatureManager and displays a quick list of which entities have appearances, transparency, or other visual properties assigned. It also shows hidden parts or bodies for assemblies and multibody parts. The Display pane is shown in Figure 5.18. I revisit the Display pane in Chapter 12 to show you how it is used in assemblies.

FIGURE 5.18

The Display pane in action

Automatic color features

The settings found at Tools➪Options➪Document Properties➪Colors can be used to automatically color certain types of features with specific colors. For example, all Shell features can be colored red as they are created.
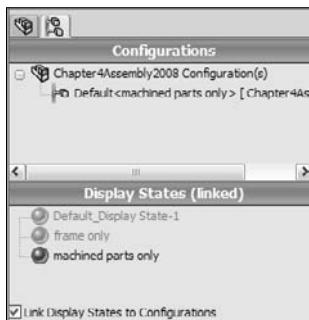
Remove appearances

You can access tools to remove appearances in two separate locations. The first is the Appearance flyout on the context toolbar for either left- or right-clicking on a model (the Appearance flyout is only available on the RMB menu if you have not disabled the context toolbar for shortcut menus).

The second location is in the Appearances PropertyManager. Both of these locations are shown in Figure 5.19.

FIGURE 5.19

Access the tool to remove appearances in the PropertyManager or the context toolbars

Using Display States

One of the most commonly used and powerful visualization aids available in SolidWorks is the Display States functionality (see Figure 5.20). Display States is simply the capability to show parts shaded, shaded with edges, wireframe, HLR (hidden lines removed), or HLG (hidden lines in gray).

Cross-Reference

Chapter 14 deals with Display States in more detail. ■
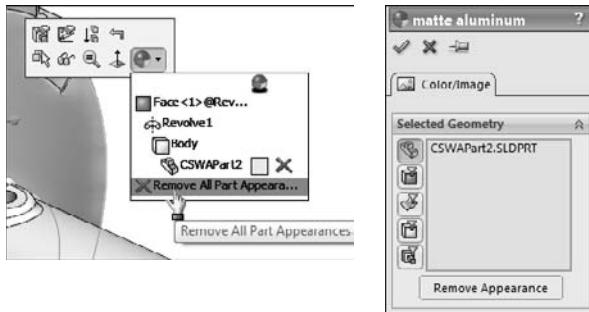
In addition to being able to use display states to differentiate parts in an assembly, you can use display states for bodies within parts. I only mention the capabilities in this chapter because display states are a huge tool for visualization in both parts and assemblies. Examples of the functionality applied to real modeling situations will come in Chapters 12 through 16 for assemblies and Chapter 28 multi-body topics.

FIGURE 5.20

Display States in an assembly

Using Edge Settings

Earlier in this chapter, I discussed the Shaded with Edges display style. Some people think that this makes the parts look "cartoony." I agree, especially when the default black edges are used, but the display improves when the edge color matches the shaded part color. In any case, sometimes this method is necessary to see the breaks between faces, especially fillets. Cartoony or not, it is also useful.
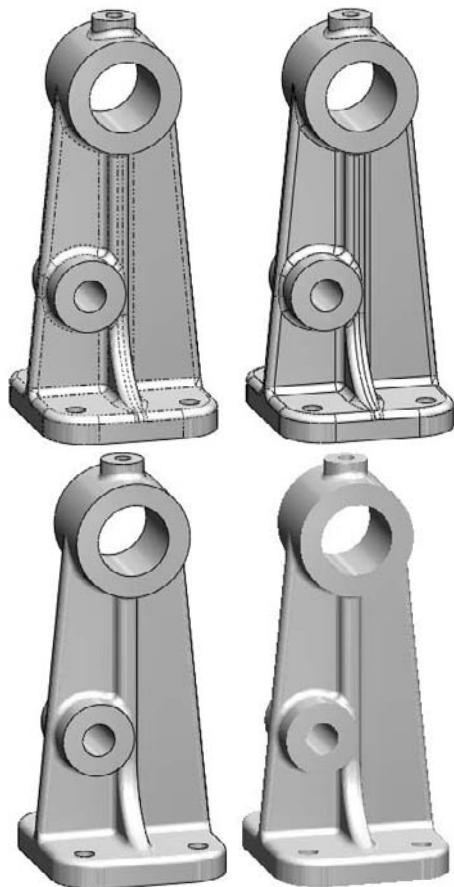
Taking this one step further, you can also make use of the tangent edge settings. These settings are found in the View➪Display menu. The settings are

- **Tangent Edges Visible.** Displays tangent edges as solid lines, just like all other edges
- **Tangent Edges as Phantom.** Displays tangent edges in a phantom line font
- **Tangent Edges Removed.** Displays only non-tangent edges

The tangent edges removed setting leaves parts looking like a silhouette. I prefer the phantom setting because I can easily distinguish between edges that will actually look like edges on the actual part and edges that only serve to break up faces on the model. The tangent edges visible setting conveys no additional information, and is the default setting. Figure 5.21 shows a sample part with all three settings.

FIGURE 5.21

Samples of the tangent edge settings

Using Assembly Visualization

Assembly Visualization is a tool that enables you to sort or display parts and subassemblies in an assembly in various ways, including by filename, quantity, mass, or a custom property value. When you click on column headers to sort the names of components in the assembly, you can move the sliders on the left side of the FeatureManager to change the display colors of the parts.
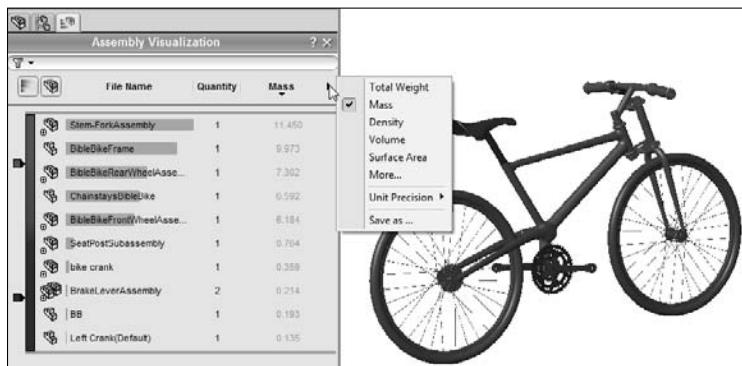
You can expand or collapse subassemblies by clicking the assembly symbol at the top of the tree, or you can disable the color display by clicking the color gradient scale. Value bars can also be displayed to show the relative value of each assembly component.

Figure 5.22 shows a model of a bicycle sorting subassemblies by mass. The gray value bars are superimposed on the text in the FeatureManager area. You will have to open an assembly on your computer and try this for yourself to see the color display.

Assembly Visualization offers several ways to sort and display the components in an assembly.

You can access the Assembly Visualization tool on the Evaluate tab of the CommandManager when an assembly is active. The toolbar icon is with the Tools icons, and you can find it by choosing Tools ➪ Customize or through the Tools menu in an assembly.

Tutorial: Applying Visualization Techniques
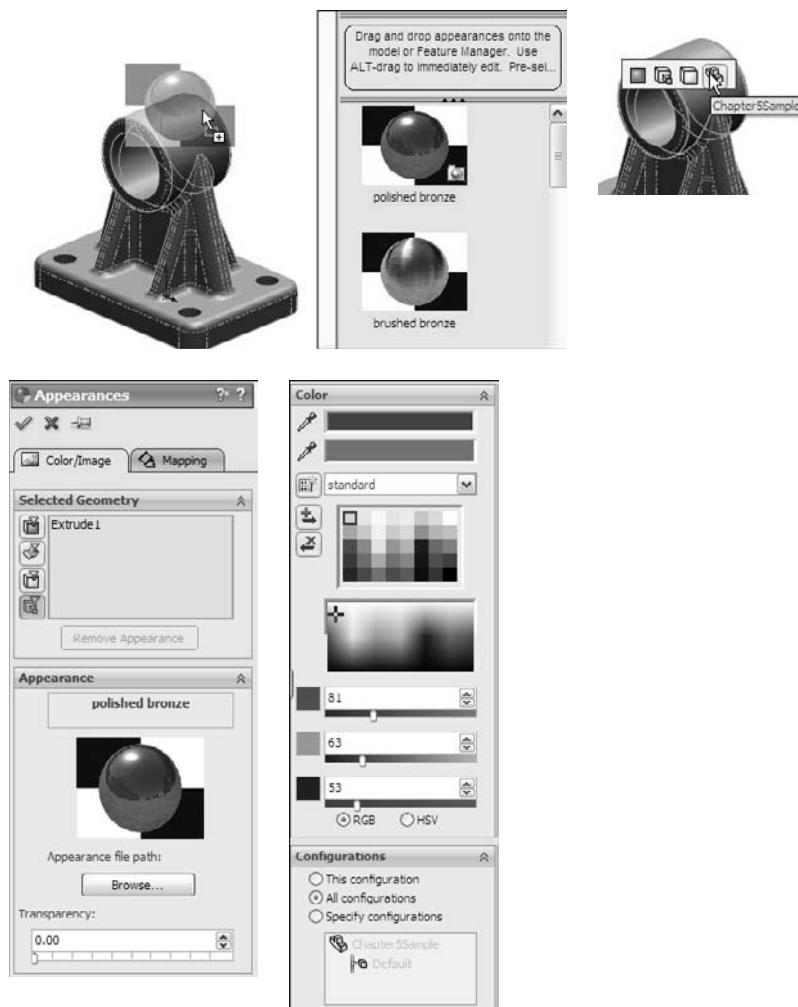
Visualization is a key factor when working with SolidWorks software. Whether it is for a presentation of your design to customers or management or simply checking the design, it is important to be able to see the model in various ways. This tutorial guides you through using several tools and techniques.

1. **If the part named** `Chapter5Sample.sldprt` **is not already open, open it from the CD-ROM.** If it is open and changes have been made to it, choose File ➪ Reload ➪ OK.

2. **Practice using some of the controls for rotating and zooming the part.** In addition to the View toolbar buttons, you should also use Z and Shift+Z (Zoom Out and In, respectively), the arrow keys, and the Ctrl+, Shift+, and Alt+arrow combinations.

3. **Use the MMB to select a straight edge on the part, and then drag it with the MMB.** This rotates the part about the selected entity. Also, apply this technique when selecting a vertex and a flat face.

4. **Select the name of the part at the top of the FeatureManager.**

5. **Click the Appearance button from the Heads-up View toolbar at the top of the graphics window.**

6. **Click the color you want in the Favorite panel.** The model should change color. If you click and drag the cursor over the colors, the model changes color as you drag over each new color. You can also drag appearances from the Task Pane. Figure 5.23 shows interfaces for both methods.

FIGURE 5.23

Use the Appearance PropertyManager to change color and material

7. **If the Color panel is not expanded, click the double arrows to the right to expand it.** Select the colors you want from the continuous color map. Again, click and drag the cursor to watch the part change color continuously.

8. **Create a swatch.** In the Favorite panel, select the Create New Swatch button and call the new swatch color file `BibleColors`.

9. **Select a color from the Color Properties continuous map; the Add Selected Color button becomes active.** Clicking the button adds the color to the swatch palette. You can add several colors to the palette to use as favorites later on.
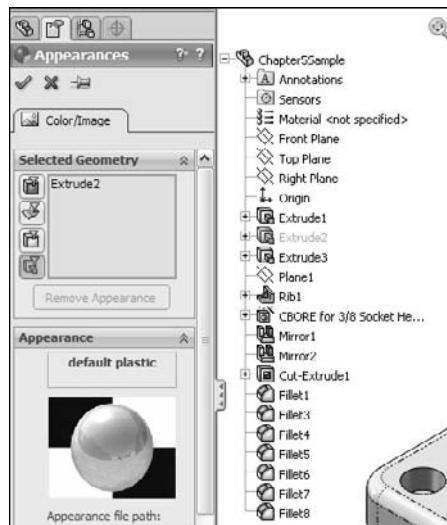
Tip

You will be able to access these colors again later by selecting `BibleColors` from the drop-down list in the Favorite panel. You can transfer the colors to other computers or SolidWorks installations by copying the file `BibleColors.slddclr` from the `<SolidWorks installation directory>\lang\english` **folder (or the equivalent file for your installed language).** ■

10. **In the Appearance panel, move the Transparency slider to the right, and watch the part become transparent.**

11. **To prevent the Appearance window from closing after every change, click the pushpin at the top of the window.**

12. **Click the green check mark icon to accept the changes; note that with the pushpin icon selected, the window remains available.**

13. **Expand the flyout FeatureManager in the upper-left corner of the graphics window, as shown in Figure 5.24, so that all the features in the part are visible.**

FIGURE 5.24

The flyout FeatureManager

14. **Select the features Extrude1, Fillet7, and Fillet6 from the FeatureManager so that they are displayed in the Selection list of the Appearances window.** Select a color from the BibleColors swatch palette that you have just created.

15. **Click the check mark icon to accept the changes and clear the Selection list.**

16. **Select the inside face of the large cylindrical hole through the part and assign a separate color to the face.**

17. **Click the check mark icon to accept the changes, and click the red X icon to exit the command.**

18. **Expand the Display pane (upper-right area of the FeatureManager).** You should see color and transparency symbols for the overall part, and color symbols for three features. There is no indication of the face color that is applied.

19. **Remove the colors.** Open the Appearances window again, re-select the three features (Extrude1, Fillet7, and Fillet6), and click the Remove Color button below the Selection list. Do the same with the colored face. Return the part transparency to fully opaque.

20. **Click the check mark icon to accept the changes.**

21. **Change the edge display to Shaded (without edges).** Then change to a Wireframe mode. Finally, change back to Shaded with Edges.

22. **Choose View ⇨ Display ⇨ Tangent Edges as Phantom.** Figure 5.25 shows the difference between Tangent Edges Visible, as Phantom, and Removed settings.

Tip

Using the Tangent Edges as Phantom setting is a quick and easy way to look at a model to determine whether face transitions are tangent. It does not help to distinguish between tangency and curvature continuity; you need to use Zebra Stripes for that. ■
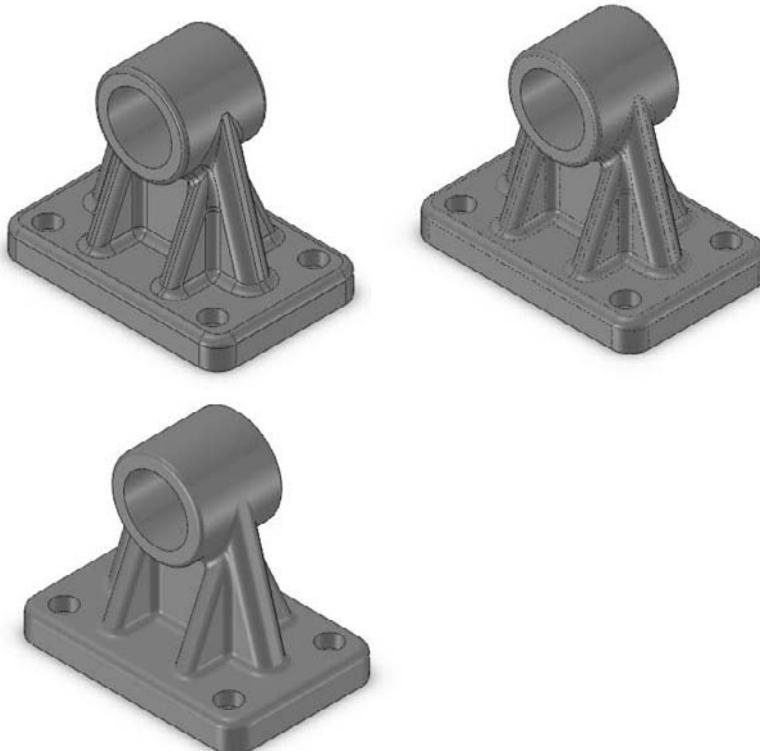
23. **Switch back to Shaded display.**

24. **If you do not have a RealView-capable computer, then skip this step.** Ensure that the RealView button in the View toolbar is depressed. Click the Appearances/Scenes tab on the Task Pane to the right of the graphics window. Expand Appearances ⇨ Metal ⇨ Steel; then in the lower pane, scroll down to the Cast Carbon Steel appearance.

25. **Turn the part over, select the bottom face, and drag and drop the appearance from the Task Pane. Apply the appearance just to the bottom face using the popup toolbar that appears.** The rest of the part should retain the semi-reflective surface, as shown in Figure 5.26. Click the check mark icon to accept the change.

26. **Click the Section View button on the View toolbar.** Drag the arrows in the middle of the section plane back and forth with the cursor to move the section dynamically through the part, as shown in Figure 5.27.

27. **Select the check box next to the Section 2 panel name and create a second section that is perpendicular to the first.**

28. **Click the green check mark icon to accept the section.** Notice that while in the Section View PropertyManager, the RealView material does not display, but once you close the dialog box, RealView returns.

Tangent Edge display settings for a shaded model
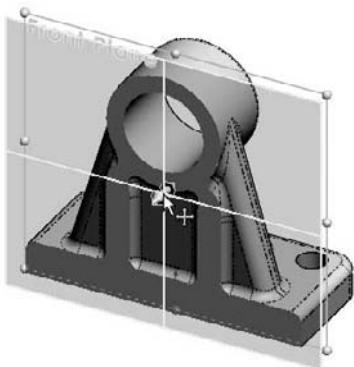
FIGURE 5.26

Applying an appearance to a face

FIGURE 5.27

A section view

Tutorial: Using Assembly Visualization

This tutorial walks you through the steps necessary to make use of the Assembly Visualization tool in SolidWorks.

1. **Close other open documents by choosing Window⇨ Close All command.** If you have any documents open from the CD-ROM, you can save them using the Save As command.

2. **Open the assembly file** `BibleBikeAssembly ch5.sldasm` **from the CD-ROM.**

3. **Select the Assembly Visualization tool from the Tools menu.** You can also select it on the Evaluate tab of the CommandManager or from a custom location by placing the icon by choosing Tools⇨Customize⇨Commands⇨Tools and dragging Assembly Visualization to the toolbar of your choice.

4. **Toggle the Flat/Nested view icon to see its effect on the tree display.**

5. **Click each available heading to see the tree sorted based on that criteria.**

6. **Turn off the assembly coloration by clicking once on the red to blue fade.**

7. **Sort the tree based on mass.**

8. **Show the tree in flat rather than nested display.**

9. **Move the red and blue sliders up and down to focus on a range of weights.**

10. **Toggle on the Value Bars to show the relative weights of parts.**

Summary

Visualization is a key function of the SolidWorks software. It can either be an end to itself if you are showing a design to a vendor or client, or it can be a means to an end if you are using visualization techniques to analyze or evaluate the model. In both cases, SolidWorks presents you with a list of tools to accomplish the task. The tools range from the analytical to the cosmetic and some of the tools have multiple uses.

Part II

Building Intelligence into Your Parts

The chapters of Part II take you beyond the basic modeling tools so you can start taking advantage of the parametric options within SolidWorks. Chapter 6 acquaints you with the entire breadth of sketching tools and techniques available in SolidWorks. Chapter 7 assists you in finding the right tool for the right job. Chapter 8 debunks some myths about patterning and mirroring and helps you establish good modeling practices. The equations in Chapter 9 are, of course, one of the quintessential strengths of parametric modeling tools. Chapter 10 introduces the concept of configurations, which will help you model variable driven models more quickly and efficiently. Chapter 11 sums it all up with what I consider to be the most important information in the book (editing), and helps you decide whether your model is "good enough" through evaluation.

Getting More from Your Sketches

P revious chapters have described the basic tools for sketching. This chapter takes you to the next level, teaching you about more advanced sketch tools, how to edit and manipulate sketches, and how to work with sketch text, sketch pictures, and sketch colors. At the end of this chapter, with a little practice to reinforce the tools and techniques, you should feel like you have mastered the topic of SolidWorks sketching and can handle almost any problem that is thrown at you.

Editing Sketch Relations

Delete is not an edit option. In time, you will find that this is good advice, even if you don't agree with it now. There are times to use the Delete command, but you should use it only when it is necessary. In my own work, I sometimes go to extreme lengths to avoid deleting sketch entities, often just to stay in practice, but also because deleting sketch entities, or even features in a part, increases the likelihood that sketch or mate relationships will be broken.

The main reason to not use Delete in a sketch is that when you are editing a sketch that has other features that are dependent on it, the dependent features may lose their references, or *go dangling*. Because of this, even when you can use the Delete command instead of making edits, it is still a good practice to edit instead. Deleting relations is not as critical as deleting sketch entities, unless the relations are referenced by equations or design tables.

Best Practice

Before deleting sketch entities, try to understand what types of relationships the change will affect downstream. Be sure to consider other sketch relationships within the current part, mates, and in-context relations in the assembly, and things of this nature. In fact, it is best to have all of this in mind when you are creating relationships to begin with. Try to make relations to the most stable entities available, which usually means having sketches and reference geometry entities as high up in the tree as possible. ■
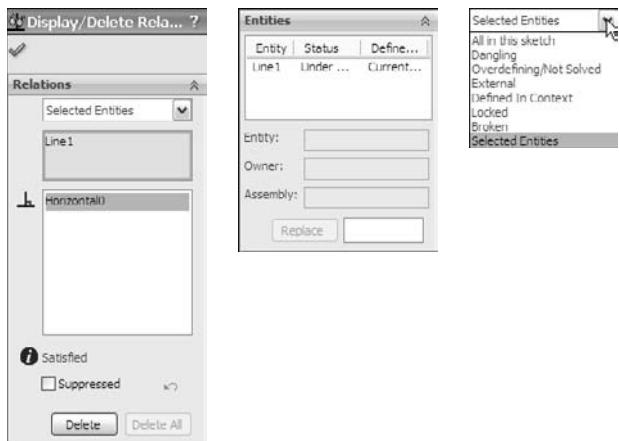
Display/Delete Relations

Display/Delete Relations is your primary tool when dealing with sketch relations. It is particularly useful for sorting relations by the various categories shown in Figure 6.1. The capability to show sketch relations in the graphics window is nice, but sorting them in a list according to their state, with the capability to delete all in any particular state, is very useful.

FIGURE 6.1

The Display/Delete Relations PropertyManager

Sketch relations in the Display/Delete Relations dialog box can be divided into the following categories:
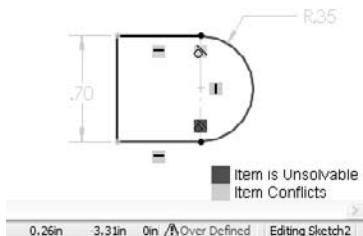
- **All in this sketch.** Shows all the relations in the active sketch.

- **Dangling.** Shows only the dangling relations. Dangling relations appear in a brownish-green or olive color, and represent relations that have lost one of the entities that drives the relation. You can repair dangling relations by selecting the entity with the dangling relation, and then dragging the red dot onto the entity to which it should have the relation.

- **Overdefining/Not Solved.** Overdefined relations are any set of conflicting or redundant instructions that are given to a sketch entity, and appear in red. For example, if a line is collinear with an edge and vertical, but the edge itself is not vertical, then both the collinear and vertical relations appear in red.

The Not Solved condition often accompanies Overdefined. Not Solved typically refers to a dimension or relation that cannot be applied because of the conflict. The lower-right corner of the screen and the status bar show flags warning that the sketch is overdefined, as shown in Figure 6.2.

When an overdefined situation exists, all the relations and dimensions in a sketch often become overdefined. This can look like a daunting task to repair, especially when the entire problem is caused by a single relation. Do not automatically delete everything. Instead, try deleting or suppressing the last dimension or relation that was added, or a single relation that looks suspect. It is also a good idea to delete red relations before deleting yellow ones. Yellow simply means conflict, while red means a condition that cannot be applied. You can suppress a dimension by setting it to Driven in the right mouse button (RMB) menu, and you can suppress relations in the Display/Delete Relations PropertyManager.

FIGURE 6.2

An overdefined sketch

- **External.** External relations connect with an entity outside the active sketch. This includes the part Origin, or any model edges. The term *external relations* can also signify any relations outside of the part.

- **Defined in Context.** Any relation between features in one part in an assembly and another part is considered an in-context relation.

- **Locked (Broken).** External relations (outside the part) may be locked or broken to increase speed and to lock out parametric changes. There is no advantage of breaking relations rather than locking them. Both are ignored, but locked relations can be unlocked; broken relations can only be deleted.

- **Selected Entities.** Sketch relations are shown only for the selected sketch entities.

Cross-Reference

In-context design, also called top-down, as well as locked and broken relations, is covered in detail in Chapter 16. ■

Caution

Some of the relations listed in the Display/Delete Relations dialog box may be colored to signify the state of the relation. Unfortunately, colored relations are typically placed at the top of the list to attract attention, but when you select them, they are always gray, and so the advantage of color-coding is always defeated for the first relation in the list. The only way around this is to select a relation other than the first one in the list. If there is only one relation in the list, you cannot see the state color. ■

A setting in Tool ⇨ Options controls the display of errors. You can choose Tools ⇨ Options ⇨ FeatureManager to find an option called Display Warnings, where you can choose Always, Never, and All but Top Level. When a sketch contains sketch relations with errors, they display as warning signs on the sketch, and will propagate to the top level of a part or assembly if you have selected the Always option.
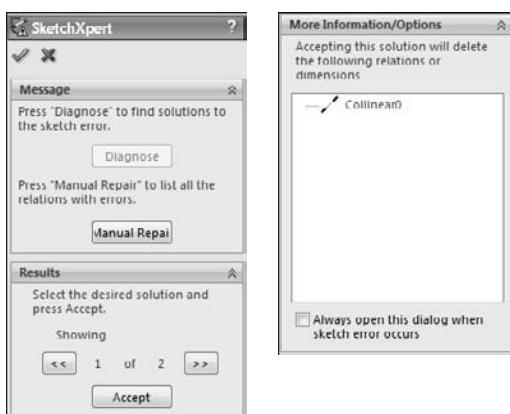
SketchXpert

The SketchXpert, shown in Figure 6.3, can help you to diagnose and repair complex sketch relation problems. The Diagnose button at the top creates several possible solutions that you can toggle through using the forward and backward arrow buttons in the Results panel. The Manual Repair button displays all the relations with errors in a window where you can delete them manually.

By selecting the option at the very bottom of the dialog box, you can make the SketchXpert appear any time that a sketch error occurs. To display the SketchXpert manually instead of automatically, you can access it by right-clicking in a sketch.

FIGURE 6.3

The SketchXpert dialog box

Copying and Moving Sketch Entities

SolidWorks offers several different tools to help you move sketch entities around in a sketch. In SolidWorks, it is usually recommended to keep the sketch as simple as you can, and to create patterns using feature patterns rather than sketch patterns. The following section discusses the main tools for moving and copying sketch entities.
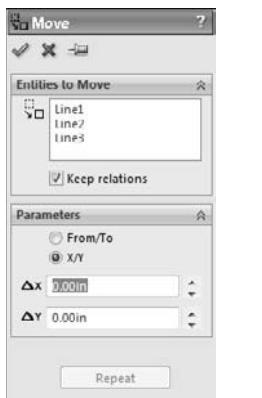
Move entities

Move entities enables you to move selected sketch entities by either selecting From and To points, or by typing in XY coordinates for the move. When the Keep relations option is unselected, the Move tool automatically detaches sketch segments whose endpoints are merged, as shown in Figure 6.4. If Keep relations is selected, SolidWorks moves the entities and tries to maintain the sketch relations and merged points. All the tools have a pushpin icon in the interface, which enables you to use them many times when the pushpin icon is pushed in; they are deactivated after one use if the pushpin icon is not pushed in.

FIGURE 6.4

Using the Move tool

Rotate entities

Rotate entities rotates selected entities in a sketch in the same way that Move entities works. You can drag the angle or type it in manually. The green check mark icon is on the RMB, as shown in the cursor display in Figure 6.5.
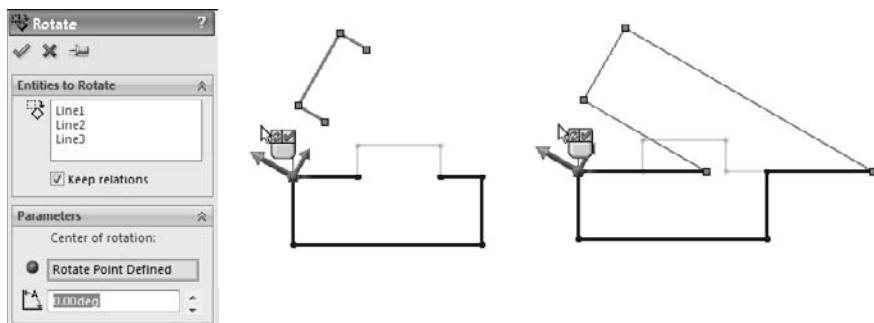
The Keep Relations option does not actually keep any relations — it deletes the Horizontal and Vertical relations in the sketch, as shown in Figure 6.5 — but it does keep the merged endpoints, as shown in the right-most image of Figure 6.5. This can be useful, especially considering how many sketch relations it would take to make a sketch move like this naturally.

Using the Rotate tool

Copy entities

The Copy entities tool works exactly like the Move entities tool, except that it copies instead of moving. The Copy entities tool enables you to move selected sketch entities by either selecting From and To points, or by typing in XY coordinates to copy. SolidWorks copies the entities and tries to maintain the sketch relations and merged points.

Scale entities

Scale entities is one of those functions probably best left alone. This is because the results are erratic and unpredictable, particularly if there are dimensions on the sketch. This tool works on a selection of entities, particularly on an isolated selection that is not connected to other entities in the sketch. The PropertyManager for the Scale Entities tool is shown in Figure 6.6.

Modify Sketch
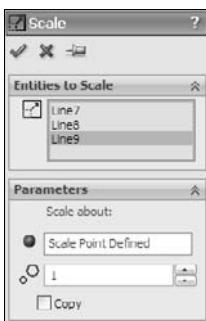
The Modify Sketch tool has been available in SolidWorks for a long time, but it has been superseded by some of the newer tools mentioned previously. However, it still has some unique functionality that is not covered by any other sketch tool. Modify sketch works on the entire sketch rather than on selections from the sketch, and it works best if there are no external relations between sketch entities and anything outside the sketch. It can also work on a sketch without the sketch being active. While most feature and tool interfaces have been moved to the PropertyManager, Modify sketch still uses a dialog box, as shown in Figure 6.7, that floats in the graphics window.
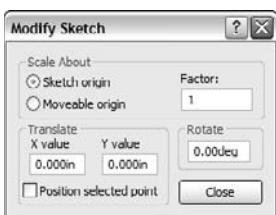
FIGURE 6.6

The Scale PropertyManager

FIGURE 6.7

The Modify Sketch dialog box

The Modify Sketch dialog box enables you to perform the following functions:

- **Scale About.** The Scaling function in the Modify Sketch tool enables you to scale about either the part Origin or the Moveable Origin. The Moveable Origin is the black origin symbol with knobs on the ends of the axes and at the intersection. The Moveable Origin can be moved and even snapped to entities that are internal or external to the sketch.

- **Translate.** The Translate function of the Modify Sketch tool enables you to click and drag to move the entire sketch, or to select a point and move it to a specific set of coordinates that you type in. If the sketch is dragged onto an external entity and picks up an automatic relation, then a message may appear saying that you can now use Modify sketch only for rotating the sketch because there is an external relation.

- **Rotate.** The Rotate function of the Modify Sketch tool enables you to position the Moveable Origin to act as the center of rotation, and to either type in a rotation angle or drag with the right-mouse button to rotate, as indicated by the cursor.
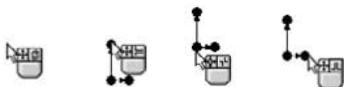
When you place the cursor over the knobs on the Moveable Origin, the cursor symbols change to indicate the functionality of the RMB. These cursors are shown in action in Figure 6.8. The cursors enable mirroring about X, Y, or both simultaneously.

FIGURE 6.8

The Modify Sketch tool cursors

Note

The Modify Sketch tool is one of my favorite sketch editing tools to use because it is straightforward and doesn't do anything unexpected. It was also an original SolidWorks development, not a tool meant to duplicate existing AutoCAD functionality like some of the previously mentioned sketch tools.

The one thing about Modify Sketch that many people find unsettling is that the red sketch origin moves and rotates along with the rest of the sketch. Once you make peace with the fact that you can't use the red sketch origin for much anyway, this becomes unimportant. ■

Copy and paste

Probably the simplest way to copy sketch entities in a sketch is to select the entities and use Ctrl+C and Ctrl+V or one of the many other methods available for this purpose (such as the RMB button menu, the Edit menu, and Ctrl+dragging).

In addition to copying selected entities within an active sketch, you can also select a sketch from the FeatureManager and copy or paste it to a selected plane or planar face. This creates a new sketch feature in the FeatureManager that is not related to the original, although it does maintain internal dimensions and relations. (External relations are not copied with the sketch.) This is particularly useful when setting up certain types of lofts that use several profiles that can be created from a single copied profile. Copying and pasting is a fast and effective method of putting sketches on planes.

Simple drag

If a selected set of sketch entities has no external relations, then you can select it as a group and move it without distorting or resizing the sketch. For best results with this, avoid dragging end points.

Derived sketch
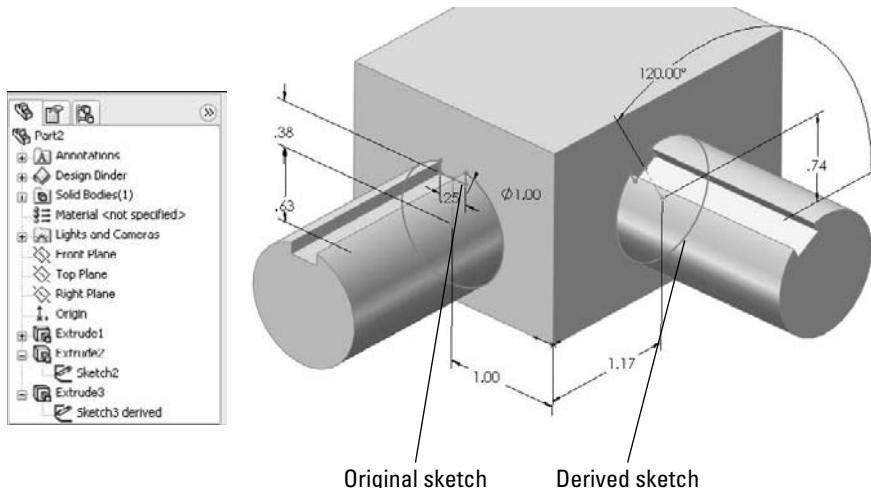
A derived sketch is a parametrically linked copy. The original parent and derived sketches do not need to have any geometrical relation to one another, but when the parent sketch is changed, the dependent derived copy is updated to stay in sync.

To create a derived sketch, you can select a plane or planar face, Ctrl+select the sketch of which you want to make the parametric copy, and then choose Insert ➪ Derived Sketch.

When you create a derived sketch, you cannot change its shape; it works like a block of a fixed shape driven by the parent. However, you can change the position and orientation of the derived sketch. Figure 6.9 shows a derived sketch and its parent. Modify Sketch is a great tool to use for manipulating derived sketches that are not related to things outside the sketch, especially for mirroring or rotating.

A derived sketch and its parent

Original sketch   Derived sketch

Using Sketch Pictures

Sketch pictures are images that are placed in a sketch on a sketch plane. You can size and rotate the images, give them a transparent background, trace over them, and suppress them. They display as a child of the sketch in the FeatureManager. Image types that you can use as sketch pictures are BMP, GIF, JPEG, TIFF, PNG, PSD, and WMF.

To bring a picture into a sketch, the sketch must first be active. Click Sketch Picture on the Sketch toolbar (it is not there by default, and so you may need to drag it onto the Sketch toolbar from the Tools➪Customize➪Commands dialog box). You can also access this command by choosing Tools➪Sketch Tools➪Sketch Picture from the menu. You cannot use sketch pictures in assembly sketches, but you can use them in a part sketch in an assembly.

To change the size of a sketch picture, you can double-click it and drag one of the handles around the outside of the image. When the picture comes into the sketch, it is usually too big, having been sized at a ratio of 1 pixel to 1 mm. To size a picture accurately, you should include a ruler or an object of a known size in the image. If you cannot do this, the next best thing is to guess the size.

Draw a line in your sketch and dimension it to approximately the size of something that is recognizable in the image, and then move the image by clicking and dragging it to lay the dimensioned sketch entity as close over the object in the image as possible.

You can rotate and mirror images, as well, using the Sketch Picture PropertyManager. Images are opaque, and you cannot see the model through them, but at the same time, you also cannot see the images through the model. They are like flat pieces of paper that are pasted to the model or hanging in space.

You can add transparency to images, either by selecting a color or by using the built-in transparency in the image file. When you select a color to be transparent, you will also need to increase both the Matching Tolerance and the Transparency value sliders, which are by default set to their minimum values.

Caution

If a sketch picture has had user defined transparency applied to it, and you double-click the picture, SolidWorks automatically bumps you into the eyedropper mode, which selects a color to be transparent. A single extra click in this mode can make a mess of your Sketch Picture transparency settings by changing the selected transparency color. ■

Sketch pictures cannot be shown on a drawing associatively. The only way to do this is to capture an image of the sketch picture that is being shown in the model, and put this image in the drawing. PhotoWorks does not use sketch pictures, either, and PhotoWorks Decals are a separate item altogether.

Tip

Although the most common use for the sketch picture is as a tracing guide, you can use it for a wide variety of other purposes. For example, any sort of logo, decal, or display that is on a flat surface can be shown as a sketch picture. ■

Best Practice

Best practice for using sketch pictures is to put them into a separate sketch near or at the top of the FeatureManager. Even though you can have sketch entities in a Sketch Picture sketch, I recommend keeping them in separate sketches. This is because when you use the sketch entities for an extrude or a loft guide curve, this sketch will be consumed under that feature, meaning the image becomes buried somewhere in your model rather than being easily accessible at the top of the FeatureManager. ■
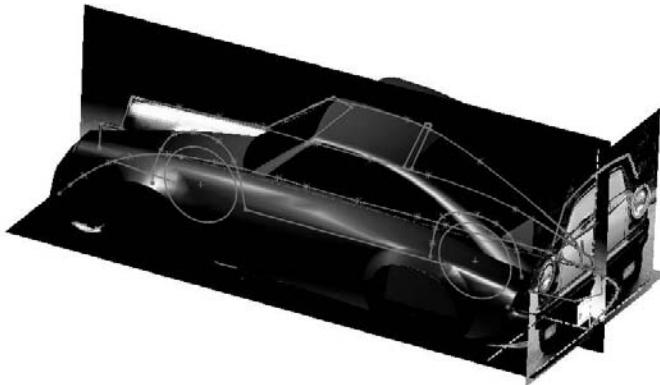
Three views

When building a model from images, it is often helpful to have three or more images from orthogonal views, similar to re-creating a part from a 2D drawing. If you have a left and a right view, it may be a good idea to put them on planes that are slightly separated so that the images are not exactly on top of one another, which makes them both hard to see. Putting them on slightly offset planes means that one will be clearly visible from one direction and the other visible from the other direction.

Each sketch picture must be in a separate sketch. Figure 6.10 demonstrates the use of multiple sketch pictures to trace the outline of a vehicle, with the partially complete model shown with the images.

FIGURE 6.10

Using multiple sketch pictures

Additionally, you can put multiple sketch pictures inside a single sketch if you want to do that. Both images show up in the FeatureManager, and both can be displayed at the same time, although you may have difficulty if you want to put them on top of one another.

Compensate for perspective

When taking digital photographs to be used as sketch pictures in SolidWorks, you have to consider how perspective affects the image. Perspective can make it difficult to size items in the foreground or background. You should be aware of this, as well as that objects at different distances from the camera will appear at distorted sizes. If you are taking the pictures that will be used as sketch pictures, you can minimize the effects of perspective by standing farther away from the object and using zoom on the camera if possible.

Estimate sharp edges

When you are drawing a sketch of an object, you are usually drawing theoretically sharp corners of the model. Real parts usually have rounded corners, so you may have to use your imagination to project where the 3D surfaces would intersect at an edge minus the fillets.

When you are reverse-modeling a part from images, you are not using an exact science. It is better than not being able to put pictures into the sketch, but there is nothing about it that can be considered precise.
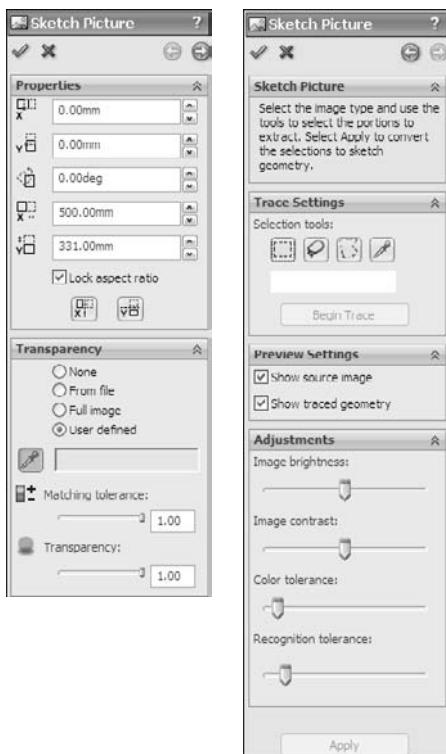
Auto Trace

Auto Trace is an add-in that you can select by choosing the Tools ⇨ Add-ins menu. Auto Trace is intended to trace between areas of contrast in sketch pictures, creating sketch entities. To use Auto Trace, make sure the add-in is activated. Activating the Auto Trace add-in activates a set of arrows at the top of the Sketch Picture PropertyManager. There is nothing to identify the functionality with the Auto Trace name. Figure 6.11 shows the Sketch Picture and Auto Trace PropertyManagers. The sliders for the Auto Trace functionality do not point out which end is high and which is low.

FIGURE 6.11

The Sketch Picture and Auto Trace PropertyManagers

Auto Trace works best with solid blocks of black and white in the Sketch Pictures. To achieve this, you may need to use image processing software and reduce your picture to a two-color (black and white) bitmap, TIF, or PNG image. Even if this pre-processing gives perfect results, don't expect much from Auto Trace.

I can't imagine a situation in which I would use this myself or recommend anyone else use it. In all cases, including idealized demonstration images or those in which traced images would be of the most benefit to the user, such as logos with complex curvature, I believe it would be faster and more accurate to do the tracing manually, even it means using splines.
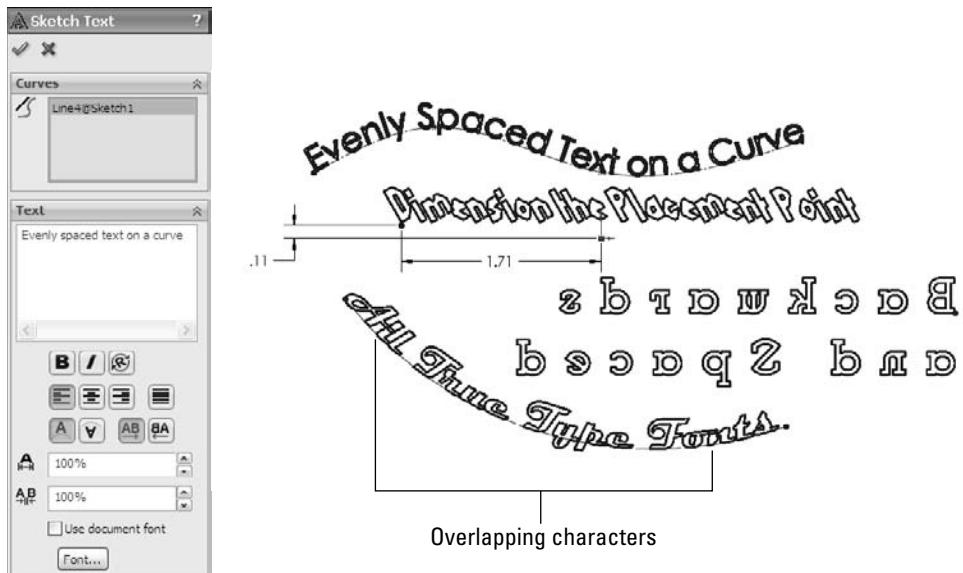
Using Sketch Text

Sketch text uses TrueType fonts to create text inside a SolidWorks sketch. This means that any TrueType font that you have can be converted to text in solid geometry; this includes Wingdings and symbol fonts. Keep in mind that some characters in certain fonts do not convert cleanly into SolidWorks sketches. Sketch text still has to follow the rules for sketching and creating features such as closed contours, as well as not mixing open and closed contours.

You can make sketch text follow a sketch curve; to space it evenly along the curve, you can control character width and spacing, as well as overall size, by specifying points or actual dimensions. Sketch text can also be justified right, left, centered, and evenly, as well as reversed, rotated, and flipped upside down. Figure 6.12 shows the Sketch Text PropertyManager and some examples of sketch text options.

FIGURE 6.12

Examples of sketch text

Overlapping characters

The icons in the Sketch Text PropertyManager are self-explanatory, other than the Rotated Text option, which rotates individual letters, and not the whole string of text.

You can use the Sketch Text tool multiple times in a single sketch to make pieces of text with different properties. Each string of text has a placement point located at the lower left of the text. This point can be given sketch relations or dimensions to locate the text.

If the text overlaps in places, as shown in Figure 6.12, you can correct this in a couple of ways. First, you can extrude it with the Merge option unselected so that each letter is created as a separate solid body. You can also explode sketch text so that it becomes simply lines and arcs in a sketch, which you can edit the same as any other sketch. You can also adjust the Width Factor and Spacing settings.

Starting in the 2010 version of SolidWorks, you can link the text to a custom property. This means that sketch text can be changed with configurations. Configurations are covered in a later chapter. The text used to extrude a feature can come from Custom Properties, which can be driven by a design table or directly through the Sketch Text PropertyManager.

Using Colors and Line Styles with Sketches

Custom colors and line styles are usually associated with drawings, not sketches; in fact, they are most valuable when used for drawings. In sketches, this functionality is little known or used, but is still of value in certain situations.

Color Display mode

In drawings, you can use the Color Display Mode button to switch sketch entities on the drawing between displaying the assigned line or layer color and displaying the sketch status color. It has exactly the same effect in part and assembly sketches.

When you select the button, the sketch state colors are used. When the button is not selected, any custom colors that you have applied to the sketch entities appear. If the button is not selected and you have not applied colors to the entities, then the default sketch state colors are used.

You can use sketch colors for emphasis, to make selected sketch entities stand out, or to make sketches with various functions immediately distinguishable. Color Display mode only has an effect on an active sketch. Once a sketch is closed, it returns to the gray default color for inactive sketch entities.

Line color

Line color enables you to assign color to entities in an active sketch. The Color Display Mode tool determines whether the assigned color or the default sketch status colors are used.

Edit sketch or curve color

You can use the Edit Sketch Or Curve Color tool to assign color to an entire sketch. The color that you assign to sketches in this way displays only when the sketch is inactive, instead of the default gray color. The sketches also follow the toggle state of the Color Display Mode button. For example, if the Color Display Mode button is selected, then inactive sketches display as gray. When the Color Display Mode button is unselected, then inactive sketches display in any color that you have assigned by using the Edit Color tool.

Line thickness and line style

The Line Thickness and Line Style tools function independently from the Color Display Mode button, but they are still used only when the sketch is active. As soon as a sketch that contains entities with edited thickness and style is closed, the display goes back to the normal line weight and font.

To assign a thickness or a style, you can select the sketch entities to be changed, click the button, and then select the thickness or style. Although a single sketch entity may have only a single thickness or style, you can use multiple thicknesses or styles within a single sketch. Figure 6.13 shows a sketch with the thickness and style edited.

Cross-Reference

Line thickness and line styles are covered in more detail in the discussion of drawings in Chapter 20. ∎

FIGURE 6.13

A sketch with edited line thickness and line style

Using Other Sketch Tools

SolidWorks has a lot of functionality that overlaps between multiple topics. The following tools could appear in other sections of the book, but I include them here because they will help you work with and control 2D sketches in SolidWorks. Almost everybody who opens the SolidWorks software at one time or another has to use a sketch, so these tools could be applied by a wide swath of users.

RapidSketch

As the name suggests, RapidSketch is meant to help you rapidly create a number of sketches on different planes. As you move a sketch cursor over flat faces of a model, the faces highlight to indicate that you can start a new sketch there.

The workflow with this tool is that you start in one sketch, with an active sketch tool, move the cursor over another plane or face without exiting the first sketch, and start sketching the entity on the new plane.

The only real downside of using RapidSketch is that if you sketch on a particular plane or face where other planes or faces might be visible in the background, SolidWorks might interpret certain selections as trying to change sketch planes. To get back to a previous sketch, deactivate the current sketch tool (for example, by pressing Esc) and double-click the previous sketch you want to return to. To move to a later sketch, use the normal sketch exiting techniques.

RapidSketch is a rarely used function in SolidWorks. It has been available for several releases now, but it has not caught on with users. I have yet to see a compelling case for its use.

Sensors

You can add Sensors in the SolidWorks FeatureManager for parts and assemblies by right-clicking the Sensors folder and selecting Add Sensor. You can find the Sensors folder at the top of the FeatureManager. If you cannot find the Sensors folder, choose Tools ➪ Options ➪ FeatureManager and make sure the Sensor folder is set to Show.
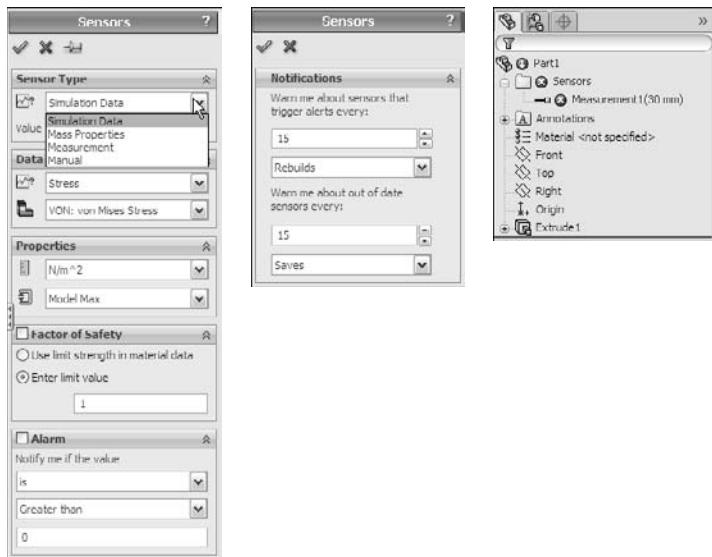
You are not limited to using sensors only when working with sketches; you can use them outside of sketches in parts and assemblies to warn you when various types of parameters meet various types of criteria.

Figure 6.14 shows the Sensor PropertyManager. You can create sensors for measurements, simulation data, or mass properties. The reason I have included Sensors in this chapter is the measurement options, which enable you to select a dimension and set a range of values or criteria for which you want to be notified. The dimension can be a driving (black) sketch dimension, a driven (gray) dimension on a sketch, or even a driven dimension placed directly on solid geometry.

The third image shows what happens when a sensor finds a condition that you asked it to notify you about.

The Sensor PropertyManager

In addition to turning Sensor alarms on or off, you can also suppress Sensors when they are no longer needed or to improve performance.

Sensors are a great way to keep an eye on particular values, such as wall thickness or clearance between parts. You can use a Sensor to monitor any value you want to monitor but don't drive directly.

Metadata for sketches

Metadata in SolidWorks is non-geometrical text information. Metadata is particularly helpful as keywords in searches as well as in Product Data Management (PDM) applications. If you don't use metadata within your CAD documents, it can be easy to forget that it is there at all.

The sources for storing metadata in SolidWorks files are

- Sketch and feature names
- Sketch and feature comments (access comments via the RMB menu)
- Custom Properties
- Design Binder documents
- Tags for features (located on Status Bar in the lower-right corner)

Metadata searches can be particularly useful in large assemblies or parts with long lists of features that you need to access for various reasons. You can conduct searches for metadata through the FeatureManager Filter at the top of the FeatureManager. The Advanced Search function in assemblies can also search metadata sources. SolidWorks Explorer is a good first-level data management solution that can search, display, and edit metadata and previews. Windows Explorer can also search properties and tags.

Construction geometry

In SolidWorks, the only construction geometry that can be created directly is the construction line. All other sketch entities can be converted to construction geometry by selecting the Construction Geometry option within the sketch entity's PropertyManager or by using the Construction Geometry toggle toolbar button.

SolidWorks terminology is inconsistent, because it sometimes refers to construction lines as centerlines. The two are really the same thing. Centerlines are used for revolved sketches and mirroring, but there is no difference between a centerline and a construction line in SolidWorks.

Construction geometry is useful for many different types of situations. I use it frequently for reference sketch data. You can make sketch relations to construction geometry, and can use it for layout sketches or many other purposes limited mainly by your needs and imagination.

Tutorial: Editing and Copying
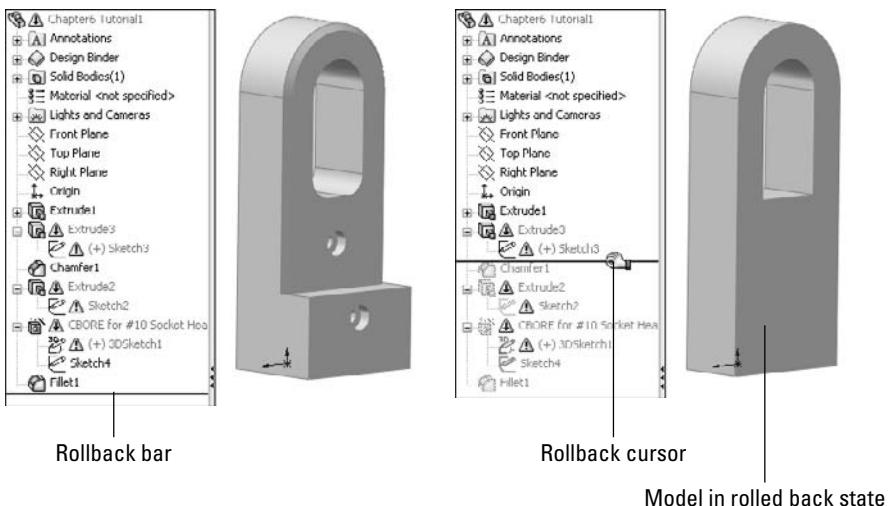
This tutorial guides you through some common sketch relation editing scenarios and using some of the Copy, Move, and Derive tools. Follow these steps to learn about editing and copying sketches:

1. **Open the part named** `Chapter6 Tutorial1.sldprt` **from the CD-ROM.** This part has several error flags on sketches. In cases where there are many errors, it is best to roll the part back and go through the errors one by one.

2. **Drag the rollback bar from just after the last fillet feature to just after Extrude3.** If Extrude3 is expanded so that you can see Sketch3 under it, then drop the rollback bar to after Sketch3. If a warning message appears, telling you that Sketch3 will be temporarily unabsorbed, then select Cancel and try the rollback again. Figure 6.15 shows before and after views for the rollback.

3. **Edit Sketch3 and deselect the Sketch Relations display (View ⇨ Sketch Relations).** Relations with errors will still be displayed. Click Display/Delete Relations on the toolbar (the Eyeglasses tool), and set it to All in This Sketch. Notice that all the relations conflict, but only one is unsolvable: the Equal Radius relation. This appears to be a mistake because the two arcs cannot be equal.

4. **Delete the Equal Radius relation.** Select the relation in red and click the Delete button in the PropertyManager. (You can also press the Delete key on the keyboard.) The sketch is still not fixed.

FIGURE 6.15

Rolling the part back to Extrude3

Rollback bar   Rollback cursor

Model in rolled back state
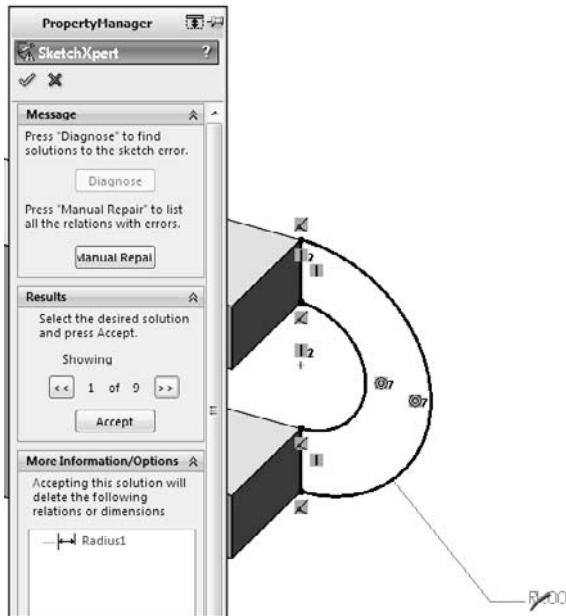
5. **Click the green check mark icon to close the Display/Delete Relations PropertyManager.**

6. **Right-click the graphics window and select SketchXpert; then click Diagnose.**

7. **Using the double arrows in the Results panel, toggle through the available solutions.** All the solutions except one remove sketch relations. Accept the one solution that removes the dimension. This is shown in Figure 6.16.The sketch no longer shows errors in the graphics window, but it still does in the FeatureManager.

8. **Close the sketch.** Notice that the error flag does not disappear until the sketch has been repaired and closed.

9. **Use the rollback bar to roll forward to after Extrude2 and Sketch2.** Figure 6.17 shows the tool tip message that appears if you place the cursor over the feature with the error. With time, you will begin to recognize the error messages by a single keyword or even by the shape of the message text. This message tells you that there is a *dangling* relation — a relation that has lost one of the entities.

10. **Edit the sketch (see Figure 6.18).** If you show the Sketch Relation icons again, the errors will be easier to identify. When you use Display/Delete Relations (Tools ⇨ Relations ⇨ Display/Delete Relations), the first two Coincident relations appear to be dangling. Clicking the relation in the Relations panel of the Display/Delete Relations PropertyManager shows that one point was coincident to a line and the other point was coincident to a point.

223
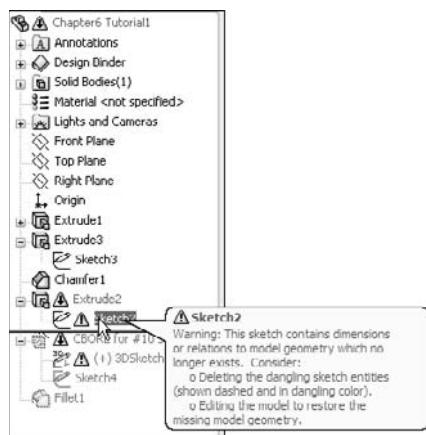
FIGURE 6.16

Using the SketchXpert to resolve an overdefined sketch
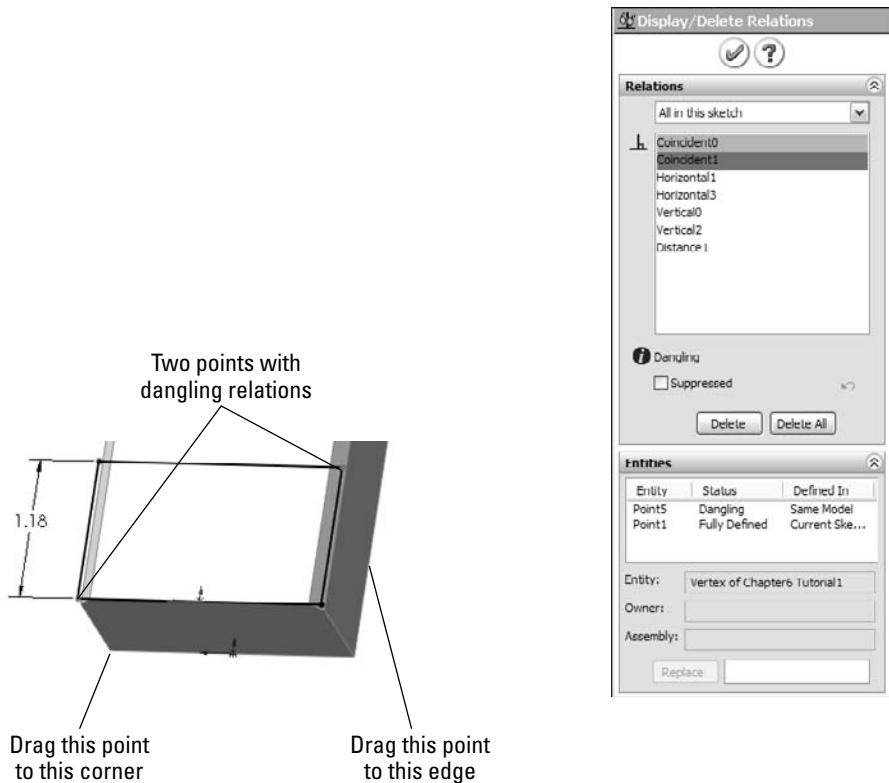
FIGURE 6.17

A tool tip gives a description of the error

FIGURE 6.18

Fixing dangling errors

11. **Click the name of the dangling entity in the Entities panel of the PropertyManager; then click the vertex indicated in Figure 6.18 in the Replace box at the bottom.** When you have fixed the errors, exit the sketch and confirm that the flag is no longer on Sketch2.

    An easier way to repair the dangling relation is to click on the dangling sketch point once. It will turn red. Next, drag the point onto an entity that you want to reattach the relation to.

12. **Exit the sketch.**

13. **Drag the rollback bar to just before CutExtrude1. Edit 3DSketch1.** This sketch is overdefined. If the Sketch Relations are not selected at this point, then select them again.

Tip

Because selecting and deselecting the display of the sketch relations in the graphics window is a task that you will perform many times, this is a good opportunity to set up a hotkey for this function. As a reminder, to set up a hotkey, choose Tools ⇨ Customize ⇨ Keyboard, and in the Search box, type relations. In the Shortcut column for this command, select a hotkey to use. ■

14. **Double-click any of the relation icons; the Display/Delete Relations PropertyManager appears.** Notice that one of the sketch relations is a Fixed relation. Delete the Fixed relation, and exit the sketch.

15. **Right-click anywhere in the FeatureManager and select Roll To End.**

16. **Click CutExtrude1 in the FeatureManager so that you can see it in the graphics window and then click a blank space to deselect the feature.**

17. **Ctrl+drag any face of the cut feature, and drop it onto another flat face.** The Ctrl+drag function copies the feature and the sketch, but the external dimensions and relations become detached. This will only work if Instant3D is unselected.

18. **Click Dangle in response to the prompt.** This means that you will have to reattach some dangling dimensions rather than re-creating them. Edit the newly created sketch, which now has an error on it.

19. **Two of the dimensions that went to external edges now have the olive dangling color.** Select one of the dimensions; a red handle appears. Drag the red handle and attach it to a model edge. Do this for both dimensions. The dimensions update to reflect their new locations. Exit the sketch and verify that the error flag has disappeared.

20. **Expand CutExtrude1, and select Sketch5 under it.** Ctrl+select a flat face on the model other than the one that Sketch5 is on. In the menu, choose Insert ⇨ Derived Sketch. You are now in a sketch editing the derived sketch.

21. **The sketch is blue, and so you should be able to resize it, right?** No, it doesn't work that way for derived sketches. You can test this by dragging the large circle; it only repositions the entire sketch as a unit.

22. **Dimension the center of the large circle to the edges of the model.**

23. **Drag the smaller circle, and notice that it swivels around the larger circle.** Create an angle dimension between the construction line between the circle centers and one of the model edges. Notice that the sketch is now fully defined.

24. **Exit the sketch, and look at the name of the derived sketch in the FeatureManager.** The term *derived* appears after the name, and the sketch appears as fully defined.

25. **Right-click the sketch and select Underive Sketch.** Notice that the sketch is now underdefined. The Underive command removes the associative link between the two sketches.

Tutorial: Controlling Pictures, Text, Colors, and Styles

This tutorial guides you through some of the miscellaneous functions in sketches, and shows you what they are used for and how they are used. Follow these steps to learn how to control these items:

1. **Open a new part using a template with inches as units.** Open a sketch on the Front plane and draw a construction line starting from the origin 12 inches down (negative Y) away from the Origin.

2. **Insert a sketch picture in this sketch.** Use `Sketch Picture 1.tif` from the CD-ROM for Chapter 6.

3. **Resize the image so that the endpoints of the construction line are near the centers of the holes on the ends of the part.** To move the image, just double-click it first, and then drag it. To resize it, drag the corners.

4. **In the Transparency panel of the Sketch Picture PropertyManager, select the Eyedropper tool and click in the white background of the image.** Make sure that the color field next to the Eyedropper tool changes to white.

5. **Slide the Transparency and Matching Tolerance sliders all the way to the right, or type** 1.00 **in the number boxes.**

6. **Close the sketch, and rename it** Sketch Image Front View.

7. **Put the image** `Sketch Picture 2.tif`, **also from the CD-ROM, in a sketch on the Right plane, and resize it to fit with the first image.** Center it symmetrically about the Origin. Also, set the transparency to the same setting as the first image.

8. **Open a new sketch, also on the Front plane, and draw two circles to match the features on the ends.** Extrude them using a Mid Plane extrusion to match the image in the other direction (about 2.5 inches), as shown in Figure 6.19.

FIGURE 6.19
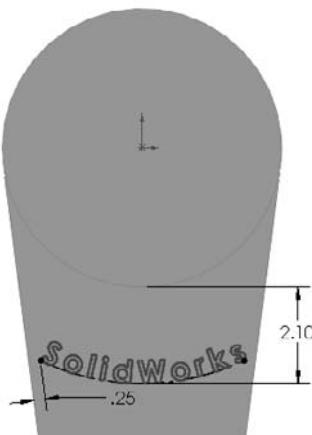
Using sketch pictures

9. **Open another new sketch on the Front plane and draw the tangent lines to form the web in the middle of the part. Use the automatic relations to draw the lines tangent to the two cylinders. It is easiest if you the Front View for this.** Close the sketch to make a solid extrusion. Extrude this part .5 inches Mid Plane.

10. **Open a new sketch on the face of the large flat web that you created in the previous step, and offset the arc edge of the larger circular boss by 2.10 inches.**

11. **Change the arc to a construction arc and drag its endpoints to approximately the position shown in Figure 6.20.** The endpoints of the arc are blue after you drag them. Give them a Horizontal relation, and then dimension them.

    To create the 2.10 dimension as shown, select the arc and circle with the Dimension cursor while pressing down the Shift key.

FIGURE 6.20

Creating an offset arc

12. **Choose File⇨ Properties.** Make sure the Custom tab is active, and type **Sketch Text** in the first open box of the Property Name column. Make sure the Type is set to Text, and in the Value field, type **SolidWorks**. Click OK when you are done.

13. **Choose Tools⇨ Sketch Entities⇨ Text to initiate the creation of sketch text.**

14. **Select the construction arc to go into the Curves window.**

15. **Below the Text window, click the Link to Property button. Select Sketch Text from the drop-down list. Select the Full Justify option, then click the green check mark icon to accept it.**

16. **Deselect the Use Document Font option, click the Font button, and then set the Units to .50 inches.** Click the Bold button to make the text thicker. Click OK to exit the dialog box. Click the green check mark icon to exit the sketch text, and then exit the sketch.

17. **Extrude the text to a depth of .050 inches with 3 degrees of draft.** The part at this point resembles Figure 6.21.

Performance

Sketch Text is a real performance killer. The more text that you use, the longer it takes to extrude. Draft on the extrusion adds to the time required.∎

FIGURE 6.21

Creating extruded text
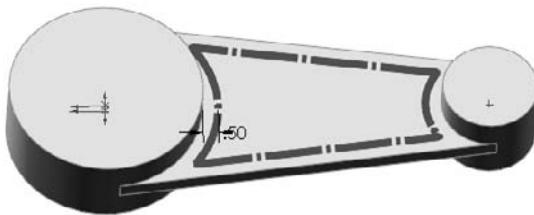
18. **Select the flat face on the other side of the part from where you just extruded the text, and open a sketch.**

19. **Select the face and click the Offset button to make a set of sketch entities offset to the inside of the face by .50 inches.** Remember that you may have to reverse the offset to get it to work properly.

20. **Open the Line Format toolbar (right-click any toolbar other than the CommandManager and select Line Format).**

21. **Select all the sketch lines and change their color using the Line Color tool.** Change the line thickness and the line style using the appropriate tools. The sketch now looks something like Figure 6.22.

Using line thickness and line style

22. **When you click the Color Display Mode tool, the colors return to regular sketch colors.** When you exit the sketch, the line weight and style also return to normal.

Tutorial: Using Metadata
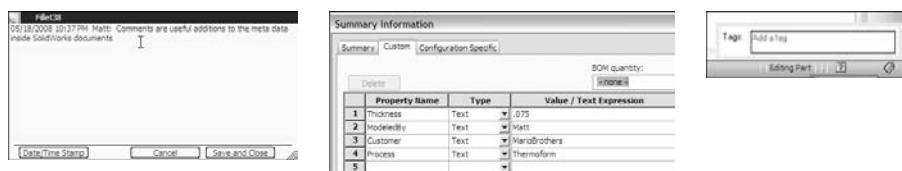
If you integrate the use of metadata into your company's modeling process, your SolidWorks models can be a resource for much more than just geometrical data. In this tutorial, discover the hidden treasure of extra information stored as metadata in this model.

1. **Open the part from the CD-ROM called** `Chapter 6 – Dial Cover.sldprt.`
2. **Check the Custom Properties in this file by choosing File ⇨ Properties.** Notice the Thickness and Process properties in particular. All the metadata entry interfaces are shown in Figure 6.23.

Metadata entry interfaces

3. **Add a Custom Property with the Property Name Material, type Text, and value ABS.** The Custom Property interface is located at File ⇨ Properties.
4. **Check the Comments in this part.** Notice that a Comments folder exists near the top of the FeatureManager. Inside it is a list of the features for which I have written comments.

5. **Add a Comment by right-clicking the VarFillet3 feature, selecting the Comment flyout arrow, clicking the Add Comment option, clicking the Date/Time Stamp button, and adding a comment that uses the word Blend.**

6. **Check the Tags for the part by clicking the small yellow tag in the lower-right corner of the Status Bar, then click any feature, and double-click in the Tags interface box.**

7. **Add a Tag by selecting the Cut-Extrude1 feature and adding the tag** pilar.

8. **Right-click any item in the FeatureManager and select Go To from the options.**

9. **Type** 37 **in the box and click the Find Next button.** The FeatureManager should highlight a feature near the bottom of the tree named Fillet37.

10. **Click Fillet37 in the Feature Manager and select the Zoom to Selection tool.** Zoom to Selection is a magnifying glass with an equal sign in it. The display zooms and pans to a fillet on one end of the part.

11. **Right-click a face of Fillet37 on the model and select Go To Feature (In Tree), which will select the FeatureManager if necessary and scroll to show Fillet37.** This sequence of tools shows the importance and interdependence of feature names and the actual geometry.

12. **Type the word** Thickness **in the filter at the top of the FeatureManager.** Figure 6.24 shows the result. Notice how quickly the results appear. Notice also that the metadata item that caused the feature to show in the list can be shown in a tool tip by hovering the mouse over the feature.

FIGURE 6.24

Using the FeatureManager Filter to search for metadata

13. **Click the X at the right end of the filter to restore the FeatureManager to its original state, and type the word** Pilar **instead.** Now filter for Thermoform.

Tutorial: Sketching Calculator

Sketches can be used as geometrical calculators. Parametrics can be extremely powerful when you can define relationships between geometry. In this tutorial, you will set up a sketch to calculate the complex size and location relationships between the rings of a child's stacking toy.
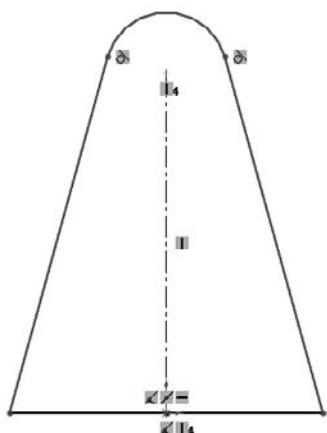
1. **Open a new part document with inch units.**

2. **Draw a pyramid with the base centered on the origin, as shown in Figure 6.25.** Do not use dimensions, but use sketch relations and construction geometry to enable it to change symmetrically.

Build a pyramid without dimensions
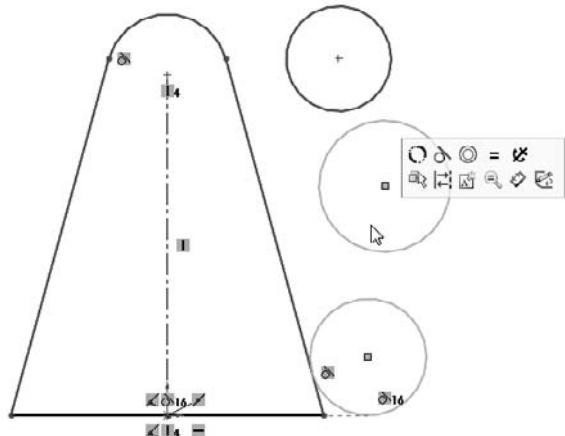
3. **Draw three circles to the right of the pyramid.** The bottom circle should be tangent to the bottom and the angled side of the pyramid. The middle and top circles should be tangent to the lower circle and the angled side of the pyramid, respectively. Figure 6.26 shows this process partially completed.

4. **Put a diameter dimension on each circle.** In the PropertyManager for each dimension, rename the dimension for **ring1dia@Sketch1**, **ring2dia**, and so on. This is shown in Figure 6.27.

5. **Place a dimension from the center of the bottom circle to the construction line, and place the dimension on the far side of the construction line from the circle.** This creates a diameter dimension.

6. **Place a dimension for the middle circle in the same way as the bottom circle, but this time, make the dimension value about 75 percent of the first diameter.**
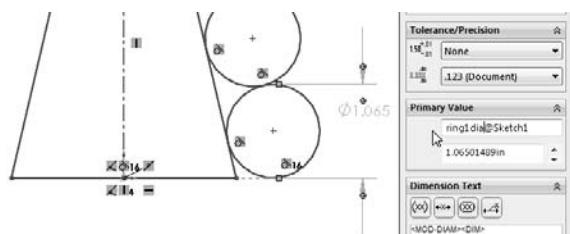
FIGURE 6.26

Sketching stacking rings

FIGURE 6.27

Naming dimensions
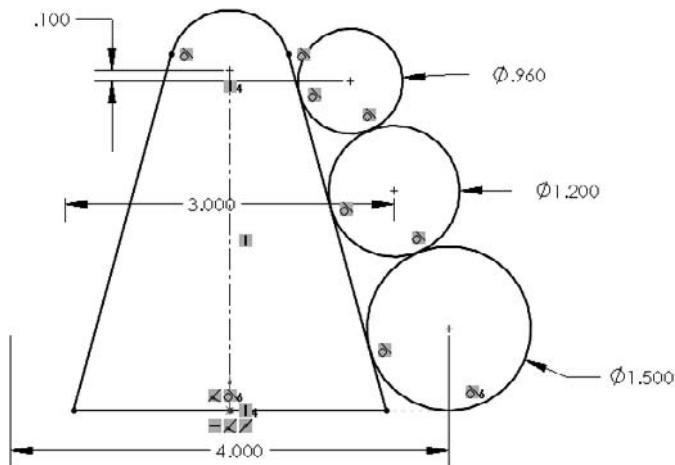
7. **Complete the sketch as shown in Figure 6.28, with all the dimensions and sketch relations.**

8. **Make changes to the sketch to see which changes it will allow and which it will not.**
   Double-click dimensions and use the wheel in the middle of the Modify dialog box to apply changes smoothly. Try changing each dimension.

FIGURE 6.28

The finished sketch

Summary

Many tools that are available in sketches are not commonly covered in the most popular sources of information, including official training manuals. The difference between a good CAD tool and a great communication tool can be minor functions that just make life a little easier, or the presentation or editing of data a little better. When you explore the capabilities of SolidWorks, it usually rewards you with functionality that is not immediately obvious.

Selecting Features

T he most frustrating part of a complex modeling job is to be able to envision a result, but not be able to create it because you do not have the tools to get it done. Worse yet is to actually have the tools but either not understand how to use them or not even realize that you have them. Getting the job done is so much more satisfying when you use the right tools and get the job done right — not just so that it *looks* right, but so that it really *is* right.

SolidWorks offers so many tools that it is sometimes difficult to select the best one, especially if it is for a function that you do not use frequently.

This chapter helps you identify which features to use in which situations, and in some cases, which features to avoid. It also helps you evaluate which feature is best to use for a particular job. With some features, it is clear when to use them, but for others, it is not. This chapter guides you through the decision-making process.

Identifying When to Use Which Tool

I am always trying to think of alternate ways of doing things. It is important to have a backup plan, or sometimes multiple backup plans, in case a feature doesn't perform exactly the way you want it to. As you progress into more complex features, you may find that the more complex features are not as well behaved as the simple features. You may be able to get away with only doing blind extrudes and cuts with simple chamfers and fillets for the rest of your career. In addition, even if you could, would you want to?

As an exercise, I often try to see how many different ways a particular shape might be modeled, and how each modeling method relates to manufacturing methods, costs, editability, efficiency, and so on. You may also want to try this approach for fun or for education.

As SolidWorks grows more and more complex, and the feature count increases with every release, understanding how the features work and how to select the best tool for the job becomes ever more important. If you are only familiar with the standard half-dozen or so features that most users use, your options are limited. Sometimes simple features truly are the correct ones to use, but using them because they are the only things you know is not always the best choice.

Using the Extrude feature

Extruded features can be grouped into several categories, with extruded Boss and Cut features at the highest level. When you use Instant3D, extruded bosses can be transformed into cuts by simply dragging them the other direction. It is unclear what advantage this has in real-world modeling, but it is an available option. As a result, the names of newly created extrude features are simply Extrude1 where they used to be Extrude-Boss1 or Extrude-Cut1.

The "Base" part of the Extruded Boss/Base is a holdover from when SolidWorks did not allow multi-body parts, and the first feature in a part had special significance that it no longer has. This is also seen in the menus at Insert ➪ Boss/Base. The Base feature was the first solid feature in the FeatureManager, and you could not change it without deleting the rest of the features. The introduction of multi-body support in SolidWorks has removed this limitation.

Cross-Reference

Multi-body parts are covered in detail in Chapter 26. ∎

Solid feature

In this case, the term *solid feature* is used as an opposite of *thin feature*. This is the simple type of feature that you create by default when you extrude a closed loop sketch. A closed loop sketch fully encloses an area without gaps or overlaps at the sketch entity endpoints. Figure 7.1 shows a closed loop sketch creating an extruded solid feature. This is the default type of geometry for closed loop sketches.

Thin feature

The Thin Feature option is available in several features, but is most commonly used with Extruded Boss features. Thin features are created by default when you use open loop sketches, but you can also select the Thin Feature option for closed loop sketches. Thin features are commonly used for ribs, thin walls, hollow bosses, and many other types of features that are common to plastic parts, castings, or sheet metal.

Even experienced users tend to forget that thin features are not just for bosses, but that they can also be used for cuts. For example, you can easily create grooves and slots with thin feature cuts.

FIGURE 7.1

A closed loop sketch and an extruded solid feature
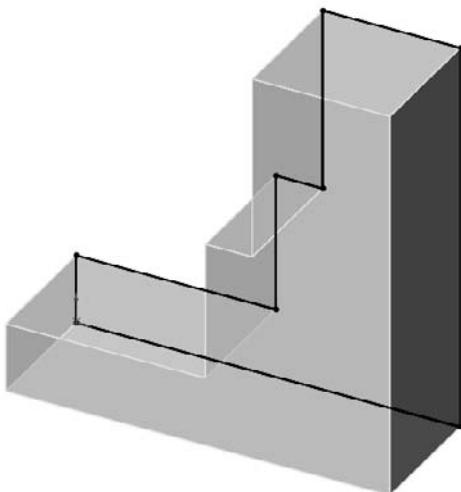
Figure 7.2 shows the Thin Feature panel in the Extruded Boss PropertyManager. In addition to the default options that are available for the extrude feature, the Thin feature adds a *thickness* dimension, as well as three options to direct the thickness relative to the sketch: One-Direction, Mid-Plane, and Two-Direction. The Two-Direction option requires two dimensions, as shown in Figure 7.2.

FIGURE 7.2

The Thin Feature portion of the Extrude PropertyManager
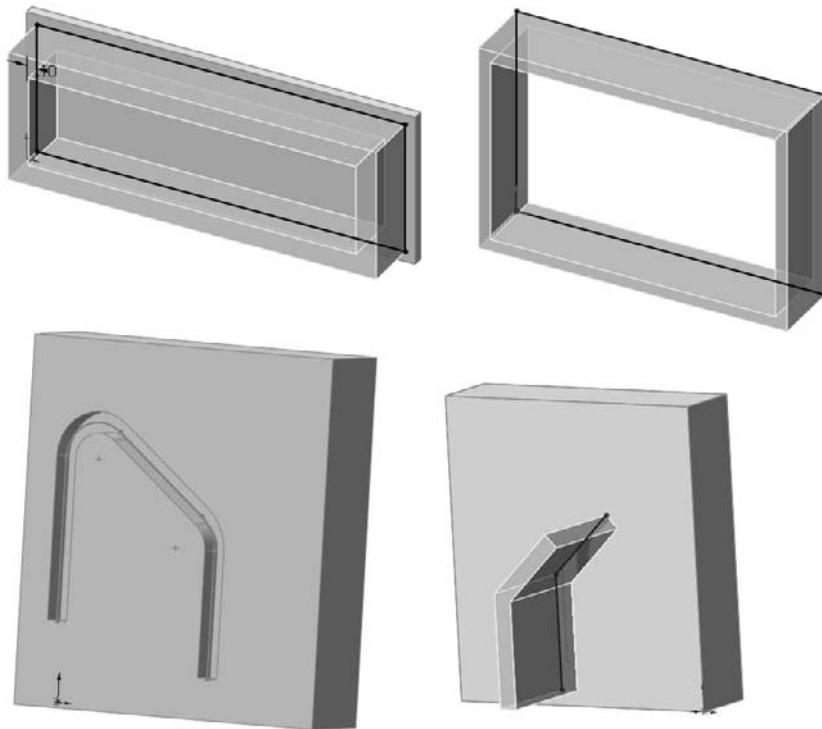
Thin feature sketches are typically simpler than closed loop sketches, which usually means that they are more robust through changes. You can create the simplest cube from a single sketch line and a thin feature extrude. However, because they are more specialized in some respects, they are not as flexible when the design intent changes. For example, if a part is going to change from a constant width to a tapered or stepped shape, thin features do not handle this kind of change. Figure 7.3 shows different types of geometry that are typically created from thin features.

FIGURE 7.3

Different types of geometry created from thin feature extrudes

Sketch types

I have already mentioned several sketch types, including closed loop and open loop. Closed loop sketches make solid features by default, but you can also use them to make thin features. Open loop sketches make thin features by default, and you cannot use them to make solid features. A nested loop is one closed loop inside another, like concentric circles. Self-intersecting sketches can be any type of sketch where the geometry crosses itself. SolidWorks also identifies sketches where three or more sketch elements intersect at a point by issuing an error if you try to use the sketch in a feature.

Sketch contours

Sketch contours enable you to select enclosed areas where the sketch entities themselves actually cross or otherwise violate the usual sketch rules. One of these conditions is the self-intersecting contour.

Best Practice
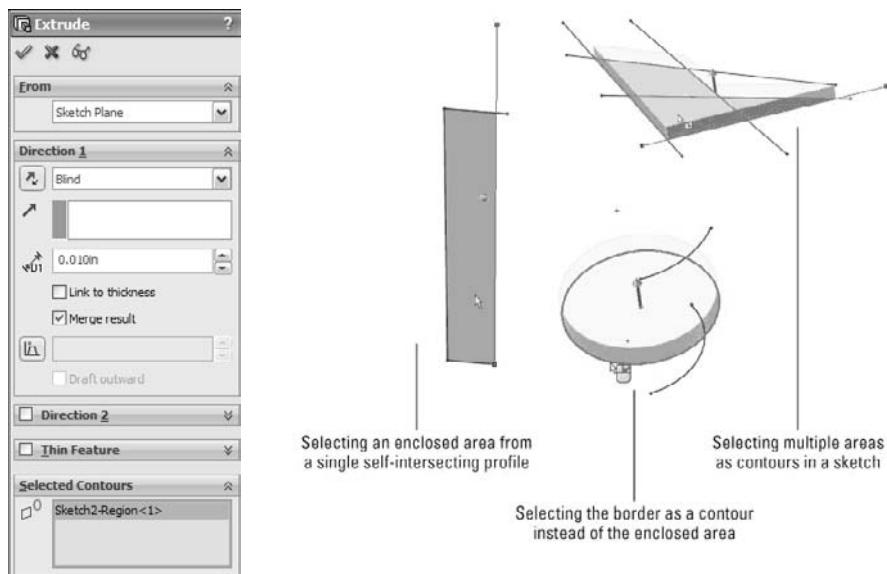
SolidWorks works best with well-disciplined sketches that follow the rules. Therefore, if you plan to use sketch contours, you should make sure that it is not simply because you are unwilling to clean up a messy sketch.

When you define features by selecting sketch contours, they are more likely to fail if the selection changes when the selected contour's bounded area changes in some way. It is considered best practice to use the normal closed loop sketch when you are defining features. Contour selection is best suited to "fast and dirty" conceptual models, which are used in very limited situations. ∎

There are several types of contour selection, as shown in Figure 7.4.

Types of contour selection

Selecting an enclosed area from a single self-intersecting profile

Selecting multiple areas as contours in a sketch

Selecting the border as a contour instead of the enclosed area

The terminology used by SolidWorks is not always logical, for example, the heading of the PropertyManager panel that you use is called Selected Contours, but the item in the selection box is called a Region. This is why you may sometimes hear Contours referred to as Regions.

3D sketch

You can make extrusions from 3D sketches, even 3D sketches that are not planar. While not necessarily the best way to do extrudes, this is a method that you can use when needed. You can establish direction for an extrusion by selecting a plane (normal direction), axis, sketch line, or model edge.

When you make an extrusion from a 3D sketch, the direction of extrusion cannot be assumed or inferred from anything — it must be explicitly identified. Extrusion direction from a 2D sketch is always perpendicular to the sketch plane unless otherwise specified.
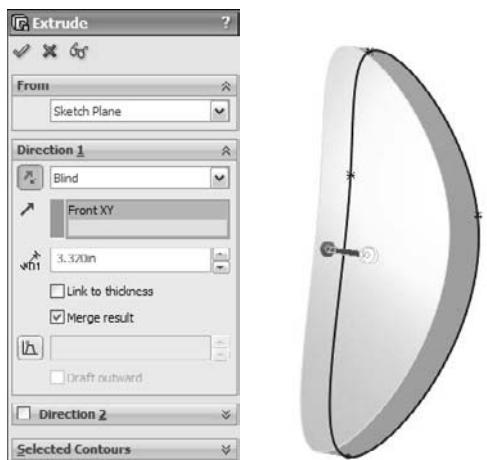
Non-planar sketches become somewhat problematic when you are creating the final extruded feature. The biggest problem is how you cap the ends. Figure 7.5 shows a non-planar 3D sketch that is being extruded. Notice that the end faces are, by necessity, not planar, and are capped by an unpredictable method, probably a simple Fill surface. This is a problem only if your part is going to use these faces in the end; if it does not, then there may be no issue with using this technique. If you would like to examine this part, it is included on the CD-ROM as `Chapter 7 Extrude 3D Sketch.sldprt`.

FIGURE 7.5

Extruding a non-planar 3D sketch

If you need to have ends with a specific shape, and you still want to extrude from a non-planar 3D sketch, then you should use an extruded surface feature rather than an extruded solid feature.

One big advantage of using a 3D sketch to extrude from is that you can include profiles on many different levels, although they must all have the same end condition. Therefore, if you have several pockets in a plate, you can draw the profile for each pocket at the bottom of the pocket, and extrude all the profiles Through All, and they will all be cut to different depths.

3D sketches also have an advantage when all the profiles of a single loft or boundary are made in a single 3D sketch. This enables you to drag the profiles and watch the loft update in real time.

Cross-Reference
Surfacing features are covered in detail in Chapter 27. Chapter 4 contains additional details on extrude end conditions, thin features, directions, and the From options. Chapter 31 also has more information on 3D sketches.■

Understanding Instant 3D

Instant 3D is the tool that enables you to use the mouse to pull arrows or handles on the screen to establish various dimension parameters for features like extrude, revolve, fillet, and even move face. Not every dimension feature parameter is editable in this way. In some cases Instant 3D offers you convenient ways to edit geometry without needing to figure out which feature is responsible for which faces. With Instant 3D, you simply pull on handles on the screen to move and resize sketches, and features including fillets.

Creating extrudes with Instant 3D

Instant 3D enables you to select a sketch or a sketch contour and drag the Instant 3D arrow to create either a blind extruded boss or cut. The workflow when using this function requires that the sketch be closed. Instant 3D cannot be used to create a thin feature, and any sketch or contour that it uses must be a closed loop. Sketches must also be shown (not hidden) in order to be used with Instant 3D.

Note

Even though the words "Instant 3D" suggest that you should be able to instantly create 3D geometry from a sketch that you may have just created, you do have to close the sketch first to get instant functionality. In this case, Instant 3D requires the sketch to be closed (as in not active) and closed (as in not an open loop. ∎

Figure 7.6 shows Instant 3D arrows for extruding a sketch into a solid and the ruler to establish blind extrusion depth. These extrusions were done from a single sketch with three concentric circles, using contour selection.

Even after you create an extruded boss, you can use Instant 3D to drag it in the other direction to change the boss into an extruded cut. When you do this, the symbol on the feature changes, but the name does not.

If you have a sketch that requires contour selection, SolidWorks automatically hides the sketch, and to continue with Instant 3D functionality using additional contours selected from that sketch, you will have to show the sketch again. This interrupts the workflow and makes using this functionality less fluid than it might otherwise be. I only mention it here so that you are aware of what is happening when the sketch disappears and the Instant 3D functionality disappears with it.

If geometry already exists in the part, and you drag a new feature into the existing solid, SolidWorks assumes you want to make a cut. However, maybe you are really trying to make a boss that comes out the other side of the part. These heads-up display icons enable you to do this. Options include boss, cut, and draft. The draft option enables you to add draft to a feature created with Instant 3D.
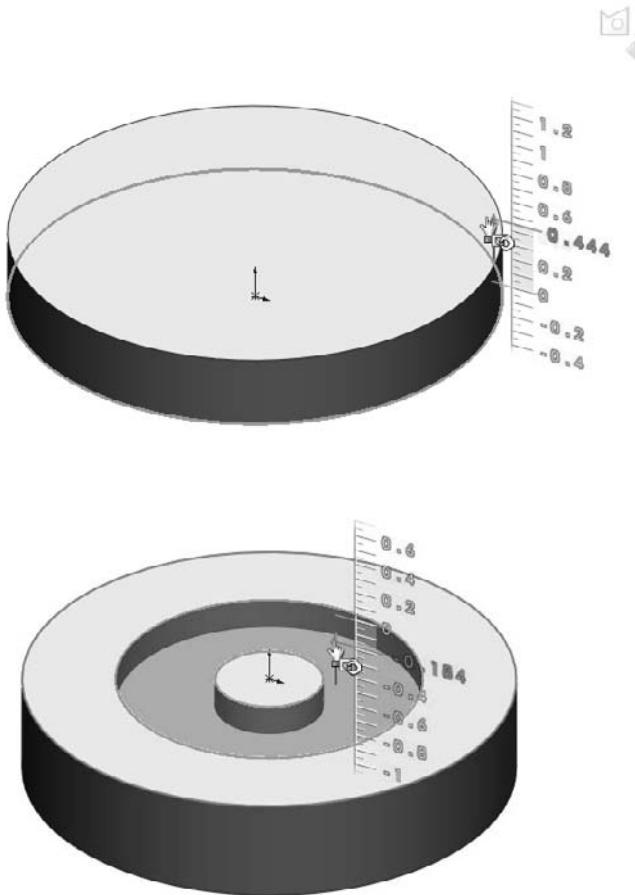
While Instant 3D can only create extruded bosses and cuts, it can edit revolves. If you create a revolved feature revolving the sketch, say, 270°, the face created at the angle can be edited by Instant 3D dragging. You can also drag faces created by any underdefined sketch elements.

FIGURE 7.6

Identifying Instant 3D interface elements

Editing geometry with Instant 3D

Instant 3D enables you to edit 2D sketches and solid geometry. You can also edit some additional feature types using Instant 3D, such as offset reference planes. It can neither create nor edit surface geometry or 3D sketches in some situations. To edit solid geometry, click on a face, and an arrow appears. Drag the arrow, and SolidWorks automatically changes either the sketch or the feature end condition used to create that face. If a dimensioned sketch was used to create that face, SolidWorks will not allow you to use the Instant 3D arrow to move or resize the face. An option exists that enables Instant 3D changes to override sketch dimensions at Tools ➪ Sketch Settings ➪ Override Dims on Drag.

Caution

Be careful with the Override Dims on Drag option. If you accidentally drag a fully defined sketch, this setting enables SolidWorks to completely resize the sketch. For working conceptually, it can be a great aid, but for final production models, you may do better to turn this off.■

Instant 3D offers different editing options depending on how a sketch is selected.
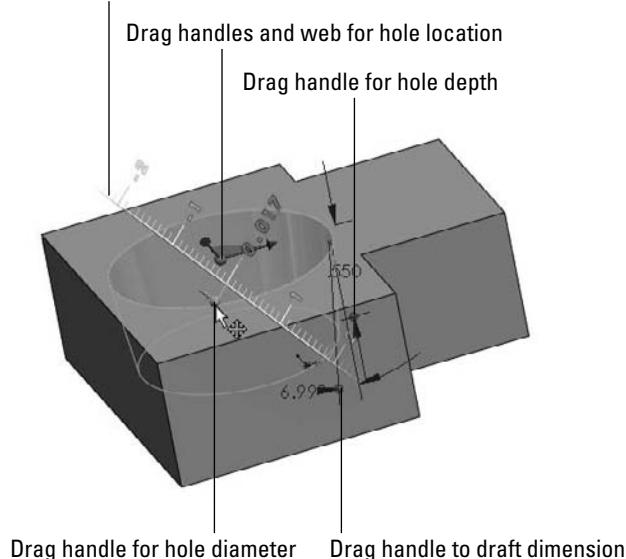
- **A sketch is selected from the graphics window.** The pull arrow appears, enabling you to create an extruded boss or cut.

- **A sketch is selected from the FeatureManager.** If the sketch has relations to anything outside of the sketch, the sketch is highlighted with no special functionality available. If no external relations exist, a box with stretch handles enable scaling the sketch, and a set of axes with a wing enables you to move the sketch in X or Y or X and Y. Figure 7.7 shows this situation.

When Instant 3D is activated, double-clicking a sketch in either the FeatureManager or on a sketch element in the graphics window opens that sketch. While you are in a sketch, if you double-click with the Select cursor in blank space in the graphics window, you close the sketch. This only works for 2D sketches; 3D sketches can be opened, but not closed, this way.

FIGURE 7.7

Sketch scaling and moving options with Instant 3D

Ruler for hole diameter

Drag handles and web for hole location

Drag handle for hole depth

Drag handle for hole diameter   Drag handle to draft dimension

Working with the Revolve feature

Like all other features, revolve features have some rules that you must observe when choosing sketches to create a revolve:

- **Draw only half of the revolve profile.** (Draw the section to one side of the centerline.)
- **The profile must not cross the centerline.**
- **The profile must not touch the centerline at a single point.** It can touch along a line, but not at a point. Revolving a sketch that touched the centerline at a single point would create a point of zero thickness in the part.

You can use any type of line or model edge for the centerline, not just the centerline/construction line type.

End conditions

There are three Revolve end conditions:

- **One-Direction.** The revolve angle is driven in a single direction.
- **Two-Direction.** The revolve angle can be driven in two independent directions.
- **Mid-Plane.** The revolve angle is divided equally in opposite directions.

There is no equivalent for Up to Vertex, Up to Next, Up to Surface, or Up to Body with the Revolve feature.

Contour selection

Like extrude features, revolve features can also use contour selection; and as with the extrude features, I recommend that you avoid using contours for production work.

Introducing Loft

A loft is what is known as an *interpolated* feature. That means that you can create profiles for the feature at certain points, and the software will interpolate the shape between the profiles. You can use additional controls with loft, such as guide curves or centerlines, and establish end conditions to help direct the shape. A loft with just two profiles is a straight line transition. If you have more than two profiles, the transition from one profile to another works more like a spline.

Many users struggle when faced with the option to create a loft or a sweep. Some overlap exists between the two features, but as you gain some experience, it becomes easier to choose between them. Generally, if you can create the cross-section of the feature by manipulating the dimensions of a single sketch, a sweep might be the best feature. If the cross-section changes character or severely changes shape, a loft may be best. If you need a very definite shape at both ends and/or in
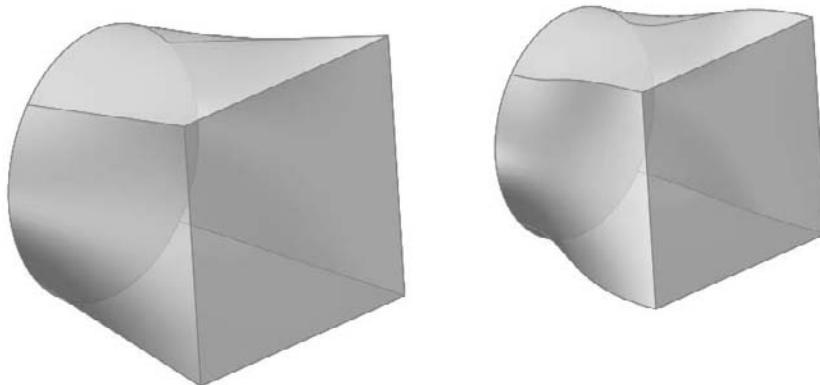
the middle, a loft is a better choice because it enables you to explicitly define the cross-section at any point. However, if the outline is more important than the cross-section, you should choose a sweep. In addition, if the path between ends is important, choose a sweep.

Both types of features are extremely powerful, but the sweep has a tendency to be fussier about details, setup, and rules, while the loft can be surprisingly flexible. I am not trying to dissuade you from using sweeps, because they are useful in many situations. However, in my own personal modeling, I probably use about ten lofts for every sweep. For example, while you would use a loft or combination of loft features to create the outer faces of a complex laundry detergent bottle, you would use the sweep to create a raised border around the label area or the cap thread.

A good example of the interpolated nature of a loft is to put a circle on one plane and a rectangle on an offset plane and then loft them together. This arrangement is shown in Figure 7.8. The transition between shapes is the defining characteristic of a loft, and is the reason for choosing a loft instead of another feature type. Lofts can create both Boss features and Cut features.

FIGURE 7.8

A simple loft
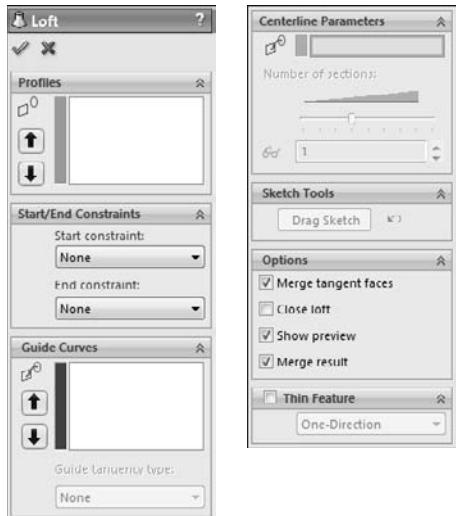
Both shapes are two-profile lofts. The two-profile loft with default end conditions always creates a straight transition, which is shown in the image to the left. A two-point spline with no end tangency creates a straight line in exactly the same way. By applying end conditions to either or both of the loft profiles, the loft's shape is made more interesting, as seen in the image to the right in Figure 7.8. Again, the same thing happens when applying end tangency conditions to a two-point spline: it goes from being a straight line to being more curvaceous, with continuously variable curvature. The Loft PropertyManager interface is shown in Figure 7.9.

FIGURE 7.9

The Loft PropertyManager

Entities that you can use in a loft

For solid lofts, you can select faces, closed loop 2D or 3D sketches, and surface bodies. You can use sketch points as a profile on the end of a loft that comes to a point or rounded end. For surface lofts, you can use open sketches and edges in addition to the entities that are used by solid lofts.

Some special functionality becomes available to you if you put all the profiles and guide curves together in a single 3D sketch. In order to select profiles made in this way, you must use the SelectionManager, which is discussed later in this chapter.

The Sketch Tools panel of the Loft PropertyManager enables you to drag sketch entities of any profile made in this way while you are editing or creating the Loft feature, without needing to exit and edit a sketch.

Note

I discuss 3D sketches in more detail in Chapter 31.■
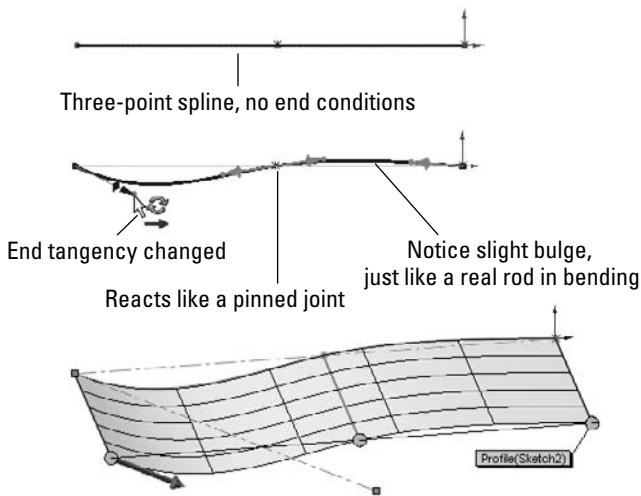
The similarities between lofts and splines

The words *loft* and *spline* come from the shipbuilding trade. The word *spline* is actually defined as the slats of wood that cover the ship, and the spars of the hull very much resemble loft sections. With the splines or slats bending at each spar, it is easy to see how the modern CAD analogy came to be.

Lofts and splines are also governed by similar mathematics. You have seen how the two-point spline and two-profile loft both create a straight-line transition. Next, a third profile is added to the loft and a third point to the spline, which demonstrates how the math that governs splines and lofts is also related to bending in elastic materials. Figure 7.10 shows how lofts and splines react geometrically in the same way that bending a flexible steel rod would react (except that the spline and the loft do not have a fixed length).

FIGURE 7.10

Splines, lofts, and bending

With this bit of background, it is time to move forward and talk about a few of the major aspects of loft features in SolidWorks. It is possible to write a separate book that only discusses modeling lofts and other complex shapes. This has in fact been done. The *SolidWorks Surfacing and Complex Shape Modeling Bible* (Wiley, 2008) covers a wide range of surfacing topics with examples in far detail. In this single chapter, I do not have the space to cover the topic exhaustively, but coverage of the major concepts will be enough to point you in the right direction.

The need for surfaces

In this chapter, I deal exclusively with solid modeling techniques because they are the baseline that SolidWorks users use most frequently. Surfaces make it easier to discuss complex shape concepts because surfaces are generally created one face at a time, rather than by using the solid modeling method that creates as many faces as necessary to enclose a volume.

From the very beginning, the SolidWorks modeling culture has made things easier for users by taking care of many of the details in the background. This is because solids are built through

automated surface techniques. Surface modeling in itself can be tedious work because of all the manual detail that you must add. Solid modeling as we know it is simply an evolutionary step that adds automation to surface modeling. The automation maintains a closed boundary of surfaces around the solid volume.

Because surfaces are the underlying building blocks from which solids are made, it would make sense to teach surfaces first, and then solids. However, the majority of SolidWorks users never use surfacing, and do not see a need for it; therefore, surface functions are generally given a lower priority.

Cross-Reference
Refer to Chapter 27 for surfacing information. For a comprehensive look at surfacing and complex shapes, see the SolidWorks Surfacing and Complex Shape Modeling Bible (Wiley, 2008).■

Loft end constraints
Loft end conditions control the tangency direction and weighting at the ends of the loft. Some of the end constraints depend upon the loft starting or ending from other geometry. The optional constraints are covered in the following sections.

None
The direction of the loft is not set by the None end constraint, but the curvature of the lofted faces at the ends is zero. This is the default end constraint for two-section lofts.

Default
The Default end constraint is not available for two-section lofts, only for lofts with three or more sections. This end constraint applies curvature to the end of the loft so that it approximates a parabola being formed through the first and last loft profiles.

Tangent to Face
The Tangent to Face end constraint is self-explanatory. The Tangency to Face option includes a setting for tangent length. This is not a literal length dimension, but a relative weighting, on a scale from 0.1 to 10. The small arrow to the left of the setting identifies the direction of the tangency. Usually, the default setting is correct, but there are times when SolidWorks misidentifies the intended tangency direction, and you may need to correct it manually.

The Next Face option is available only when lofting from an end face where the tangency could go in one of two perpendicular directions. This is shown in Figure 7.11.

Apply to All refers to applying the Tangent Length value to all the tangency-weighting arrows for the selected profile. When you select Apply to All, only one arrow displays. When you deselect it, one arrow should display for each vertex in the profile, and you can adjust each arrow individually.

Curvature to Face
The difference between tangency and curvature is that tangency is only concerned with the direction of curvature immediately at the edge between the two surfaces. Curvature must be tangent
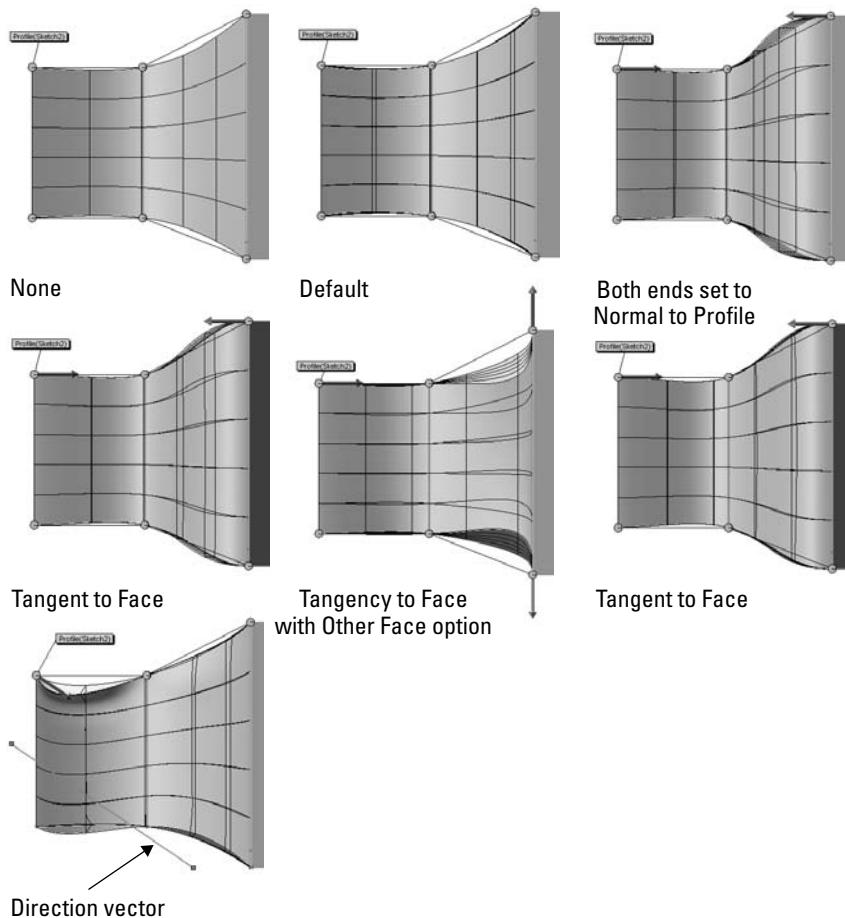
and match the radius of curvature on either side of the edge between surfaces. This is often given many names, including curvature continuity and c2 or g2. Lofted surfaces do not usually have a constant radius; because they are like splines, they are constantly changing in local radius.

Direction Vector

The Direction Vector end constraint forces the loft to be tangent to a direction that you define by selecting an axis, edge, or sketch entity. The angle setting makes the loft deviate from the direction vector, as shown in Figure 7.11. The curved arrows to the left identify the direction in which the angle deviation is going.

FIGURE 7.11

Examples of end constraints

None

Default

Both ends set to
Normal to Profile

Tangent to Face

Tangency to Face
with Other Face option

Tangent to Face

Direction vector

Isoparameter U-V lines

The mesh or grid shown in the previous images appears automatically for certain types of features, including lofts. The grid represents *isoparameter* lines, also known as NURBS mesh or U-V lines. This mesh shows the underlying structure of the faces being created by the feature. If the mesh is highly distorted and appears to overlap in places, then it is likely that the feature will fail.

You can show or hide the mesh through the right mouse button (RMB) menu when editing or creating a Loft feature, unless the SelectionManager is active. In this case, you can see only SelectionManager commands in the RMB menu. In addition, planar faces do not mesh, only faces with some curvature.

Guide curves

Guide curves help to constrain the outline of a loft between loft profiles. Although it is best to try to achieve the shape you want by using appropriately shaped and placed loft profiles, this is not always possible. The most appropriate use of guide curves for solid lofts is at places where the loft is going to create a hard edge, which is usually at the corners of loft profile sketches. Guide curves often (but not always) break up what would otherwise be a smooth surface, and you should avoid them in these situations, if possible.

Best Practice

Do not try to push the shape of the loft too extremely with guide curves. Use guide curves mainly for tweaking and fine-tuning rather than coarse adjustments. Use loft sections and end constraints to get most of the overall shape correct. Pushing too hard with a guide curve can cause the shape to kink unnaturally.■
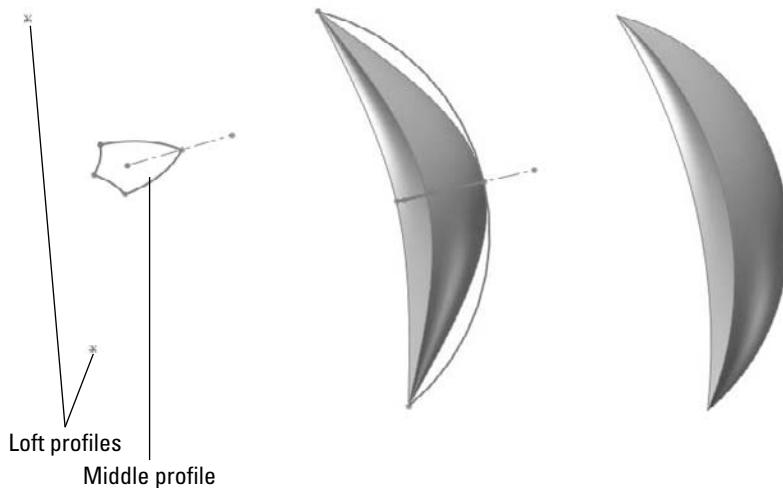
Although guide curves can be longer than the loft, they can not be shorter. The guide curve applies to the entire loft. If you need to apply the guide curve only to a portion of the loft, then split the loft into two lofts: one that uses the guide curve and one that does not. The guide curve must intersect all profiles in a loft.

If you have more than one guide curve, the order in which they are listed in the box is important. The first guide curve helps to position the intermediate profiles of the loft. It may be difficult to visualize the effects of guide-curve order before it happens, but remember that it does make a difference, and depending on the difference between the curves, the difference may or may not be subtle.

Guide curves are also used in sweeps, which I address later in this chapter. Figure 7.12 shows a model that is lofted using guide curves. The image to the left shows the sketches that are used to make the part. There are two sketches with points; you can use points as loft profiles. The image in the middle shows the Loft feature without guide curves, and the one to the right is the part with guide curves. If you would like to examine how this part is built, you can find it on the CD-ROM with the filename `Chapter 7 Guide Curves.sldprt`.

FIGURE 7.12

A loft with and without guide curves

Loft profiles
Middle profile

Centerline lofts

The Centerline panel of the Loft PropertyManager is used to set up a centerline loft. You can use the centerline of a loft in roughly the same way that you use a sweep path. In fact, the Centerline loft resembles a sweep feature where you can specify the shape of some of the intermediate profiles. Centerline lofts can also create intermediate profiles. You may prefer to use a centerline loft instead of either a sweep or a regular loft because the profile may change in ways that the Sweep feature cannot handle, and the loft may need some guidance regarding the order of the profiles, or how to smooth the shape between the profiles.

I cover sweep features in this chapter. If you are creating a centerline loft, you may want to examine the sweep functionality as well.

You can use centerlines simultaneously with guide curves. While guide curves must touch the profile, there is no such requirement for a centerline; in fact, the centerline works best if it does not touch any of the profiles.

The slider in the Centerline Parameters panel enables you to specify how many intermediate sections to create between sketched profiles.

SelectionManager

The SelectionManager simplifies the selection of entities from complex sketches that are not necessarily the clean, closed loop sketches that SolidWorks works with most effectively.

The SelectionManager has been implemented in a limited number of features. Selection options in the SelectionManager include the following:

- **OK.** Accepts the selection. This feature is also available on the RMB menu.

- **Cancel.** Quits the SelectionManager.

- **Clear All.** Clears the current selection set.

- **Push Pin.** Keeps the SmartSelection window available, even when it is not required for sketch entity selections.

- **Select Closed Loop.** You can select two different types of loops with this tool:

    - A parametric closed loop in a 2D or 3D sketch.

    - A parametric loop of edges around a surface.

- **Select Open Loop.** Selects a chain (end-to-end sketch entities).

- **Select Group.** Selects entities individually. If you click the Propagate symbol, all tangent edges are selected.

- **Select Region.** Works like the Contour Selection described earlier in this chapter.

- **Standard Selection.** Disables special functions of the SelectionManager. This feature works like a regular selection tool.
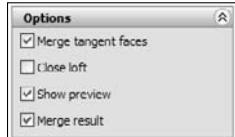
- **Auto OK Selections.** Becomes enabled when you use the Push Pin. Auto OK Selections works for closed and open loop selection.

Loft options

You can choose from the following Loft options, as shown in Figure 7.13:

Loft options

- **Merge tangent faces.** Merges model faces that are tangent into a single face. This is done behind the scenes by converting profiles into splines, which make approximations but are smoother than sketches with individual tangent line and arc entities.

- **Close loft.** The loft is made into a closed loop. At least three loft profiles must exist in order to use this option. Figure 7.14 shows a loft where the Close Loft option is used, and the loft sections are shown. This model is on the CD-ROM with the filename `Chapter 7 — Closed Loft.sldprt`.

FIGURE 7.14

A closed loft

- **Show preview.** Selecting and deselecting this option turns the preview of the Loft feature on or off, respectively, if the feature is going to work. All the following loft preview options are system options and remain selected until you deselect them.
  - **Transparent/Opaque Preview** is available from the RMB menu when you edit a loft, if the SelectionManager is not active.
  - **Mesh Preview** is also available on the same RMB menu.
  - **Zebra Stripe Preview** is also available on the same RMB menu, and is covered in more depth in Chapter 11.
- **Merge result.** Merges the resulting solid body with any other solid bodies that it may contact.

Controlling Sweep features

The Sweep feature uses multiple sketches. A sweep is made from a profile (cross-section) and a path, and can create a boss or a cut feature. If you want, you can also use guide curves. Sweeps can run the gamut from simple to complex. Typical simple sweeps are used to create wire, tubing, or hose. Sweeps that are more complex are used for creating objects such as bottles, involutes, springs, and corkscrews.
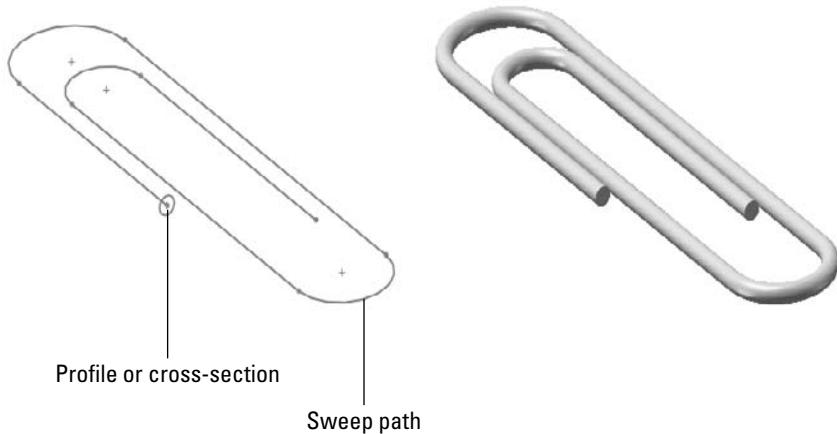
The main criteria for selecting a sweep to create a feature are that you must be able to identify a cross-section and a path. The profile (cross-section) can change along the path, but the overall shape must remain the same. The profile is typically perpendicular to the path, although this is not a requirement.

Simple sweep

An example of a simple sweep is shown in Figure 7.15. The paper clip uses a circle as the profile and the coiled lines and arcs as the path.

The sweep profile follows the path.

Profile or cross-section

Sweep path

Sweep with guide curves

Sweeps that are more complex begin to control the size, orientation, and position of the cross-section as it travels through the sweep. When you use a guide curve, several analogies can be used to visualize how the sweep works. The cross-section/profile is solved at several intermediate positions along the path. If the guide curve does not follow the path, the difference between the two is made up by adjusting the profile. Consider the following example. In this case, the profile is an ellipse, the path is a straight line, and there are guide curves that give the feature its outer shape. Figure 7.16 shows all these elements and the finished feature.

On the CD-ROM
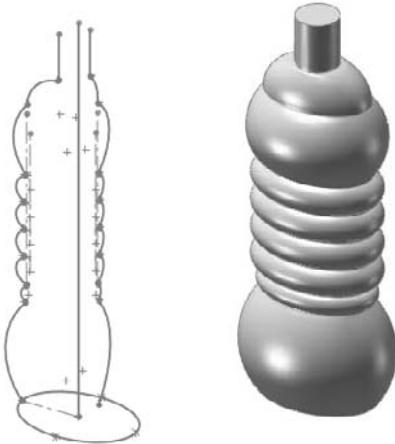
The part shown in Figure 7.16 is on the CD-ROM with the filename `Chapter7 Bottle.sldprt.` ■

The PropertyManager for the Sweep function includes an option for Show Sections, which in this case creates almost 200 intermediate cross-sections. These sections are used behind the scenes to create a loft. You can think of complex sweeps with guide curves or centerlines as an automated

setup for an even more complex loft. It is helpful to envision features such as this when you are troubleshooting or setting up sweeps that are more complex. If you open the part mentioned previously from the CD-ROM, you can edit the Sweep feature to examine the sections for yourself.

FIGURE 7.16

The sweep profile follows the path and is controlled parametrically by guide curves.

In most other published SolidWorks materials that cover these topics, sweeps are covered before lofts because many people consider lofts the more advanced topic. However, I have put lofts first because understanding them is necessary before you can understand complex sweeps, as complex sweeps really are just lofts.

Pierce relation

The Pierce sketch relation is the only sketch relation that applies to a 3D out-of-plane edge or curve without projecting the edge or curve into the sketch plane. It acts as if the 3D curve is a length of thread and the sketch point is the eye of a needle where the thread pierces the needle eye. The Pierce relation is most important in the Sweep feature when it is applied in the profile sketch between endpoints, centerpoints, or sketch points and the out-of-plane guide curves. This is because the Pierce relation determines how the profile sketch will be solved when it is moved down the sweep path to create intermediate profiles.

Figure 7.17 illustrates the function of the Pierce relation in a sweep with guide curves. The dark section on the left is the sweep section that is sketched. The lighter sketches to the right represent the intermediate profiles that are automatically created behind the scenes, and are used internally to create the loft.

FIGURE 7.17

The effects of the Pierce relation

Guide curve   Pierce relation forces contact

Sketched sweep profile
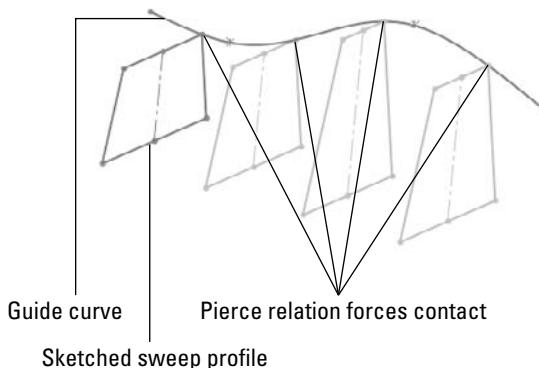
Figure 7.17 shows what is happening behind the scenes in a sweep feature. The sweep re-creates the original profile at various points along the path. The guide curve in this case forces the profile to rebuild with a different shape. Pierce constraints are not required in simple sweeps, but when you start using guide curves, you should also use a pierce.

Tip

If you feel that you need more profile control, but still want to create a sweep-like feature, try a centerline loft. The centerline acts like a sweep path that doesn't touch the profiles, but unlike a sweep, you can use multiple profiles with it.■

Figure 7.18 shows a more complicated 3D sweep, where both the path and the guide curve are 3D curves. I cover 3D curves toward the end of this chapter; you can refer to these sections to understand how this part is made.

On the CD-ROM
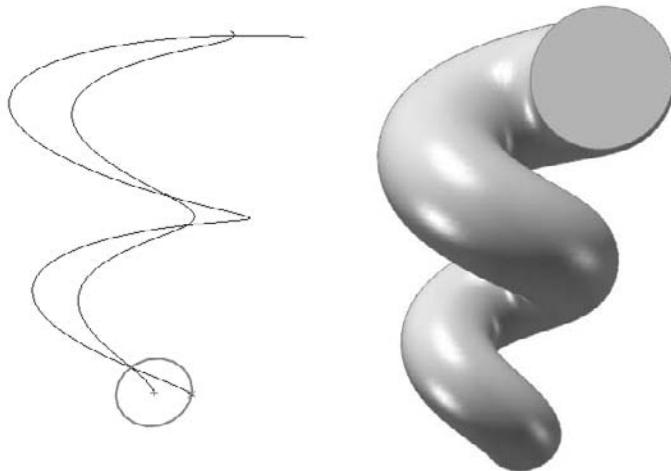
The part shown in Figure 7.18 is on the CD-ROM with the filename `Chapter 7 3D Sweep.sldprt`.■

This part is created by making a pair of tapered helices, with the profile sketch plane perpendicular to the end of one of the curves. The taper on the outer helix is greater than on the inner one, which causes the twist to become larger in diameter as it goes up.

To make the circle follow both helices, you must create two pierce relations, one between the center of the circle and a helix, and the other between a sketch point that is placed on the circumference of the circle and the other helix. This means that the difference in taper angles between the two helices is what drives the change in diameter of the sweep.
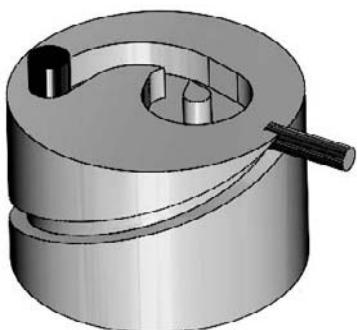
A 3D sweep

Cut Sweep with a solid profile

The Cut Sweep feature has an option to use a solid sweep profile. This kind of functionality has many uses, but is primarily intended for simulating complex cuts made by a mill or lathe. Figure 7.19 shows a couple of examples of cuts you can make with this feature. The part used for this screen shot is also on the CD-ROM.

Cuts you can make with the Cut Sweep feature using a solid profile
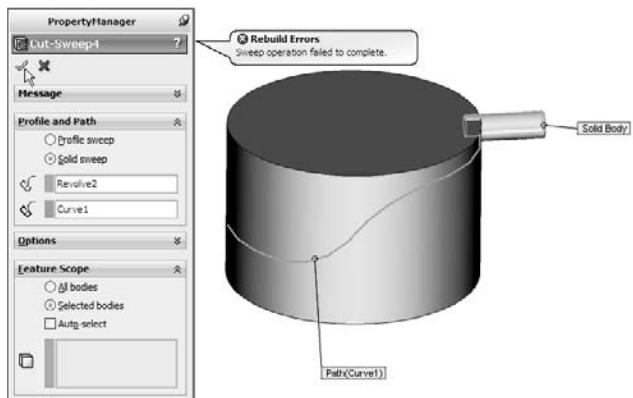
The solid profile cut sweep has a few limitations that I need to mention:

- It uses a separate solid body as the cutting tool, so you have to model multi-bodies.
- The path must start at a point where it intersects the solid cutting tool body (path starts inside or on the surface of the cutting tool).
- The cutting tool must be definable with a revolved feature.
- The cutting tool must be made of simple analytical faces (sphere, torus, cylinder, and cone; no splines).
- You cannot use a guide curve with a solid profile cut (cannot control alignment).
- The cut can intersect itself, but the path cannot cross itself.

You can create many useful shapes with the solid profile cut sweep, but because of some of the limitations I've listed, some shapes are more difficult to create than others. For these shapes, you might choose to use regular cut sweep features. Figure 7.20 shows an example of a cam-like feature that you may want to create with this method, but may not be able to adequately control the cutting body.

FIGURE 7.20

Controlling a cam cut may be a challenge

Creating Curve Features

Curves in SolidWorks are often used to help define sweeps and lofts, as well as other features. Curves differ from sketches in that curves are defined using sketches or a dialog box, and you cannot manipulate them directly or dimension them in the same way that you can sketches. Functions that you are accustomed to using with sketches often do not work on curves.

Several features that carry the curve name are actually sketch-based features:

- Equation driven curve
- Intersection curve
- Face curves

Tip

When you come across a function that does not work using a curve entity, but that works on a sketch (for example, making a tangent spline), it may help to use the Convert Entities function. Converting a helix into a 3D sketch creates a spline that lies directly on top of the helix and enables you to make another spline that is tangent to the new spline.■

You can define the following types of curves in SolidWorks:

- Helix/tapered helix/variable helix/spiral
- Projected curve
- Curve through XYZ points
- Curve through reference points
- Composite curve

You can find all the curve functions on the Curves toolbar or by choosing Insert ➪ Curve from the menu.

Curve features in general have several limitations, some of which are serious. You have to be prepared with workaround techniques when using them. When curves are used in features, you often cannot reselect the curve to re-apply sketch broken sketch relations. (The work around for this is to select the curve from the FeatureManager, or if that doesn't work, you have to delete the feature and re-create it). In addition, curves cannot be mirrored, moved, patterned, or manipulated in any way. (A workaround for this may be to use Convert Entities to create a sketch from the curve.)

Working with helix features
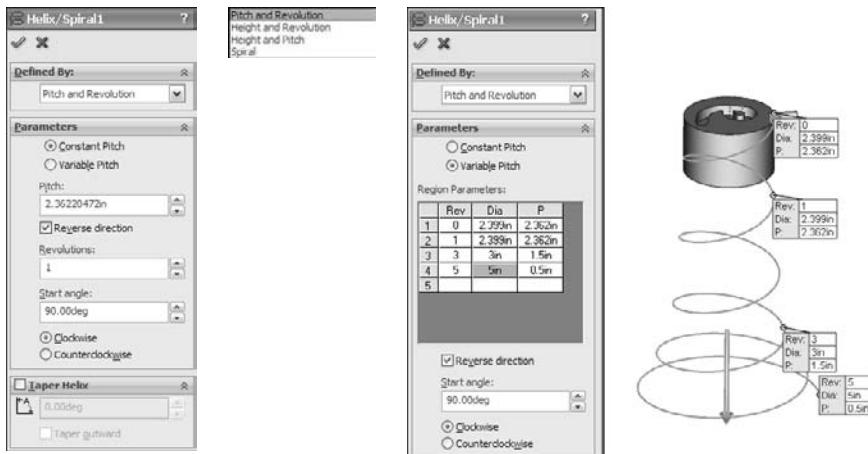
The helix curve types are all based on a circle in a sketch. The circle represents the starting location and diameter of the helix. Figure 7.21 shows the PropertyManagers of the Constant Pitch and Variable Pitch helix types.

You can create all the helical curve types by specifying any combination of total height, pitch, and the number of revolutions. The start angle is best thought of as a relative number. It is difficult to predict where zero degrees starts, and this depends on the relation of the sketch plane to the Origin. The start angle cannot be controlled outside of the PropertyManager, and cannot be driven by sketch geometry. The term *pitch* refers to the straight-line distance along the axis between the rings of the helix. Pitch for the spiral is different and is described later.

FIGURE 7.21

The Helix PropertyManager
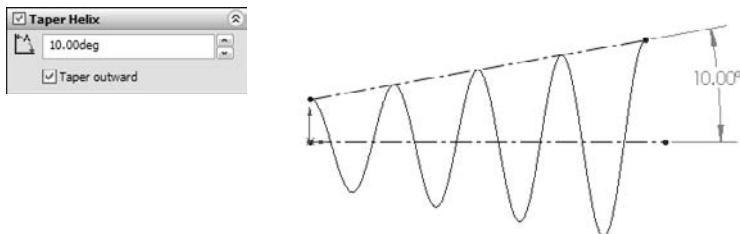
Tapered helix

The Tapered Helix panel in the Helix PropertyManager enables you to specify a taper angle for the helix. The taper angle does not affect the pitch. If you need to affect both the taper and the pitch, then you can use a variable pitch helix. Figure 7.22 shows how the taper angle relates to the resulting geometry.

FIGURE 7.22

The Tapered Helix panel
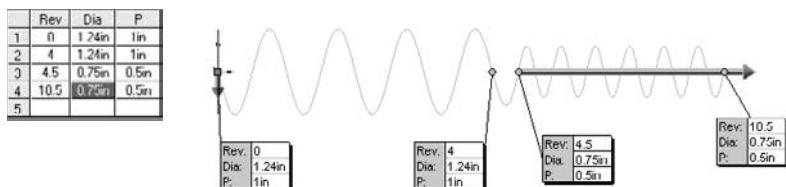
Variable pitch helix

You can specify the variable pitch helix either in the chart or in the callouts that are shown in Figure 7.23. Both the pitch and the diameter are variable. The diameter number in the first row cannot be changed, but is driven by the sketch. In the chart shown, the transition between 4 and 4.5 revolutions is where the pitch and diameter both change.

The variable pitch helix

Spiral

A spiral is a flattened (planar) tapered helix. The pitch value on a spiral is the radial distance between revolutions of the curve.

Creating projected curves

Many users have difficulty envisioning the concept of the projected curve. The two options available for projected curves are

- Sketch On Faces
- Sketch On Sketch

These names can be misleading if you do not already know what they mean. In both cases, the word *sketch* is used as a noun, not a verb, so you are not actively sketching on a surface; instead, you are creating a curve by projecting a sketch onto a face.

Sketch On Faces
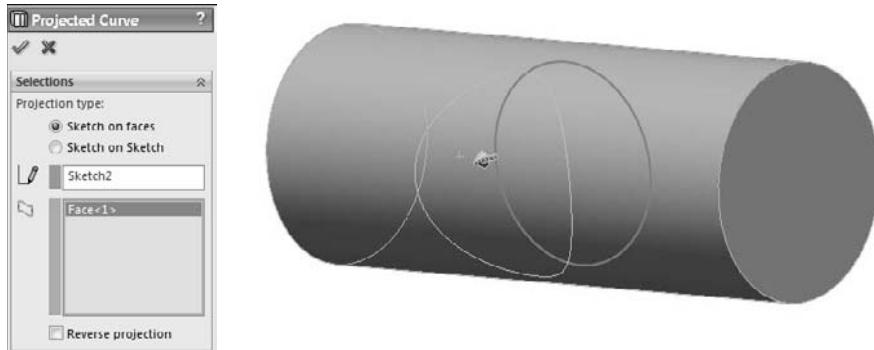
The Sketch On Faces option is the easiest to explain, so I will describe this one first. With this option set, the projected curve is created by projecting a 2D sketch onto a face. The sketch is projected normal (perpendicular) to the sketch plane. This is like extruding the sketch and using the Up To Surface end condition. The sketch can be an open or closed loop, but it may not be multiple open or closed loops, nor can it be self-intersecting. Figure 7.24 shows an example of projecting a sketch onto a face to create a projected curve.

FIGURE 7.24

A projected curve using the Sketch On Faces option

Sketch On Sketch

This is the concept that most frequently causes difficulty for users. The Sketch Onto Sketch Projected Curve option can be visualized in a few different ways.

Reverse 2D drawing visualization method

One way to visualize Sketch Onto Sketch projection is to think of it as being the reverse of a 2D drawing. In a 2D drawing, 3D edges (you can think of the edges as curves) are projected onto orthogonal planes to represent the edge from the Front or Top planes. The Sketch Onto Sketch projection takes the two orthogonal views, placed on perpendicular planes, and projects them back to make the 3D edge or curve. This is part of the attraction of the projected curve, because making 3D curves accurately is difficult if you do it directly by using a tool such as a 3D sketch spline; however, if you know what the curve looks like from two different directions, then it becomes easy. Figure 7.25 illustrates this visualization method.
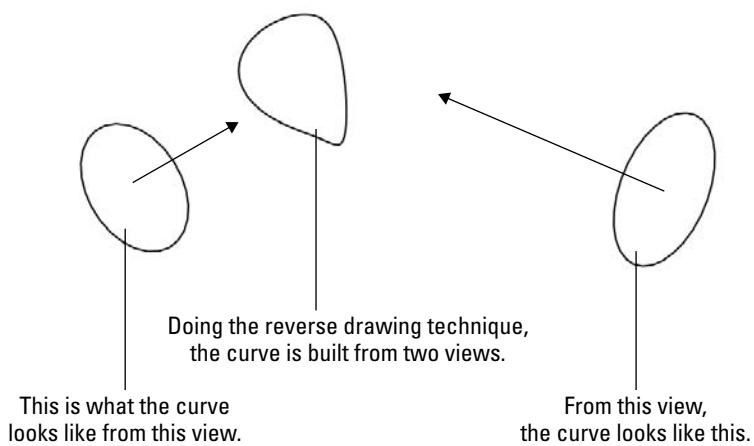
When you think of describing a complex 3D curve in space, one of the first methods that usually comes to mind is describing it as two 2D curves from perpendicular directions, exactly in the same way as you would if you created projected drawing views from it. From this, it makes sense to see the creation of the curve as the reverse process, drawing the 2D views first, from which you can then create the 3D curve.

Intersecting surfaces visualization method

A second method used for visualizing Sketch On Sketch projected curves is the intersecting surfaces method. In this method, you can see the curve being created at the intersection of two surfaces that are created by extruding each of the sketches. This method is shown in Figure 7.26.
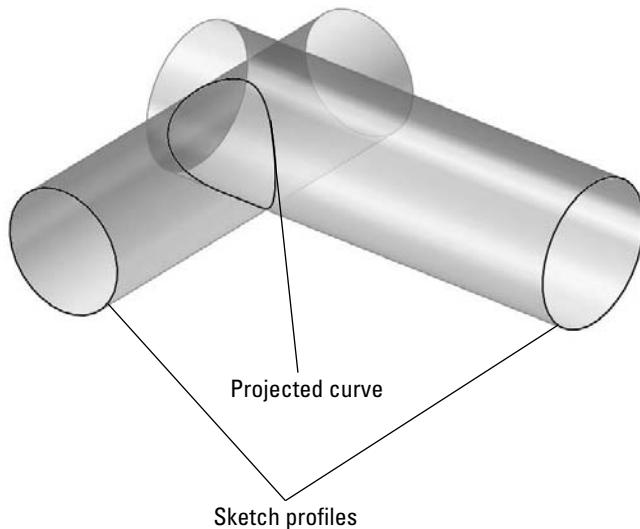
FIGURE 7.25

The reverse 2D drawing visualization method for projected curves

Doing the reverse drawing technique,
the curve is built from two views.

This is what the curve
looks like from this view.

From this view,
the curve looks like this.

FIGURE 7.26

Using intersecting surfaces to visualize a Sketch on Sketch projected curve

Projected curve

Sketch profiles

Making Curve Through XYZ Points
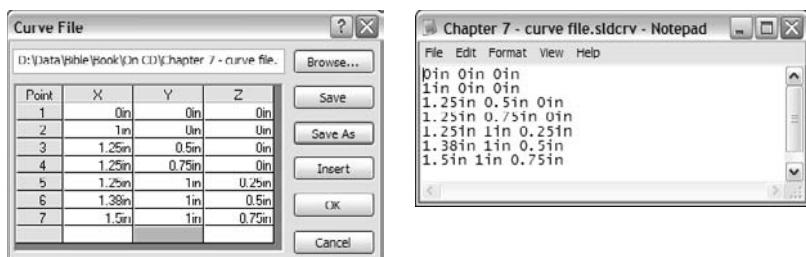
The Curve Through XYZ Points feature enables you to either type in or import a text file with coordinates for points on a curve. The text file can be generated by any program that makes lists of numbers, including Excel. The curve reacts like a default spline, so the teeter-tottering effect may be noticeable, especially because you cannot set end conditions or tangency. To avoid this effect, it may be a good idea to overbuild the curve by a few points on each end, or have a higher density of points.

If you import a text file, the file can have an extension of either `*.txt` or `*.sldcrv`. The data that it contains must be formatted as three columns of X-, Y-, and Z-coordinates using the document units (inch, mm, and so on), and the coordinates must be separated by comma, space, or tab. Figure 7.27 shows both the Curve File dialog box displaying a table of the curve through X, Y, and Z points, and the `*.sldcrv` Notepad file. You can read the file from the Curve File dialog box by clicking the Browse button, but if you manually type the points, then you can also save the data out directly from the dialog box. Just like any type of sketch, this type of curve cannot intersect itself.

FIGURE 7.27

The Curve File dialog box showing a table of the curve through X, Y, and Z points, and a Notepad text file with the same information

Building Curve Through Reference Points

The Curve Through Reference Points feature creates a curve entity from selected sketch points or vertices. The curve can be an open or closed loop, but a closed loop requires that you select at least three points. You cannot set end conditions of the curve, and so this feature works like a default spline in the same way as the XYZ curve.

The most common application of this feature is to create a wire from selected points along a wire path. Another common application is a simple two-point curve to close the opening of a surface feature such as Fill, Boundary, or Loft.

Putting together a Composite curve

The composite curve joins multiple curves, edges, or sketches into a single curve entity. The spring shown in Figure 7.28 was created by using a composite curve to join a 3D sketch, variable pitch helix, and a projected curve. You can also use model edges with the composite curve. The curve is shown on half of the part; the rest of the part is mirrored. Curves cannot be mirrored.

Composite curves overlap in functionality with the Selection Manager to some extent. In some ways, the Composite Curve is nicer because you can save a selection in case the creation of the feature that uses the Selection Manager fails. (If you can't create the feature, you can't save the selection.) On the other hand, Composite Curves don't function the same way that a selection of model edges do for settings like tangency and curvature.

FIGURE 7.28

A part created from a composite curve

Using Split lines

Split lines are not exactly curves; they are just edges that split faces into multiple faces. Split lines are used for several purposes, but are primarily intended to split faces so that draft can be added. They are also used for creating a broken-out face for a color break or to create an edge for a hold line fillet, which I discuss later in this chapter.

There are some limitations to using split lines. First, they must split a face into at least two fully enclosed areas. You cannot have a split line with an open loop sketch where the ends of the loop are on the face that is to be split; they must either hang off the face to be split or be coincident with the edges. If you think you need a split line from an open loop, try using a projected curve instead.

The SolidWorks 2010 version removes some other long-standing limitations, such as splitting on multiple bodies, using multiple closed loops, and using nested loops. These much needed improvements will help users avoid workarounds.

Caution

A word of caution is needed when using split lines, especially if you plan to add or remove split lines from an existing model. The split lines should go as far down the tree as possible. Split lines change the face IDs of the faces that they split, and often the edges as well. If you roll back and apply a split line before existing features, you may have a significant amount of cleanup to do. Similarly, if you remove a split line that already has several dependent features, many other features may also be deleted or simply lose their references. ■

Equation Driven Curve

The Equation Driven Curve is not really a curve feature; it is a sketch entity. It specifies a spline inside a 2D sketch with an actual equation. Even though this is a spline based sketch entity, it can only be controlled through the equation, and not by using spline controls. This feature is covered in more detail in Chapter 3, with other sketch entities.

Understanding Fillet Types

SolidWorks offers very powerful filleting functions. The Fillet feature comprises various types of fillets and blends. Simple fillets on straight and round edges are handled differently from variable-radius fillets, which are handled differently from the single or double hold line fillet or setback fillets. Once you click the OK button to create a fillet as a certain type, you cannot switch it to another type. You can switch types only before you create an established fillet feature.

Many filleting options are available, but most of them are relatively little used or even known. In fact, most users confine themselves to the Constant Radius or Variable Radius fillets. The following section describes all the available fillet types and options:
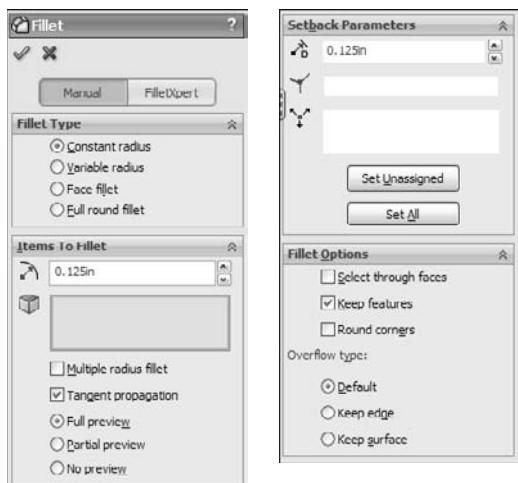
- Constant Radius Fillet
  - Multiple Radius Fillet
  - Round Corners
  - Keep Edge/Keep Surface
  - Keep Feature
- Variable Radius Fillet
- Face Fillet
  - Curvature Continuous Fillet
  - Face Fillet with Help Point
  - Single Hold Line Fillet
  - Double Hold Line Fillet
  - Constant Width Fillet

- Full Round Fillet
- Setback Fillet
- Setback Fillet with Variable Radius

Figure 7.29 shows the Fillet PropertyManager. Other options affect preview and selection of items, and these options are discussed in this section.

FIGURE 7.29

The Fillet PropertyManager

Creating a constant radius fillet

Constant radius fillets are the most common type that are created if you select only edges, features, or faces without changing any settings. When applying fillets in large numbers, you should consider several best-practice guidelines and other recommendations that come later in this chapter.

There are still some long-time users who distinguish between fillets and rounds (where fillets add material and rounds remove it). SolidWorks does not distinguish between the two, and even two edges that are selected for use with the same fillet feature can have opposite functions; for example, both adding and removing material in a single feature.
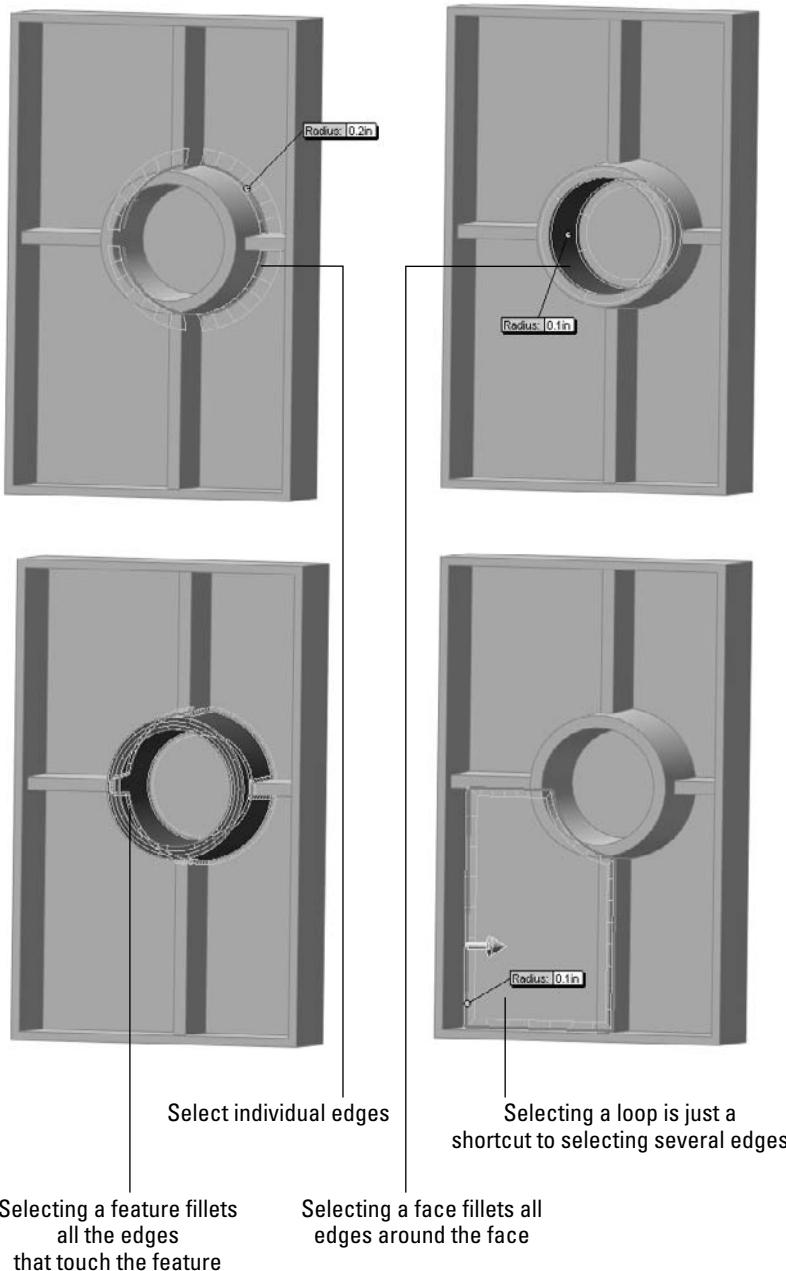
Selecting entities to fillet

You can create fillets from several selections, including edges, faces, features, and loops. Edges offer the most direct method, and are the easiest to control. Figure 7.30 shows how you can use each of these selections to create fillets on parts more intelligently.

FIGURE 7.30

Selection options for fillets

Select individual edges

Selecting a loop is just a
shortcut to selecting several edges

Selecting a feature fillets
all the edges
that touch the feature

Selecting a face fillets all
edges around the face

Tip

To select features for filleting, you must select them from the FeatureManager. The Selection Filter only filters edges and faces for fillet selection. You can select loops in two ways: through the right-click Select Loop option, or by selecting a face and Ctrl+selecting an edge on the face.

Another option for selecting edges in the Fillet command is the Select Through Faces option, which appears on the Fillet Options panel. This option enables you to select edges that are hidden by the model. This can be a useful option on a part with few hidden edges, or a detrimental option on a part where there are many edges due to patterns, ribs, vents, or existing fillets. You can control a similar option globally for features other than fillets by choosing Tools ⇨ Options ⇨ Display/Selection, Allow Selection In HLR [Hidden Lines Removed], and Shaded Modes. ■

Faces and Features selections are useful when you are creating fillets where you want the selections to update. In Figure 7.31, the ribs that are intersecting the circular boss are also being filleted. If the rib did not exist when the fillet was applied, but was added later and reordered so that it came before the Fillet feature; then the fillet selection automatically considers the rib. If the fillet used edge selection, this automatic selection updating would not have taken place.

Tangent propagation

By default, fillets have the Tangent Propagation option turned on. This is usually a good choice, although there may be times when you want to experiment with turning it off. Tangent propagation simply means that if you select an edge to fillet, and this edge is tangent to other edges, then the fillet will keep going along tangent edges until it forms a closed loop, the tangent edges stop, or the fillet fails.

If you deselect Tangent Propagation, but there are still tangent edges, you may see different results. One possible result is that it could fail. One of the tricks with fillet features is to try to envision what you are asking the software to do. For example, if one edge is filleted and the next edge is not, then how is the fillet going to end? Figure 7.31 shows two of the potential results when fillets are asked not to propagate. The fillet face may continue along its path until it runs off the part or until the feature fails.

Tip

This may sound counterintuitive, but sometimes when fillet features fail, it may be useful to deselect propagation and make the fillet in multiple features. There are times when creating two fillets like the one shown in Figure 7.31 will work, and making the same geometry as a single feature will not. This may be due to geometry problems where the sharp edges come together and are eliminated by the fillet. ■

Best Practice
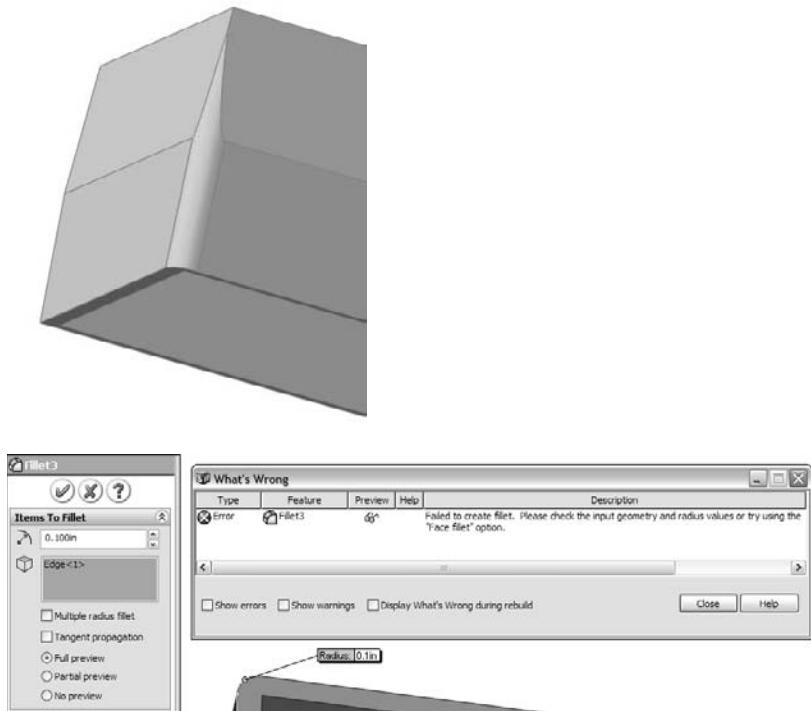
In general, fillets should be the last features that are applied to a model, particularly the small cosmetic or edge break fillets. Larger fillets that contribute to the structure or overall shape of the part may be applied earlier.

Be careful of the rock-paper-scissors game that you inevitably are caught up in when modeling plastic parts and deciding on the feature order of fillets, draft, and shell. Most fillets should come after draft, and large fillets should come before the shell. Draft may come either before or after the shell, depending on the needs of the area that you are dealing with on the part. In short, there is no single set of rules that you can consistently apply and that works best in all situations. ■

FIGURE 7.31

Deselecting the Tangent Propagation option
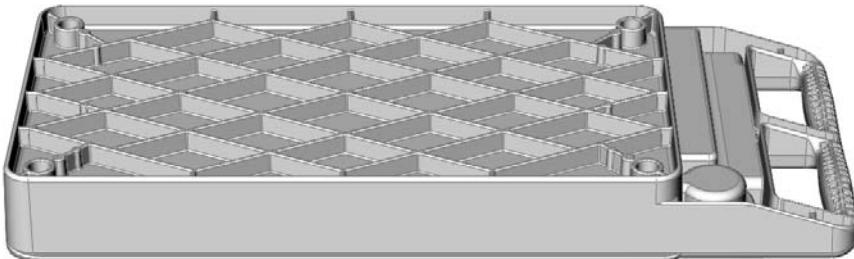
Dealing with a large number of fillets

Figure 7.32 shows a model with a bit of a filleting nightmare. This large plastic tray requires many ribs underneath for strength. Because the ribs may be touched by the user, the sharp edges need to be rounded. Interior edges need to be rounded also for strength and plastic flow through the ribs. Literally hundreds of edges would need to be selected to create the fillets if you do not use an advanced technique.

FIGURE 7.32

A plastic tray with a large number of fillets

Selecting entities

Some of the techniques outlined previously, such as face and feature selection, can be useful for quickly filleting a large number of edges. Another method that still selects a large number of edges, but is not as intuitive as the others, is window selection of the edges. To use this option effectively, you may want to first position the model into a view where only the correct edges will be selected, turn off the Select Through Faces option, and use the Edges Selection filter.

Using the FilletXpert

The FilletXpert is a tool with several uses. One of the functions is its capability to select multiple edges. A part like the one shown in Figure 7.32 is ideal for this tool. To use the FilletXpert, click the FilletXpert button in the Fillet PropertyManager. Figure 7.33 shows this. When you select an edge, the FilletXpert presents a pop-up tool bar giving you a choice of several selection options. Notice that Figure 7.33 shows the majority of the edges selected that are needed for this fillet.

The FilletXpert is also a tool that automatically finds solutions to complex fillet problems, particularly when you have several fillets of different sizes coming together.

The Corner tab of the FilletXpert enables you to select from different corner options, which are usually the result of different fillet orders. To use the CornerXpert, make sure the FilletXpert is active; then click on the corner face, and toggle through the options.

Using preview
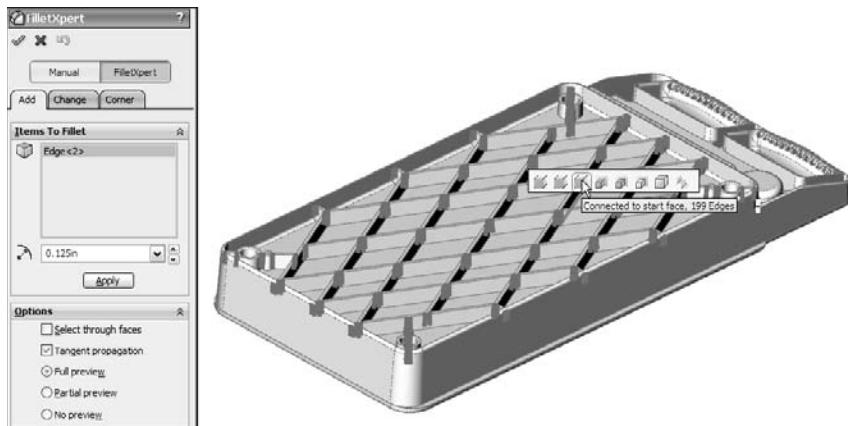
I like to use the fillet preview. It helps to see what the fillet will look like, and perhaps more important, the presence of a preview usually (but not always) means that the fillet will work.

Unfortunately, when you have a large number of fillets to create, the preview can cause a significant slowdown. Deselecting and using the Partial Preview are both possible options. Partial Preview shows the fillet on only one edge in the selection and is much faster when you are creating a large number of fillets.

Using the FilletXpert selection technique

Performance

For rebuild speed efficiency, you should make fillets in a minimum number of features. For example, if you have 100 edges to fillet, it is better for performance to do it with a single fillet feature that has 100 edges selected rather than 100 fillet features that have one edge selected. This is the one case where creating the feature and rebuilding the feature are both faster by choosing a particular technique. (Usually if it is faster to create, it rebuilds more slowly.)■

Best Practice

Although creation and rebuild speed are in sync when you use the minimum number of features to create the maximum number of fillets, this is not usually the case. (There had to be a downside.) When a single feature has a large selection, any one of these edges that fail to fillet will cause the entire feature to fail. As a result, a feature with 100 edges selected is 100 times more likely to fail than a feature with a single edge. Large selection sets are also far more difficult to troubleshoot when they fail than small selection sets that fail.■

Using folders
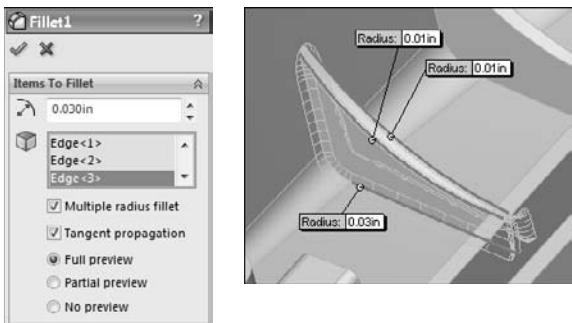
When you have a large number of fillet features, it can be tedious to navigate the FeatureManager. Therefore, it is useful to place groups of fillets into folders. This makes it easy to suppress or unsuppress all the fillets in the folder at once. Separate folders can be particularly useful if the fillets have different uses, such as fillets that are used for PhotoWorks models and fillets that are removed for FEA (Finite Element Analysis) or drawings.

Multiple Radius Fillet

The Multiple Radius Fillet option in the Fillet PropertyManager enables you to make multiple fillet sizes within a single fillet feature. Figure 7.34 shows how the multiple radius Fillet feature looks when you are working with it. You can change values in the callout flags or in the PropertyManager.

FIGURE 7.34

Using the Multiple Radius Fillet option

This may seem like an attractive way to group several fillets into as small a space on the FeatureManager as possible, but I cannot think of a single reason that would drive me to use this option. While there may be a small performance benefit to condensing several features into one, many more downsides adversely affect performance:

- Loss of control of feature order.
- A single failed fillet causes the whole feature, and thus all the fillets, to fail.
- Troubleshooting is far more difficult.
- Smaller groups of fillets cannot be suppressed without suppressing everything.
- You cannot change the size of a group of fillets together.

Best Practice

While this may be more personal opinion than best practice, I believe that there are good reasons to consider using techniques other than single features that contain many fillets, or single features that drive fillets of various sizes. Best practice would lean more toward grouping fillets that have a similar use and the same size. For example, you may want to separate fillets that break corners on ribs from fillets that round the outer shape of a large plastic part.

Another consideration is feature order when it comes to the fillet's relationship to draft and shell features. If the fillets are all grouped into a single feature, then controlling this relationship becomes impossible.■
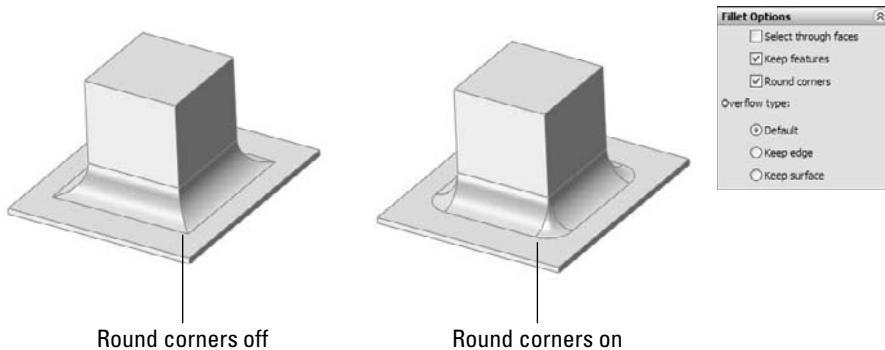
Round corners

The Round corners option refers to how SolidWorks handles fillets that go around sharp corners. By default, this setting is off, which leaves fillets around sharp corners looking like mitered picture frames. If you turn this setting on, the corner looks like a marble has rolled around it. Figure 7.35 shows the resulting geometry from both settings.

**FIGURE 7.35**

The Round corners option, both on and off

Round corners off   Round corners on
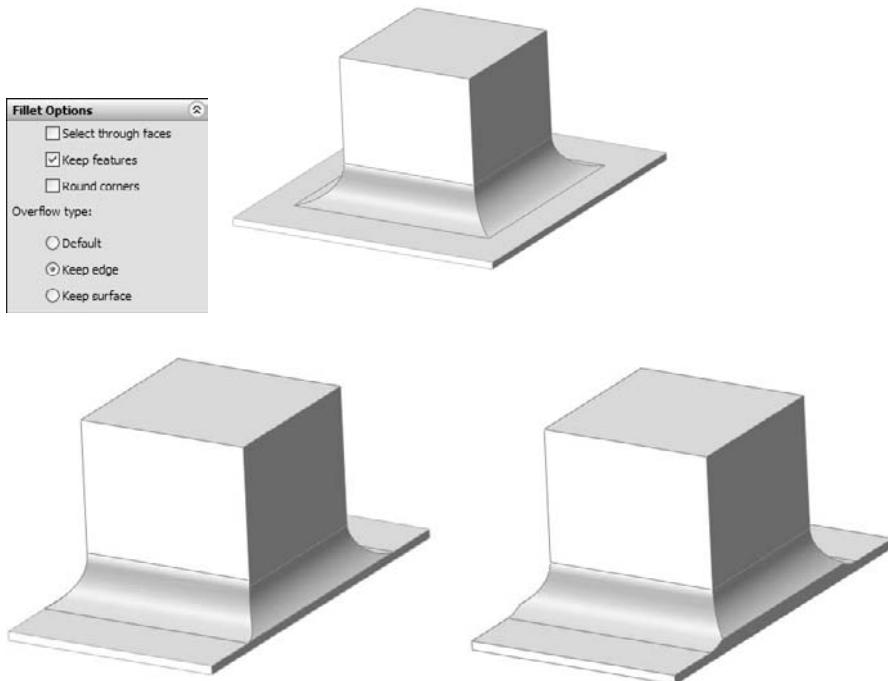
Keep edge/Keep surface

The Keep edge/Keep surface toggle determines what SolidWorks should do if a fillet is too big to fit in an area. The Keep Edge option keeps the edge where it is and tweaks the position (not the radius) of the fillet to make it meet the edge. The Keep surface option keeps the surfaces of the fillet and the end face clean; however, to do this, it has to tweak the edge. There is often a tradeoff when you try to place fillets into a space that is too small. Sometimes it is useful to try to visualize what you think the result should look like. Figure 7.36 shows how the fillet would look in a perfect world, followed by how the fillet looks when cramped with the Keep edge option and how it looks when cramped with the Keep surface option.

The Default option chooses the best option for a particular situation. As a result, it seems to use the Keep edge option unless it does not work, in which case it changes to the Keep surface option.

FIGURE 7.36

The Keep edge option and the Keep surface option

Keep features
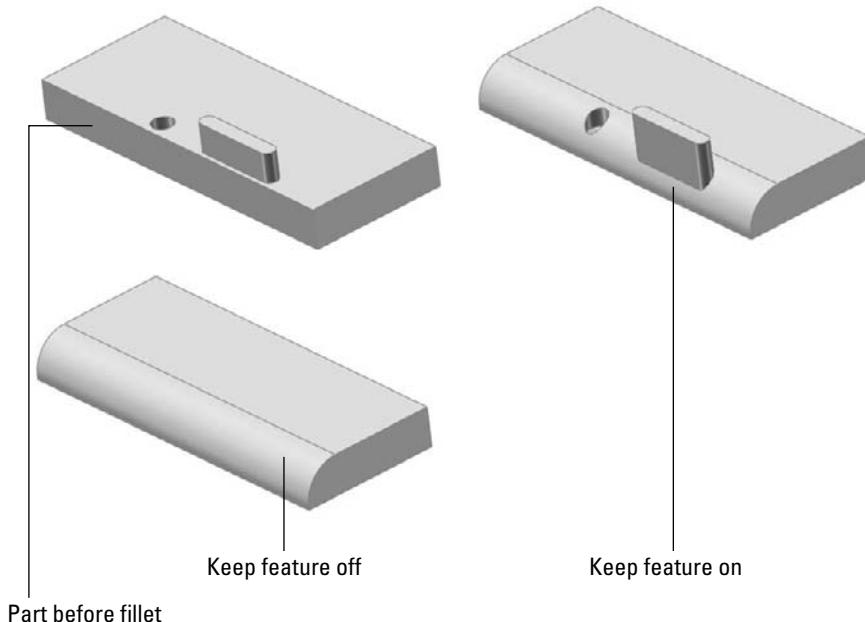
The Keep Feature option appears on the Fillet Options panel of the Fillet PropertyManager. By default, this option is turned on. If a fillet surrounds a feature such as a hole (as long as it is not a through hole) or a boss, then deselecting the Keep Feature option removes the hole or boss. When Keep Feature is selected, the faces of the feature trim or extend to match the fillet, as shown in Figure 7.37.

FIGURE 7.37

The Keep Feature option, both selected and unselected

Keep feature off   Keep feature on

Part before fillet

Creating variable radius fillets

Variable radius fillets are another powerful weapon in the fight against boring designs; they also double as a useful tool to solve certain problems that arise. Although it is difficult to define exactly when to use the variable radius fillet, you can use it when you need a fillet to round an edge, and it has to change in size to fit the available geometry.

Best Practice

It may be easier to identify when not to use a variable radius fillet. Fillets are generally used to round or break edges, not to sculpt a part. If you are using fillets to sculpt blocky parts and are not actively trying to make blocky parts with big fillets, you may want to consider another approach and use complex modeling, which gives the part a better shape and makes it more controllable. Other options exist that give you a different type of control, such as the double hold line fillet.■
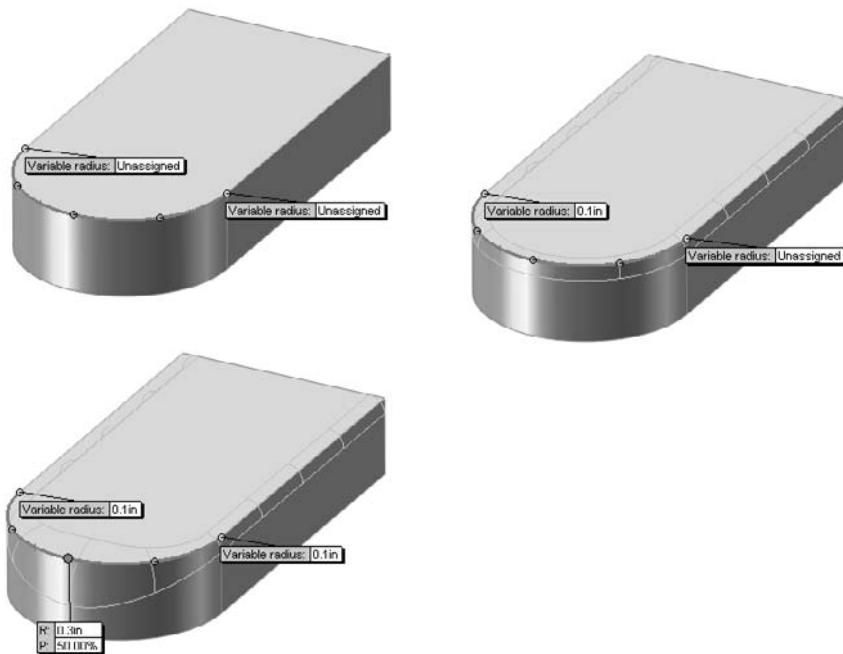
In some ways, variable radius fillets function like other fillets. For example, they offer propagation to tangent edges and preview options.

Applying the values

When you first select an edge for the variable radius fillet, the endpoints are identified by callout flags with the value *unassigned*. A preview does not display until at least one of the points has a radius value in the box. You can also apply radius values in the PropertyManager, but they are easier to keep track of using the callouts. Figure 7.38 shows a variable radius fillet after the edge selection, after one value has been applied, and after three values have been applied. To apply a radius value that is not at the endpoint of an edge, you can select one of the three colored dots along the selected edge. The preview should show you how the fillet will look in wireframe display.

Assigning values to a variable radius fillet

By default, the variable radius fillet puts five points on an edge, one at each endpoint, one at the midpoint, and one each halfway between the ends and middle. If you want to create an additional control point, there are three ways to do it:

- Ctrl+drag an existing point
- Select the callout of an existing point and change the P (percentage) value
- Change the Number of Instances value in the Variable Radius Parameters panel of the PropertyManager
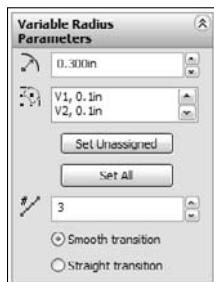
If you have selected several edges, and several unassigned values are on the screen, then you can use the Set Unassigned option in the PropertyManager to set them all to the same value. The Set All option sets all radius values to the same number, including any values that you may have changed to be different than the rest. Figure 7.39 shows the Variable Radius Parameters panel.

FIGURE 7.39

The Variable Radius Parameters panel of the Fillet PropertyManager

Another available option with the variable radius fillet is that you can set a value of zero at an end of the fillet. You need to be careful about using a zero radius, because it is likely to cause down-stream problems with other fillets, shells, offsets, and even machining operations. You cannot assign a zero radius in the middle of an edge, only at the end. If you need to end a fillet at a particular location, you can use a split line to split the edge and apply a zero radius at that point. Figure 7.40 shows a part with two zero-radius values.

FIGURE 7.40

Zero-radius values in the variable radius fillet
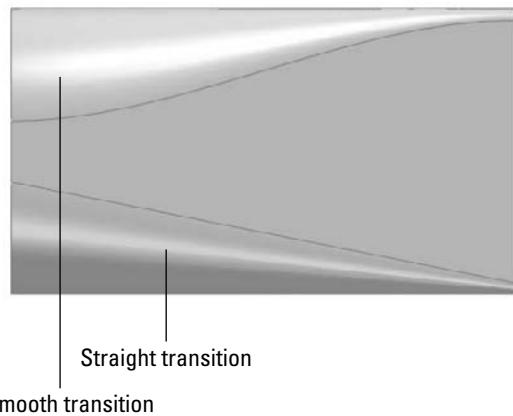
Straight versus smooth transitions

Variable radius fillets have an option for either a straight transition or a smooth transition. This works like the two-profile lofts that were mentioned earlier in this chapter. The names may be somewhat misleading because both transitions are smooth. The straight transition goes in a straight line, from one size to the next, and the smooth transition takes a swooping S-shaped path between the sizes. The difference between these two transitions is demonstrated in Figure 7.41.

FIGURE 7.41

Straight versus smooth transitions of a variable radius fillet

Straight transition

Smooth transition

Other reasons to use the variable radius fillet

Variable radius fillets use a different method to create the fillet geometry than the default constant radius fillet. Sometimes using a variable radius fillet can make a difference where a constant radius fillet does not work. This is sometimes true even when the variable radius fillet uses constant radius values. It is just another tool in the toolbox.

Using face Fillets

Face Fillets may be the most flexible type of fillet because of the range of what they can do. Face Fillets start as simply an alternate selection technique for a constant radius fillet and extend to the extremely flexible double hold line face fillet, which is more of a blend than a fillet.

Under normal circumstances, the default fillet type uses the selection of an edge to create the fillet. An edge is used because it represents the intersection between two faces. However, there can sometimes be a problem with the edge not being clean or being broken up into smaller pieces, or any number of other reasons causing a constant radius fillet using an edge selection to fail. In cases like this, SolidWorks displays the error message, "Failed to create fillet. Please check the input geometry and radius values or try using the 'Face Fillet' option."
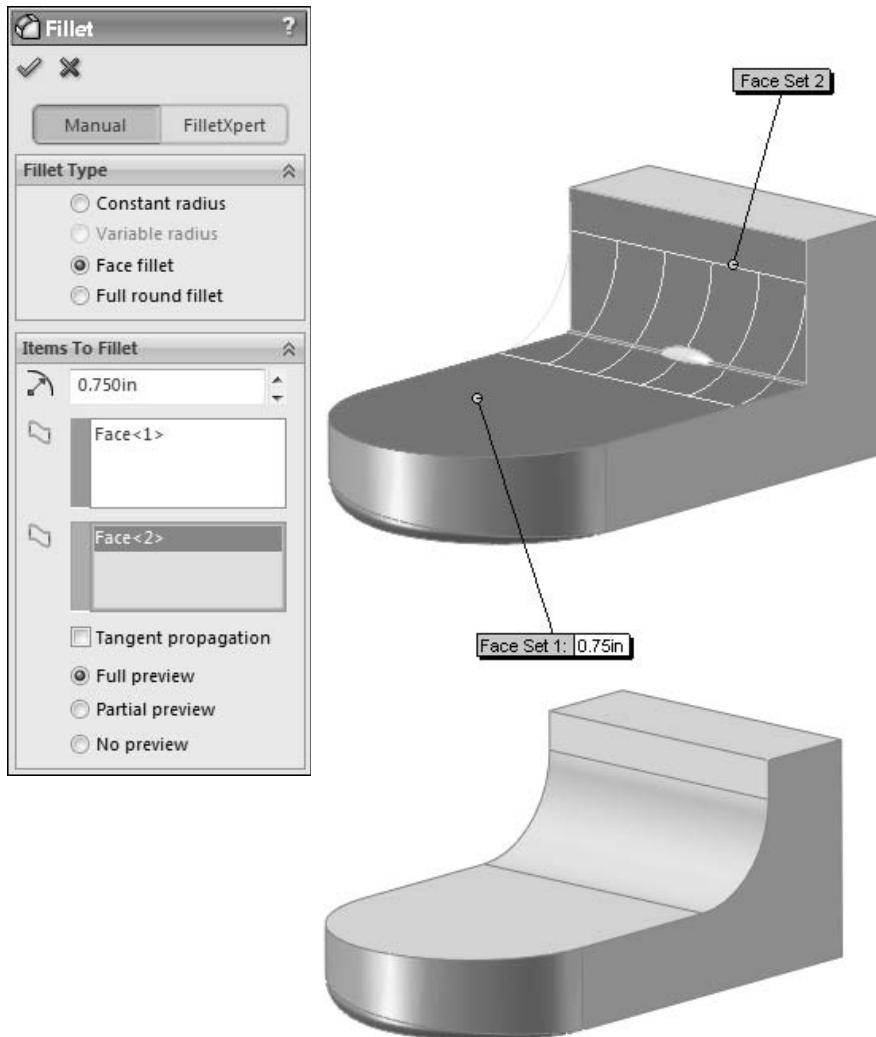
Users almost universally ignore these messages. In the situation shown in Figure 7.42, the Face Fillet option suggested in the error message is exactly the one that you should use. Here the face fillet covers over all the junk on the edge that prevents the fillet from executing.

FIGURE 7.42
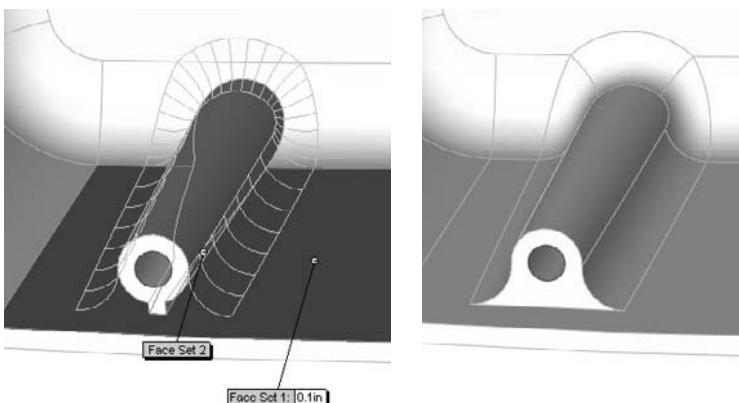
A face fillet covering a bad corner

Face Fillets are sometimes amazing at covering over a mess of geometry that you might think you could never fillet. The main limitation on fillets of this type is that the fillet must be *big* enough to bridge the gap. That's right, I said *big* enough. Face Fillets can fail if they are either too small or too large. Figure 7.43 shows a complex fillet situation that is completely covered by a face fillet.

FIGURE 7.43

A face fillet covering complex geometry

On the CD-ROM

The model used for this image can be found on the CD-ROM, with the filename `Chapter 7 Plastic Cover Fillets.sldprt.`■

Continuous curvature Face Fillets

Curvature continuity refers to the quality of a transition between two curves or faces, where the curvature is the same or continuous at and around the transition. The best way to convey this concept is with simple 2D sketch elements. When a line transitions to an arc, you have non-continuous curvature. The line has no curvature, and there is an abrupt change because the arc has a specific radius.

Note

**Radius is the inverse of curvature, and so r = 1/c. For a straight line, r = ∞, in which case c = 0.**■
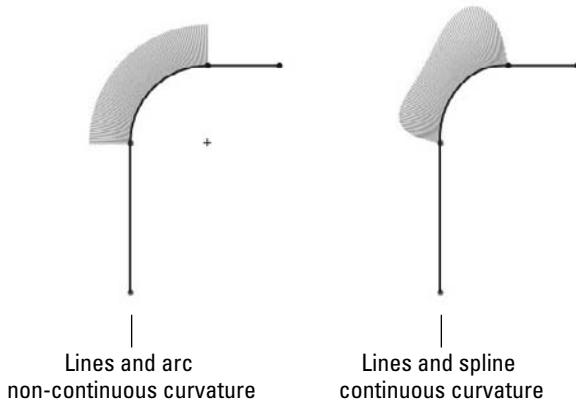
To make the transition from r = ∞ to r = 2 smoothly, you would need to use a variable radius arc if such a thing existed. There are several types of sketch geometry that have variable curvature, such as ellipses, parabolas, and splines. Ellipses and parabolas follow specific mathematical formulas to create the shape, but the spline is a general curve that can take on any shape that you want, and you can control its curvature to change smoothly or continuously. Splines, by their very definition, have continuous curvature within the spline, although you cannot control the specific curvature or radius values directly.

All this means that continuous curvature Face Fillets use a spline-based variable-radius section for the fillet, rather than an arc-based constant radius. Figure 7.44 illustrates the difference between continuous curvature and constant curvature. The spikes on top of the curves represent the curvature (1/r, and so the smaller the radius, the taller the spike). These spikes are called a *curvature comb*.

FIGURE 7.44

Using curvature combs to evaluate transitions

Lines and arc
non-continuous curvature

Lines and spline
continuous curvature
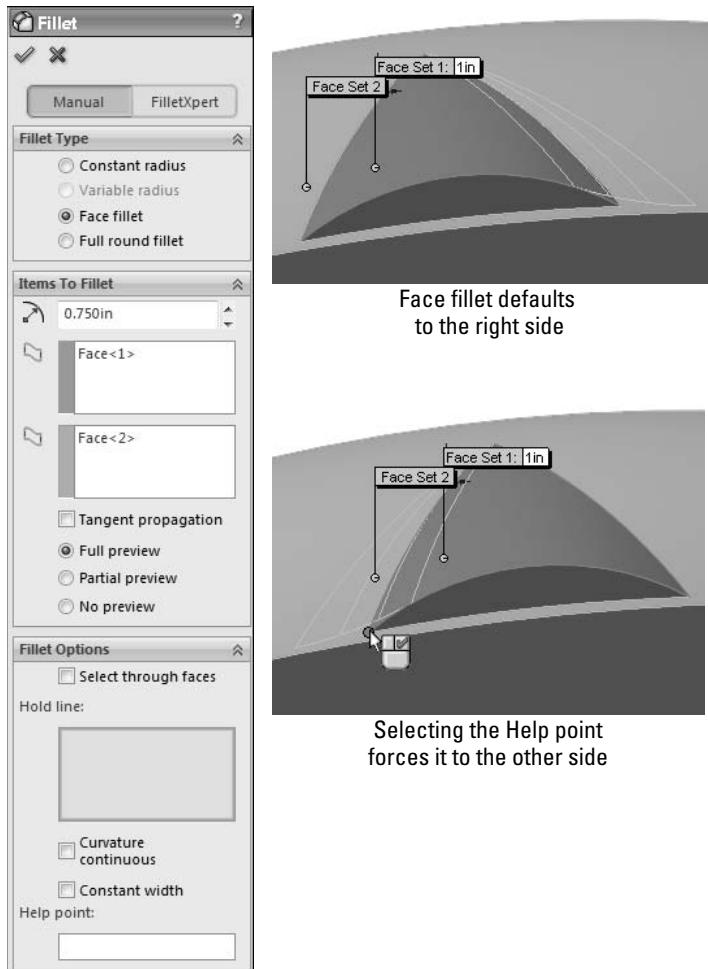
In Comparison to Figure 7.45, notice how the curvature comb immediately jumps from no curvature to the constant arc radius, but the spline image ramps up to a curvature that varies.

Face Fillets with Help Point

The Help Point in the Face Fillet PropertyManager is a fairly obscure option. However, it is useful in cases where the selection of two faces does not uniquely identify an edge to fillet. For example, Figure 7.45 shows a situation where the selection of two faces could result in either one edge or the other being filleted (normally, I would hope that both edges would be filleted). The fillet will default to one edge or the other, but you can force it to a definite edge using the Help Point.

FIGURE 7.45

Using a Help Point with a Face Fillet

Face fillet defaults
to the right side

Selecting the Help point
forces it to the other side

In some cases, the Help Point is ignored altogether. For example, if you have a simple box, and select both ends of the box as selection set 1, and the top of the box as selection set 2, then the fillet could go to either end. Consequently, assigning a Help Point will not do anything in this case, because multiple faces have been selected. The determining factor is which of the multiple
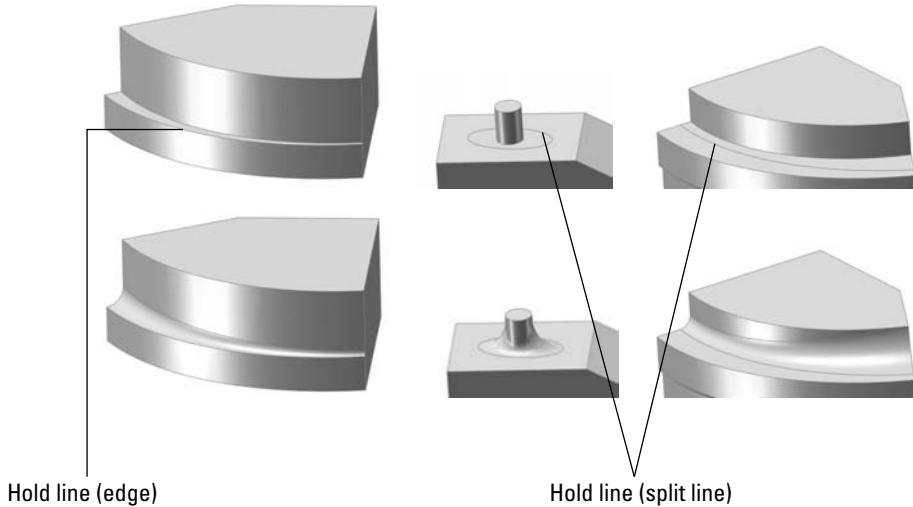
faces is selected first. If this were a more commonly used feature, the interface for it might be made a little less cryptic, but because this feature is rarely, if ever, used, it just becomes a quirky piece of trivia.

Single hold line fillet

A single hold line fillet is a form of variable radius fillet, but instead of it being defined using the variable radius fillet type, it is created using the face fillet. Rather than the radius being driven by specific numerical values, it is driven by a hold line, or edge, on the model. The hold line can be an existing edge, forcing the fillet right up to the edge of the part, or it can be created by a split line, which enables you to drive the fillet however you like. Figure 7.46 shows these two options, before and after the fillets. Notice that these fillets are still arc-based fillets; if you were to take a cross-section perpendicular to the edge between filleted faces, it would be an arc cross-section with a distinct radius. However, in the other direction, hold line fillets do not necessarily have a constant radius, although they may if the hold line is parallel with the edge between faces.

FIGURE 7.46

Single hold line fillets

Hold line (edge)   Hold line (split line)
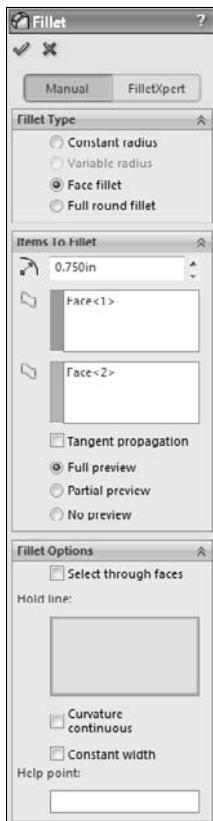
You can select the Hold line in the Fillet Options panel of the Face Fillet PropertyManager, as shown in Figure 7.47. The top panel, Fillet Type, is available only when the feature is first created.

When you edit it after it has been created, the Fillet Type panel does not appear. As a result, you cannot change from one top-level type of fillet to another after it has been initially created.

FIGURE 7.47

The PropertyManager interface for the hold line face fillet


Double hold line fillet

There are times when a single hold line does not meet your needs. The single hold line controls only one side of the fillet, and in order to control both sides of the fillet, you must use a double hold line fillet. SolidWorks software does not specifically differentiate between the single and double hold line fillets, but they are radically different in how they create the geometry. When both
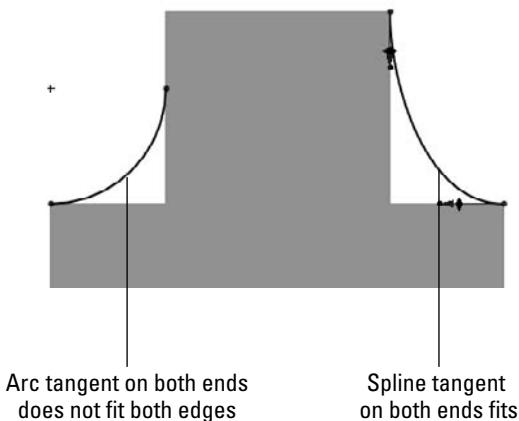
sides of the fillet are controlled, it is not possible to span between the hold lines with an arc that is tangent to both sides unless you were careful about setting up the hold lines so that they are equi-distant from the edge where the faces intersect. This means that the double hold line fillet must use a spline to span between hold lines, as shown in Figure 7.48.

A double hold line uses a spline, not an arc.

Arc tangent on both ends
does not fit both edges

Spline tangent
on both ends fits
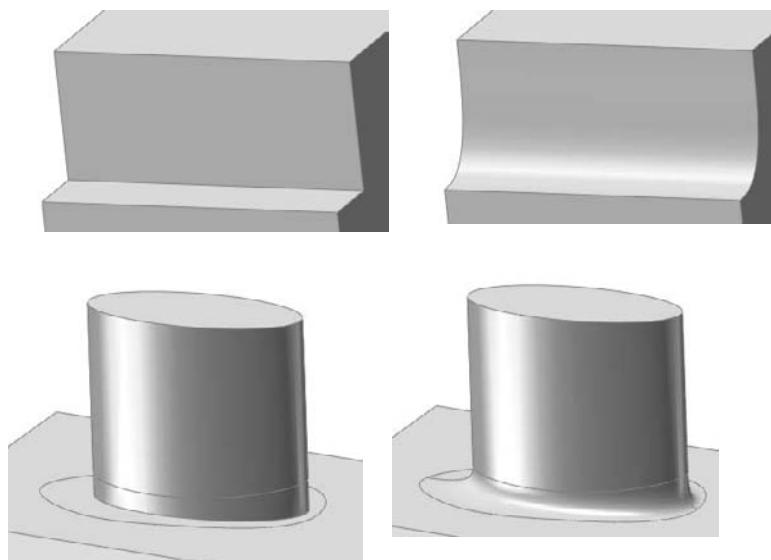
To get this feature to work, you need to use the curvature continuous option in the Fillet Options panel. Remember that this option creates a spline-based fillet rather than an arc-based fillet, which is exactly what you need for a double hold line fillet. This makes the double hold line fillet more of a blend than a true fillet. Figure 7.49 shows examples of the double hold line fillet.

Constant width fillet

The Constant width option of the Face Fillet PropertyManager drives a fillet by its width rather than by its radius. This is most helpful on parts where the angle of the faces between which you are filleting is changing dramatically. Figure 7.50 illustrates two situations where this is particu-larly useful. The setting for constant width is found in the Options panel of the Face Fillet PropertyManager. The part shown in the images is on the CD-ROM as `Chapter 7 Constant Width.sldprt`.
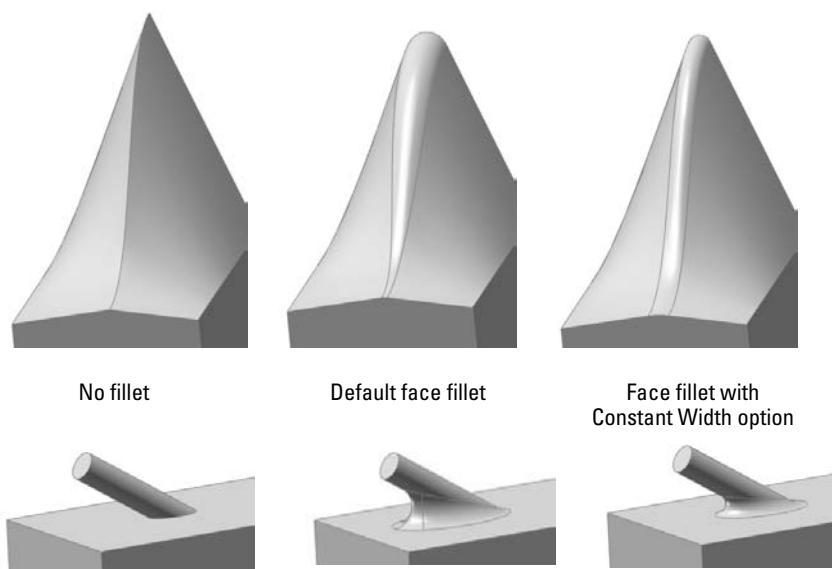
FIGURE 7.49

Examples of the double hold line fillet

FIGURE 7.50

The constant width fillet

No fillet   Default face fillet   Face fillet with
   Constant Width option
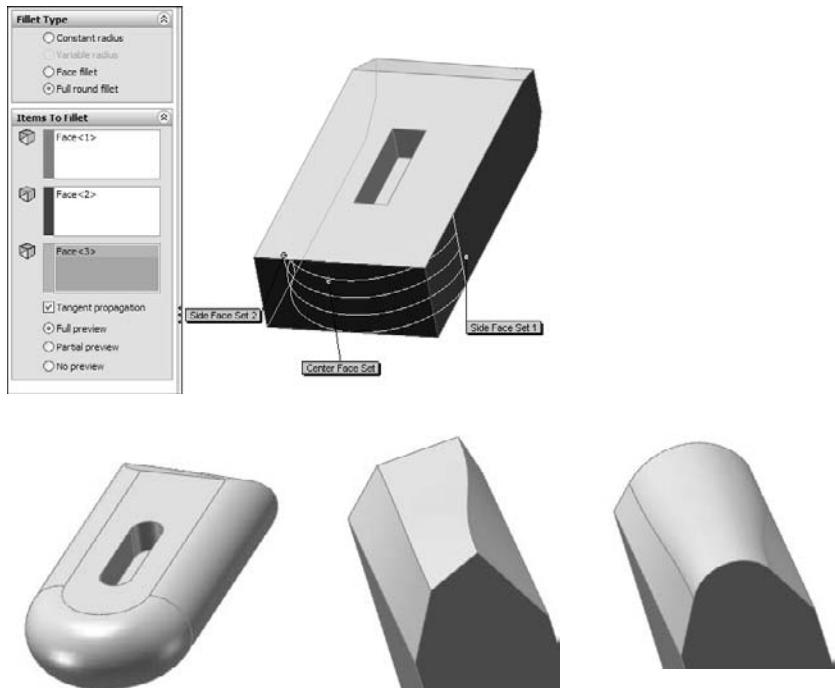
Applying a full round fillet

The full round fillet is very useful in many situations. In fact, it may actually work in situations where you would not expect it to. It does require quite a bit of effort to accomplish the selection, but it compensates by enabling you to avoid alternate fillet techniques.

To create a full round fillet, you have to select three sets of faces. Usually one face in each set is sufficient. The fillet is tangent to all three sets of faces, but the middle set is on the end and the face is completely eliminated. Figure 7.51 shows several applications of the full round fillet. Notice that it is not limited to faces of a square block, but also propagates around tangent entities and can create a variable radius fillet over irregular lofted geometry.

FIGURE 7.51

A full round fillet
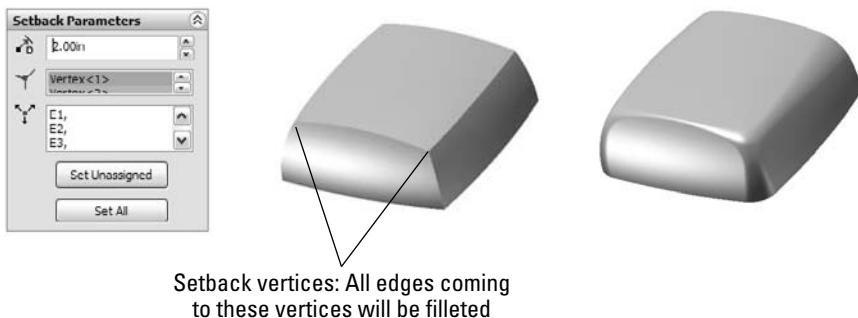
Building a setback fillet

The setback fillet is the most complex of the fillet options. You can use the Setback option in conjunction with constant radius, multiple radius, and variable radius fillet types. A setback fillet blends several fillets together at a single vertex, starting the blend at some "setback" distance along each filleted edge from the vertex. At least three, and often more, edges come together at the setback vertex. Figure 7.52 shows the PropertyManager interface and what a finished setback fillet looks like. The following steps demonstrate how to use the setback fillet.

The Setback Fillet interface and a finished fillet

Setback vertices: All edges coming
to these vertices will be filleted
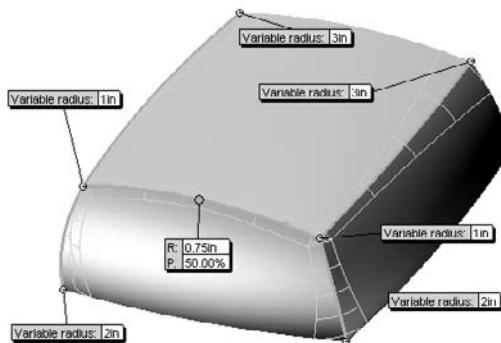
Setting up a setback fillet can take some time, especially if you are just learning about this feature. You must specify values for fillet radiuses, select edges and vertices, and specify six setback distances. If you are using multiple radius fillets or variable radius fillets, then this becomes an even larger task. The steps are as follows:

1. **Determine the type of fillet to be used:**
   - Constant radius fillet
   - Multiple radius fillet
   - Variable radius fillet

2. **Select the edges to be filleted.** Selected edges must all touch one of the setback vertices that will be selected in a later step.

3. **Assign radius values for the filleted items.** Figure 7.53 shows a sample part that illustrates this step.

FIGURE 7.53

The setback fillet setup for Steps 1 through 3

4. **Select the setback vertices.** In the Setback Parameters panel of the PropertyManager, with the second box from the top highlighted, select the vertices. Although this box looks like it is only big enough for a single selection, it can accept multiple selections.

5. **Enter setback values.** As shown in Figure 7.54, the setback callout flags have leaders that point from a specific value to a specific edge. The dimensions refer to distances, as shown in the image to the right in Figure 7.54. The setback distance is the distance over which the fillet will blend from the corner to the fillet.
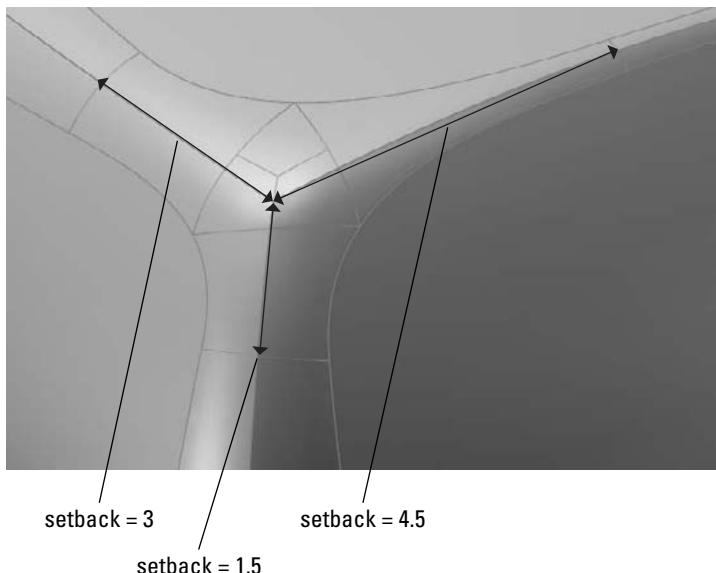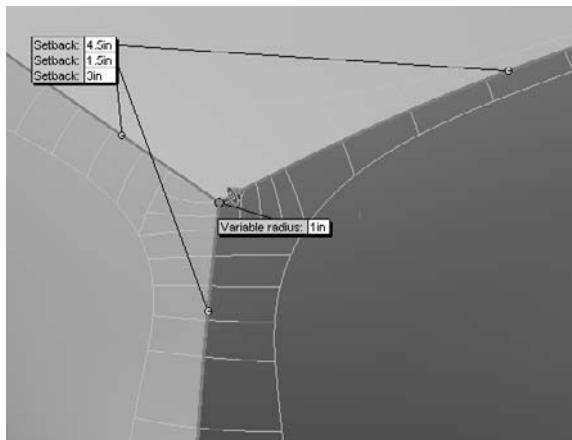
Caution

When you select multiple vertices, the preview arrows that indicate which edge you are currently setting the setback value for may be incorrect. The arrows can only be shown on one vertex; therefore, you may want to rely on the leaders from the callouts to determine which setback distance you are currently setting.■

6. **Repeat the process for all selected setback vertices.** If you are using a preview, then you may notice that the preview goes away when starting a second set of setback values. Don't worry. This is probably not because the feature is going to fail. Once you finish typing the values, the preview will return. When you have spent as much time setting up a feature as you will spend on this, seeing the preview disappear can be frustrating; however, persevere, and it will return.

Entering setback values

setback = 3

setback = 4.5

setback = 1.5

Selecting a Specialty Feature

SolidWorks contains several specialty features that perform tasks that you will use less often. Although you will not use these features as frequently as others, you should still be aware of them and what they do, because you never know when you will need them.

Using the Dome feature

The Dome feature in SolidWorks is generally applied to give some shape to flat faces. A great example of where a Dome fits well is the cupped bottom of a plastic bottle, or a slight arch on top of buttons for electronic devices.

Until SolidWorks 2010, another very similar feature existed, which was called Shape. You can no longer make Shape features, but you may find one from time to time in old parts. If you find a Shape feature on an old part, it will continue to function unless any of its parent geometry changes. Shape features will not update in SolidWorks 2010 or later. SolidWorks recommends you re-create the geometry as another feature, possibly a Dome or Freeform feature.

Best Practice

Dome features are best used when you are looking for a generic bulge or indentation, and are not too concerned about controlling the specific shape. Occasionally, a dome may be exactly what you need, but when you need more precise, predictable control, then you should use the Fill, Boundary, or Loft feature.■
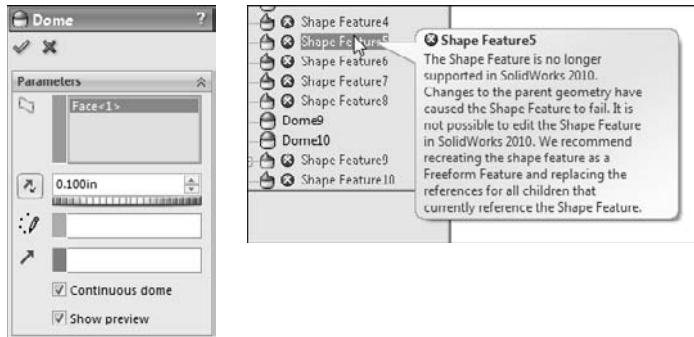
The Dome feature has several attributes that will either help it qualify for a given task, or disqualify it. These attributes can help you decide if it will be useful in situations you encounter:

- The Dome feature can create multiple domes on multiple selected faces in a single feature, although it creates only a single dome for each face.
- Using the Elliptical Dome setting, Dome can create a feature that is tangent to the vertical.
- Dome can use a constraint sketch to limit its shape.
- Dome works on non-planar faces.
- Dome cannot establish a tangent relationship to faces bordering the selected face.
- Dome cannot span multiple faces.
- Dome displays a temporary untrimmed four-sided patch that extends beyond the selected face when you use it on a non-four-sided face.
- Dome functions only on solids, not on surfaces.

The error caused by a Shape feature being forced to update in SolidWorks 2010 is shown in Figure 7.55.

FIGURE 7.55

Shape features may fail in SolidWorks 2010

> **Shape Feature5**
> The Shape Feature is no longer supported in SolidWorks 2010. Changes to the parent geometry have caused the Shape Feature to fail. It is not possible to edit the Shape Feature in SolidWorks 2010. We recommend recreating the shape feature as a Freeform Feature and replacing the references for all children that currently reference the Shape Feature.
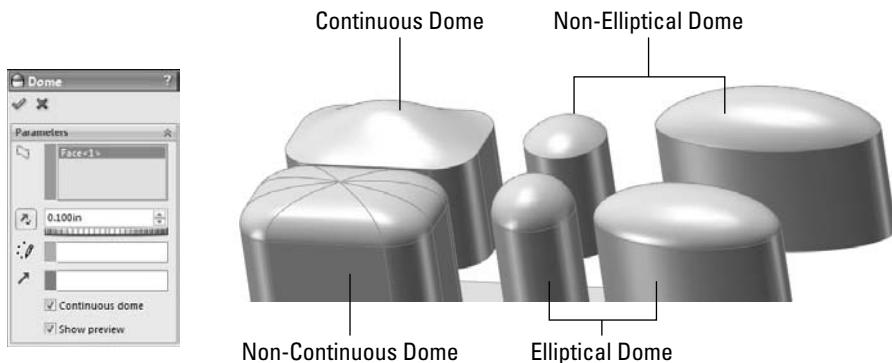
The Dome feature has two notable settings: the Elliptical Dome and Continuous Dome.

The Elliptical Dome is available only on flat faces where the boundary is either a complete circle or an ellipse. The cross-section of the dome is elliptical, and does not account for draft, which means that it is always tangent to the perpendicular from the selected flat face.

The Continuous Dome is a setting for any non-circular or elliptical face, including polygons and closed-loop splines. The setting results in a single unbroken face. If you deselect the Continuous Dome setting, it functions like the Elliptical Dome setting. Figure 7.56 shows the most useful settings for the Dome feature.

FIGURE 7.56

Examples of various types of domes

Continuous Dome   Non-Elliptical Dome

Non-Continuous Dome   Elliptical Dome

Using the Wrap feature

The Wrap feature enables you to wrap 2D sketches around cylindrical and conical faces. However, trying to wrap around 360 degrees can cause some difficulties. Although all the available information on the Wrap feature says that you can wrap onto a conical surface, it fails to mention that the point of the cone must be cut off in order for it to work.

The Wrap feature works by flattening the face, relating the sketch to the flat pattern of the face, and then mapping the face boundaries and sketch back onto the 3D face. The reason why it is limited to cylindrical and conical faces is that these types of geometry are *developable*. This means that the faces can be mapped to the flat pattern through some relatively simple techniques that happen behind the scenes. Developable geometry can be flattened without stretching. You will see in a later chapter that sheet metal functions are limited in the same way and for the same reasons.
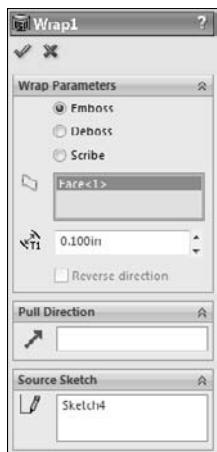
SolidWorks does not wrap onto other types of surfaces, such as spherical, toroidal, or general NURBS surfaces, because you cannot flatten these shapes without distorting or stretching the material. There is software that can flatten these shapes, but it is typically done for sheet-metal deep drawing applications, which highly deform the metal. Figure 7.57 shows the Wrap PropertyManager interface.

The Wrap feature has three main options:

- Emboss
- Deboss
- Scribe

FIGURE 7.57

The Wrap PropertyManager interface

Scribe

Scribe is the simplest of the options to explain, and understanding it can help you understand the other options. Scribe creates a split line-like edge on the face.

Several requirements must be met in order to make a wrap feature work:

- The face must be a cylindrical or conical face.
- The loop must be a closed loop or nested closed loop 2D sketch.
- The sketch must be on a plane that is either tangent to or parallel to another plane that is tangent to the face.
- Wrap supports multiple closed loops within a single feature.
- Wrap supports wrapping onto multiple faces.
- The wrap should not be self-intersecting when it wraps around the part. (Self-intersection will not cause the feature to fail, but on the other types, Emboss and Deboss, it may produce unexpected results.)

Scribes can be created on solid or surface faces. Scribed surfaces are frequently thickened to create a boss or a cut.

Emboss

The Wrap Emboss option works much like the scribe, but it adds material inside the closed loop sketch, at the thickness that you specify in the Emboss PropertyManager. Embossing can only be done on solid geometry. If the feature self-intersects, then the intersecting area is simply not embossed, and is left at the level of the original face. One result is that creating a full wrap-around feature, such as the geometry for a barrel cam, requires a secondary feature. This is because the Wrap feature always leaves a gap, regardless of whether the sketch to be wrapped is under or over the diameter-multiplied-by-pi length.

Tip
To work around this problem, you can use a loft, extrude, or revolve feature to span the gap.■

When you use the Emboss option, you can set up the direction of pull and assign draft so that the feature can be injection molded. This limits the size of the emboss so that it must not wrap more than 180 degrees around the part.

Deboss
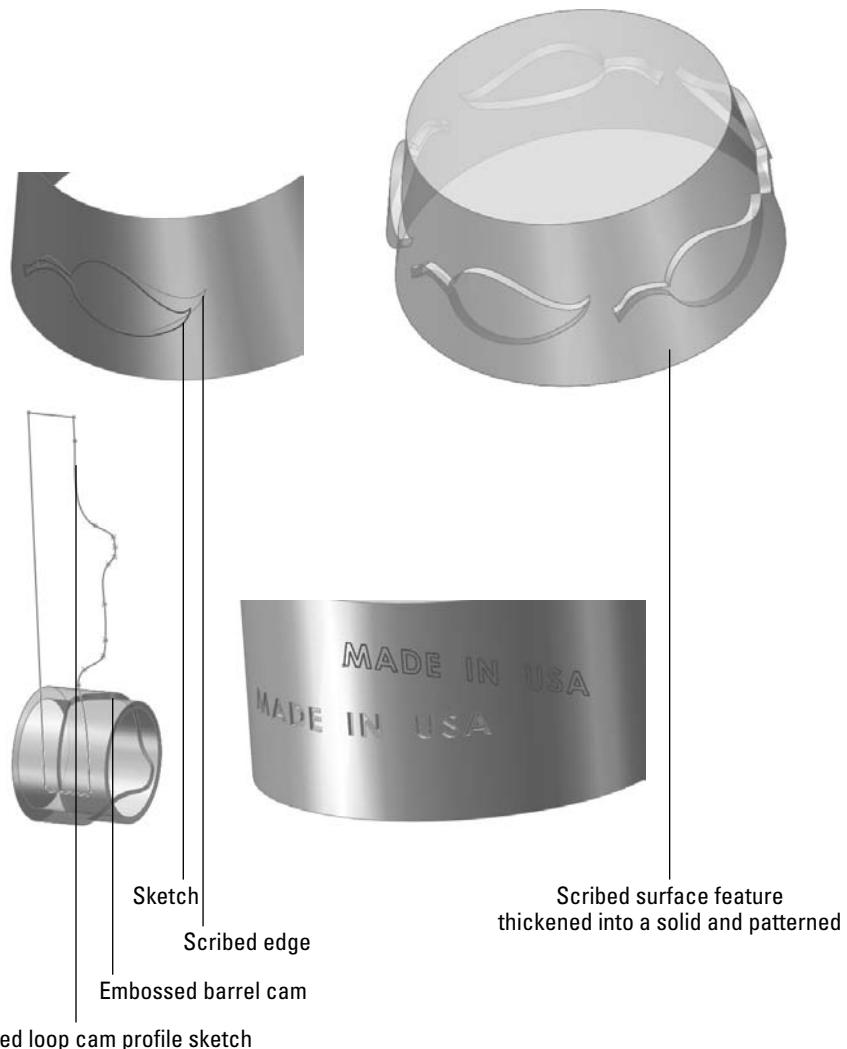
Deboss is just like emboss, except that it removes material instead of adding it. Figure 7.58 demonstrates all these options. The part shown in the images is available on the CD-ROM with the filename `Chapter 7 Wrap.sldprt`. For each of the demonstrated cases, the original flat sketch is shown to give you some idea of how the sketch relates to the finished geometry.

FIGURE 7.58

The Wrap feature options

Sketch

Scribed edge

Embossed barrel cam

Closed loop cam profile sketch

Scribed surface feature
thickened into a solid and patterned

Keep in mind that this feature is not like the projected sketch. A projected sketch is not foreshortened on the curved surface, but is projected normal from the sketch plane. A sketch that is one-inch long will measure one-inch along the curvature of the surface and will measure less than one inch linearly from end to end.

The scribed part in the previous figure was created on a conical surface body. The surface was then thickened as a separate body and patterned.

Cross-Reference

Chapter 26 covers working with multi-bodies, and Chapter 27 covers surfaces.■

The embossed cam employed a workaround with a revolve feature to close the gap that is always created when wrapping all the way around a part.

The example with the debossed text employs a direction of pull and draft so that the geometry can be molded.

Flex feature

The Flex feature is different from most other features in SolidWorks. Most other features create new geometry, but Flex (and Deform, which follows) takes existing geometry and changes its shape. Flex can affect the entire part, or just a portion of it. Flex works on both solid and surface bodies, as well as imported and native geometry.

Figure 7.59 shows the Flex PropertyManager interface. Flex has four main options and many settings. The four main options are as follows:

- **Bending.** Establishes two trim planes to denote the ends of the bent area, and specifies an angle or radius for the bend.

- **Twisting.** Establishes two trim planes to limit the area of the twist, and enters the number of degrees through which to twist.

- **Tapering.** Establishes two trim planes to limit the area of the taper. The body will be larger toward one end and smaller toward the other end.

- **Stretching.** Establishes two trim planes to limit the area to be stretched. You can stretch the entire body by moving the trim planes outside of the body.

Best Practice

Flex is not the kind of feature that you should use to actually design parts, but it can be extremely valuable when you need to show a part in an "in use" state. A simple example would be a rubber strap that stretches over something when it is used, but that is designed and manufactured in its free state. The geometry that you can create by using the flex functions is not generally production-model quality, but it is usually adequate for a looks-like model. ■

FIGURE 7.59

The Flex PropertyManager interface
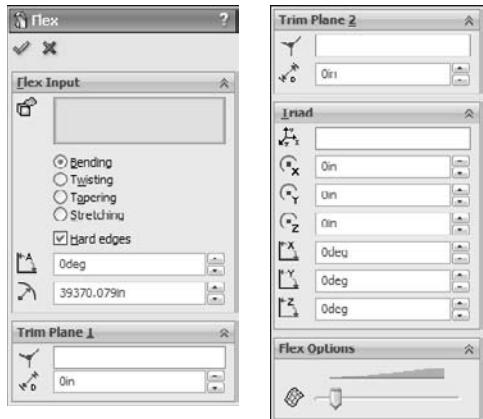
Figure 7.60 shows examples of each flex option using a model of a rubber grommet. The part shown in the figure can be found on the CD-ROM with the filename `Chapter 7 Flex.sldprt`.

In some cases, the triad and trim planes are slightly disoriented. The best thing to do in situations like this is to simply reorient the triad using the angle numbers in the Triad panel of the PropertyManager. This is also a solution if the planes are turned in such a way that the axis of bending is not oriented to the bend that the part requires.

The Flex feature is very conscious of separate bodies. In some cases this can be helpful, but in default situations when there is only one body in the part, it can be annoying. Remember to select the body to be affected in the very first selection box at the top of the PropertyManager.

Tip

If you want to bend only one of the tabs on the grommet, then the best solution is to split the single body into two bodies and flex only one of the bodies. The examples shown for twisting and stretching use this technique.■
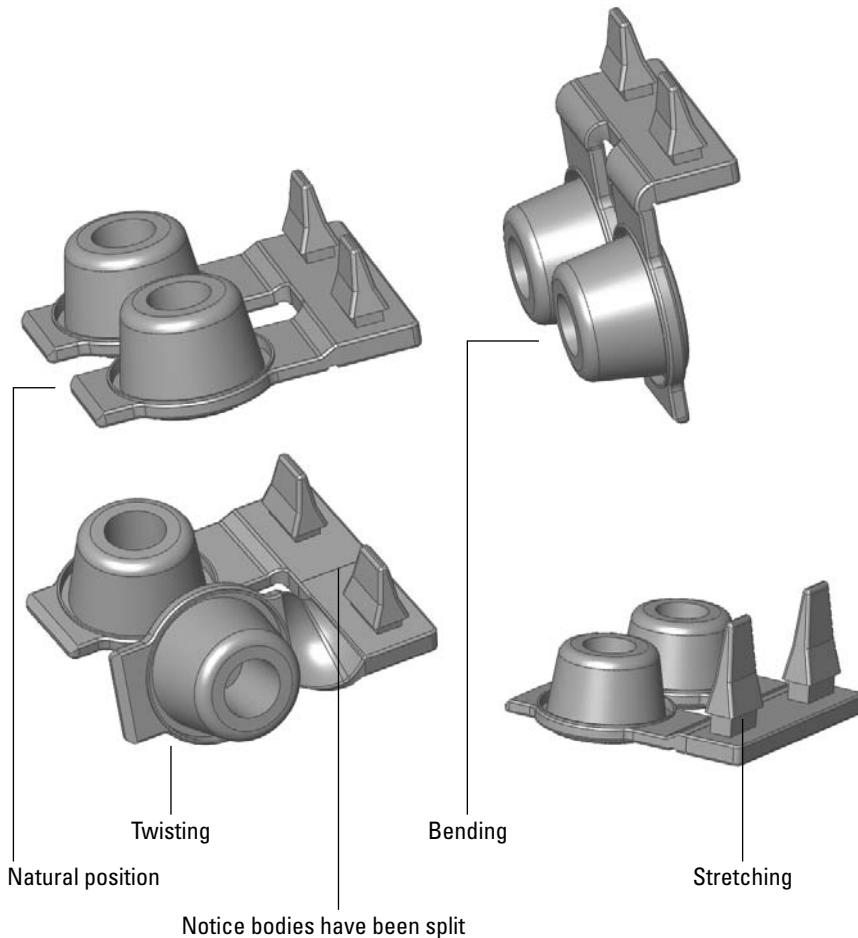
Cross-Reference

Splitting a single body into multiple bodies is covered in Chapter 26.■

You can place the trim planes by selecting a model vertex, by dragging the arrow on the plane, or by typing in a number. Be careful when dragging the plane arrows because dragging the border of the plane drags the flex value for the feature. (Dragging the plane in a bending operation is like changing the angle or radius for the bend.)

FIGURE 7.60

A rubber grommet in various flex states

Twisting

Natural position

Notice bodies have been split

Bending

Stretching

Using the triad can be very tricky. Moving the triad in the bending option moves the axis of the bend, and so it determines whether the bend will compress or stretch the material. The position of the triad also determines which side of the bent body will move or stay stationary, or if both sides will move. Placing the triad directly on a trim plane causes the material outside the bend on that side of the trim plane to remain stationary.

I highly recommend taking a look at the models that are provided with this chapter to examine the various functions of the Flex feature more carefully. The model uses configurations, which are covered in Chapter 10.

Applying the Deform feature

Like the Flex feature, the Deform feature changes the shape of the entire model without regard to parametrics, features, history, or dimensions. Some software packages call this technique *global shape modeling*. Also like Flex, Deform works on surface bodies as well as solids. Deform can also handle imported geometry as well as SolidWorks native parts. Model complexity is not an issue unless the part runs into itself during deformation.

The Deform feature is also another feature type that you may not use to actually design anything, but that you may use to show a model in a deformed state.

Also, it is a minor point, but if you use the large icons setting, notice that the toolbar icon used for Deform is the wrong one. The icon used for large icons is the Boundary Cut icon. Just be careful if you use large icons.

Best Practice

Typically, if you want a model to have a certain shape, then you need to intentionally and precisely model it with that shape. The problem with using deform and flex geometry for actual design data is that they both create fairly approximate geometry, and this process yields a result that is not completely intentional. The shape that you finally achieve is the result of arbitrary uncontrolled function of the feature, not necessarily creating a shape that you had clearly envisioned beforehand.■

Deform has three types:

- **Point.** This type deforms a portion of the model by pushing a point and the geometry around it.

- **Curve to curve.** The most precise and useful deform type, this type selects an existing edge and forces the edge to match a curve.

- **Surface push.** This type of deform, while conceptually a very interesting function, is nearly unusable in practice. The part is deformed into a shape vaguely resembling an intermediate shape between the existing state of the part and a "tool" body.
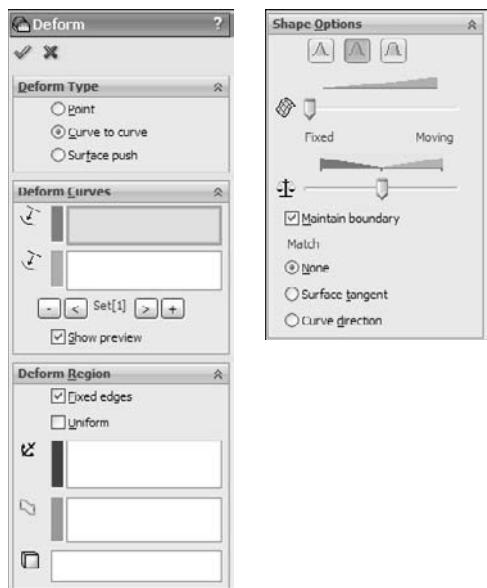
Figure 7.61 shows the PropertyManager interface for the Deform feature. The interface is different for each of the three main types, and also changes, depending on selections within the individual types. The interface shown is for the Curve to curve type because I believe this to be the most useful type.

Point deform

The Point deform option enables you to push a point on the model, and the model deforms as if it were rubber. Figure 7.62 shows the PropertyManager, as well as a before-and-after example of the Point deform function. The key to using this feature is to ensure that the Deform region option is unselected. Aside from that, you just have to use trial and error when applying the Point deform option. The depth, diameter, and shape of the deformation are not very precise. Also, you cannot specify the precise location for the point to be deformed. Again, this is best used for "looks-like" models, not production data.
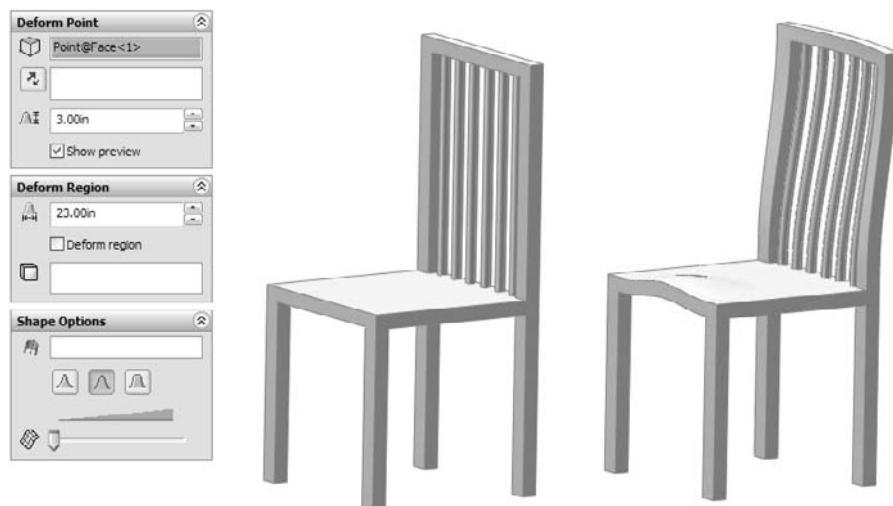
FIGURE 7.61

The PropertyManager interface for the Curve to curve deform

FIGURE 7.62

The Deform Point PropertyManager and a before-and-after example

In the model from Figure 7.62, two Point deform features are used, one to apply some shape to the back and one to apply some shape to the seat.
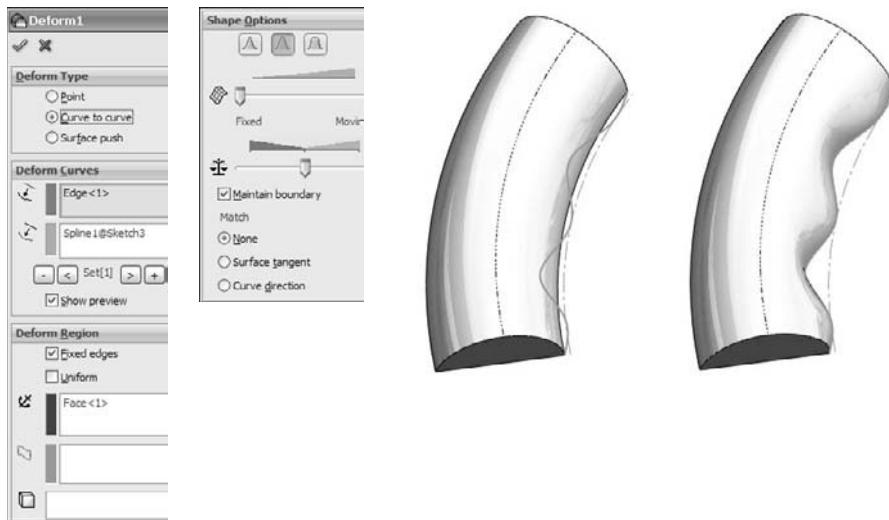
Curve to curve deform

Because the Curve to curve uses curve (or sketch or edge) data, it is a more precise method than the other deform types. The main concept here is to transform a curve on the original model to a new curve, thus deforming the body to achieve the new geometry.

The model shown in Figure 7.63 has been created using the Curve to curve deform. The part starts as a simple sweep (sweep an arc along an arc), and then a split line is created to limit the deform to a specific area of the model. The model is on the CD-ROM with the filename `Chapter 7 Deform Curve to Curve.sldprt`.

FIGURE 7.63

Using the Curve to curve deform option

Surface push deform

I do not go into much detail on the Surface push deform type because it is not one of the more useful functions in SolidWorks. In order to use it, you must have the body of the part that you are modeling, and a tool body that you will use to shape the part that you are modeling. The finished shape does not fit the tool body directly, but looks about half-way between the model and the tool body, blended together in an abstract sort of way. It looks like the dent that would result from an object being thrown very hard at a car fender, in that neither the thrown part nor the fender is immediately recognizable from the result.
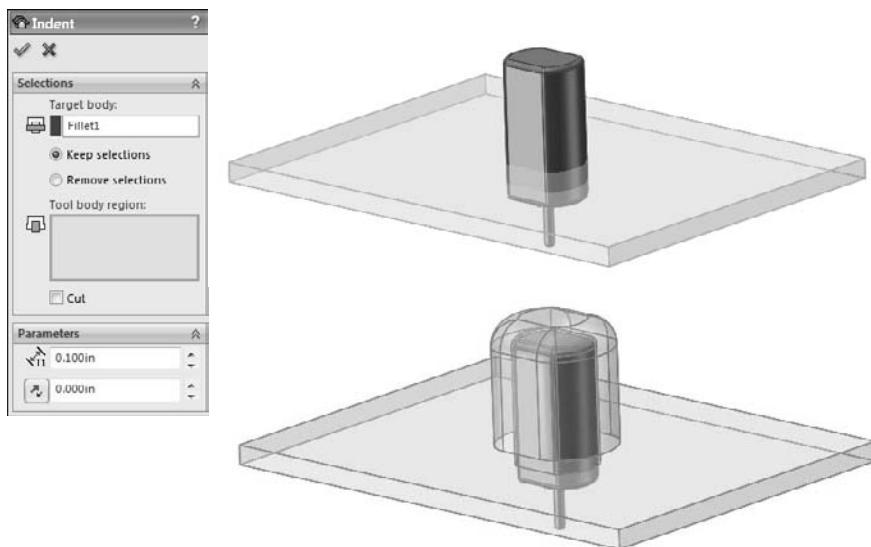
Indent feature

The Indent feature is what the Surface push deform is trying to be, or should try to be. Indent uses the same ingredients as the Surface push, but it produces a result that is both intelligible and useful. For example, if you are building a plastic housing around a small electric motor, then the Indent feature shapes the housing and creates a gap between the housing and the motor. Figure 7.64 shows the PropertyManager interface for the Indent feature, as well as what the indent looks like before and after using the feature.

Using the Indent feature

In this case, the small motor is placed where it needs to be, but there is a wall in the way. Indent is used to create an indentation in the wall by using the same wall thickness and placing a gap of .010 inches around the motor. The motor is brought into the wall part using the Insert ⇨ Part command. This is a multi-body technique. Multi-bodies are examined in detail in Chapter 26.

Tutorial: Bracket Casting

When you follow this tutorial, you are encouraged to follow the directions the first time to make sure that you understand the concepts involved, and then to go through it again, this time deviating from the instructions to see if you can expand your understanding by experimentation. To try bracket casting, follow these steps:
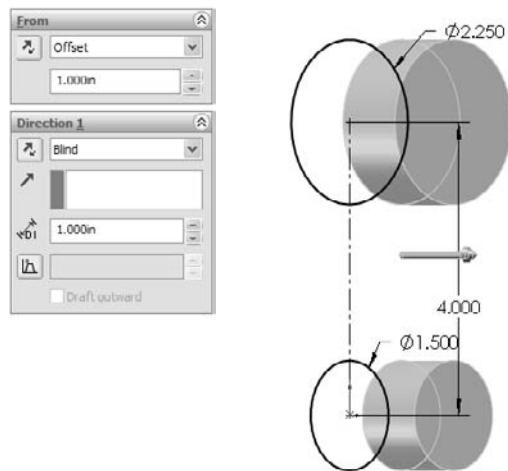
1. Open a new part using an inch-based template.

2. On the Right plane, draw a circle centered at the origin with a diameter of 1.50 inches, and a second circle placed 4.000 inches vertically from the first, with a diameter of 2.250 inches.

3. Exit the sketch, and make sure Instant 3D is selected. The Instant 3D icon is on the Features toolbar, and looks like a ruler with an arrow. Click on the sketch in the graphics window, and pull the Instant 3D arrow to create a solid. Edit the feature (right- or left-click the feature either in the graphics window or in the FeatureManager and click the Edit Feature icon, which is the yellow and green block with a hand pointing to it). Now enter numbers by hand so that you extrude the sketch 1.000 inch using a From condition of Offset by 1.000 inch, such that the offset and the extrude depth are in the same direction. Rename this feature **Bosses in the FeatureManager**. Figure 7.65 shows the results of these steps.

FIGURE 7.65

The results of Steps 1 to 3
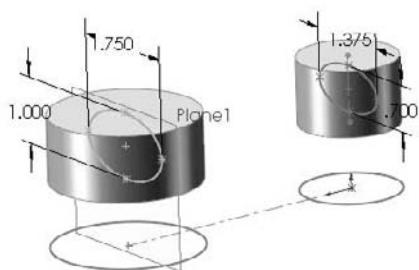
Note

These steps produce multiple bodies that will be merged in a later step. Multi-body parts are covered in more detail in Chapter 26. You can tell that there are multiple bodies by looking at the Solid Bodies folder near the top of the tree and expanding the folder. The bodies are listed in the folder.■

4. **On the Top plane, open a new sketch and draw a horizontal construction line across the cylinder, from the midpoint of one side to the midpoint of the other side.** To pick up the automatic relations for the midpoints more easily, it is recommended that you orient the view, normal to the sketch, or use the Top view. It does not matter if you make the relations to the top or bottom cylinder, because the midpoints of the sides are in the same place when they are projected into the sketch plane.

5. **Next, draw an ellipse (Tools ⇨ Sketch Tools ⇨ Ellipse) centered at the midpoint of the construction line and that measures .700 inches horizontally and 1.375 inches vertically.** You may want to assign a relation between the center of the ellipse and one of the control points to prevent the ellipse from rotating and fully define the sketch Exit the sketch when you are satisfied with the result.

6. **Show the sketch for the Bosses feature.** (Click the plus icon next to the Bosses extrude to show the sketch, and then right-click the sketch and select Show.)

7. **Create a plane parallel to the Top plane at the center of the larger circle.** You can access the Plane creation interface by choosing Insert ⇨ Reference Geometry ⇨ Plane. If you pre-select the Top plane from the flyout FeatureManager and the center of the larger sketch circle from the graphics window, the interface automatically creates the correct plane. Click OK to create the plane. Rename this plane **Top Boss Plane.**

8. **Draw a second ellipse on the Top Boss Plane.** Do not draw a construction line as you did for the first ellipse; instead, you can just make the centerpoint of the second ellipse directly on top of the first ellipse's centerpoint. The dimensions should be 1.000 inch horizontal by 1.750 inches vertical. Figure 7.66 shows the results up to this point.

FIGURE 7.66

The results up to Step 8

305

Tip

When you are sketching on parallel planes that are separated by some distance and trying to pick up automatic relations, it is often very helpful to be looking "normal to" the sketch, so that you can see how other entities are projected into the sketch plane.■

9. **Use the Loft feature to loft between the two ellipses.** Be sure to select the ellipses in approximately the same location so that they do not twist. If the loft preview accidentally twists, then use the connectors (light-blue square dots on the sketches that are connected by a straight line) to straighten out the loft.

Note

Notice that this feature joined together the other two disjoint bodies with the body that was created by the loft into a single body. This is a result of selecting the Merge Result option in the Options panel.■

Tip

If you want to experiment, expand the Start/End Constraints panel and apply end conditions for the loft. This causes the loft to change from a straight loft to a curved loft.■

10. **Right-click all sketches that are showing, and select Hide.** Do the same for the Top Boss Plane. This cleans up the display to prevent it from becoming confusing. However, if you prefer to see the sketches, then you can leave them displayed.
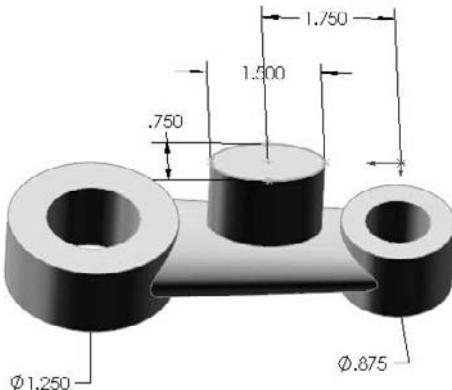
Tip

You can either hide or show different types of entities in groups by using the View menu. Hide All Types hides everything, and disables the options for individual entity types to be used.■

11. **Open a sketch on the Right plane.** Sketch an ellipse such that the center is oriented 1.750 inches vertically from the Origin, and the ellipse measures .750 inches horizontally and 1.500 inches vertically.

12. **Extrude this ellipse using the Up To Next end condition.** If Up To Next does not appear in the list, then change the direction of the extrude and try it again.

13. **Show the sketch of the Bosses feature by expanding the feature (click the "+" next to it), right- or left-clicking the sketch icon, and clicking the Hide/Show icon (eyeglasses).** Next, open a sketch on the Right plane. Sketch two circles that are concentric with the original circles, with the dimensions of .875 inches and 1.250 inches. Exit the sketch.

14. **Use Instant 3D to create an extruded cut that goes through the large circular bosses.** This feature will look like a boss extrusion at first, so when you have finished dragging its depth, a small toolbar with two icons appears. One of the icons enables you to add draft; the other enables you to turn the boss into a cut. Figure 7.67 shows the state of the model up to this step.

The results up to Step 14

15. **Start a fillet feature, and select the face of the Loft feature.** Assign a radius of .200 inches.

Note

Although this fillet is created by selecting a face, it is not a face fillet. Selecting a face for a regular constant radius fillet simply fillets any edge that is on the face.∎
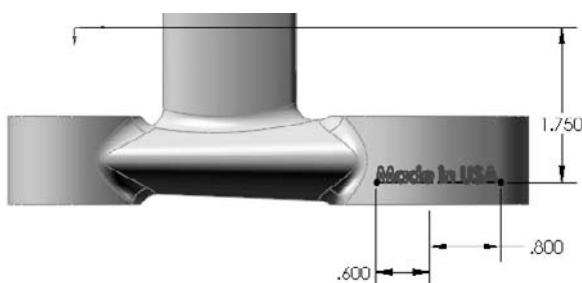
16. **Create a mirror feature, using the Right plane as the mirror plane.** In the Mirror PropertyManager, expand the Bodies To Mirror panel, and select anywhere on the part. Make sure that the Merge Solids option is selected. Click OK to accept the mirror.

17. **Orient the view to the Front view, and then turn the view on its side (hold down Alt and press the left- or right-arrow key six times).**

18. **Open a new sketch on the Front plane.** From the View menu, make sure that Hide All Types is not selected, and show Temporary Axes. Draw and dimension a horizontal construction line, as shown in Figure 7.68.

19. **With the construction line selected, start the Sketch Text command (Tools⇨Sketch Entities⇨Text).** Make sure that the line appears in the Curves selection box.

20. **Click in the text box, and type** Made in USA **(or your name or company name).** Select the text and click the Bold button. Deselect the Use Document Font option, change the font to use units, and set the height to .175 inches.

21. **Click OK to exit the Sketch Text PropertyManager, and click OK again to exit the sketch.** You can turn off the Temporary Axis display.

FIGURE 7.68
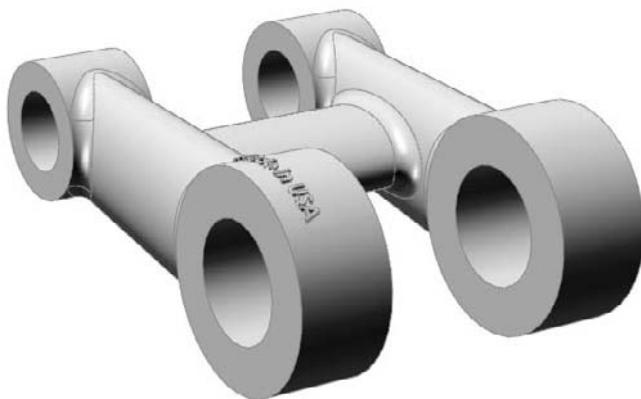
The results up to Step 18

22. **Choose Insert ⇨ Features ⇨ Wrap.** You should be prompted to select a plane or a sketch. Use the Flyout FeatureManager to select the sketch that you just created with the sketch text in it. Next, select the cylindrical face of the boss to see a preview of the text wrapped onto the face. If the text appears backward, then select the Reverse Direction option in the Wrap PropertyManager.

23. **Select the Emboss option, and assign a thickness of .025 inches.** Click in the Pull Direction selection box and select the Front plane. Click OK to accept the feature.

24. **Save the part and close it.** If you would like to examine the reference part, you can find it on the CD-ROM with the filename `Chapter 7 Tutorial Bracket Casting.sldprt`. The finished part is shown in Figure 7.69.

FIGURE 7.69

The finished part
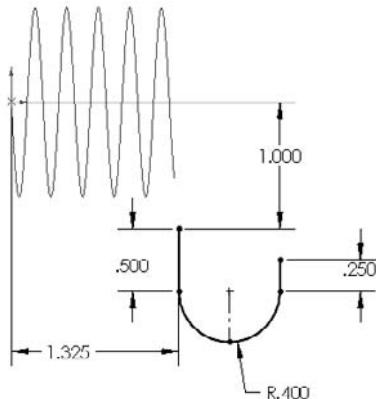
Tutorial: Creating a Wire-Formed Part

Follow these steps to create a wire-formed part:

1. **Open a new part using an inch-based template.**

2. **Open a sketch on the Right plane and sketch a circle that is centered on the Origin with a diameter of 1.500 inches.**

3. **Create a Helix, Constant Pitch, Pitch, and Revolution, where the Pitch = .250 inches, Revolutions = 5.15, and Start Angle = 0.** The Helix command is found at Insert ⇨ Curve ⇨ Helix/Spiral.

4. **Create a sketch on the Front plane, as shown in Figure 7.70.** Pay careful attention when adding the construction line, as shown. This line is used in the next step to reference the end of the arc.

FIGURE 7.70

The results up to Step 4
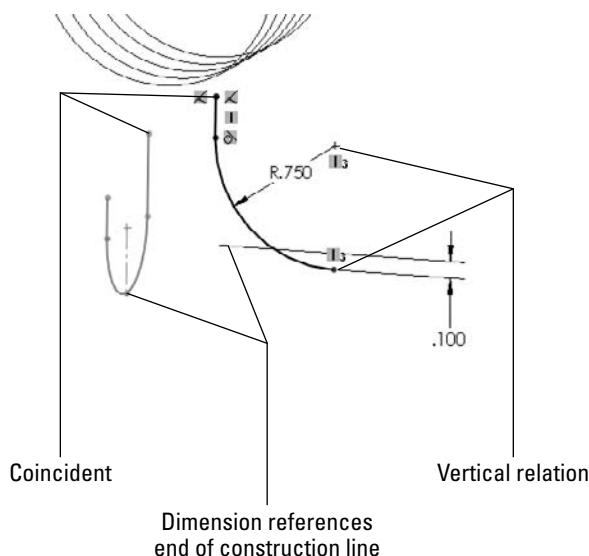
5. **Open a sketch on the Right plane and use Figure 7.71 to add the correct relations and dimensions.** Be aware that the two sketches shown are on different sketch planes, which makes it difficult to depict in 2D. You can also open the part from the CD-ROM for reference.

6. **Exit the sketch and create a projected curve.** The Projected Curve function is found at Insert ⇨ Curve ⇨ Projected Curve. Use the Sketch on Sketch option.

7. **Open a 3D sketch.** You can access a 3D sketch from the Insert menu. Select the helix and click Convert Entities on the Sketch toolbar. Then select the projected curve and click Convert Entities again. You now have two sections of a 3D sketch that are unconnected in space.

The sketch for Step 5

Coincident

Vertical relation

Dimension references
end of construction line
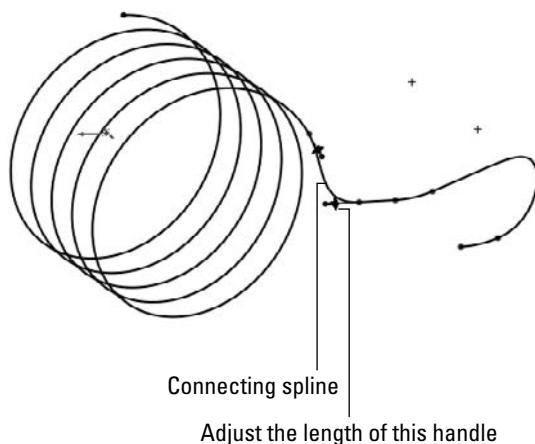
8. **Draw a two-point spline to join the ends of the 3D sketch entities that are closest to one another.** Assign tangent relations to the ends to make the transition smooth. Figure 7.72 illustrates what the model should look like at this point.

The results up to Step 8

Connecting spline

Adjust the length of this handle

Tip
You may have to adjust the length of one of the spline tangency length arrows to keep the spline from remaining inside the cylinder of the helix.

9. **Open a sketch on the Right plane, and draw an arc that is centered on the Origin and coincident with the end of the 3D sketch helix.** The 185-degree angle is created by activating the dimension tool and clicking first the center of the arc, and then the two endpoints of the arc. Now place the dimension. This type of dimensioning allows you to get an angle dimension without dimensioning to angled lines. Exit the sketch.

10. **Create a Composite Curve (Insert ⇨ Curve ⇨ Composite) consisting of the 3D sketch and the new 2D sketch.**

11. **Create a new plane using the Normal to Curve option, selecting one end of the composite curve.**

12. **On the new plane, draw a circle that is centered on the end of the curve with a diameter of .120 inches.** You need to create a Pierce relation between the center of the circle and the composite curve.

13. **Create a sweep feature using the circle as the profile and the composite curve as the path.** To create the sweep, you must first exit the sketch.

14. **Hide any curves that still display.**

15. **Choose Insert ⇨ Cut ⇨ With Surface.** From the Flyout FeatureManager, select the Right plane. Make sure that the arrow is pointing to the side of the plane with the least amount of material. Click OK to accept the cut. The finished part is shown in Figure 7.73.

FIGURE 7.73

The finished part

Summary

SolidWorks has a wide range of feature types to choose from, spanning from simple extrudes and revolves to more complex lofts and sweeps. It also offers a selection of specialty features that may not be useful on a day-to-day basis, but that have their place in the modeling techniques that you need to know to get the job done.

Some features, such as extrude, fillet, and flex, have so many options that it may be difficult to take them all in at once. You should browse through the models on the CD-ROM for this chapter and use the Rollback bar (described in detail in Chapter 11) to examine how the parts were built. You can then try to create a few on your own.

Patterning and Mirroring

P atterning and mirroring in SolidWorks are great tools to help you improve your efficiency. SolidWorks software provides many powerful pattern types that also help you accomplish design tasks. In addition to the different types of patterns, there are options that enable functionality that you may not have considered. A solid understanding of patterning and mirroring tools is necessary to be able to build the maximum amount of parametric intelligence into your models.

Patterning in a Sketch

You can use both pattern and mirror functions in sketch mode, although sketch patterns are not a preferred choice. The distinction between patterning and mirroring in sketch mode is important when it comes to sketch performance.

Performance

Although there are many metrics for how software performs, in SolidWorks, the word performance means the same thing as speed. Sketch patterns have a very adverse effect on speed, and do not offer the same level of control as feature patterns. ∎

You might hear a lot of conflicting information about which features are better to use in different situations. Users coming from a 2D background often use functions such as sketch patterning because it's familiar, without questioning whether there is a better approach. When in doubt, you can perform a test to determine which features work best for a given situation.

In this test, I made a series of 20-by-20 patterns using circles, squares, and hexagons. The patterns are both sketch patterns and feature patterns, and I created them with both Verification on Rebuild and Geometry Pattern turned on and off. Verification on Rebuild is an error checking setting that you can access through Tools➪Options➪Performance, and Geometry Pattern is a setting that is applicable only to feature patterns.

Table 8.1 shows the rebuild times (in seconds) of solid geometry created from various types of patterns as measured by Feature Statistics (found at Tools➪Feature Statistics). Sketch patterns are far slower than feature patterns, by a factor of about ten. The biggest speed reduction occurs when you use sketch patterns in conjunction with the Verification on Rebuild setting, especially as the number of sketch entities being patterned increases.

Generally, the number of faces and sketch relations being patterned has a significant effect on the speed of the pattern. The sketch pattern times are taken for the entire finished model, including the sketch pattern and a single extrude feature, using the sketch with the pattern to do an extruded cut. The sample parts are on the CD-ROM for reference. Look for the filenames beginning with Reference1 through Reference7.

TABLE 8.1

Pattern Rebuild Times

Pattern Type	Default	Geometry Pattern	Verification on Rebuild
20 × 20 sketch circle	.87	n/a	5.52
20 × 20 sketch square	4.5	n/a	60
20 × 20 sketch hex	8.6	n/a	126
20 × 20 feature circle	.06	.53	.08
20 × 20 feature square	.23	.53	.23
20 × 20 feature hex	.36	.55	.36

The most shocking data here is the difference between a sketch pattern of a hex when a patterned sketch cuts into a flat plate compared to a feature pattern of a single extruded hex with each using the Verification on Rebuild option — 0.36 seconds compared to 126 seconds.

Always keep this information about sketch patterns in mind:
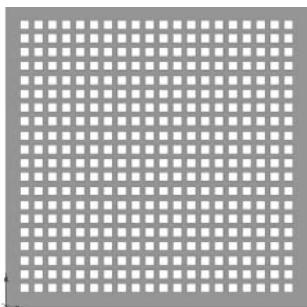
- Sketch patterns are bad for rebuild speed.
- The more faces created by a pattern, the longer it takes to rebuild.
- The more sketch relations a sketch pattern has, the longer it takes to rebuild.
- Geometry pattern does not improve rebuild speed (unless a special end condition like Up to Surface has been used)

- Verification on Rebuild dramatically increases rebuild time with the number of faces, but is far less affected by feature patterns than extruded sketch patterns.

Figure 8.1 shows one of the parts used for this simple test.

A pattern part used for the test

One interesting result of this test was that if a patterned extruded feature creates a situation where the end faces of the extruded features have to merge into a single face, the feature could take ten times the amount of time to rebuild as a pattern with unmerged end faces. This was an inadvertent discovery. I'm sure you would make your own discoveries if you were to investigate rebuild speeds for end conditions for cuts such as Through All, Up to Face, Up to Next, and so on, as well as the difference between cuts and boss features. Further, using Instant 3D can be an impediment when you're editing very large sketches simply due to the effects of the preview.

Debunking more sketch myths

People often say that it is best practice to define your sketches fully. I completely agree with this statement. However, I have heard people go to the extent to say that fully defined sketches solve faster, with the rationale being that SolidWorks has to figure out how to solve the underdefined sketch, but the fully defined sketch is already spelled out. Let's find out.
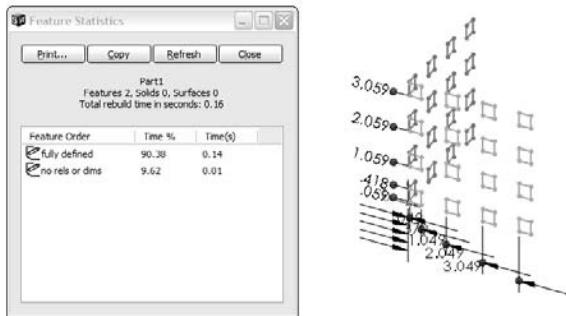
In this example, I created a sketch pattern of 4 × 4 rectangles and used the Fully Define Sketch tool to add dimensions. Then I copied and pasted the sketch and removed all the dimensions and relations. Figure 8.2 shows the Feature Statistic results.

It is safe to say that fully defined sketches are best practice, but it is not due to rebuild speed. Sketch relations are costly from a rebuild time point of view. Patterning sketch relations are even more costly. The rebuild time does not even come close to the time that it takes the Fully Define Sketch tool to create all the dimensions and relations in the first place. This combination of geometry, software, and hardware took about 30 seconds of CPU time to add the relations and dimensions.

Comparing rebuild times of a fully defined sketch to a completely undefined sketch

Patterning a sketch

It is best to pre-select the sketch entities that you want to pattern before using the Sketch Pattern tool. If you do not pre-select, then after the PropertyManager is open, you can only select entities to pattern one by one, because the window select is not available for this function.

Tip

When creating a linear sketch pattern, be sure to select the Add Spacing Dimension check boxes. If these dimensions are not added, then editing the pattern becomes more difficult. ∎

Linear Sketch Pattern

The Linear Pattern PropertyManager is shown in Figure 8.3.

Unlike other PropertyManagers, the selected entities for the sketch pattern functions are found at the bottom of the PropertyManager instead of at the top. This is a little confusing. Sketch tool PropertyManagers that are new for the 2010 version, such as Convert Entities and Mirror, place the selection box at the top.

The Direction 1 panel works predictably by establishing the direction and spacing, and then the number. The Angle setting enables you to specify a direction that does not rely on anything outside of the sketch.

The Direction 2 panel works a little differently. You must first specify how many instances you want, and then the other information becomes available. The spacing is grayed out until you tell it you want more than one instance in Direction 2.

FIGURE 8.3

The Linear Pattern PropertyManager
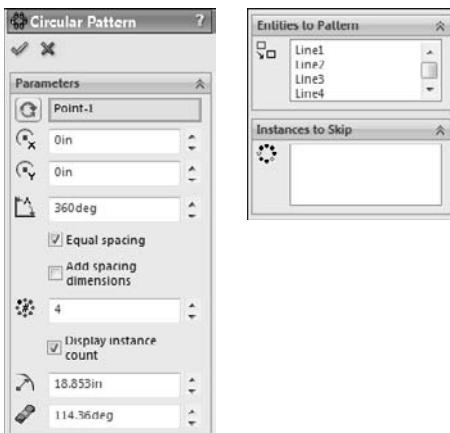
Circular Sketch Pattern

The Circular Sketch Pattern defaults to the sketch Origin as the center of the pattern. You can move and position this point using the numbers in the PropertyManager, but you cannot dimension it until after the pattern is created. Again, this is another feature where you need to pre-select because window selection is not available (patterned sketch entities must be selected one by one to go into the Entities to Pattern panel). Figure 8.4 shows the Circular Pattern PropertyManager.

FIGURE 8.4

The Circular Pattern PropertyManager

Mirroring in a Sketch

Mirroring in a sketch is a completely different matter from patterning in a sketch. It offers superior performance, and the interface is better developed. Mirrored entities in a sketch are an instrumental part of establishing design intent.

Two methods of mirroring items in a sketch are discussed here, along with a method to make entities work as if they have been mirrored when in fact they were manually drawn.

Mirror Entities

Mirror Entities works by selecting the entities that you want to mirror along with a single centerline, and clicking the Mirror Entities button on the Sketch toolbar. You can use this simple and effective tool on existing geometry. This method is the fastest way to use the tool but there are other methods. You can pre-select or post-select, using a dialog box to select the mirror line, which does not need to be a centerline.

One feature of Mirror Entities may sometimes cause unexpected results. For example, in some situations, Mirror Entities will mirror a line or an arc and merge the new element with the old one across the centerline. This happens in situations where the mirror and the original form a single line or a single arc. SolidWorks may delete certain relations and dimensions in these situations.

Dynamic Mirror

As the name suggests, Dynamic Mirror mirrors sketch entities as they are created. You can activate it by selecting a centerline and clicking the Dynamic Mirror button on the Sketch toolbar. Dynamic Mirror is not on the toolbar by default; you need to choose Tools ➪ Customize ➪ Commands to add it to the toolbar. You can also access Dynamic Mirror by choosing Tools ➪ Sketch Tools ➪ Dynamic Mirror from the menu.

When you activate this function, the centerline displays with hatch marks on the ends and remains active until you turn off or exit the sketch. Figure 8.5 shows the centerline with hatch marks.

Symmetry sketch relation
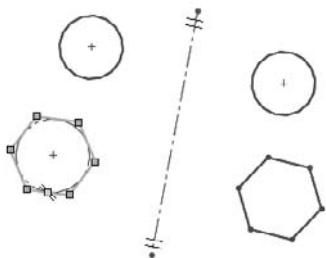
I have covered the Symmetry sketch relation in previous chapters on sketching, but I mention it here because it offers you a manual way to mirror sketch entities. There are editing situations when you may not want to create new geometry, but instead use existing entities with new relations driving them. To create the Symmetry sketch relation, you must have two similar items (such as lines or endpoints) and a centerline selected.

Mirroring in 3D sketches

Chapter 31 deals with 3D sketches in more detail, but I discuss the mirror functionality here to connect it with the rest of the mirroring and patterning topics. 3D sketches can contain planes and if you are sketching on a plane in a 3D sketch, you can mirror items on it. You cannot mirror general 3D sketch entities.

FIGURE 8.5

The Dynamic Mirror centerline with hatch marks

Sketch patterns are also unavailable in the 3D sketch, but you can use the Move, Rotate, and Copy sketch tools on planes in 3D sketches. Combining one questionable functionality (3D sketches) with another (sketch patterns) does not usually improve either one.

Exploring the Geometry Pattern Option

The SolidWorks Help file says that the Geometry Pattern option in feature patterns results in a faster pattern because it does not pattern the parametric relations. This claim is valid only when there is an end condition on the patterned feature such that the feature will actually pattern the end condition's parametric behavior. The part shown in Figure 8.6 falls into this category. The improved rebuild time goes from .30 to .11 seconds. Although a 60 percent reduction is significant, the most compelling argument for the use of the Geometry Pattern has nothing to do with rebuild time. It is to avoid the effect of patterning the end-condition parametrics.

In fact, the Geometry Pattern option's main intent is to pattern existing geometry without the parametric intelligence. The main mission of Geometry Pattern has nothing to do with rebuild speed.

Under some conditions, Geometry Pattern will not work. One example is any time a patterned face merges with a face that cannot be patterned. Figure 8.7 shows two patterns, one that can use Geometry Pattern and one that cannot.
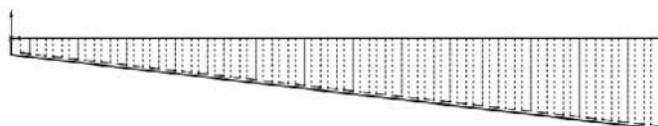
The pattern of the rectangular feature cannot use Geometry Pattern because the face that is merged is not merged in all pattern instances. The circular pattern can use Geometry Pattern because the flat face is merged in every pattern instance. The circular pattern also allows Geometry Pattern for the same reason.
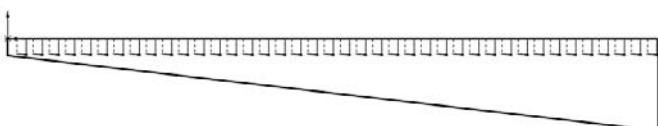
FIGURE 8.6

A geometry pattern test
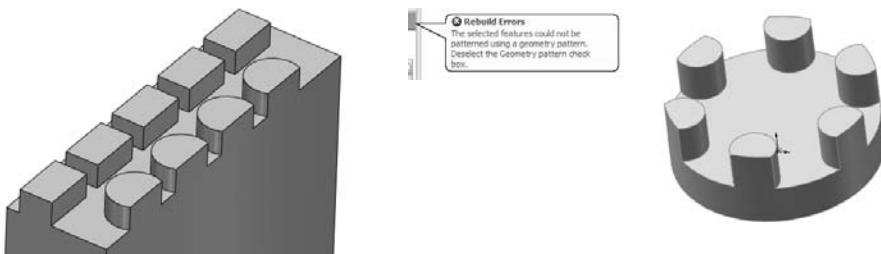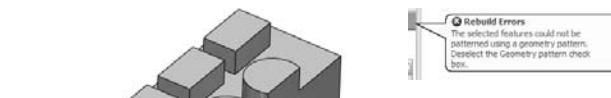
Geometry pattern off — Parametrics are patterned

Geometry pattern on — Parametrics not patterned

FIGURE 8.7

Merged faces

In some situations, SolidWorks error messages may send you in a loop. One message may tell you that the pattern cannot be created with the Geometry Pattern turned on, so you should try to turn it off. When you do that, you may get another message that says the pattern will not work, and that you should try to use the Geometry Pattern setting. In cases like this, you may try to use a different end condition, or change the selection of features patterned along with the feature, such as fillets. You may also try to pattern bodies or even faces rather than features.

Patterning Bodies

I cover multi-bodies in depth in Chapter 26, but need to deal with the topic here briefly. Any discussion of patterning is not complete without a discussion of bodies because using bodies is an available option with all the pattern and mirror types.

SolidWorks parts can contain multiple *solid* or *surface* bodies. A solid body is a solid that comprises a single contiguous volume. Surface bodies are different, but they can also be patterned and mirrored as bodies.

There are both advantages and disadvantages to mirroring and patterning bodies instead of features. The advantages can include the simplicity of selecting a single body for mirroring or patterning. In cases where the geometry to be patterned is complex or there is a large number of features, patterning bodies also can be much faster. However, in the example used earlier with patterning features in a 20-by-20 grid of holes, when done by patterning a single body of 1" × 1" × .5" with a .5" diameter hole, patterning bodies gives a rebuild time of about 60 seconds with or without Verification on Rebuild. The function combines the resulting bodies into a single body that takes most of the time. This says that for large patterns of simple features, patterning bodies is *not* an efficient technique. Although I do not have an experiment in this chapter to prove it, it seems intuitive that creating a pattern of a smaller number of complex bodies using a large number of features in the patterned body would show a performance improvement over patterning the features.

Another disadvantage of patterning or mirroring bodies is that it does not allow you to be selective. You cannot mirror the body minus a couple of features without doing some shuffling of feature order in the FeatureManager. Another disadvantage is that if the base of the part has already been mirrored by a symmetrical sketch technique, then body mirroring is not going to help you mirror the subsequent features. In addition, the Merge Bodies option within the mirror feature does not work, as you would want it to. It merges only those bodies that are part of the mirror to bodies that are part of the mirror. Pattern Bodies does not even have an option to merge bodies. Both of these functions are often going to require an additional combine feature (for solid bodies) or knit (for surface bodies) to put the final results together.

Some of these details may seem obscure when you're reading about them, but when you begin to work patterning bodies and begin trying to merge them into a single body, read over this section again. The inconsistency between the Merge option existing in Mirror but not in Pattern is unexplainable, and a possible opportunity for an enhancement request.

Cross-Reference

Bodies are discussed in more detail in Chapter 26. Surface modeling is covered in Chapter 27. ■

Patterning Faces

Most of the pattern types have an option for Pattern Faces. This option has a few restrictions, the main limitation being that all instances of the pattern must be created within the boundaries of the same face as the original. Figure 8.8 shows an example of the Pattern Faces option working with a Circular Pattern feature. To get around this limitation, you can knit and pattern the surface body, as shown in Figure 8.9.
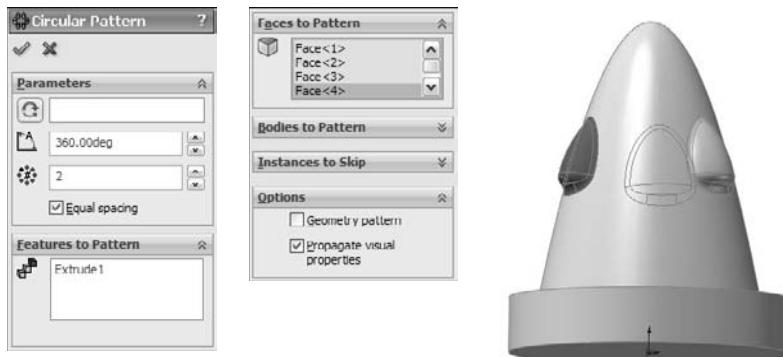
Patterning faces is another way of patterning geometry within SolidWorks without patterning the feature intelligence that was built into the original. It is also a way to make patterns on imported parts from existing geometry. Chapter 30 comes back to this topic briefly in the discussion on imported geometry and direct edit techniques.
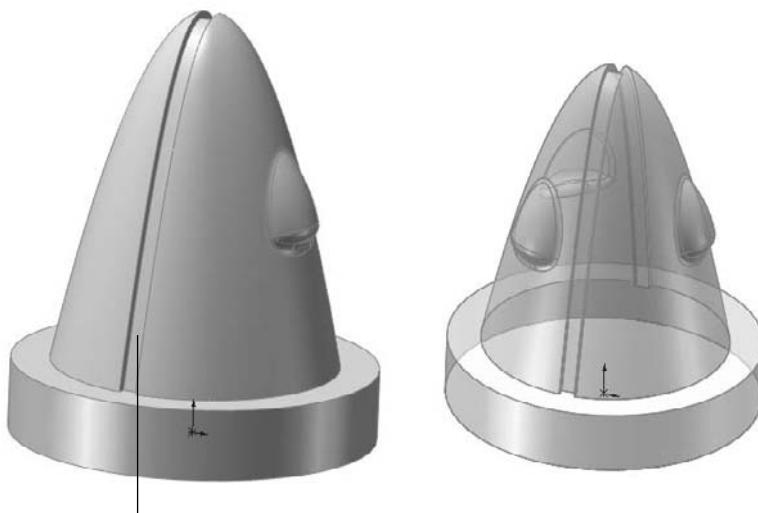
FIGURE 8.8

A circular pattern using the Pattern Faces option

FIGURE 8.9

Patterning a surface body

Split in face means faces
from feature on side cannot be
patterned all the way around

Cross-Reference
Working with surface bodies is covered in Chapter 27. ∎

Patterning Fillets

You may hear people argue that you cannot pattern fillets. This is only partially true. It is true that fillets as individual features cannot be patterned. For example, if you have a symmetrical box and a fillet on one edge and want to pattern only the fillet to other edges, this does not work. However, when fillets are patterned with their parent geometry, they are a perfectly acceptable candidate for patterning. This is also true for the more complex fillet types, such as variable radius and full radius fillets. You may need to use the Geometry Pattern option, and you may need to select all the fillets affecting a feature, but it certainly does work.

Understanding Pattern Types

Up to now, I have discussed patterns in general; differentiated sketch patterns from feature patterns, face patterns, and body patterns; and looked at some other factors that affect patterning and mirroring. I will now discuss each individual type of pattern to give you an idea of what options are available.

Linear Pattern

The Linear Pattern feature has several available options:
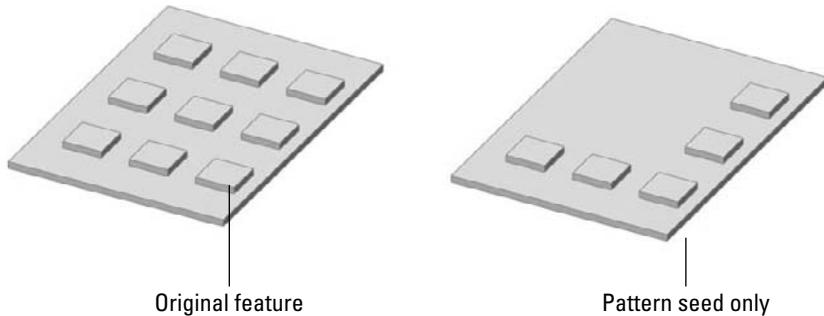
- **Single direction or two directions.** Directions can be established by edge, sketch entity, axis, or linear dimension. If two directions are used, the directions do not need to be perpendicular to one another.

- **Spacing.** The spacing represents the center-to-center distance between pattern instances, and can be driven by an equation.

- **Number of Instances.** This number represents the total number of features in a pattern, which includes the original seed feature. It can also be driven by an equation. Equations are covered in detail in Chapter 9.

- **Direction 2.** The second direction works just like the first, with the one exception of the Pattern Seed Only option. Figure 8.10 shows the difference between a default two-direction pattern and one using the Pattern Seed Only option.

- **Instances to Skip.** This option enables you to select instances that you would like to leave out of the final pattern. The pink dots are the instances that remain, and the red dots are the ones that have been removed. Figure 8.11 shows the interface for skipping instances. You may have difficulty distinguishing the red and pink colors on the screen.

- **Propagate visual properties.** This option patterns the color, texture, or cosmetic thread display, along with the feature to which it is attached.
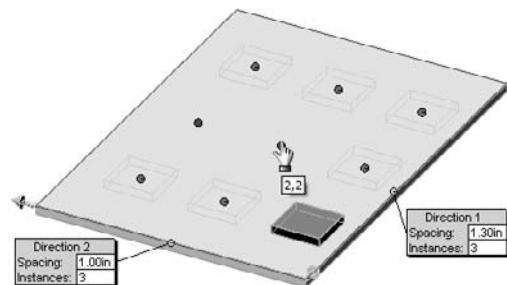
FIGURE 8.10

Using the default two-direction pattern and the Pattern Seed Only option

Original feature   Pattern seed only

FIGURE 8.11

Using the Instances to Skip option

- **Vary Sketch.** This option in patterns is often overlooked and not widely used or understood. While it may have a niche application, it is a powerful option that can save you a lot of time if you ever need to use it.
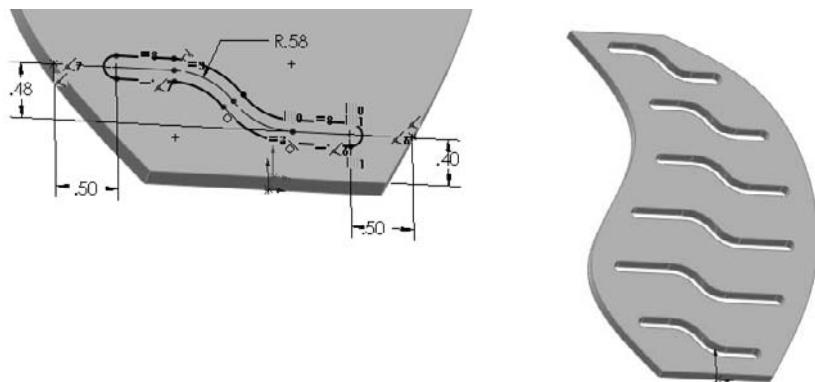
  Vary Sketch allows the sketch of the patterned feature to maintain its parametric relations in each instance of the pattern. It is analogous to Geometry Pattern. Where Geometry Pattern disables the parametric end condition for a feature, Vary Sketch enables the parametric sketch relations for a pattern.

  To activate the Vary Sketch option, the Linear Pattern must use a linear dimension for its Pattern Direction. The dimension must measure in the direction of the pattern, and adding the spacing for the pattern to the direction dimension must result in a valid feature.

  The sketch relations must hold for the entire length of the pattern. Figure 8.12 shows the sketch relations and the resulting pattern. The preview function for this feature does not work.

Using the Vary Sketch option

On the CD-ROM

To better understand how this feature works, open the sample file from the CD-ROM called `Chapter 8 Vary Sketch.sldprt,` **and edit Sketch2.** ∎

Edit the .40-inch dimension. Double-click it and use the scroll arrow to increase the dimension; watch the effect on the sketch. If a sketch reacts to changes properly, then it cannot be used with the Vary Sketch option. In this case, the .40-inch dimension is used as the direction. The direction dimension has to be able to drive the sketch in the same way that this one does. These dimensions cannot pass through the Zero value and cannot flip directions or move into negative values.

To make the sketch react this way to changes in the dimension, the slot was created using the bi-directional offset that was demonstrated in an earlier chapter, which means that the whole operation is being driven by the construction lines and arcs at the centerline of the slot. Sketch points along the model edges are kept at a certain distance from the ends of the slots using the .50-inch dimensions. The arcs are controlled by an Equal Radius relation and a single .58-inch radius dimension. The straight lines at the ends of the slots are controlled by an Equal Length relation.

This type of dimensioning and relation creation is really what parametric design is all about. The Vary Sketch option takes what is otherwise a static linear pattern and makes it react parametrically in a way that would otherwise require a lot of setup to create individual features. If you model everything with the level of care that you need to put into a Vary Sketch pattern feature sketch, then your models will react very well to change.

Circular Pattern

The Circular Pattern feature requires a circular edge or sketch, a cylindrical face, a revolved face, a straight edge, an axis, or a temporary axis to act as the Pattern Axis of the pattern. All the other options are the same as the Linear Pattern — except that the Circular Pattern does not have a Direction 2 option and the Equal Spacing option works differently.
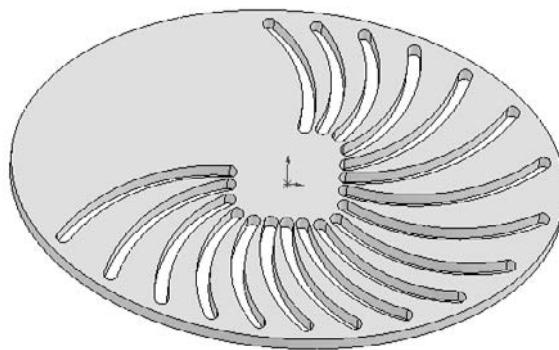
Equal Spacing takes the total angle and evenly divides the number of instances into that angle. The name *equal spacing* is a bit misleading because all Circular Patterns create equal spacing between the instances, but somehow everyone knows what they mean.

Without using the Equal Spacing option, the Angle setting represents the angular spacing between instances.

The Vary Sketch option is available in Circular Pattern as well. The principles for setup are the same, but you must select an angular dimension for the direction. The part shown in Figure 8.13 was created using this technique.

FIGURE 8.13

A Circular Pattern vary sketch

Curve Driven Pattern
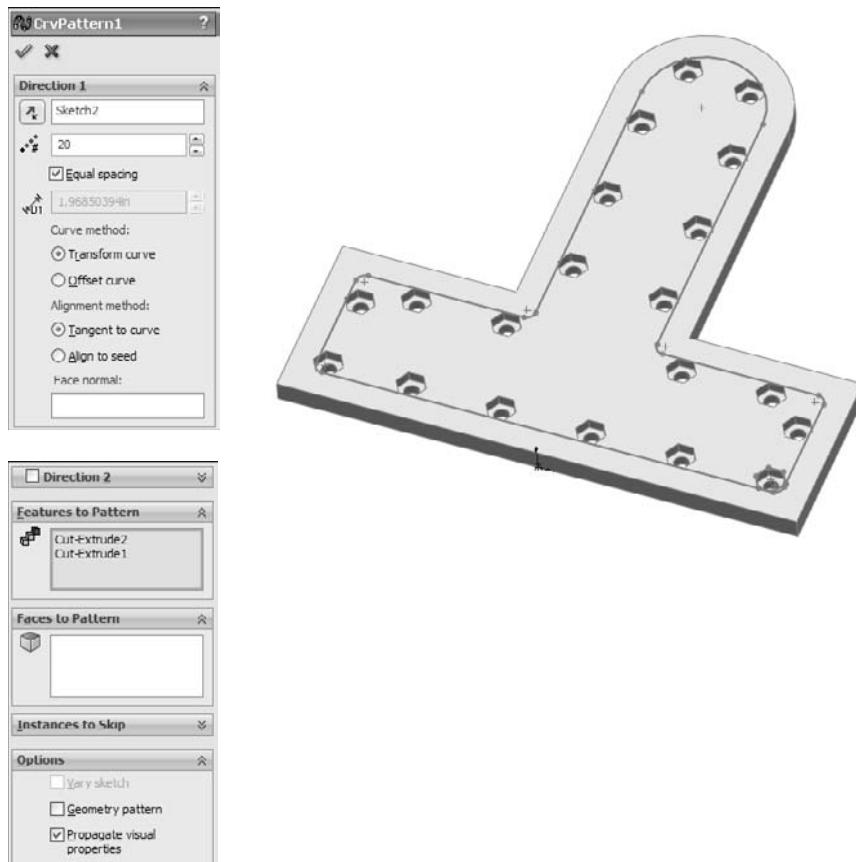
A Curve Driven Pattern does just what it sounds like: it drives a pattern along a curve. The curve could be a line, an arc, or a spline. It can be an edge, a 2D or 3D sketch, or even a real curve feature. An interesting thing about the Curve Driven Pattern is that it can have a Direction 2, and Direction 2 can be a curve. This pattern type is one of the most interesting, and has many options.

For an entire sketch to be used as a curve, the sketch must not have any sharp corners — all the entities must be tangent. This could mean using sketch fillets or a fit spline. The example shown in Figure 8.14 was created using sketch fillets. This pattern uses the Equal Spacing option, which spaces the number of instances evenly around the curve. It also uses the Offset Curve option, which maintains the patterned feature's relationship to the curve throughout the pattern, as if an offset of the curve goes through the centroids of each patterned instance. The Align to Seed option is also used, which keeps all the pattern instances aligned in the same direction.

Figure 8.15 shows the same part using the Transform Curve positioning option and Tangent to Curve alignment option.

FIGURE 8.14

The Curve Driven Pattern using sketch fillets
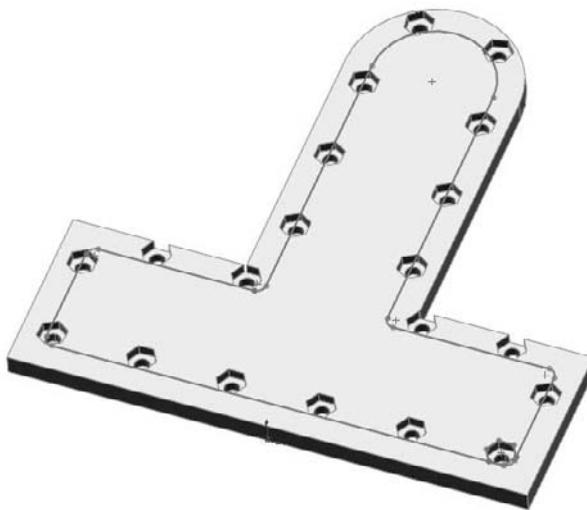
FIGURE 8.15

Using the Transform Curve and Tangent to Curve options

Instead of an offset of the curve going through the centroids of each patterned feature instance, in the Transform Curve, the entire curve is moved rather than offset. On this particular part, this causes a messy pattern. The Tangent to Curve option gives every patterned instance the same orientation relative to the curve as the original.

The Face Normal option is used for a 3D pattern, as shown in Figure 8.16. Although this functionality seems a little obscure, it is useful if you need a 3D curve-driven pattern on a complex surface. If you are curious about this example, it is on the CD-ROM with the filename `Reference 3d Curve Driven.sldprt`.

Using a Direction 2 for a Curve Driven Pattern creates a result similar to that in Figure 8.17. This is another situation that, although rare, is good to know about.
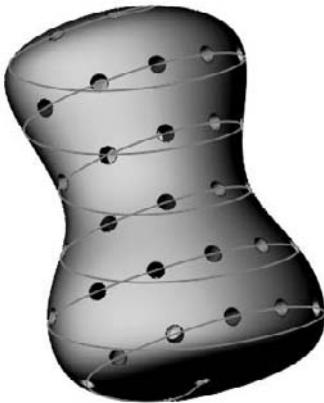
The rest of the Curve Driven Pattern works like the other pattern features that have already been demonstrated.

Sketch Driven Pattern

Sketch Drive Patterns use a set of sketch points to drive the locations of features. The Hole Wizard drives the locations of multiple holes using sketch points in a similar way. However, the Sketch Driven Pattern does not create a 3D pattern in the same way that the Hole Wizard does. Figure 8.18 shows a pattern of several features that has been patterned using a Sketch Driven Pattern. A reference point is not necessary for the first feature.
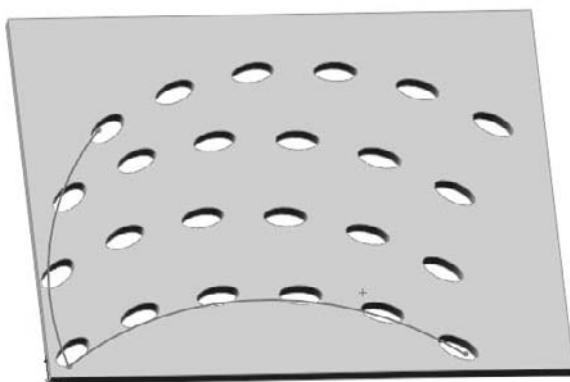
FIGURE 8.16

Using a 3D Curve Driven Pattern

FIGURE 8.17

Using Direction 2 with a Curve Driven Pattern
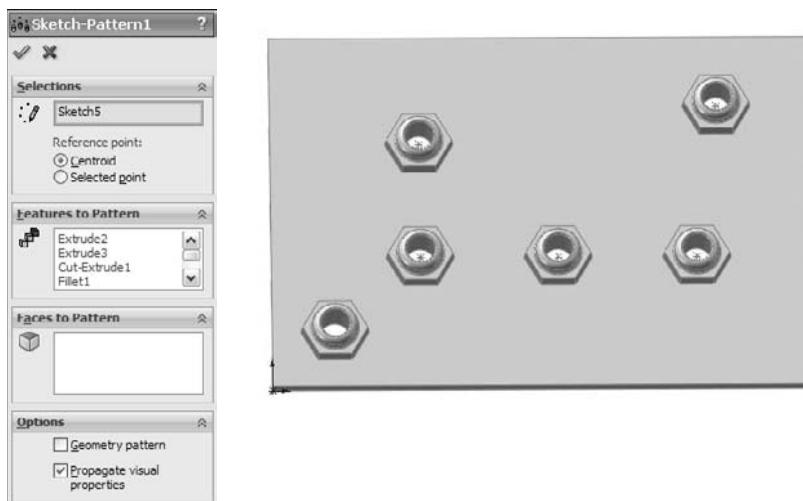
The Centroid option in the Reference Point section is fine for symmetrical and other easily definable shapes such as circles and rectangles, where you can find the centroid just by looking at it, but on more complex shapes, you may want to use the Selected Point option. The Selected Point option is shown in Figure 8.19.
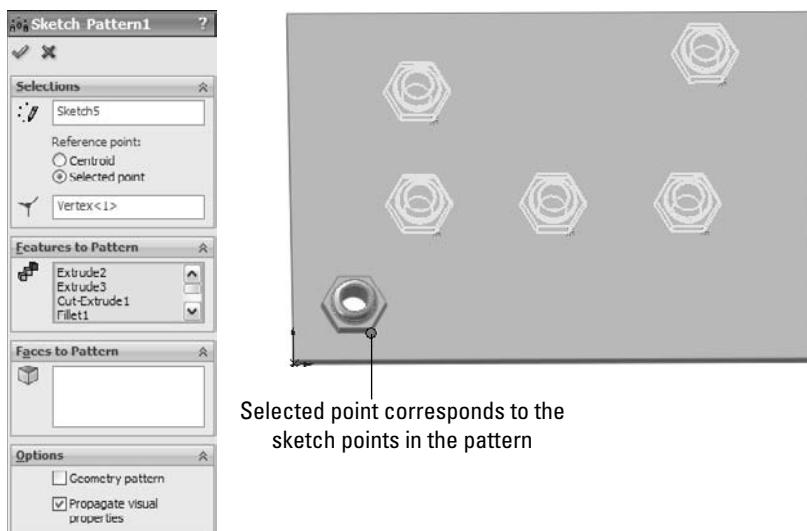
FIGURE 8.18

Using a Sketch Driven Pattern

FIGURE 8.19

Using the Selected Point option in a Sketch Driven Pattern

Selected point corresponds to the
sketch points in the pattern

Table Driven Pattern

A Table Driven Pattern drives a set of feature locations, most commonly holes, from a table. The table may be imported from any source with two columns of data (X and Y) that are separated by a space, tab, or comma. Extraneous data will cause the import to fail.
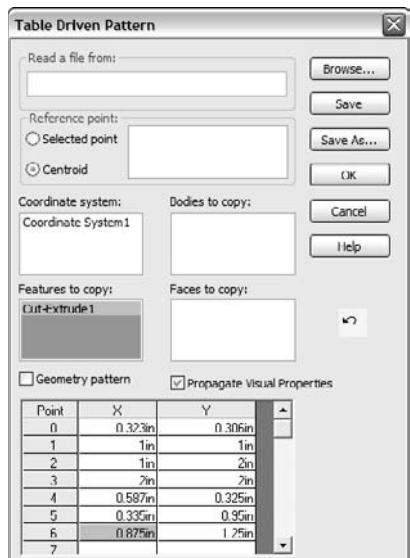
The X, Y Origin for the table is determined by a Coordinate System reference geometry feature. The XY plane of the Coordinate System is the plane to which the XY data in the table refers.

You can access the Coordinate System command by choosing Insert ➪ Reference Geometry ➪ Coordinate System from the menu. You can create the Coordinate System by selecting a combination of a vertex for the Origin and edges to align the axes. Like the Sketch Driven Pattern, this feature can use either the centroid or a selected point on the feature to act as the reference point.

The fact that this feature is still in a floating dialog box points to its relatively low usage and priority on the SolidWorks upgrade schedule. The interface for the feature is rather crude in comparison to some of the more high-usage features. This interface is shown in Figure 8.20.

FIGURE 8.20

The Table Driven Pattern dialog box
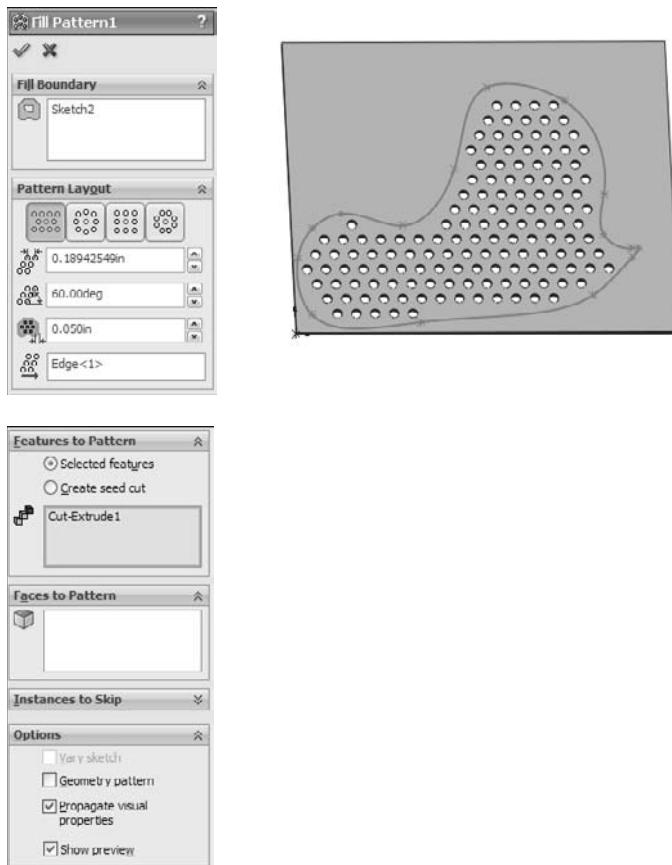
Fill Pattern

The Fill Pattern feature fills a face or area enclosed by a sketch with a pattern of a selected feature. The type of pattern used to fill the area is limited to one of four preset patterns that are commonly used in gratings and electronics ventilation in plastics and sheet metal. These patterns and other options for the Fill Pattern are shown in Figure 8.21.

Using the Fill Pattern feature

The Pattern Layout panel enables you to control spacing and other geometrical aspects of the selected pattern layout, as well as the minimum gap from the fill boundary. This is most useful for patterns of regularly spaced features with an irregular boundary.

Cosmetic Patterns

Cosmetic Patterns are not patterns in the same sense as all the other pattern types in SolidWorks. Cosmetic Patterns do not actually create any geometry, just the appearance of geometry. They are applied using RealView functionality, which may or may not be available to you depending on your hardware, in particular your video card.

Note
More information is available on RealView capable video cards from the SolidWorks corporate Web site, at `www.solidworks.com/pages/services/VideoCardTesting.html?lsrc=quick_links.` ∎

Cosmetic Patterns are appropriate if your manufacturing method does not require actual geometry. For example, rapid prototyping requires explicit geometry in order to build a part, but a perforated sheet metal panel or a knurled cylindrical handle may require only a note on a drawing for the shop to set up a manufacturing process to create the geometry.

To apply a Cosmetic Pattern to a face, feature, body, or entire part, click the RealView tab from the Task Pane, and choose Appearances ⇨ Miscellaneous ⇨ Pattern or ⇨ RealView Only Appearances. Drag and drop the desired pattern onto the model, and use the popup menu to apply it to a face, feature, body, or the entire part. Figure 8.22 shows the RealView tab of the Task Manager with some of the Cosmetic Pattern options.
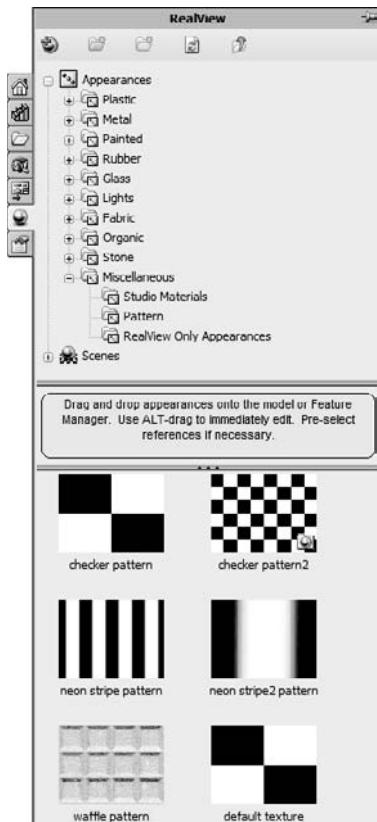
Cross-Reference
You can find more details about RealView appearances in Chapter 5. ∎

FIGURE 8.22

Cosmetic Pattern options in the RealView tab of the Task Manager

Mirroring in 3D

Because symmetry is an important aspect of modeling parts in SolidWorks, mirror functions are a commonly used feature. This is true whether you work on machine parts, sheet metal, injection-molded, cast, or forged parts. I discussed sketch-mirroring techniques earlier in this chapter, and now I will discuss 3D mirroring techniques.

Mirroring bodies
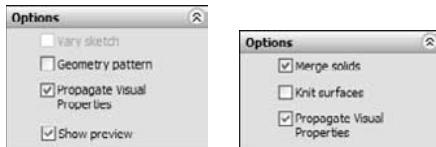
Earlier in this chapter, I discussed patterning bodies. I mentioned that the patterning and mirroring tools in SolidWorks do not have adequate functionality when it comes to body management. Neither tool allows the patterned or mirrored bodies to be merged with the main body if the main

body is not being patterned or mirrored. Figure 8.23 shows the Options panels for both the Linear Pattern (on the left) and the Mirror (on the right) features. Here you can see that the pattern function has no provision whatsoever for merging bodies. The Mirror appears to have the functionality, but it applies only to bodies that are used or created by the Mirror feature.

In future versions of SolidWorks, we hope these features will be outfitted with more complete merge and feature scope functionality, such as Extrude features.

FIGURE 8.23

Options panels from the Linear Pattern and Mirror PropertyManagers

Best Practice

Mirroring bodies is the fastest and simplest method when a part has complete symmetry. However, this may not be an option if the part is not completely symmetrical. In addition, the decision to mirror must often be made when you are creating the first feature. If the first feature is modeled as a sketch that is built symmetrically around the Origin, then you may need to cut the part in half to mirror it. This is an adequate modeling technique, although it is not as clean as it could be. ■

Mirroring features

Features can be mirrored across planes or flat faces used as the plane of symmetry. If you are mirroring many features, then it is best to mirror them all with a single mirror feature rather than to make several mirror features. You may have to do this by moving the mirror feature down the tree as you add new features. Depending on your part and what you are trying to accomplish, it may be better to mirror bodies than features, but you should not go too far out of your way or model in a contrived manner to make this happen.

Mirroring entire parts

Often when modeling, you are required to have a left- and a right-handed part. For this, you need to use a method other than body or feature mirroring. The Mirror Part command creates a brand new part by mirroring an existing part. The new part does not inherit all the features of the original, and so any changes must be created in the original part. If you want different versions of the two parts, you need to use Configurations, which have not been covered yet in this book.

Cross-Reference

Configurations are covered in detail in Chapter 10. ■

You can use the Mirror Part command by pre-selecting a plane or planar face. You should be careful when choosing the plane because the new part will have a relationship to the part Origin, based on the plane on which it was mirrored.

The Mirror Part command is found in the Insert menu. When mirroring a part, you can bring several entity types from the original file to the mirrored part. These include axes, planes, cosmetic threads, and surface bodies. Sketches and features are two commonly requested items to be brought forward by the Mirror Part command, but this is not possible in the current version of the software.

Mirror Part invokes the Insert Part feature, which is covered in more detail in Chapters 26 and 28, on multi-bodies and Master Model techniques, respectively.

One of the options available when you make a mirrored part is to break the link to the original part. This option brings forward all the sketches and features of the original part, and then adds a Move/Copy Body feature at the end of the tree that simply mirrors the body.

Note
Under normal circumstances, you cannot get the Move/Copy Body feature to mirror a body. SolidWorks has applied some magic pixie dust behind the scene to make this happen. ■

Tutorial: Creating a Circular Pattern
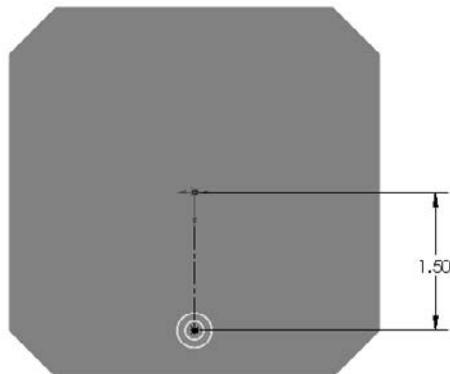
Follow these steps to get practice with creating circular pattern features:

1. **Draw a square block on the Top plane centered on the Origin, 4 inches on each side, .5-inch thick extruded Mid Plane with .5-inch chamfers on the four corners.**

2. **Pre-select the top face of the block and start the Hole Wizard.** Select a counterbored hole for a 10-32 socket head cap screw, and place it as shown in Figure 8.24.

3. **Create an axis using the Front and Right planes.** Choose Insert ⇨ Reference Geometry ⇨ Axis. Select the Two Planes option, and select Front and Right planes from the flyout FeatureManager. (Click the bar that says Axis at the top of the PropertyManager to access the flyout FeatureManager.) This creates an axis in the center of the rectangular part.

4. **Click the Circular Pattern tool on the Features toolbar.** Select the new Axis in the top Pattern Axis selection box in the Circular Pattern PropertyManager. Select the Equal Spacing option and make sure that the angle is set to 360°. Set the number of instances to 8.

5. **In the Features to Pattern panel, select the counterbored hole.** Make sure that Geometry Pattern is turned off.

6. **Click OK to finish the part, as shown in Figure 8.25.**
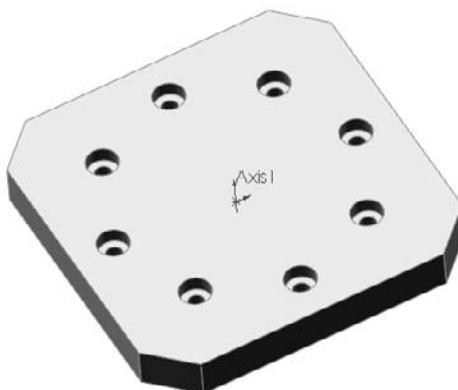
FIGURE 8.24

Start drawing a plate with holes.

FIGURE 8.25

The finished circular pattern

Tutorial: Mirroring Features

Follow these steps to get some practice with creating mirror features:
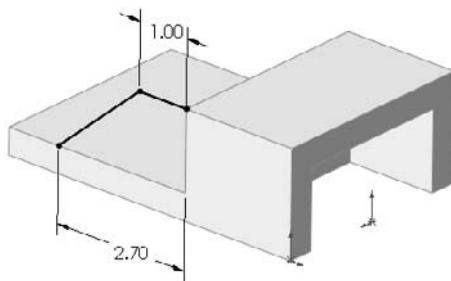
1. **Open the file from the CD-ROM called** `Chapter8 Tutorial2.sldprt`.

2. **Open a sketch on the side of the part, as shown in Figure 8.26.** The straight line on top is 1.00 inch long, and the angled line ends 2.70 inches from the edge, as shown.

FIGURE 8.26
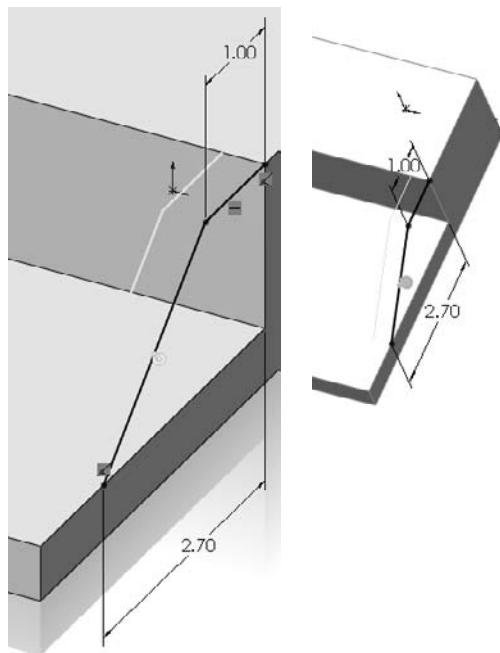
The sketch for the Rib feature

3. **Click the Rib tool on the Features toolbar or select it from the menu at Insert ⇨ Features ⇨ Rib.** Set the material arrow to go down toward the block, and the thickness setting to go to the inside by .375 inches. The PropertyManager and the preview should look like Figure 8.27.

FIGURE 8.27
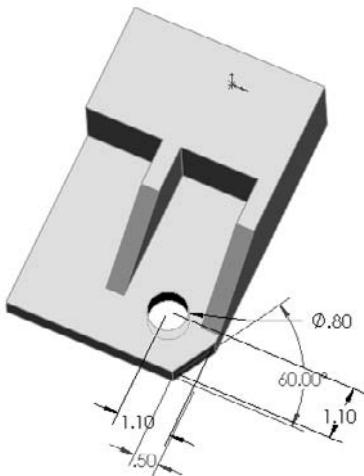
Applying the Rib feature

4. **Create a linear pattern using the rib, making the pattern reach 2 inches into the part.**

5. **Create a chamfer on the same side of the part as the original rib, as shown in Figure 8.28.** The chamfer is an Angle-Distance using 60° and .5 inches.

6. **Create a round hole, sized and positioned as shown.**

Additional features on the part

7. **Mirror the hole and the chamfer about the Right plane.** The parametrics of the chamfer will have difficulty patterning, and so you need to use the Geometry Pattern option. The finished part is shown in Figure 8.29.

The finished part
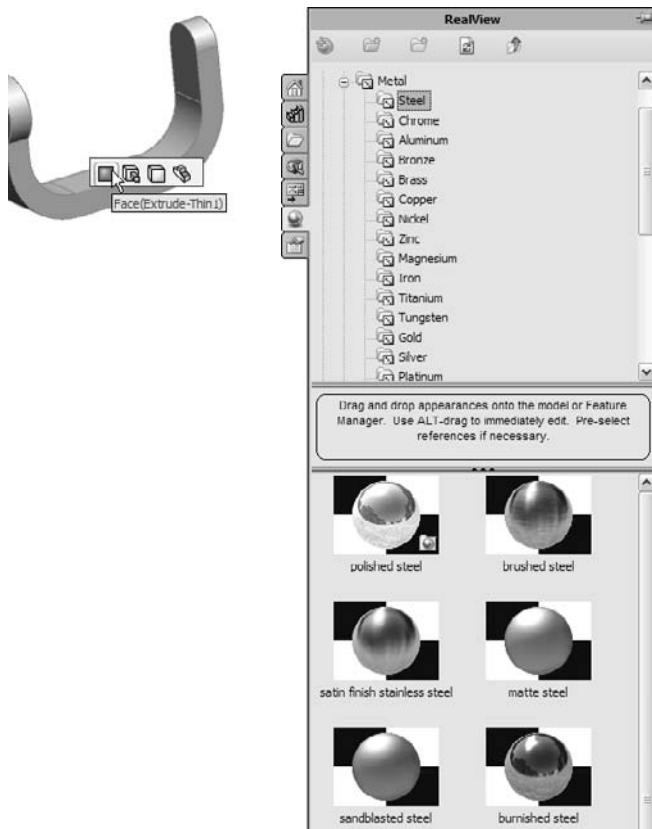
Tutorial: Applying a Cosmetic Pattern

1. **Open the file from the CD-ROM for Chapter 8 called** `Chapter 8 – tutorial – cosmetic pattern.sldprt`.

2. **Click the Appearances tab in the Task Pane.** These steps will work whether or not you have RealView actually selected.

3. **Expand the Appearances heading, then the Metal heading, then Steel, and then drag the Sandblasted Steel icon from the lower panel onto the part.** When the popup menu appears, select the Part icon, to apply the appearance to the entire part. Figure 8.30 shows the Task Pane and the popup menu.

FIGURE 8.30

Applying an appearance to a part
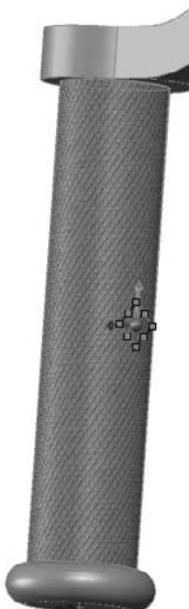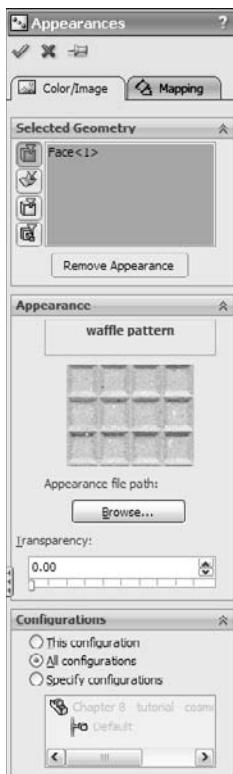
4. **Expand the Miscellaneous listing (under Appearances) and the Pattern heading.**
Drag the Waffle Pattern onto the large cylindrical face of the part, and then Alt-click
the Face icon in the popup toolbar. Using the Alt key while dragging or to select face,
feature, body, or part automatically activates the PropertyManager to edit the appearance.
Figure 8.31 shows the Appearances PropertyManager.

FIGURE 8.31

The Appearances PropertyManager

5. **In the Mapping tab of the Appearances PropertyManager, select the cylindrical
mapping under the Mapping Style section of the Mapping Controls panel.**
6. **Change the Rotation to 45 degrees, and choose the smallest Mapping Size.**

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Summary

Feature patterns and mirrors are powerful tools, but you need to have some discipline to benefit from their usefulness. Patterns in particular are extremely flexible, with many types of functions and options available. You should avoid sketch patterns if possible, not only because of performance considerations, but also because complex sketches (sketches with a lot of entities and relations) tend to fail more often than simple sketches.

Using Equations

IN THIS CHAPTER

Using equations to create relationships between dimensions

Linking dimensions together

Assigning global variables

Entering expressions

Using equations tutorial

Parametric sketch relations are not the only way to drive dimensions with intelligence. You can also use equations, link values, and global variables. Equations help you create simple or complex mathematical relations between dimensions. Link values are essentially a quick way of making two dimensions equal. Global variables can be used in equations like other dimension names. These three techniques are all very similar and related to one another in the interface, but are used in different ways in different situations.

Equations can cause problems if used incorrectly, and have functionality that may appear incorrect at times, but if you are familiar with how the tools work, you can avoid the common pitfalls and get maximum benefit by adding intelligence to your designs.

Understanding Equations
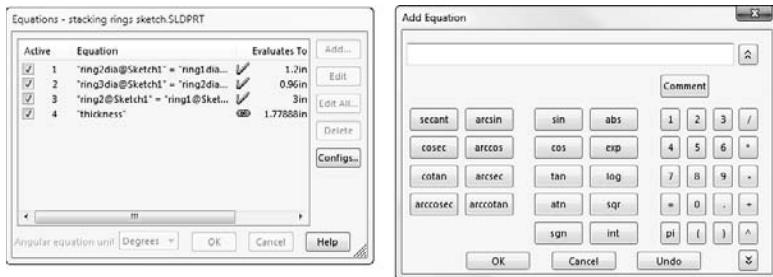
Σ You can use the Equations tools to create simple or complex mathematical relations between dimensions. You can find the Equations tool on the Tools toolbar or by choosing Tools ⇨ Equations from the menu. Equations are stored in a folder at the top of the FeatureManager, Figure 9.1 shows the Equations main interface along with the Add Equation window. As I have noted with other areas of the interface, Equations still uses a floating dialog box. SolidWorks has put most functions in the PropertyManager, but equations tend to be more horizontal than vertical, while the PropertyManager is more vertical than horizontal.

The Equations interface

Using the Equations interface, you can turn off equations temporarily by deselecting the Active check box in front of the equation. Equations can also be deactivated by a design table. I will discuss design tables in more detail in Chapter 10, which discusses configurations.

Best Practice

Although I do not cover configurations until Chapter 10, I will mention part of the relationship between equations and configurations here. Equations and configurations (particularly those that are driven by a design table) should probably not be mixed. This is not because they do not work together, but more for the sake of organization. When controlling dimensions, it can become confusing if the changes are being driven from multiple sources. In addition, there is no reason not to bring your equations into Excel rather than using the comparatively limited equation functionality offered by SolidWorks. Of course, every user will have his or her own reasons for working one way or another, and this is really just a question of personal preference. ■

Creating equations

Equations are easy to create and useful for many purposes. A common situation where you would use an equation is to space a pattern of holes evenly along an edge, including the gap on both ends, where the gap at the ends is half of the regular spacing. Before you write an equation, you need to take care of a few organizational details.

Naming dimensions
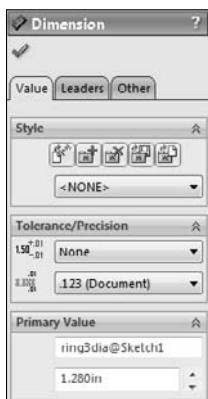
It is not necessary to name every entity in every SolidWorks document, but you should get in the habit of naming important features, sketches, and even dimensions. Dimensions become particularly important when you use them in equations, configurations, and design tables. Under most circumstances, you do not use or even see dimension names, but with equations, you do.

Named dimensions make a huge difference when you want to recognize the function of an equation by simply reading it. A most obvious example would be the difference between D3@Sketch6 and Length@WindowExtrusionSketch. The first name means nothing, but the second one is descriptive if you are familiar with the part.

To name a dimension, click the dimension and go to the PropertyManager. In the Primary Value panel shown in Figure 9.2, type the new name for the dimension in the Name text box. You cannot use the symbol @ in dimension names because it is used as a delimiter between the name of the dimension and the feature or sketch to which it applies. Also, be aware that even though the software allows you to change the name of the sketch or feature in the Dimension PropertyManager, it will not accept this change.

FIGURE 9.2

Renaming a dimension

Best Practice

You should keep dimension names as short as possible while still making them unique and descriptive. This is because space in the interface is often limited, and when combined with sketch or feature names (and even part names when used in an assembly), the names can become difficult to display in a readable fashion. ■

Tip
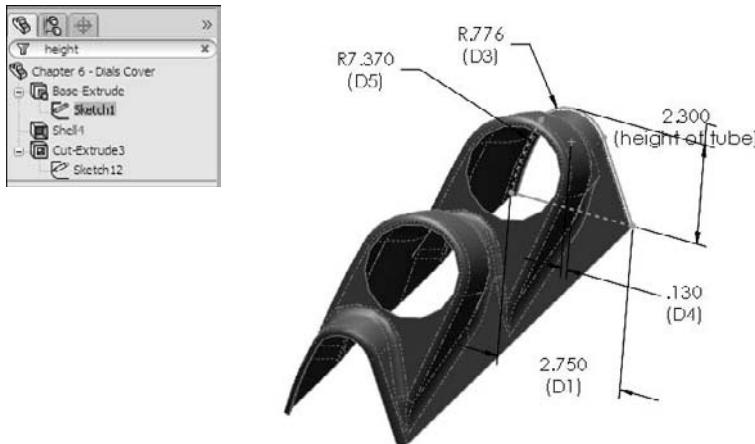
You can show dimension names as a part of the dimension itself; choose View ⇨ Dimension Names and select the option. It's also helpful to know the FeatureManager Filter filters dimension names, which makes named dimensions easy to find. Figure 9.3 shows the filter displaying features and sketches that contain a dimension containing the filtered word "height." Other filtered words display in tool tips, but dimension names appear not to. ■

FIGURE 9.3

Using the FeatureManager Filter to filter dimension names
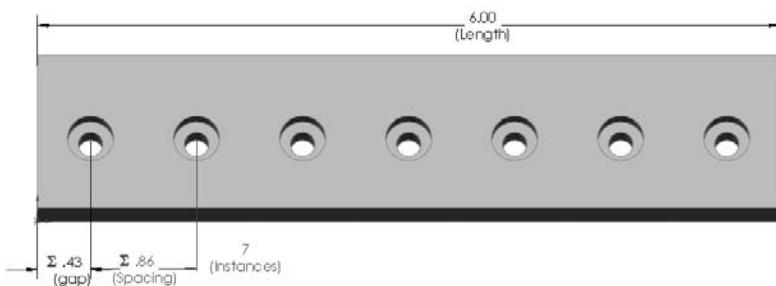
Building the equation

When creating an equation in SolidWorks, it is often a good idea to write it out on paper first. Examine the part shown in Figure 9.4, where the relevant dimensions have been named and displayed. The number of holes — called Instances here — is the driving variable. From that number, the spacing of the holes is calculated over the length of the part. There is also a gap on each end of the pattern of holes. This gap (measured between the center of the last hole and the end of the part) always needs to be half of the spacing between the holes. The sigma symbols to the left of the dimensions indicate that an equation is driving it. Dimensions driven by equations cannot be directly edited.

FIGURE 9.4

Variables for the hole pattern

In this case, a more sophisticated equation has not been implemented to account for the diameter of the holes possibly interfering with one another when there are a large number of holes. In other words, because there are two values that need to be calculated (the spacing and the gap), you need to create two equations. Because the gap dimension is always half of the spacing, the spacing needs to be calculated first, as follows:

```
Spacing = Length / ((Instances-1)+1)
```

The *Instances –1* term stands for the number of spacings. If you have two holes, then there is only one spacing. The *+1* term stands for the two half-spacings for the two ends. The second equation is simpler and looks like this:

```
Gap = Spacing / 2
```

The order of the equations is important. SolidWorks solves the equations in the order in which they are listed in the Equations dialog box. Because the gap is dependent on the spacing, the spacing must be calculated before the gap. If it is done the other way around, you can get into a situation where it takes two rebuilds to finalize a set of equations, or even a situation where in every rebuild all the numbers change. This is called a circular relation, and is a common error in order or history dependent functions, not just in SolidWorks, but in many CAD applications. Figure 9.5 shows the resulting set of equations.

FIGURE 9.5

Equations for the hole pattern

Before beginning to build the equation, you should first display the dimensions that you need to use to create the equation. You can add dimensions to the equation by clicking them from the graphics window. To do this, right-click the Annotations folder at the top of the FeatureManager, and select Show Feature Dimensions. You should also select the Display Annotations option if it is not already selected. When you have done this, all the dimensions that you need to create every feature are displayed. Also, be sure to turn on the Show Dimension Names option by choosing Tools ➪ Options ➪ General.

Tip

For models that have more than a few features, showing all the dimensions in the entire model may overload the screen with information. In this case, you can double-click a feature from the FeatureManager to show all the dimensions on that feature. ∎

To build the equation, first click the Equation button on the Tools toolbar to open the Equations dialog box. Then click the Add button to display the Add Equation dialog box. To add dimensions to the equation section, just select the dimension. You can use the keypad on the dialog box or on your keyboard to add operators and syntax. All standard rules of syntax apply for the order of operations, use of parentheses, and driving versus driven sides of the equation.

Using comments

Notice the comment to the right of the first equation in Figure 9.5. Comments can be very useful for annotating equations for yourself or others. Two important reasons to annotate are to remember the significance of variables or dimensions and to add special notes about the logic of the equation.

You can make comments for equations by using a single quote after the end of the equation, or by clicking the Comment button in the Add Equation dialog box. In the following example, the comment, "This must be solved first," is applied to the equation using the single quote before the comment.

```
"Spacing@LPattern1" = "Length@Sketch1" / ("Instances@LPattern1")
    'This must be solved first
```

To expand on the earlier discussion about projected changes to the Equation interface, several standard selection functionalities do not work in the Edit Equation dialog box. These include triple-clicking to select all (although double-clicking works to select a single word) and pressing Ctrl+A to select all.

Tip

You can make general comments for the model in the Design Journal, a Microsoft Word document that is embedded into the SolidWorks file. The Design Journal is found in the Design Binder folder near the top of the FeatureManager. ■

On the CD-ROM

You can find the part used in this section on the CD-ROM with the filename `Chapter 9 Equations.sldprt.` ■

Using driven dimensions
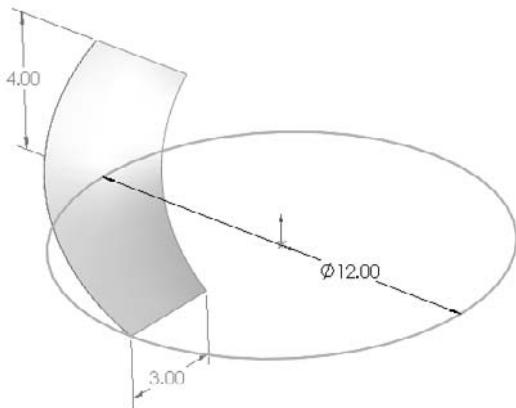
Sometimes it is more convenient to use a driven (reference) dimension in an equation. This is particularly true when using geometry is the best way to calculate a number. For example, if you are manufacturing a helical auger in 90-degree sections from flat steel stock, then you need to design the auger in 3D, but begin to manufacture it in 2D.

What is the shape of the auger when flat? The best way to figure this out (aside from lofted bends, which are discussed in Chapter 29) is to use a little high school geometry, a construction sketch, and some simple equations.

Figure 9.6 shows a 90-degree section of an auger blade. The outside diameter is 12 inches, and the blade width is 3 inches. The overall height is 4 inches. In this case, the auger is represented as a surface because the thickness is ignored. Surface features can be useful in situations like this (used as construction geometry) and are discussed in Chapter 27.

FIGURE 9.6

A representation of the auger

On the CD-ROM

You can find the part for Figure 9.6 on the CD-ROM with the filename `Chapter 9 Auger.sldprt.` ■
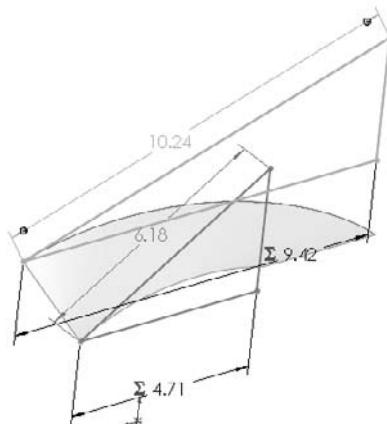
With this information, you can calculate the lengths of the 3D edges using a sketch and a simple equation. In Figure 9.7, the hypotenuses of the triangles represent the helical edges of the inside and outside of the auger. By making the triangles the same height as the auger section, and by making the horizontal side of the triangle the same length as a quarter of the inside or outside diameter by using simple equations, the geometry and sketch relations automatically calculate the flat lengths of the inside and outside edges of the auger (length of triangle side = diameter of circle × pi / 4). In this way, the triangle is used to simplify the calculation, and give it a visual result.

FIGURE 9.7

Triangles calculate the length of the helical edge.

From this point, you can calculate the flat pattern again, using SolidWorks' sketch-solving capabilities as the calculator. Think of the auger as being the cardboard tube inside a roll of paper towels. If you examine one of these tubes closely, you see that it is simply a straight and flat strip of cardboard that has been wound around a cylinder. What was the flat, straight edge of the original board is wound into a helix. This method simply reverses that process.

This example requires the little-used arc-length dimension to drive the size of the arc. The hypotenuse dimensions are shown by driven or reference dimensions, and these are used to drive the arc-length dimensions, as shown in Figure 9.8. Remember that you can create arc length dimensions by using the Smart Dimension tool to click both endpoints of the arc and then the arc itself.

The reasoning behind this example may be a little difficult to grasp, but the equations and the sketches are certainly simple.

Caution
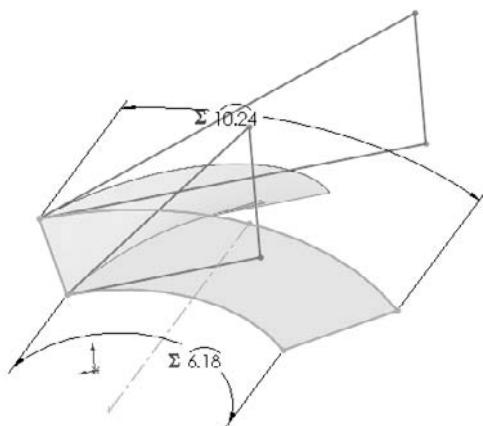
Using reference dimensions on the driving (independent or right) side of the equation can, and in some situations, require more than one rebuild to arrive at a stable value (meaning a value that does not change with the next rebuild). SolidWorks issues a warning when it sees that you are using a reference dimension in an equation, but it does allow it. ■

Σ Equations are listed in the Equations folder in the FeatureManager. You can edit, add, or delete them through the right mouse button (RMB) menu.

FIGURE 9.8

Figuring the flat pattern of the auger

Using equation tricks

Some functions that are permitted in SolidWorks equations are often viewed as parlor tricks, but they actually do have some practical applications. The two functions that fall into this category are IIF and SWITCH. If you are familiar with any programming language, you may already be familiar with these two functions.

IIF

In words, this is how an IIF statement is used:

If some relationship is fulfilled, then the IIF function returns a value. If the relationship is not fulfilled, then it returns a different value.

A more technical description is

```
IIF(expression, value if true, value if false)
```

In practice, you could use it like this:

```
IIF(x>5, x-1, x+1)
```

which reads, "if x is greater than 5, then subtract 1 from x; if not, then add 1 to x." One of the reasons why this is considered a parlor trick is that this function causes the value of x to oscillate between two numbers (depending on the number that it starts with) with each rebuild. It may be difficult to imagine an application where this sort of behavior would be desirable, but when you combine it with a macro that simply rebuilds a model a number of times, you can use it to create a certain animation effect.

On the CD-ROM
A simple example of the `IIF` function can be found on the CD-ROM with the filename `Chapter 9 Oscillate.sldprt`**. The equation is shown in Figure 9.9.** ■

FIGURE 9.9

An equation using `IIF`

Tip
You can find some great examples of this function at `www.mikejwilson.com`**, along with many other extremely creative examples of SolidWorks modeling. The model on this site called** `Ship in a Bottle. sldprt` **also includes a macro that will rebuild the model a certain number of times, which is useful for animations that are created in this way.** ■

SWITCH

The `SWITCH` function enables you to have a list of relationships with associated values. The value of the first relationship in the list that is satisfied is returned by the `SWITCH` function. For example,

```
switch (x>2, 1.5, x>1, .5 x<1, 2.5)
```

reads as follows: "if x is greater than 2, then the answer is 1.5; if x is not greater than 2 but greater than 1, then the answer is .5; if x is less than 1, then the answer is 2.5."

As you can see, this function does not cover all situations, but it does create a condition where the value cycles through three different numbers in a specific order. Is this useful? Possibly. Again, the main application for this function would be a simple animation for changing the size or shape of SolidWorks components that cannot be changed in other more conventional ways.

Using Link Values
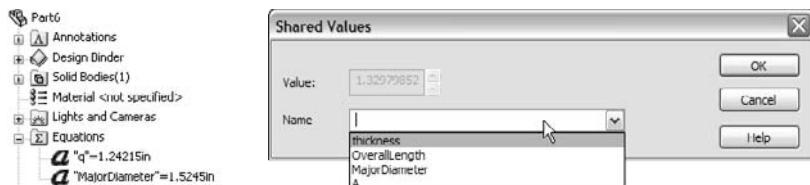
Link values are simply a way to link several dimensions together, making them equal. A link value is not exactly like an equation that sets the dimensions equal, because it does not depend on order like an equation does. All dimensions are set to the same value simultaneously.

Link values are available by right-clicking on a dimension. You can also get to link values by clicking the down arrow on the right side of the Modify dialog box. Unfortunately, they are not available from the RMB menu when the dimension tool is active. To apply a link value to a new dimension, you must place the dimension, exit the dimension tool, right-click the dimension, and select Link Value.

Link values are listed under the Equations folder in the FeatureManager. Figure 9.10 shows the link values in a listed part, and the drop-down list from which you can select them or type them. Notice again that the Link Values feature also operates from a dialog box instead of the PropertyManager.

FIGURE 9.10

Link values listed in the FeatureManager, and the Shared Values interface

Note

Another way to access link values is through the Modify dialog box. If you click the down arrow at the right end of the dimension value box, you can select between Link Values and Equations. In fact, if you press the Down Arrow key on the keyboard, the Equation interface becomes available. There is no similar trick to get Link Values to appear. ∎

You must type in the first link value that is assigned in a part. After you add the first one, you can link other dimensions to this link value by using the drop-down arrow shown in Figure 9.10. You cannot edit link values, meaning you cannot change a dimension from linking to a value called "height" to a value called "length." In order to change the value to which a dimension is linked, you must first unlink the value and then relink it. The Unlink function is available from the RMB menu in the same way that you assign link values. Dimensions that have a link value have the small chain symbol displayed to the left of the dimension.

To link several dimensions to the same value at the same time, you can Ctrl+select multiple dimensions and then right-click one of them and select Link Value. It will link all the dimensions selected at once. (Thanks to Brian McElyea for this suggestion!)

Tip

One link value name has a special significance. If you use the name thickness, then a Link To Thickness option appears in all extrude dialog boxes. This is intended to reflect sheet metal functionality, but it is useful for models of various manufacturing techniques.

To take this one step further, you can save a part template with a thickness link value; all your new parts will also have this functionality right from the start. To save the template with a link value, you must create at least one dimension to assign the link value, and then delete the geometry (and the dimension); however, the link value will remain. ■

Link values of different types are not necessarily interchangeable. You cannot use angular dimension link values on radius, diameter, or linear dimensions. You can use linear and diameter link values interchangeably, but not angle link values.

Using Global Variables

Global variables are assigned in the Equations dialog box as simply the variable name equaling the value. Figure 9.11 shows a list of equations, link values, and global variables. When you type in a variable name, you do not need to add the quotation marks; they are added automatically.

FIGURE 9.11

Equations, link values, and global variables

Despite the word *variable* in the name global variable, the values are not variable. They are fixed, and only changeable through the Equations dialog box. The only place you can use global variables is in equations. You cannot directly enter them into dialog boxes for dimension values, or use them like Link Values.
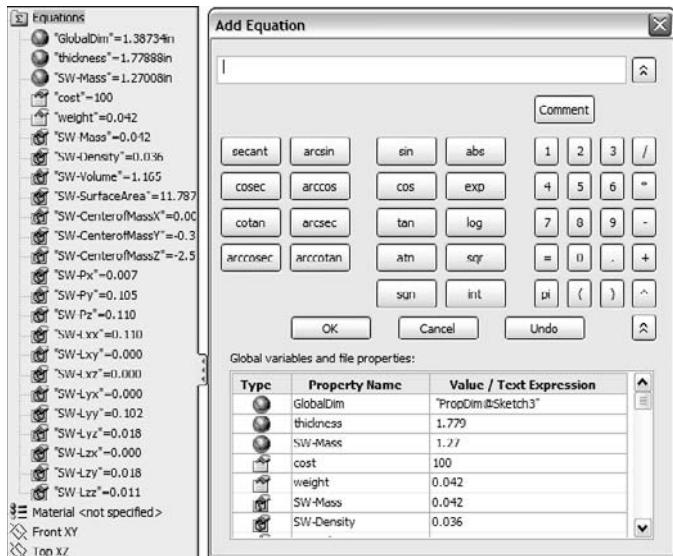
You can use custom and file properties to drive equations. If you right-click your Equations folder and select Show Properties, you see that the default file properties already exist in the list:

Global Variable

Custom Property

Default File Property

In the equation editor shown on the right in Figure 9.12, you can expand the list of global, custom, and default properties for easy selection and placement into equations. Any custom properties you add that are of the type "number" are automatically added to this list and can be used in equations. Notice that the custom property "cost" is a property saved in my template and gets picked up for use here.

FIGURE 9.12

Equations and properties

Using Expressions

Expressions, unlike all the previous variables, values, and equations, can be entered directly into dimension dialog boxes in the Modify dialog box and PropertyManager value boxes. The expressions have to be composed of numbers and mathematical operators. An expression such as

```
2.375+(4.8/3) -1.1
```

355

is perfectly acceptable, as is

        1+1/2

or

        1 1/2

In the second case in this example, the plus symbol is understood.

Other types of operations are also available, such as ones for changing units in a dimension box. For example, if you are editing a part in inches, and enter 40mm, then SolidWorks does the conversion for you. You can even mix units in a single expression such as 4.875+3.5mm, where the inch part is assumed as the document units.

SolidWorks does not remember the expression itself, only the final value. Expressions can be entered into any place where you enter dimensions for SolidWorks features.

Tutorial: Using Equations

Follow these steps to get some practice with using equations:

1. **Start from the part on the CD-ROM with the filename** `Chapter9 Tutorial Start.sldprt`.

2. **Show the dimension names.** Choose View ⇨ Show Dimension Names to find this setting.

3. **Double-click the Circular Pattern feature to display the angle and number of instances of the feet and related features.** You may have to move the angle dimension to see the pattern instance number.

4. **Click the instance number.** Change the name of the dimension to **#** (pound or number sign) in the Dimension PropertyManager. Make sure that Instant3D is unselected when doing this.

5. **Double-click the first feature, which is the revolve, and rename the 3.60-inch dimension** CapRad, **again by selecting it and using the PropertyManager.**

6. **Write an equation that drives the number of legs by CapRad/7.**

   a. Open the Equations dialog box by choosing Tools ⇨ Equations.

   b. Click Add to add an equation.

   c. Double-click the Circular Pattern and click the # dimension. Make sure that the name of the dimension is listed in the equation box, and type an equal sign.

   d. Double-click the Revolve feature and select the CapRad dimension; then type the characters **/1.5**.

   e. Add a comment to the equation to reflect which dimension is driving which dimension.

7. **Click Rebuild, press Ctrl+B or Ctrl+Q to rebuild the model, and observe whether any update takes place.**

8. **Rename the 6.00-inch dimension for the height of the revolved feature to** DomeHt.

9. **Create a second equation that drives the DomeHt dimension at the current ratio of the height to the radius.**

   a. Create a global variable called Ratio = 6/3.6 (1.66667) in the Equations dialog box.

   b. Create the equation. The equation will take the form of DomeHt = (Ratio) × CapRad. You can use the drop-down list under the calculator pad to select the Ratio variable from the list.

10. **Use a link value to make the radii of Fillet1 and Fillet2 the same.**

11. **Double-click the revolve feature.** Change the .CapRad dimension to 5 and rebuild. You should observe 3 feet. Change it again to 6 and you should see 4 feet.

12. **Give the part a new name, including your initials or the date, and save and close it.**

Summary

SolidWorks equations and related dimension-management tools are powerful, but often leave you wishing for a little more flexibility and control. The interface is not up to date with the rest of the SolidWorks interface, and so I would look to see an updated equation interface soon that integrates dimension input, link values, and global variables.

If you want to encourage SolidWorks to revise certain features, then you can go to the SolidWorks Web site and submit an enhancement request. They do look at customer input when developing or updating functionality.

Working with Part Configurations

Configurations, also known as simply *configs*, are variations of a part in which dimensions are changed, features are suppressed (turned off), and other items such as color or custom properties may be controlled. Configurations enable you to have these variations within a single part file, which is both convenient and efficient.

This chapter deals only with part configurations, but assemblies can also have configurations. Assembly configurations can use different part configurations, among other things. This will mean more to you as you learn about part configurations.

Cross-Reference

Assembly configurations are discussed in Chapter 14. ■

One example of configurations is having many sizes of a fastener within a part file. Socket Head Cap Screws have thousands of potential sizes. Configured parts can also have features that you can turn off and on (suppress and unsuppress, respectively), such as a cross drive or a slotted drive. Changing dimensions and suppressing or unsuppressing features are the most commonly used techniques available through configurations.

There is some overlap between the topics of configurations and display states, with colors and hide/show states being controlled by both methods. When you have an option, it is best to control visual properties using display states because they require fewer resources (they're faster).

Controlling Items with Configurations

With every new release of SolidWorks software, it seems that more items become "configurable;" that is, able to be driven by configurations. Configurable items for parts include the following:

- Feature dimensions, tolerances, driving/driven state
- Suppression of features, equations, sketch relations, and feature end conditions
- Sketch plane used by a sketch
- Configuration-specific custom properties
- Part, body, feature, and face colors
- Derived configurations
- Properties that can be assigned, such as mass and center of gravity
- Configuration of base or split parts
- Sketch pattern instances
- Sketch text
- Size of Hole Wizard holes

You can control configurations in several ways:

- Making changes manually to dimensions and features
- Using the Configure Feature/Modify Configurations table
- Using an Excel-based design table
- Using the Configuration Publisher

Finding configurations

SolidWorks lists configurations in the ConfigurationManager. This is a tab at the top of the FeatureManager area.

Tip

You can split the FeatureManager interface into two by dragging the splitter bar at the top of the panel. This very useful function enables you to see the FeatureManager in the upper panel and the PropertyManager or ConfigurationManager in the lower panel. Remember also that starting with SolidWorks 2009, you can detach the PropertyManager from the left-hand side panel area. ∎

Deleting configs

Each part has a default config named "Default." There is nothing special about this config; you can rename it and even delete it. At least one config must always remain in the tree, and you cannot delete the configuration that is currently active. If you would like to remove a config, then you need to switch to another config (by double-clicking the other configuration in the ConfigurationManager), and then delete the one you want to remove.

If you have used the software for a while, you may remember not being able to delete or rename configurations that are referenced by open documents. This limitation (at least for renaming configurations) was removed by a service pack in SolidWorks 2009. Being able to rename configurations referenced by open documents such as assemblies or drawings is an important change that new users will probably take for granted, and veteran users need to be aware of.

Cross-Reference

Chapter 14 deals with configurations of assemblies in depth. This chapter deals only with configurations of parts. Configurations of drawings do not exist. ■

If you try to delete a part configuration being used by an open assembly, SolidWorks simply issues the message "None of the selected entities could be deleted" without explanation.

If you delete a configuration of a part that is used in an assembly, but the assembly is not currently open, the next time the assembly is opened, it issues the message "The following component configurations could not be found . . . . If the configuration was renamed the same configuration will be used, otherwise the last active configuration will be substituted for each instance."

You can delete groups of configs by window select, Shift+select, or Ctrl+select in the ConfigurationManager. You can also use the right mouse button (RMB) menu, much like regular features in the FeatureManager. None of the configurations selected for deletion may be active, or referenced by other open and resolved documents.

Sorting configs

In the ConfigurationManager, configs are listed alphabetically, not in the order in which they are created. This has several advantages, especially when you have a large number of configs. For example, if configs are named by size in a part that you are working with, then when you select a configuration, you can type in a number and the selection scrolls to that place in the list of configs. This makes it easier to select the one you are looking for, much the same as it works in Windows Explorer.

Alphabetization

This alphabetized order is significant because many other sections of the SolidWorks interface are not alphabetized, which causes problems when you are searching for items in larger lists. Sections that are not alphabetized include Help/Contents, Files of Type lists in Open and Save dialog boxes, and the File Locations settings (Tools⇨Options⇨File Locations), Entity Color list, and several others. If you are inclined to submit an Enhancement Request to SolidWorks, alphabetization of lists is one topic that would benefit everyone and should be easy for SolidWorks to implement.

Naming configs

In order for this sorting and alphabetization to work, you must first name the configs properly. For example, if you have a list of sizes or config names from 1 to 100, then you should use 001, 002…100 as your syntax. This makes the config names easier to browse and type in. Syntax becomes most important when you place a part with many configs into an assembly, because you must select a config from the list, and typing in the first few numbers is often faster and easier than scrolling to it.
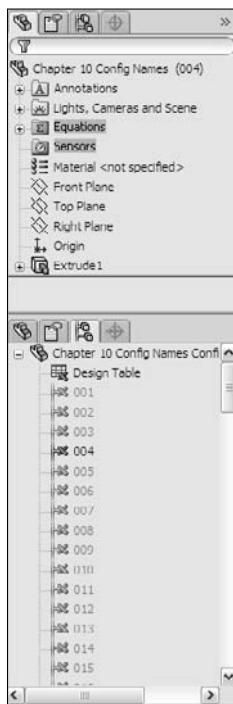
On the CD-ROM

The CD-ROM contains a part called `Chapter 10 Config Names.sldprt,` **which illustrates proper naming and alphabetization.** ■

To understand this technique better, you can open the part called `Chapter 10 Config Names.sldprt` from the CD-ROM, split the FeatureManager area, and change one of the panes to display the ConfigurationManager. Click one of the configuration names, and type in a number between 001 and 100. The highlight scrolls to the number that you typed in. Thoughtful selection of the configuration names can save you and your co-workers a lot of time when you need to insert select configs into an assembly. Figure 10.1 shows this arrangement.

FIGURE 10.1

The split FeatureManager, displaying the ConfigurationManager

Cross-Reference

The splitter bar and other portions of the FeatureManager interface appear in Chapter 2. ■

Activating configurations

Within a part file, to change the display from one configuration to another, you must first switch to the ConfigurationManager panel, and then either double-click the desired config or right-click it and select Show Configuration.
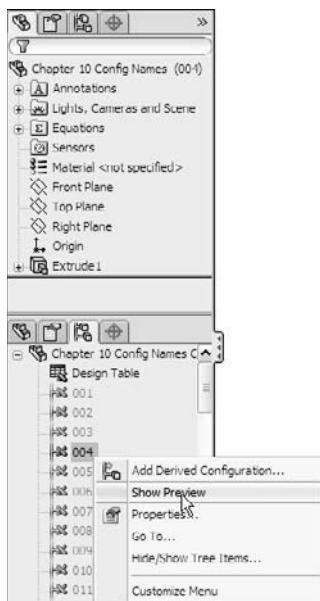
Alternatively, you can right-click the config in the ConfigurationManager and select Show Preview, as shown in Figure 10.2. A small preview thumbnail displays in the PropertyManager panel. However, not all configurations will have previews. For example, in a part with many configs that have been generated automatically by a design table, the configurations may not have previews because the config itself has never actually been rebuilt. Previews exist only when the configuration has been activated at least once, the image on the screen generated, and the part then saved. SolidWorks stores both the body (geometry) and the preview image of the part so that next time you access the configuration, the software does not have to rebuild everything again.

FIGURE 10.2

Showing a configuration preview
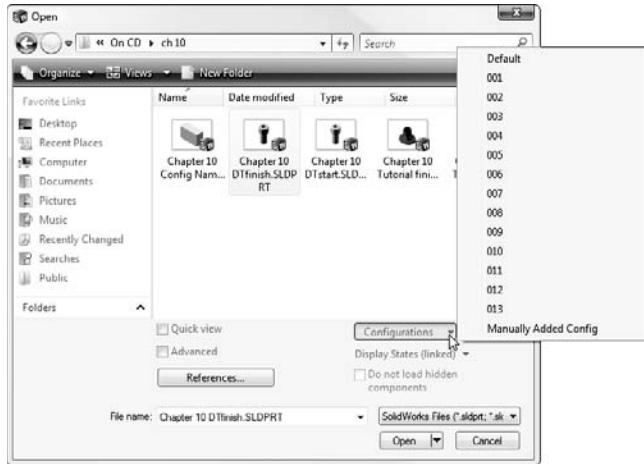
You can even select a configuration while opening a file. This enables you to save time by avoiding rebuilding the model. To take advantage of this option, you must use the File ⇨ Open interface, which is shown in Figure 10.3. You can select the config from the lower-right drop-down Configurations list.

FIGURE 10.3

Selecting a configuration from the Open dialog box

Creating configurations

You can create configs manually, using the Configure Feature/Modify Configurations table or through Excel-driven design tables. Design tables are extremely useful for situations where there are more than a few configs, or more than a few items are being controlled. You should use design tables because they keep things very organized within the spreadsheet grid.
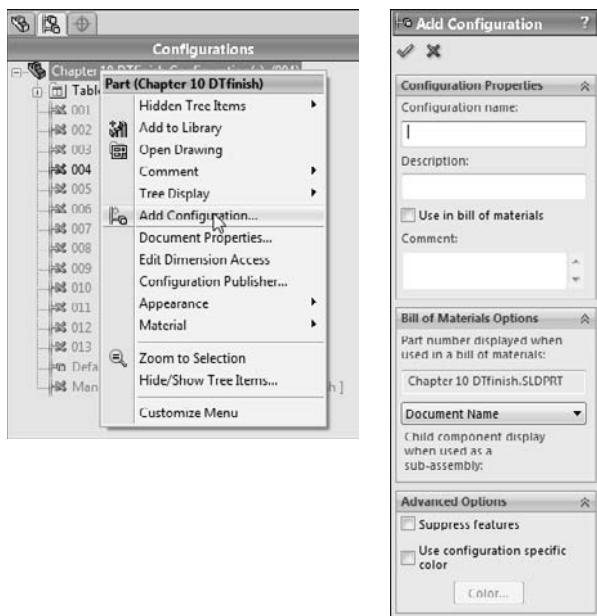
For now, I am going to focus on creating and manipulating configs manually so that you can become familiar with them without also worrying about Excel and design table syntax.

A new config

To make a new config, you can right-click the top-level icon in the ConfigurationManager, which displays a part symbol and the name of the part, and select Add Configuration. If you right-click an existing configuration, SolidWorks will make a derived config, which I discuss later in this chapter. Figure 10.4 shows the RMB menu and the Properties dialog box that you can use to set up the new config.

FIGURE 10.4

Creating a new configuration
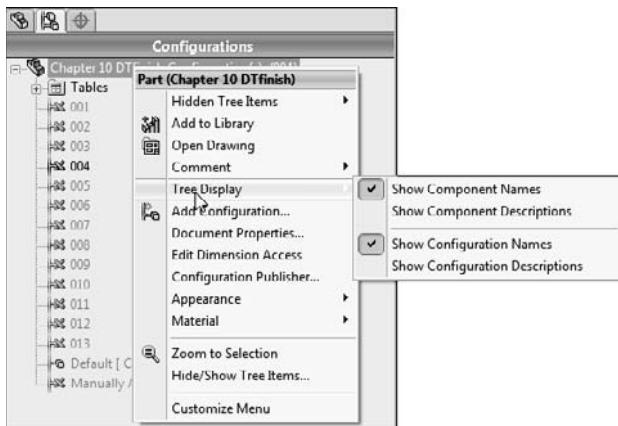
Configuration properties and options

The name of the config is important mainly for quick access and organization purposes. The configuration description is also important, because it can display in the ConfigurationManager, and even in the Assembly tree. (You can also use the FeatureManager Filter to search configuration descriptions.) This is important when the name of the config is numerical rather than descriptive, and you would like to also have a description but not include it in the name. The config description can also appear in place of the filename in the Assembly tree display. I discuss this in more detail in Chapter 12. Config descriptions can be driven manually through the Configuration Properties dialog box or through a design table if you have many configs to manage. You can display config descriptions through the RMB menu, as shown in Figure 10.5.

FIGURE 10.5

Enabling configuration descriptions

I discuss the Bill of Materials (BOM) options in Chapter 24, but the option is set in the Configuration Properties to use the filename, the configuration name, or a custom name that the user specifies. You can save this setting with a template. You achieve control over configurations through the combination of the Configuration Properties and the Advanced Options, which I discuss next.

Best Practice

Although you can change the preferred settings at any time, it is definitely a best practice to make a template early on when you are using SolidWorks to model parts. SolidWorks remembers the BOM options and Advanced options that you set for the Default configuration and uses them in document templates. This is true for both part and assembly templates. ■

Advanced options

Two advanced configuration options are found in the bottom panel of the Configuration Properties PropertyManager: Suppress features and Use configuration specific color. While the second option is self-explanatory, the first one is not, and often catches new and even experienced users off guard.
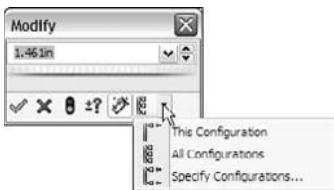
Suppress features refers to how inactive configurations should handle new features that are added to the part. For example, if you have two configs, 1 and 2, and config 1 is active and you add a new Fillet feature, what happens to that feature in config 2? If this option is turned on, the new features are suppressed in the inactive configs. If it is turned off, the new features will be unsuppressed when the inactive configs are activated. This creates a much bigger challenge for manually created configurations than for design table-driven configs because changing suppression states for several features across multiple configs is much easier in a design table than in manual config management.

The Modify dialog box

The Modify dialog box enables you to change dimensions by just double-clicking the dimension and changing the value. When you change a dimension using the Modify dialog box in a part that has more than one configuration, an additional button appears on the Modify dialog box, shown in Figure 10.6.

FIGURE 10.6
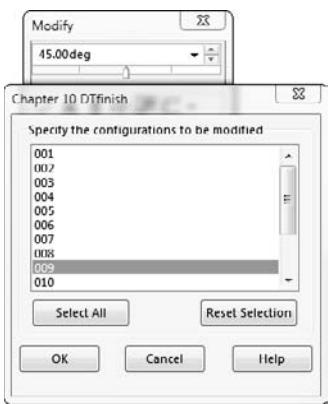
The Modify dialog box

Modify has three options for configuring dimensions: This Configuration, All Configurations, and Specify Configurations. The All Configuration is the default option. Choosing the Specify Configurations option opens a dialog box, which is shown in Figure 10.7.

FIGURE 10.7

Using the Specify Configurations dialog box to change a dimension in multiple configurations

Negative dimensions

SolidWorks 2009 brought into play the use of negative dimensions. Negative dimensions can only be used within the Modify dialog box, and cannot (or should not) be used in other places. The negative sign serves only to change the direction of the dimension, and then it is discarded. It does not stay with the dimension.

An equation that results in a negative dimension will flip the sense of the dimension every time it is rebuilt. This may be a useful trick, but will probably serve as an annoyance for most modeling. When you put a negative dimension in a Modify dialog box, the dimension changes sense (direction) and the negative sign disappears after one rebuild. If you put a negative dimension into a Modify Configurations dialog box, it will also disappear, and change the direction of the dimension for all configurations. When you enter a negative dimension into a design table, the negative is retained (until the next time you open the design table), and the sense of the dimension is retained only for the configs to which you assigned negative dimension values.

Note

This design table functionality is something that arouses my suspicion. I would not build a design intent scenario based on negative dimensions in design tables. The functionality seems unintentional, unstable, or otherwise subject to change. ■
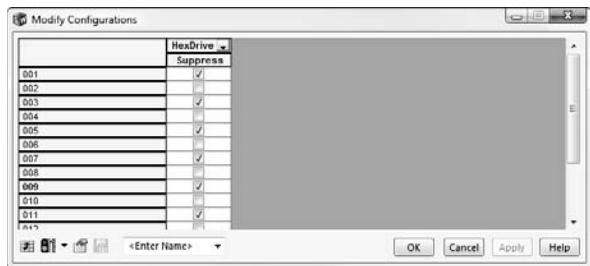
Negative dimensions can only be assigned to sketch dimensions, not to feature dimensions. You cannot change the extrusion direction by making the blind depth negative.

Using the Modify Configurations dialog box

The Modify Configurations dialog box, shown in Figure 10.8, enables you to create and modify configured features and dimensions in a more organized way than by using the simple manual methods described earlier, but without getting involved in an Excel-based design table, described later in this chapter. Do not confuse the Modify Configurations dialog box with the Modify dialog box, which is used to change dimensions.

FIGURE 10.8

Using the Modify Configurations dialog box to configure a feature

You can access the Modify Configurations dialog box by right-clicking a dimension or feature you want to drive via configurations and selecting Configure Dimension or Configure Feature.

With the Modify Configurations dialog box active, double-click a dimension to add it to the configured features list. You can add configurations on the fly by typing in the appropriate box, and change values or states of features by double-clicking and entering numbers or selecting the check box in the appropriate column.
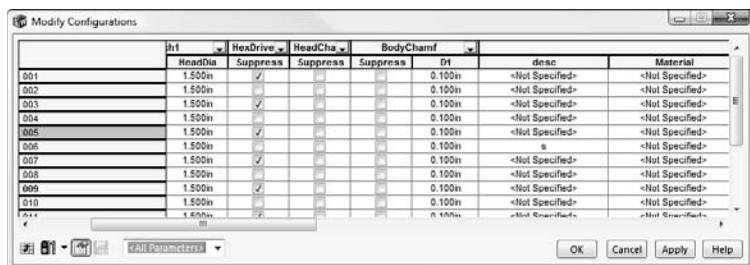
The Modify Configurations interface is still a relatively new part of the software. I still tend to use either the manual or Excel-based techniques. I have not yet found a use for the middle ground offered by the Modify Configurations dialog box, but I can see where it might be valuable for people who might want to configure a couple of features without getting involved in a big spreadsheet.

In addition, I should mention that much folklore exists surrounding what is perceived as a problematic relationship between Excel and SolidWorks. Some users claim that Excel often causes SolidWorks to crash. Beyond that, many workplaces may not have Excel available to them, either because of the cost or because they use a non-Microsoft solution for spreadsheet applications. These users still want the functionality of design tables even if Excel is not installed on their machines.

The icons in the lower left of the Modify Configurations dialog box enable you to use the dialog box for more than just dimensions and feature suppression. Figure 10.9 shows that you can also use it to control custom properties. Some of these changes are new in SolidWorks 2010, so even if you are familiar with this dialog box from a previous release, you might want to have another look at it to see whether there is some new functionality of which you can take advantage.

FIGURE 10.9

Using Modify Configurations to control dimensions, features, and custom properties

The Modify Configurations dialog box does not give you control over everything. Some things that you can configure, you cannot drive from this dialog box, such as part color. Design tables are still the most powerful way to go, but Modify Configurations offers a lot of flexibility and immediacy.

Using Table Views

The Modify Configurations dialog box adds significantly to your options for creating and editing configurations. SolidWorks 2010 adds two very nice tools that add to the convenience and power of SolidWorks configurations. The first of these two is called Table Views.

Table Views enable you to keep a small table of only the parameters that you want to show. The Parameters drop-down list enables you to select which Table View you want to display, or to create a new one. Using drop-down lists on the parameter headings, you can suppress the display of any parameter in a new Table View.
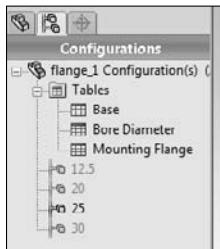
Dividing down what might otherwise be a large design table into several Table Views helps to keep your data organized and easy to access. Figure 10.10 shows the ConfigurationManager of a part with configurations managed with Table Views.

Displaying Table Views in the Configuration Manager of a configured part
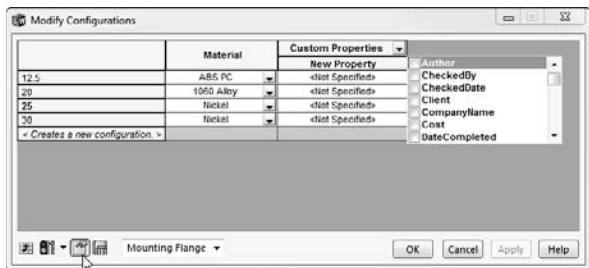
Adding custom properties with the Modify Configurations dialog box

The second nice option added in SolidWorks 2010 to the Modify Configurations dialog box is the Hide/Show Custom Properties option. When this button is depressed, the Modify Configurations dialog shows columns for existing custom properties, as well as a drop-down list that enables you to add more custom property columns. This makes it very easy to control configuration specific custom properties within a configured part.

Figure 10.11 shows these tools in action.

Using Modify Configurations to control configuration specific custom properties

Using Custom Property Managers

The Property Tab Builder enables an administrator to create special Task Pane tabs that are used to control custom properties and configuration specific custom properties. As an administrative function, the Property Tab Builder is covered in more detail in the *SolidWorks Administration Bible* (Wiley, 2009). Configuration-specific custom properties can be controlled through design tables, Custom Property tabs, or the regular SolidWorks configuration-specific custom property interface. Select the method that works best for the way you and your company use the software.

Figure 10.12 shows the Property Tab Builder interface. You can access the Property Tab Builder by clicking the Windows Start button and choosing SolidWorks ⇨ SolidWorks Tools from the menu. This is the place where you actually construct the interface that shows up in the Properties tab of the Task Pane on the right side of the SolidWorks graphics window.

FIGURE 10.12

Using the Property Tab Builder to construct a Custom Properties tab for the Task Pane
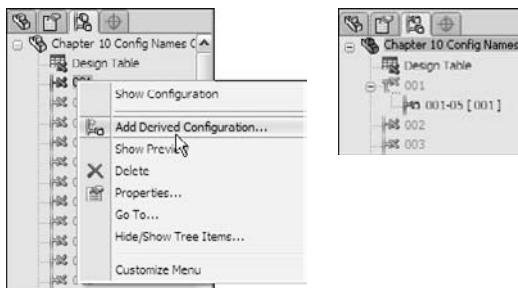
Using this type of interface enables the administrator to standardize custom properties values due to spelling or difference in interpretation. The Task Pane interface is easy to use and highly customizable. Notice that you can create regular custom properties or configure custom properties, and that option can be set independently for each property value entered using this interface.

Derived configurations

Derived configurations are configs that are dependent on other configs. You can create them from the RMB menu on a configuration, and they appear indented underneath the parent config. Figure 10.13 shows the RMB menu and the position of the derived config in the tree.

Creation and placement of the derived config

Derived configurations maintain the same values and properties of the parent config unless you break the link to the child config by explicitly changing a value in the child config. For all other values, the child config value changes when the parent config value changes.

One very nice application of derived configs is to use them for simplified configurations, and set the properties so that any features that are added to the parent config are also added to the derived config. You can do this by deselecting the Advanced Option Suppress Features to Off in the PropertyManager for the configuration. This causes the derived config to inherit *only* features that are added to the parent, and not to other configs. You can use the simplified configs for Finite Element Analysis (FEA), making drawings of models where all of the edge breaks have actually been modeled. You can also used use them for the reverse (a complex config rather than simple one) to have a config that includes fillets for rendering purposes that are otherwise not there. In addition, you can create and maintain derived configs using design tables, which are discussed in the next section.

File size considerations

A long-standing dispute has raged over the effects of file size on speed. Here are the facts: When SolidWorks creates a configuration, it stores information about the 3D geometry and a preview thumbnail of the configuration inside the part file. This makes it faster to access the configuration the next time because it has only to read the data, rather than read other data and then recalculate the new data. As a result, saving the stored data makes the file larger, but also enables you to avoid having to recalculate it.

Many people assign more importance to file size than I do, and use it as a criterion on which to base decisions about which features or techniques to use or not use. If I can use a single file instead of

multiple files by using configurations, I prefer the technique. Libraries of parts can often be made more manageable by using configured parts rather than many individual parts. Large hard drives for storage are cheaper and easier to upgrade than processors. In the end, reading stored data rather than recalculating it is faster. This is why when SolidWorks can store data that will probably be needed again at some point. The result is that configurations definitely increase file size.

The ideal situation would be if SolidWorks would give users control over which data is stored in model files. This is a good place for an enhancement request.

Performance

File size has a negative effect on speed when you are sending data across the Internet or working across a network. If the data is on your hard drive, then storing data instead of calculating it offers a big benefit. If you are sending data across a slow network connection, you should take measures to decrease the size of the file before sending it, such as using a zip or compression utility. ■
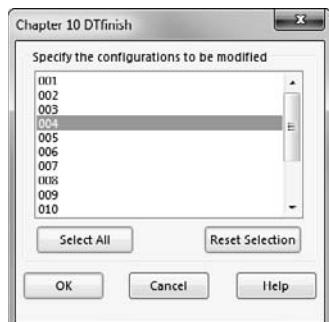
Controlling dimensions

Controlling dimensions with configurations is simple. You need only three things to start: one dimension and two configurations. Because you already know how to create these elements, you are ready to start. Configurations require that you spend some time developing "design intent" for parts. Configurations drive changes in models, and if they are properly modeled, you can avoid feature or sketch failures due to dimension changes.

I will start with the example of a simple block. A fully dimensioned block has three dimensions. Make sure that you have manually created at least two configurations. Double-clicking a model face opens all the dimensions, and double-clicking one of the dimensions opens the familiar Modify dialog box. Figure 10.6 shows that there is a small difference in the Modify dialog box. It now has a drop-down list where you can specify whether this change applies only to this config, to all configs, or to specified configs. If you select specified configs, then a dialog box shown on the top in Figure 10.14 appears, where you can select which configs this dimension change applies to.

FIGURE 10.14

Specify Configurations supplies a list of configs from which to choose

Once you are finished, you can toggle back and forth between the configs by double-clicking each of the configs in the ConfigurationManager. Although this is simple, if you forget to change the drop-down list from the All Configurations setting to either the This Configuration or the Specify Configurations setting, then you apply the change to all of the configurations. This shows that building a configuration manually is fine for a few simple changes, but it can become unwieldy if you are changing more than a few dimensions in this way. You would then have to remember which dimensions were changed to what. As you can see, using design tables is a better method for multiple dimensions.

Controlling suppression

Suppressing a feature is just like turning it off; the feature appears as grayed-out text in the FeatureManager. With configurations, you can suppress a feature in one config and unsuppress it in another. When dealing with manual configuration techniques, there are two methods for controlling suppression: manually suppressing features and creating configurations with the appropriate options for the inclusion of new features, which I discussed previously in this chapter.

In addition to the Suppress toolbar button, you can also use the Unsuppress and Unsuppress buttons with Dependents functions. When you suppress a feature, any feature that is dependent on it is also suppressed. If you then use the Unsuppress feature, it unsuppresses only the feature itself. However, Unsuppress with Dependents brings back all the dependent features, as well.

Performance

Suppressing complex features is a great way to improve performance. Experienced users often create a configuration of a part that they use as a simplified config, where patterns, fillets, and extruded text features are suppressed. This becomes more important as you start working with assemblies. For reasons I discuss in the assemblies chapters, it can be a great advantage to make a configuration for each part called "simplified." You can make a second configuration in template files so that new parts and assemblies automatically have this built in to them. ∎
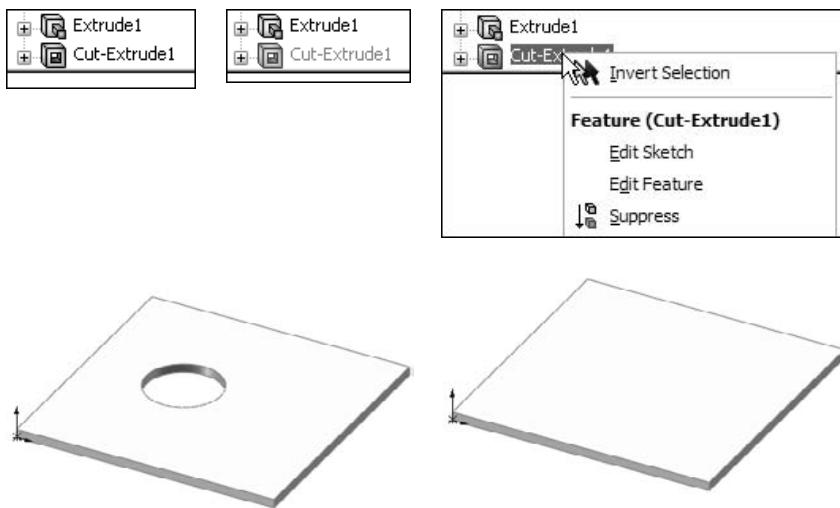
Generally, SolidWorks users employ a combination of these methods, mainly because configurations are not usually started on a complete model; they are often added when the model is still in progress, and so features are added after the users create the configurations.

On the left side of Figure 10.15, you can see a feature that is alternately unsuppressed and suppressed in the tree. The text and icon for the suppressed feature are grayed out. You can suppress features from the RMB menu on the feature, from the Edit menu, or through a tool on a toolbar. The Suppress button is not on a toolbar by default, but you can find it in the Commands dialog box (Tools ⇨ Customize ⇨ Commands), along with the other buttons for the Features toolbar. Only the Edit menu offers the options of Unsuppress With Dependents and This Configuration ⇨ All Configurations ⇨ Specify Configurations options for each of the Suppress, Unsuppress, Unsuppress With Dependents functions.

Using Unsuppress With Dependents can save you a lot of time or the hassle of looking for all the features dependent upon a feature that has been suppressed. Because it is not available on the RMB menus, this function is used less than it might otherwise be.

FIGURE 10.15

Suppressing a feature

Controlling custom properties

Several reasons may compel you to use custom properties, including integration with searches for a Product Data Management system, automatically filling out drawing title blocks, or adding information to the BOM.

When you are using custom properties with configurations, you must use the Configuration Specific Custom Properties interface (or an appropriately configured custom property tab or the Modify Configurations dialog box), which enables you to have custom properties that change with each configuration. Standard custom properties apply to the top-level part and keep the same value for all configurations. The configuration-specific functionality is useful for situations such as different part numbers for configurations, and many other situations that are limited mostly by your use of configs. The Custom tab of the Summary Information dialog box, shown in Figure 10.14, still applies custom properties that do not change with the configurations to the part.
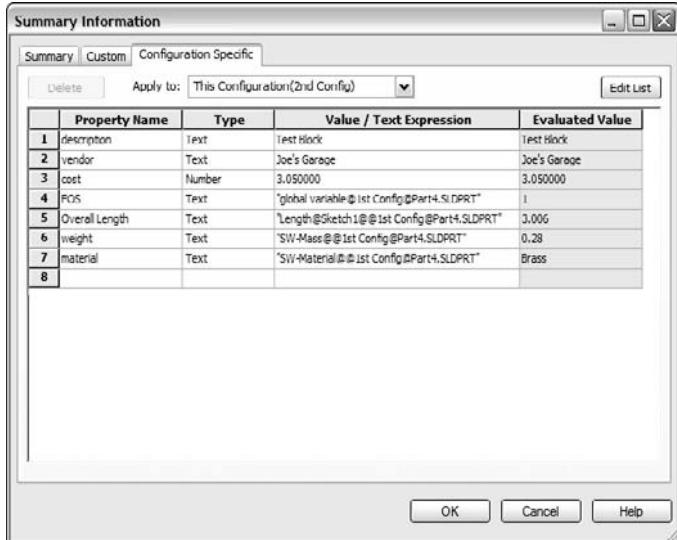
The interface for managing custom properties manually is shown in Figure 10.16. You can access this dialog box by choosing File ➪ Properties from the menu.

You can also link custom properties to mass properties, model dimensions, link values, sketch text, and global variables by selecting from the drop-down list under the Value/Text Expression column, which appears when you select a cell in the column, as shown in Figure 10.14. To link a custom property to a model dimension, simply place the cursor in the Value/Text Expression box that you want to populate and click a dimension in the graphics window. Again, managing this data for a single config or only a few configs is easy enough; however, it can quickly become unwieldy, which is where using design tables can make a huge difference.

FIGURE 10.16

The Configuration Specific tab in the Summary Information dialog box

Controlling sketch relations

You can individually suppress or unsuppress sketch relations using configurations. Figure 10.17 shows the Display/Delete Relations PropertyManager interface, at the bottom of which is the Configurations panel. To suppress a relation, select it from the list and select the Suppressed option in the Relations section above the Delete buttons.

Tip

This is another situation where Delete is not used as an editing option. Using this technique, you can save sketch relations, or activate different sets of relations in different configs; this technique allows a single sketch to react to changes differently. ■
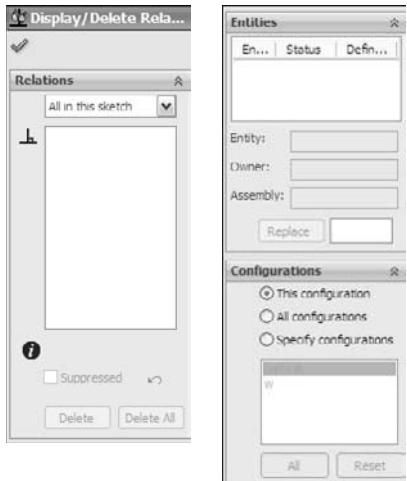
Controlling sketch planes

You cannot configure the Offset distance in the From option for extrudes, but you *can* configure the sketch plane for the sketch that is used in the feature. The Sketch Plane PropertyManager interface expands when configurations are present, as shown in Figure 10.18.
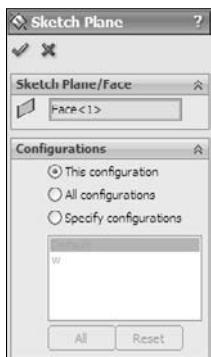
FIGURE 10.17

The Display/Delete Relations dialog box for configuring sketch relations

FIGURE 10.18

The Sketch Plane PropertyManager interface for configuring a sketch plane

Tip

Another way to change the sketch plane is to put the sketch on an offset plane or a plane that can otherwise be driven by a dimension (for example, using reference sketch geometry). Actually moving a sketch to another plane can cause the sketch to rotate or flip. Moving the plane it is on is a better option that does not cause the sketch to rotate or flip. ■

Caution

Changing sketch planes indiscriminately can have serious consequences for your model. "Face/Plane Normals" sometimes point in different directions, and can cause a sketch to flip, rotate, or mirror when you change it from one plane to another. One strange result is that changing it back to the original location can cause the sketch to flip again, but in a different way so that it does not go back to its original location/orientation. As a result, every time you change the configuration, the sketch could appear in a new and unexpected location or orientation. ■

Controlling configurations of inserted parts

Inserted parts have a long history in SolidWorks. They have had several names in the past, and some sources (including SolidWorks documentation such as training documents and even help files) still use some of these legacy names out of habit or precedence. For example, you will sometimes hear inserted parts called *derived* or *base* parts. Both of these terms are obsolete.

Cross-Reference

Inserted parts are discussed in detail in Chapter 28, which describes master model techniques. ■

Inserted parts use one part as the starting point for another part. The inserted part sits as a feature in the FeatureManager of the child part. You can insert just the body geometry itself, or you can bring forward reference geometry, sketch data and all features, and break the link to the original part if you wish to.

The role of configurations with inserted parts is that the configuration of the inserted part can be controlled from the child component. For example, you may have designed an engine block for an automobile. This engine block is a casting, and using configurations, you have both the six-cylinder and the eight-cylinder blocks in a single-part file. This model represents the "as cast" engine block. The next step is to make the block with all the secondary machining operations, such as facing mating surfaces, boring cylinders, drilling and tapping holes for threaded connections, and so on. As a result, the as-cast part is inserted into the as-machined part, and the configuration is selected before you add the cut features. As the name suggests, you add inserted parts by choosing Insert ⇨ Part from the menu.

The interface for assigning the configuration is shown in Figure 10.19. Simply right-click the inserted part feature and select List External References. It would seem to make more sense if the configuration could be selected when the part is first inserted, but it does not work this way; you have to select the configuration after the part is inserted.

Library features
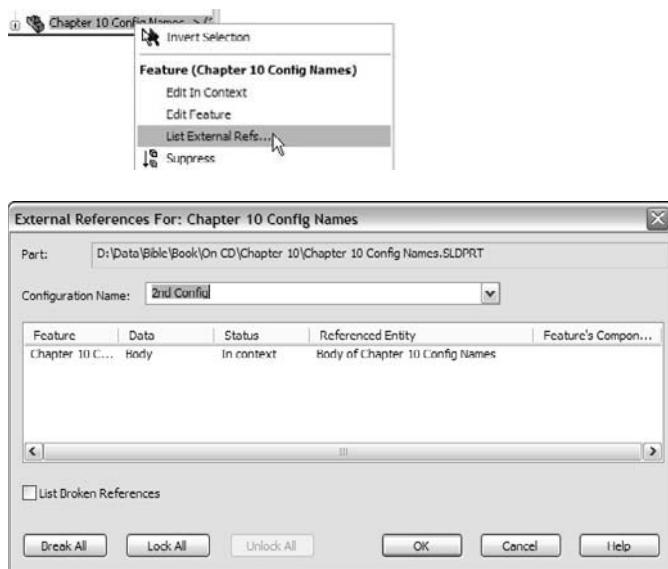
Library features can have configurations, and they carry those configurations with them into the part in which they are placed. Unfortunately, part configs cannot reference different library feature configs.

Cross Reference

Chapter 18 discusses library features. ■

Assigning the configuration of an inserted part

Configurations for library features are created in exactly the same ways that configurations are created for other parts. The technique for saving the configs to the library feature is discussed in Chapter 18.

Unconfigurable items

As important as it is to know what you *can* do, it is equally important to know what you *cannot* do. The following is a list of items that are not configurable. Although this list is not complete, it contains many of the more relevant items that cannot be configured:

- Library feature configs
- Blocks
- Extrude direction or From Offset dimension or direction
- Most of the values in features such as Deform, Freeform, and Twist

While Library Features can be configured, once you drop them into a part and click OK, the configurations are no longer accessible (unless you have selected Link To Library Part), so a part's configurations cannot select the configuration of a Library feature. A part's configurations can change the dimensions of a library feature.

Using Design Tables

In addition to describing some of the basic concepts involved with configurations, the first part of this chapter presents reasons for using design tables. For example, while manual configuration management can be haphazard, and is highly prone to mistakes, design tables lay everything out in an Excel spreadsheet. Although many new users ask whether they can use a different replacement spreadsheet program, you must use Excel for design tables.

Note
The versions of Excel that are supported by SolidWorks for design tables are 2000, 2003, and 2007. Although Excel 97 may still work, Microsoft no longer supports that product. ■

Excel is a format that is easy to read and print out, and even non-SolidWorks users can understand and work with it. Although there is some special syntax that you need to use with design tables, for most uses, SolidWorks can create the syntax automatically for you; therefore, there is minimal manual data entry. If you are careful to name dimensions, features, and configurations properly, design tables should be easy to understand and manage. In Excel, you can also color cells, rows, and columns in such a way that large amounts of tabulated data are easier to sort through. In addition, because design tables use Excel, they can also use all of Excel's calculation capabilities.

Best Practice
When using equations and design tables, it is considered best practice to name dimensions, sketches, features, and other configured items. However, it is not recommended to mix design tables with SolidWorks equations. Besides the fact that Excel equations are far more sophisticated than those of SolidWorks, driving dimensions from too many locations can be confusing when you edit the part after you have forgotten the details of how the part was built.

It is a great idea to document design intent using comments in the features or the Design Journal. You should also add comments to design tables as needed. ■

What Can Be Driven by a Design Table?

Just because something can be configured does not necessarily mean that it can also be driven by a design table. Here is a small list of items that fit into this category:

- Sketch plane configuration
- Suppressed sketch relations
- Suppressed dimensions (suppressed dimensions become driven dimensions)

However, the good news is that there are many items that can be driven by a design table. Table 10.1 lists these items, along with their associated syntax.

TABLE 10.1

Items That Can Be Driven by a Design Table
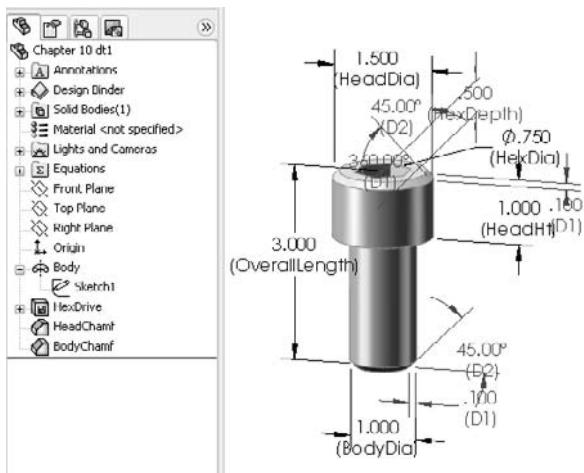
Item	Syntax (Goes in Column Header)	Possible Values (Goes in Field Cell)	Default Value If Field Is Blank
Configs of Inserted Parts	$configuration@ <part name>	<config name>	not evaluated
Configs of Split Parts	$configuration@ <split feature name>	<config name>	not evaluated
Comment Column	$comment	comment text	blank
Configuration Description	$description	description text	<config name>
BOM Part No.	$partnumber	$d, $document = document name $p, $parent = parent config name $c, $configuration = config name <text> = custom name	config name
Feature Suppression State	$state@<feature name>	suppressed, s unsuppressed, u	present suppression state
Dimension Value	dimension@<feature name> dimension@<sketch name>	allowed numerical values	not evaluated
Parent Config (creates a derived config)	$parent	parent config name text	not evaluated
Config Specific Custom Property	$prp@<property name>	property name text	not evaluated
Equation State	$state@<equation number>@equations	suppressed, s unsuppressed, u	unsuppressed
Light Suppression State	$state@<light name>	suppressed, s unsuppressed, u	unsuppressed
Sketch Relation Suppression	$state@<relation name> @<sketch name>	suppressed, s unsuppressed, u	unsuppressed
User Notes (same as comment)	$user_notes	Text	blank
Part or Feature Color	$color $color@<feature name>	see SolidWorks Help, Colors, Parameters in design tables	0, black
Assigned Mass	$sw-mass	allowed numerical values	value from Mass properties
Assigned Center of Gravity X, Y, Z Coordinates	$sw-cog	allowed numerical values in the format of x, y, z	value from Mass properties
Dimension Tolerance	$tolerance@ <dimension name>	see SolidWorks Help, Tolerance Keywords, and Syntax in Design Tables	none

Creating a simple design table

When you prepare to create a design table, you generally need to give appropriate names to dimensions, sketches, and features. Remember that while the feature is the most visible item and the easiest to rename, most of the dimensions probably belong to the sketch, which you may also need, or want, to rename. Names should reflect the function or location of the item. It is a good idea to show dimension names when renaming items (remember that you can show dimension names by selecting the Dimension Names option by choosing View ⇨ Dimension Names). Figure 10.20 shows the result of renaming the feature and dimension.

Renamed features and dimensions

You can use one of the following three techniques to add a design table to a SolidWorks part by choosing Insert ⇨ Tables ⇨ Design Tables from the menus:

- **Insert Blank Design Table.** This method starts from a blank template that contains the underlying framework, but no values.

- **Auto-create Design Table.** This method populates the new design table with any existing configurations and items that are different between the configs.

- **From file.** This method enables you to create a design table externally and then import it.

Although I prefer the Auto-create method, it is most appropriate for when you have existing configurations. The From file method is best when a design table has been exported from another part, saved externally, and brought into the current part. For the following example, I am using the Insert Blank Design Table method.

On the CD-ROM

If you would like to follow along with these steps to create the design table, you can use the part from the CD-ROM with the filename `Chapter 10 DTstart.sldprt`. ∎

Figure 10.21 shows the results of starting with the new blank design table. You may notice that the window title bar at the top says SolidWorks, but the toolbars look a lot like the Excel interface. This is because Excel is actually running inside of SolidWorks. Clicking outside of the Excel window can cause the Excel window to close, although there are several items outside of the Excel window that you can select without the window closing, such as features in the FeatureManager and dimensions in the graphics window. You can also rotate and pan the view in the graphics window without closing the Design Table window. If you are very careful, you can also drag the thin hatched border of the Excel window to adjust its size or location.

You can also edit Design Tables in a separate window, which makes editing easier, but makes adding dimension and feature names more difficult. To edit the table in its own window, right-click on the Design Table in the FeatureManager and select Edit Table In New Window.

FIGURE 10.21

The interface where you can create the design table, and the resulting blank design table
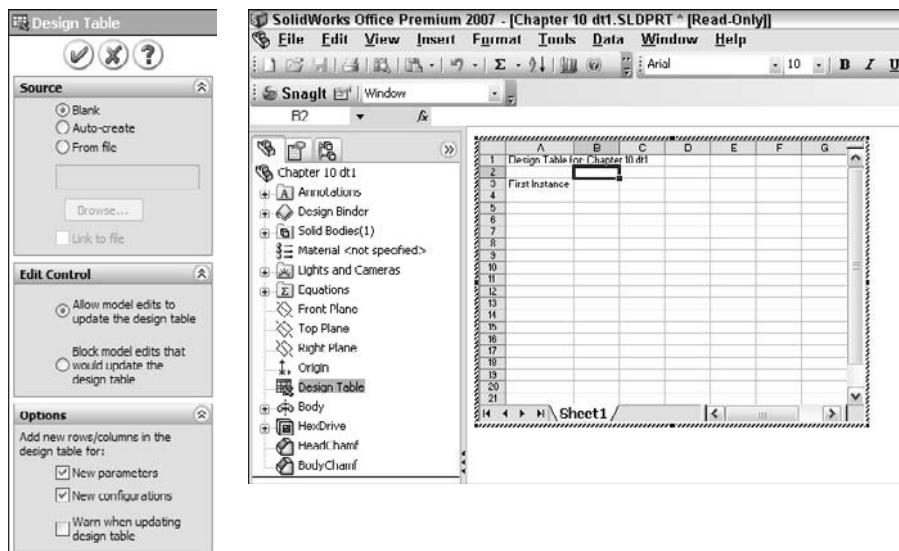
Figure 10.22 shows a fully developed design table, with some complexity. Although your first design table does not need to be this complex, this example demonstrates what you can do with this feature.

FIGURE 10.22

A fully populated design table

Config Names   Feature or Property Syntax from Table

The config names go in the first column and the feature or property names go in the second row. The first row is reserved for the name of the table. All this is automatically set up by SolidWorks.

Note

Because you are actually working in Excel when working with design tables, you can use Excel formatting, which is how the text in Figure 10.20 is rotated 90 degrees for the column headers. (To rotate text in a table, right-click the cell, group of cells, or row; select Format Cells; and then select the Alignment tab.) ∎

In the new design table, the next step is to type in some configuration names. Because you are working in Excel, all the fill functionality is available. In the example shown in Figure 10.23, I have typed in the first three values of 001, 002, and 003, then window-selected the cells, and dragged the fill handle on the selection window to fill the number pattern to populate a larger area. To find more information about this technique, look for Fill or Automatically Number Rows in the Excel Help files.

The next step is to fill in some feature and dimension names in the second row. The first thing that you do is to suppress the HexDrive feature. To make this the first feature in the list, click in cell B2, and then double-click the HexDrive feature in the FeatureManager. The name of the feature and its current suppression state are added to the design table with all the necessary syntax and correct spelling.

FIGURE 10.23

Filling in configuration names

To rotate the text in this row vertically, right-click row number 2, select Format Cells, click the Alignment tab, and turn the orientation to 90 degrees. The word *unsuppressed* displays in all capitals and fully spelled out, while all you need is a U or an S. Replace the word with an S, and double-click the line between the column heading letters B and C at the top of the Excel window to condense column B as much as possible. Alternate the rest of the rows between *U*s and *S*s to either suppress or unsuppress the HexDrive feature in various configurations. Figure 10.24 shows the current state of the design table.

Close the Design Table window, and click OK on the message box that lists the new configurations created by the Design Table. Now split the FeatureManager, set the lower pane to the ConfigurationManager, and double-click some configurations. Notice that in the configs where you specified an S, the HexDrive is suppressed and no longer appears in the model.

You can now add a dimension to the design table. To add a dimension, it is most convenient to display the dimensions on the screen at all times. To show all the dimensions in the part, right-click the Annotations folder in the FeatureManager and select Display Annotations. If the dimensions do not display, you may have to go back and select Show Feature Dimensions. Arrange the dimensions so that you can clearly see them all, as shown in Figure 10.25.
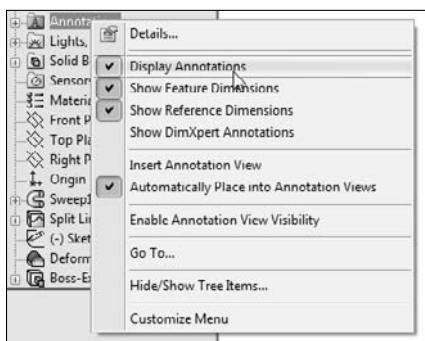
FIGURE 10.24

Building the design table

FIGURE 10.25

Dimension and annotation display settings

To display the design table again, locate it in the ConfigurationManager, right-click it, and select Edit Table. Editing the feature changes the settings used for the design table. Edit Table in New Window is an option that you will use later because it simplifies many things; however, for now, the Edit Table option makes it easiest to add new items to the design table.

Note

If a window appears with the name Add Rows and Columns, just click OK for now. This window lists parameters that have changed in its lower pane, and it is asking you if you would like to add any of the changed parameters to the design table. If you would like to add them, just select the parameter in the lower pane and click OK. If not, just click OK. ∎

If the design table displays on top of your model, you can either move the model or move the design table. Moving the design table is a bit tricky and involves dragging the striped-line border of the Excel window; remember not to grab it at the corners or midpoints, because this will simply resize it. If you click inside the border, nothing happens. If you click outside of the border, the Excel window closes. Moving the model may be easier. To do this, Ctrl+drag in blank space in the graphics window; it pans the display so that you can see the part dimensions.

With cell C2 selected, or whatever the next available cell is in the second row, double-click the OverallLength dimension in the graphics window. SolidWorks adds the proper syntax to the design table, along with the current value for the first configuration in the list. Fill in values for the rest of the configurations. You can then calculate these values in Excel using any of the available techniques.

Exit the design table and toggle through the various configurations to see their different lengths. These examples should get you started on more complex configurations and design tables. Any dimensions that are controlled by the design table (and that are therefore locked) display in pink on the screen.

Design table settings

Figure 10.19 shows the PropertyManager for design tables. After you have created the table, you can edit the table settings by right-clicking the table and then selecting Edit Feature. Edit Feature enables you to edit the settings for the table only; it does not enable you to edit values within the table.

Linked design table

By selecting the From File source option, you can create a design table from an external file; you can also link the table to the external file. When you use the other two options, Blank and Auto-Create, SolidWorks stores the Excel file within the SolidWorks document. Linking to an external file may be useful if you have a non-SolidWorks user who is entering data into the design table, or if a single table controls multiple parts.

Edit control

The Edit Control panel has two options, which act as a toggle. The Allow model edits to update the design table option is self explanatory, as is its opposite, the Block model edits that would update the design table option. If the Allow model edits option is selected, and you make a manual change to the model, the next time you open the design table, SolidWorks warns you about the change and that it will update the design table. Likewise, if you try to make a manual change and the Block model edits option is selected, you receive a warning that the value cannot be changed.

Options

The Options settings determine the behavior when you are using the Allow model edits option and a new item has been configured. For example, the design table may already exist and you manually add a configuration and suppress a feature.

Configurations that have been added manually are displayed somewhat differently from configs that are being managed by the design table. Figure 10.26 shows the two configurations at the bottom of the tree with square symbols, while the design table configs have Excel symbols.

Manually created configs versus design table–created configs

After you manually add the config and suppress the feature, the next time you open the design table, the Add Rows and Columns dialog box appears. Many users are simply annoyed by this, but that may be because they do not understand what it does or why it appears. In the example shown in Figure 10.27, a new configuration has been manually added; it appears in the Configurations box as ManuallyAddedConfig, and in the Parameters box, it looks like a feature named BodyChamfer has been either suppressed or unsuppressed manually. The appearance of this dialog box means that SolidWorks is asking you if you would like to include these items in the design table. If so, simply select the items you would like to add to the design table and click OK. If you do not want to include the items in the design table, then simply click OK or Cancel. If you click OK, you will not be offered these choices again; if you click Cancel, the next time you open the table, the dialog box with the same choices will reappear. If you never want to see this dialog box again, then make sure that all the options in the Options panel shown in Figure 10.21 are deselected.

Editing the design table

As I mentioned earlier, when you open the design table inside the SolidWorks window, it can sometimes be difficult to work with. One way to handle this problem is to only edit the design table inside SolidWorks when you want to add new features to the column headers, and when adding new configurations or editing the field values, edit the table in a separate window. This option appears on the RMB menu as Edit Table in New Window. It gives you much more flexibility in resizing the Excel window, changing zoom scale, and other operations, but it does not enable you to double-click a dimension so that it is added automatically to the column header.

FIGURE 10.27

The Add Rows and Columns dialog box

Caution

When working on design tables, it is a good idea to avoid conflicts with other sessions of Excel by closing any other Excel windows. The combination of operating Excel spreadsheets inside both SolidWorks and Excel has been known to cause crashes, or the "Server Busy" warning message. If you are diligent about having only one session of Excel active at a time when you are working on design tables (or Excel BOMs), there is less likelihood of a crash or conflict. ■

Using the Configuration Publisher

The Configuration Publisher is the tool formerly known as Create PropertyManager. The Configuration Publisher enables you to create an interface that creates configurations on the fly based on rules that you establish. The interface that you create appears when you put the part into an assembly, enabling you to create a custom size and a new configuration to go with it. This is similar to putting a Toolbox part into an assembly, and getting a special interface to specify sizes and create new configurations if necessary. In order to create a Configuration Publisher for a part, the part must contain a Design Table with at least a single row. You can use an auto-created Design Table if you need to.

You access the Configuration Publisher by right-clicking the name of the part at the top line of the ConfigurationManager, and selecting Configuration Publisher from the menu.
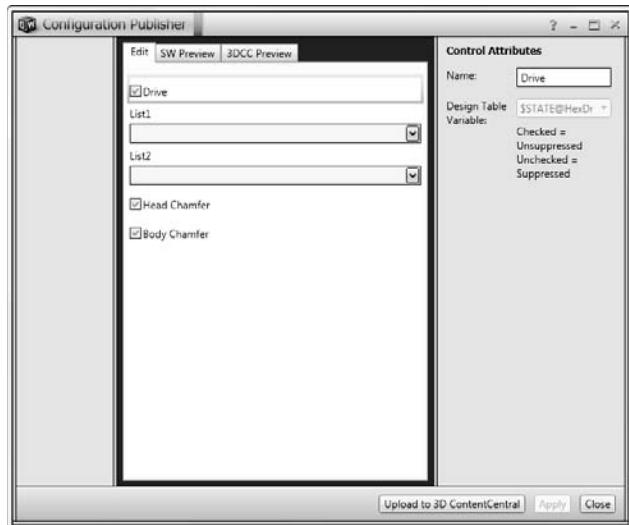
Figure 10.28 shows the Configuration Publisher interface. It is like the Property Tab Builder in that you use it to place interface controls on a PropertyManager that will pop up when the part is placed into an assembly.

The Configuration Publisher helps you build an interface to create configurations as you place a part into an assembly
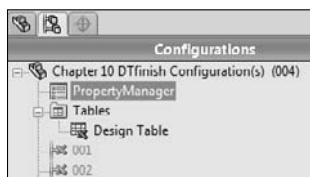
Aside from users putting parts into assemblies, the interface you create using the Configuration Publisher can also be used in 3D Content Central to specify a new component size. If you are a supplier and make parts in a wide variety of sizes, instead of creating all the sizes and uploading to 3D Content Central, you can just create the rules in the Configuration Publisher interface and allow the configurations to be created automatically according to the rules.

When the interface has been created, a PropertyManager entry shows up under the name of the part in the ConfigurationManager, as shown in Figure 10.29.
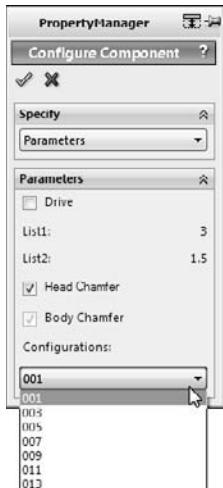
The PropertyManager entry in the ConfigurationManager

When the part is placed into an assembly, the PropertyManager shown in Figure 10.30, which is the one created in the Configuration Publisher, appears and enables you to access options to size and configure the new instance. A similar interface is also used if you have uploaded the part to 3D Content Central to allow other people to download the part.

FIGURE 10.30

A custom PropertyManager makes it easy to size and configure a part as it is inserted into an assembly.

Tutorial: Working with Configurations and Design Tables

Throughout this book, parts that I use for one purpose may also be useful for other purposes. For example, the part used in this tutorial uses a loft with guide curves where both guide curves are created in the same sketch. The guide curve sketch is made from symmetrical splines where I have used the spline handles to change the shape smoothly and in a controlled way. I have also used a curve-driven pattern to go around an elliptical shape.

Tip

If at some point, you decide that you have made mistakes from which you cannot recover, or you would simply like to start over again, you can choose File ⇨ Reload. This is the same as exiting the part without saving, and then reopening the part to start from the beginning. ■

To start working with configurations and design tables, follow these steps:

1. **From the CD-ROM, open the part called** `Chapter 10 Tutorial start.sldprt`. Take a moment to become familiar with this part by using the rollback bar to see how it was made. In particular, look at the two patterns, which need to be parametrically linked. Figure 10.31 shows the part.

The `Chapter 10 Tutorial start.sldprt file`

2. **Manually create a configuration for the part called Size 1.** Remember that to create a configuration, you must show the ConfigurationManager tab in the FeatureManager area, and right-click the name of the part at the top level. It is better to do this by splitting the FeatureManager window and setting the lower pane to the ConfigurationManager.

3. **Set the Advanced option by selecting both Suppress Features and Use Configuration Specific Color.**

4. **Before closing the Add Configuration PropertyManager, click the Color button on the Advanced Options panel of the Configuration PropertyManager and select a different color for the Size 1 configuration.** The color does not change immediately. It will change after you close the PropertyManager.

5. **Choose View⇨Dimension Names.**

6. **Double-click the feature CrvPattern1 in the FeatureManager.** A number 6 with a D1 under it will appear on one of the holes in the pattern. If you have changed your part to a blue color, it may be difficult to see because the text will also be blue.

7. **Change the name of the dimension to Hole# by clicking the dimension and using the PropertyManager.**

8. **Change the value of the number to 8, and be careful to also change the drop-down setting to This Configuration Only instead of All Configurations.** If you forget to do this, then you will have to go to the other configuration and set it back to 6.

9. **Click the Rebuild symbol (which resembles a traffic light) to show the changes before exiting the Modify dialog box.** Notice that the CrvPattern2 fails after rebuilding CrvPattern1 with eight instances. Click the green check mark icon to exit the Modify dialog box, and then make the same changes to the CrvPattern2: change the dimension name and the number of patterned instances to eight (remember to use the This Configuration Only setting). The part should now look like Figure 10.32.

FIGURE 10.32

The model after Step 9

10. **When you double-click to change configurations, the SolidWorks interface now shows a part with a different color and a different number of holes and ribs.** After the first change between configurations, the changes should happen quickly because SolidWorks has stored the geometry.

11. **Choose File ⇨ Properties and select the Configuration Specific tab.** Set the Apply To drop-down list to Default, and type a Property Name of **description** and a Value of **Gray Vent Cover**. Now change the Apply To drop-down setting to Size 1 and type a description for the new configuration using the name of the color that you applied to this config. Figure 10.33 shows the two states of the data.

12. **Exit the Custom Properties dialog box.** Now that you have made a few changes manually, the following steps guide you through bringing these changes into a design table and using the design table to make additional changes.

FIGURE 10.33

Setting the Configuration Specific custom properties

13. Choose Insert ⇨ Tables ⇨ Design Table from the menus. Use Auto-create as the Source, allow model edits, and select all three options in the Option panel. Click OK to create the design table. Figure 10.34 shows the design table that you have automatically created.

FIGURE 10.34

The automatically created design table

14. Use the striped border to move the window without closing it. This may take some practice. If the window closes, right-click the design table in the FeatureManager and select Edit Table. Move the window to a place where you can see the model clearly.

15. **If a cell in the second row of the design table is selected, select a different empty cell that is not in the second row (this prevents data from automatically populating cells until you have the correct data).** Now double-click the Extrude1 feature in the FeatureManager. Find the .500" (D1) dimension on the screen. Right-click the dimension and rename it **BaseThk**.

16. **Click the next open cell in the second row, and double-click the .500" dimension that you just renamed.** You may have to use the handles at the corners and side midpoints to resize the Excel window to see everything. Add another configuration row and the additional values in the cells, as shown in Figure 10.35. The color number is determined by a formula that you can find in the help section under the topic Color Parameter.

FIGURE 10.35

Make additions to the design table.

	A	B	C	D	E	F	G
1	Design Table for: Chapter 10 Tutorial start						
2		$DESCRIPTION	$COLOR	$PRP@description	Hole#@CrvPattern1	Rib#@CrvPattern2	BaseThk@Extrude1
3	Default	Default	12632256	Gray Vent Cover	6	6	0.5
4	Size 1	Size 1	255	Red Vent Cover	8	8	0.8
5	Size 2	Size 2	16711680	Blue Vent Cover	10	10	1
6							

17. **Remember that this part needs to have the number of ribs always equal to the number of holes.** This is simple to do in Excel. Click in the first row value for the Rib# number. This is cell F3 in Figure 10.34. Type the equal sign and then click in the cell to the left, E3. You can also simply type **=E3** in this cell. This links the Rib# cell to the Hole# cell.

18. **Use the Window Fill feature by selecting the dot at the lower-right corner of the selected F3 cell and dragging it down to include cells F4 and F5, as shown in Figure 10.36.**

19. **Click in a blank space to exit the design table.** Double-click through the configurations in the ConfigurationManager to see the results of your efforts.

FIGURE 10.36

Copying the equation to other cells

Drag this handle

Summary

Configurations are a powerful way to control variations of a design within a single part file. Many aspects of the part can be configured, while a few cannot. Manually created configurations are useful for making a small number of variations and a small number of configurations, but they become unwieldy when you need to make more than a few variations of either type.

Design tables are recommended because they enable you to see all the changes more clearly that have been made for all the configurations. Having the power of Excel available enables you to access many functions that are not shown here, such as using lookup tables and Concatenate functions to build descriptions or configuration names. I briefly revisit design tables in Chapter 14 on assembly configurations to expand on the information here and to incorporate the additional assembly configuration information.

Editing and Evaluation

When you use CAD programs, you typically create a part once, but edit it many times. *Design for change* is at the core of most of the modeling work that you will do in SolidWorks, and deletion is not an editing option.

The initial stages of modeling are the most crucial. This is when you set up parametric relations between the features and sketches that form the foundation for an assembly or complex part. For this reason, editing often quickly turns into repair. Granted, some changes are simply unavoidable, but a thorough knowledge of editing — and repairing — can help you to understand the how, what, and why of modeling best practice.

This chapter starts with some very basic concepts of editing, which you may have picked up if you have been reading this book from the beginning. Because this is the last chapter in Part II, and the last that deals strictly with part modeling, it also contains a summary of part modeling best practice techniques and a set of model evaluation tools that can help you evaluate the manufacturability and aesthetic properties of parts. I have included these evaluation tools in a chapter on editing because the create-evaluate-edit-evaluate cycle is one of the most familiar in modeling and design practice.

Using Rollback

Rolling back a model simply means looking at the results of the design tree up to a certain point in the model history. In SolidWorks, you can actually change history — that is, you can change the order in which operations are completed. The order in which you create features is recorded, and if you change this order, you get a different geometric result.

You can use several methods to put the model in this rolled-back state:
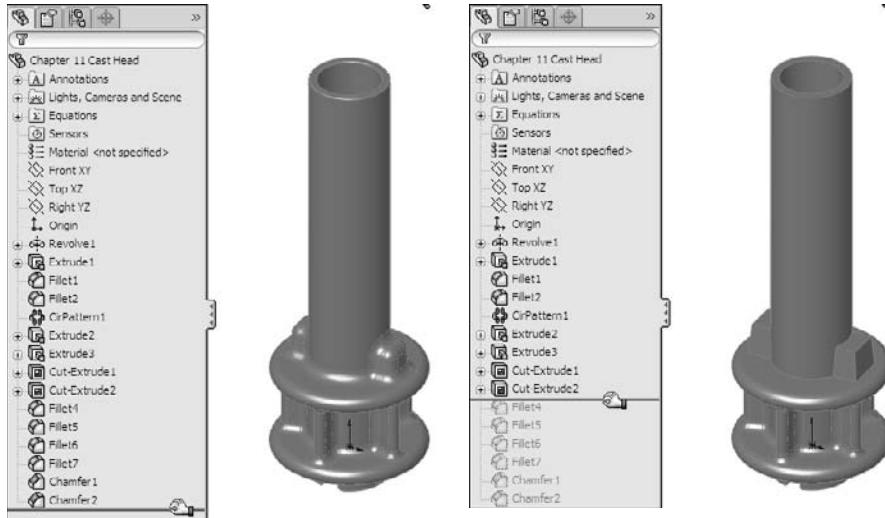
- Dragging the Rollback bar with the cursor.

- Right mouse button (RMB) clicking and selecting one of the Rollback options.

- Editing a feature other than the last one in the design tree. (SolidWorks rolls back the model automatically.)

- Choose Tools ⇨ Options ⇨ FeatureManager ⇨ Arrow key navigation to control the Rollback bar with the arrow keys.

- Saving the model while editing a feature or sketch, and then exiting the model. When the part is opened again, it is rolled back to the location of the sketch that was being edited.

- Pressing Esc during a long model rebuild. This method is supposed to roll you back to the last feature that was rebuilt when you pressed Esc; however, in practice, I have rarely seen it do this, and it usually rebuilds the entire model anyway.

Using the Rollback bar

The Rollback bar, which typically appears at the bottom of the FeatureManager in SolidWorks part documents, enables you to put the part into almost any state in the model history. This is not the same as the Undo command, but is the equivalent of going back in time to change your actions, and then replaying everything that you did after that point. Figure 11.1 shows the Rollback bar in use. Notice how the cursor changes into a hand icon when you move it over the bar.

FIGURE 11.1

Using the Rollback bar

Consumed features
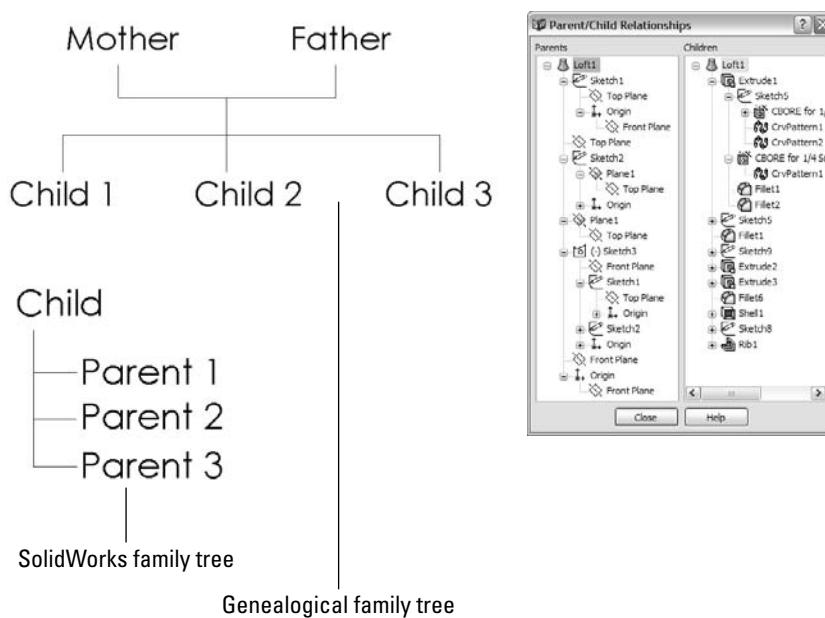
When you use a sketch for a feature such as a Sketch Driven Pattern, the sketch is left in the design tree, in the place where it was created. However, most other features, such as extrudes, *consume* the sketch, meaning that the sketch disappears from its normal order in the FeatureManager and appears indented under the feature that was created from it. Consumed sketches are sometimes also referred to as *absorbed* sketches.

Examining the parent-child relationship

In genealogical family tree diagrams, the parent-child relationship is represented with the parent at the top, and the children branched below the parent. In SolidWorks, parent-child relationships are tracked differently. Figure 11.2 shows the difference between a genealogical family tree and the SolidWorks design tree.

Different interpretations of the structure of parent-child relationships

You can display the parent-child relations between SolidWorks features, as shown in Figure 11.2, by right-clicking on any feature and selecting Parent/Child. This helps you determine relationships before you make any edits or deletions because you can see which features will be removed or go dangling (lose their references).

When SolidWorks puts the child feature at the top, it is, in effect, turning the relationship upside down. In the SolidWorks FeatureManager, the earlier point in history is at the top of the tree, but the children are listed before than the parents. The SolidWorks method stresses the importance of solid features over other types of sketch or curve features.

For example, you create an extrude from a sketch, and, therefore, the sketch exists before the extrude in the FeatureManager. However, when you create the extrude, SolidWorks places the sketch underneath the extrude. This restructuring can become more apparent when a sketch (for example, Sketch1) is created early in the part history, and then not used to create a feature (for example, Extrude5) until much later. If you roll down the FeatureManager feature by feature, you arrive at a point at the end of the design tree where Extrude5 appears and Sketch1 suddenly moves from its location at the top of the tree to under Extrude5 at the bottom of the tree.

This scenario may cause a situation where many sketches and other features that are created between Sketch1 and Extrude5 are dependent on Sketch1, but where Sketch1 suddenly appears after all these other features. This can be difficult to understand, but is key to effectively editing parts, especially parts that someone else created.

The main point here is that SolidWorks displays many relationships upside down. You need to understand how to navigate and manage these history-bound relationships.

One way to get around difficulties in understanding the chronological order of features when compared against the relationship order of features is to roll back a model tree item by item. This can help you sort through the issues. Also remember that from SolidWorks's point of view, the solid feature is the most important item in the tree and is what the rest of the items in the tree support. SolidWorks has made the solid features easily visible and accessible in the tree.

Rolling back features with multiple parents
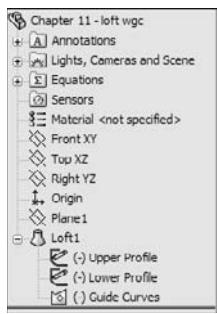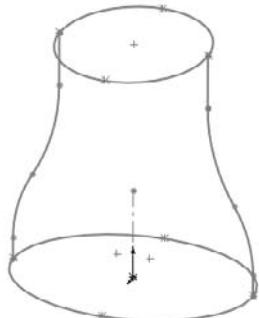
Take an example such as a loft with guide curves. If you create the guide curves first, and then you create the loft profiles by referencing the guide curves, the loft automatically reorders these sketches when they display under the loft feature such that the profiles are listed in the order in which they were selected, followed by the guide curves in the order in which they were selected. This is shown in Figure 11.3. This restructuring can be confusing if you want to go back and edit any of the relationships between the sketches. You can find this example on the CD-ROM with the filename `Chapter 11 Loftwgc.sldprt`.

Note
In this example, the two guide curves were created as part of a single sketch, and the SelectionManager was used to select them as individual open curves. This is why the Guide Curves sketch is represented with the contour symbol rather than a regular sketch symbol. ■

FIGURE 11.3

Multiple parents and sketch reordering

Viewing consumed features in their original order
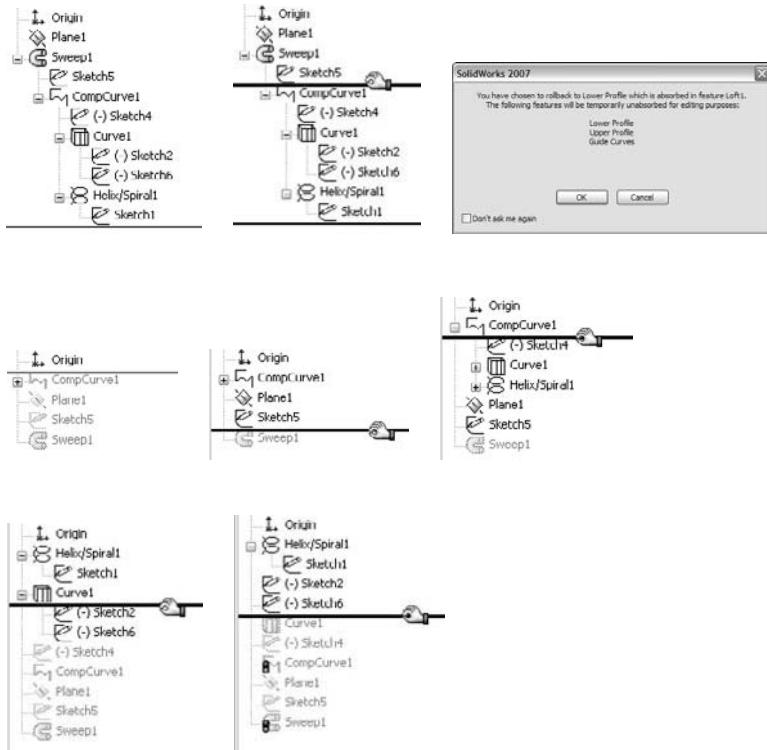
If you want to view consumed sketches in their original order — for example, the sketches in the loft feature in Figure 11.3 — you must first expand the feature by clicking the plus (+) symbol next to it so that you can see the consumed sketches, and then roll back between the feature and the first sketch. At this point, a warning message appears, stating that the sketches will be temporarily unabsorbed during editing. You can then move the Rollback bar again to show the sketches.

This maneuver can become complicated when you have two sketches absorbed by a projected curve, the projected curve absorbed by a composite curve and the composite curve absorbed by a sweep feature. To isolate the original projected sketches, you need to repeat the technique with the Rollback bar as described earlier, once for each parent/child relationship. This is shown in Figure 11.4, and the steps to accomplish it are as follows.

FIGURE 11.4

Rollback of nested, absorbed features

To view consumed sketches in their original order, follow these steps:

1. **Expand the sweep.**
2. **Roll back between the sweep and the profile sketch.**
3. **Answer the prompt that appears.**
4. **Roll down so that you can see the unabsorbed sketches and curves.**
5. **Expand the composite curve.**
6. **Roll back to just after the composite curve.**

7. Roll back so that you can see the contents of the composite curve.

8. Expand the projected curve.

9. Roll back to just after the projected curve.

10. Roll back so that you can see the contents of the projected curve.

Using other Rollback techniques

The Rollback feature is located on the RMB menu. Simply right-click a feature and select either Rollback or Roll to Previous. If you are already rolled back and you right-click below the Rollback bar, you can access additional options to Roll Forward and Roll to End.

Editing any feature other than the last feature also serves to roll back the model while you are in Edit mode. As soon as you rebuild the feature or sketch, SolidWorks rebuilds the entire design tree.

The Tools ➪ Options ➪ View setting for Arrow Key Navigation enables you to use the up- and down-arrow keys to manipulate the Rollback bar. Under normal circumstances, the arrow keys control the view orientation, but after you have moved the Rollback bar once using the cursor, the up- and down-arrow keys control the Rollback bar. The left- and right-arrow keys have no effect on the Rollback bar.

Caution
The one situation where this technique does not work as expected is when you are working on a part in the context of the assembly, with the design tree rolled back. The down arrow simply causes the Rollback bar to roll immediately to the end of the design tree. ∎

Reordering Features

The ability to reorder features is one that offers SolidWorks users control over the modeling process and its outcome. Without this capability, editing would not be one of the strengths of the software. As you already know, feature order can make a difference in the final shape of a part. For example, this order:

1. Extrude

2. Cut

3. Fillet

4. Shell
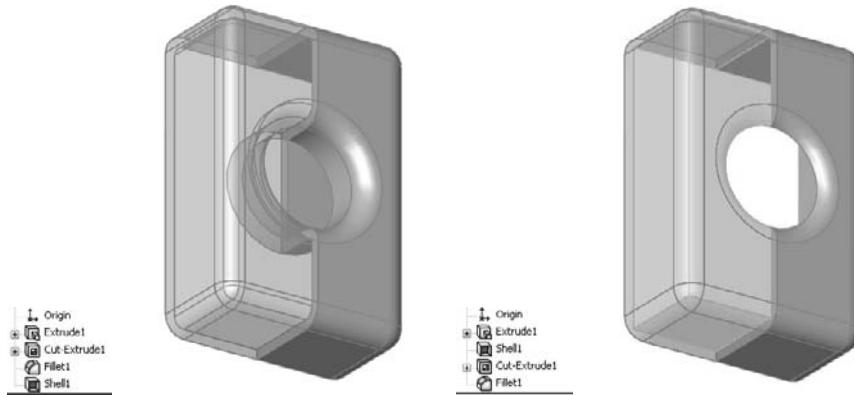
gives you a very different part from this order:

1. **Extrude**
2. **Shell**
3. **Cut**
4. **Fillet**

The results of these different orders are shown in Figure 11.5. (The part is split and partially transparent for demonstration purposes only.) You can view this part on the CD-ROM; it is filename `Chapter 11 Reorder.sldprt`.

FIGURE 11.5

Parts that use a different feature order

On the part in the example shown in Figure 11.5, it is fairly simple to reorder the Shell feature by dragging it up the design tree. As a result, the well created by the Cut feature is not shelled around (to create a tube) if the cut comes after the shell. Also, notice the effect of applying the fillets after the shell rather than before it. The corners inside the box are sharp, while the outside corners have been filleted. When you apply the fillet before the shell, fillets that have a radius larger than the shell thickness are transferred to the inside of the shell.

When you are reordering the features, a symbol may appear on the reorder cursor that says that you cannot reorder the selected feature to the location you want. In this case, you may want to select the Parent/Child option from the RMB menu to investigate. Sketch relationships, feature end conditions, and faces or edges selected for features such shell, patterns, and mirror can cause relationships that prevent reordering.

If two adjacent features are to swap places, it generally does not matter whether you move one feature up the design tree or you move the other one down. However, there are isolated situations that are usually created by the nested, absorbed features discussed earlier, where one feature cannot go in one direction, but the other feature can go in the opposite direction, achieving the exact same result. If you run into a situation where you cannot reorder a feature in one direction even though it appears you should be able to, try moving another feature the other direction.

Note
While this behavior may appear to be a bug, it is more likely just a product of parent/child relationships hidden by the way SolidWorks reorders consumed parent sketches and other types of features. If you were able to stretch the FeatureManager out into a straight line from earliest to latest, the difficulties in reordering would become apparent. ■

Reordering Folders

There are times when, regardless of which features you choose to move and which direction you choose to move them in, you are faced with the task of moving many features. This can be time-consuming and tedious, not to mention have the potential to introduce errors. To simplify this process, you can put all the features to be moved into a single folder, and then reorder the folder. Keep in mind that the items in the folder need to be a continuous list (you cannot skip features), and you can only reorder the folder if each individual feature within the folder can be reordered.

Best Practice
Folders are frequently used for groups of features that go together and that may be suppressed or unsuppressed in groups. You can also use folders in assemblies. Folders are frequently used to group cosmetic fillet features that are often found at the end of design trees for plastic parts or for groups of hole features. ■

To create a folder, right-click a feature, or a selected group of features, and select Add to New Folder. Folders should be renamed with a name that helps identify their contents. You can reorder folders in the same way as individual features. When you delete a folder, the contents are removed from the folder and put back into the main tree; they are not deleted.

You can add or remove features to or from the folders by dragging them in or out. If a folder is the last item in the FeatureManager, the next feature that is created is not put into the folder; you must place it in the folder manually. You cannot drag features out of a folder and place them immediately after it, because they will just go back into the folder. If you want to pull a feature out of a folder and place it after the folder, there must be another feature between the feature that you are moving and the folder. However, you can pull a feature out of the folder and place it just *before* the folder.

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Using the Flyout and Detachable FeatureManagers

The Flyout FeatureManager resides at the top-left corner of the graphics window. The PropertyManager goes in the same space as the FeatureManager, and is sometimes too big to allow this area to accommodate both managers in a split window.

The Flyout FeatureManager enables you to select items from the design tree when the regular FeatureManager is not available because it is covered by the PropertyManager. It usually appears collapsed, so that you can only see the name of the part and the part symbol. To expand it, click the plus icon next to the name of the part in the Flyout FeatureManager.

You can use the Flyout FeatureManager in parts or assemblies. However, you cannot use the Flyout FeatureManager to suppress or roll back the tree.

Cross-Reference
Other functionality and limitations of the Flyout FeatureManager that relate to its function in assemblies can be found in Chapters 12 to 15.■
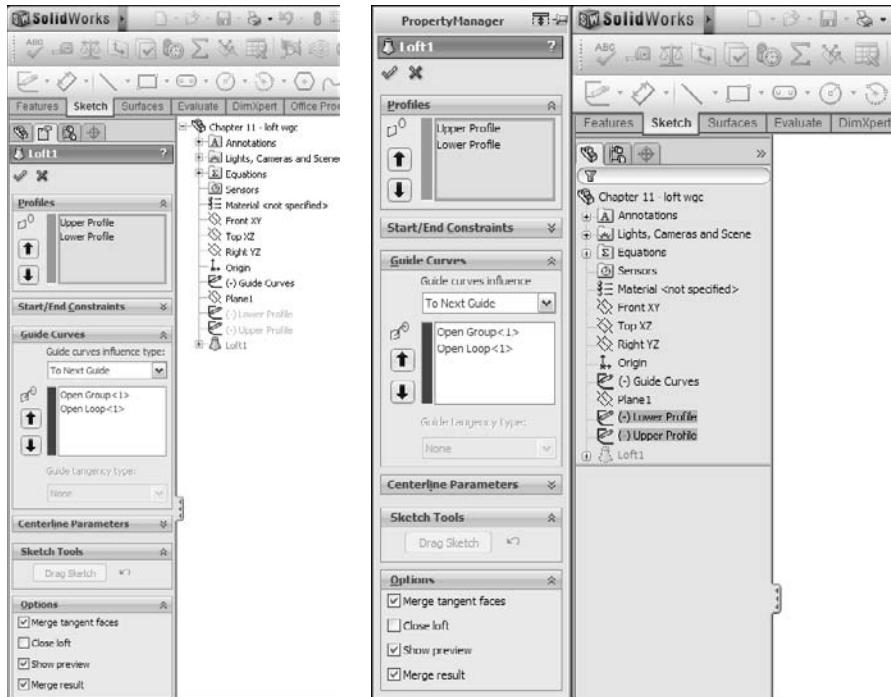
You can access the settings for the Flyout FeatureManager by choosing Tools ⇨ Options ⇨ Feature Manager ⇨ Use Transparent Flyout FeatureManager in Parts/Assemblies.

You may prefer not to work with the flyout FeatureManager because it interrupts your workflow by covering the regular FeatureManager with a PropertyManager. If this is the case, you can use the detachable PropertyManager instead. Detaching the PropertyManager removes the need for the fly-out. I often dock the detachable PropertyManager where the flyout FeatureManager would go, or even use it undocked on a second monitor. The main advantage of using the detachable PropertyManager instead of the flyout FeatureManager is that with the detachable PropertyManager you don't have to re-locate features in the FeatureManager that were already in view.

Figure 11.6 shows the difference between the flyout FeatureManager on the left, and the detach-able PropertyManager on the right. My preference is clearly the detachable PropertyManager. When you use this, everything is predictable, and you don't have to go hunting for features that were listed right in front of you when you do something that opens a PropertyManager. I usually decrease the overall size of the SolidWorks application window, and place the PropertyManager to the left of the SolidWorks application. This works best on a wide aspect monitor or multiple moni-tors. If you use a small monitor or a normal aspect monitor, using the Auto Collapse option with the PropertyManager docked to the right of the FeatureManager (where the flyout FeatureManager would otherwise go) is also a good option.

FIGURE 11.6

Comparing the flyout FeatureManager with the detachable PropertyManager

You may correctly ask "What's the difference?" The difference is that when you do something like editing a sketch plane, the current state of the FeatureManager is covered and replaced by the PropertyManager. You may have had the new plane you wanted to use in view. Especially with long FeatureManagers, in both parts and assemblies, when the flyout appears, you have to again scroll to find the plane that was right in view. However you use the detachable PropertyManager, I think you will find it an improvement over the flyout.

Summarizing Part Modeling Best Practice

This section is a summary of best practice suggestions for modeling parts. Best practice lists are important because they lay the groundwork for usage of the software, which is helpful for new users and users who are trying to experiment with the limits of the software.

I believe that it is only after you respect the rules and understand why they are so important, that you know enough to break them. However, best practice lists should not be taken too seriously. They are not inflexible rules, but conservative starting places; they are concepts that you can default to, but that can be broken if you have good reason. Following is a list of suggested best practices:

- Always use unique filenames for your parts. SolidWorks assemblies and drawings may pick up incorrect references if you use parts with identical names.

- Using Custom Properties is a great way to enter text-based information into your parts. Users can view this information from outside the file by using applications such as Windows Explorer, SolidWorks Explorer, and Product Data Management (PDM) applications.

- Learn to sketch using automatic relations.

- Use fully dimensioned sketches when possible. Splines are often impractical to fully dimension.

- Limit your use of the Fixed constraint.

- When possible, make relations to sketches or stable reference geometry, such as the Origin or standard planes, instead of edges or faces. Sketches are far more stable than faces, edges, or model vertices, which change their internal ID at the slightest change and may disappear entirely with fillets, chamfers, split lines, and so on.

- Do not dimension to edges created by fillets or other cosmetic or temporary features.

- Apply names to features, sketches, and dimensions that help to make their function clear.

- When possible, use feature fillets and feature patterns rather than sketch fillets and sketch patterns.

- Combine fillets into as few fillet features as possible; this also enables you to control fillets that need to be controlled separately — such as fillets to be removed for Finite Element Analysis (FEA), drawings, and simplified configurations — or added for rendering.

- Create a simplified configuration when building very complex parts or working with large assemblies.

- Model with symmetry in mind. Use feature patterns and mirroring when possible.

- Use link values or global variables to control commonly used dimensions.

- Do not be afraid of configurations. Control them with design tables where there are more than a few configs, and document any custom programming or automated features in the spreadsheet for other users.

- Use display states when possible instead of configurations.

- Use multi-body modeling for various techniques within parts; it is not intended as a means to create assemblies within a single part file.

- Cosmetic features — fillets, in particular — should be saved for the bottom of the design tree. It is also a good idea to put them all together into a folder.

- Use the Tools ➪ Options ➪ Performance ➪ Verification on Rebuild setting in combination with the Ctrl+Q command to check models periodically and before calling them "done." The more complex the model, or the more questionable some of the geometry or techniques might be, the more important it is to check the part.

- Always fix errors in your part as soon as you can. Errors cause rebuild time to increase, and if you wait until more errors exist, troubleshooting may become more difficult.

- Troubleshoot feature and sketch errors from the top of the tree down.

- Do not add unnecessary detail. For example, it is not important to actually model a knurled surface on a round steel part. This additional detail is difficult to model in SolidWorks, it slows down the rebuild speed of your part, and there is no advantage to actually having it modeled (unless you are using the model for rapid prototype or to machine a mold for a plastic part where knurling cannot be added as a secondary process). This is better accomplished by a drawing with a note. The same concept applies to thread, extruded text, very large patterns, and other features that introduce complex details.

- Do not rely heavily on niche features. For example, if you find yourself creating helices by using Flex/Twist or Wrap instead of Sweep, then you may want to rethink your approach. In fact, if you find yourself creating a lot of unnecessary helices, then you may want to rethink this approach as well, unless there is a good reason for doing so.

- File size is not necessarily a measure of inefficiency.

- Be cautious about accepting advice or information from Internet forums. You can get both great and terrible advice from people you don't know, along with everything in-between. Sometimes even groups of people can be dead wrong. Get someone you trust to verify ideas, and as always, test them on copied data to determine if they're effective.

If you are the CAD Administrator for a group of users, you may want to incorporate some best practice tips into standard operating procedures for them. The more users that you manage, the more you need to standardize your system.

Cross-Reference

Best practice development is covered at length in the SolidWorks Administration Bible (Wiley, 2009). ■

Using the skeleton or wide tree approach

The term *Skeleton* in Pro/ENGINEER has a different significance than the way it is being used here. SolidWorks does not have any feature or function named "skeleton." The term is just being used to refer to a set of sketches, planes, axes, and reference points used to lay out the major faces and features of a part.

The SolidWorks Help files, tutorials, and training curricula have encouraged users in some respects to take a "fast and loose" approach to modeling, which lends itself best to simple models that are not changed frequently. Little thought is given to the structure of the part; the focus is on the final shape. The main consideration seems to be the simplest way to do something, or how it *could* be done rather than how it *should* be done. This mentality fit well with the initial several releases of the SolidWorks software, which at that time was marketed as being simple and fast.

The software has progressed immensely since those days. It is now entirely plausible to create complex castings and plastic parts with many hundreds of features, weaving in and out of surface and solid techniques, multi-bodies, and external references. This is a far cry from the typical tutorial or training part, which still tend to have fewer than 15 features, half of which may be fillets. With the simpler parts, you hardly give a thought to parent/child relationships, rebuild times, or the consequences of making changes that cause a feature to fail, because the whole part can be rebuilt from scratch in ten minutes anyway.

SolidWorks users have traditionally been taught to build each feature linearly, on top of the one that came before. This is the genealogical equivalent of each generation having a single child, and then that child having a single child, and so on. The family tree, or FeatureManager, winds up looking like a long staircase, with each generation related only to the generation immediately before it. In the SolidWorks world, this creates long, linear, daisy-chained relationships between consecutive features.

It turns out that this is not a great idea, especially as the parts begin to get more complex. When each feature is dependent upon the one before it, all the features must be solved in a particular order, and if one feature fails, so do all the features that come after it. This also slows down the rebuilding process. Especially as we move into the age of parallel multi-threaded processing, a linear set of commands or features must be executed in order one after the other, and there is really little room for parallel processes.

The sophistication of the documentation provided with SolidWorks software has not kept pace with the sophistication of the software itself, which I suppose is why you are reading this book rather than the help files provided with the software. The documentation is still based on the simple scenarios, and the advanced user is left to figure things out on his or her own.

As the software gets more sophisticated, the models created with the software can become more sophisticated, and the methods used to build the models must become more sophisticated, as well. It's time to leave the linear modeling approaches behind.

Rather than using a linear daisy-chain modeling scenario, it is better practice to base features on entities that are less likely to fail or change in such a way that dependent downstream features also fail. In earlier chapters, I have suggested that you make sketch relations to other sketches when possible instead of model edges for this very reason.

Taking that scenario one step further, what if a handful of sketch and plane features were used to centralize control of all of the rest of the features? What if every feature, to the extent possible, related back to these "skeleton" features? Features such as fillets, shell, and draft by design require selections from solid geometry, but other features, such as any feature created from sketches, could be made with only reference to those original skeleton sketches and planes. The parent/child relationship would look very different for a model made in this way. Instead of looking like a long staircase, this tree would look more like a tree that gets wide very quickly. There would be fewer "generations," but each generation would be more populated.

The first thing to notice is that errors in features at the top of the tree do not cascade down the tree as they do in the "stairstep" model. Second, it is always much easier to find how a model is constructed, because all the reference geometry used to build it is set up in the first few features. This scenario also has the potential to make better use of multi-threaded processing because the logic is less linear and more parallel.

On the CD-ROM

Examine the part on the CD-ROM for Chapter 11 called `Chapter 11 Sketleton.sldprt`**. Roll back the FeatureManager and examine it. Notice that sketches do not reference faces or edges of the part, but other sketches and planes. ∎**

Using Evaluation Techniques

You can use evaluation techniques to evaluate geometry errors, demonstrate the manufacturability of a given part, or to some degree quantify aesthetic qualities of a given part, or section of a part. I discuss evaluation techniques here because the design cycle involves iterations around the combination of create-evaluate-edit-evaluate functions. I discuss the following techniques in this section:

- Verification on rebuild
- Check
- Zebra Stripes/RealView/Lights and specularity
- Curvature display
- Deviation analysis
- Tangent Edges as Phantom
- Geometry Analysis
- Feature statistics
- Curvature comb
- SimulationXpress

Many of these techniques apply specifically to plastic parts and complex shapes, but even if you do not become involved in these areas of design or modeling, these tools may help you to find answers on other types of models as well.
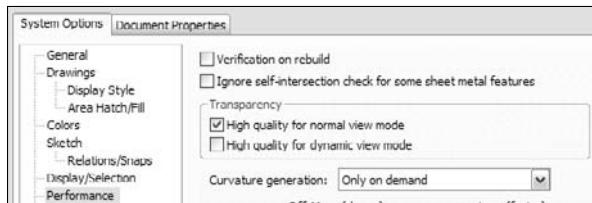
In SolidWorks 2010, a new tab, Evaluate, appears on the default Command Manager. You can use the commands on this tab to evaluate parts in several ways. Some focus on plastic parts or thin walled parts or symmetric parts, and so on. Most of these tools are from the Tools toolbar, but are found on the Evaluate tab in the Command Manager.

Using Verification on rebuild

Verification on rebuild is an option that you can access by choosing Tools ⇨ Options ⇨ Performance ⇨ Verification on rebuild. Under normal circumstances (with this setting turned off), SolidWorks checks each face to ensure that it does not overlap or intersect improperly with every adjacent face. Each face can have several neighbors. This option is shown in Figure 11.7.

FIGURE 11.7

The Verification on rebuild option

With the setting selected, SolidWorks checks each face with every other face in the model. This is a better check than with the setting off but greatly increases the workload. The switch is deselected by default to prevent rebuild times from getting out of control. For most parts, the default setting is sufficient; however, when parts become complex, you may need to select the more advanced setting.

If you are having geometry or rebuild error problems with a part and cannot understand why, then try turning Verification on rebuild on and pressing Ctrl+Q. Ctrl+Q applies the Forced Rebuild command, and rebuilds the entire design tree. Ctrl+B, or the Rebuild command, only rebuilds what SolidWorks determines needs to be rebuilt.

If you see additional errors in the design tree that were not there before, then the combination of Verification on rebuild and Forced Rebuild has identified problem areas of the model and the features that caused the errors failed. If not, then your problem may be elsewhere. You still need to fix any errors found this way.

Performance
For speed reasons, it is normal practice to turn Verification on rebuild off, and to use it selectively to check models with potential errors. The type of speed degradation that you can see is dependent on the number of faces in the model. Some of the performance degradation as relates to patterns is documented in Chapter 8. ■

Check
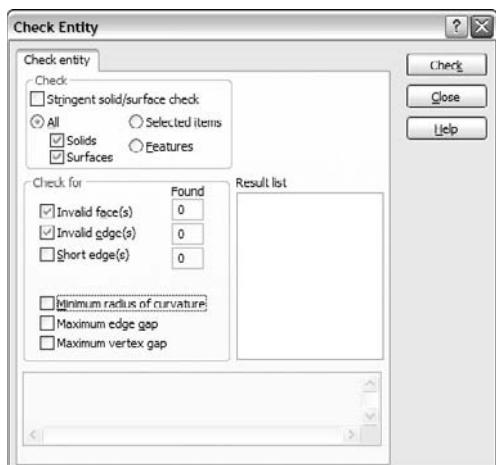
Check is a tool that checks geometry for invalid faces and other similar geometry errors. It is also often used to find open edges of surface bodies, short edges, and the minimum radius on a face or entity. I usually apply the Check tool before selecting the Verification on rebuild option. The Check tool points to specific face or edge geometry (not features or sketches) that is the cause of the problem. When the Check tool finds general faults, the locations it points to may or may not have something obvious to do with a possible fix.

Much of the time, the best tool for tracking down geometry errors is the combination of experience and intuition. It is not very scientific, but you come to recognize where potential problems are likely to arise, such those that occur when you attempt to intersect complex faces at complex edges. Figure 11.8 shows the Check Entity dialog box.

The Check Entity dialog box

Evaluating reflective techniques

Evaluating complex shapes can be difficult. A subjective evaluation requires an eye for the type of work you are doing. An objective evaluation requires some sort of measurable criteria for determining a pass or fail, or a way for you to assign a score somewhere in the middle.

One way to subjectively evaluate complex surfaces, and in particular the transitions between surfaces around common edges, is to use reflective techniques. If you look at an automobile's fender, you can tell whether it has been dented or if a dent has been badly repaired by seeing how the light

reflects off of the surface. The same principle applies when evaluating solid or surface models. Bad transitions appear as a crease or an unwanted bulge or indentation. The goal is to turn off the edge display and not be able to identify where the edge is between surfaces for the transition to be as smooth as if the whole area were made from a single surface.

Zebra Stripes

Zebra Stripes can be activated one of two ways, by choosing View ⇨ Display ⇨ Zebra Stripes from the menus, or by clicking a toolbar button on the View toolbar. Zebra Stripes place the part in a room that is either spherical or cubic, where the walls are painted with alternating black-and-white stripes (although you can change the colors and the spacing of the stripes). The part is made to be perfectly reflective, and the way that the stripes transition over edges tells you something about the qualities of the faces on either side of the edge. Four conditions are of particular interest:

- $c0$ = faces contact at edge
- $c1$ = faces are tangent at edge
- $c2$ = curvature of each face is equal at the edge and the transition is smooth
- $c3$ = rate of change of curvature of each face is equal at the edge

The Zebra Stripes tool can only help you identify $c0$, $c1$, and $c2$, and only subjectively. This feature is of most value between complex faces. Figure 11.9 illustrates how the Zebra Stripes tool shows the differences between these three conditions.

Notice how on the Contact-only model, the Zebra Stripe lines do not line up across the edge. On the Tangent example, the stripes line up across the edges, but the stripes themselves are not smooth. On the Curvature Continuous example, the stripes are smooth across the edges. The part shown in Figure 11.9 is a surface model, and can be found on the CD-ROM with the filename `Chapter 11 Zebra Stripes.sldprt`.

Tip

You should rotate the model a lot when you are using the Zebra Stripes tool. Changing the density of the lines can also help, as can increasing the image quality (Tools ⇨ Options ⇨ Document Properties ⇨ Image Quality). Turning off the edge display may also help. ∎

RealView
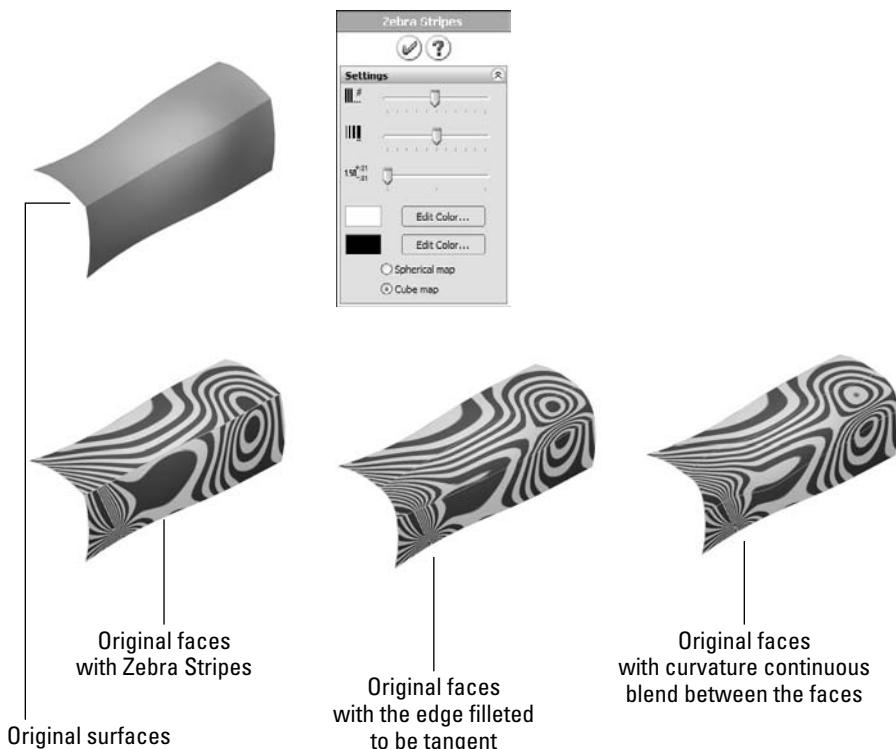
RealView Graphics display is only available to users with certain types of video cards. To see whether your card supports RealView, consult the SolidWorks Web site.

RealView causes reflections that can be used in a way similar to the reflections in Zebra Stripes. Rotate the part slowly and watch how the reflections flow across edges. Instead of black and white stripes, it uses the reflective background that is applied as part of the RealView Scene.

FIGURE 11.9

Contact, tangency, and curvature continuity

Original surfaces

Original faces
with Zebra Stripes

Original faces
with the edge filleted
to be tangent

Original faces
with curvature continuous
blend between the faces

Cross-Reference

RealView techniques and usage are covered in more depth in Chapter 5. ■

Curvature display

Model curvature can be plotted onto the model face using colors, as shown in Figure 11.10. The accuracy of this display leaves a bit to be desired, but it does help you identify areas of very tight curvature on your part. Areas of tight curvature can cause features such as fillets and shells to fail.

FIGURE 11.10

Curvature display
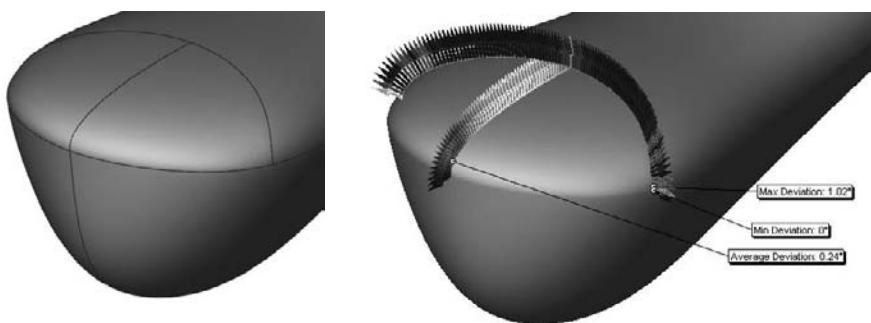
Deviation analysis

Deviation Analysis measures how far from tangent the surfaces on either side of a selected edge actually are. For example, the edges shown in Figure 11.11 are found to be fair, but not very good. I prefer deviations of less than 0.5 degrees. Often with some of the advanced surface types such as Fill, Loft, and Boundary, SolidWorks can achieve edges with less than 0.05-degree maximum deviation.

FIGURE 11.11

An example of Deviation Analysis

While Deviation Analysis helps to quantitatively measure how close to tangent the faces on either side of the selected edge are, it does not tell you anything about curvature, so you must still run Zebra Stripes to get the complete picture of the flow between faces. Both tests have to return good results to have an acceptable face transition.
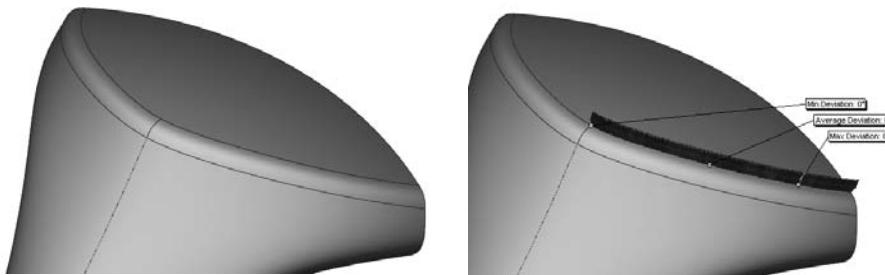
Tangent Edges as Phantom

Using the Tangent Edges as Phantom setting is an easy way to evaluate a large number of edges all at once. This function does not do what the Zebra Stripes tool does, but it gives you a good indication of the tangency across a large number of edges very quickly. Again it only represents tangency, and tells you nothing about curvature continuity.

Although this is an easy method to use, it is not completely reliable. I have not seen this function deliver false positives (edges displayed as tangent when in fact they were not), but I have seen many false negatives (edges that display as non-tangent when in fact they were). Figure 11.12 shows a situation where the edges are displayed with solid edges, but Deviation Analysis shows them to have a zero-degree maximum deviation.

Using the Tangent Edges as Phantom setting

The measure of tangency has some tolerance. Users cannot control the tolerance, nor does the documentation say what it is. If SolidWorks says two faces are *not* tangent at an edge, you can believe that, but if SolidWorks says that the faces *are* tangent, you still have to ask *how* tangent. That is the question that Deviation Analysis can answer.

Geometry Analysis
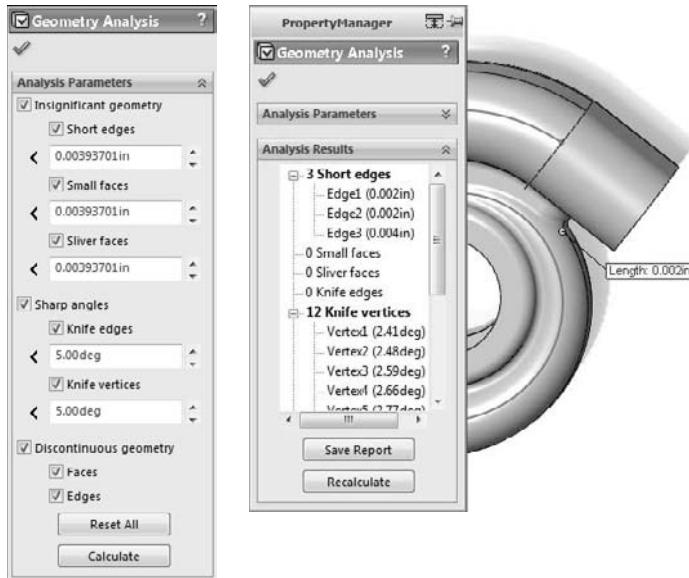
Another tool that is fairly new is the Geometry Analysis tool. You can find it in the Tools menu or the new Evaluate tab in the Command Manager. It is an extremely useful tool for troubleshooting problematic geometry. The PropertyManager, shown in Figure 11.13 allows you to look for several specific items:

- Short edges
- Small faces
- Sliver faces
- Knife edges/vertices
- Discontinuous faces or edges

FIGURE 11.13

Using Geometry Analysis to find typical problem spots

These specific types of geometry typically cause problems with other features, such as shells or fillets. If you are having difficulty with a feature failing for a reason that you can't explain, use the Geometry Analysis tool to point out potential problem spots. This is not a tool that will do your job for you, but it is a tool that gives you useful information to help you do your job better and faster.

Feature Statistics
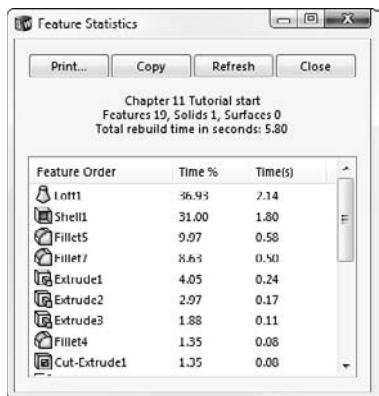
The Feature Statistics tool has been used previously in this book to measure rebuild times for individual features in parts. You can find it either in the Tools menu or the Evaluate tab of the Command Manager.

Feature Statistics lists the rebuild times of each individual feature in a part. This is useful for researching features, benchmarking hardware or versions of SolidWorks, and developing best practice recommendations for different tools and techniques. Figure 11.14 shows the Feature Statistics interface.

Overall, I do not recommend relying too heavily on the data the Feature Statistics tool provides; this is not because it is inaccurate, but because rebuild time is not always the best way to evaluate a model. You can certainly use the information, but you also need to keep it in perspective. A feature that takes a long time to rebuild but gives the correct result is always better than any feature that does not give the correct result, regardless of rebuild time.

FIGURE 11.14

Feature Statistics helps you analyze rebuild times for features.

Curvature Comb
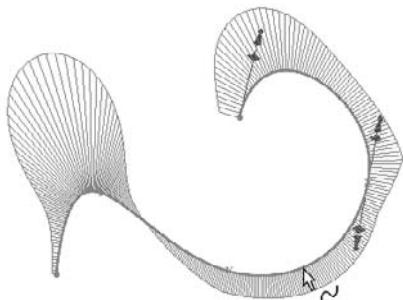
The Curvature Comb is a graphical tool you can apply to a spline, circle, arc, ellipse, or parabola to indicate the curvature along the length of the curve. You cannot apply a Curvature Comb to a straight line because it has no curvature. The height of the comb indicates the curvature. Curvature is the inverse of radius (c=1/r), so that as the radius gets smaller, the curvature gets bigger.

Figure 11.15 shows a curvature comb applied to a spline. Notice that the spline continuously changes curvature. An arc has constant curvature.

When the comb crosses the spline, it means that the direction of curvature has changed. When the comb intersects the spline, it means that the spline at that point has no curvature (it is straight).

FIGURE 11.15

A Curvature Comb shows the constantly changing curvature of the spline.

Cross-Reference

The Curvature Comb is discussed in depth in the SolidWorks Surfacing and Complex Shape Modeling Bible (Wiley, 2008). ■

Analyzing with SimulationXpress

SimulationXpress (formerly COSMOSXpress) is a limited version of SolidWorks Simulation (formerly COSMOS Works) that is bundled with SolidWorks to acquaint users with FEA. The full version of SolidWorks Simulation does a wide range of analysis, from vibrations to large deformations. SimulationXpress is a very quick and easy wizard for simple stress analysis on stand-alone parts with simple constraints. It does simple linear stress analysis on a single part with a single material using only fixed constraints and a load. You can also use SimulationXpress to do a simple stress/weight optimization based on dimensions that you select to be altered.

You can start SimulationXpress through the Tools menu, or from the Evaluate tab of the CommandManager. The interface guides you through a very simple wizard. If you have any familiarity with FEA applications, you will find SimulationXpress easy to understand and use, but possibly lacking in flexibility or capability.

Starting in SolidWorks 2010, SimulationXpress uses a combination of the Task Pane on the right of the graphics window and the PropertyManager on the left side of the graphics window. In this book, I am using the detached PropertyManager, so I can position it for more compact screen shots.

SimulationXpress is intended as a quick and dirty analysis run to see whether you are even close to having a part that will stand up to the loads you apply to it. As with any type of analysis, the results are only of value if the setup was correct and if you are able to interpret the results.

In this chapter I do not aim to teach analysis theory or best practice, this is simply a quick overview of how to use the tool. FEA is a field of study unto itself, and you should learn it from a dedicated resource. Analysis is a field that depends on approximations and simplifying assumptions. Knowing how and when to make the correct approximations and assumptions along with being able to interpret approximated results are some of the keys to success not provided by this book.
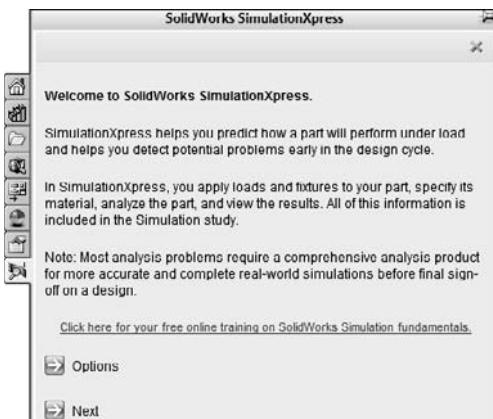
As you step through the process, the notes in the Task Pane seem to continually prompt you that various conditions will require the next level higher (not free) in order to correctly solve the model. SimulationXpress is clearly a sales and marketing tool, but if your needs are simple, and you understand the process, you may be able to get useful information from it.

Welcome to SolidWorks SimulationXpress

When you start SimulationXpress, an image welcomes you to the tool and lets you know what to expect from the process. It also issues a disclaimer that seems to downplay the value of this tool. There is also a link here for some valuable web-based training if you are new to analysis or simulation. Figure 11.16 shows the new Welcome screen.

FIGURE 11.16

Welcoming you to the SimulationXpress software

If you are a veteran SimulationXpress user, starting with SolidWorks 2010, most of what you see here is new, including the order of the steps and some of the terminology.

Applying Fixtures

Every analysis must have at least one face that is held stationary in order for the analysis to work. These are usually faces on the bottom of the part.

In SolidWorks 2010, Fixtures are what was formerly referred to as Restraints. You must apply Fixtures to the part to be analyzed as the first step in the process. A PropertyManager appears to collect the faces you select. Fixtures are fixed, meaning they are the faces in the analysis that remain perfectly stationary. These are usually some sort of mounting location, or a welded area.
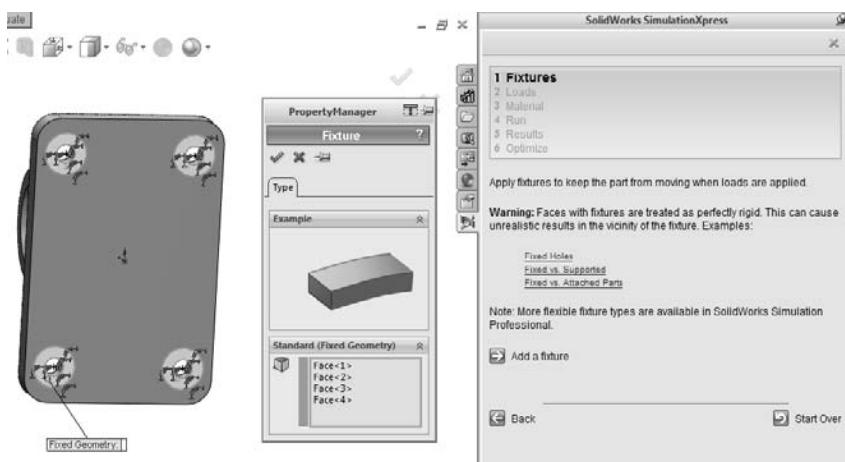
On this part, I have used the Split Line feature to split out areas of the bottom face around the bolt holes to represent sections of the part that will not move regardless of how much force is applied to the part.

The dark blue underlined text in the Task Pane are links to simple animations showing some possibilities for assigning Fixtures. Clicking the Add a fixture link in the Task Pane brings up the Fixture PropertyManager, shown in Figure 11.17.

With early versions of SolidWorks 2010, the display of the Fixture symbols seems erratic and incorrect depending on the rotation angle of the part.

Assigning fixed faces using the Fixture PropertyManager
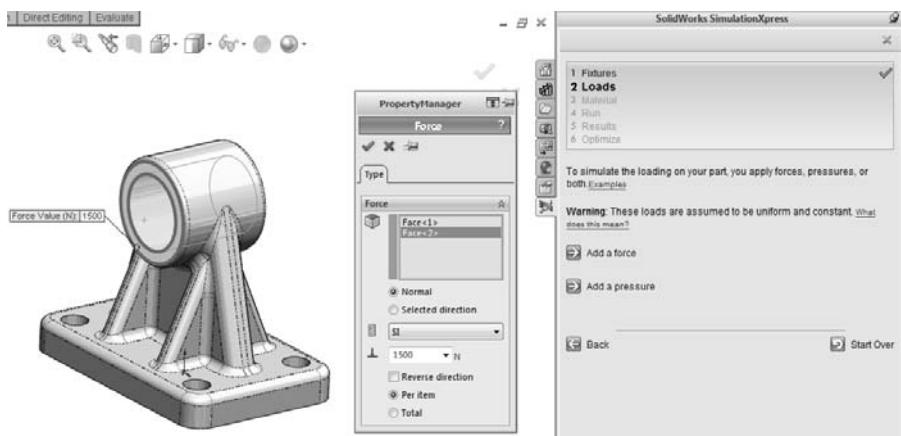
Applying Loads

You can apply forces or pressures to model faces. Forces are normal to selected faces by default, but you can select an axis, edge or sketch element to specify a selected direction for the force. Again links in the Task Pane enable you to see small video hints or tool tips. Figure 11.18 shows the Loads interfaces.

Applying loads to the model in SimulationXpress
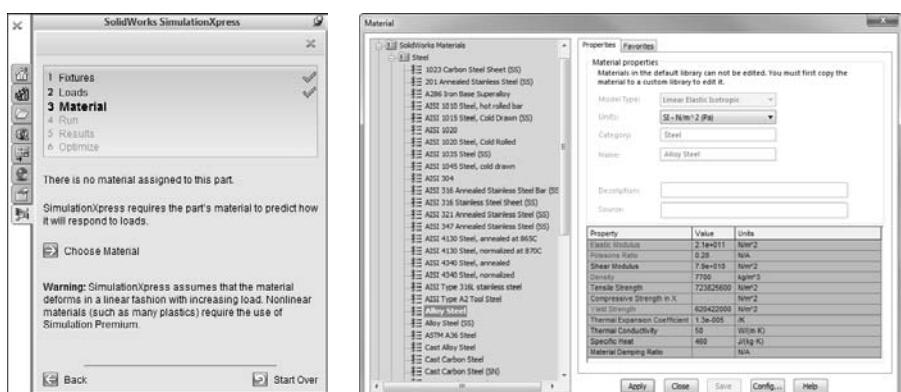
Selecting a Material

The third step in setting up SolidWorks SimulationXpress is to assign a material. Once you have made your selection, click Apply. Figure 11.19 shows some of the materials selection. Materials can be created or customized in the same way that you would create or customize standard SolidWorks materials.

Assigning a material in SimulationXpress

Tip
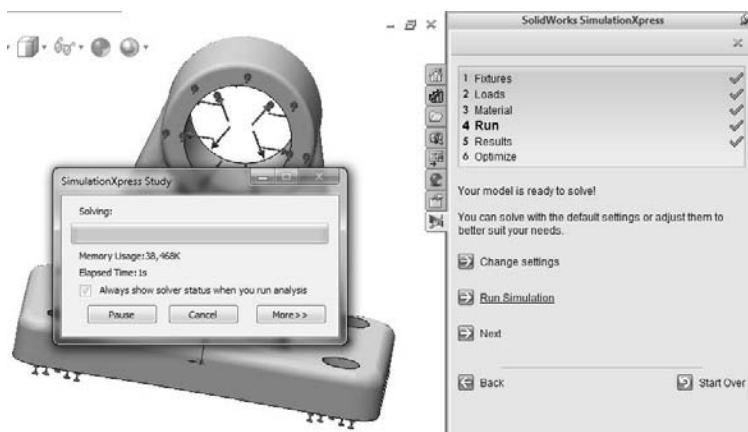
My experience with the early versions of SimulationXpress in SolidWorks 2010 sp0.0 has been that much of the tool did not work properly the first time. It took several tries to get a material to apply, the loads appeared to be applied incorrectly, and the initial screen did not appear when I first started the software. ■

Running the analysis

Analyzing simple parts on modern computers usually takes a matter of seconds. Because SimulationXpress is limited to single parts, most analyses you run will not take very long. You need to make sure that you have green check marks to the right of Fixtures, Loads, and Material before you can run the analysis. Figure 11.20 shows the analysis of this part in the process of running.

FIGURE 11.20

Running the analysis

Visualizing the results

SimulationXpress makes three different types of results available to you:
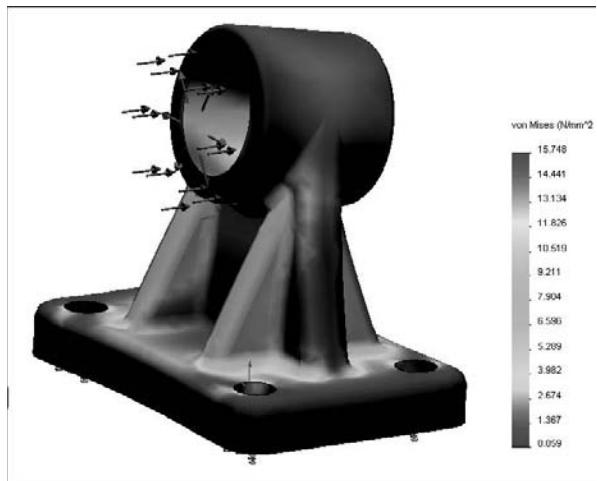
- vonMises stress plot
- Displacement plot
- Factor of Safety (FOS) plot

Each of these can be animated. The deformation shown by the model is exaggerated by a deformation scale factor, which is shown as part of the legend on the screen. It is typically around 1000x. Deformation is exaggerated to make it easier to visualize how the structure flexes under load.

Figure 11.21 shows a deformed vonMises stress plot.

FIGURE 11.21

Visualizing the results of the stress analysis
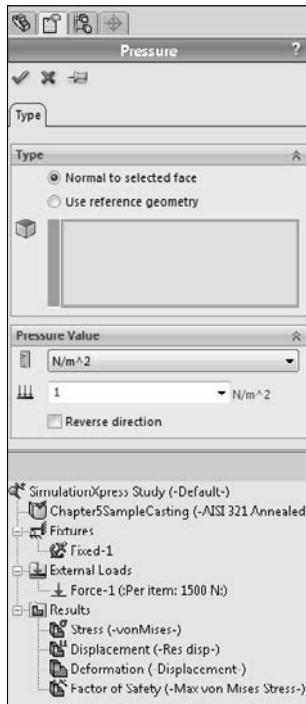
All the values that you enter and faces that you select are stored in the SimulationXpress Study panel, which shows up by default on the bottom in a split PropertyManager, as shown in Figure 11.22. This is saved with the part, and is displayed any time you have the SolidWorks SimulationXpress panel open in the Task Pane.

FIGURE 11.22

Accessing the results and inputs for the study in the lower panel of the PropertyManager

Optimizing the design
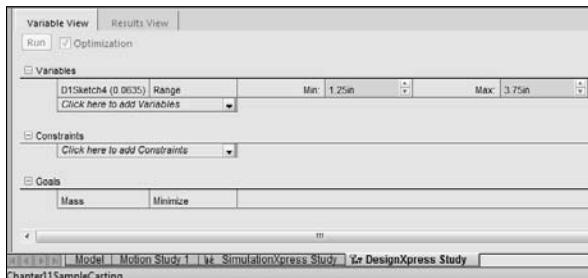
The final step in the Simulation process, if you choose to use it, is the Optimization. Optimization enables you to vary a dimension of the part and run multiple studies to determine what value for that dimension produces the best result.

Optimization displays another panel at the bottom of the graphics window that enables you to establish a range for the dimension that will be varied during the optimization. This panel is shown in Figure 11.23.

When you run the optimization, SolidWorks will change the model dimension within the range that you specified, and run an analysis for each dimension value. For this to be most successful, you must have a robust model where all the features can adapt to the range of dimensions for the feature being changed.

Entering a range of sizes for the optimized feature

Tutorial: Making Use of Editing and Evaluation Techniques
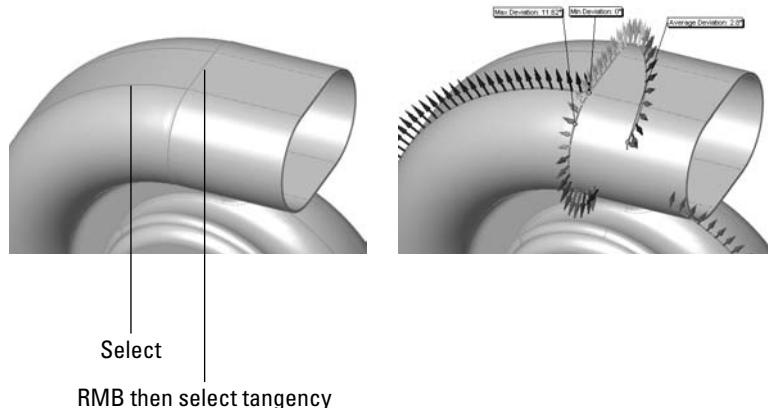
Using this tutorial, you make some major edits to an existing part. You use some simple loft and spline commands, and work with the rollback states and feature order, as well as some evaluation techniques. Please follow these steps:

1. **Open the existing part with the filename** `Chapter 11 Tutorial Start.sldprt`. Roll the part back and step through it feature by feature to see how it was made. Edit the loft feature to see which sketches were used to create it. This can help you to understand how the part was built. Exit the loft command and move the rollback bar back to the bottom of the tree.

2. **Open the Deviation Analysis tool (Tools ⇨ Deviation Analysis).** Select the edges, as shown in Figure 11.24.

   The maximum deviation is about ten degrees, which is far too much. This part needs to be smoothed out, which you can do using splines in place of lines and arcs.

3. **The first step is to make the outlet all one piece with the spiral.** You can do this with a Fit Spline. You need to create the Fit Spline before the loft profiles and after the spiral.

   Expand the loft, and roll back between the loft feature and the first sketch. Click OK in response to the prompt, and then roll back to just after the spiral, as shown in Figure 11.25.

4. **Right-click the spiral in the FeatureManager and show it.** Open a new sketch on the Top plane.

FIGURE 11.24

Deviation analysis of an existing part

Select

RMB then select tangency

FIGURE 11.25

Rolling back to just after the spiral

5. **Try to draw a horizontal line from the outer end of the spiral.** You will notice that you cannot reference the end of the spiral.
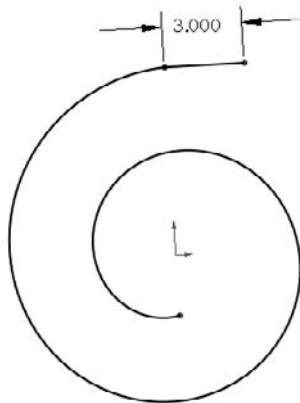
Tip

Curves that are absorbed into other features are notoriously difficult to work with. Generally, you need to select them from the FeatureManager to do anything at all with them. Also, if you need to reference an end of an absorbed curve, you are better off using Convert Entities to make it into a sketch entity. ■

6. **Notice that you cannot select the spiral from the graphics window.** Even when selected from the FeatureManager, it appears not to be selected in the graphics window. Ensure that it is selected in the FeatureManager, and then click the Convert Entities button on the sketch toolbar.

7. **Draw a horizontal line from the outer end of the spiral and dimension it to be three inches long, as shown in Figure 11.26.**

FIGURE 11.26

Preparing for the Fit Spline

8. **Select both the converted spiral and the line, and click Tools ⇨ Spline Tools ⇨ Fit Spline.** Set the Tolerance to .1 and make sure that only the Constrained option is selected. Click OK to accept the Fit Spline. Test to make sure that a single spline is created by moving your cursor over the sketch to see whether the whole length is highlighted.

Note

The Fit Spline feature fits a spline to a set of sketch entities within the specified tolerance. It can be a useful tool for smoothing out sketch geometry. ■
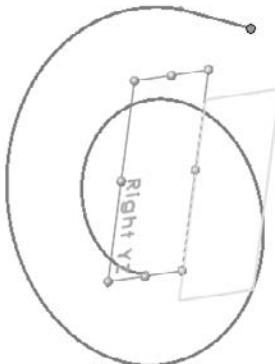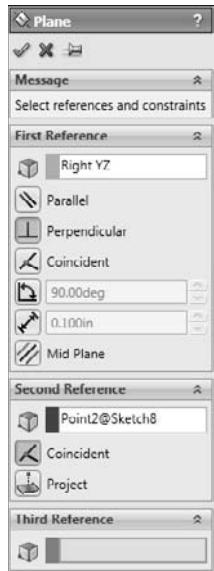
Caution

Do not exit the Fit Spline by pressing the Enter key as you do with other commands, because it simply exits you out of the command without creating a spline. ■

9. **Right-click on spline and select the Curvature Comb.** Notice how the comb is affected by the transition from the spiral to the straight line.

10. **Exit the sketch, and create a new plane.** Choose Insert ➪ Reference Geometry ➪ Plane from the menus. Select the Right plane from the Flyout FeatureManager as the first reference and the outer end of the Fit Spline that you have just created as the second reference. Click OK to accept the new plane. This is illustrated in Figure 11.27.

FIGURE 11.27
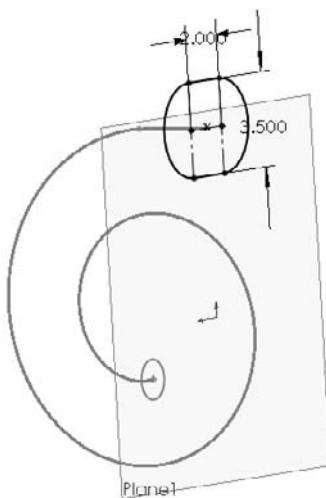
Creating a new plane

11. **Drag the Rollback bar down between Sketch3 and Loft1.** If it goes beyond Loft1, then you need to navigate back to this position again.

12. **Right-click Sketch3 and select Edit Sketch Plane.** Select the newly created Plane1 from the Flyout FeatureManager, and click OK to accept the change.

13. **Notice that the loft profile has moved to a place where it does not belong.** This is because the sketch has a Pierce constraint to the spiral, and there are multiple places where the spiral pierces the sketch plane.

    Edit Sketch3 and delete the Pierce constraint on the sketch point in the middle of the construction line. Create a Coincident relation between the sketch point and the outer end of the Fit Spline, as shown in Figure 11.28. Do not exit the sketch.

FIGURE 11.28

Sketch3 in its new location
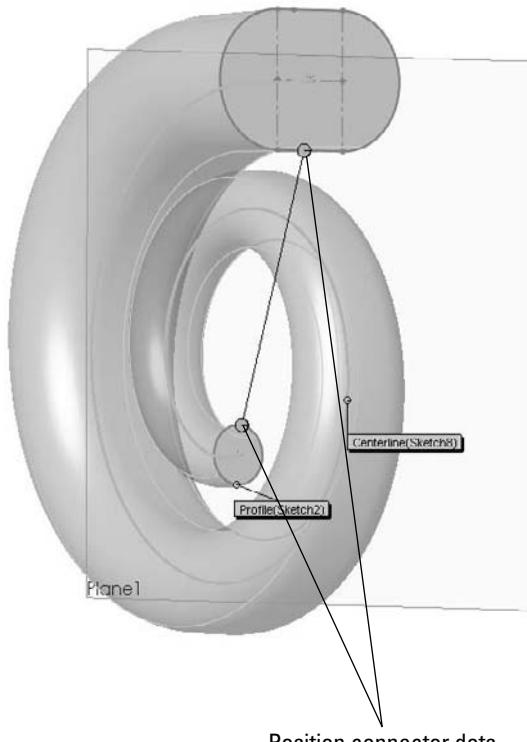
14. **One of the goals of these edits is to smooth out the part.** Remember that the Deviation Analysis told you that the edges created between the lines and arcs in Sketch3 were not very tangent. For this reason, it would be a good idea to replace the lines and arcs in Sketch3 with another Fit Spline.

    Right-click one of the solid sketch entities in Sketch3, and click Select Chain.

15. **Create another Fit Spline using the same technique as in Step 8.** Exit the sketch.

16. **Drag the Rollback bar down one feature so that it is below the Loft.** Notice that the Loft feature has failed. If you hold the cursor over the feature icon, the tooltip confirms this by displaying the message, "The Loft Feature Failed to Complete."

17. **Edit the Loft feature.** Expand the Centerline Parameters panel if it is not already expanded, and delete the Spiral from the selection box. In its place, select the Spiral Fit Spline.

18. **If the loft does not preview, check to ensure that the Show Preview option is selected in the Options panel, at the bottom.**

19. **If it still does not preview, right-click in the graphics window and select Show All Connectors.** Position the blue dots on the connector so that it looks like Figure 11.29.

20. **Click OK to accept the loft.** The loft should be much smoother now than it was before. In addition, the spiral feature should no longer be under the loft; it should now be the first item in the design tree.

21. **Drag the Rollback bar down to just before the Shell feature.** Notice that Fillet5 has failed. Move the mouse over Fillet5. The tooltip tells you that it is missing some references. Edit Fillet5 and select edges in order to create fillets, as shown in Figure 11.30.

FIGURE 11.29

Positioning the connectors
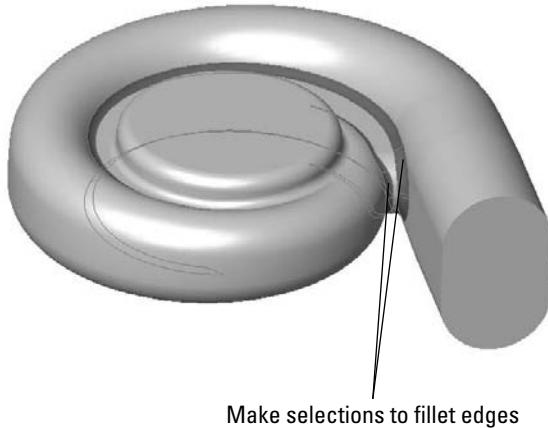
Position connector dots
in approximately corresponding
locations on the two loft profiles

22. **Right-click in the design tree and select Roll To End**. This causes the FeatureManager to become unrolled all the way to the end.

23. **The outlet of the involute is now longer than it should be.** This is because the original extrude was never deleted from the end. Right-click the Extrude1 feature and select Parent/Child. The feature needs to be deleted, but you need to know what is going to be deleted with it.

24. **The Shell is listed as a child of the extrude because the end face of the extrude was chosen to be removed by the Shell.** Edit the Shell feature and remove the reference to the face. (A Shell feature with no faces to remove is still hollowed out.)

25. **If you right-click Extrude1 and select Parent/Child again, the Shell feature is no longer listed as a child.**

26. **Delete Extrude1, and when the dialog box appears, press Alt+F to select Also Delete Absorbed Features.**

FIGURE 11.30

Repairing Fillet5

Make selections to fillet edges

27. Edit the Shell feature and select the large end of the loft. Exit the Shell feature. The results up to this step are shown in Figure 11.31.

FIGURE 11.31

The results up to Step 27

28. Drag a window in the design tree to select the four fillet features. Then right-click and select Add to New Folder. Rename the new folder Fillet Folder.

29. Click the Section View tool, and create a section view using the Front plane.

30. Reorder the Fillet folder to after the Shell feature.
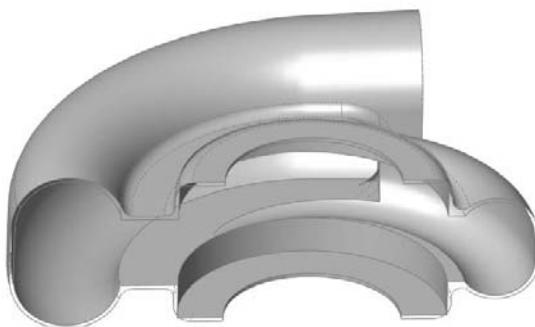
31. **At this point, you should notice that something does not look right.** This is because creating the fillets after the Shell causes the outside fillets to break through some of the inside corners. The fillets should have failed, but have not, as shown in Figure 11.32.

FIGURE 11.32

Fillets that should have failed

32. **Choose Tools ➪ Options ➪ Performance, and select Verification on rebuild.** Then click OK to exit the Performance menu and press Ctrl+Q. The fillets should now fail.

33. **Click Undo to return the feature order to the way it was.**

34. **Save the part.**

Summary

Working effectively with feature history, even in complex models, is a requirement for working with parts that others have created. When I get a part from someone else, the first thing that I usually do is look at the FeatureManager and roll it back if possible to get an idea of how the part was modeled. Looking at sketches, relations, feature order, symmetry, redundancy, sketch reuse, and so on are important steps in being able to repair or edit any part. Using modeling best practice techniques helps ensure that when edits have to be done, they are easy to accomplish, even if they are done by someone who did not build the part.

Evaluation techniques are really the heart of editing, as you should not make too many changes without a basic evaluation of the strengths and weaknesses of the current model. SolidWorks provides a wide array of evaluation tools. Time spent learning how to use the tools and interpret the results is time well spent.

Part III

Working with Assemblies

The chapters of Part III detail the tools you need to be familiar with in order to get the most from your assemblies. Of these, Chapters 12 and 16 are my favorites. These are loaded with best-practice suggestions and tips for efficient workflow. Chapter 16, the in-context chapter, is particularly important for SolidWorks users from many different fields who need or want to make parametric relations between parts. A lot of erroneous information floats around the SolidWorks community on this topic, and this chapter will help you cull the reliable information.

Building Efficient Assemblies

Chapter 4 provides a brief introduction to the basics of assemblies, how to put parts together, the basics of mating, and so on. The basic process for putting assemblies together remains the same for assemblies of any size, but once the assembly passes a certain point — and this point is likely different for each user or application — the assembly will benefit from some sort of organization or management techniques. This chapter introduces you to the tools and techniques that are available to help you manage performance issues as well as general-use issues, efficiency, browse-worthiness, or searchability.

Identifying the Elements of an Assembly

From Chapter 4, you know that an assembly can contain parts and mates. Real-world assemblies can become much more complex. As the number of parts and design requirements for an assembly grows, you may need to add some of the following types of assembly elements. (You may already be familiar with some of these from having worked with part documents.) The assembly elements are listed here, and described in detail either later in this chapter or in other chapters.

- Assembly equations
- Assembly Layout feature
- Assembly layout technique
- Assembly reference geometry (plane, axis, point, coordinate system)

- Parts
- Subassemblies
- Folders for parts
- Folders for mates
- Mates
- Assembly features (cuts that are made once the parts are assembled)
- Component patterns
- Mirror components
- In-context reference placeholders
- Smart Fasteners
- Smart Components
- Virtual components
- Envelopes
- Assembly configurations
- Assembly Design Table
- Assembly Bill of Materials (BOMs)
- Hidden/Suppressed/Lightweight/SpeedPak
- Sensors
- Hole Series

These elements are shown in Figure 12.1.

Standard reference geometry items

The three standard planes and the Origin in the assembly FeatureManager design tree are all familiar to you, as are the other standard items, such as the Annotations, Design Binder, Sensors, and Lights and Cameras folders. These items offer the same standard functionality of their part document counterparts.

Note
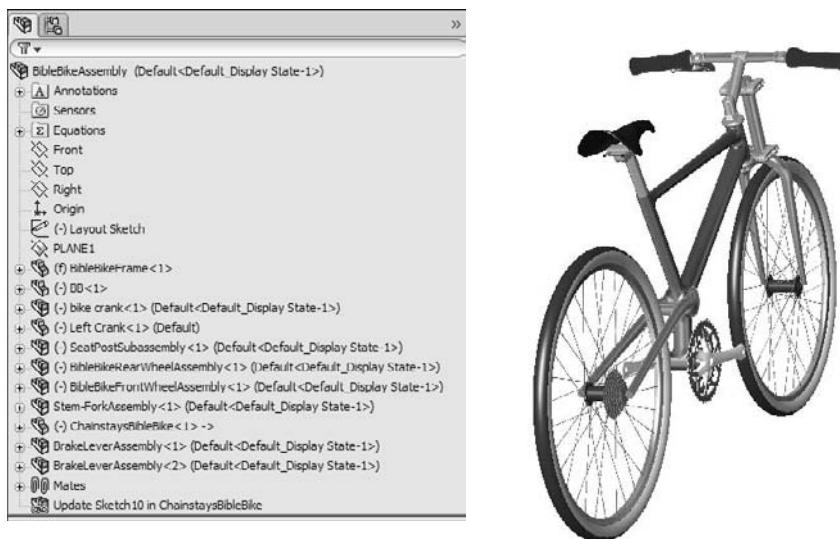
Remember that you can use choose Tools➪Options➪FeatureManager page and permanently select or deselect various folders under the FeatureManager header. Also be aware that when some folders are set to Automatic, they do not automatically turn on when they should. In cases like this, choose Tools➪Options➪ FeatureManager and manually set them to Show. ■

Assembly equations

Assembly equations work mainly like part equations, but with some additional complications and considerations. For example, one of the additional features of assembly equations is the ability to drive the dimensions of one part from another part. The syntax is slightly different for this application, as shown in Figure 12.2. Overall, issues with equation order and using driven dimensions on the right side of the equation are the same between parts and assemblies.

FIGURE 12.1

Elements of an assembly
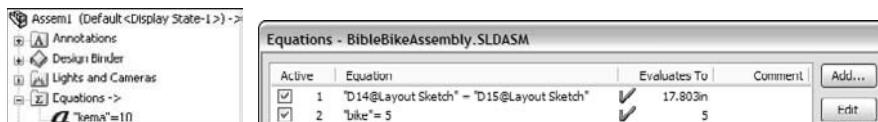
Cross-Reference

Equations are discussed in detail in Chapter 9. ■

FIGURE 12.2

An assembly equation driving one part from another

External references

Notice the "->" symbol after the Equations folder in the Assembly FeatureManager. This means that there is an external or in-context reference. An external reference means that an aspect of the part is dependent upon something outside of the part. This has file management implications because you must maintain the names of the files so that they always recognize the other file involved in the external relation. In-context means that one part has a relation to another part in positions determined by an assembly. So in this case, the in-context external reference can only be solved if the original part, the referenced part, and the assembly where the relationship was created are all open at the same time.

Cross-Reference

In-context references are discussed in depth in Chapter 16. ■

When one part drives another part in this way, the assembly must also be open to drive the relationship. If just the two parts are open individually, then changing the driving part does not update the driven part; because the relationship was created in the context of the assembly, the assembly must also be open to facilitate the change.

Link values and global variables

Link values and global variables also work in assemblies, but they do not work between parts. Local assembly sketches can use these functions, and the parts can use them when edited in the context of the assembly, but they cannot cross any document barriers (links must remain within a single document).

Renaming

Equations update with new part names regardless of how the part is renamed. Names of subassemblies also update when assembly files are renamed. This includes renaming a document using the Save As command, using SolidWorks Explorer, or using Windows Explorer. It also includes redirecting the assembly to the new part name, as well as renaming the assembly using each of these techniques. If the assembly can find the part and recognizes the part as the one that it is looking for, then the equation will work.

Some of the methods named previously for renaming parts are not recommended; for testing purposes I specifically tried to break the relationships in the equations by using them. SolidWorks Explorer and the Save As methods can be effective when used properly. References between files are a different issue altogether from an equation's references to local file names.

Recommendations

While assembly equations are certainly a valid way to control part sizes, I would recommend using assembly or part configurations, possibly with design tables, to accomplish something similar. Equations and configurations do not mix well because the two methods compete to control the dimensions. I recommend configurations with design tables over equations.

Cross-Reference

Assembly configurations are discussed in Chapter 14. Design tables are discussed in Chapter 10. ∎

Caution

You may have unexpected results if a single dimension is controlled from more than one location. For example, if you have a part-level equation and an assembly-level equation, then one of the equations will be automatically set to Read Only and will not be used. ∎

Assembly layout sketch

SolidWorks has an assembly feature called Layout that uses a 3D sketch to lay out the major functions of an assembly, and even details of parts. The word *layout* also refers to a technique using 2D sketches in an assembly to do exactly the same thing. The distinction between the technique and the formal assembly feature is bound to be confusing. SolidWorks' Layout feature only works in assemblies, but layout techniques have been used in parts as well as assemblies for many years. In this chapter I describe the technique, and leave the Layout feature for Chapter 16. When you look at the two functionalities, the feature is definitely intended to be used as an in-context tool, while the technique can be used most easily as a reference for controlling part position (through mating) rather than a way to directly control the sizes and shapes of the parts. So when I refer to a *Layout* (capital), I'm referring to the formal feature. When I refer to a *layout* or *layout sketch* (lower case), I'm referring to a technique where a sketch is used at the assembly level to control the assembly in some way.

Cross-Reference

The Layout feature is described in more detail in Chapter 16, while the technique using assembly sketches to lay out an assembly is described here. The material in this chapter is written as if the Layout feature does not exist, mainly to give you a straightforward view of how the technique works without worrying about two different functions at the same time. ∎

The layout sketch is a very useful tool for laying out a mechanism in an assembly or even details on parts within the assembly. Sketches in the assembly have the same characteristics as they do in the part environment. In Figure 12.3, the assembly layout sketch is indicated with a heavy dashed line for emphasis.

When combined with in-context techniques, assembly layout sketches can help to determine the shape of parts, or the location, size, or shape of features within the parts. You can also use layout sketches to mate assembly components to far more robust and dependable mates, rather than mating part to part. The sketch shown in Figure 12.3 is used for both of these techniques. The shape of the frame and the major pivot points are established in the 2D sketch. The wheels are also mated to the sketch.
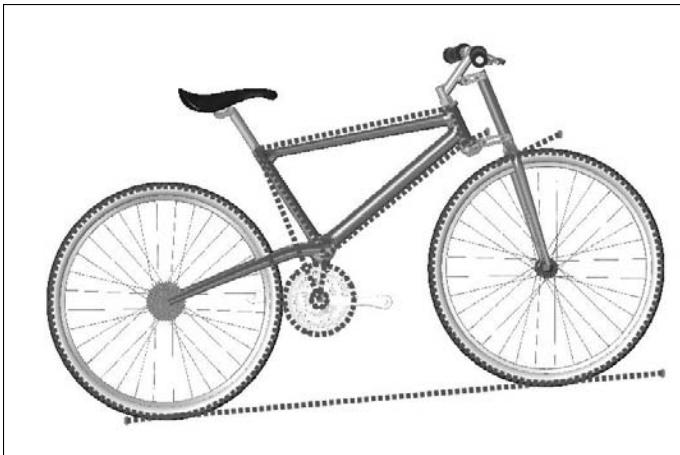
When you use an assembly layout sketch for either the in-context part building or simply part positioning, the main advantage that it offers is giving you a single driving sketch that enables you to change the size, shape, and position of the parts. You can use as many layout sketches as you want, and you can make them on different sketch planes. This enables you to control parts in all directions.

FIGURE 12.3

An assembly layout sketch

Caution

When using layout sketches, it is assumed that the relationships are created such that the sketch drives everything else. However, there is nothing preventing you from using other things in the assembly to drive the sketch. You should avoid this type of conflict, called a circular reference. It can create sketches that change with every rebuild and can seriously impact rebuild times. When using any type of in-context relations, you need to be careful to establish one or more driving entities, which are not in turn driven by other entities.

To take this a step further, it is best to avoid daisy chaining, where A drives B, B drives C, and so on. It is better practice to make A drive both B and C directly. This saves on rebuild times and troubleshooting. See the sidebar on using the skeleton or wide tree approach in Chapter 11 for more details on the benefits of this type of modeling and an example part. ■

One of the drawbacks of this technique is that you give up dynamic assembly motion. To move the parts, you have to move the sketch and rebuild. The part does not move until the sketch is updated. If you need to combine layout functionality with dynamic assembly motion, see the Layout feature in Chapter 16.

Virtual components

Virtual components are covered in more depth in Chapter 16. Virtual components are parts that are saved so they are internal to the assembly. You can save them out so that they are external to the assembly and can be reused in other assemblies. You can also convert external components to virtual components. Virtual components, as the name suggests, can be either parts or subassemblies.

Best Practice

Using virtual components is a technique that is useful for concept work in assemblies, but you will not see them show up on any best practice list. The main limitation of this technique shows up in the form of data management and reuse. I recommend limiting your use of virtual components because the technique promotes what many users and administrators consider to be sloppy practice. ■

Assembly reference geometry

Planes and axes are frequently created within assemblies to drive symmetry or placement of parts. You can use assembly layout sketches to create the reference geometry entities. When you create reference geometry within the assembly in this way, be aware that the normal history-based parent/child relationships are still followed. The familiar icons for reference geometry entities are also used in the assembly tree.

History-based and non-history-based portions of the assembly tree

Because features such as sketches and reference geometry are history based and found in the assembly tree, at least a portion of the assembly FeatureManager is history based. However, not all of it is. For example, the list of parts and subassemblies is not history based.

Sketches and reference geometry may appear before or after the list of parts, subassemblies, and mates. All the remaining entity types that can be found in the assembly FeatureManager are also history-based features, and you can reorder them in the tree. However, several situations can disrupt the process. Under normal circumstances, sketches and reference geometry at the top of the assembly FeatureManager are solved, then the parts are rebuilt if required, and then the mates. This ensures that the sketches and reference geometry are in the correct locations so that if parts are mated to them, all the components end up being the correct size and in the right position.

Assembly-level reference geometry can be created that references component geometry instead of layout sketches. This creates a dependency that changes the usual order. For example, the planes are usually solved before the part locations, but when the plane is *dependent* on the part location, the plane has to be solved *after* the part. If a part is then mated to the plane, you are beginning to create a dependency loop, such that the plane is solved, followed by the part, then the plane again because the part has moved; and then the mate that goes to the plane has to resolve the part.

Best Practice

If you are a bit confused by all of this, don't worry. You can simply follow this rule: Do not mate to anything that comes after the mates in the assembly FeatureManager tree. This includes assembly planes or sketches that are dependent on part geometry, assembly features such as cuts, in-context features, component pattern instances, Series Holes, or Smart Fasteners.

This is probably a lot of information if you are a new user, but if you remember this rule, you can avoid creating models with circular references, where A is dependent on B, which is dependent on A — a never-ending loop that causes major problems for large assembly rebuild times. ■

Parts and subassemblies

Parts and subassemblies are shown with their familiar icons in the design tree. You can reorder and group them in folders, which is covered in the next section.

Parts are sometimes shown with a feather, which indicates a lightweight part, and assemblies can have an icon that indicates a flexible subassembly.

Special icons also exist for hidden and suppressed components.

Folders

You can create folders to organize and group both parts and mates. I discuss this technique in detail later in this chapter.

Mates

The Mates area remains a constant, single folder, but you can organize it by reordering the mates and grouping them into folders. Each mate is shown with a symbol corresponding to the type of mate it is, but the mate folder is shown as a pair of paperclips.

Assembly features

In manufacturing, once parts are assembled, secondary machining operations are sometimes applied to them to ensure that holes line up properly, or for other purposes. For example, assembly features can be cut extrudes, cut revolves, or hole features. These features appear only in the assembly, not in the individual parts.

You should not confuse assembly features with in-context features. In-context features are created when you are editing a part in the assembly with a reference between parts, but the sketch and feature definition are in the part itself.

Component patterns and mirror components

Component patterns can pattern either parts or assemblies by creating either a pattern defined in the assembly, or a pattern that follows a pattern feature created in a part. The pattern is listed as a feature in the assembly FeatureManager, and all the instance parts appear indented from the pattern feature in the design tree. You can hide or suppress each instance, change its configuration, and in most ways control it as if it were a regular part in the design tree.

Because the options for locally defined patterns are comparatively limited, users generally like to use part feature patterns to drive the component patterns when possible.

Component patterns are listed at the bottom of the assembly FeatureManager with a set of components under a LocalPattern icon. The component instances under the LocalPattern can be controlled in several ways, including assigned configurations, colors, and display states. The pattern can even be dissolved, leaving the components, but dissolving the intelligent pattern that places them.

Mirror components are revamped in 2010 and are also listed under a special MirrorComponent icon after the mates.

Performance

To improve performance, it is best to pattern subassemblies if possible. If it is not possible, then patterning a group of parts is the next best option. Making multiple patterns, one for each part, is an inefficient way to accomplish the same thing. ∎

In-context reference update holders

It is difficult to get a good picture of assemblies in general without including a discussion about in-context references, but to treat the subject properly, it also requires its own section, and in fact, this book gives in-context modeling its own chapter (Chapter 16). When you create a reference between parts in an assembly, the assembly needs to remember which parts are involved in the reference, and what assembly creates the spatial relationship between them.

When you create the relation, a placeholder has to remain in the assembly to hold this information. This placeholder is called an Update Holder. The Update Holders do not display by default. To see them, you must right-click the top level in the FeatureManager and select Show Update Holders. They only exist when in-context references exist in the assembly, and there is one Update Holder for each in-context sketch or feature. You cannot do very much with the Update Holders, other than query them for parent/child relations and to list the external relations, but they serve as a reminder that you have in-context references to maintain. For more information on this feature, see Chapter 16.

Popular perceptions of in-context techniques aside, in-context modeling is a powerful extension of parametric design techniques. If you follow the best practice suggestions outlined in Chapter 16, you will soon gain confidence and master this technique rather than being frightened by it. The functionality works, and if you do not abuse it, it will serve you well.

Smart Fasteners

Smart Fasteners are assembly features that automatically select Toolbox parts for use in standard-sized holes, and you can use them in many different ways. The Smart Fastener feature in the assembly FeatureManager is used to edit the definition of the Smart Fastener, which can include adding items such as nuts and washers. You can also use Smart Fasteners in conjunction with the Hole Wizard to place appropriate holes and matched fasteners, all in a single step.

Cross-Reference
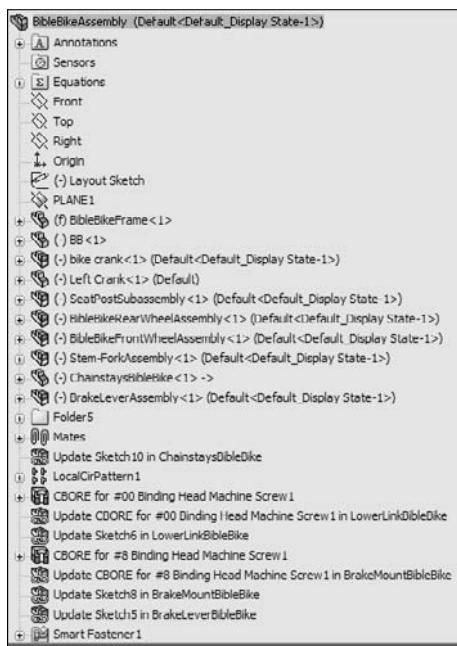
Smart Fasteners, Toolbox, and the Hole Wizard are discussed in detail in Chapter 17. This is the only functionality beyond what is found in SolidWorks Standard edition that I deal with in this book. ■

Hole Series

The Hole Series is a Hole Wizard–type feature that you apply in an assembly. This wizard leaves the feature icon in the assembly, but also adds features directly to the individual parts. It also adds in-context Update Holders to the assembly FeatureManager, as shown in Figure 12.4. The Series Hole is designed to go through a series of parts, placing the appropriate hole type in each part, counterbore, through, threaded, and so on.

FIGURE 12.4

Adding in-context Update Holders

Using SpeedPaks

A SpeedPak is a derived configuration of an assembly that keeps only selected solid bodies and faces, but can represent the rest of the assembly with non-selectable display data. A SpeedPak can be used to replace an entire subassembly within an upper-level assembly. SpeedPaks are intended to increase performance with very large assemblies and drawings.

Figure 12.5 shows first the SpeedPak PropertyManager, which you access by right-clicking an active configuration, and selecting Add SpeedPak. Each configuration can have only one SpeedPak.

Figure 12.5 also shows the configuration list with the SpeedPak listed indented under the Default config, and the entire assembly. The final image shows the SpeedPak inserted into an assembly document, consisting of a single face and two solid bodies. Notice the special icon associated with SpeedPaks. You can change a part in an assembly from or to a SpeedPak in the same way that you would change a configuration using Component Properties.

Remember this is a tool for increasing assembly speed, and to increase speed, there is always something that you have to give up. SpeedPak can be thought of in some ways like Lightweight assemblies and components in that it is display-only data. If your expectations of the tool are in line with the actual functionality, you will be very satisfied with the functionality SpeedPaks offer. For this reason it is important to understand the abilities and limitations of SpeedPak.

Using ghosts

You can use any faces or bodies that you select in the Include lists either manually or through the Quick Include sliders (which automatically select bodies and faces based on size) in assemblies to mate to or in drawings to dimension to. Any geometry that is not selected is included as a ghost — it displays, but cannot be selected. When the cursor gets near ghost geometry, the ghost fades away, revealing only selectable geometry. Notice at the bottom of the SpeedPak PropertyManager that you can also choose to remove the ghost data and further increase the memory savings.

Sharing self-contained data

The SpeedPak is self-contained. All the selected face and body geometry is saved inside the assembly. If you want to send someone a visual representation of an assembly, make a SpeedPak configuration and send only the assembly file — no parts are required. This is the equivalent of being able to put an eDrawing file into an assembly.
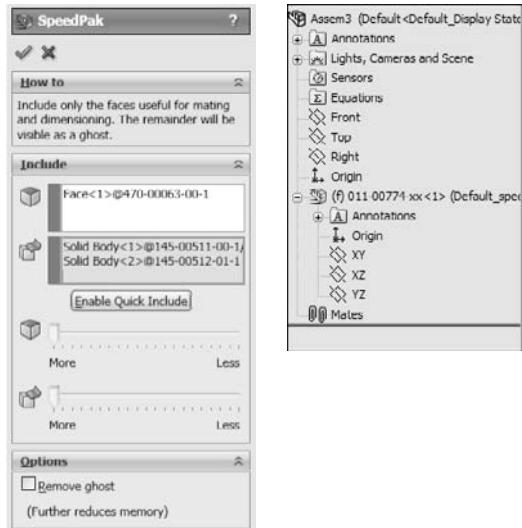
Using SpeedPak with drawings

You can even use SpeedPaks with drawings. Just remember that only edges created by the faces or bodies in the Include lists can be dimensioned to. Some functionality exists for the ghost data, such as BOM inclusion and numbered balloons. Ghost data displays as gray on the drawing, while geometry in the Include list is black.

FIGURE 12.5

Managing SpeedPaks

Model of Garmin assembly from the SolidWorks demo sets

Using Subassemblies

The primary tool for organizing assemblies is the subassembly. A subassembly is just a regular assembly that is used as a component in another assembly.

Best Practice

The number of levels of subassemblies is not limited to a specific number, although for different sizes and types of assemblies, I encourage you to establish a best practice for your company. For example, establish a guideline that suggests that subassemblies of 100 parts or less go no deeper than three levels. ∎

You can use several criteria to determine how subassemblies are assigned:

- Performance
- BOM
- Relative motion
- Pre-fabricated, off-the-shelf considerations
- According to assembly steps for a process drawing
- To simplify patterning

The underlying question here is based on the multiple functions of your SolidWorks assembly model. Is the assembly intended primarily for design? For visualization? For documentation? For process documentation? When used primarily for design, the assembly is used to determine fits, tolerances, mechanisms, and many other things. As a visualization tool, it simply has to look good and possibly move properly if that is part of the design. As a documentation tool, it is important how the model relates to the BOM, and the order in which subassemblies are added. As a process tool, you need to be able to show the assembly in various intermediate states of being assembled, likely with configurations.

I have seen companies create multiple assembly models for different purposes. Sometimes the requirements between the different methods are contradictory and cannot all be met at the same time with a single set of data. Again, depending on what information you need to be able to extract from your SolidWorks models, you may want to approach assembly modeling and organization differently, and you may need to create multiple assembly models to accomplish everything.

Creating subassemblies from existing parts

You can create subassemblies from parts that already exist in an assembly. To do this, select the parts that you want to add to the subassembly using Shift+, Ctrl+, or box select techniques, and then select Form New Subassembly Here from the right mouse button (RMB) menu. You are then prompted to assign a name or possibly select a template for the new subassembly.
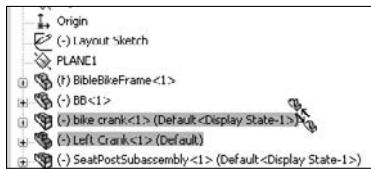
Caution

When creating a new subassembly from existing parts or when moving parts into or out of a subassembly from the upper-level assembly, some things may be lost. For example, mates are moved from the upper level to the subassembly. If you have in-context relationships, they may be removed. Operations that create subassemblies cannot be undone easily. ■

Once you have created the subassembly, you can add or remove components using the drag-and-drop method. For example, Figure 12.6 shows the cursor that indicates that the part named Left Crank is being moved into the subassembly named bike crank. To move a part out of a subassembly, simply drag the part into the upper-level assembly.

FIGURE 12.6

Moving parts into a subassembly

Note

When you are dragging a part out of an assembly and into another one, you may again see the cursor symbol that appears in Figure 12.6. If you do not want this to happen, hold down the Alt key while dragging. The cursor symbol changes to the Reorder cursor (a reversed, L-shaped arrow), and the part is placed after the subassembly rather than within it. ■

Insert a new subassembly

Along with the RMB menu option Form New Subassembly Here, which takes existing parts and puts them into a newly created subassembly, you can use another option called Insert New Subassembly. The names of these functions do not adequately describe the difference in what they do. Insert New Subassembly inserts a blank subassembly at the point in the design tree that you indicate by right-clicking it. You can place components into the subassembly by dragging and dropping them from the main assembly, or you can open the assembly in its own window and insert parts by using the usual methods.

Dissolving subassemblies

If you would like to get rid of a subassembly but want to keep its parts, then you can use the Dissolve Subassembly option through the RMB menu. This option has some of the same consequences of the Form New Subassembly Here option in that mates are moved from the subassembly to the upper-level assembly, and you may lose in-context relations and assembly features.

Organizing for performance

Performance in SolidWorks is a word used to mean speed. Subassemblies can contribute to speed-saving modeling techniques by segmenting the work that the software needs to do at any one time.

Solving mates

The mates that contribute to putting the pieces of an assembly together are solved at the level of the top assembly. Under normal circumstances, subassemblies are treated as static selections of parts that are welded together, and their mates are not solved at the same time that the top-level assemblies' mates are solved. This segmenting of the mates leads to improved performance by only solving one set of mates at a time.

Mates are usually solved as a single group unless there is a special situation, such as mates to in-context features, component pattern instances, or an assembly feature, all of which have already been described in this chapter. When one of these situations occurs, the mates have to be divided into separate groups or solved multiple times. This is done transparently behind the scenes so that the user does not have to worry about it. Multiple rebuilds affect the user only in terms of rebuild times.

Flexible subassemblies

When you create subassemblies, the mates for the parts of the subassembly are not solved in the upper-level assembly. This means that if a subassembly is a mechanism, the mechanism does not allow Dynamic Assembly Motion in the upper-level assembly. For example, in Figure 12.7, the front fork is a linkage mechanism, but it is also a subassembly. Without reassembling the parts of the fork in the upper-level assembly, you can allow the mates from the fork subassembly to be solved in the upper-level assembly by using the Flexible Subassembly option in the Component Properties dialog box, which is also shown in Figure 12.7. When you select the Flexible option in the Solve As section, you enable the mates of this subassembly to be solved in the upper-level assembly, which allows the parts of the subassembly to move in the upper-level assembly.

To access the Component Properties dialog box, right-click the subassembly and select Component Properties from the menu.

Flexible subassemblies have become more reliable and easier to use over the last several releases. I encourage you to work with them or do some experimentation to see if they assist your modeling process. If you find they cause trouble in some situations, they are easy enough to simply deactivate.
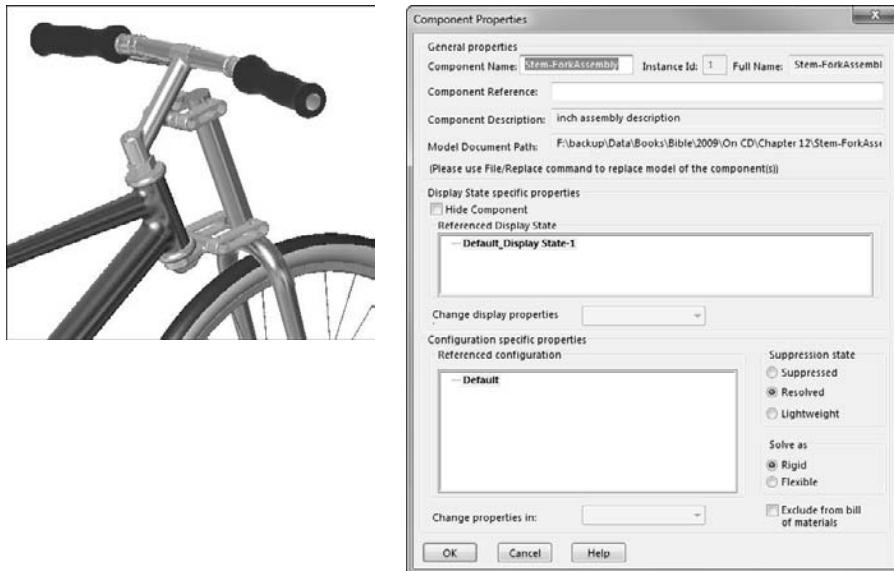
Legacy data

If you have assemblies that were built in older versions of SolidWorks (such as SolidWorks 2001+), mates used to be split up into multiple *mate groups*, which represented the groupings that mates were solved in. This was forced by mating to the history-based features in the assembly FeatureManager. SolidWorks no longer displays mate groups, but the groups are still used in the background to solve mates. This is another change that SolidWorks has made to the software that simplifies the user's interaction with the software, but also makes it obvious that things are now happening behind the scenes that you can't control.

Creating a flexible subassembly

Organizing for the BOM

The Bill of Materials, or BOM, is a table that is placed either into a drawing of an assembly or in an assembly itself. This table shows the parts used in the assembly and includes other information, such as part numbers, quantities, descriptions, and custom property data.

Cross-Reference

SolidWorks BOM functionality is discussed in depth in Chapter 24. ∎

Businesses often represent assemblies and subassemblies in various ways by using MRP (Manufacturing Resource Planning) or ERP (Enterprise Resource Planning) software. The methods that accountants and manufacturers use to organize assemblies are not always the same as those that engineers or designers might choose, but some companies require the BOM on the drawing to match the MRP or ERP Bill of Materials.

Best Practice
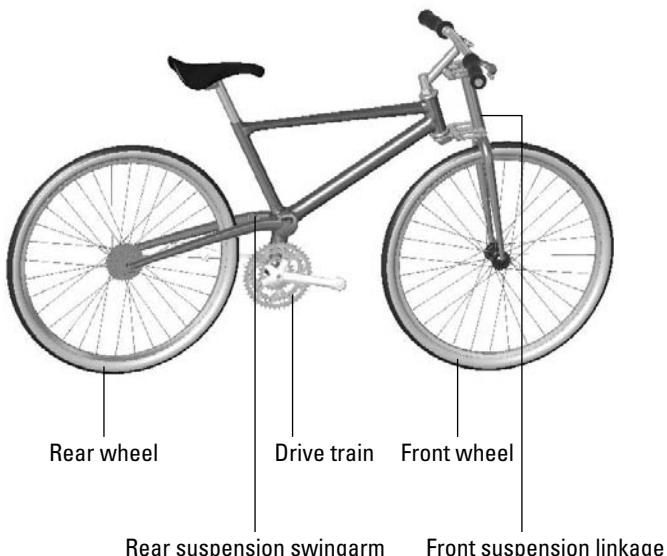
When you are forced into modeling something in an unnatural way to satisfy an outside requirement such as special BOM requirements, it might be best to detach the unnatural part and model normally. In the situation mentioned here — where MRP is forcing how the assembly is put together by requiring the BOM to match MRP — I recommend separating the BOM from the assembly structure rather than building an assembly that makes other SolidWorks' functions difficult. This ensures that the BOM becomes a manually maintained document.. Alternatives to this approach would be to make configurations or entirely new assembly documents to drive the BOM. ∎

Grouping subassemblies by relative motion

A more natural way to group subassemblies is by considering relative motion. In the bicycle example, each wheel is a separate subassembly because it moves as a unit relative to the rest of the assembly. Figure 12.8 illustrates where relative motion might be on the bicycle.

Grouping subassemblies by relative motion

Grouping subassemblies by relative motion is great for assembly modeling, but it does not usually reflect product reality very well. Using this method, you often end up with parts in the subassembly that will have to be disassembled in order to actually put the physical parts together. However, if your only consideration is ease of modeling, then this is probably the method to use.

Organizing groups of purchased components

If you are modeling a product that is created from a shopping list of purchased components, then it may make the most sense to organize your subassemblies into groups of parts that are purchased together. In fact, purchased subassemblies are often modeled as single parts, except when relative motion is required in the purchased assembly.

For example, in the bicycle assembly, the sprockets on the rear wheel are purchased as a separate unit, and yet the part that mounts onto the wheel moves relative to the sprockets that are driven by the chain. This is an example of a purchased part that would be modeled as a subassembly to show relative motion. The bicycle chain, another purchased subassembly, has not yet been added to this

assembly, and is a more complex model. The desire to show all of the individual links moving through the path may override both the complexity of assembling it and the performance considerations of exercising all of the mates.
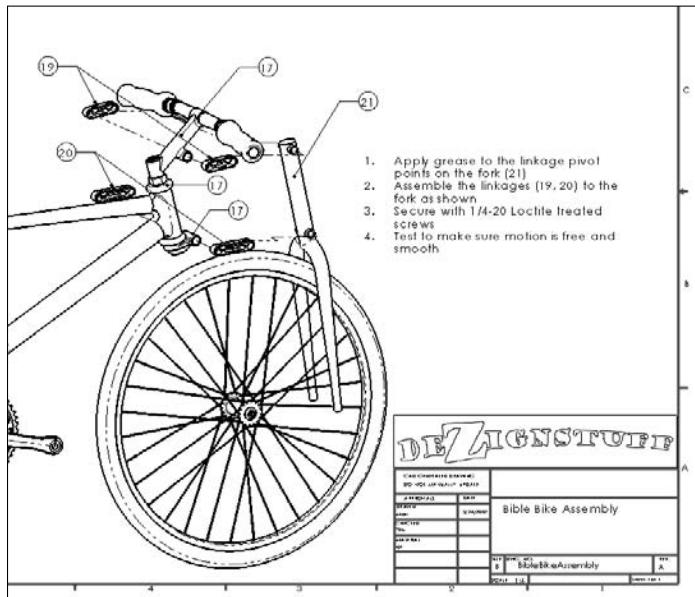
Although the BOM method of organizing assemblies sometimes leads to unnatural solutions, you should not discard it altogether. If you can devise concessions in order to make the BOM work automatically, then you should do this.

Depicting an assembly process

Manufacturing and assembly processes need to be documented as well as individual part design. You often need to create exploded-view assembly instructions for manufacturing or service documentation at each step of a multi-step assembly process. Figure 12.9 shows a page of this type of process documentation.

FIGURE 12.9

Assembly process documentation

This is certainly a task that is different from the initial design or modeling of the assembly, and it may require an entirely separate assembly model. Generally, you can perform the different steps by using a separate configuration for each process step, with exploded views for each configuration.

Item numbering

Balloons number the parts according to the item number that is used in the BOM, but, of course, you do not know the item numbers until the BOM is created. You can influence the item numbers by reordering the parts in the assembly (which I discuss later in this chapter), by manually editing item numbers, or by manually numbering the balloons. I cover all of this in detail in Chapter 24.

Individual steps

Each step corresponds to an assembly configuration (discussed in Chapter 14), and you can place them on a separate sheet of the drawing (discussed in Chapter 21). Each configuration can have multiple exploded views, if necessary, to show all the steps.

Patterning considerations

The most efficient way to pattern large numbers of components in an assembly is to pattern a single subassembly with all the components to be patterned in it. While this may not be easily combined with some of the other considerations that I mentioned previously, it is another option that you can use to organize assemblies.

Using Folders

Folders are primarily used in the assembly FeatureManager for grouping parts and mates into either special classifications for easy browsing, or groups that can be easily hidden and shown, or suppressed and unsuppressed, as appropriate. Figure 12.10 shows some examples of these folders.

FIGURE 12.10

Folders that are used to organize components and mates

Creating folders in the FeatureManager

You can add folders to the assembly FeatureManager in one of two ways:

- Add existing components to a new folder
- Create an empty new folder

Add To New Folder tool

To use the Add To New Folder tool, right-click a component or mate (or selection of components or mates) and select Add To New Folder from the menu. This moves the component or mate into the folder. Folders do not affect the assembly in any functional way; they are simply for organization, to speed browsing and selection.

Create new folder

To simply create a new folder without putting anything into it right away, right-click either the Mates area or the components list, and select Create New Folder from the menu.

Adding items to existing folders

To move an item into an existing folder, just drag the item (component or mate) onto the folder. If the folder is expanded, showing its contents, then you can also drag the item as if you were reordering a feature in the FeatureManager of a part, and drop it in the list of items where you would like it.

If you are dragging a part or assembly and trying to put it immediately after an assembly, a cursor like the one shown in the center in Figure 12.11, may appear. This cursor means that the part is going to become part of the assembly. If this is not what you are trying to do, then hold down the Alt key while dragging; the part is placed into the folder immediately after the assembly, instead of being made a part of the assembly. The third image in Figure 12.11 shows the cursor with the Alt key pressed.

Reordering items in the tree

There are times when you may want to reorder items in the assembly tree. For example, you may want to place items close to one another in the tree, or you may be preparing to put items that are next to one another into a single folder. You may want to reorganize components for the BOM display.

You can reorder mates simply by dragging them. Mates display in the order in which they are created, but the order is not significant. They can be reordered however you like.

Components also display in the order in which they are added to the assembly, and you can reorder them in any way you like.

FIGURE 12.11

Moving items into folders

Best Practice

It is often useful to have an ordering strategy that helps you work with the model. I usually try to keep the biggest parts, the parts that everything else is mated to, or the part that is treated as "ground," as the first part(s) in the assembly. I put the fasteners and other cosmetic or BOM-driving parts at the end of the tree, usually in a descriptive folder. ■

Working with Tree Display Options

Display options for items in the FeatureManager are often overlooked, but can be useful for displaying data about parts, subassemblies, mates, and features. Figure 12.12 shows the RMB options. You must right-click the top-level assembly name in the FeatureManager to access this menu.
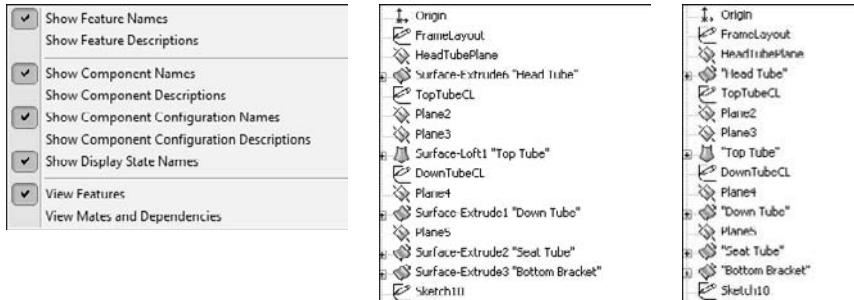
Note

All these options are available for parts and drawings as well, except for the View Features option and the View Mates and Dependencies option, which are related to assemblies. ■

Tree display options in assemblies

Showing feature names and descriptions

If you are so thorough that you have added descriptions to your features, then you are doing well. This option refers to the descriptions of features in the parts. The middle image in Figure 12.12 shows a section of the FeatureManager for the bicycle frame part; some of the features have had descriptions added with both of the options for feature names and feature descriptions turned on. The image to the right in Figure 12.12 shows the result of turning off the feature name, with only the feature description option turned on. If no feature description has been created, then the feature name displays. Feature descriptions always appear in double quotes.

Showing component and config names and descriptions
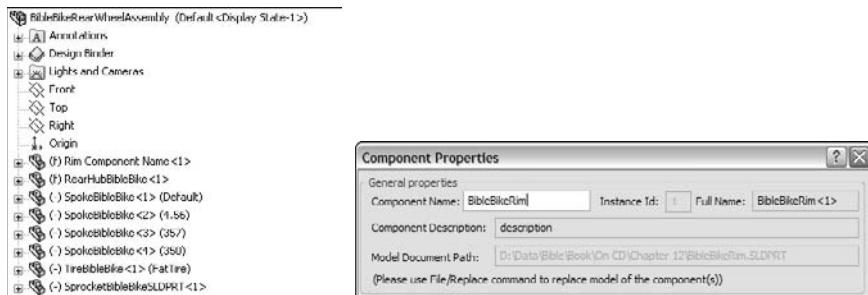
The image on the left in Figure 12.13 shows the default arrangement of displaying component and configuration names in the assembly FeatureManager. This example uses the rear wheel assembly from the bicycle assembly. In assemblies, you cannot turn off the component names and the component descriptions. SolidWorks issues this statement when you try to turn off both the name and the description: "You cannot hide both the component name and the file name. At least one must be visible in the tree."

Using names other than the part filename in the assembly FeatureManager

The message in the previous paragraph distinguishes between the *filename* and the *component name* that is listed for individual parts or subassemblies in the assembly FeatureManager. You can specify the component name in the Component Properties dialog box by right-clicking the component in the assembly FeatureManager and selecting Component Properties.

FIGURE 12.13

Component name options

However, you can assign a component name for a component that is different from the filename only if a special setting is turned off. (The setting defaults to on.) You can access this setting by choosing Tools ➪ Options ➪ External References ➪ Update Component Names When Documents Are Replaced. What the name of the setting has to do with showing a component name in the FeatureManager is not exactly clear, but these are the steps that you have to go through if you want to change how the part and subassembly names display in the assembly FeatureManager.

After going through the preceding steps, you still receive the warning message about the filename and the component name. This is apparently an "unintended feature," or bug, and should likely say that you cannot turn off both the component name and the component description at the same time.

There may be a situation where you want to show a name other than the filename in the FeatureManager. For example, your company may be using sequential part numbers for the filenames that are difficult to read, and you want something descriptive in the design tree so that you do not need a cross-reference sheet next to your computer that equates filenames to meaningful descriptions.

If you prefer, you can avoid a lot of problems by just using the part description instead of the filename or the component name. The choice is up to you.
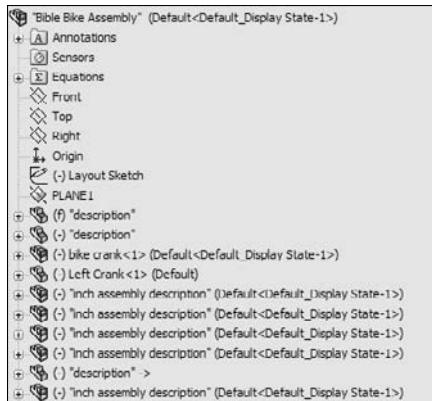
One of the problems with using the component description is shown in Figure 12.14, where the user did not enter proper descriptions for the parts, and SolidWorks used the default description in the templates. The tire and spokes use configurations, which display in the figure.

FIGURE 12.14

An example of what can occur when you do not enter proper descriptions for parts

Using Component Reference per Instance

SolidWorks 2010 adds Component Reference capabilities. In the Component Properties dialog, shown earlier in Figure 12.7, the second row at the top enables you to enter Component Reference information. This is typically used in electrical diagrams for similar components with different values, such as power ratings, capacitance or resistance. Instances of the same component in a SolidWorks assembly that have the same Component Reference can be listed together on a BOM. Instances with different Component References are listed separately on the BOM.

Figure 12.15 shows parts listed in a pattern that have Component References listed for each instance.

In order for Component References to be used in balloons on an assembly drawing, the drawing must have a BOM with a Component Reference column. BOMs are handled in detail in Chapter 24, where this topic of Component References will be revisited.

FIGURE 12.15

Listing parts in an assembly containing Component Reference information
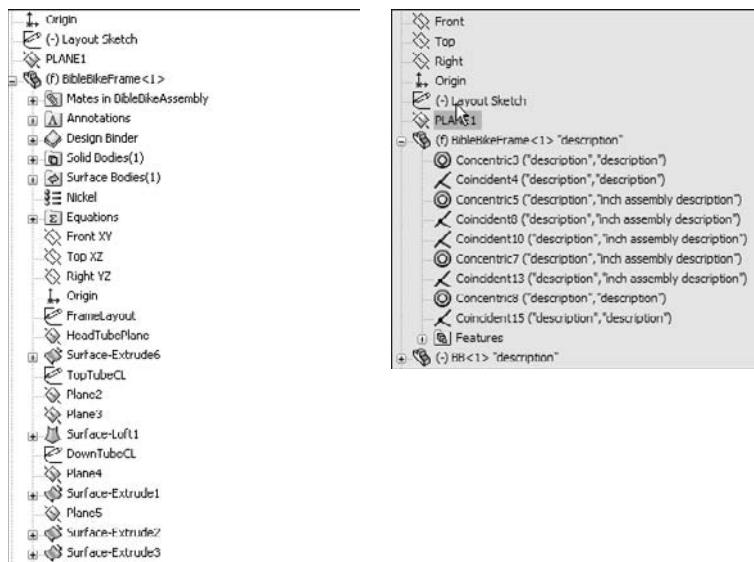
Viewing features, mates, and dependencies

The last set of options shown in Figure 12.12, determines if you see the part features or the assembly mates after the name of each component in the assembly tree. The default setting is for the part's features or the subassembly's components to display, just as if the part or subassembly were open in its own window.

The View Mates and Dependencies option can also show the features, but they are placed into a separate folder. This option makes it very easy to see the mates that are assigned to an individual component. For example, in Figure 12.16, the image to the right shows the mates directly under the BibleBikeFrame part. This often makes troubleshooting much easier because it isolates the mates for a single part. Notice also that the first folder under the part name in the image to the left in Figure 12.16 is the Mates folder. This indicates that, regardless of whether you choose to display mates or features, you always have easy access to the other type.

FIGURE 12.16

You can view features, as well as mates and dependencies.

The View Mates tool is extremely valuable for looking at how an assembly is held together with mates. When you right click on a component in the assembly and choose View Mates from the RMB menu, SolidWorks highlights the component you clicked on, and makes all parts that mate to that component transparent. Any parts that are not related are hidden. SolidWorks also opens up a small dialog box with the list of mates touching the component you clicked on.
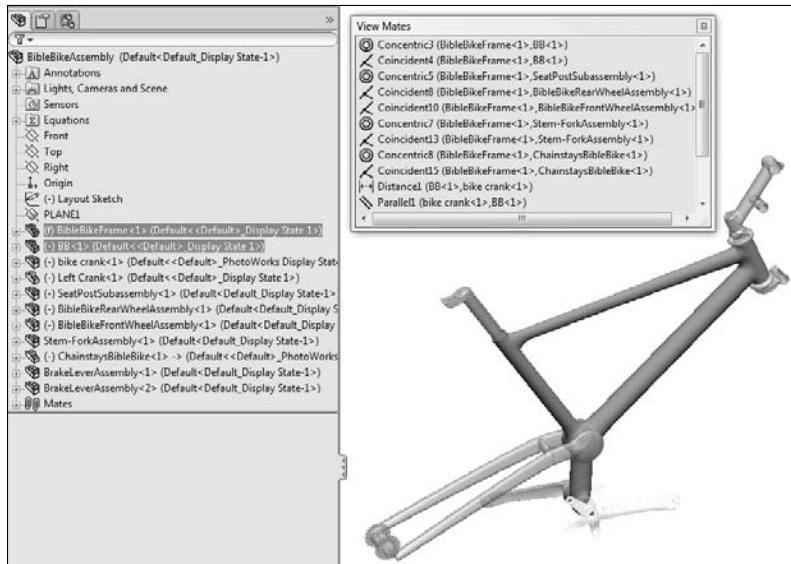
Figure 12.17 shows this arrangement using the Bible Bike assembly. If you are a SolidWorks veteran, this works very differently from the way View Mates worked in the past, but I hope you will agree with me that this is a big help in mate visualization.

If you Ctrl-select multiple components before starting the View Mates tool, SolidWorks no longer puts the common mates in bold, it just lists them at the top of the dialog box.

FIGURE 12.17

Showing mates in the PropertyManager pane

Using Assembly Tools

SolidWorks is incredibly complex software, and there are many tools to do many small tasks. This is true nowhere more than it is in assemblies. Users have such wide expectations of the software in various areas, and everyone needs something different. Many tools exist to help you put together, maintain, and evaluate efficient assemblies. The tools in this section are important tools that don't fit into the other categories.

Using Sensors
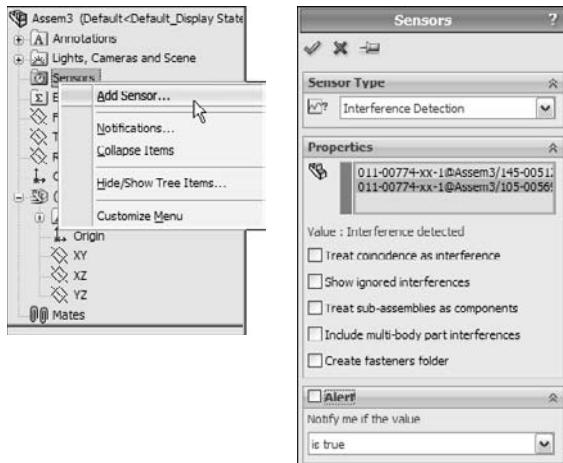
Sensors provide an alert if a monitored value goes outside of a specified range. Sensors can be used in parts or in assemblies. The types of values that you can monitor with Sensors are dimensional values, mass, volume or surface area, interference detection between select components, and Simulation data (stress analysis values).

To create a sensor in an assembly, right-click the Sensors folder in the assembly FeatureManager. If the Sensors folder is not there, choose Tools ➪ Options ➪ FeatureManager page and turn it on. Figure 12.18 shows the Sensor setup interface.

Setting up a sensor in an assembly

Using the AssemblyXpert

You can find the AssemblyXpertcan on the Tools menu, or choose Tools ➪ Customize from the menu to place it on the Assembly toolbar. AssemblyXpert gives suggestions about things you can do to improve the performance of an assembly, such as updating files to the new version and looking at Large Assembly Mode, and existing errors in the assembly mates, in-context, and supposedly circular references.

I was not able to confirm that AssemblyXpert would find a circular reference, but SolidWorks and I might be using different definitions of circular reference. My definition is a list of parts referencing other parts where the references form a loop, with one part as the start and end point. AssemblyXpert could not find this kind of reference loop. SolidWorks may be using a more explicit definition where specific geometry within a part is both driving and driven.

Figure 12.19 shows the AssemblyXpert results. Notice that the results include the information formerly included with the Assembly Statistics: part and subassembly count, along with mates, unique parts, and so on.
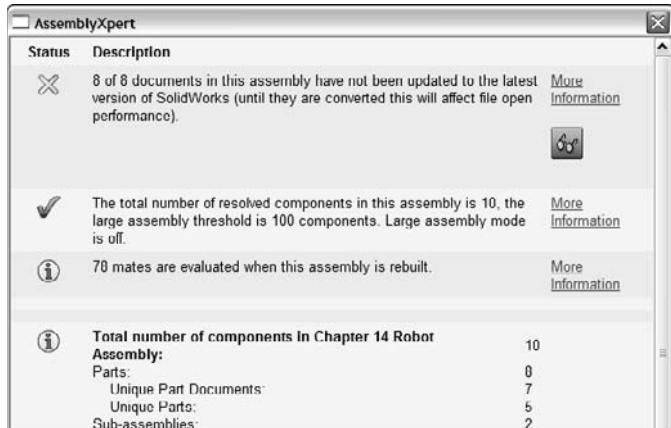
The information in the AssemblyXpert is certainly useful, particularly for newish users who may not know to look for things like this already. You can use it as both a diagnostic and a learning tool.

FIGURE 12.19

AssemblyXpert results

Tutorial: Arranging Assemblies

In this tutorial, you take an assembly that is already put together and group its components into subassemblies, and then convert one subassembly into a flexible subassembly. Note that some of the commands and RMB options you are asked to select may not be shown on the truncated RMB menus. To remedy this, click the double-arrow at the bottom of the RMB menu or choose Tools ⇨ Customize ⇨ Options from the menu, and click the Show All button for both shortcuts and menu customization.

Follow these steps to learn how to effectively arrange items in an assembly:

1. **Start by opening the** `Chapter 12 Robot Assembly .sldasm` **file from the CD-ROM, Chapter 12 folder.**

   Notice that the names for the files are long and somewhat difficult to read. This would also apply for files that use sequential numbers for the file name instead of a descriptive name.

2. **To display a more readable name, right-click the name of the assembly at the top of the FeatureManager, select Tree Display from the menu, and then turn on Show Component Descriptions.** Repeat these steps and this time turn off Show Component Names.

   Figure 12.20 shows the display of the FeatureManager after the change. Even the top-level assembly uses its description rather than its file name.

   Notice that two components still use their clumsy file names rather than easy-to-read descriptions. This is because descriptions were never entered for those two components.

FIGURE 12.20

Simplifying the FeatureManager display to include descriptions

3. **Open the Large Cylinder Piston part by left-clicking the part in the FeatureManager and clicking the Open icon.** Choose File ➪ Properties from the menu and make sure the Custom tab is active. Create a new property called **description**, assign a type of text, and then enter **large cylinder piston** for the value. Save the part (Ctrl+S is a fast way to do that) and then flip back to the assembly (the fastest way to do that is to hold down Ctrl and press Tab).

   If the display has not yet updated, press Ctrl+Q to force the tree to rebuild.

4. **Open the Large Cylinder Body part.** In this part, choose File ➪ Save As from the menu. Leave the name the same, but where it says Description, enter **large cylinder body**, as shown in Figure 12.21. Click Yes when asked if you want to replace the document of the same name. Flip back to the assembly when you are done. You may have to rebuild to see the change update.

5. **Ctrl+ select the large cylinder piston and the large cylinder body parts from the FeatureManager.** Then right-click and select Form New Assembly Here from the menu. If your assembly template has a description, it will appear in the FeatureManager. If not, the file name will appear.

   You have just created an assembly as a virtual component while the parts are external documents. If you switch the tree display to show file names briefly, you will see what is shown in Figure 12.22, the name of the assembly is Assem1^Chapter 12 Robot Assembly. So the virtual component gets a default name (Assem1) followed by the name of the parent assembly (Chapter 12 Robot Assembly) to ensure that it has a unique name, if there are other virtual components in other assemblies.

Adding a description in the Save As dialog box
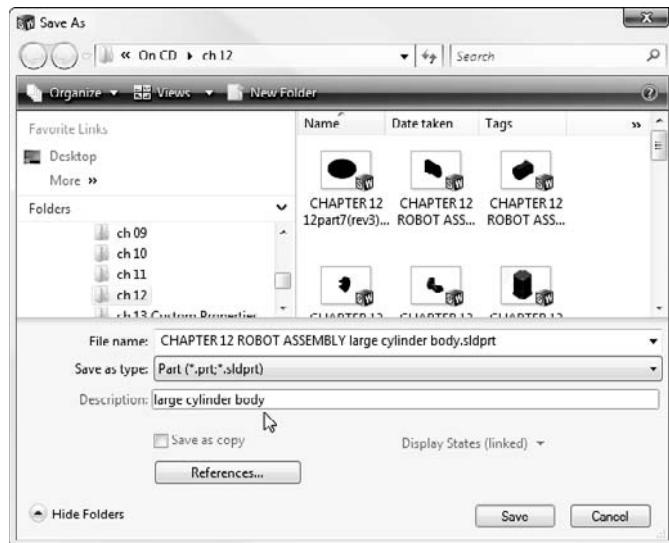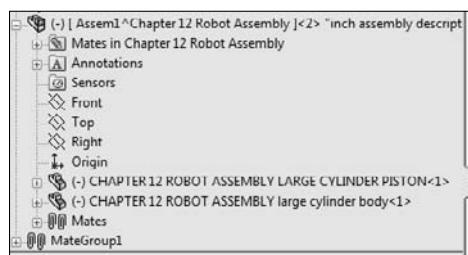
The newly created virtual component subassembly and its external parts
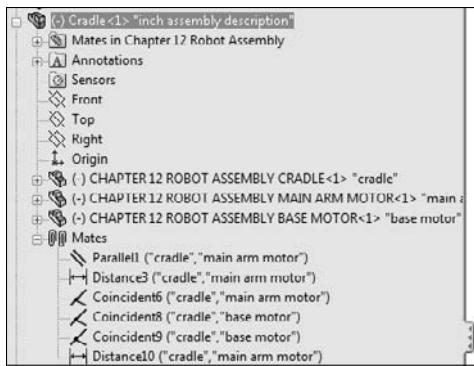
6. **Ctrl+ select the Base Motor, the Main Arm Motor, and the Cradle, and make another new subassembly.** After this one is created, right-click the subassembly and select Save Assembly (In External File) from the menu. Change the name of the assembly from Assem2 to Cradle. Save it in the same path as the rest of the parts.

Once a virtual component is saved externally, you cannot use undo to reverse it, but you can right-click the external file in the FeatureManager and select Make Virtual from the menu.

In Figure 12.23, notice that several of the mates came along into the new subassembly. These are the mates between the motors and the cradle. The mates that locate the cradle to the other parts in the assembly have remained in the upper-level assembly.

FIGURE 12.23

The cradle assembly brings along internal mates

Note that you cannot undo the subassembly operations. If you need to remove the assembly but keep the parts, right-click the assembly and select Dissolve Subassembly from the menu.

If you try to move the parts in the assembly, you should notice that all parts work as they should except that the Main Arm does not move up and down. This is because you turned the cylinder and piston into a subassembly, and the subassembly mates are not solved in the top level, which means the subassembly cannot move within itself.

To get around this, you need to make the subassembly into a flexible subassembly, which will solve the subassembly's mates in the top-level assembly,

7. **Right-click the cylinder subassembly in the FeatureManager, and select Component Properties from the menu.** In the lower-right corner of the dialog box, there is an option to Solve as Rigid or Flexible. Change this setting to Flexible.

After you click OK and return to the assembly, the main arm should move as it did originally, and the piston should move in and out of the cylinder. Notice that the symbol for the assembly changes when it is changed to a flexible subassembly.

Tutorial: Managing the FeatureManager

This tutorial uses the `BibleBikeAssembly.sldasm` file found on the CD-ROM for Chapter 12. Open the file and follow these steps to learn about managing the FeatureManager:

1. **Create a new subassembly within the existing assembly using the parts BibleBikeFrame and ChainstayBibleBike.** Name this new assembly `FrameAssembly.SLDASM`.

2. **Reorder the new FrameAssembly to the top of the design tree**.
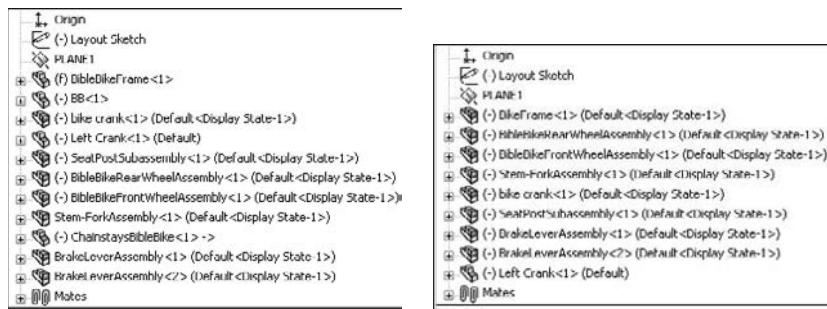
3. **Reorder the other parts and assemblies so that the bigger assemblies appear higher on the list, and the parts appear at the bottom.** (Remember that the Alt+drag option prevents a component from being placed into a subassembly.)

4. **Drag the part called BB (for Bottom Bracket) into the Frame Assembly (drag without using the Alt key).** The assembly FeatureManager at this point is shown in Figure 12.24.

FIGURE 12.24

The starting state and the state as of Step 4

5. **Select both wheels and then select Add To New Folder from the RMB menu.** Name the new folder **Wheels**, and move it to the bottom of the tree.

6. **Expand the Mates folder, select the first four mates, and put them in a new folder (select Add To New Folder from the RMB menu).** Name the new folder **Centering Mates**.

Summary

Assemblies are more than simply parts and subassemblies put together with mate relationships; several other types of features and placeholders can also exist in the assembly FeatureManager. Organizing assembly components is fairly straightforward and can offer benefits for finding parts as well as controlling suppression and display states globally.

The assembly FeatureManager contains several options for the data to display for subassemblies, parts, configurations, and features within. Remember that all the data that you include in your SolidWorks documents can be accessed and reused later on, so it is worth the effort to name it properly. Descriptions can be very important, both at the part level and also for features and configs.

Getting More from Mates

I n SolidWorks assemblies, mates are the basic units that either make everything work together and function properly, or fill the tree with errors and warning symbols. When properly handled, mates enable your assembly to react predictably to changes in parts exactly the same way that sketch relations drive changes in part features. As a result, mates and sketch relations often have the same function and even the same weaknesses to watch out for.

This chapter goes one step further with mates, by not just simply putting parts together with Coincident and Concentric mates, but also mating parts when tolerances, gaps, and symmetry become issues. You will also learn about the more advanced mate types that may be useful for special situations.

One of the assumptions that I make in this chapter is that assembly mates are not just used for positioning parts, but also for motion. Making motion work takes a little more than just establishing the right spatial relationship between parts; it usually also involves analyzing the open degrees of freedom.

From time to time, I have met users who take a static approach to putting parts together into assemblies, by simply placing parts at the correct X- and Y- coordinates without assigning any relationships to the parts around them. This defeats most of the purpose of creating parametric, associative assemblies in the first place. Assembly mates are an extremely powerful tool for enabling your assemblies to react predictably to change.

Applying Mates

An average assembly of 100 parts is likely to have almost 300 individual mates. If you create these parts one at a time, taking perhaps a minute for each mate, you would spend five hours just applying mates. Any time that you can save applying mates is a benefit to you — assuming you still get the correct results — because it is time that you can spend doing something else. In this section, you will learn efficient mating strategies, as well as speedy techniques.

Mating through the Mate PropertyManager

The Mate PropertyManager is the default method for applying mates, and was used briefly in Chapter 4. The Mate PropertyManager interface is shown in Figure 13.1. You can create mates by pre-selecting entities before applying the Mate command or selecting them after you open the Mate PropertyManager. The three types of mates are Standard, Advanced, and Mechanical.

FIGURE 13.1

The Mate PropertyManager interface

Mate workflow

If you make a lot of mates, it is important to have an efficient rhythm when working with the interface. Assuming you have the Mate PropertyManager already active, the most efficient way to use the Mate interface is as follows:

1. **Click the first entity.**

2. **Click the second entity.**

3. **Click OK on the right mouse button (RMB) cursor icon to accept the default.**

   Or, if the automatic default mate type is not the mate that you want to apply, select it from the popup list, which is shown in Figure 13.2.

   FIGURE 13.2

   The Mate selection popup list

4. **Click the green check mark icon from the popup list.**

5. **Repeat Steps 1 to 4.**

6. **After the last mate, press Esc, the green check mark icon, or the red X icon from either the PropertyManager or the confirmation corner in the upper-right corner of the graphics area.**

View and model positioning

Sometimes you will have to rotate the model to achieve the correct view in order to select faces or edges. There are also times when you will want to *pre-position* so that the model snaps into the correct position automatically. You can rotate individual parts in an assembly by clicking and dragging with the RMB. (Dragging the RMB over a part rotates that part). You rotate the view by dragging with the middle mouse button, or MMB. You can move parts by dragging them with the left mouse button, or LMB. You can pan the view by pressing Ctrl and dragging with the MMB. When you drag a part with the LMB while the Mate PropertyManager is active, SolidWorks does not add the selected entity to the Mate Selections list.

To summarize these actions:

- To rotate an individual component in an assembly, click and drag with the RMB.

- To move an individual component in an assembly, drag with the LMB.

- To rotate an assembly view, drag with the MMB.

- To pan an assembly view, Ctrl+drag with the MMB.

Also be aware of the view manipulation tools, available by clicking the triad in the lower-left corner:

- To rotate normal to an axis, click its triad axis.
- To rotate by 15° about an axis (angle can be specified at Tools ⇨ Options ⇨ View), Alt+click its triad axis.
- To rotate by 90° about an axis, Shift+click its triad access.
- To activate the mouse gesture wheel, drag the RMB in blank space (dragging the RMB over a part rotates the part).
- To zoom to fit, double MMB-click in the graphics window (same as the F hotkey)

Tip

If you have a Spaceball or 3D motion controller, you can perform all these actions more easily and simultaneously using one hand for view rotations and the other hand for selections. You can also use a Spaceball to move parts. ■
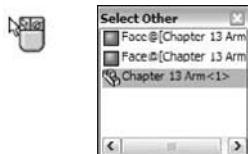
Select Other

The Select Other command enables you to select items that are hidden by other items. It is often used to select faces that are hidden behind other faces without rotating the part. You can apply the Select Other command through the RMB menu. Right-click where the face would be if you could see it. A list of entities displays and you can select the entity you want from this list or from the graphics window.

Moving your mouse over an entity in the list highlights the entity in the graphics window. Pressing Tab or scrolling the mouse wheel cycles through the entities one by one. Clicking faces with the RMB hides them, which enables you to see further down into the part or assembly. Clicking with the LMB either in the graphics window or the selection list box selects the item. Figure 13.3 shows the Select Other cursor and dialog box.

FIGURE 13.3

The Select Other cursor and dialog box

Although this selection method is also used for other purposes, it is often used for selecting faces for mating.

Multiple Mate mode

Multiple Mate mode enables you to select one face in order to mate multiple parts to it. Figure 13.4 shows the interface for this mode, which you can toggle to from the Mate PropertyManager interface. Multiple Mate mode is the paperclip with the lightning bolt icon without any text on it. This function works only with the Standard Mate types, not with any of the Advanced mates, which I discuss later in this chapter.

FIGURE 13.4

The Multiple Mate mode interface

You can create a special folder for all the multiple mates by selecting the Create multi-mate folder check box in the Mate Selections PropertyManager. You can also automatically link the values for distance and angle mates with link values by selecting the Link dimensions check box.

Taking advantage of SmartMates

SmartMates are mates that you can create automatically by dragging one part onto the other without invoking the Mate command. There are three different methods that you can use to apply SmartMates:

- Alt+dragging the part
- Dragging the part from one window to another
- Using Mate references

Alt+dragging a SmartMate

Probably the easiest way to quickly create a SmartMate is by Alt+dragging. One, two, or even three mates can be applied at once by holding down the Alt key while dragging a face or edge from one part onto a face or edge on another part.
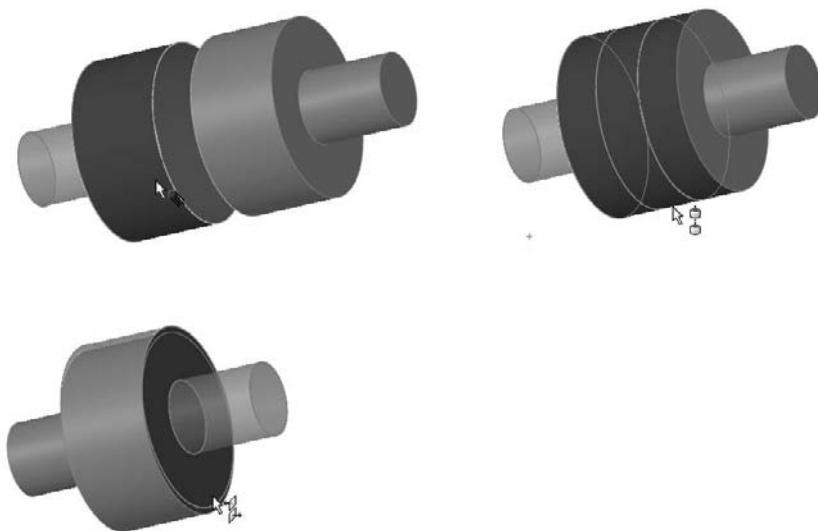
When you are dragging a part while pressing the Alt key, the part is made transparent to enable you to see other part faces that you may want to mate it to. A special cursor appears when a SmartMate is about to be applied. Figure 13.5 shows the cursors that appear for adding Concentric and Coincident mates.

Applying a SmartMate

When you drop the face or edge onto the mating face or edge to complete the mate, you must use the popup Mate toolbar to accept or alter the mate. In the examples in Figure 13.6, a face is being dragged onto another face. However, you can also drag edges and vertices. Mates are limited to being either Coincident or Concentric.
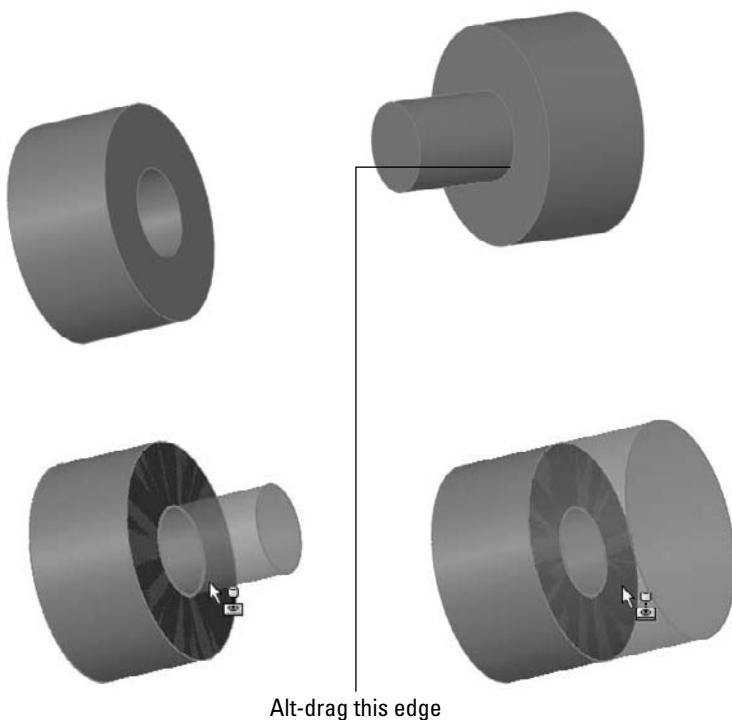
The *peg-in-hole* mate is actually the combination of a Concentric mate and a Coincident mate. This is the type of mate that is created between a screw and a hole, and is the result of Alt+dragging a circular edge onto a circular edge. When the circular edges are created by the intersection of a cylindrical face and a flat face, the Concentric mate goes between the two cylindrical faces, and the Coincident mate goes between the flat faces. The peg-in-hole mate is illustrated in Figure 13.6. The top two images show the state of the parts before the SmartMate. The image in the lower left shows the SmartMate orienting the part in the wrong way so that the two parts interfere. In the image in the lower right, the part to which the SmartMate is applied has been reoriented by the Tab being pressed before the SmartMate is accepted by the part being dropped.

Tip

You can press the Tab key to flip the alignment if a SmartMate tries to put parts together in the wrong way. If you are in the process of Alt+dragging, make sure to release the Alt key before pressing Tab. The Alt+Tab combination is a Windows shortcut to show a list of open applications. ∎

Using SmartMate to create the peg-in-hole mate combination

Alt-drag this edge

> You can also Alt+drag a SmartMate to mate origins and coordinate systems. When you Alt+drag one origin onto another, you get the special cursor shown here, and when you release the mouse button, the popup option box shown in Figure 13.7 appears. If you only want the origin points mated Coincident, turn off the Align Axes option. With the Align Axes option on, the origin points will be Coincident and the X, Y, and Z directions will be parallel to the same directions in the counterpart origin.

Using Align Axes option on the popup option box for origin-to-origin SmartMates

Drag between windows

You can simply Ctrl+drag a face of the part to the face of another part in a different SolidWorks window. It is probably most useful to tile windows before creating this kind of SmartMate.

Mate references

Mate references are model faces, edges, or vertices that are pre-selected and used in a SmartMate-like fashion when a part is dragged in from Windows Explorer or from a library window. Mate references are discussed in Chapter 19 in the course of discussing library parts. They are a great way to automate common mates with commonly used parts, such as library parts.

Mating with macros

If all the confirmations and extra mouse-clicks to open and close windows are not for you, and you are just applying simple mates, then you may want to use macros to mate parts. Macros are not going to give you the same flexibility, but for simple and predictable mates, they do vastly improve speed. You have to have the parts ready to go when you press the macro button, or you will create the wrong mate.

You can find macros for Coincident, Concentric, Parallel, Perpendicular, and Tangent mates in the CD-ROM folder for Chapter 13. For example, to use the concentric macro, you would need to pre-position the parts so that they are within 90 degrees of the proper alignment, have one of the parts mated in place such that that only one part will move, select the two cylindrical faces, and then run the macro. Ideally, the macro would be connected to a hotkey, so the workflow for this process would be extraordinarily fast. You would click one face, click the other face, then press the hotkey, and the parts would fly together.

Note

To connect a macro to a hotkey, first put the macro in a folder called macros under the SolidWorks installation directory, and then restart SolidWorks. Then use the Keyboard dialog box (Tools ⇨ Customize ⇨ Keyboard) to assign hotkeys to the macros in the list. ■

Like SmartMates, macros work best for the simpler mate types where you do not need to select any options. The workflow with macros can be very fast, but you have to have the parts pre-positioned and be very sure of what you want.

Mating for Motion

Dynamic Assembly Motion is a powerful tool for visualizing the motion of mechanisms in SolidWorks. It works best if there is a single open *degree of freedom*.

Analyzing degree-of-freedom

When working with motion in SolidWorks, you need to be comfortable with the concept of degrees of freedom. When inserted into an assembly, each model begins with six degrees of freedom:

- Translation in X (tX)
- Translation in Y (tY)
- Translation in Z (tZ)
- Rotation about X (rX)
- Rotation about Y (rY)
- Rotation about Z (rZ)

When applying mates, and especially when troubleshooting motion or overdefinition problems, it is important to look at how each mate translates into degrees of freedom being tied down. For example, a Coincident mate, planar face to planar face, ties down one translation degree of freedom (in the direction perpendicular to the faces) and two rotational degrees of freedom (about directions which lie in the plane of the faces). What remains are two translational degrees of freedom in the plane of the faces and one rotational degree of freedom about an axis perpendicular to the planar faces.

A point-to-point Coincident mate ties down three translational degrees of freedom, and the part can only rotate.

An edge-to-edge Coincident mate ties down two translational and two rotational degrees of freedom. As a result, a part that you mate in this way can only slide along the mated edge and rotate around the mated edge.

Tip

When using face-to-face Coincident mates, it takes three mates to fully define a block type part. When using edge-to-edge Coincident mates, it only takes two mates.

Something to be careful about is that a degree-of-freedom analysis frequently predicts an over-defined mate scenario when SolidWorks does not in fact display any errors or warnings. For example, if one block is mated to another with the simple case of three face-to-face Coincident mates, and each Coincident mate ties down one translational and two rotational degrees of freedom, then the mating scenario ties down 9 degrees of freedom, so the part is overconstrained by three rotational degrees of freedom. However, SolidWorks has a lot of forgiveness built in, so it frequently allows situations like this, where parts are severely overconstrained. When troubleshooting any overconstrained situation, you should not take this forgiveness for granted. If SolidWorks reports an assembly as overconstrained and the reason is not intuitively obvious, try reducing some of the degrees of freedom constrained. For example, instead of making two faces coincident, consider making them simply parallel, or mate a point to a face instead of two faces.

Best Practice

This may be an overly cautious approach, but it can mean the difference between an assembly that works and one where errors are frustratingly persistent. If you are careful to approach all parts with the degree-of-freedom analysis in mind such that any newly added mate does not duplicate any of the degrees of freedom that are already tied down, you will have fewer assembly mate errors and fewer problems with assembly motion.

This means that instead of having the traditional three face-to-face Coincident mates, you would have one face-to-face Coincident mate (one translational degree of freedom, two rotational degrees of freedom), one edge-to-face Coincident mate (one translational degree of freedom, one rotational degree of freedom), and one point-to-face Coincident mate (one translational degree of freedom). This accounts for three translational and three rotational degrees of freedom without over-defining any of them.

It is true that SolidWorks internally compensates for over-defined degrees of freedom, but relying on it to do so and then tempting fate by methodically over-defining all assemblies is a risk that you do not have to take, even though it is common practice. ■

Setting up successful motion
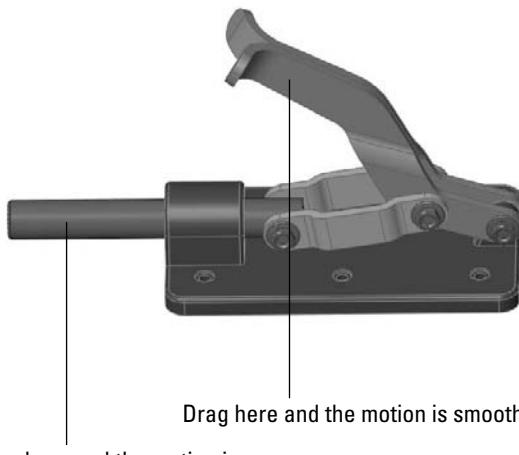
The best bet for creating motion in a SolidWorks assembly is to leave open a single degree of freedom. This means that there is only one way the part can move, back and forth, by translation or rotation. Computers in general do not respond well to ambiguity. Dragging an item that might move in several ways is more likely to cause jerky or hesitant motion.

A good example of this kind of problem with motion can be found in one of the sample assemblies that installs with SolidWorks. I have included this example on the CD-ROM for your convenience, and it is shown in Figure 13.8. The filename for the assembly is `Plunger.sldasm`.

FIGURE 13.8

An assembly displaying best bet for motion

Drag here and the motion is smooth

Drag here and the motion is poor

If you drag the assembly parts from the location shown in Figure 13.8, the performance varies. This is because when you drag the handle parts, for every position of the handle, there is only one solution for the rest of the parts. However, when dragging the plunger bar, for every position of the plunger bar, there are two possible positions for both the links and the handle (one possibility is as shown, and the other would be with the handle interfering with the base of the assembly). This kind of ambiguity causes problems in SolidWorks assemblies such as assemblies that have open degrees of freedom but will not move or only move in a jerky fashion.

Another example of difficulties related to open degrees of freedom and motion is shown in Figure 13.9. The grippers at the end of the arm move when the rest of the arm moves, but the grippers cannot be independently controlled by dragging. To fix this problem, you may want to either use the Fix/Float option (available through the RMB menu) or use configurations with mates suppressed or unsuppressed. Fix the part that you want to remain stationary closest to the part you want to move. Remember to Float the part when you are done. Be aware also that fixing a part may over-define some mates. You can open this assembly from the CD-ROM, in the filename called `Chapter 13 Robot Assembly.sldasm`.

FIGURE 13.9

A robot arm assembly with degree-of-freedom conflicts

Working with Advanced and Mechanical Mate Types

Advanced and Mechanical mate types greatly expand the number of ways that you can put parts together into assemblies. Advanced mate types include Symmetric, Width, Path Mate, Linear Coupler, Limit, and others. Mechanical mate types include Cam, Hinge, Gear, Rack and Pinion, Screw, Universal Joint, Belts /Chains. You can access Advanced and Mechanical mates by expanding the corresponding panels on the Mate PropertyManager shown in Figure 13.1.

I have not spent any time explaining the standard mates. If you understand sketch relations, the mate relations fall into place easily. One exception to this is the Lock mate. Lock is different than Fix. Fix pins a part to the background. The Lock mate locks two parts to one another, so that they always maintain the same relationship to one another, regardless of how they move with respect to other parts. This section goes into some detail on all the advanced and mechanical mates, with a brief example of each.
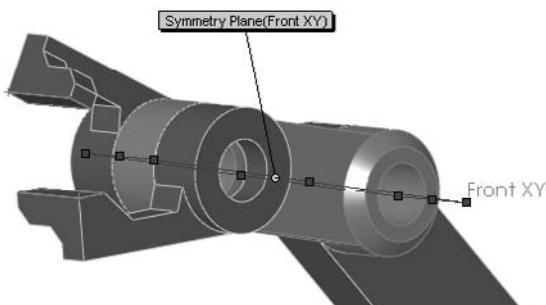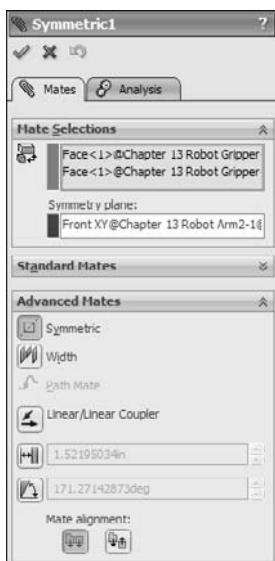
Symmetric mate

The Symmetric mate works a lot like the Symmetry relation in sketches, except that a plane is used as the plane of symmetry instead of a construction line. Figure 13.10 shows a Symmetric mate being applied to the gripper jaws. The Symmetric mate is listed in the Advanced Mates pane of the Mate PropertyManager.

FIGURE 13.10

Applying a Symmetric mate

Cam mate
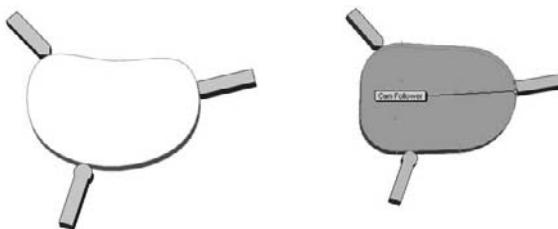
The Cam mate creates a special instance of either the Coincident or Tangent mate. Four conditions exist with the Cam mate:

- **Coincident.** The vertex on the follower mated to a cam that is created from a single closed-loop face (spline, circle, ellipse).

- **Tangent.** The cylindrical or planar face mated to a cam that is created from a single closed-loop face.

- **CamMateCoincident.** The vertex on the follower mated to a cam that is created from multiple faces. This condition enables the follower to go all the way around the cam, not stopping at the broken faces or following the extension of a single face.

- **CamMateTangent.** The cylindrical or planar face mated to a cam that is created from multiple faces. This condition enables the follower to go all the way around the cam, not stopping at the broken faces or following the extension of a single face.

Figure 13.11 shows both single-face and multi-face cams, along with the Cam Mate interface. The two assemblies are available from the CD-ROM in the file named `Chapter 13 Cam.sldasm`.

Using Cam mates

If you open the assemblies and spin the cam plate, you will notice that in both cases, the flat follower does not work very well. In fact, in the single face cam assembly, it does not work at all.

Note

Barrel (cylindrical) cams cannot use the Cam mate to create cam motion, but they do work with the Path mate. Path mates are covered in more detail later in this section. ∎
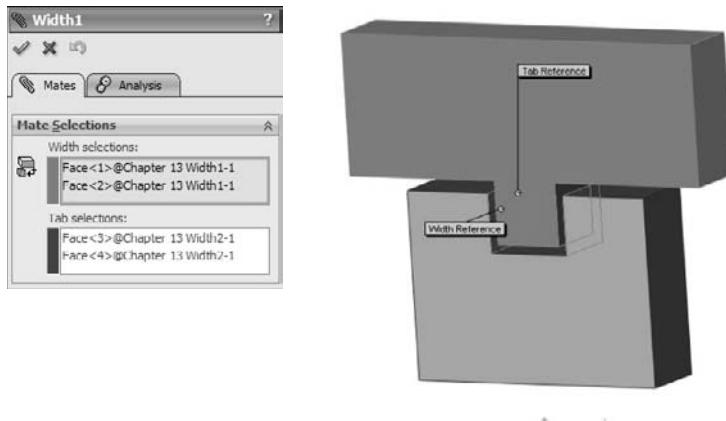
Width mate

The Width mate is often used as a replacement for the Symmetric mate in situations where parts are modeled with some tolerance and have a gap rather than touching face to face. The Width mate requires two pairs of faces to be selected, and works particularly well when a part has to be spaced evenly between two faces and there is no mid-plane; for example, when a square key is placed in a square keyway that is somewhat larger than the key. If a mid-plane is available, the Symmetric mate may be a better option, or at least a faster one to mate given the Symmetric mate only requires two faces and a plane. Figure 13.12 shows a good application for a Width mate as well as the PropertyManager interface for the mate.

FIGURE 13.12

Applying a Width mate
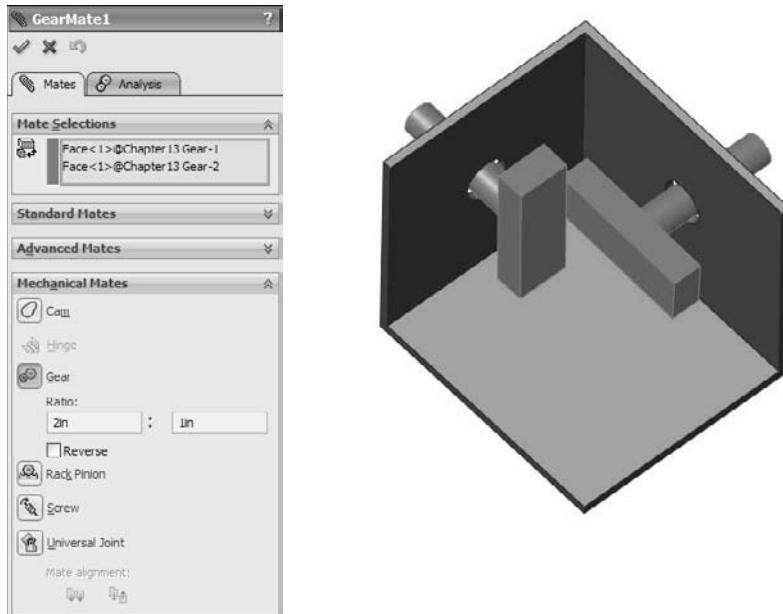
Gear mate

The Gear mate enables you to establish gear type relations between parts without making the parts physically mesh. You can also apply gear ratios and directions without physical connections, so that you can have a shaft in and a shaft out of a black-box transmission. You can open the assembly shown in Figure 13.13 from the CD-ROM. It is named `Chapter 13 Gear Mate.sldasm`. To see the effect of the mate, open the assembly and rotate the parts. Then edit the mate and change the ratio and direction. The selection for the Gear mate is just two cylindrical faces.

FIGURE 13.13

Applying a Gear mate

Rack and Pinion mate
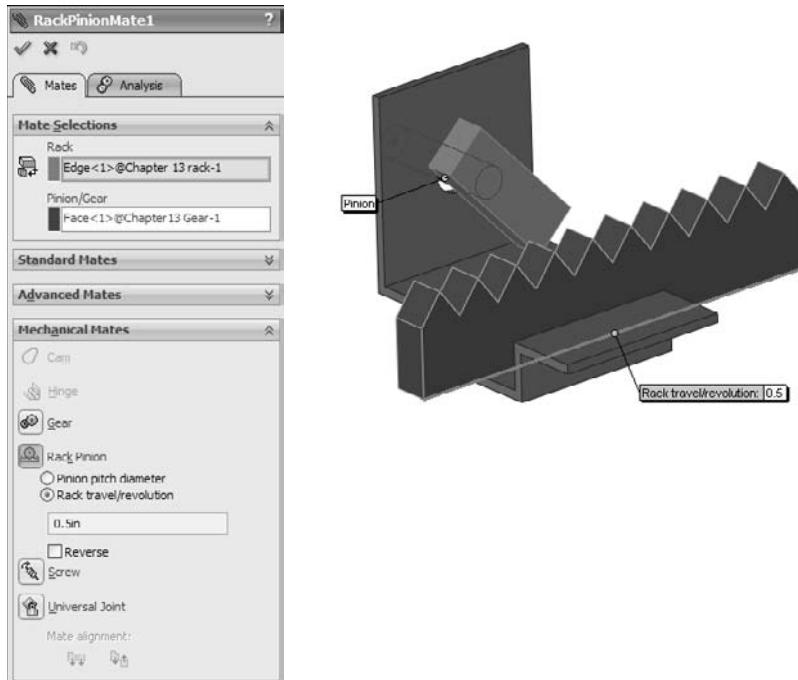
The Rack and Pinion mate takes rotational motion of one part and turns it into translational motion for a second part. Again, the parts do not need to be physically connected and can be simple representations of the actual geometry that is needed to drive the motion in the real world. Figure 13.14 shows an assembly that uses the Rack and Pinion mate. You can find this assembly on the CD-ROM with the filename Chapter 13 RackPinionMate.sldasm.

FIGURE 13.14

Applying a Rack and Pinion mate

Limit mates
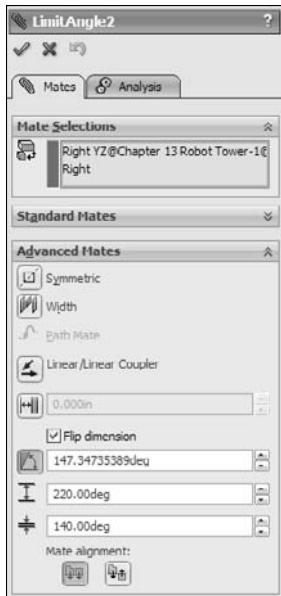
You can apply limits to distance and angle mates in order to allow the parts to move within a certain range of values. Figure 13.15 shows the PropertyManager interface for the Limit Angle mate. Limit mates accept zero and negative values that are not normally accepted for dimensions in SolidWorks. When used properly, Limit mates can be an extremely powerful tool for creating more realistic motion in assemblies.

On the CD-ROM, open the assembly named `Chapter 13 Robot Limit Mate.sldasm`. Drag the Robot Tower part. Notice that it only rotates within a limited angle. LimitAngle2 is the mate that is driving this motion.

The Limit Angle PropertyManager

Screw mate
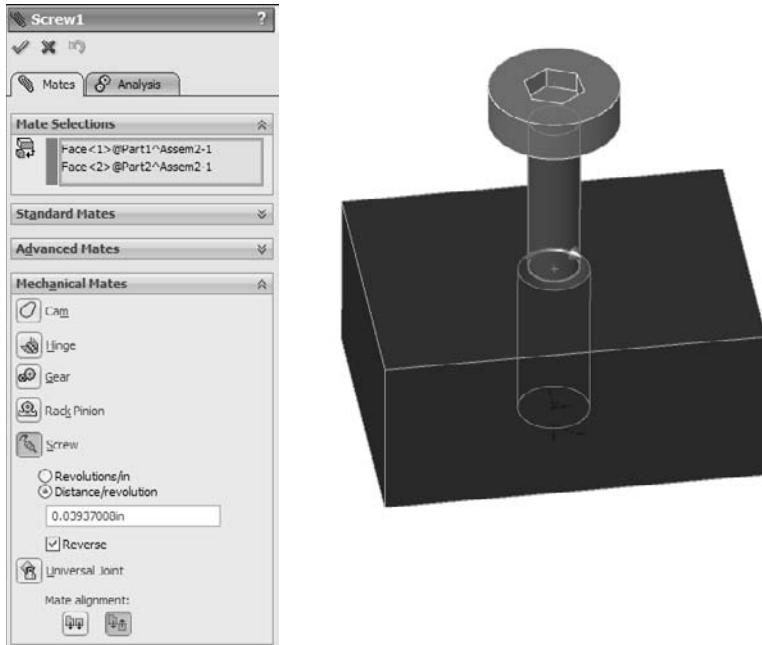
The Screw mate functions just the way the name suggests. For every revolution of a part relative to another part, the part moves in a linear direction by a specified amount. This mate requires two cylindrical faces and a pitch value, as shown in Figure 13.16.

FIGURE 13.16

Setting up a Screw mate

Screw mates can be handy for lead screw animations. I would probably not recommend them for general modeling, but for animations, they are a fantastic addition to the mate toolbox.

Path mate
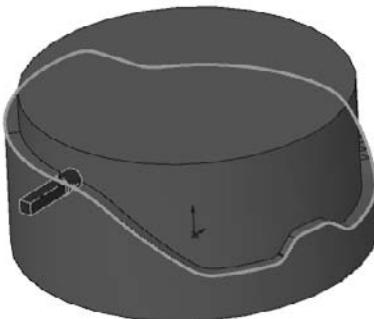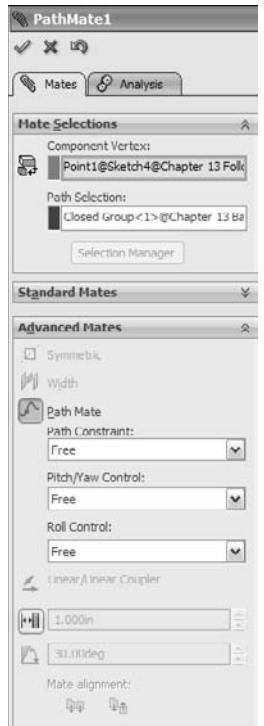
The Path mate is the one that makes complex barrel cam motion possible, as well as other types of path driven motion. Another application for this mate type beyond barrel cams is for the motion of a camera in a fly-through animation. The Path mate requires a point or vertex on one part and a curve selection in a second part. The path selection, if it is not just a single sketch or curve entity, requires the use of the SelectionManager, which enables you to select multiple end-to-end entities to form closed or open paths. Figure 13.17 shows the setup for a Path mate.

Note

On the barrel cam in Figure 13.17, notice that a sketch point is being driven along the path. In reality, this does not exactly reflect the motion of the follower around the cam surface. The Path mate does not take into account the tangent contact point between the surfaces; it simply drives the point along the curve. There is a slight amount of error in this scenario, such that the leading or trailing surface of the follower will interfere with the cam, depending on the angle of the cam surface. Also note that the Path sketch entity has nothing to do with the Path mate; it is not required to make the Path mate work. ■

FIGURE 13.17

Setting up a Path mate

Linear Coupler mate

The Linear Coupler mate relates the motion of one part in one direction to another part in either the same or a different direction. It also enables you to apply a ratio between the motions. The directions do not have to be parallel or anti-parallel; they can be at right angles, or at any angle. The mate only controls motion in one direction, so other directions are free to move.

You can use this mate to simulate symmetric motion, or geared motion without modeling the rest of the detailed mechanism. Figure 13.18 shows the setup for this mate.

Hinge mate
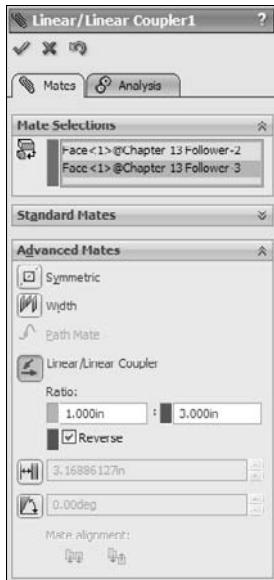
The Hinge mate is just a shortcut for making a Concentric and a Coincident mate, but it does it all in a single feature, in a single interface. Figure 13.19 shows the PropertyManager interface for the Hinge mate.

FIGURE 13.18

Setting up a Linear Coupler mate

Belt/Chain
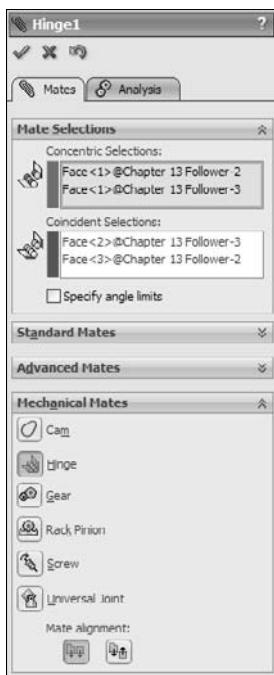
The Belt/Chain assembly feature is not technically a standalone mate type, but it uses mates to accomplish its task. You can use the Belt/Chain feature in two ways: to create relationships between sketch blocks or to create relations between parts. This feature also creates a sketch and a solid part representing the belt or chain.

Chapter 4 has a tutorial that discusses the Belt/Chain functionality as it relates to sketch blocks. The functionality using solid parts as pulleys and sprockets is very similar. You can initiate the Belt/Chain function from a toolbar button on the Assembly toolbar or through the menus by choosing Insert ➪ Assembly Feature ➪ Belt/Chain.

FIGURE 13.19

Setting up a Hinge mate

Editing and Troubleshooting

You should become proficient with editing and troubleshooting assembly mates. If you are not comfortable with repairing and modifying mates, you may find assemblies frustrating to work with. However, once you master the techniques, you will be more confident and willing to experiment with assembly changes.

Editing existing mates
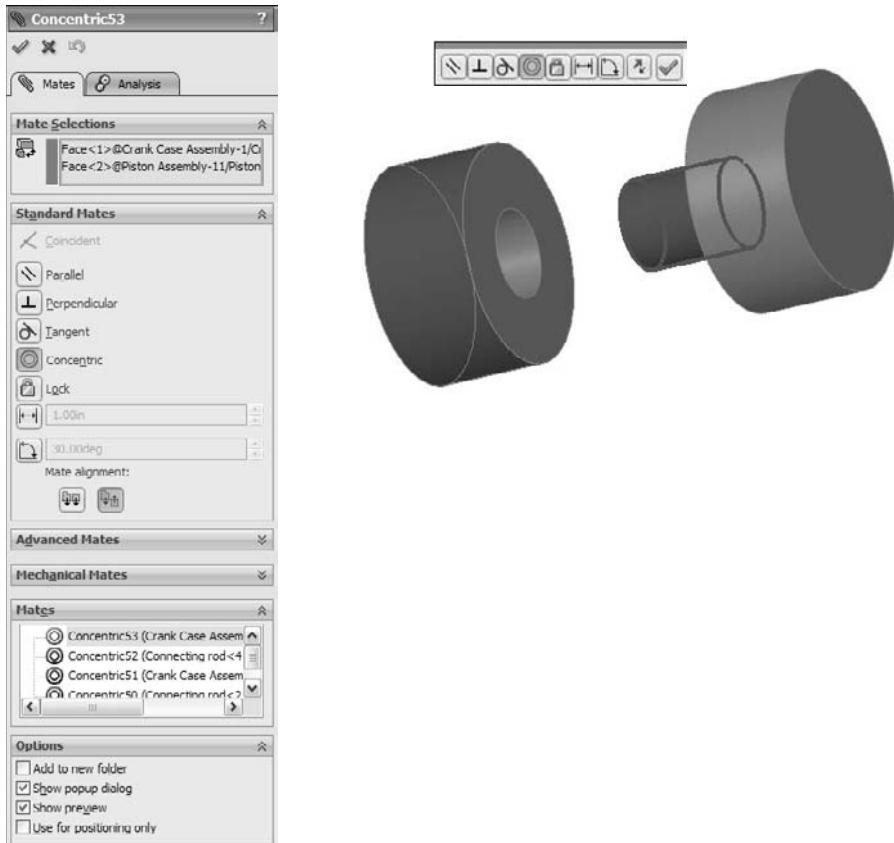
If you are editing just one mate, then you can simply right-click it and select Edit Feature (or if you are using Context Toolbars, left-click it and click the Edit Feature button). Remember that you can find mates in places other than the Mates folder at the bottom of the assembly FeatureManager; most notably, you can find them in folders under the parts that they are mating together.

You can make several types of changes to mates, including changing the selections, the mate types, and the mate alignment. These types of changes are all shown in Figure 13.20, which displays a mate being edited. The selected faces are highlighted in the graphics window.

FIGURE 13.20

Editing a mate

To edit multiple mates consecutively without exiting the Mate PropertyManager, it is best if you pre-select the mates. Pre-selected mates are shown in the Mates panel, as shown in Figure 13.20. You can switch from editing one mate to another by simply selecting the new mate in the Mates panel. If you select only one mate before clicking Edit Feature, but realize later that you want to edit multiple mates, you can select more mates through the FeatureManager.

When mate entities are lost, the mate displays as grayed out, as shown in Figure 13.21. Also shown is a mate that cannot be resolved; for example, a face coincident to two points separated in space. You can repair the missing reference problem by selecting the Invalid reference in the Mate Selections window and then selecting the correct item from the graphics window.

The yellow triangle is a warning symbol that shows that a mate is satisfied, but it is in conflict with another mate that is not satisfied.

FIGURE 13.21

Repairing mates with missing references

Troubleshooting assembly mates

It is best to troubleshoot an assembly mate problem as soon as it appears, and not after it has time to become complicated by other issues. Failed mates also cause performance problems because SolidWorks keeps trying to solve the mates that are in conflict with one another.

Assembly problems often appear to be far larger than they actually are. For example, the entire tree may light up with warnings and error symbols when one extra mate is applied. You can use several approaches to troubleshoot situations like this. In fact, I sometimes purposefully over-define mates just to locate a leftover mate or a mate that is not supposed to be there.

Two types of symbols may help you distinguish the kinds of errors that are present in different mate features. The yellow triangle that contains an exclamation point is really not an *error*; it is actually more of a *warning*. It tells you that this mate is in conflict with other mates (this symbol is used for a variety of warnings), but that the mate is still satisfied. One of the other mates with which it conflicts is probably not valid, and so this type of warning is usually accompanied by an actual error symbol where the mate is not satisfied.

The red circle containing the X is a failed mate. This is a mate that is in conflict *and* is invalid. If it is also a Coincident mate, then the two Coincident entities are not coincident.

Distinguishing between warnings and errors

You can use the following troubleshooting techniques:

- **Last in first out.** When a mate is added that causes warning and error signs to appear throughout the design tree, you can usually correct the problem by removing this last mate.

- **Single elimination.** If you are sure that the last mate added is correct, then you may want to go backward up the tree starting at the bottom, suppressing individual mates until you find one that causes the warning and error signs to disappear from the tree.

- **Single addition.** It may be easier to take the opposite approach, by suppressing all but the mates that you are sure of, and then gradually unsuppressing mates until the conflict reappears.

- **Suppress a part.** With all the mates active, try suppressing an individual part to see whether this makes a difference. If it does, then unsuppress the part and look at the mates for that part in the Mates folder under the part.
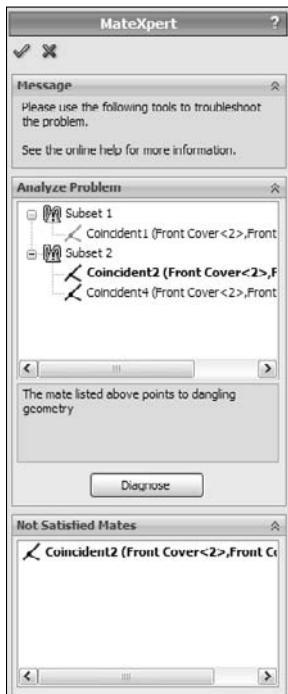
- **Mate Xpert.** The Mate Xpert is an automated routine that creates subsets of groups of conflicting mates. Each subset of mates has one mate that is not satisfied because of the conflict. This may help you to find the cause of the conflict. Figure 13.22 shows the Mate Xpert interface. You can access the Mate Xpert from the RMB menu on mates with errors.

The Mate Xpert interface

Examining Mate Options

The Options pane of the Mate PropertyManager is shown in Figure 13.23. Most of the options are self-explanatory, except for Use for positioning only. This option positions a part but does not apply a parametric mate. Some users make extensive use of this option for various applications where you need the part located precisely, but do not need or want a mate feature in the tree. Positioning parts for Animator animations where the part does not move according to a mate is one example of a use for this option.

FIGURE 13.23

The Options pane of the Mate PropertyManager

Reviewing Mate Best Practices

Sometimes best practice recommendations can contradict one another, and for each best practice recommendation that you find, there are likely several specific situations where the recommendation is invalid. As a result, you should apply the following recommendations carefully.

- Each assembly should have at least one part that is either fixed or fully mated to the standard planes of the assembly so that it cannot move relative to the assembly.

- You should use fixed parts sparingly. One part that serves as a "ground" for the assembly should be fixed. Other than that, the parts of imported assemblies are sometimes fixed to keep them from being moved accidentally.

- Do not mate to time-dependent features in the assembly tree, or to in-context features in parts. You may want to refer to Chapter 12 for a refresher on time-dependent features in the assembly tree. This can create circular references where the assembly must be rebuilt multiple times to fully resolve the positions of all parts and sketches.

- When possible, it is best to mate all parts to the "ground" part. Creating *daisy-chain* mates (where A mates to B, which mates to C, and so on) forces the mates to be solved in a particular order, which may take more time to solve than otherwise. If all the mates relate to established assembly references, the mates may be more stable. Chapter 11 describes using a skeleton in a part to make sketch and feature relations. A similar concept can be applied in an assembly, by mating parts to an assembly sketch.

- When possible, leave part positions fully defined, especially when other geometry is dependent upon the position of parts. Some examples include in-context features, assembly features, or assembly-level reference geometry, which are dependent on part geometry.

- Constraining the rotational degree of freedom for components such as screws, washers, and nuts is usually considered excessive. At times, too many open degrees of freedom may cause problems with complex motion, such as a gripper on the end of a robotic arm. SolidWorks functions well when there is a single, well-defined path between two points, but when there are multiple options, the software may become confused.

- Do not leave errors unresolved in the tree.

- Remember to use subassemblies to break up the number of mates that are solved in the top-level assembly.

- Limit the use of flexible subassemblies.

- Do not mate to entities that may be removed later by suppressing or unsuppressing features, especially edges or faces that are created by features such as fillets. For this reason, it is usually best to wait until parts are complete before you use them to create an assembly, although this is rarely practical.

- Use a degree-of-freedom analysis to prevent mates from becoming over-defined.

Tutorial: Mating for Success
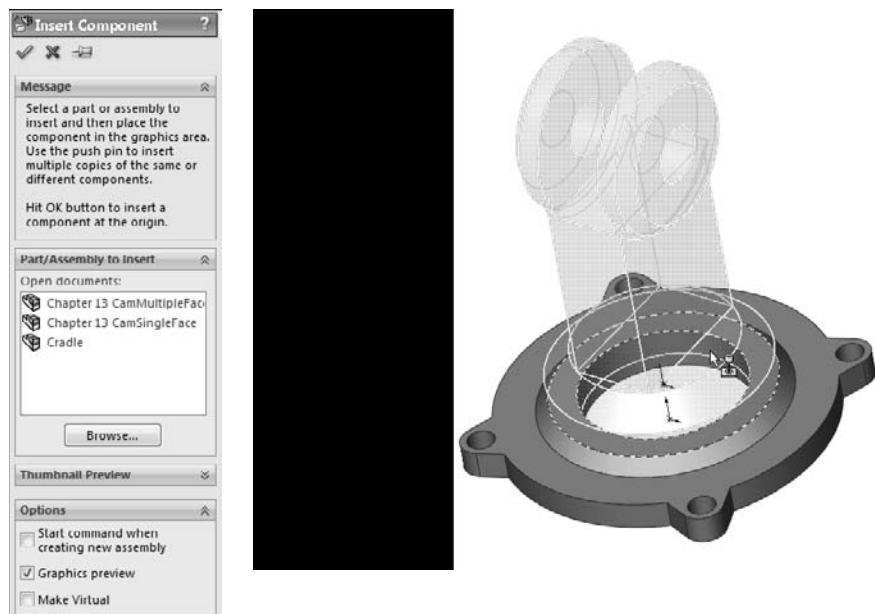
In this tutorial, you will put together a model of a robotic arm to better understand some of the mate issues discussed in this chapter. Follow these steps to mate for success:

1. **Open the part named** `Chapter 13 Robot Base.sldprt` **from the CD-ROM.**

2. **In the part document window, click the Make Assembly From Part icon, and click the cursor on the Origin of the assembly to place the part Origin at the assembly Origin.** The part is automatically fixed in place.

3. **Choose Insert ➪ Component ➪ Existing Part/Assembly.** Click the Browse button in the PropertyManager and find the part called `Chapter 13 Robot Tower.sldprt`. This part contains a Mate reference to help you mount it to the base. If you bring the cursor near the big circular hole in the base, you can see the preview of the tower snap into place. Click to accept this placement. Figure 13.24 shows this placement in progress. Notice that the cursor appears as a SmartMate cursor for the peg-in-hole mate. When the part is dropped, check the mate list to confirm that a Concentric and a Coincident mate have been applied by the Mate reference.

4. **Open the part with the filename** `Chapter 13 Arm.sldprt`, **the Default configuration, in its own window, and choose Window ➪ Tile Vertically.** The part and the assembly should be open in adjacent windows.

5. **Click the face inside the hole without the chamfer around it in the Arm part, as shown in Figure 13.25.** Then drag it into the assembly to the cylindrical face inside the hole at the top of the Robot Tower part. The concentric SmartMate symbol should appear on the cursor.

FIGURE 13.24

A Mate reference being used to SmartMate a component
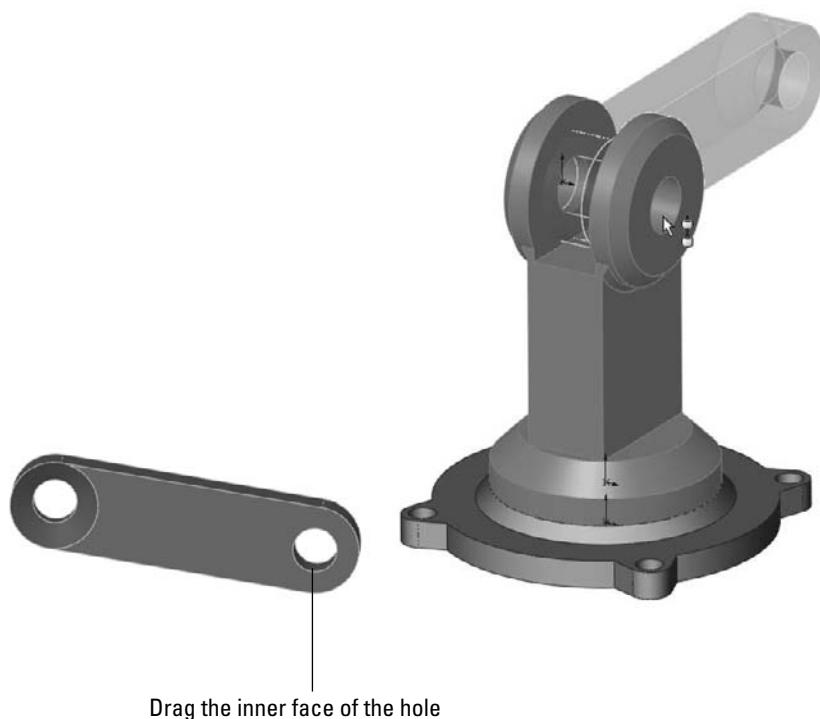
6. **Click the green check mark icon to accept the Concentric mate. Move the part to test that the mates are correct.**

7. **Click the Mate tool on the Assemblies toolbar.** Expand the Advanced Mates panel and click the Width mate.

8. **In the Width Selections box, select the two inner faces of the Robot Tower part, and in the Tab selections box, select the outer faces of the Arm part.** The selection should look as shown in Figure 13.26.

FIGURE 13.25
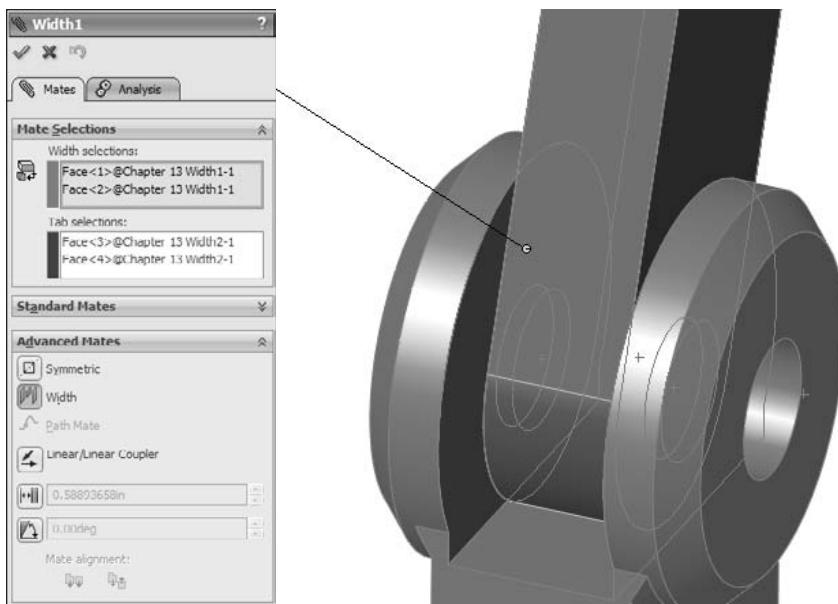
Displaying a SmartMate when dragging between windows

Drag the inner face of the hole

9. **Open a Windows Explorer window, and select the following parts: Chapter 13 Robot Arm2 and Chapter 13 Robot Gripper.** Drag these parts into the SolidWorks assembly window and drop them in a blank space.

10. **Select the chamfered faces of the Arm and Arm2 parts and create a Coincident mate between them.** You can make Coincident mates between conical faces as long as the cones are the same angle. This special case acts like a combination of Concentric and Coincident mates. Figure 13.27 shows the selections and the results.

11. **Create a copy of the gripper part so that there are two instances of it in the assembly.** You can do this by Ctrl+dragging the part within the assembly window with the Mate PropertyManager closed.

FIGURE 13.26

Creating a Width mate
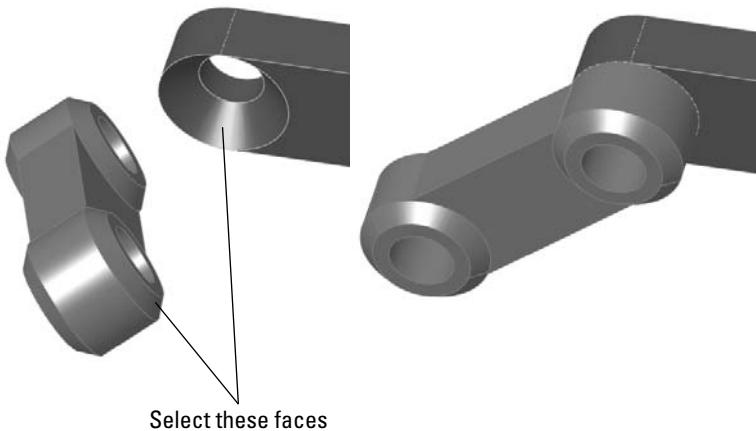
12. **Mate both of the grippers to the Arm2 end using the same mating technique that you used for the previous conical face Coincident part.**

13. **Once you have applied these parts, try moving the various joints of the assembly.** Notice that it is difficult, if not impossible, to isolate the motion of just a single part. This is because there are too many open degrees of freedom, and a lot of ambiguity.

14. **Fix Arm2 to allow you to move the gripper parts as you want.** Create a Symmetric mate between the indicated faces of the grippers and the Front plane of the Arm2 part, as shown in Figure 13.28.

15. **Practice making angle mates, suppressing mates, and fixing parts to limit motion.**

16. **Save the assembly and exit the file.**
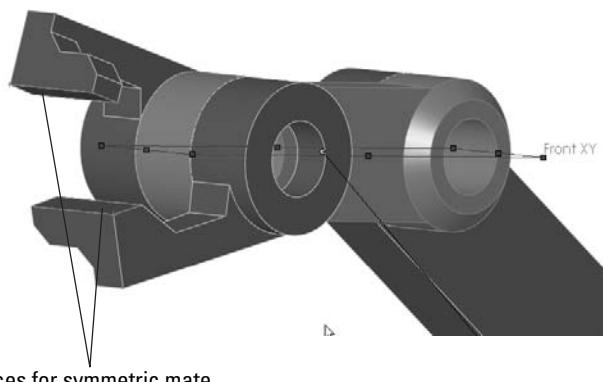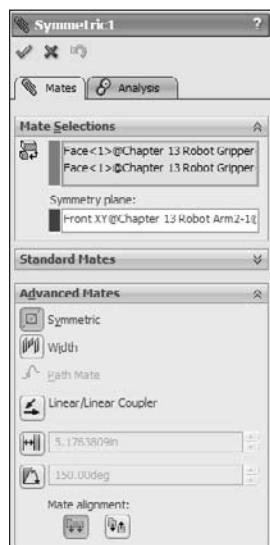
FIGURE 13.27

Making conical faces coincident

Select these faces

FIGURE 13.28

Creating a Symmetric mate

Faces for symmetric mate

Summary

A thorough understanding of mates, and editing and troubleshooting techniques in particular, makes the difference between a real assembly artist and a user who struggles through or avoids certain tasks. There is a lot about mates that is not straightforward, but with practice, you can understand and master them. You can put assemblies together quickly, with a focus on rebuild performance and Dynamic Assembly Motion.

Although best practice concepts should not dominate your designs, they are great guidelines to start from. Watch out for the pitfalls outlined in the section in this chapter that summarizes mate best practices to follow to avoid making big mistakes.

Controlling Assembly Configurations and Display States

Assembly configurations enable you to control many things, including part configurations, suppression, visibility, color, and assembly feature sizes. They also enable you to control assembly layout sketch dimensions, mate values, suppression states, and several other items. What you learn in this chapter about assembly configurations builds on the information in Chapter 10, which discusses part configurations. In this chapter, you also learn how you can use design tables in conjunction with SolidWorks assemblies.

Display States are a better performance alternative than configurations for controlling visibility and displaying part styles in assemblies. I discuss Display State options at length in this chapter.

Using Display States

Display States enable you to change visual properties more quickly than configurations. Configurations save a lot of extra data if all you need is to hide or show parts, and they can be slow to change from one to another, but you can change between Display States almost instantaneously.
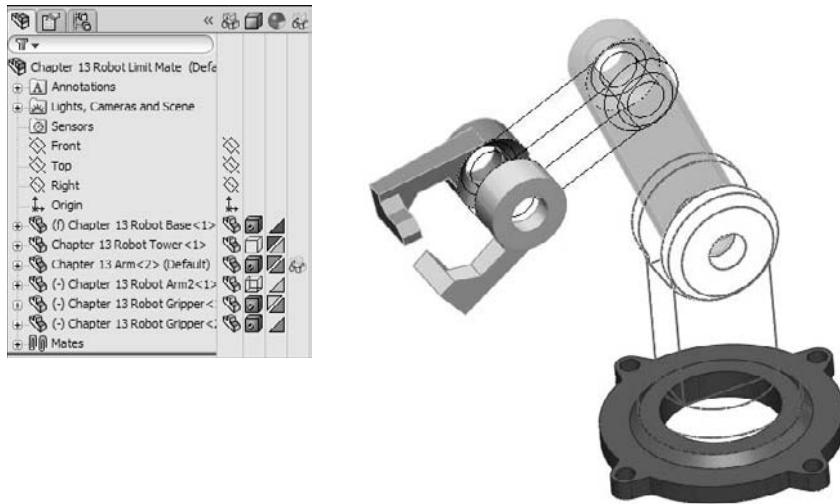
Assembly display states can also control part display states, and different instances of a part in an assembly can use different display states.

Controlling Display States and configurations

Display States can be either independent of configurations or linked to them, depending on your settings. To control the display, you can use the Display pane that flies out from the FeatureManager when you click the double-arrow icon in the upper-right corner of the FeatureManager. This is shown in Figure 2.1 in Chapter 2. Figure 14.1 shows the Display pane in action, along with an assembly showing parts in different Display States.

The Display pane and an assembly with parts in different Display States

The column symbols for the Display pane are as follows:

- Hide or Show state of the part
- Display Mode options for each component:
- Appearances
- Transparency
- Default Display
- Component/Part Color (see Note)

Note

The difference between a component and a part in SolidWorks assemblies is that a component is a generic way of identifying any top-level item in an assembly, and may be a single part or a subassembly. It always refers to a specific instance of the part within the assembly. In the case shown in Figure 14.1, the gripper jaw part is used twice; this creates two instances of the gripper jaw. One instance has its component color set to a custom color, and the other instance uses the part color. (The component color is also referred to as an override of the part color.) The part color is what you see when you open the part in its own window. The component color is only set in the assembly, and you can only see it in that particular assembly; it never affects how the part displays in any other assembly that the part is shown in.

When there is a difference between the part and component display properties (when an override exists), the component property appears as the upper-left triangle, in the color column of the Display Pane, and the part property appears as the lower-right triangle. You can only see these triangles in the Appearance column of the Display Pane. ■

Appearance overrides are discussed in Chapter 3, but I will briefly summarize it here, showing the lowest priority at the top:

- Part
- Body
- Feature
- Face
- Component

If you override the appearance or display mode for a component in a subassembly, and the upper-left triangle appears in the Display pane, you can remove the override through either the left mouse button (LMB) or right mouse button (RMB) menu. Figure 14.2 shows the LMB menu from a component of a subassembly with overrides.

FIGURE 14.2

You can remove overrides in the Assembly Display pane.
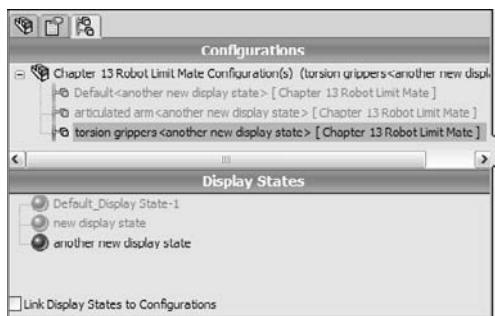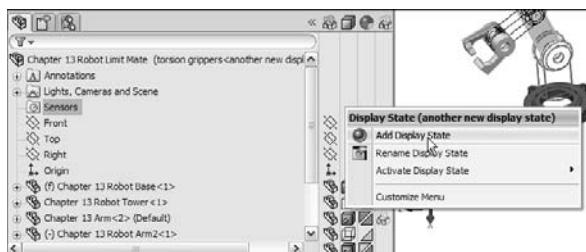
When you select Clear Override, SolidWorks clears any overrides for the currently selected subassembly component. Clear All Top Level Overrides clears all overrides in all subassemblies in the entire top-level assembly. There is no intermediate option to clear all top-level overrides for a particular subassembly; if you want to distinguish between overrides at that level, you need to clear several individual overrides. The options to remove overrides do not affect top-level components.

The active Display State appears in angle brackets after the configuration name and the filename at the top of the FeatureManager, as shown in the image on the left in Figure 14.3. Display States are created and managed in the ConfigurationManager, in a panel at the bottom of the ConfigurationManager, as shown in the image on the right in Figure 14.3. To create a new Display State, simply right-click in the Display Pane and choose Add Display State. It seems a little counterintuitive that in the place where you create Display States, you cannot see the list of Display States.

FIGURE 14.3

Display States shown in the FeatureManager and the ConfigurationManager

Performance

Display States offer a huge performance gain over configurations when used to control display of parts. The reason for this is that SolidWorks saves some model information for each configuration. When a configuration is created only for the purpose of hiding or coloring a part, this takes up a lot of additional file space and CPU (central processing unit) time. Display States change much faster than configurations — almost instantaneously — and they add very little space when you use them. ∎

Using Display States with drawings

Display States can be shown on drawings. If you only show or hide parts in Display States, you will escape most of the problems. For Display States that change the display mode (wireframe, shaded, and so on) to work properly, you have to set the view itself to Shaded, then select the display state from the PropertyManager for the view. The big catch here is that you have to change the Display State of the parent view; changing the Display State of a projected view does nothing, even though the controls are available. Projected views cannot have a Display State that is independent from the parent view. Drawing views are discussed in depth in Chapter 21.

Using part Display States in assemblies

Starting in SolidWorks 2010, parts can have Display States, including separate bodies within parts. You can control the part Display State for specific instances of a part within an assembly in the Component Properties dialog. Using part Display States has the same advantages that using assembly Display States has (mainly display speed), especially when compared against configurations.

Understanding Assembly Configurations

Assembly configurations are used for many different purposes, including assembly performance, simplified assemblies, variations of assemblies, assemblies in different positions or states, and many others. Like part configurations, assembly configs also have a few best practice type suggestions. Configuration settings for assemblies control how the assembly appears in a Bill of Materials (BOM), and what happens to parts, features, or mates that are added to other configurations, and so on. All these are discussed in this section.

Configurations for performance

One of the best tools to make large assemblies easier to work with is assembly configurations. You can use several techniques to improve the speed of working with assemblies. Although this information is presented in a list of techniques, it is important to select a method that fits the situation. Each method has strengths and weaknesses.

Suppressing components and features

The most obvious use of configurations for improving assembly speed is to have a configuration or several configurations with suppressed components. Suppressed components are not loaded or displayed, so memory and video power are conserved.

Tip

Remember that you can use a folder for parts and suppress the folder. If you are just using configurations to hide parts, consider using Display States, given they are more efficient for that purpose. Also remember that SpeedPak, discussed in Chapter 12, is a subset of configurations. SpeedPak is a simplified representation, enabling you to select faces and bodies to represent the entire subassembly for performance reasons. ∎

Schemes that you may want to use for suppressing parts need to have configurations that isolate functional areas of an assembly, configs that remove the fasteners or purchased components, configs that remove complex parts, configs that only leave the parts used in in-context relations, configs that suppress patterns and assembly features, assembly configs that use simplified part configs, configs that show the assembly in different positions, or variations of the assembly using different part configurations.

If you suppress the "ground" part or any part that connects groups of parts, keep in mind that this can cause other parts to float in space unattached. Obviously this is not a good situation, and you should avoid it if possible. One way to avoid it is to use an assembly layout sketch and mate the parts to the sketch instead of to the ground part.

Aside from components, other items can also be suppressed to improve performance, such as assembly features and component patterns. Do you really need to see all those parts patterned around the assembly to work on it in a simplified representation? You may be able to suppress the parts. If you feel that you cannot suppress parts, then consider at least using Display States to hide parts that are needed to complete the parametrics but do not need to display.

Performance

The biggest killer of assembly speed is the circular reference. You can make circular references in a couple of different ways, but they are usually the result of mixing history-based functions (mates, in-context sketch relations, feature references) with non-history-based functions (parts shown in the Assembly FeatureManager). This enables you to create partial or complete loops of references, where A references B, which references A. These are a particular problem with in-context references, which are discussed in more depth in Chapter 16. ■

Configuring SpeedPaks

SpeedPaks are described in more detail in Chapter 12. A SpeedPak is a configuration that uses only selected faces and bodies to represent an entire subassembly, instead of opening all the parts in the assembly. In fact, a SpeedPak stores the geometry in the assembly file so it doesn't have to open any part files at all.

Part of the reason I mention SpeedPak in the configuration chapter is that it is a form of configuration, essentially a derived configuration; another reason is that SpeedPaks are configurable. So you can have top-level assembly configurations that call on subassemblies to use their SpeedPaks. That can be of significant help with very large assembly performance.

Using part configurations for speed

I have discussed simplified part configurations in Chapter 10, and they can consist of configs with cosmetic features such as small fillets and extruded text, or other cosmetic details that are suppressed. Assembly configurations can use different part configurations, which, for example, would enable you to make an assembly config called "Simplified," and in it reference all the Simplified part configurations.
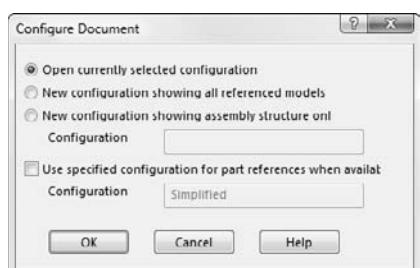
Tip

When opening an assembly through the Open dialog box, the Advanced option enables you to open the assembly and create a new assembly configuration that uses part configurations of a given name, if available. The default part configuration name entered in the text box is, I think, suggestive of how SolidWorks intended for this function to be used. As shown in Figure 14.4, it is "Simplified." In previous versions, the Advanced button was conspicuously placed on the front of the Open dialog box, but by 2009 it was changed to a selection cleverly hidden in the list of configurations, as shown in the image of the configuration drop-down list. ∎

FIGURE 14.4

The Advanced option for assemblies in the Open dialog box
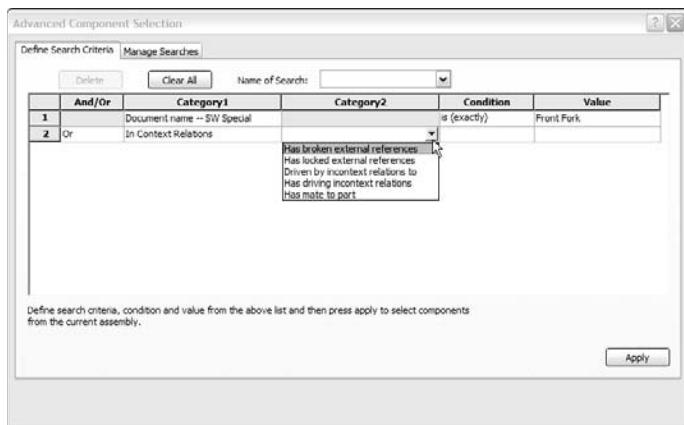
Other special operations for assembly configurations in the Open dialog box include creating a new configuration that has all the components suppressed. This enables you to see the structure of the assembly without fully resolving all the components. Another option is to open the assembly with a new configuration, where all the components are resolved. Beyond that, the Open dialog box also enables you to select a specific configuration to open to so that you do not have to wait for the last saved config to load and then make the change.

Advanced Component Selection

The Advanced Component Selection dialog box was formerly called Advanced Show/Hide Components. It is shown in Figure 14.5. You can access this dialog box by right-clicking the configuration name in the ConfigurationManager and selecting Advance Select.

FIGURE 14.5

The Advanced Component Selection dialog box
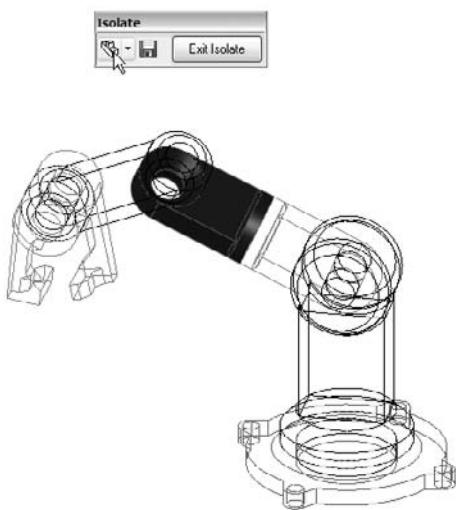
This tool enables you to establish search criteria and show or hide parts based on the criteria. Multiple criteria can be used, stored, and retrieved. This tool is generally underused, and in my experience, users are always surprised to find it in the software. It has been there since about 1998, having undergone a face lift in the last two releases. The Category 1 options enable you to search on things like document name, in-context status, part mass, and several other standard SolidWorks info. Category 2 can be either custom property info or structured options for Category 1 such as specific in-context conditions.

Isolate

Isolate works like the inverse of the Show command. If you select multiple parts and click Isolate from the RMB menu, the selected parts remain shown, and everything else becomes hidden. A little popup menu gives you the option of showing the removed components in a Wireframe or Transparent display mode, or of saving the current display as a new Display State. This is a very useful function, as shown in Figure 14.6.

The Isolate function can create display states
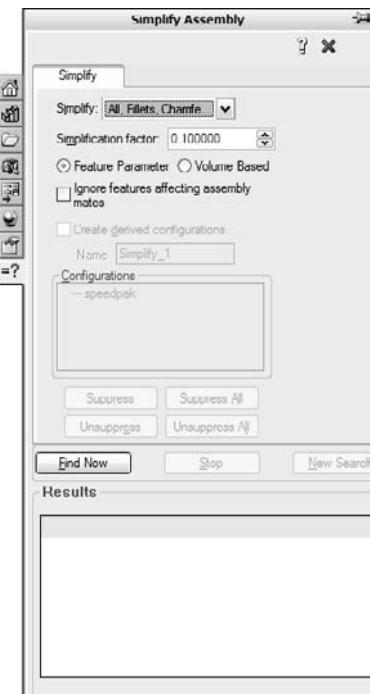
SolidWorks Utilities Simplify Assembly

If you have the SolidWorks Office bundle or higher, then you can activate the Utilities add-in. You can do this by choosing Tools ⇨ Add-ins, and then selecting Utilities. This displays a Utilities menu with the Simplify option. The Simplify Assembly tool is shown in action in Figure 14.7. This tool shows up in the Task pane (right side of the screen).

FIGURE 14.7

The Simplify Assembly tool

The Simplify Assembly tool can help you find features in the parts of the assemblies that are under a certain size or that take up less than a certain percentage of the volume of the part. You can then suppress these features in special derived configurations.

Controlling display performance

Overall, SolidWorks performance is split into two categories: CPU (central processing unit) processing and GPU (graphics processing unit) processing. Which of these functions your computer performs better can vary widely, depending on your hardware, drivers, and system maintenance, among other factors.

When trying to speed up the performance of an assembly, the biggest impact is obviously made if you can reduce the load on both the CPU and the GPU. You can do this by suppressing a part. When a part is suppressed, it is neither calculated nor displayed; this means the load on each processor for that part is zero.

When you hide a part, its parametric features are still calculated by the CPU; however, because the part is hidden, it creates no load on the GPU. If you have a good main processor and a questionable video card, then you will achieve a greater benefit from removing graphics load from your display.

Lightweight parts

On the other hand, if you want to still show a part but not calculate any of its parametric relations, you should use Lightweight parts. You can find Lightweight default settings by choosing Tools ➪ Options on both the Assemblies and Performance pages. You can make parts lightweight through the RMB menu. The opposite of lightweight is resolved. *Resolved* means that the part is fully loaded, its parametrics are loaded and calculated by the CPU, and its graphics display data is calculated and shown by the GPU.
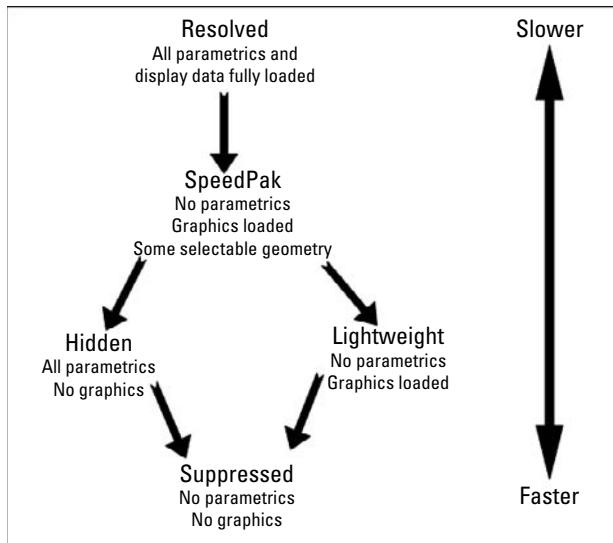
SpeedPak

There is some confusion about where SpeedPak falls into this scheme of things. With SpeedPak, the parametrics are not loaded, but the graphics are. Also, some of the geometry is selectable, as if it were imported geometry (actual geometry but without rebuildable parametrics). But SpeedPak applies only to subassemblies, where the need for improvement is a lot higher.

To summarize this section, there is a five-way relationship between the Resolved, Lightweight, Hidden, and Suppressed states, as shown in Figure 14.8.

FIGURE 14.8

The relationship between the Resolved, SpeedPak, Lightweight, Hidden, and Suppressed states

Resolved versus unsuppressed

The terminology becomes a little convoluted here because of the relationship between the five different states. In parts, the feature states are easy to remember because features can be either suppressed or unsuppressed. However, in assemblies, there are five states instead of two, and so *unsuppressed* could mean anything that is not suppressed, which still leaves three states. For this reason, *resolved* is used instead of *unsuppressed* when dealing with components in an assembly.

Configurations for positions

When you use configurations to display an assembly in various positions, you can do it a couple of ways: by changing mates or by changing a layout sketch. Mates are configurable in two ways: mates can be suppressed and unsuppressed, and angle and distance mate values are configurable in the same way that sketch dimensions are configurable. Although creating a mate scheme that enables you to reposition the assembly using mate suppression states and values is essential to this method, it may not be the best approach.

Using a skeleton or layout sketch to mate parts may be a better approach, although this also has its drawbacks. If you mate to a layout sketch, you cannot make use of Dynamic Assembly Motion. If you use the mate scheme discussed previously, this generally means having a fully defined assembly, and this also does not allow for Dynamic Assembly Motion.

As a compromise, a good way to handle this is by using one configuration for Dynamic Assembly Motion, with one or more open degrees of freedom. You can use other configurations to fully define the mechanism and show it in particular positions using either method. Probably the best way to demonstrate this is with an example using the robot arm assembly.

Positioning with mates

First, take a look at positioning with mates. On an assembly such as this one, the goal is to position the grippers. You can do this a couple of ways, both directly and indirectly. In the assembly used for this chapter, the grippers have been rebuilt as a subassembly, which allows different types of control. Notice that the subassembly has a configuration for the closed position and one that allows Dynamic Assembly Motion. Also, the subassembly is being solved as Flexible. Figure 14.9 shows the assembly and the FeatureManager.

Driving the position directly
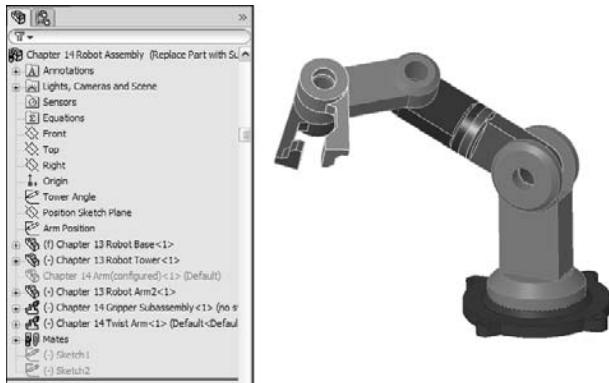
A sketch point has been added to the subassembly to precisely identify the point on the gripper that is to be positioned. Sketch points have also been added to the main assembly to represent parts that need to be picked up by the robotic arm.

Check the derived configurations under the default config. Notice that when you switch between certain configurations, the parts seem to separate. Moving one of the links causes the parts to snap back together again. This is probably because there are so many options when moving between configurations that the software has difficulty choosing a final position. This is definitely one of the potential problems when using configured mates to show an assembly in various positions.

FIGURE 14.9

The assembly used for this example

Notice also that although the grippers are positioned correctly, the arm is still allowed to swivel around the intended target point. You can correct this by defining an orientation for the grippers for each location. If an additional pivot were added to the assembly, then fully defining the parts would become more difficult. The arm would not be able to reach any additional points, but it would not be so limited in orienting the grippers at each point.

Driving the position indirectly

You can also use mates to drive configured positions of the assembly using a series of angle mates. This makes it more difficult because to get to a particular location, you have to do some calculations, but the angle mates are to be more stable than it would be to simply reply on moving the parts to unconstrained positions.

If you cycle through the derived configurations under the Indirect top-level configuration, notice that mates are not suppressed and unsuppressed; rather the values are changed. This makes it somewhat more difficult to precisely position the grippers, but because it is specific about the positions of the individual parts, there is no ambiguity.

Positioning with sketches
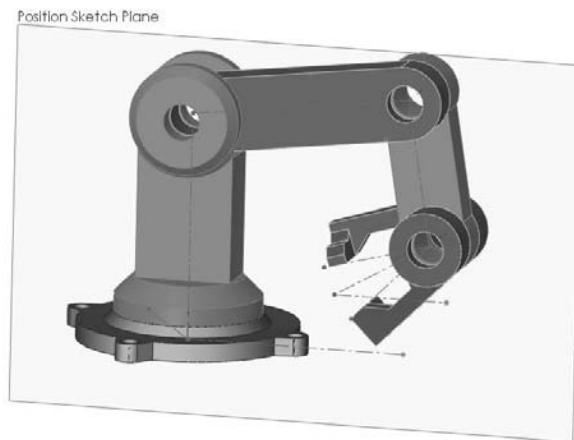
Although this technique still uses mates to position the parts and to change the position, you change sketch dimensions rather than mate values. Sketches used to drive parts from an assembly are sometimes called *layout sketches* or *skeletons*. I also discuss them in Chapter 16 for in-context or top-down assembly techniques and Chapter 11 as a way of controlling its parent-child relationships. Figure 14.10 shows the same assembly that is used for the rest of this chapter.

Positioning assembly components with sketches

This particular assembly is driven by two sketches on different planes to govern the position of the parts. Keep in mind that this assembly has been used for all the other techniques as well; this means all these techniques can exist together simultaneously, and are controlled by configurations.

Examine the assembly to see how the parts are mated to the sketches. This is important. The first time you create a part such as this, you may be tempted to mate part planes to the sketch lines.

Caution

Mating planes to sketch lines has a very serious drawback that you must be aware of. Unlike other types of mates, which have an alignment that you can control, plane-to-sketch line mates cannot be aligned. This means that the software is as likely to align elements correctly as incorrectly on any plane-to-line mate. ■

Best Practice

A better way to mate part planes to sketch lines is to mate the Temporary Axes through the joints with the sketch endpoints. This solves the alignment problem. ■

Configurations for product variations

In this case, *product variations* means variations in size or part replacement. Some examples are a 4-foot cabinet and an 8-foot cabinet, or a two-button mouse and a three-button mouse.

As a simple example, Figure 14.11 shows the familiar robotic arm assembly, but with a variation: one of the arms has been replaced with a subassembly. The subassembly is made of the original replaced part using configurations, and there are configurations of the subassembly, which is again being used as a flexible subassembly.

A part that is replaced by a subassembly

Through the course of this chapter, the robot arm assembly has greatly increased in complexity, but it has retained the original information that was in the first version. Maintaining valid assembly data through manually managed configurations is difficult, and all it takes is a simple mistake to wipe out a lot of assembly configuration data. Appropriately, the next section discusses assembly design tables.

Design tables for assembly configurations

Chapter 10 dealt with part configurations and created a good framework for design tables in general. This chapter augments that information with what you need to know to use design tables effectively in assemblies.

Assembly design tables can do everything that part design tables can do, except for selecting configurations of base parts and split parts, which are not valid assembly functions. Assembly design tables can also do some things that a part design table cannot. These include:

- Suppressing the state of a part (R for Resolved or S for Suppressed)
- Assigning the component configuration for the assembly config
- Enabling you to activate the Never Expand in BOM option

If you have been using design tables for a while and are familiar with older versions, then you may have noticed that the $show parameter, which specified whether the part was shown or hidden, has become obsolete due to the new functions of Display States.

Figure 14.12 shows the design table that results from auto-creation using the robot arm assembly. Some of the columns have been hidden to make it small enough to fit on the page. If you want to see the entire table, you need to open the assembly. If you edit the design table, you will probably want to use the Open in Separate Window option, which is easier to navigate and control.

FIGURE 14.12

An auto-created design table from the robot arm assembly

Modify Configurations and Configuration Publisher

Modify Configurations and Configuration Publisher, two configuration tools, are covered in Chapter 10 in sections on part configurations. Modify Configuration is a dialog box you can use to configure dimensions, features, and custom properties in parts, and can also use in assemblies. It is sometimes used in the place of design tables when Excel's advanced functionality is not needed.

The Configuration Publisher is demonstrated in Chapter 10 on parts: it is used to create a PropertyManager that pops up when the part is placed into an assembly. The PropertyManager enables you to select a configuration to go into the assembly. Configuration Publisher is also available in assemblies; this means you can use a popup PropertyManager when assemblies are placed into other assemblies as well. Library subassemblies may be less common than library parts, where the popup PropertyManager would be used to best effect.

Both tools are useful for managing assembly configurations. Design tables offer the most complete control, but also have some drawbacks associated with being tied to Microsoft Excel. Both Modify Configurations and the Configuration Publisher are internal to SolidWorks, regardless of the level of the software you have purchased, so if you use one of them, you can be sure that whomever you send data to will be able to access all the functionality

Assembly configuration dos and don'ts

Assembly configurations have some potential pitfalls that you can avoid if you pay attention to some of these dos and don'ts.

- Avoid using Delete as an editing option when working with configurations. Delete is forever and removes all configs.

- Avoid the use of in-context relations to size parts when you are also using configurations to size parts. A non-configured part driven by a configured part only causes confusion.

- Avoid using configurations to represent document control type revisions. I have seen people attempt to do this, but in the end, it limits the kinds of edits you can make to your parts and assemblies, and it is far too easy to make a mistake that wipes out all your diligence. In the end, this is not a viable technique.

- If you are working with manually created configs, then you should create a new configuration and activate it before making the changes. Otherwise, you will end up trying to set the original config back to the way it was.

- Remember to select the This Configuration Only option for changed dimensions instead of leaving it at the default All Configurations setting.

Creating Exploded Views

Exploded views enable you to display an assembly taken apart so that you can see all the parts. They are great for assembly documentation, assembly instructions, and for visualizing assemblies with concealed internal parts.

I have included exploded views in the assembly configurations chapter because, like Display States, exploded views are found in the ConfigurationManager under the configuration. Each config can have only a single exploded view with multiple steps, and you can copy exploded views between configurations.

When you are creating the exploded view of the top-level assembly, and a subassembly already has one, you can include the subassembly's exploded view in the top-level exploded view. While you are creating exploded views, mates are temporarily suspended.
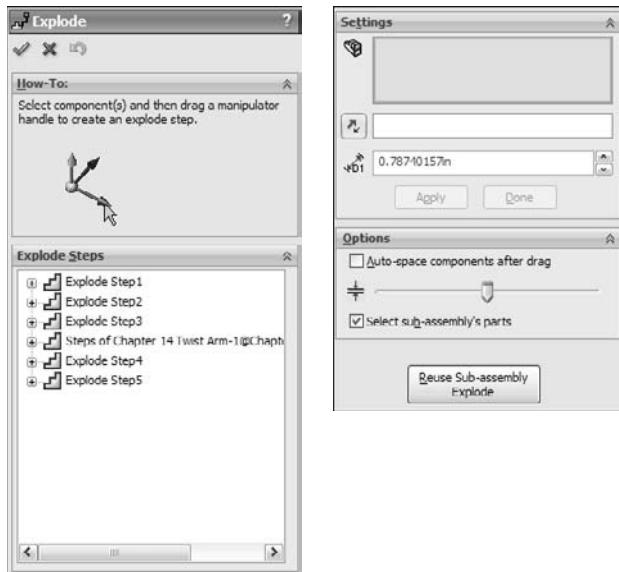
To initiate a new exploded view, switch to the ConfigurationManager, right- click a configuration name, and select New Exploded View, as shown in Figure 14.13.

Figure 14.13 also shows the Explode PropertyManager interface. This interface includes a helpful How-To section at the top to give you a hint of where to start. You can initiate Exploded View from an assembly toolbar button or through the RMB menu on a configuration.

If you are creating assembly instructions or an animation from the exploded view (using SolidWorks Motion or the RMB options, Animate Explode and Animate Collapse), then you may need to be more careful about *how* the parts are exploded. You can create explode lines that show how the parts go back together.

FIGURE 14.13
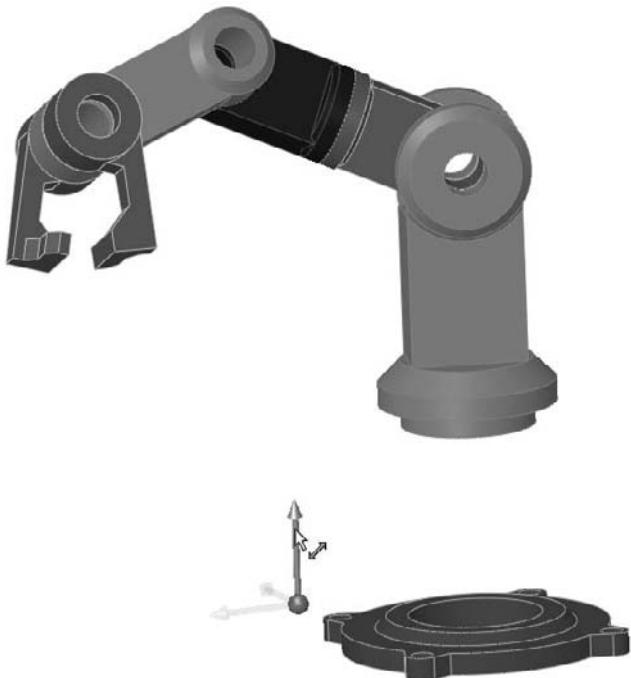
Initiating a new exploded view

To begin, you can explode the Base and the Tower down and back, respectively. A single part can explode in multiple directions, or multiple parts can explode in a single direction. These two parts are shown exploded in Figure 14.14. Select the base, and then drag the arrow of the Triad that moves in the direction that you want the part to move.

FIGURE 14.14

Exploding the base

The Tower part is a little more difficult because it is not lined up with the direction in which it needs to be exploded. To remedy this, highlight the Direction box in the PropertyManager and select a face that is normal to the direction that you want to drag, or an edge that is in this direction. Then drag the appropriate arrow on the triad again, as shown in Figure 14.15.

Tip
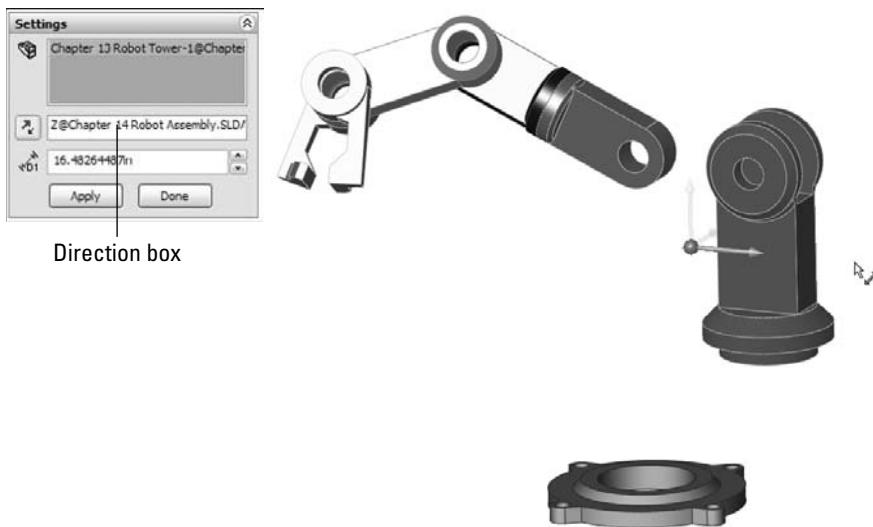
SolidWorks Help says that you can drag the sphere of the triad onto a face to change the direction, rather than selecting a face in the Direction box. You must also hold down the Alt key while dragging it in order for it to stick to an entity. ∎

Exploding the Twist Arm subassembly provides the opportunity to show a couple of useful subassembly functions. You can explode the parts of a subassembly either together as a unit or individually. You can even reuse explode steps from the subassembly, which is what you will do here.

FIGURE 14.15

Changing the direction of the explode

Direction box

To explode the subassembly as a single part and then reuse its explode steps from the subassembly file, ensure that the Select sub-assembly's parts check box is unselected, as shown in Figure 14.16.

Caution

You cannot reuse a subassembly's explode steps if the subassembly is set to Flexible. SolidWorks will tell you that there are no explode steps to reuse if you try to reuse the explode steps of a flexible subassembly. In order to work around this, you can set the subassembly to solve as Rigid, reuse the explode steps, and then set the subassembly back to Flexible. ■

Tip

While exploding the parts, you should rotate the view from time to time. Unless you are creating the explode for a particular point of view, the explode may look very different if you rotate it a little. ■

For the final explode step, the grippers will explode individually in opposite directions. Remember that these parts belong to another subassembly. If you create an explode step with the Select sub-assembly's parts option turned unselected or selected, you will not be able to change it later, so you need to pay careful attention to what you are doing.

Select the Select sub-assembly's parts option, select one gripper, Alt+drag the triad to set the direction, and then drag the distance of the explode. If you are in the mood to submit an enhancement request to SolidWorks, then you may want to request a Symmetrical Explode function for situations such as this. Figure 14.17 shows the finished result of the explode.

FIGURE 14.16

Exploding a subassembly

FIGURE 14.17

The finished explode

You may have noticed that each explode step creates a dashed line to show where the explode came from. Unfortunately, these dashed lines cannot be turned into usable explode lines on documentation. You must create explode lines manually, using 3D sketches. Although 3D sketches can be very tricky to use, for this purpose, you can limit their function to simple straight lines. As long as you try for simple results, the Explode Lines feature should work well.
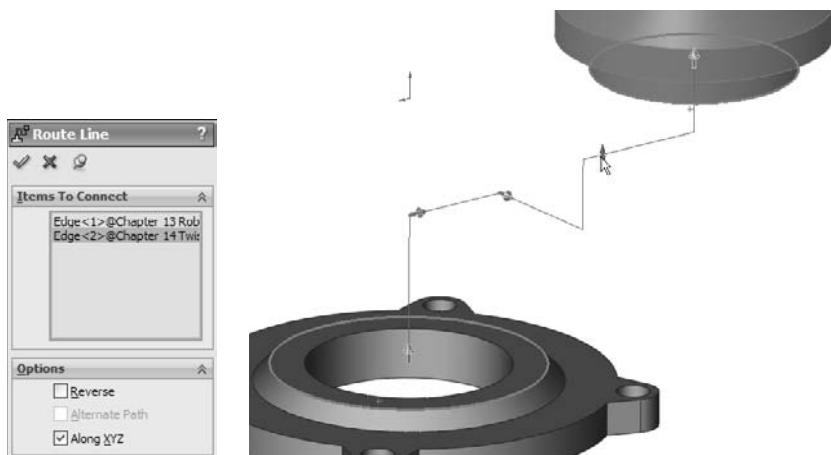
Collapse Items in the RMB menu refers to expanded ConfigurationManager lists, while *Collapse* refers to un-exploding the Exploded View.

To initiate the Explode Line Sketch function, you can choose Insert ➪ Explode Line Sketch. This displays the Explode Sketch toolbar and the Route Line PropertyManager interface. Creating nice explode lines takes a little practice, but it is easiest with circular parts, or circular features on parts with other shapes. Selecting circular edges makes the line start from the center of the circle.

If after you have selected two circular edges, the explode line goes the wrong way, you can click the arrow at the start of the line. Notice in Figure 14.18 that the line seems to take an unnecessarily circuitous route. This is because the explode directions were not square to the assembly Origin. To work around this problem to some extent, you can deselect the Along XYZ option in the Route Line PropertyManager. You can move the jogs by bringing the cursor close to the line and selecting the arrows that pop up.

Redirecting explode lines
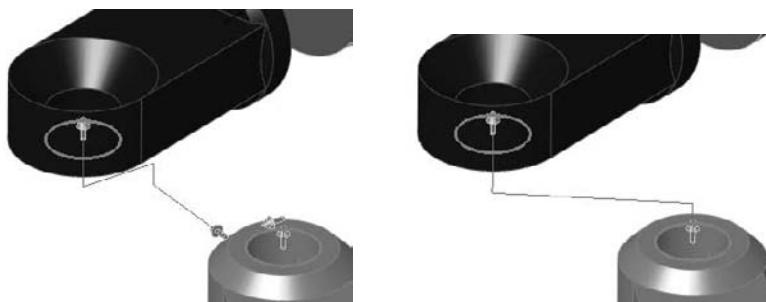
If you need to route an explode around other lines or parts, you can use the Jog Line in the explode sketch.

Figure 14.19 shows the difference between using the Along XYZ option (image to the left) and deselecting it when the explode is not square to the assembly Origin (image to the right). The completed explode lines are shown in Figure 14.20.
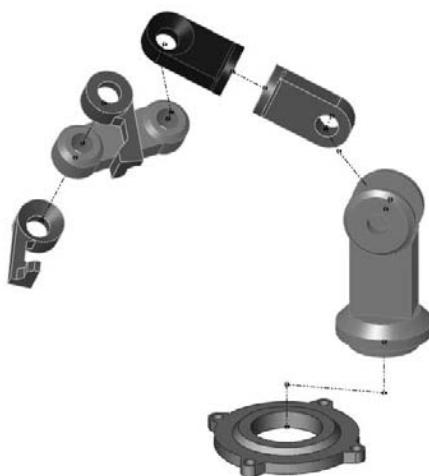
FIGURE 14.19

The Along XYZ option, selected and deselected

FIGURE 14.20

Completed explode lines

You can animate the explode or collapse from the RMB menu. To do this, right-click the exploded view and select Animate Explode or Animate Collapse. This method does not offer recording or Photoworks rendering like Animator software, but it is fast and easy. You cannot see explode lines during an animation.

Tutorial: Working with Assembly Configurations

To begin this tutorial, open the assembly named `Chapter 14 Bike.sldasm`. This assembly is made from the same parts as the assembly that was used in Chapter 12, but it will be used differently here. This file contains all the aspects that you need to work with in this chapter, including subassemblies, motion, and part configurations.
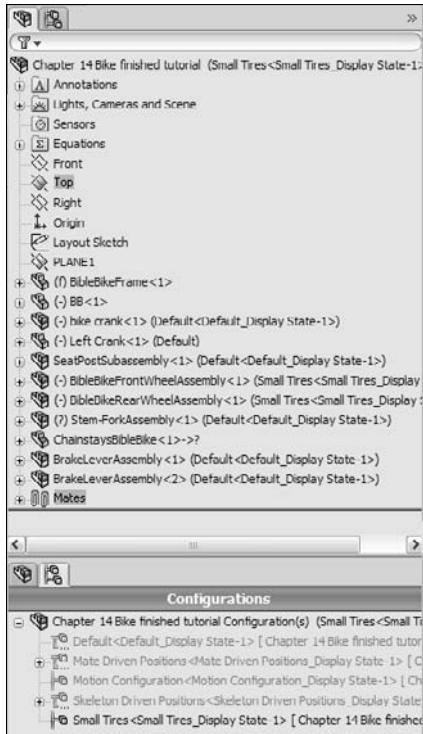
To learn how to work with assembly configurations, follow these steps:

1. **Prepare to use configurations by splitting the FeatureManager window into an upper and lower pane.** Place the FeatureManager on the top and the ConfigurationManager on the bottom.

2. **Before starting to make changes to this assembly**, add the top-level configurations that you will need, as follows:

   - Small Tires

   - Motion Configuration

   - Skeleton Driven Positions

   - Mate Driven Positions

   The configurations will list alphabetically.

3. **Make sure that the Advanced options for each configuration are set to Suppress new features and mates and suppress new components.**

4. **Activate the Small Tires configuration.** Figure 14.21 shows the FeatureManager up to this point.

FIGURE 14.21

The FeatureManager and ConfigurationManager up to Step 4
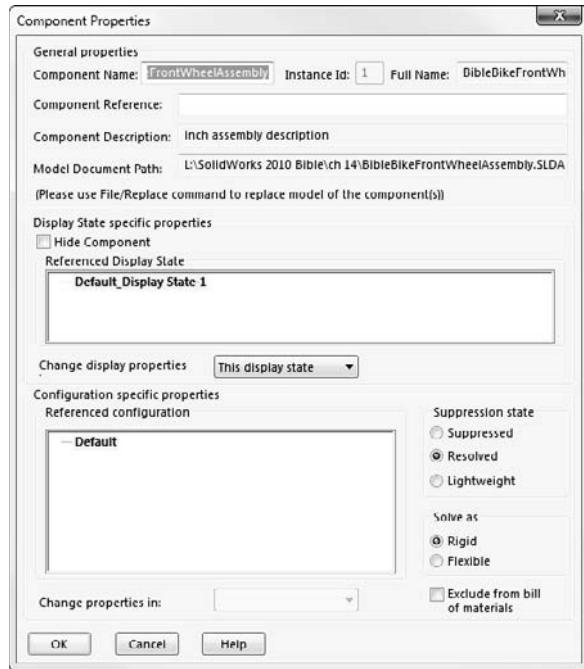
5. **Open the Front Wheel Assembly in its own window and switch to the ConfigurationManager.** Add a configuration called Small Tires, and change the tire to the configuration called Small Tire, which has already been created.

6. **Switch back to the main assembly window (using Ctrl+Tab), RMB-click the Front Wheel Assembly in the FeatureManager, and select Component Properties.** Select the Small Tire configuration for the Front Wheel assembly, as shown in Figure 14.22.

FIGURE 14.22

Changing the tires in the Component Properties dialog box
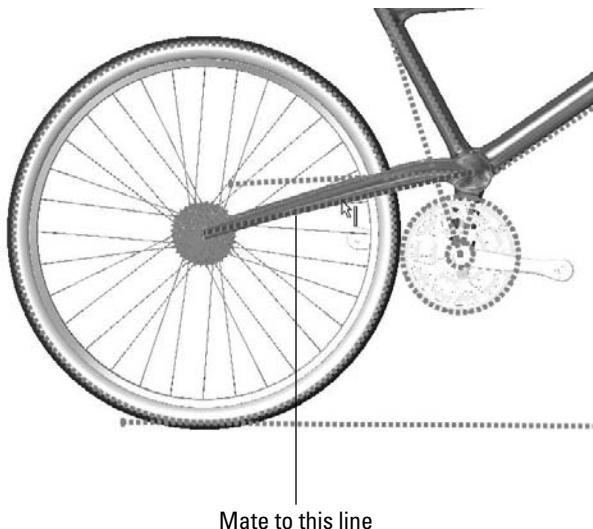
7. **Repeat Steps 4 to 6 for the Rear Wheel assembly.**

8. **Double-click another configuration from the list and watch the assembly change from small to fat tires.**

9. **Change to the Motion configuration.** RMB-click the Stem-Fork assembly, select Component Properties, and set the assembly to be solved as Flexible.

10. **Exit the dialog box and check to see that the fork linkage mechanism moves by dragging the fork.** Notice that the fork works but that the front wheel does not move with it. The bike design is not yet complete, so you do not need to worry about that at this point. Putting the front wheel in the fork assembly could be used to make the wheel move with the fork.

11. **Switch to the Skeleton Driven Positions configuration.**

12. **Display the assembly layout sketch at the top of the FeatureManager.**

13. **Create two new derived configurations under the Skeleton Driven Positions config-uration, one called Default Position and the other called Compressed Position.**

14. **Activate the Default Position configuration, and make a coincident mate between the Top plane of the Chainstay part and the sketch line indicated in Figure 14.23.** Again, the wheel does not move at this time.

FIGURE 14.23

Positioning the rear of the bike

Mate to this line

15. **Activate the Compressed Position configuration and make a coincident mate between the same plane and the line that is angled up at 10 degrees.**

Note

For these configs you also need to set the Advanced options just as you set the top-level configs in Step 3. If you do not do this, you may need to manually suppress the unwanted mates in the appropriate configurations. ■

16. **Switch to the Mate Driven Position configuration.** Change the stem-fork assembly to a flexible subassembly (RMB-click and choose Component Properties ➪ Solve as Flexible).

17. **Add new derived configurations called 1, 2, and 3.** While creating the new configs, ensure that the Suppress new features and mates and Suppress new components options are selected. Leave the 1 configuration activated.

18. **Make an angle mate between the Bike assembly Top plane and the face of the link, as shown in Figure 14.24.**

FIGURE 14.24

Using angles to position the fork

19. **Once the mate is complete, double-click the angle dimension (you may have to double-click the angle mate to get it to display and then zoom out to see it), and change the value to 18 degrees.** Again, with the change, the fork may fly to an unexpected location. Pressing Ctrl+Q brings it back.

20. **Switch to the configuration 2, unsuppress the angle mate made in Step 18 and change the value to 25 degrees.** You may have to change the configuration 2 to Flexible, although it should inherit this property from the parent config.

Note

You need to set the Fork assembly to solve as Flexible for each configuration. You may also need to control the alignment for the angle mate manually for each configuration. ■

Summary

Display States in the assembly can save you a lot of time because they change faster than configurations and offer more options for visualization, including mixed display modes. Assembly design tables can select Display States and drive many other parameters in assemblies. Remember also that the Modify Configurations and Configuration Publisher work in assemblies as well as in parts. Assembly configurations are a very powerful tool for product variations and performance, especially when combined with SpeedPak.

Using Component Patterns and Mirrors

I n SolidWorks assemblies, the word *component* can refer to either parts or subassemblies at the top level of an assembly. Component patterns can therefore be patterns of parts, subassemblies, or combinations of parts and subassemblies.

Performance

For best pattern performance, you should use subassemblies as the patterned unit as much as is practical. Multiple patterns of individual components are not as efficient as a single pattern of multiple components. A single pattern of a single component, where the single component is itself a subassembly, is the best choice, if available.

Another performance issue is the fact that component patterns require external references (for the direction or center of the pattern). These external references have the potential to increase rebuild times if you do not choose them carefully. Reference geometry internal to the assembly would be the best choice.

Although you can experience possible performance problems with patterns, patterns can also significantly decrease the number of mates in an assembly, which always improves performance. ∎

Component patterns come in two varieties: local patterns, which include linear and circular patterns, and feature-driven patterns, which are driven by a feature pattern in a part. The local patterns are obviously somewhat limited, but because feature-driven patterns follow patterned features, they can also be driven by sketch-driven patterns. Curve-driven and fill patterns can also be used.

It is possible to focus only on the basics, to simply make patterns that exist in the present moment. However, if you are interested in creating features that will adapt to future changes, then you will find this chapter useful.

Using Local Component Patterns

Local component patterns are limited to linear and circular patterns. The linear pattern directions work just like the linear pattern feature in parts, and must reference a line, axis, edge, and so on to establish the direction. In an assembly, this means that the feature uses model geometry from a part (solid or surface edges, sketches, reference geometry), an assembly sketch entity, or an assembly reference geometry entity (such as an axis). This is important to keep in mind if you are concerned about circular references.

Best Practice

If you have a feature pattern in a part, you should take advantage of it and use a feature-driven pattern instead of a local pattern. The rebuild time may be longer, but associativity between the part and assembly helps maintain design intent. ■

Local pattern references

If you still need to create a local pattern, it is best to use a reference that is not dependent on part geometry. Remember that when part geometry is used for this purpose, the parts must be solved first (sketches and features rebuilt), then the mates must be solved (to position the parts), then any in-context references must be solved (which may change the part geometry), and then any assembly features or component patterns must be solved. As a result, it is best practice to use as pattern direction references assembly sketches that do not reference anything else. The assembly sketches should sit at the top of the assembly FeatureManager to ensure that they are not picking up references from the history-based features in the design tree (mated components, patterns, assembly features, and so on).

When a local pattern really requires a reference from a part, you have no alternative. However, if you can avoid this by using a sketch assembly skeleton to which the parts are mated and also used for the pattern references, then you should do so. At all costs, you should avoid using in-context features, assembly reference geometry that is dependent on part geometry, and assembly features (other than sketches) for the local pattern reference.
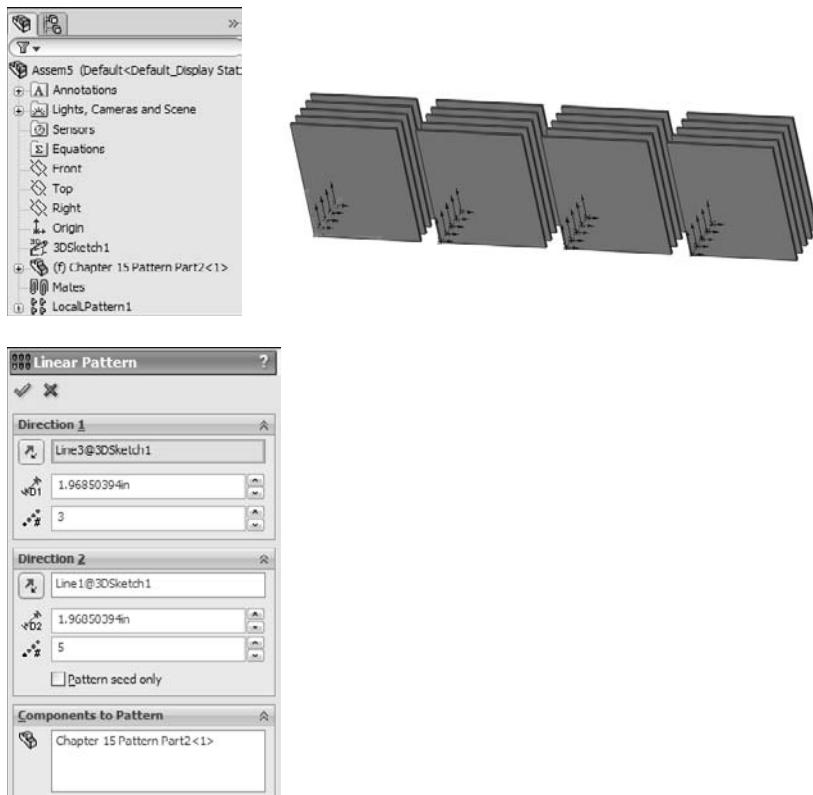
Figure 15.1 shows a sample organization of one way that you can set up an assembly to properly control local component patterns. The lines shown can be created in either two 2D sketches or a single 3D sketch. The lines are dimensioned from planes, which enables them to be angled for patterns that are not square to the coordinate system of the assembly, but still lie on its main planes.

In most situations the rebuild time penalty of using model geometry to establish pattern direction is fairly slight. The sketch method is probably most justifiable in large, complex assemblies, or in assemblies requiring long rebuild times. Figure 15.1 also shows the PropertyManager interface for the local pattern.

The Assembly FeatureManager for local component pattern setup

Notice where the pattern is placed in the FeatureManager. You can reorder the pattern feature in the design tree, but you cannot move it above the mates. Interestingly, you can move the sketches after the pattern, even though the pattern is dependent on one of the sketches. Obviously, SolidWorks is working with the order behind the scenes in such a way that the user cannot make mistakes.

Pattern seed only

All the aspects of the interface should be familiar, such as the direction, instances, and spacing. The Pattern seed only option is also used in feature patterns.

Patterning the seed only is designed to allow you to create a single pattern in two directions that are separated by 180 degrees, where the internal instances do not overlap one another. For example, if you take a basic two-directional pattern and change the angle between the directions so that
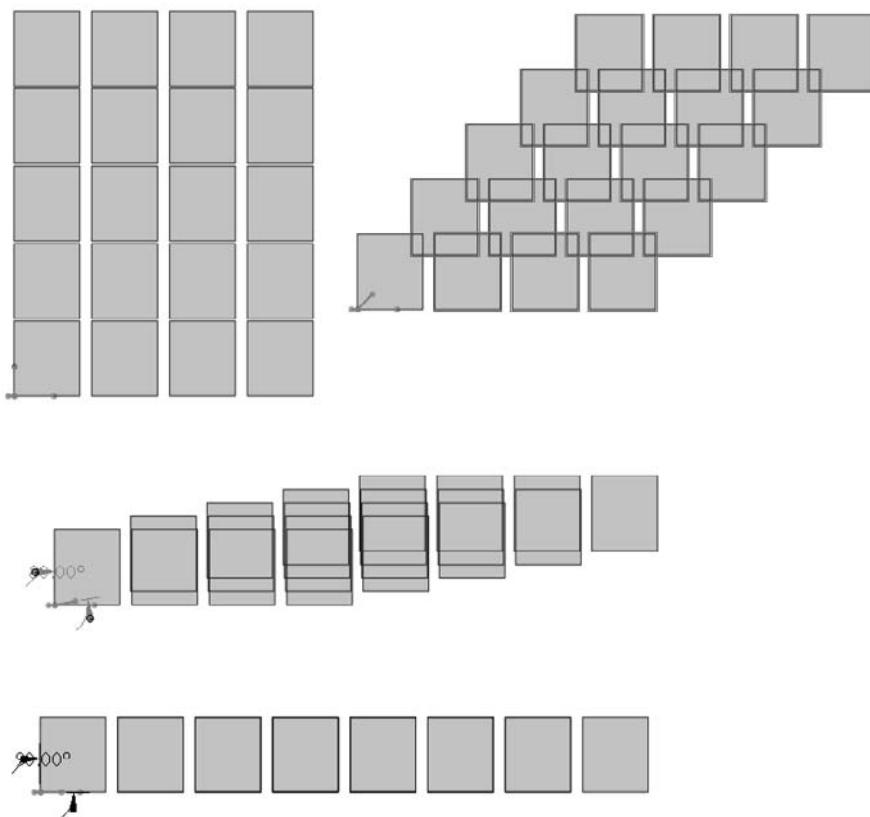
they are anti-parallel (parallel but going in opposite directions), then all the component instances that were between the two legs of the L created by the two directions will come to overlap one another when they are laid out in a straight line.

Figure 15.2 shows how a 5-by-4 pattern with 20 instances becomes a 1-by-8 (the seed is not counted twice) pattern. To be clear, the figure shows a two-direction pattern where the angle between the directions becomes shallower and shallower, until the two directions are parallel or anti-parallel. When this happens, the other 12 instances overlap the remaining ones. When you use the Pattern seed only option, you are only patterning the two legs of the L, and not the instances in between. Having parts that overlap can cause problems with Bills of Materials (BOMs) and mass properties due to having duplicate parts.

FIGURE 15.2

Use the Pattern seed only option for two directional pattern
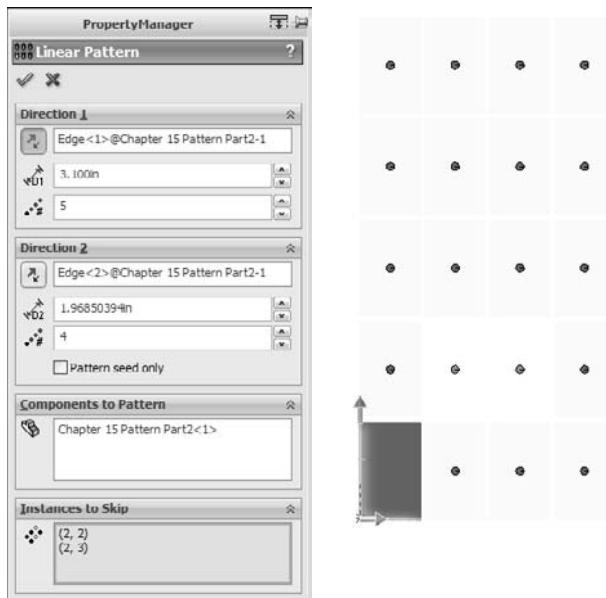
Instances to Skip

The Instances to Skip option for component patterns, shown in Figure 15.3, works just like the equivalent option for features. Click the dots in the graphics window to toggle each instance of the pattern. On the screen, the instances to keep use pink dots and the instance to skip use orange dots. The colors are almost indistinguishable at a relatively wide spacing.

FIGURE 15.3

The Instances to Skip option

Mirror Components
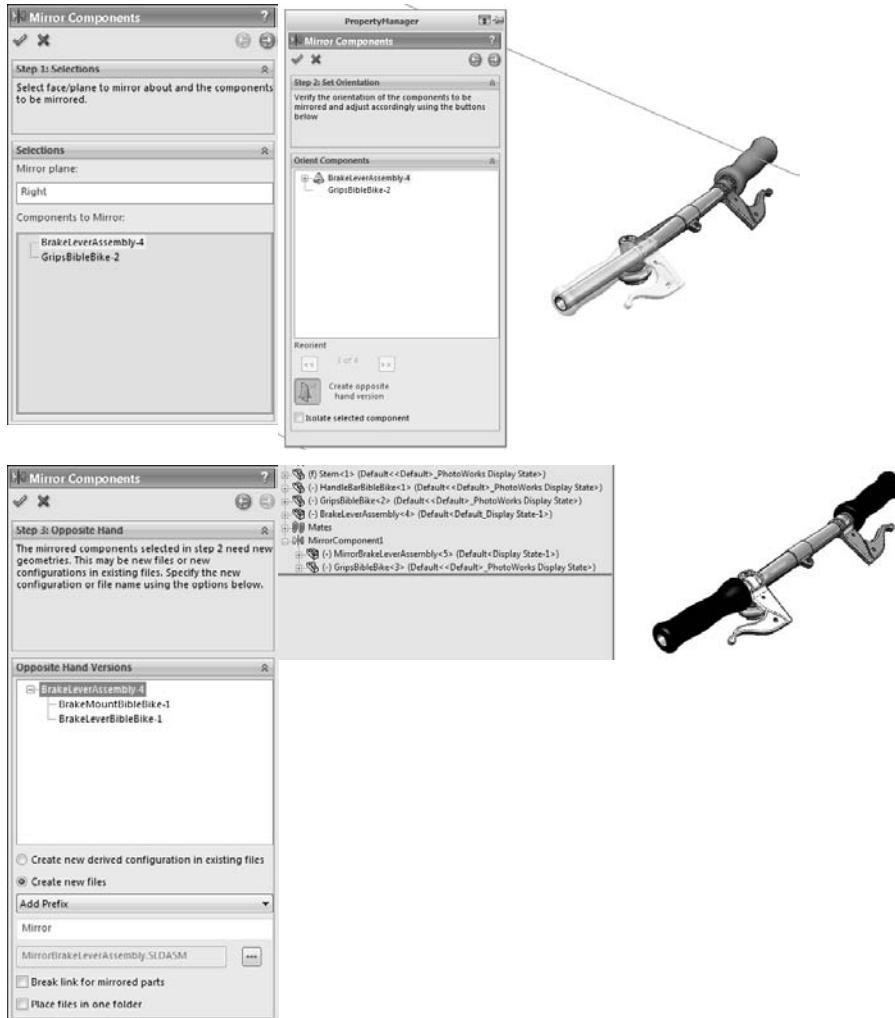
Mirror Components is a revamped feature in SolidWorks 2010 that leaves a feature in the tree just as pattern features do. It gives you the option to align each mirrored part, and to decide which parts require opposite handed versions and which can just use the original in a mirrored location.

Figure 15.4 shows the PropertyManager for this assembly feature.

FIGURE 15.4

Mirroring components of an assembly with the Mirror Components tool

The process to mirror components in the assembly has 3 steps, identified in the PropertyManager screen shots in Figure 15.4:

1. **Selections.** Select the parts and subassemblies that you want to mirror.
2. **Set Orientation.** Identify the components that need to have opposite hand versions, and for the others, you can toggle through the available options for placement.

3. **Opposite Hand Versions.** For the parts that need opposite hand versions, you can assign them to use new parts or derived configurations.

You should find this version of the mirror components capability to be more robust and thorough than previous versions.

Using Feature-Driven Component Patterns
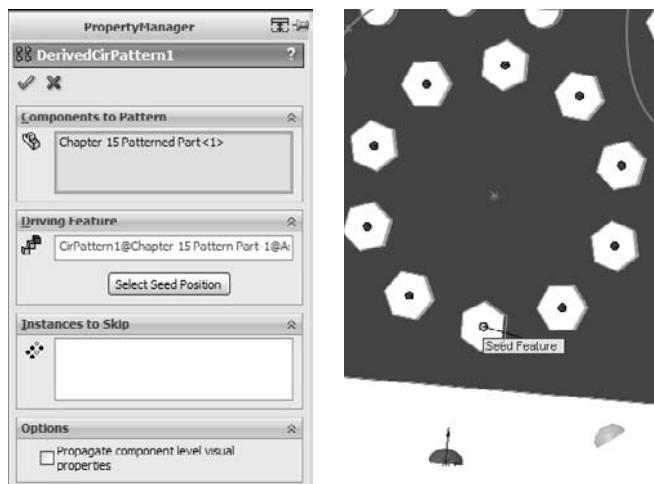
By their very nature, feature-driven component patterns defy some of the best practice suggestions in this book. This is because the pattern is driven by part geometry, and the part must first be solved (by solving features internal to the part) and then placed (by solving mates); only then can the feature-driven pattern be solved. Nevertheless, I recommend using feature-driven patterns over local patterns when available because of the parametric link.

Note

For the feature-driven component patterns, the location of the initial component is important. You need to match the placement of the initial component with the position of the original feature from which the pattern was created, not one of the patterned instances. You can get around this requirement if you use the Select Seed Position option. When you do this, the feature pattern instances all appear with dots and you can select which instance to use as the seed. Again, the selected dot is blue and unselected dots are purple, nearly indistinguishable at the size and spacing of the dots. Figure 15.5 shows the PropertyManager interface for a feature-driven component pattern. ■

FIGURE 15.5

The feature-driven Component Pattern interface

You can nest feature-driven component patterns such that one component pattern is patterned by another component pattern, just as you do using feature patterns. The second pattern can be a local pattern or a feature-driven pattern.
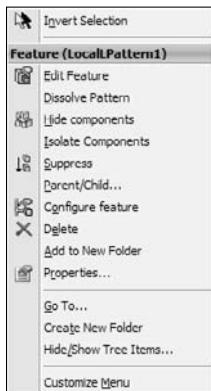
Understanding Other Pattern Options

Figure 15.6 shows the RMB menu for a component pattern. The components of the RMB menu are as following:

FIGURE 15.6
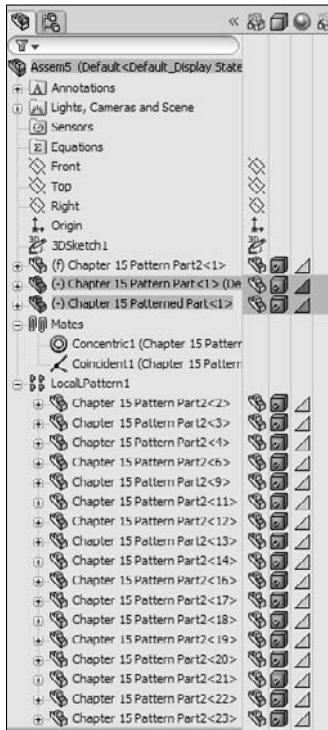
The component RMB menu

- **Dissolve Pattern.** The Dissolve Pattern option removes the component instances from the pattern feature and puts them in the main part of the assembly FeatureManager. The components just become normal components in the assembly without the intelligence of the pattern feature placing them. The components are left in the assembly without any mates, simply floating in position.

- **Add to New Folder**. You can add patterns to folders. If you have a list of patterns at the end of an assembly, it may make sense to group them into related folders for the purpose of organization. This is the same as using folders for features, mates, or components.

- **Component pattern display options.** You can change the appearance of individual component pattern instances individually or collectively as a pattern feature. Figure 15.7 shows the display pane where you can control these display options.

FIGURE 15.7

The display pane for controlling display options

- **Component patterns and configurations.** Individual instances of the component pattern also enable you to control configurations. After you create the pattern, you can select individual instances and change their configs. This can be extremely useful if you have a mechanism subassembly shown in various positions; for example, patterned around an indexing dial.

Tutorial: Creating Component Patterns

To learn how to create component patterns, follow these steps:

1. **Open a new assembly.** Create a new 3D sketch and draw three lines from the Origin out at odd angles so that they do not pick up horizontal or vertical automatic relations. Draw two of the lines, then rotate the view and press the Tab key and draw the third line.
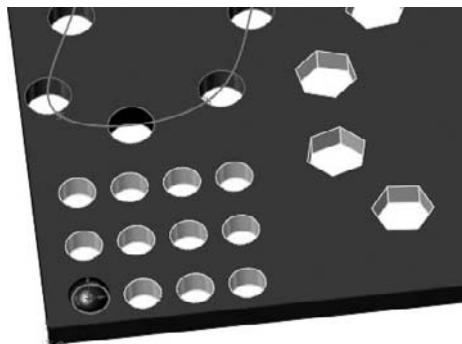
2. **Apply sketch relations such that each line lies on a plane: one line on the Front, one on the Top, and one on the Right.**

3. **Click Close to exit the sketch when you are done.**

4. **Open the part from the CD-ROM named** `Chapter 15 Pattern Part.sldprt.` This part already contains several features you can use to practice using feature-driven component patterns.

5. **Insert the part into the new assembly created in Step 1.** Locate the part at the assembly Origin such that the part Origin matches the assembly Origin.

6. **Open the part called** `Chapter 15 Patterned Part.sldprt` **and place it in the assembly.**

7. **Place the small part on the original feature of the rectangular pattern of round holes near the origin, as shown in Figure 15.8.** All the original features are colored red. Remember that Alt+dragging the circular edge on the flat side of the part enables you to SmartMate the part to the round holes. It cannot help you with the rectangular or hex holes. For these, it may be best to show the sketch for the holes and place the part with respect to the sketch entities.

FIGURE 15.8

Placing the Patterned Part
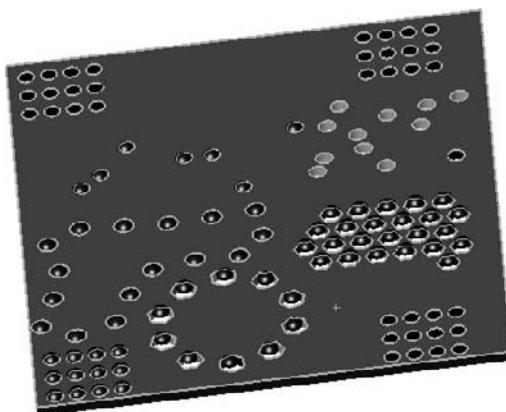
8. **Create feature driven patterns (Insert ⇨ Component Pattern ⇨ Feature Driven).** Try to use each of the patterns from the pattern part. For each new pattern, make a copy of the patterned part and place it in one of the holes. Remember to use Select Seed Position option to pick a feature pattern instance instead of the original feature.

9. **Once you have created a few feature-driven patterns and have a better understanding of how it is done, right-click the top level of the assembly FeatureManager and select Collapse Items (near the bottom of the menu).** The point is simply to practice placing a part and patterning it with an existing feature pattern. The assembly might look like Figure 15.9 when you are done.

Several Feature-driven patterns
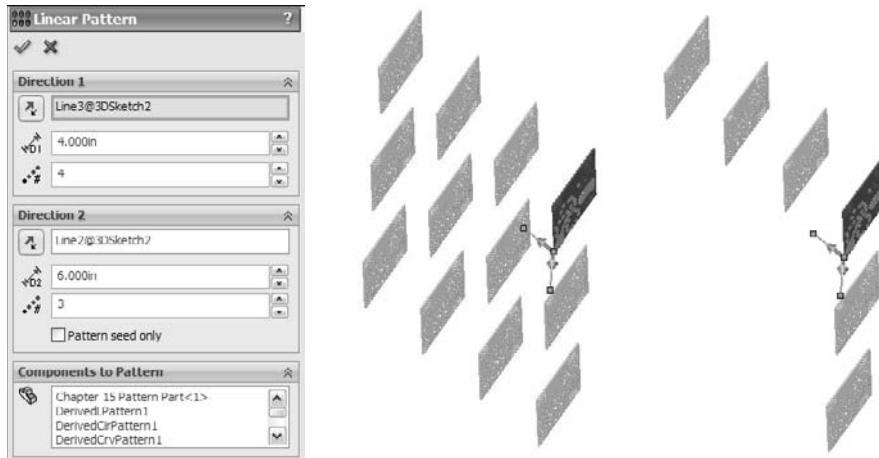
10. **Create a local pattern (Insert ⇨ Component Pattern ⇨ Linear Pattern). Select one of the sketch lines drawn in Step 1 as a pattern direction.**

11. **Highlight the Components to Pattern selection box**. Select the first part in the FeatureManager and then Shift+select the last pattern feature. This patterns everything in the assembly.

12. **Make the spacing four inches with three instances.**

13. **Create a second direction using another of the sketch lines with six-inch spacing and four instances.**

14. **Notice how the preview shows 12 instances of the patterned assembly.** Select the option for Pattern seed only and see how the preview changes to seven instances. Figure 15.10 shows this difference. Click OK to accept the feature.

FIGURE 15.10

Two direction patterns, one with and one without the Pattern Seed Only option

Summary

Performance and best practice are both issues that require compromise. Patterns can cause a performance reduction because of the nature of the references. However, they can also improve performance because the need for extra mates is reduced and it is easier to simplify the assembly by suppressing the pattern feature.

Feature-driven patterns are driven by feature patterns and transgress best practice suggestions, but they also add a parametric link, which updates the component pattern automatically. And they offer many more options that are driven by the pattern options available to features in a part.

16

Modeling in Context

I n this chapter, I present best practice suggestions and offer a balanced explanation so that you can evaluate the benefits and potential problems associated with in-context modeling. I give you enough facts so that you can decide for yourself how in-context techniques fit into your process.

In-context modeling is an extremely powerful technique for extending parametric design from individual parts to top-level assemblies. With the power comes potential for mistakes. If you are not careful, in-context modeling can lead to unexpected results.

Understanding In-Context Modeling

In-context modeling is also known as *top-down* modeling. It is a technique used to create relationships between parts in the context of an assembly, where the geometry of one of the parts is controlled by both the other part and the mates that position them relative to one another.

In-context, or top-down, modeling may be contrasted against *bottom-up* modeling. Bottom-up modeling involves making the parts in their own individual windows and assembling the finished parts into an assembly with mates.

In its most common form, a sketch in one part in an assembly is related to an edge in another part in the assembly. The relationship is specific to that particular assembly, and is only relevant *in the context of* that assembly.

For example, you may create a box and put it into an assembly. You must then create a lid that is parametrically linked to the size and shape of the box. You can create a lid part in the context of the assembly such that the lid always matches the box. Sketch relationships, dimensions, and feature end conditions from the lid can reference the box. When the box changes, the lid also changes *if the assembly is open*.

The assembly maintains a record of each in-context reference. If the box is changed with both the assembly and the top open, then the top updates, but if the box is changed without the assembly being open, then the lid will not update until the assembly is opened. The record of the reference that the assembly maintains is held in what is called an *update holder*, and in recent versions, it is all but forgotten.

Cross-Reference

Chapter 12 discusses in-context reference update holders. These are the pointers in the assembly that hold the reference information. These holders are hidden by default and do not enable any real functionality, but they do serve as a reminder that the assembly has in-context references and can be queried to tell you what parts the in-context relations go between. ∎

Advantages of in-context modeling

The advantages of in-context modeling are obvious. I spent the first couple of chapters of this book discussing the strengths of parametric design, and in-context modeling is just an extension of parametric techniques to include parts in the context of an assembly. Making a change to one part and having all related parts update offers indisputable advantages.

Some users approach modeling haphazardly, where if it works, it's good enough. For some types of work, this is acceptable and "good enough." For example, it usually works when you create something that will never be changed or if you are working on concept models.

On the other hand, users who need to build models that will be reused frequently and changed often must approach the decisions they make during modeling as if they were playing chess: Each decision has consequences; you rarely know exactly how things are going to turn out but need to prepare for the most likely contingencies.

In-context modeling presents the same types of decisions. You can do it fast and dirty or methodically and with much consideration beforehand.

Potential problems with in-context modeling

Some issues may arise from the technique of driving changes from a different part model within an assembly. There are no problems with the *overall concept* of in-context modeling; the problems occur with the *practical application of the technique*. In particular, the biggest problems seem to arise when in-context techniques are combined with other techniques. You must be very careful about file management issues when in-context references exist in your assembly.

Major potential problems include:

- Lost references due to renamed assemblies
- Convoluted references causing long rebuild times
- Circular references causing changes with each rebuild
- Frustrated users who don't understand how to manage change in an in-context scheme

However, you can overcome these difficulties. In the rest of this chapter, I show you how to make the most of in-context modeling.

Identifying alternatives to in-context modeling

The kinds of problems users typically have with in-context modeling are related to file management. Inexperienced users may rename a file and break links, or try to use an in-context part in some location other than where the in-context relation was created, and may not be able to make changes that they want to make, or changes happen when they do not want them to.

Sometimes these problems are the result of the user simply not understanding what to expect from the tool, and sometimes it is because the tool is not capable of what they are trying to achieve. Right or wrong, many users have developed an irrational fear of references that can control a part from outside of the part itself. In-context modeling in itself is not a bad technique, but sometimes it is not the best option. I recommend thoroughly understanding in-context and all related techniques before passing judgment on any of the techniques.

It's always important to identify alternative techniques because one tool never solves all possible problems. In-context modeling is powerful, but in some situations other techniques are better suited. In this case I want to introduce you to a couple of techniques that share with in-context the ability to control individual parts from a centralized location, but achieve that in ways that are somewhat different: Assembly layout modeling enables you to control individual parts not from other parts, but from an assembly level sketch. Multi-body modeling enables you to control several parts from a single part without worrying about the file management issues of having another assembly as the middle agent.

Assembly layout modeling

Assembly layouts are powerful tools that remove much of what some people object to in in-context modeling. Relationships in this technique are controlled by top-level sketches, where a single or multiple sketches can control most of the features on all parts through relationships between part sketches or features and the assembly sketch. This still creates an external reference that requires the existence of an assembly to update the relationships, but it is not a direct link between different parts in the context of the assembly.

Assembly layouts do not lend themselves well to dynamic assembly motion, but they are great for having a single location to drive an entire assembly. Assembly layouts come in two flavors, the generic layout which is simply done using sketches in an assembly; and the formal Layout feature,

which is essentially a 3D sketch in the assembly with special properties. The Layout feature is described in more detail later in this chapter. Assembly layout sketches are covered in more detail in Chapter 12.

Multi-body modeling

Multi-body modeling, like in-context modeling, is a powerful technique with positives and negatives. If you model what will later turn out to be separate parts together in a single part, you can avoid in-context modeling altogether. It is not recommended that you replace assemblies with multi-body modeling for a number of reasons, such as limitations of multi-body techniques for common assembly operations like dynamic assembly motion and interference detection. Used judiciously, multi-body modeling can help you save time making models that hold up well to changes. Multi-body modeling is discussed in depth in Chapter 26, along with a direct comparison to assembly modeling and a justification for why you shouldn't replace assemblies with multi-body parts.
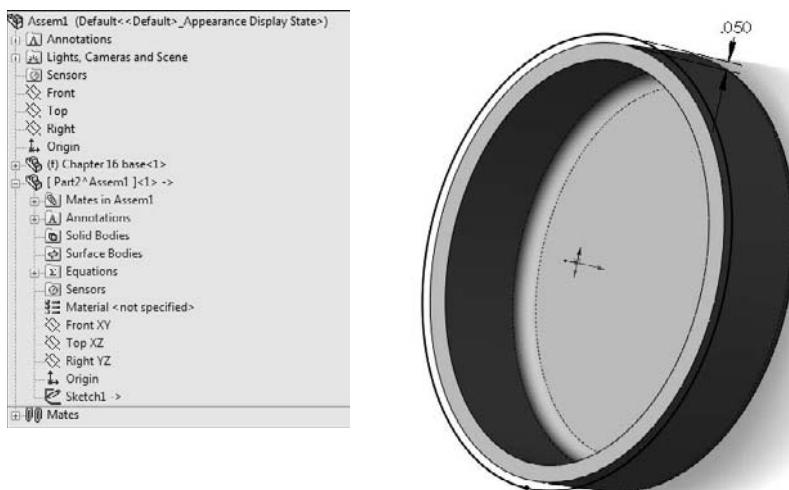
Dealing with the Practical Details of In-Context Modeling

Figure 16.1 shows a simple box with the sketch of a simple top for the box. Notice in the FeatureManager that two parts are listed as the top and base. The .050-inch offset is creating a sketch in the top part that is driven by the edges of the base part. This simple assembly demonstrates the in-context process in the sections that follow.

FIGURE 16.1

The top of the box being built in-context

The in-context process

You can perform in-context modeling using one of two basic schemes. You can build parts from the very beginning in the context of the assembly (using the Insert ➪ Component ➪ New Part menu option) or you can start them using bottom-up techniques, creating the parts in a separate part window, adding them to the assembly, and then adding additional in-context features later.

Starting out in-context

To start a new part in the context of an assembly, you will first assume that the assembly contains another part. Creating a new part in a blank assembly is not very interesting. In this case, I am using the assembly shown in Figure 16.1. To create the new part, choose Insert ➪ Component ➪ New Part. This command is also available through a toolbar button (shown to the left) that you can place on the Assembly toolbar. At this point, SolidWorks prompts you to select a face or plane on which to locate the new part. When you select the face or plane to place, SolidWorks places the Front plane of the new part on it, opens a new sketch and adds an InPlace mate to the assembly. In-context parts start as virtual parts, saved inside the assembly; you can choose to save it as an external or internal part the next time you save the assembly. I discuss virtual part functionality later in this chapter.

The InPlace mate

The mate that SolidWorks adds automatically when a part is created in-context is called an *InPlace* mate. It works like the Fixed option, but is actually a mate that is listed with the other mates and that may be deleted but not edited.

The InPlace mate clamps the part down to any face or plane where it is applied. It is meant to prevent the in-context part from moving. I will explain later in this chapter why it is so important for in-context parts not to move.

Alternative technique

Instead of using the Insert ➪ Component ➪ New Part command, you can simply create a blank part in its own window and save it to the desired location. Then insert the blank part into the assembly and mate the origins coincident. You can then edit the part in-context, the same as if you had created it in-context from the beginning. The only difference between parts developed this way and parts created in-context is the InPlace mate. The InPlace mate cannot be edited and does not have relations to other geometry in the usual sense. Many users feel more secure with real mates to real geometry, which they can identify and change if necessary.

Valid relations

Sketches, vertices, edges, and faces from the other parts in the assembly can be referenced from the in-context part as if they were in the same part as the sketch. Most common relations are concentric for holes, and coincident for hole centers. Converted entities (On-Edge relations) make a line-on-edge relation between the parts, and Offset sketch relations are also often used.

Other types of valid in-context relations include in-context sketch planes and end conditions for extrude features such as Up to Face and Up to Body. Beyond that, you can copy surfaces from one part using the Knit Surface feature or the Offset Surface feature. I discuss surfacing in more detail in Chapter 27.

Working in-context

When you are working in-context or using in-context data, visual cues offer information about the part that you are working on. The following topics are meant to help you understand what is going on while you are working in-context.

Text color
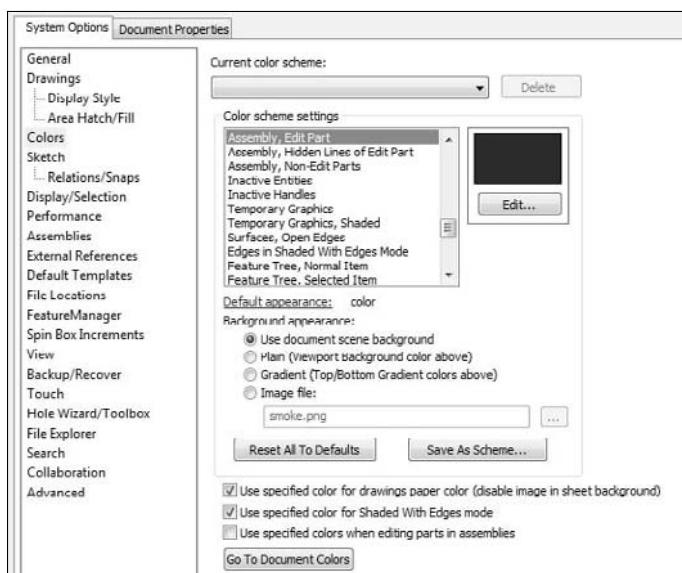
When you are working in-context, the FeatureManager text of the part that you are working on turns blue. This should make it immediately obvious first that you are working in-context, and, second, which part is being edited.

Part color and transparency

You can control the color and transparency behavior of parts in the assembly where a part is being edited in-context by choosing Tools ➪ Options ➪ Colors page. Figure 16.2 shows a detail of this page. The option at the bottom of the dialog box determines whether the colors specified in the list at the top are used or ignored. If they are ignored, the parts are the same colors they would be if you were not using in-context techniques.

FIGURE 16.2
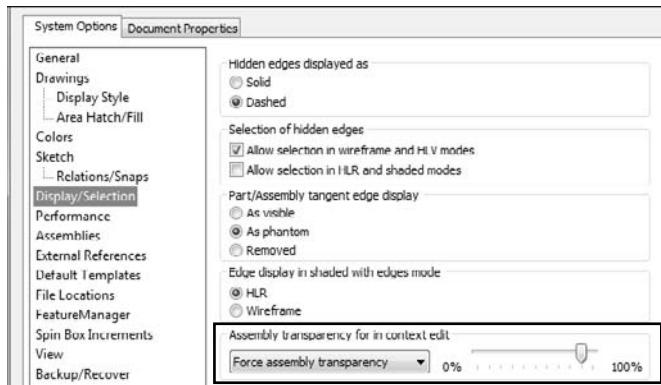
Part-color settings for in-context control

The Tools ➪ Options ➪ Display/Selection Assembly transparency for in context edit controls the transparency of the parts not being edited. Figure 16.3 shows this setting. Forcing the non-edited parts to become transparent helps you keep focus on the part you are editing in the assembly.

FIGURE 16.3

Part transparency for in-context control

The options in the Assembly Transparency drop-down list are

- **Opaque Assembly.** All parts that are not being edited when an assembly component is being edited in-context turn opaque, even if they are otherwise transparent.

- **Maintain Assembly Transparency.** Leaves all assembly components in their default transparency state.

- **Force Assembly Transparency.** Forces all the parts, except for the one being edited in the assembly, to become transparent.

These options reflect personal preference more than anything else, but it is useful to have a reminder as to whether a part is being edited in the assembly or the assembly document is being edited in its own window.

Tip

The color selected in the box shown in Figure 16.2 controls both the text color and the color of the part shown in the graphics window. ∎

Edit Component button

You can use the Edit Component button in two ways. First, after you have created a part in-context, seeing the Edit Component button depressed serves as a reminder that you are editing the part rather than editing the assembly. Along with the part color and transparency displays, this is important feedback because assembly functions such as mates, exploded views, and others are not available when you are editing the part.
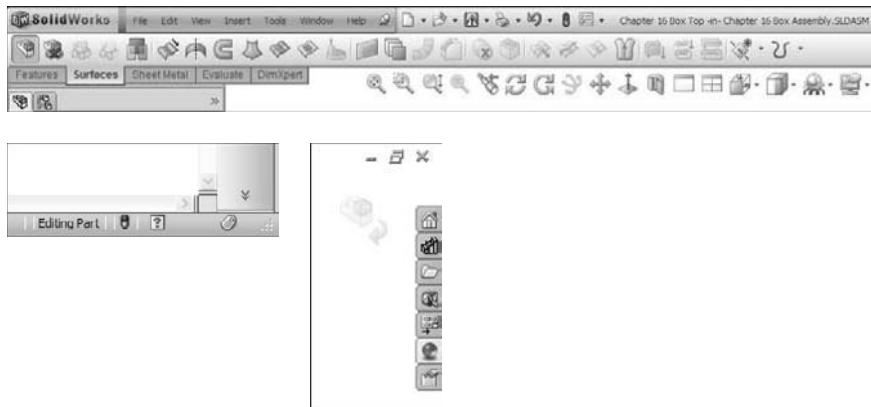
Second, you can use the Edit Component button to begin or finish editing a part that is already in an assembly. When you are editing a part in the context of an assembly, the title bar of the SolidWorks window reflects the fact that you are editing a part in an assembly, the toolbar changes to a part-editing toolbar, and the lower-right corner of the taskbar displays the words *Editing Part*, as shown in Figure 16.4.

Third, a confirmation corner image exists in the upper-right corner of the graphics window when you are editing a part in the context of the assembly. This makes it easier to exit Edit Component mode.

FIGURE 16.4

Indicators that you are editing a part in-context

Editing a component can also mean editing a subassembly in the context of the top-level assembly. You can create in-context assembly features and mates if necessary; however, you will do this far less frequently than editing parts in-context.

Note

Creating in-context relations is not the only reason to edit a part or subassembly in the context of the top-level assembly. Sometimes it is simply more convenient to do normal editing when you are in the top-level assembly; this way you can see how the part relates to other parts after making changes in the assembly without making relations between the parts.
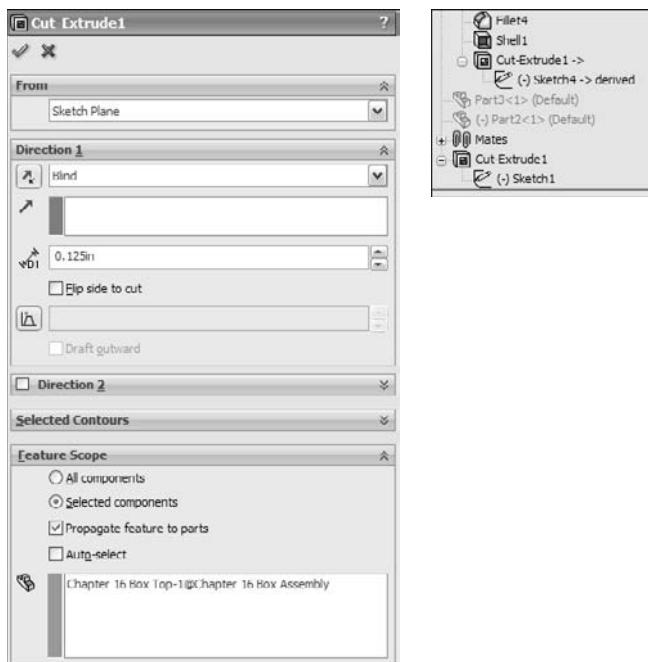
Editing a subassembly in the context of the upper-level assembly is often useful as well, to see how changing subassembly mates affects the top level. ■

Probably the most common mistake you can make with in-context editing has to do with editing the part versus editing the assembly when they add a sketch. If you intend to add a sketched feature to a part in the context of an assembly, but you fail to switch to Edit Part mode before creating the sketch, then the sketch ends up in the assembly rather than the part; you can only do limited things with a sketch in an assembly. Likewise, if you intend to make an assembly layout sketch, but you do not switch out of Edit Part mode, you end up with a sketch in a part that cannot do what you want it to do.

Fortunately, SolidWorks has added a remedy for the first situation. When you make a sketch in the assembly but need to make a feature in the part, you can choose the Propagate feature to parts option in the Feature Scope area of the PropertyManager for the feature, as shown in Figure 16.5.

FIGURE 16.5

Propagating an assembly feature to the part

Notice in the image on the right that the last sketch in the part appears as derived. This means that the sketch and the feature are still driven from the assembly, but they have been propagated to the part enough to allow the feature to be edited in the part. I wouldn't go this route just because you made a mistake and it's simpler to do this than to move the sketch to the part, but it is an option that is valid in some situations.

Interestingly, this feature cannot be deleted from the part; you must delete it from the assembly.

External reference symbol
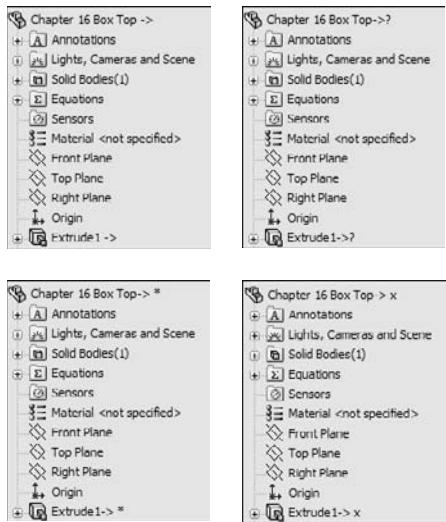
The external reference symbol appears as a dash followed by a greater-than sign (->). External references indicate more than just in-context features. You can also create external references by using the Split Part command as well as the Insert Part (base or derived part) or the mirrored part functions. Figure 16.6 shows the expanded FeatureManager for a part with an in-context reference in a sketch.

External references can have four states, which Figure 16.6 shows. These are

- In-context (->)
- Out-of-context (->?)
- Locked reference (->*)
- Broken reference (->x)

FIGURE 16.6

The in-context "carrot" on Extrude1 and Sketch1

In-context ->

The in-context symbol signifies that the relation created between two parts within the current assembly is fully resolved. It can find both parts involved in the relationship and the assembly where the relationship that was created is active.

Out-of-context ->?

Out-of-context means that the document — usually but not necessarily an assembly — where the reference was created is not open at the time. It is indicated by an in-context symbol followed by a question mark. You can open the document where the reference was created by clicking the right mouse button (RMB) and selecting the Edit In Context option from the menu. Edit In Context opens either the parent part of an inserted part or the assembly where the reference was created for an in-context reference. When you open the referencing document, the out-of-context symbol changes to the in-context symbol.

Locked reference ->*

You can lock external references so that the model does not change, even if the parent document changes. The symbol for this is ->*. Other features of the part may be changed, but any external reference within the part remains the way it is until the reference is either unlocked or removed. In the top and base example I mentioned earlier, this means that if the Bottom part is changed, and the external reference on the Top is locked, then the Top will no longer fit the Bottom.

One of the best things about locked references is that you can unlock them. They are also flexible and give you control over when updates take place to parts with locked references.

Broken reference ->x

The broken reference is another source of controversy. Some users believe that if you make in-context references, the best way to respond to them is to break them immediately. However, I would argue that using the Break References function is *never* a good thing to do. I believe that you should remove the reference by editing the feature or the sketch or change it to make it useful.

The problem with a broken reference is that it has absolutely no advantage over a locked reference. For example, while locked references can at least be unlocked, broken references cannot be repaired. The only thing that you can do with a broken reference is to use Display/Delete Relations or to manually edit features to completely remove the external reference. Perhaps it would be better for SolidWorks to replace Break References with a function called *Remove References*. Would anyone like to make an enhancement request?

Best Practice

Best practice is to not put yourself in a situation where you are using broken references. Parametric relations should not change if the driving geometry does not change.

You cannot selectively lock or break external relations. For example, all the external relations in the part can be locked, all the external relations can be broken, or none of them can be locked or broken. If you need to selectively disable relations, then you should consider suppressing features, sketch relations, end conditions, or sketch planes. ∎
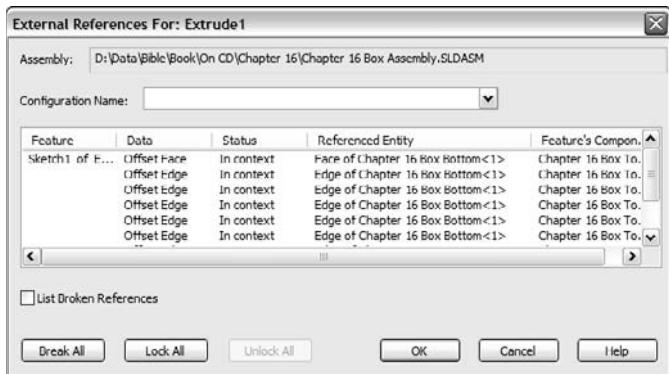
List External References

You can access the locked and broken references through the List External References option on the RMB menu of any feature with an external reference symbol. Figure 16.7 shows the name and path of the assembly where the external reference was created, as well as the part names and entity types.

The External References dialog box

No External References
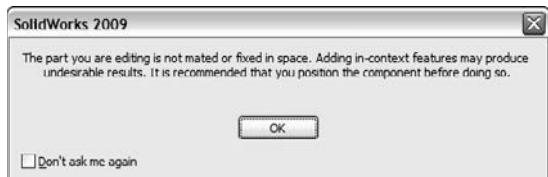
To access the No External References button on the Assembly toolbar, choose Tools ➪ Options ➪ External References ➪ Do Not Create References External To The Model from the menus. As its name suggests, this setting prevents external relations from being created between parts in an assembly. When you offset in-context edges or use Convert Entities, the resulting sketch entities are created without relations of any type.

This lack of references includes the InPlace mate, which is not created when a part is created in-context. As a result, when you add the part to the assembly, if you exit and later re-enter Edit Part mode, SolidWorks reminds you that the part is not fixed in space by displaying the warning shown in Figure 16.8.

This message should remind you that in-context features should be used only on parts that are fully positioned in the assembly.

The dialog box that warns you about adding in-context relations to an underdefined part
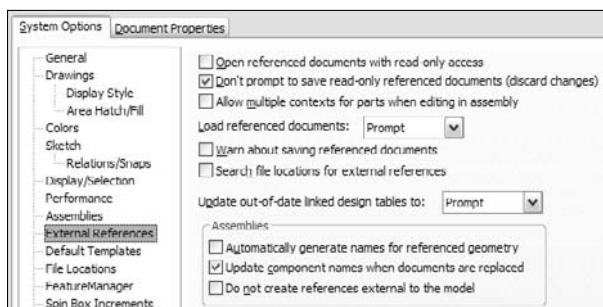
External reference settings in Tools ⇨ Options

The Tools ⇨ Options ⇨ External References pane of settings controls many aspects of the behavior of external references. I discussed one of these references earlier — No External References — and I discuss the other reference, Multiple Contexts, next. This page in the Tools ⇨ Options dialog box is shown in Figure 16.9.

The Tools ⇨ Options ⇨ External References pane

In-context best practice suggestions

However, this is a technique that requires a fair amount of discipline, restraint, foresight, and judgment. The potential problems associated with overuse or misuse of in-context techniques primarily include performance problems (speed) and lost references due to file management issues. Users may also experience problems with features or sketches that change with each rebuild. The following section contains best practice suggestions that can help you avoid these situations.
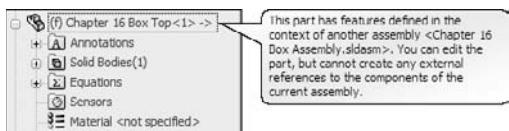
Multiple contexts

Multiple contexts occur when a part has references that are created in multiple assemblies. By default, multiple contexts are prevented from happening. If you place a part that already has external references into a different assembly, a warning appears, as shown in Figure 16.10.

FIGURE 16.10

The warning message that appears about multiple contexts

Although SolidWorks displays many warnings about multiple contexts, you may still run into situations where you need to use them. For example, you may have a subassembly where a part, such as a top plate of a stand, has in-context references to locate a set of mounting holes for legs of the stand. When you place the subassembly into the top-level assembly and mount another assembly to the top plate, another set of in-context holes is required in the top plate.

Figure 16.11, at the top, shows the first table and points out the in-context relations. At the bottom, the large bracket appears for the machine that is mounted to the table top using more in-context relations. The External References dialog boxes for the two different in-context features appear in Figure 16.12. Notice that the Assembly fields at the top of the External References dialog boxes are different. You can only achieve this by selecting the Allow multiple contexts for parts when editing in assembly option shown in Figure 16.9.

Note

The Tools ⇨ Options setting for multiple contexts is a system option. This means that this option is either on or off for every document on a single machine, but when the assembly is used on another machine, the option may be off. ∎

Multiple context modeling is something that should be the exception rather than standard practice. If you do not have all the assemblies open where the in-context references were created, then you will have some out-of-context references. This can make for a troubleshooting nightmare if someone ever has to try to reconstruct how the assembly is driven.
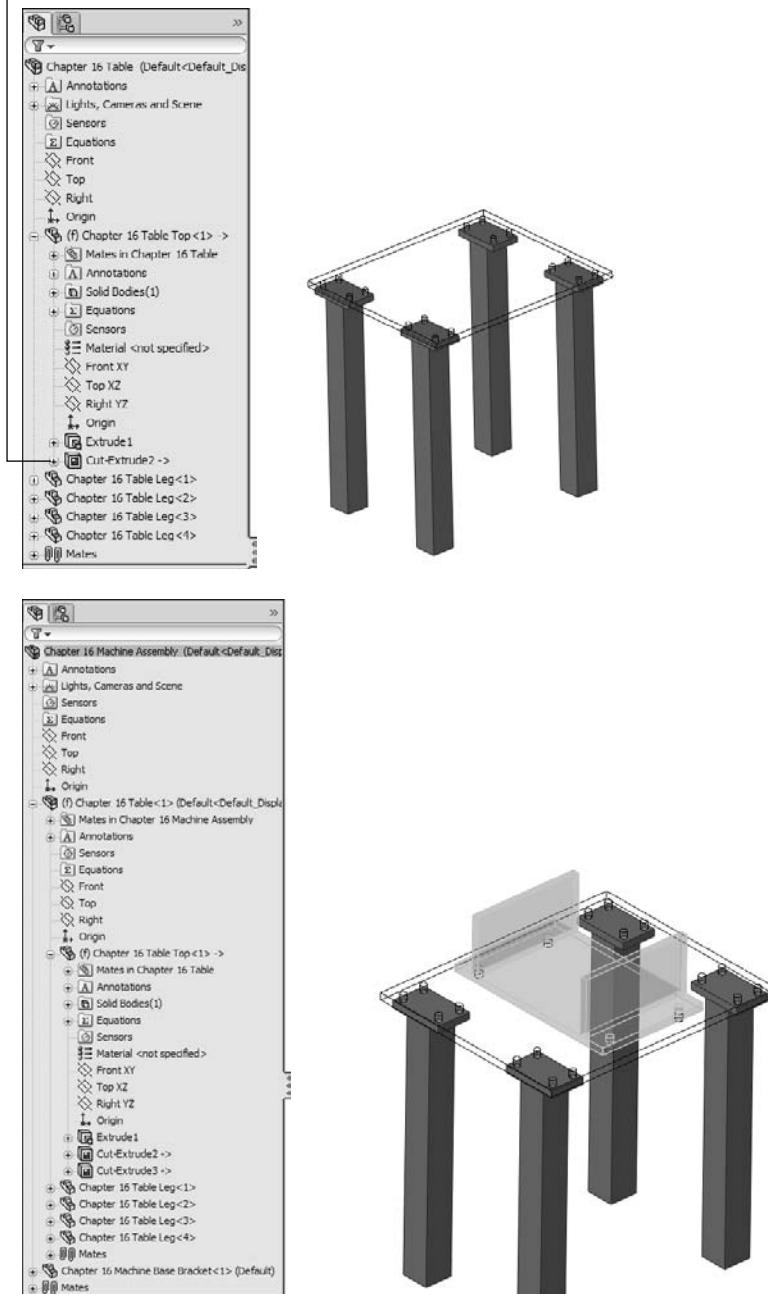
Best Practice

The best practice is to avoid creating multiple-context references. If you need to do this, then be very careful about naming files, and remember to turn off the multiple-context option when you have finished creating the reference. ∎

FIGURE 16.11

Using multiple contexts
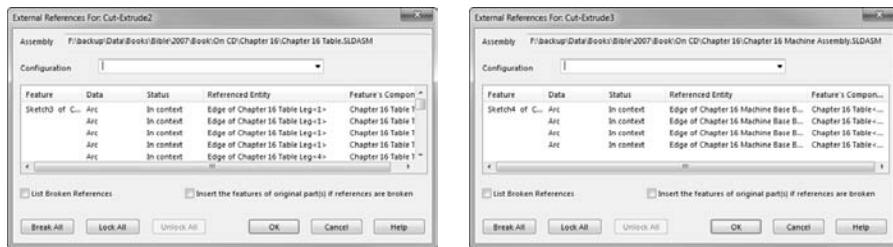
In-context feature

FIGURE 16.12

External References dialog boxes

If you receive a multiple-context part from someone else, the best thing to do is to determine whether you have all the files required to make it work. Right-click the external reference symbol and select Edit In Context to determine whether SolidWorks can find the right files. Also looking for an out-of-context symbol will tell you if any of the necessary files are not currently open.

Aside from doing some programming, the only way to find out whether a part was created as a multiple-context part is to examine the External References list for each in-context feature. This can be very time-consuming. Although multiple-context parts should be very rare, it is impossible to determine ahead of time whether or not a part that you have received is a multiple-context part, at least without programming. The one exception to this is when some features are in-context and some are out-of-context.

In-context with configurations
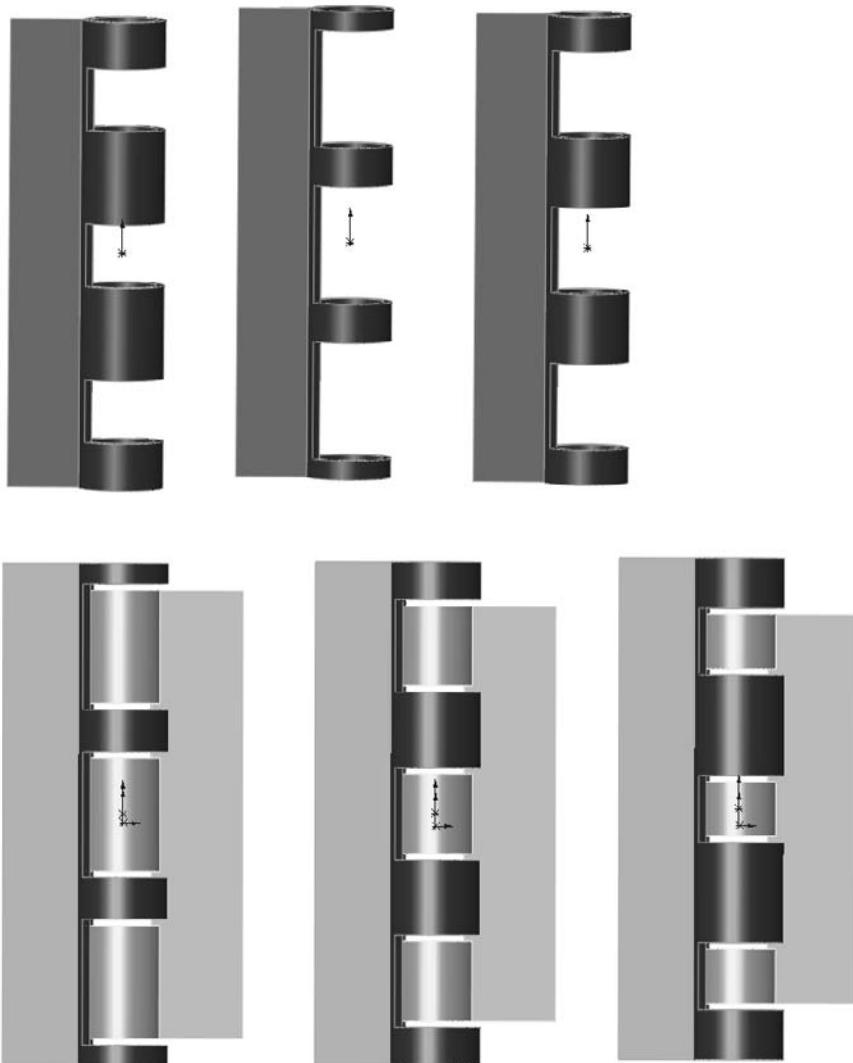
On the surface, mixing in-context references with configurations sounds like it is combining two powerful techniques that should offer you great control over models. Although this may sometimes be true, I want to caution you about some of the effects that combining these two techniques may cause. In particular, you should be careful about part configurations, particularly configurations of the *referenced* part.

If you are using in-context relations to parts with configurations, then you may want to consider a few things. First, look at the door-hinge part shown in Figure 16.13. At the top are three configurations of one of the hinge plates. The second hinge plate is built in the context of the assembly so that it will always match the first plate. At the bottom are the results of changing the first hinge-plate part configuration in the assembly. This looks like an ideal situation because the second hinge plate always changes to match the first hinge plate. What could be wrong with this?

The problem here is that you can only show the size of the second hinge plate that corresponds to the configuration of the first plate that is active in the assembly. If you had two instances of the hinge assembly in a top-level assembly, then you would be able to show only one size for the second plate.

FIGURE 16.13

Combining in-context references with configurations

A second situation where combining in-context references and configurations can cause you trouble is if you have referenced the edges of a part from another part, and a configuration of the referenced part either adds or removes fillets or chamfers, thus breaking the edges. Either of these situations can cause the in-context sketches or other features to fail. This may be a reason to reference the underlying sketches, rather than the model edges or faces themselves.

In some situations, configurations work well with in-context relations. One example of this would be when an assembly has many configurations used for positioning parts. In this case, use one configuration for the sole purpose of creating in-context relations.

In-context with motion

You should make in-context references between parts where there is no relative motion. The parts themselves can move relative to the rest of the assembly, but they should remain stationary relative to one another. The parts should also be fully defined to ensure that they will not move; you should not simply count on avoiding dragging underdefined parts.

In some cases, such as an assembly of imported parts, it may make sense to fix parts in bulk rather than to mate them. When you are using in-context relations, you need to take extra care to ensure that the parts do not move around. When parts move around, in-context features also move.

Obviously, if the motion is around a circular hole and the in-context feature is circular and is not affected by the rotation of the referenced part, then it makes less difference; however, if there is a keyway, that may change things. You need to pay attention when combining underdefined parts and in-context features.

Best Practice

For best practice, you should avoid in-context relations between parts when relative motion is allowed between these parts. ∎

In-context with multiple instances

Another situation that can cause problems is when multiple instances of an in-context part are being used in the assembly. In cases like this, you need to be careful and consistent, by always using the same instance to create the in-context relations. You can do this by putting parts into folders, or by giving the in-context part a special component color.

One trick used by some people is to use one instance of an in-context part for the in-context relation, and a second instance of the part to allow motion. In-context relations are tied to one specific instance of a part, regardless of how many of those parts are in the assembly. You might want to set the driving in-context part aside by putting it in a folder, changing its color, or hiding it.

In-context and file management

Understanding what you are doing with file management is imperative when working with parts that depend on in-context features. Because the references are stored in both the part that is doing the referencing and the assembly where the reference is created, improperly changing the name of either document or even the referenced document is bound to cause problems. For example, if you rename an in-context part using Windows Explorer, then the assembly will not recognize the part, as I demonstrated in an earlier chapter. This also means that any in-context references will not update. The part will show the out-of-context symbol.

Best Practice

For best practice, you should use either the SolidWorks Save As command or SolidWorks Explorer to rename parts and assemblies. This applies to all parts and assemblies, but doubly to in-context documents. ■

In-context and mates

I mentioned this earlier, but a section on in-context best practices would not be complete without issuing the warning against mating to in-context features. Mating parts to in-context features creates a parametric daisy chain, thus establishing an order in which assembly features and mates must be solved. This always creates performance problems in assemblies, especially large ones. The SolidWorks AssemblyXpert looks for this condition when examining assemblies.

Circular references

Circular references in assemblies are a bigger problem than most people realize. In fact, most people do not realize that circular references *are* a problem, or, for that matter, that they even exist.

A circular reference takes the form of "Part A references Part B, which references Part A." It creates a circular loop that really wrecks assembly rebuild times. Part feature design trees are not susceptible to this sort of looping because the part FeatureManager operates in a linear fashion (at least when it comes to applying relations between sketches or features).

The Assembly FeatureManager is solved in this order, or an order that is very similar:

1. Solve reference geometry and sketches that are listed before parts in order, at the top of the design tree.

2. Rebuild individual parts as necessary.

3. Solve the mates and locate the parts.

4. Solve in-context features in parts.

5. Solve reference geometry and sketches listed after the mates.

6. Solve assembly features and component patterns.

7. Loop to Step 3 to solve mates that are connected to anything that was solved after the first round on the mates.

8. Continue to loop until complete.

As you can see, even if you do not have a reference such as "Part A references Part B, which references Part A," it is still possible to get a highly convoluted, if not entirely circular, loop. Many users with smaller assemblies in the hundreds of parts complain about very poor performance.

Skeletons and layouts

When you are making in-context references, a technique that can help you avoid circular references is to always create references to parts that are higher in the design tree. You can expand on this idea until a single entity is at the top of the design tree, to which all in-context references are

made. This could take the form of a layout sketch, or a skeleton. These concepts are discussed in Chapters 11, 12, and 13. I discuss the Layout feature, which is different from the layout sketch, later in this chapter as an additional in-context tool.

Remember that the layout sketch consists of a single or even multiple sketches that control the overall layout of the assembly, as well as all the relationships between parts. When you refer all the relations to a single entity that does not change with part configurations, or lose or gain filleted edges, the intra-part parametrics become much stronger and more stable.

When you are building a mold for plastic injection molding, a single sketch can control the size and position of the plates, pins, and so on. If all the 3D parts are mated to the 2D sketch, or use the 2D sketch by converted entities, then the parts will move with the sketch. This same technique is important and useful for any type of die or punch design, along with many other types of design.

In-context and libraries

Library parts should never contain in-context references, especially if the in-context references are out of context. Small library assemblies may have in-context references between the parts, but a single part should not have features created in-context. External references may be unavoidable in the form of mirrored or inserted parts, but in-context references are completely avoidable.
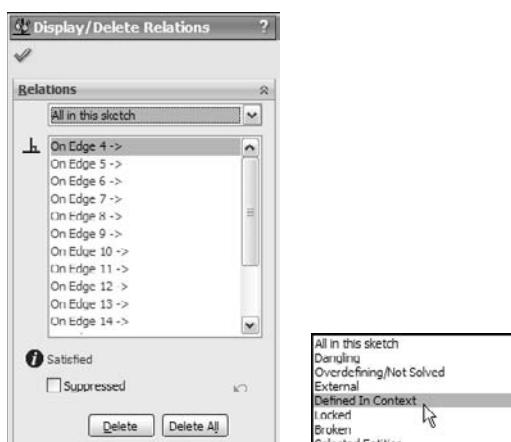
Removing relations

The correct way to remove in-context sketch relations is by using the Display/Delete Relations tool. You can sort the relations by selecting the Defined in Context option, as shown in Figure 16.14.

FIGURE 16.14

Sorting sketch relations by type

If you are considering using the Break Relations tool, then you should either reconsider and use Lock Relations instead or simply remove all the in-context relations altogether.

Other types of in-context references are not as easy to remove as sketch relations. When you see the External Reference symbol on a sketch, it could be the sketch relations or it could be the sketch plane that was in-context. In order to remove the reference from an in-context sketch plane, you must redefine the plane locally in the part.

You should also not forget end conditions such as Up To Surface, Offset From Surface, or even From Surface. If an external reference symbol remains on a feature, you can use the Parent/Child option on the RMB menu to locate it. Remember that using an edge or vertex for a plane definition can cause an in-context relation.

Should you use mating or in-context?

In-context is initially so fast and easy to use that it can be addictive, but you need to think before you use it because of the speed and file management implications these relations will have later on in your design process.

Communicating design intent

If someone else needs to use your model after you are done with it and possibly edit it, then you should leave some clues to help this person understand how the model works, and how it is best changed. For example, you can use descriptive feature and sketch names, comments that are associated with features, the Design Binder to add documentation, and the Design Journal to write notes. You can even put HTML (Hypertext Markup Language) links in notes that display in the graphics window.

In-context design intent may not always be obvious, and an impatient user may find it more expedient to delete the in-context references and replace them with either local relations or no relations at all. The more you document your intent, the more likely others will be to follow it.

Using Other Types of External References

The external reference symbol (- >) indicates in-context features that have been created in the context of an assembly, but it also indicates three other types of external references —Inserted parts, split parts, and mirrored parts.

Understanding inserted parts

I discuss inserted parts to some extent in Chapter 10, and in more detail in Chapter 28. In the past, inserted parts have also been called base parts and derived parts, and some users still use those names.

An inserted part is simply an entire part that has been inserted into another part. This is sometimes referred to as a *pull* operation because the data is pulled from the original part into the child part. The part may be inserted at any point in the history of the design tree, and it may create an additional body within the part or be added to the existing one. Additional features can also be added to the inserted part.

Items that can be brought along with the inserted part are solid bodies, surface bodies, planes, axes, sketches, cosmetic threads, and even features. You can also use a particular configuration of the inserted part in the child part. I discuss this aspect in Chapter 14, dealing with configurations, and also in Chapter 28, dealing with master models.

You can use inserted parts for many modeling applications, such as cast parts and secondary operations. You first insert the original cast part into a new blank part. Then you add cut and hole features until the part resembles the finished part.

Another application for inserted parts is a single part that has been built from several models. For example, I once worked on a large, rather complicated plastic basket, where the basket was modeled as three individual parts, and then reassembled into a single part. Another application may be to insert a part as a body into a mold block to create a mold cavity. To insert a part into another part, you can choose Insert ⇨ Part.

Understanding split parts

I discuss split parts in detail in Chapter 28, in the section about master models. Inserted and split parts are both master model techniques, as are a few more techniques that I discuss in Chapter 28. Some people also include in-context techniques with the master model tools because this is a way of making several parts update together.

Split parts are sometimes called a *push* operation because the data is pushed from the original multi-body part to the individual child parts. The *split* function takes a single body and splits it into several bodies, optionally saving the bodies out as individual parts. This is done for various reasons, such as creating a single, smooth shape out of several different parts; for example, automobile body panels or the various covers and buttons on a computer mouse. You can use the split parts technique for other applications, as well. Sometimes a product is designed as a simple, single solid to keep the modeling simple, and because it is not known how the parts will be assembled or manufactured. When the manufacturing decisions are made, the part can be split into several models that have the engineering details added to them.

Understanding mirror parts

You can mirror a right-handed part to create a left-handed part. To activate the Mirror Part command, you must select a plane or planar face. Then choose Insert ⇨ Mirror Part to initiate the Mirror Part command.

Mirror parts can also use configurations, and so if you have one of those "mirrored exactly except for . . ." parts, you can select the configuration of the parent from the child document.

Using the Layout feature

I described the Layout feature briefly in Chapter 12, where I also described layout sketches. This is an unfortunate naming conflict. Two highly useful functions do nearly the same thing, one of which is simply a technique that has existed for years, and the other, a newly added formal feature. For this reason, I will always capitalize the name of the new Layout feature and refer to it as a feature, while I'll refer to the layout sketch in lowercase and as either a sketch or a technique.

The Layout feature is simply a 3D sketch that is given special treatment within an assembly. It works best with sketch blocks. To initiate a Layout, click the Layout button on the Layout tab of the assembly CommandManager or activate it from the Insert menu. Once you are in a Layout, SolidWorks puts you into a 3D sketch with the Front (XY) plane activated, so it displays a small grid.

Cross-Reference
3D sketches are addressed in Chapter 31. ■

For now, you primarily treat the 3D sketch as much like a set of 2D sketches as possible. The main difference is that you can double-click on a different plane to start sketching on the new plane, and you will always see this small grid when a plane is active.

3D sketches have some limitations when you are working with Layouts, such as lacking the capabilities to use sketch patterns and Sketch Pictures.

Using the Layout workflow
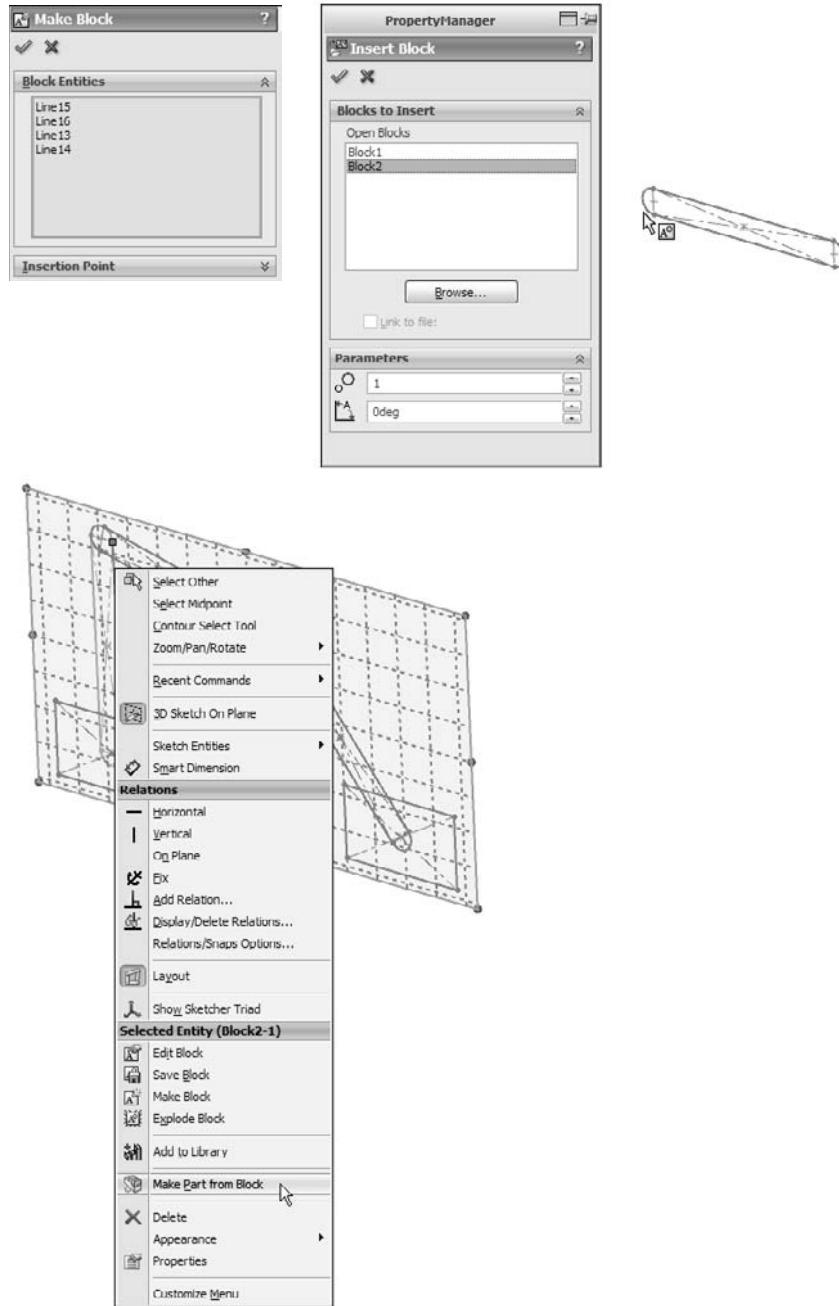
Here is the general workflow for working with the Layout feature:

1. **Open a new or existing assembly.**
2. **Click the Layout toolbar button on the Layout tab of the CommandManager.**
3. **Sketch on the plane in the 3D sketch to create 2D sketches representing parts of a mechanism or other assembly.**
4. **Make selections of the sketch into blocks representing individual parts.**
5. **Insert multiple instances of the blocks to represent multiple instances of the parts**.
6. **Use sketch relations to put the blocks together like mating parts in an assembly.**
7. **Test the mechanism by dragging sketches.** (Blocks function like a single sketch entity, so you can drag them within the sketch like parts in an assembly.)
8. **Right-click the block (from inside or outside the Layout) and select Make Part From Block (also a button on the Layout toolbar), as shown in Figure 16.15.**

FIGURE 16.15

Tools you encounter when using Layout

Note

The icons for Make Part From Block differ slightly between large and small icon sizes. The icon shown previously is a large icon. ■

Virtual components

Virtual components always exist with in-context workflows, and frequently with the Layout workflow. Virtual components are parts that are saved within the assembly. You can save a virtual component externally, and you can also make an externally saved part into a virtual component.

The advantages of virtual components are that you don't have to worry about saving out additional files, plus the assembly will never lose track of any virtual component.

Virtual components are primarily intended to be used as quick, temporary, conceptual tools, rather than as a way to make parts that will be a permanent part of the assembly. I have also seen some SolidWorks users use virtual components to represent non-geometric parts such as glue or paint. Any time you choose Insert ➪ Component ➪ New Part from the menus and select a template and a plane to put the part on, the part is placed immediately into the assembly, and you can start working without worrying about having to save the assembly and the part. This saves a lot of time initially. Later on, when you save the assembly, SolidWorks prompts you to save the parts externally as well, or you may choose to leave the parts internal to the assembly.

Virtual components are named Part1^Assem1, where Part1 and Assem1 are default names. You can easily rename the part by clicking the RMB menu and selecting Rename Part. You cannot do this for external parts. If you make an external part virtual, the name in the assembly becomes Copy of *file name*^Assem1 where *file name* was the name of the external file. The name of the assembly is always included (and cannot be removed) to ensure that if you have subassemblies that also have virtual components, you will always have unique filenames for all the parts.

Virtual components can also be accessed in their own window, which makes them easier to edit for some purposes. Bills of Materials (BOMs) and numbered balloons work correctly with virtual components.

Best Practice

It is considered best practice to save any parts that will be a permanent part of the assembly as external files. Virtual components should be limited to temporary parts or possibly non-geometry, BOM-only parts like glue or paint. ■

Balancing advantages and limitations

In theory, the Layout feature has several advantages:

- You can make parts from blocks within the Layout.
- You can move parts by moving blocks in the Layout.

- It is a great way to structure your relations within an assembly.
- It is useful for motion analysis studies.

In practice, this feature needs some enhancements before it is ready for use on real assemblies. While attempting to create a Layout tutorial for this chapter, I came across several limitations that would make the tutorial unmanageable, and I think you will agree that using 2D sketches as assembly layout sketches is still a better idea than trying to avoid the limitations of the formal Layout feature. The limitations listed are presumably not bugs because Layout was introduced in 2009 and this testing was done on 2010 SP 0.0. Here are some of the limitations of Layout:

- The 3D sketch used for Layout has all the limitations that come with 3D sketches.
- Sketch relations are listed in the Mates folder.
- Gaining access to edit the Layout once it has been closed requires a method you don't expect from a sketch: you click the Layout button on the toolbar rather than right-click and edit an icon in the FeatureManager.
- It requires that you use blocks to get all of the functionality.
- A fully defined 3D sketch with blocks is very unstable.
- Part creation from blocks does not offer a time savings.
- You cannot paste copied sketch entities from a 2D sketch into the Layout.
- You cannot use autodimension (or polygons or ellipses) in the Layout.

Although the formal Layout feature has serious advantages over regular layout sketches, at this time the limitations outweigh them. The rest of my discussion on layouts addresses the generic layout technique rather than the formal feature.

Tutorial: Working In-Context
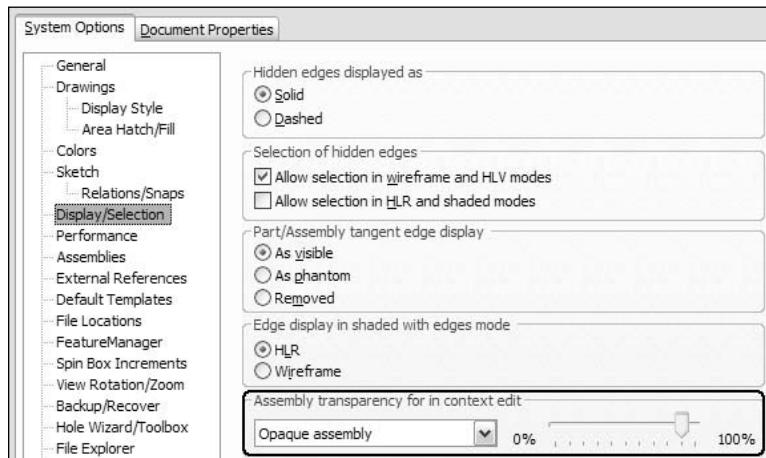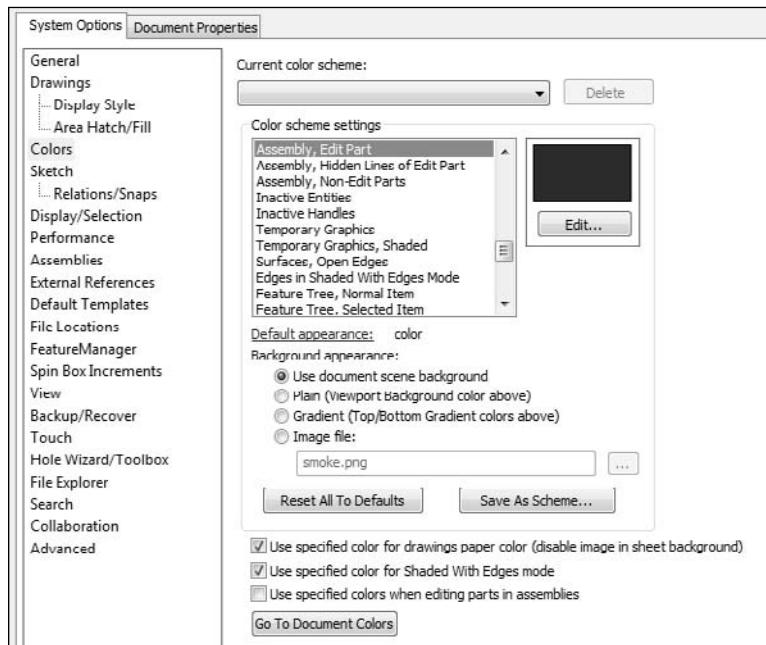
Follow these steps to get a feel for the workflow of working with parts in the context of an assembly:

1. **Open the assembly from the CD-ROM named** `Chapter 16 Tutorial Table. sldasm`.

2. **Set the colors that are to be used during in-context editing.** Remember that two settings control this — one at Tools ➪ Options ➪ Colors, and the other at Tools ➪ Options ➪ Display/Selection — as shown in Figure 16.16.
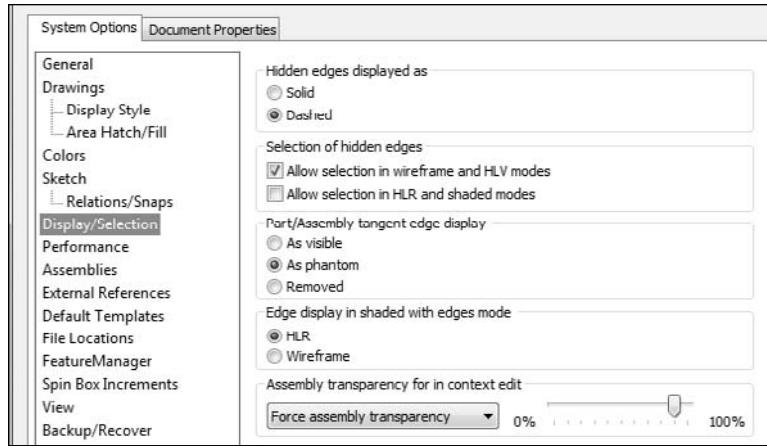
FIGURE 16.16

Setting in-context colors

Set the Assembly Edit Part color to a shade of blue, and the Assembly Non-Edit Parts to a shade of gray.
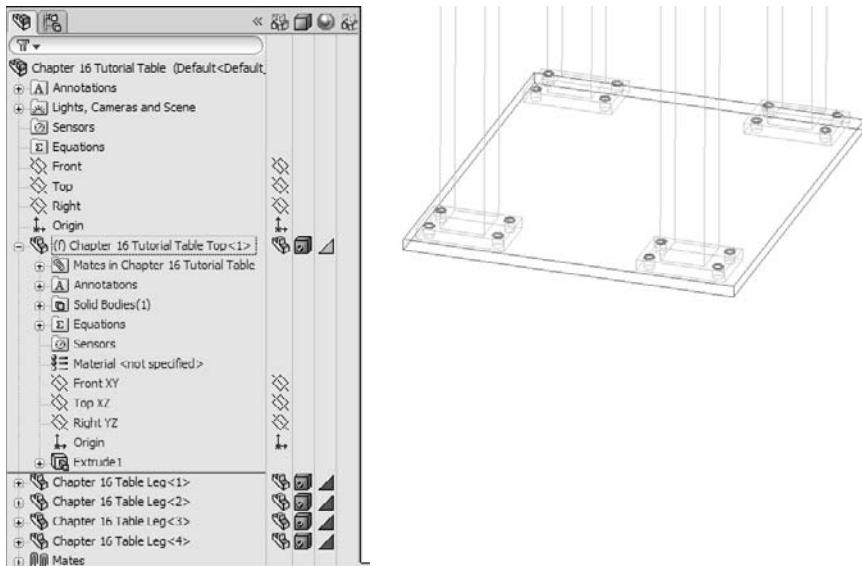
Also set the Assembly transparency for in context edit setting to Force Assembly Transparency, with the slider at around 90 percent.

3. **Now you are ready to begin.** Select the Table Top part, and click the Edit Component button on the Assembly toolbar. This command is also available through both the RMB menu and the drop-down menu as Edit Part. (If you right-click a subassembly, the Edit Subassembly option becomes available.) Notice that the Table Top part and the FeatureManager text turn the same color.

4. **Expand the Table Top part in the Assembly FeatureManager, select the Front plane, and open a new sketch on it.** Notice that you cannot select the edges of the transparent parts through the transparency, even if the Select Through Transparency option is selected (Tools ➪ Options ➪ Display/Selection). This setting applies only to faces, not to edges. Instead, change the display mode for the entire assembly to Wireframe.

5. **Now select the 16 hole edges on the legs.** It does not matter whether you select the top edges or the bottom, or even a combination of top and bottom. Use the Convert Entities command to project the edges into the sketch plane as circles, as shown in Figure 16.17.

6. **Create a cut that goes Through All.** You may have to change the direction of the extrude to get it to work. Exit Edit Component mode using the confirmation corner and save the tutorial assembly.

7. **Now open the file named** `Chapter 16 Tutorial Machine Assembly.sldasm`. Notice that the Table Top part in this assembly is using the Wireframe display state, which is assigned in the Display pane.

8. **Right-click the part and select Edit Part from the list, or select the part and click the Edit Component button on the toolbar.** A warning will display that the part has features that were created in the context of another assembly. You can edit the part, but you cannot add any more external references (in-context features) to it.

9. **Toggle off the Edit Component button on the Assembly toolbar to exit Edit Part mode.**

10. **Choose Tools ⇨ Options ⇨ External References and select the Allow Multiple Contexts for Parts When Editing in Assembly option.** Now try to edit the Table Top part again in the context of the assembly. This time, no warning message displays.

FIGURE 16.17

Converting entities in-context
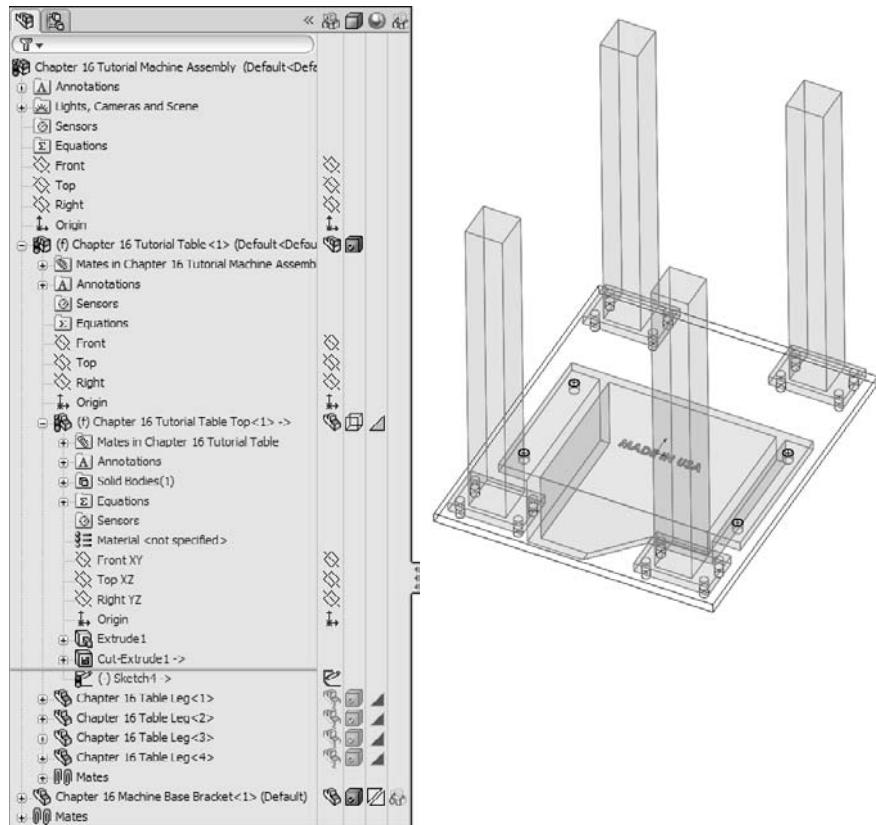
11. **Make sure that you are editing the Table Top part.** It will not change colors as specified in the Tools ⇨ Options ⇨ Colors settings because it is using the Wireframe display mode. Ensure that the status bar in the lower-right corner displays *Editing Part* rather than *Editing Assembly*.

12. **Open a sketch on the Front plane, and convert the four edges of the holes, as shown in Figure 16.18.**
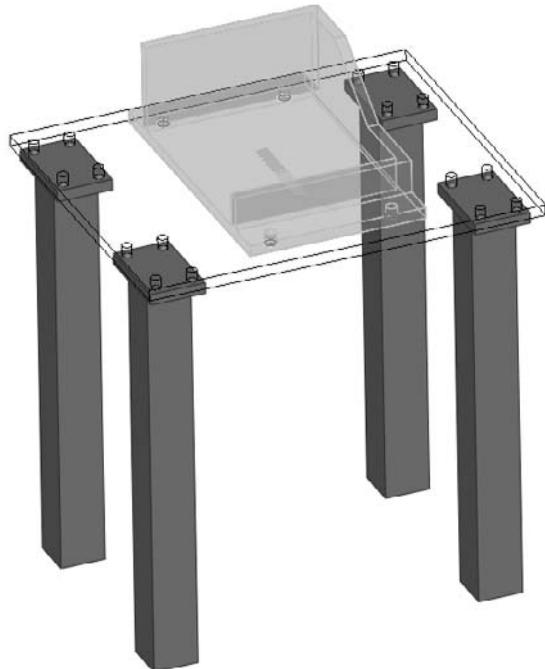
FIGURE 16.18

Creating holes in-context

13. **Cut the holes using the Through All setting.** Again, be aware of the direction of the cuts. Toggle out of Edit Component mode and press Ctrl+S to save the assembly. Figure 16.19 shows the finished assembly.

14. **Open the Machine Base Bracket part in its own window by selecting Open Part from the RMB menu.** The part is shown in Figure 16.20.

15. **Select the Front plane and choose Insert ⇨ Mirror Part.** This creates a new part and opens a new PropertyManager interface, as shown in Figure 16.21.

    In this case, select Solid bodies and click OK.

FIGURE 16.19

The assembly as of Step 13

Note

Notice that I used the Insert⇨Mirror Part command, but the PropertyManager says Insert Part. The Mirror Part functionality uses the Insert Part function, but adds a feature to mirror the body once it is inserted. Notice all the entity types you can transfer, and the fact that you can break the link to the original part. Also note that the template used for this part was chosen based on the settings at Tools⇨Options⇨Default Templates⇨ Always Use These Default Document Templates or Prompt User To Select Document Template. ∎

Cross-Reference
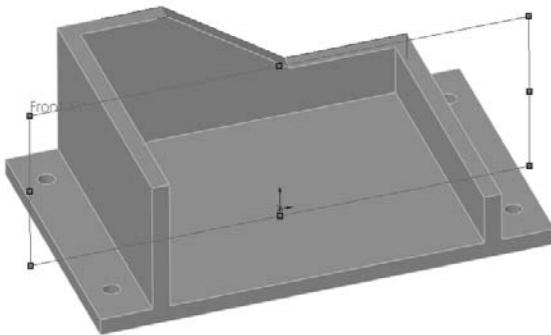
I discuss the function of Mirror/Insert part in more depth in Chapter 26. ∎
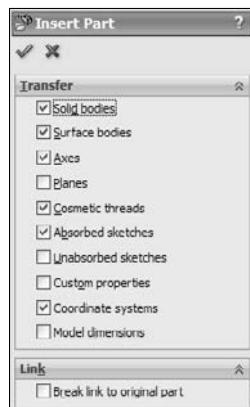
FIGURE 16.20

The Machine Base Bracket part, ready for mirroring

FIGURE 16.21
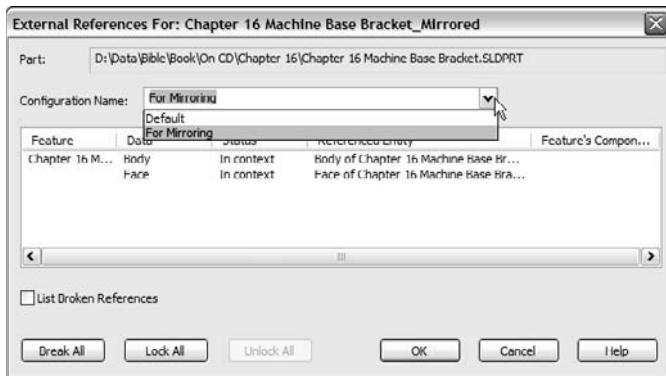
The Mirror Part PropertyManager

16. **Notice that the new part is indeed a mirrored copy of the original.** You can see that the "MADE IN USA" text on the bottom is backwards. Fortunately, a configuration exists specifically for this purpose. Change the configuration by selecting For Mirroring in the Configuration Name drop-down list in the External References dialog box (from the RMB selection List External References), as shown in Figure 16.22. Notice that this configuration removes the extruded text from the model.

17. **Add your own "MADE IN . . ." extruded text to the bottom of the part.** Save the part.

Selecting a configuration

Tutorial: Working with a Layout

In this tutorial, you will use regular assembly sketches to lay out and build a tooling die.

1. **Open the assembly from the CD-ROM named** `Chapter 16 tutorial layout start.sldasm`**.** Notice that three layout sketches and some of the parts have been added already. The existing parts are virtual components, saved inside the assembly.

2. **Click Add New Part on the Assembly tab of the Command Manager.** The cursor appears with a green check mark, and in the lower left corner, the Task Bar prompts you to select a plane on which to place the Front plane of the part. A sketch will automatically be opened on that plane. Click the Front plane of the assembly.

3. **Click the Corner Rectangle sketch tool from the Sketch toolbar.** Create a rectangle from the two corners indicated in Figure 16.23. It may be helpful to bring the model into a Front view before drawing these rectangles.

4. **Extrude the rectangle using the Up To Vertex end condition for both Direction 1 and Direction 2.** Select sketch endpoints in the Plane Depth Layout sketch for both directions so that the new plate matches the other existing plates.

    Be careful not to click on model faces, edges, or vertices when creating these depth references. Make sure that all your references stay in the sketch.
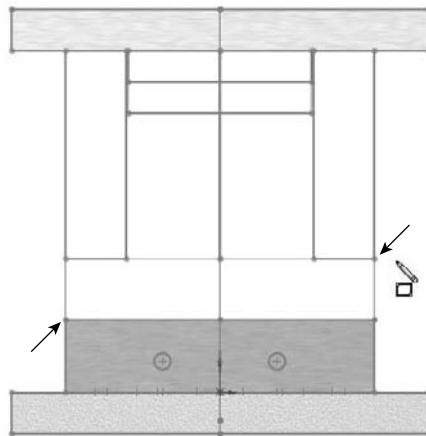
5. **Click the Exit Edit Part icon in the ConfirmationCorner, the upper-right corner of the SolidWorks graphics window.** Right-click on the new part in the FeatureManager and select Rename Part. Rename it **Plate4**. You can also use the Windows standard method of slow double-clicking to rename parts. The ^*Chapter 16 tutorial layout start* part of the name is automatically added. Assign a material from the Appearances tab for the new plate.
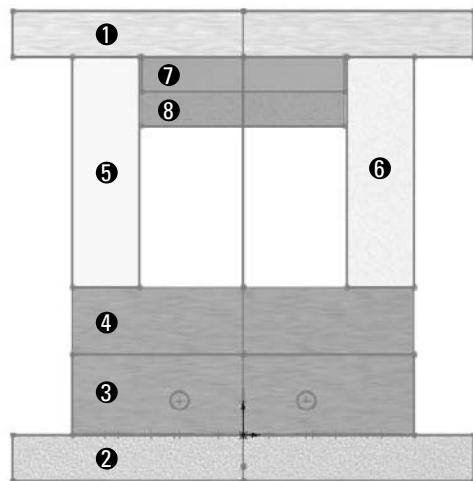
Creating a new plate in the context of an assembly with layout sketches.

6. **Follow this procedure for four remaining plates, as shown in Figure 16.24.** To summarize the steps again, they are add new part, create sketch, extrude block.

All of the plates controlled by the layout sketches
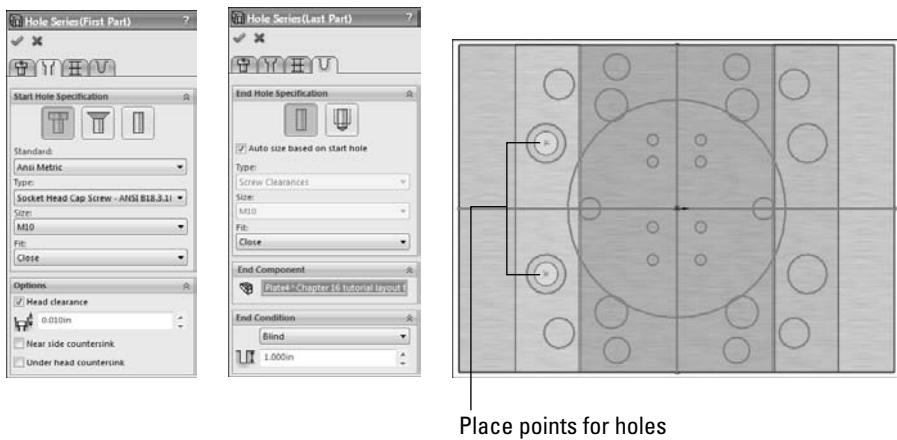
Note

Be careful with the plates labeled 5, 6, 7, and 8. There is a clearance gap between the sides of the 7 and 8 plates and the vertical plates 5 and 6. ∎

7. **Reorient the view to the Top view. Make sure you can see the Pin Layout sketch.**

8. **Select the top face of Plate 1, and click on the Hole Series toolbar.** The Hole Series may be hidden under the Assembly Features flyout on the Assembly tab of the CommandManager. The Hole Wizard no longer requires pre-selection to avoid 3D placement sketches, but the Hole Series as of 2010 has not been updated to follow the same rules as the Hole Wizard.

9. **Place two sketch points at the centers of the circles, as shown in Figure 16.25.** Make sure that the two points are both over the same plate 5 or 6. You cannot cut both plate 5 and 6 in the same Hole Series feature. Use the settings shown in Figure 16.25.

FIGURE 16.25

Placing screw holes through multiple parts in the die

Place points for holes
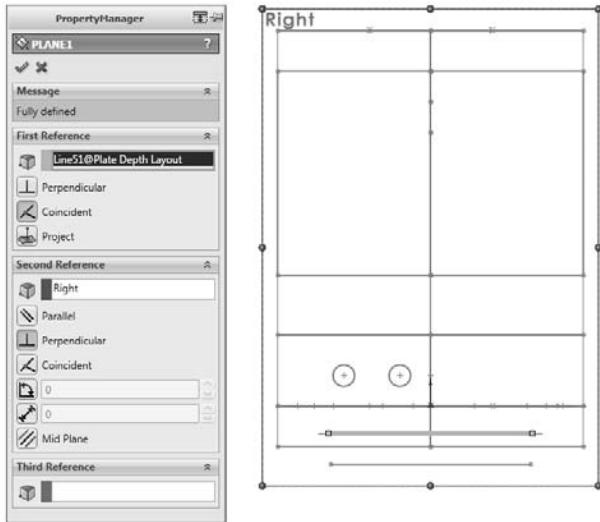
Make sure the holes are Counterbored, ANSI Metric, Socket Head Cap Screw, M10, with a head clearance of 0.10 inch. All other conditions should follow Figure 16.25.

10. **Place two more new holes on the other side.** Make the parts transparent to see how the holes have been placed.

11. **Initiate the Plane feature. Create a plane perpendicular to the Right plane and coincident to the line from the Plate Depth Layout shown in the Right view in Figure 16.26.** Rename the new plane Sprue Bushing Seat. (The new plane should show up right underneath the initial three layout sketches.)

FIGURE 16.26
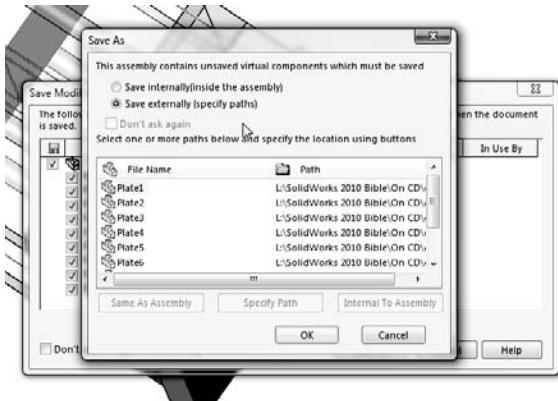
Creating a plane in the assembly driven by the layout sketch

12. **Create a new part on the Sprue Bushing Seat plane.**

13. **Activate the Convert Entities sketch tool, select the large (approximately 4-inch diameter) circle from the Pin Layout sketch, and convert it into the sketch plane of the new part.**

14. **Draw a 1/2 inch circle in the center (at the origin).** Extrude the sketch 0.875 inch so that it protrudes from Plate 2.

15. **Exit the Edit Part mode (using the ConfirmationCorner).** Rename the new part Sprue Bushing.

16. **Left-click on Plate2 and select Edit Part from the shortcut toolbar.**

17. **Open a new sketch on the Sprue Bushing Seat plane (which is part of the assembly, not part of the part).**

18. **Select the large circle used in Step 13, and use the Offset sketch tool to offset the circle 0.005 inch to the outside.** Create a Through All cut that comes out the exposed side of Plate2, clearing an area for the Sprue Bushing. You may need to switch to Wireframe display to accomplish this.

19. **Apply a chamfer to the outer edge of the new cut, 0.010 inch.**

20. **Exit Edit Part mode, and save the assembly to a new location by choosing File ⇨ Save As**. Click the Save All button, and then select the Save Externally option, as shown in Figure 16.27.
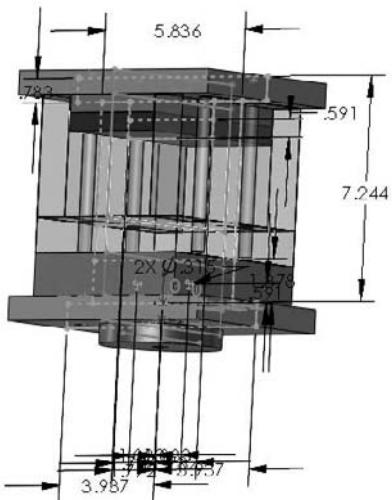
Saving the internal virtual components to external parts

21. **Exit Edit Part mode using the icon in the ConfirmationCorner.**

22. **Double-click the Plate Layout sketch, and change the 5.836 inch dimension to 6 inches.** Change the 7.244 inch dimension to 7 inches. Make sure the model rebuilds, and watch the individual parts update. Figure 16.28 shows these dimensions for reference.

Editing layout sketch dimensions to drive the size of the individual parts

You will need to do more to finish this tooling die than is covered in this tutorial. The point is to give you some experience using the in-context features that are used with a layout sketch technique.

Summary

Although in-context functions are powerful and seductive, you should use them sparingly. In particular, be careful about file management issues such as renaming parts and assemblies. The best approach is to use SolidWorks Explorer or the Save As command with both the parts and assemblies open.

In-context techniques, including the Layout feature, are the pinnacle of true parametric practice and enable you to take the concepts of design intent and design for change to an entirely new level.

Part IV

Creating and Using Libraries

Part IV contains arguably some of the most challenging material in the entire book. The topics treated here go well beyond basic modeling and into administration of automated libraries. Libraries always take time to set up, but the custom libraries especially can have huge design automation payoffs if done properly. Explore this part with an eye toward the possibilities rather than focusing on roadblocks.

Using Hole Wizard and Toolbox

The Hole Wizard and Toolbox are two applications that go together because they both work from a single database of matching hole and fastener sizes. One of the most useful examples of combining these two applications is the capability to automatically place holes through multiple parts and put appropriately sized screws and hardware into the holes, all in one step. The hole knows which fastener, or stack of fasteners, needs to go together.

Many automatic aspects of Toolbox exist, and the concept behind the combination of Toolbox and the Hole Wizard is compelling due to the level of shared information and automation. This chapter aims to give you the information you need to decide how to implement and use Toolbox in your work.

Using the Hole Wizard

The Hole Wizard enables you to place holes for many types of screws with normal, loose, or close fits. You can create Hole Wizard holes as assembly features in an assembly or as features in individual parts that are built in the context of an assembly using the Series Hole functionality. This tool is called a *wizard* because it guides you through the process step by step. A summary of the process of creating a Hole Wizard hole is as follows:

1. **Pre-select the face to put the holes on, although this is not required.** SolidWorks 2010 no longer requires pre-selection to avoid 3D placement sketches; the Hole Wizard now used 2D sketches by default.

2. **Select the type of hole; for example, counterbored, countersunk, drilled hole, tapped hole, pipe tap, or legacy.**
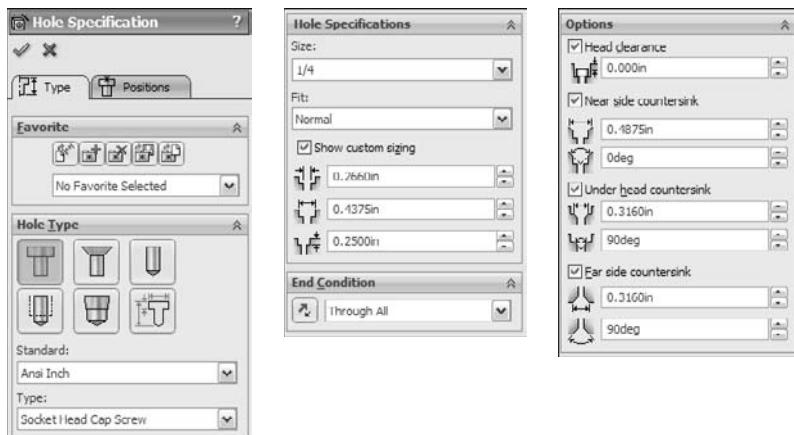
3. **Set the standard to be used, such as ANSI** (American National Standards Institute) **inch, ANSI metric, or ISO** (International Organization for Standardization)**.**

4. **Select the type of screw.** For example, a counterbored hole can accommodate a socket head cap screw or a hex head screw, among others.

5. **Select the size of the screw.**

6. **Select the fit of the screw into the hole, such as normal, loose, or close.**

7. **Select the end condition of the hole.**

8. **Select options for clearance and countersinks or edge breaks.**

   Alternatively, you can use or assign a favorite. A favorite is a hole with settings that you use frequently and want to save. I discuss these later in this chapter.

   You can use Custom Sizing when you need a hole with non-standard dimensions.

9. **Locate the center of the hole or holes.** You can place multiple holes in a single Hole Wizard feature, even on different faces and curved faces. I address the specifics of this step later in this chapter.

10. **Click OK to accept the type, size, and placement of the hole.** Figure 17.1 shows the Hole Wizard PropertyManager interface.

FIGURE 17.1

The Hole Wizard PropertyManager interface

Anatomy of a Hole Wizard hole

Hole Wizard holes are made of two sketches: a center placement sketch and a revolved cut profile. Figure 17.2 shows a simple part with an expanded Hole Wizard feature. Notice that the feature is named for the size and type of the hole.

FIGURE 17.2

A design tree containing a Hole Wizard hole

Placement sketch

The placement sketch is listed first under the Hole Wizard feature. It contains one or more sketch points marking the hole centers. It may also contain construction geometry with relations and dimensions to parametrically locate the hole centers. I discuss placement sketches in more detail in the next section.
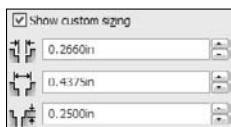
Hole sketch

The revolve profile sketch is not on an identifiable sketch plane that you can reuse for other features, although that would be useful. You can change the sketch dimensions outside of the wizard interface, and if you later use the wizard to edit it, then the changes appear in the Custom Sizing panel. Figure 17.3 shows the Custom Sizing panel with the changed counterbore diameter highlighted.

FIGURE 17.3

The Custom Sizing area of the Hole Specifications panel

If you select any of the choices from the Options panel, the revolved sketch profile is altered to accommodate the change. For example, if you select the check box for a near side countersink, the sketch changes to add a line for the countersink; a separate chamfer feature is not added.

2D versus 3D placement sketches

In SolidWorks 2010, SolidWorks changed the Hole Wizard so that it defaults to a 2D sketch. Pre-selection of a plane or planar face is still recommended because it eliminates a step later on, but it is no longer required in order to avoid being forced into a 3D placement sketch. 3D placement

sketches are still available and still useful, but because they are so much more difficult to use and rarely truly needed, SolidWorks decided to change the default to a 2D sketch.

3D placement sketches are needed for Hole Wizard holes if you have a set of similar holes that are placed on different levels or are always perpendicular to non-planar surfaces. If you need 3D placement sketches, they are still available, but they are no longer the default.

For most people who have learned to pre-select a face before opening the Hole Wizard, this change will have no effect. It will have a positive effect on new users and those who frequently forget to pre-select.

Advantages and limitations of the 2D sketch

The main advantages of the 2D sketch method are the simplicity and completeness of the available tools. Everyone knows how to manage 2D sketches, sketch planes, dimensions, and construction geometry.

A limitation of the 2D sketch is that the holes that you create through this method are limited to a single planar face, and the holes will all be perpendicular to that face. Sometimes this creates a great limitation, while other times it does not matter.

Advantages and limitations of the 3D sketch

The obvious advantage of the 3D placement sketch is that it can put a set of holes on any set of solid faces, regardless of whether they are at different levels, are non-parallel, or are even non-planar. This function offers multiple holes, multiple faces, and multiple directions. In situations where that is what you need, nothing else will do.

A limitation of the 3D sketch is that it can be fairly cumbersome. Dimensions work very differently in 3D sketches compared to 2D sketches. For example, to create and place a hole in a specific position on a cylinder, you need to follow these steps:

1. **Begin with a circle with a diameter of one inch, drawn on the Top plane and extruded using the Mid-plane option one inch.**

2. **Start the Hole Wizard without any pre-selection, either through the Features toolbar or by choosing Insert ➪ Features ➪ Hole ➪ Wizard.**

3. **Set the interface to use an ANSI inch, one-quarter-inch, and counterbored hole for a socket head cap screw.** Use a Normal fit and Through All for the End Condition, with a .100-inch head clearance (in the Options panel) and no custom sizing changes. These settings are shown in Figure 17.4.

4. **Click the Positions tab, which is located at the top of the PropertyManager window.** The interface asks you to select a face where you would like to put the holes or to select the 3D sketch option. In this case, click the 3D Sketch button.
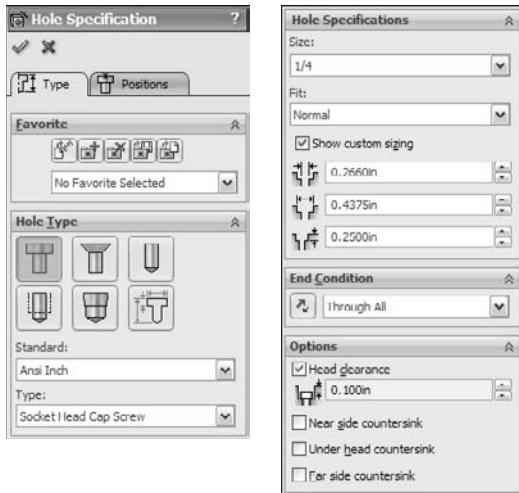
Note

Be careful with clicking when the Point tool is turned on. For example, if you click in a blank space, the Point tool places a point off the part. SolidWorks will try to use the point later to create a hole in empty space, which usually causes an error. ∎
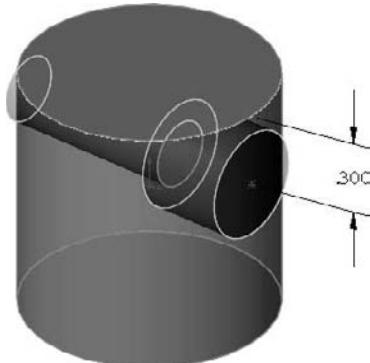
The Hole Wizard settings for the socket head cap screw

5. **Click the cylindrical surface of the part.** The surface appears orange when you move the cursor over it to indicate that an OnSurface sketch relation will be created between the sketch point and the cylindrical surface.

6. **The hole should be positioned from one end of the cylinder.** Using the SmartDimension tool, click one flat end face of the cylinder and the sketch point. Place the dimension and give it a value of .300 inches, as shown in Figure 17.5.

   Locating the point angularly around the cylinder is more difficult. You can use several methods to do this, but this example shows one using construction sketch geometry.

Dimensioning the 3D Placement sketch point

Tip

To force a 3D dimension to have a certain orientation, dimension from a plane or planar face rather than from an edge, vertex, or sketch entity. A dimension from a plane is always measured in a direction perpendicular to the plane, but a dimension from a line or point is always measured by the shortest distance between the entities. Two-dimensional sketches can force dimensions to be horizontal or vertical, but 3D sketches cannot. ■

7. **With the Line tool activated while still in the 3D sketch, Ctrl+click the flat end face that the previous dimension referenced.** This moves the red "space handle" origin to the selected face and constrains any new sketch entities to that face. You are still in the 3D sketch, but are constrained to the selected plane and still must play by all the 3D sketch rules. The elements of 3D sketches are described in detail in Chapter 31.

8. **Select the Temporary Axes view by choosing View⇨Temporary Axes.**

9. **Place the cursor near the center of the activated end face; a small, black circle appears, indicating that the end point of the line will pick up a coincident relation to the temporary axis.** Draw the line so that it picks up an AlongX sketch relation. The cursor shows the relations about to be applied, just like in a 2D sketch.

10. **Draw a second line again from the center, but this time do not pick up any automatic relations. This line should also be on the flat end face.**

Note
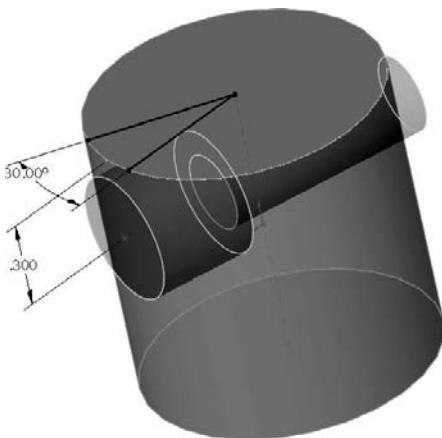
Although you can set these lines to display as construction lines if you like, this is not required for the feature to work; the lines also work as regular solid lines. ■

11. **Put an angle dimension between the lines and change the angle to 30 degrees.** To be thorough (which is always recommended in 3D sketches, which have a tendency to handle underconstrained sketch geometry unpredictably), constrain the ends of the lines to the circular edge of the cylinder. At this point, the part looks like Figure 17.6.

FIGURE 17.6
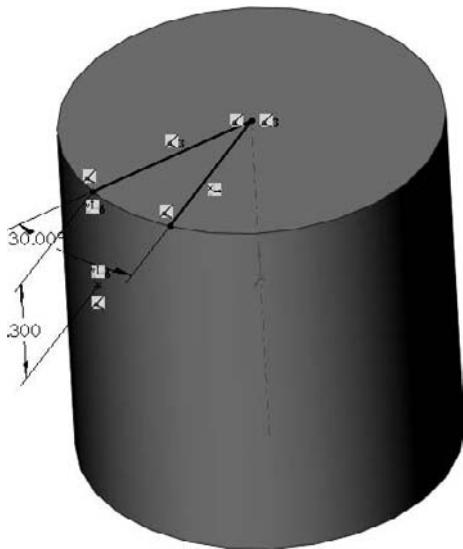
The example part at the end of Step 11

12. **Create an AlongY sketch relation between the points indicated in Figure 17.7.** The hole centerpoint on the cylindrical face is one of the points, as well as the endpoint of the angled line. Change the angle dimension to ensure that it is controlling the sketch point as expected. Click the green check in the upper-right corner of the graphics window to accept the result and exit the command.

FIGURE 17.7

Control the placement of the 3D sketch point around the cylinder.

On the CD-ROM

You can find the finished part from this example on the CD-ROM named `Chapter 17 3D Hole Placement.sldprt.` ■

Making and using Favorites

Hole Wizard Favorites store types of holes that you use frequently so that you can simply recall a favorite, rather than manually making all the changes every time you use the same hole. Favorites are saved to a database named Default.mdb as you create them, and are immediately available from all other part documents. You can also save favorites to a special file type with the extension `*.sldhwfvt`. Other users can then load these files and add your favorites to their Default.mdb databases. This is a convenient way to create company standards for hole features.

Shared Toolbox installations share a `SWBrowser.mdb` between several users, making Hole Wizard Favorites available to everyone. I cover how to set up shared Toolbox installations later in this chapter.

Creating a Hole Wizard Favorite

To create a Hole Wizard Favorite, set up a Hole Wizard hole as you normally would, and then use the Add Favorite button to add it to the Favorites database. The Hole Wizard Favorite panel contains five buttons:

- **Apply Defaults/No Favorites.** Removes favorite settings from the current interface, setting all values back to their defaults.

- **Add or Update Favorites.** Either adds a new favorite to the database or changes the name or other settings for an existing favorite.

- **Delete Favorite.** Removes a favorite from the database.

- **Save Favorite.** Saves a favorite to an external file with the extension `*.sldhwfvt`, which can be loaded by other users and added to their databases.

- **Load Favorite.** Loads a saved favorite file.

Storing custom holes

You can use Hole Wizard Favorites to store custom holes. Create the hole with its custom sizes, and then add the favorite and give it a recognizable name. The custom hole will now be available to anyone who connects to the same database file.

Administering Hole Wizard Favorites

The database file is typically found in the `Data` subdirectory of the SolidWorks installation directory, but an option in Tools ➪ Options ➪ File Locations ➪ Hole Wizard Favorites Database theoretically enables you to move the file to somewhere else.

Further, the `*.sldhwfvt` files do not have an entry in the File Locations list, but seem to always default to the `lang\english` subdirectory of the SolidWorks installation directory. Neither this location nor the `Data` directory makes sharing among multiple users very convenient, but both file types can be copied to other installations. You may want to read through Chapter 18 to learn about setting up libraries for all file types.

Best Practice

It is a best practice to create a folder for library type files that you want to save and use with a future version of SolidWorks. You can specify the locations for these files by choosing Tools ➪ Options ➪ File Locations. I recommend a location such as `D:\Library`. This moves the file off of the same drive as the operating system, in case you need to reformat, and it keeps it out of the Program Files area to prevent it from being lost or overwritten when SolidWorks is installed, uninstalled, upgraded, or changed in other ways. Even for files that need to remain in the SolidWorks installation directory (such as macros), it is best to also have these backed up in a library location. ■

Favorites quirks

Hole Wizard Favorites seem to have a couple of quirks that are possibly "sub-optimal," as they say. First, you can only see the favorites for a specific type of hole when that type of hole is activated in the interface. For example, if you have a number of favorites for countersunk holes, but you currently have the counterbored hole icon activated, you will not be able to see the countersunk favorites until you switch to the countersunk icon.

If you have a lot of favorites, this may be beneficial, but if you have only a few favorites, or you do not use favorites frequently, it may be confusing and can create some unnecessary steps to find all your favorites.

A second quirk occurs when you allow SolidWorks to name the favorites and you have fractional values such as ¼ — which happens now and then in hole sizes — and then try to save the favorites. Each favorite is saved as a separate file, using the name that was automatically assigned to it by SolidWorks as the filename. Unfortunately, the character "/" is not allowed in a filename, so it fails.

Using the Hole Series

The Hole Series enables you to make a series of in-context hole features in individual parts that are connected by a Hole Series assembly-level feature. It is intended for a stack of parts where, for example, the top part has a counterbored hole, the middle part has a clearance through hole, and the final part has a blind threaded hole.

Hole Series interface

The Hole Series used to be part of the Hole Wizard, but has since been exported as a separate tool. It is now a five-step, wizard-based feature, ending with populating the new hole with a fastener using Smart Fasteners functionality. The Toolbox add-in is required to use Smart Fasteners. Figure 17.8 shows the interface for the various steps.

Basic Hole Series steps
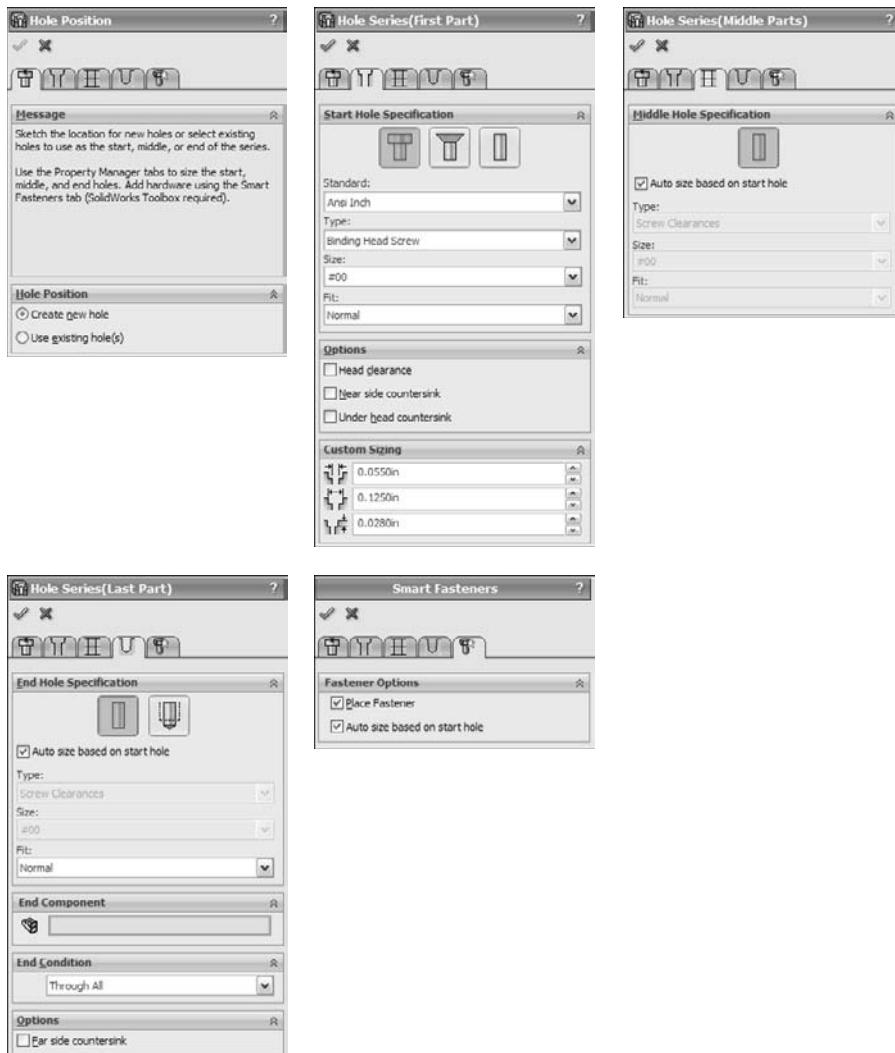
When using the Hole Series feature, you must follow these basic steps:

1. **Have an assembly open with two or more parts in it that need to be fastened together.**

2. **Initiate the Hole Series tool by choosing Insert ⇨ Assembly Features ⇨ Hole ⇨ Hole Series**. It is also available as a toolbar button, but it is not on the toolbar by default.

3. **If the Hole Series is to be started from an existing hole, then select it in the Hole Position panel.** If not, then use sketch points, construction geometry, dimensions, and sketch relations to locate the hole centerpoints.

4. **Use the tabs at the top of the PropertyManager to advance from one panel to the next.**
   - The Start Hole Specification refers to the part where the series of holes starts.
   - The Middle Hole Specification is for all parts between the first part and the last part.
   - The End Hole Specification is the last part and is either a through clearance hole or a threaded hole.

589
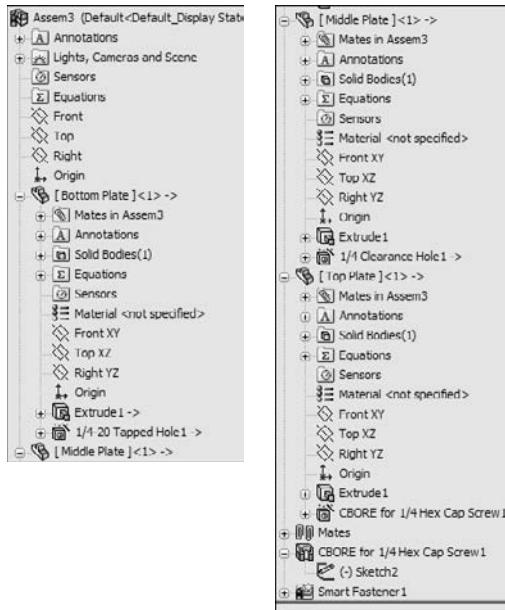
FIGURE 17.8

The Hole Series interface

The finished feature leaves an in-context feature in each part, with the Hole Series part in the assembly, as shown in Figure 17.9.

FIGURE 17.9

The finished Hole Series

Hole Series quirks

The Hole Series feature also has some quirks that you should know about so you don't spend too much time trying to verify that things are not working as you would expect. The first is that if you start the Hole Series without preselection, the cursor that comes up initially is a 3D sketch cursor, but once you click the cursor to place a point on a face, the Hole Series tool creates a 2D sketch feature.

The second example is actually from a tutorial in Chapter 16, and is where the Hole Series drills multiple holes and the holes will not all go through the same parts. This is like the example shown in Figure 17.10, where a hole drilled down from the top of plate 1 might drill into plate 5 or plate 6. In this case, you would have to create holes that drill into different parts separately, as was done in Chapter 16.
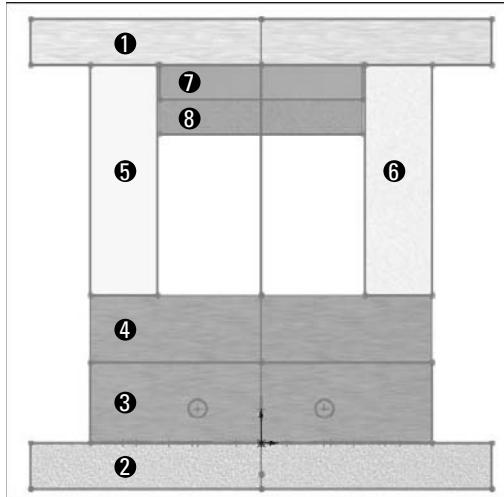
Also, you may find that the Hole Series does not retain the information you give it for "End Part," or the depth of the tapped hole at the end of the series. Behavior like this makes the Hole Series difficult to use reliably, so make sure to double-check any data created with the Hole Series feature. It is a highly convenient feature, but may also be flawed.

FIGURE 17.10

Create holes that drill into different parts separately

Understanding Toolbox

Toolbox is a complex add-in that helps you add hardware to your SolidWorks assemblies. Much of the functionality in it is intended to be automated, and to get the most from Toolbox, you need to configure it deliberately. The advanced functionality might not be obvious to users and should probably be set up by an administrator.

If you are an individual user who is using Toolbox on your own and not sharing data with others, the default Toolbox settings should be fine. But if you are one among a large group of users, a dedicated implementation effort is needed.

This book primarily covers Toolbox for the end user perspective. If you are looking for information from the administrative perspective, I recommend the *SolidWorks Administration Bible* (Wiley, 2009). That book covers various schemes for implementing Toolbox.

Caution

Improper installation, maintenance, or management of Toolbox can cause the loss of all useful information about fasteners and hardware in your assemblies. ■

Toolbox creates fasteners and other hardware components on the fly or reuses existing parts when possible. Technically, it is not a library, but a *configurator*. Libraries store existing components, while configurators build them on the fly from information supplied by the user.

One advantage of configurators is that the parts start out very compact because there is only the default size, and the sizes are efficiently stored in a database and created as needed.

The advantage of a library is that it enables you to simply plug in the parts to make them work. All Toolbox really needs to do for users is provide a library of parts. Anything more than that is only beneficial if it offers some improvement over a simple library of existing parts without introducing any risks or setbacks.

How Toolbox works

No one asks how a staircase works, because it does not work; it simply exists and people use it. An escalator, however, is a different issue. With an escalator, there is a complex installation, and then to use it, you have to know how to get on and get off and what to do if it stops working. The end results of using the staircase and using the escalator are the same (you start at the bottom and arrive at the top), but the complex automation is supposed to save you some effort.

That is one way you can look at Toolbox. The end product is supposed to be the same as using a static library of parts, but there is some mechanism behind the scenes that has to be set up and maintained properly in order for it to work in the way you expect. Most SolidWorks books, tutorials, or training materials are going to ask you to accept what happens inside Toolbox as a "black box" and to just assume that the end results are exactly what you need and intend. Here, I supply you with information about how it works, so you can decide how useful it will be for you.

The database

Toolbox has three major components:

- Default parts of one size, with named dimensions and features
- A database containing all size information for all parts and Hole Wizard holes
- A software application with settings and an interface

When Toolbox is installed, it starts as a set of SolidWorks parts with named features and dimensions, some suppressed features (depending on settings), some dlls (executable programs), and a database. The parts have a single Default configuration, which is typically one of the size extremities, either the largest or smallest. The database starts out about 87MB and includes all the size information for all the parts, as well as all the standards information.

If you create a *custom standard* in Toolbox, it actually replicates a section of the database. By doing this, the database file can easily double in size.

Later, you will see that a network installation of Toolbox requires the database to be on the network, and every time you create a new fastener, it has to open the database. As a result, simply placing a screw in an assembly can mean that even if your assembly is located on your local hard drive, you still have to open a very large database file across the network. The first rule about performance with SolidWorks is to work locally rather than across a network.

By default, the database is located at `C:\Solidworks Data\lang\English`. You can open this file with Microsoft Access or Excel.

Note

When specifying network paths, it is best to specify a Universal Naming Convention, or UNC, path rather than a mapped address. A UNC address follows the format, `\\Server\Shared Folder`. The advantage of the UNC over the mapped drive is that mapped drives can vary from one computer to another, but the UNC is always the same. ■
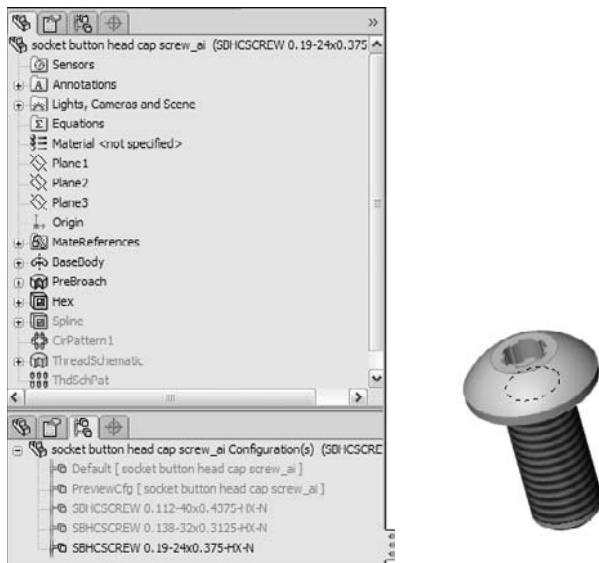
The Configurator application

If you have just installed Toolbox the way that most, if not all, new users do, then you will accept all defaults and trust the software that you just purchased to not give you bad advice. In this situation, the database is installed locally and Toolbox is set to use configurations for sizes.

When you put a Toolbox part into an assembly, you do not even notice anything other than the part going into the assembly, although it may hesitate while the large database is opened. If you check the part configurations, you may notice that there is a Default config and a new config that represents the size that you just created. Every new size that you create makes another new configuration. Figure 17.11 shows a Toolbox part with the FeatureManager and ConfigurationManager open, showing several configurations that Toolbox created in this particular fastener.

FIGURE 17.11

A Toolbox part showing the FeatureManager and ConfigurationManager

Next, you may receive an assembly from a client. Often, because Toolbox parts are located in an area where you would not necessarily look for parts, users send assemblies and parts, but do not send Toolbox parts. You may think that this is okay; after all, you have Toolbox on your system; therefore, it should pick up your toolbox parts. The truth is that when receiving an assembly from someone else, you are better off if one of the parties does not have Toolbox on their system.

Huge Screws

If both you and the client who sent the assembly have Toolbox, then you should be okay, right? Well, yes and no. Yes, your client's assembly will pick up your Toolbox parts, but no, it will not work properly because you do not have all the same configurations and sizes that your client has. In cases like this, you will experience what I have come to refer to as the Huge Screws syndrome. When SolidWorks finds the right file but cannot find the right configuration, it uses another configuration, usually the Default, which is generally the biggest size. This is where the Huge Screws name came from.

Part of the really bad news is that if you save your assembly with the Huge Screws, SolidWorks has no way of knowing that the huge screws are not the correct screws, and you can only solve the problem manually by going through the assembly and reassigning sizes to the huge screws.

You can work around this by opening an assembly that has not yet been saved with the Huge Screws, using the Advanced option in the Open dialog box (you can find this in the Configurations list), and then selecting the New configuration showing assembly structure only option. With this option, all components are suppressed. You can unsuppress any non-Toolbox parts and continue working. Ask your client to send you his Toolbox parts and then unsuppress those parts in the assembly, making sure that it finds the right parts; this is best done by having the correct parts already open before you open the assembly. These options are shown in Figure 17.12.

If you replace your Toolbox parts with the Toolbox parts from the client, you may experience the same problem in reverse if you had configs that your client did not. In the end, it would be great to be able to merge the two parts to combine all the available sizes into a single file. There is a way of doing that, which I will describe later, but it is a convoluted work-around. Files that have the same names and different content are at the top of the list of things you shouldn't do in file management, and yet the SolidWorks Toolbox system frequently creates this very situation.
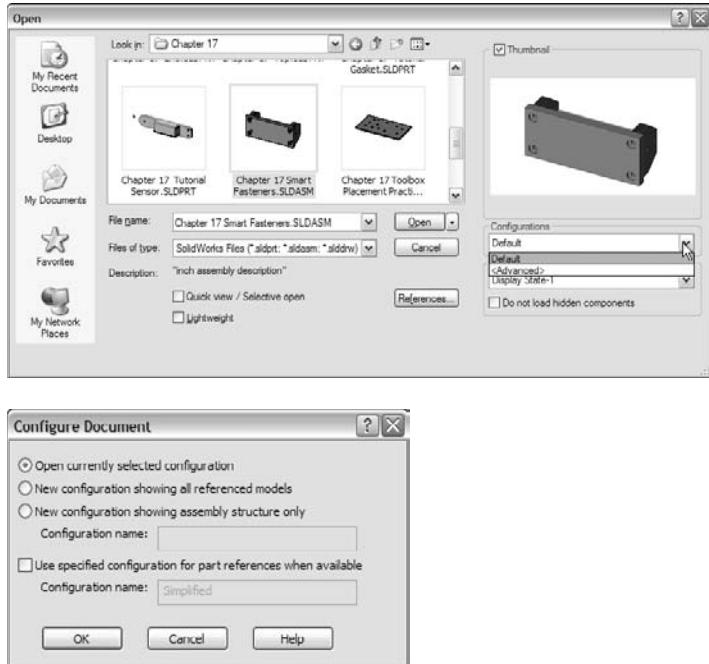
A slight retraction

To be fair, SolidWorks fixed the Huge Screws problem in the 2007 version by coming up with a clever method for figuring out which size is missing and building it on the fly when the assembly is opened. Additional information about the Toolbox parts is now stored in the assembly, which helps identify the missing parts. Unfortunately, the fix only works for assemblies that use the parts from the 2007 or later library and assemblies that have been built in SolidWorks 2007 or later. To sum up, if you have assemblies built in an older version of SolidWorks and your Toolbox library becomes corrupted or lost or you are sent an assembly that uses a different Toolbox library, even if you are working in a version later than SolidWorks 2007, you cannot benefit from this fix.

FIGURE 17.12

Opening an assembly with all parts suppressed

This is disappointing in many respects because if you have existing Huge Screws problems, you will continue to have them until you rebuild the assembly or manually repair the configurations. It is doubly disappointing because the information needed to re-create the correct configuration has always been stored in the assembly — the filename and the configuration name are enough — but SolidWorks has missed an opportunity to really fix this problem.

I provide some recommendations before the tutorial at the end of this chapter if you are still interested in using Toolbox.

Toolbox organization

Toolbox parts can be organized in a number of ways. The raw parts are organized as follows:

- Standard and Units (for example, ANSI Inch or ANSI Metric; most standards do not include multiple units, they assume metric).

- Hardware Type (such as bearings, bolts, and bushings).

● Each type is organized differently, but bolts and screws are organized by drive or head type. (For example, you have socket head screws, hex head, and thumb screws.)

● Filenames look like `Socket Button Head Cap Screw_AI.SLDPRT`, where the `AI` represents ANSI Inch.

Figure 17.13 shows this organization in part.

Configurations or parts?

By now you might be unsure about using configurations in general. If so, that is not the message I am trying to convey. Configurations in themselves are not the problem; the problem here is in the file management practice of having files with the same names but different content. Mixing that with the practice of trying to treat "configurator" software like a "library" exacerbates the problem.

FIGURE 17.13

Toolbox content organization

That said, you have two options regarding how you create different sizes. The default option is that sizes are created as configurations within a single part. The other option is that sizes are created as individual files.
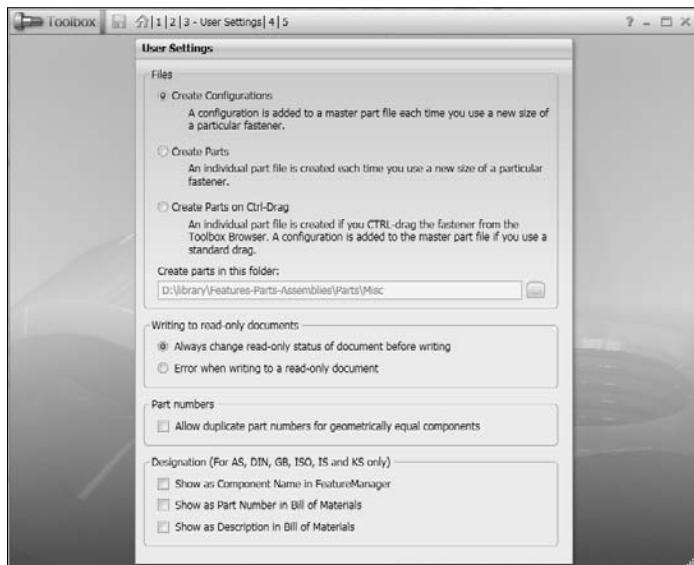
The best time to make this choice is before you install SolidWorks. Unfortunately, before you install SolidWorks, you probably do not have any idea that these issues exist. The reason for making this decision not just early, but immediately, is that if you start using the default setting (configurations), and make a few configurations for some parts and then switch to using the Save Parts setting, the parts that are saved out will all have the pre-existing configurations and thus different sizes.

If you find yourself in this situation, it is better to reinstall Toolbox or simply to copy over a new default set of parts with no configurations.

You can access the Create Configurations and Create Parts options by choosing Toolbox ⇨ Configure ⇨ Define User Settings, as shown in Figure 17.14. I discuss the other settings in this dialog box later in this chapter.

FIGURE 17.14

Toolbox settings for the Create Configurations and Create Parts options

Which is better?

The following list contains some pros and cons for each option.

Configurations are better for:

- Controlling data across several sizes. For example, a design table can drive custom properties that are added to all configurations. Doing this with many individual parts would be very messy.

- The interface to select configurations from a list is easier to work with than the interface to select a part from a list.

- File management organization is somewhat easier for configured parts.

Separate parts are better for:

- Keeping the file size small.

- Replacing all of one size part with another.

- Guaranteeing that you will never have the Huge Screws problem.
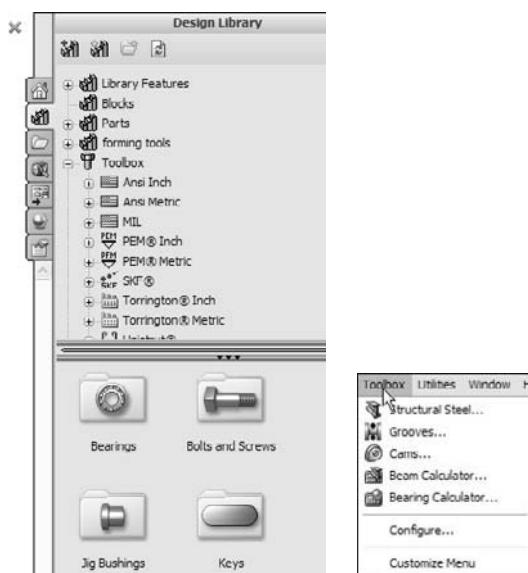
Using Toolbox

Toolbox has two components: Toolbox and Toolbox Browser. In practice, the *Toolbox* component is often ignored, and the *Toolbox Browser* component is generally referred to as Toolbox.

The Toolbox Browser is the Task Pane interface and is found on the Design Library tab, as shown in Figure 17.15. The Toolbox component is found in the Toolbox drop-down menu. It includes structural steel shapes, grooves, cams, and beam and bearing calculators.

FIGURE 17.15

Toolbox and the Toolbox Browser
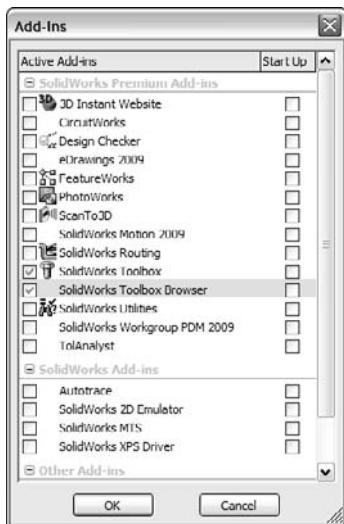
Turning on Toolbox and the Toolbox Browser

You can turn on Toolbox and the Toolbox Browser through the Tools ➪ Add-Ins dialog box. The column of check boxes on the left indicates that the add-in will be active for the current session of SolidWorks only. The column of check boxes on the right indicates that the add-in will be active every time the software starts up, as shown in Figure 17.16.

Turning Toolbox on in the Tools ➪ Add-ins interface

Once the Toolbox Browser is turned on, you can use it by expanding the Task Pane at the right of the SolidWorks graphics window and clicking the Design Library, which looks like a stack of books. In this panel, you will see the Toolbox screw symbol. Expand icons until you find the fastener or other hardware that you are looking for, and then drag the part into the assembly.

Populating holes

Holes can be populated in several ways, such as dragging-and-dropping, populating multiple holes at once, using the right mouse button (RMB) menu, and using feature-driven component patterns. I discuss manual and patterning options here, and Smart Fasteners in the next section.
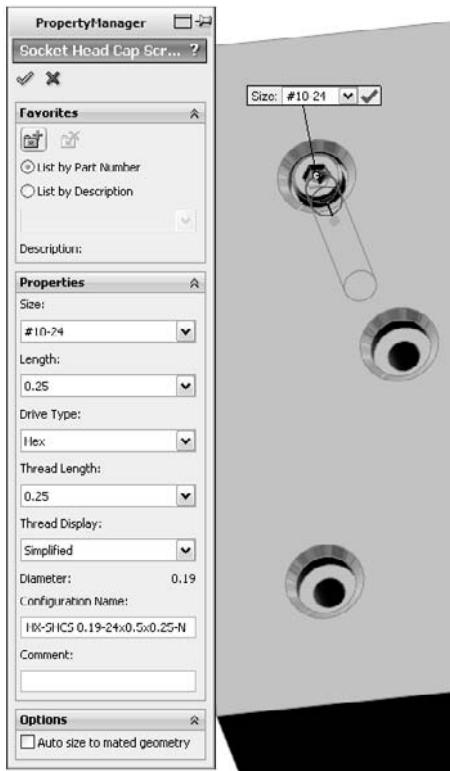
Drag-and-drop

The simplest way to bring Toolbox parts into an assembly is to drag-and-drop them. Position the part that the fastener goes into so that you can see the edge of the hole where the screw head will go. Then browse to the correct fastener and drop the fastener onto the edge, as shown in Figure 17.17.

FIGURE 17.17

Dropping a fastener onto a hole

Toolbox parts will even automatically size themselves based on the hole. It is best to use Hole Wizard holes if you are going to use this function of Toolbox parts. Hole Wizard and Toolbox are meant to work together.

When you place the fastener, a PropertyManager appears that enables you to select various properties of the part, including the length, if you want to override the automatically selected size, the thread representation, and if you want the fastener to change in size when the hole changes. Also after placing the fastener, a handle at the end of it enables you to drag the length of the fastener, which will snap to predetermined lengths. You can also drag the arrow that appears at the end of the fastener to graphically select the length.
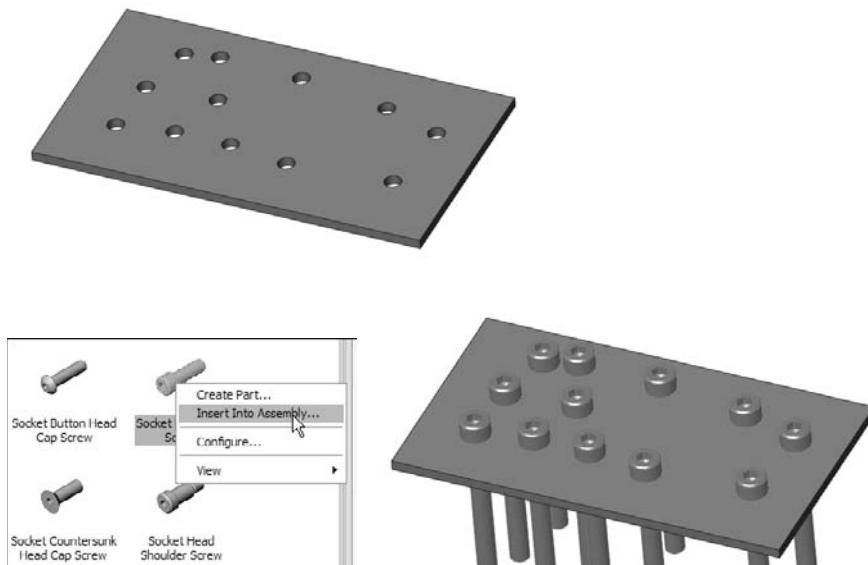
Populating multiple holes at once

Figure 17.18 shows the progression from a plate with holes in an assembly. In this example, you would Ctrl+select the edges of the holes, then select a fastener, and then choose Insert Into Assembly from the RMB menu to populate multiple holes.

Populating multiple holes at once in an assembly

Feature Driven component patterns

Chapter 15 covers Feature Driven component patterns (also known as derived patterns), where a pattern of parts in an assembly is driven by a feature pattern in a part. You can find this assembly feature by choosing Insert ➪ Component Patterns ➪ Feature Driven in the assembly menus.

Smart Fasteners

Smart Fasteners are Toolbox parts that know what holes they go into automatically. The database that holds all the information for Toolbox part types and sizes also holds the information for the sizes of the holes. It is only natural that SolidWorks tries to combine this information and use it to its best advantage. You can use Smart Fasteners in two ways: asking SolidWorks to automatically place fasteners in existing holes or placing hole series holes and fasteners simultaneously in the context of the assembly.

Smart Fasteners with Hole Series

One way to use Smart Fasteners is in conjunction with Hole Wizard Hole Series. Hole Series creates the holes through multiple parts at once, creating the appropriate type of hole through each part, and then Smart Fasteners automatically places fasteners in the holes, even including nuts and washers. To do this, you can select the option on the last panel of the Hole Series PropertyManager interface, as shown in Figure 17.19. If you are planning on using Smart Fasteners, using them in conjunction with the Hole Series holes is your best bet.

The Place Fastener option

The Smart Fasteners with Hole Series is a function that you should be careful with. It is very effective, but it may cost you some performance (speed). The Hole Series is an assembly feature that drives several in-context features, and then parts are mated to those in-context features (fasteners).

Smart Fasteners Populate All

Smart Fasteners functionality has an even more automatic component. Once an assembly has parts mated into place, you can place fasteners into parts with appropriate holes by face, by part, or for the entire assembly at once.

Caution

You may not want to spend a lot of time trying to use this type of the Smart Fasteners functionality. I have tried to find examples where Smart Fasteners works well and predictably, but with limited success. I have searched through training examples, through tutorial files from SolidWorks, and have even made some of my own example files. I have looked for presentations from user groups and SolidWorks World that use Smart Fasteners, but no one appears to be talking about this functionality. Although in theory, it offers interesting functionality, in reality, it receives very little attention — definitely a warning sign.

The one assembly that I did find where Smart Fasteners worked surprisingly well (in fact, almost perfectly) was from the sample files that installed with SolidWorks. Upon closer examination, the reason this worked well was because it used assembly features for the holes, and so the holes did not appear in the individual parts. If that is the price that you have to pay just to get fasteners to populate automatically, then you will probably rather put them all in manually. ■

The limitations of Smart Fasteners

Smart Fasteners have some documented limitations where you should not expect them to work:

- Holes in single parts
- Holes created by extruding a nested loop
- A mirrored hole or cut features
- Holes in mirrored, imported, or derived parts
- Misaligned holes
- Holes with a large difference in diameter
- Holes with large gaps between them (a large gap in the axial direction)
- Holes made using different techniques (such as sketch pattern versus feature pattern)

If you would like to try out Smart Fasteners, then you can use the assembly included on the CD-ROM called `Chapter 17 Smart Fasteners.sldasm`. In this assembly, half of the holes are done correctly and the other half are not: the screws are put in either backward or headfirst. The documented method for flipping the fasteners is to expand the Smart Fastener, right-click the series, and choose Flip. In this case, my attempts resulted in success about half of time, which was somewhat higher than my attempts with other assemblies. In some cases, screws were put in the ends of shafts without holes, on filleted edges, and unfortunately missed most of the places that I *did* want the screws to go.

Organizing Toolbox parts in an assembly

Assembly FeatureManagers are hard enough to manage when they become full of parts; they become even more unmanageable when they also need to include the many types of fastener parts. As a result, I recommend organizing fasteners, as well as any other type of part that is found in large quantity in the assembly, into folders, as shown in Figure 17.20. You should also group parts of the same size or function together.
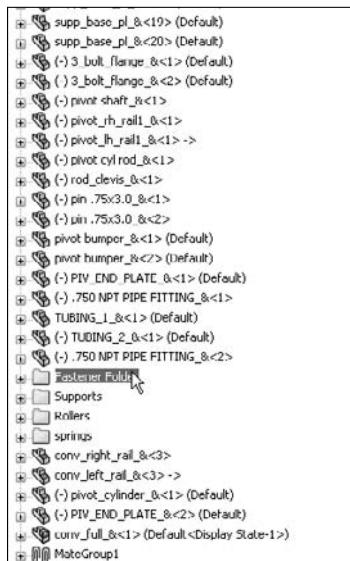
Recommendations

After spending almost an entire chapter saying what you *should not* do, it is finally time to say what you *should* do. Toolbox can be downright dangerous if you install and use it improperly; however, the following recommendations work in most situations.

Just to be clear about this, the most serious file management problems with Toolbox show up when you use configurations, which just happens to be the default option. Still, I'm a big fan of using configurations in general, and especially with library parts, but a Toolbox implementation with configurations is challenging. Again, this is because of the file management issue of different users having the same part names with different content (Toolbox parts with or without certain configurations created).

FIGURE 17.20

Organizing Toolbox parts into folders

The simplest setup that works

If you are a single user who does not share files over a network with other users, then installing SolidWorks and Toolbox with the default settings should work for you. This appears to be the arrangement that the developers had in mind when they programmed the tool, because it is the only scenario in which it works as expected.

Be careful if you ever receive an assembly from another Toolbox user, because this is the one situation that can cause immediate trouble. If the user also sends his Toolbox parts, then I recommend that you open all his Toolbox parts before you open his assembly so that the assembly is certain to access his Toolbox parts instead of yours.

If you need to include materials and mass-populate custom properties, then I recommend that you go through the exercise of building all the configurations of all the parts, and then use an auto-created design table to drive the properties. If you have more than one user, then this technique will not work for you, unless both users work independently from one another.

A complete setup that works

If you have multiple users that share assemblies, then you need to also share the Toolbox library. If you share assemblies only among yourselves, meaning only with other users who are also sharing

Toolbox, then sharing Toolbox should be good enough. However, if you share assemblies with Toolbox users who do not share your Toolbox library, then you should probably go through the exercise of populating all your parts with all the available configurations. If you do not receive assemblies from outside of your group with Toolbox parts in the assembly and you have network performance problems, it may be a good idea to install Toolbox locally, but to set it to use the Create Parts setting, where the parts are on a shared network location.

If you use a Product Data Management (PDM) system, then I would definitely install Toolbox locally and use the Create Parts setting. The sharing occurs through the PDM system. Library parts should be non-revision managed parts, but you may want to have a representation of the fasteners so you can do where-used searches and BOMs.

The least problematic technique is to turn Toolbox off altogether and either buy or make your own library of static parts. You can then distribute these files internally in your organization, as well as to any other people upstream or downstream from you who also share files with you. You can build this type of library by using Toolbox's config population tool; materials or other custom properties are then dealt with the way you want, likely using auto-created design tables.

Of course, there is a downside to this too, and it is that you lose all the nice automation features available with Toolbox. The best option if you want to keep Toolbox is to use the Copy Parts option, install locally, use a PDM system, and if you get assemblies from Toolbox users who aren't part of your network, insist that they either use your parts or send you their parts.

For a more complete discussion of the administration issues surrounding SolidWorks Toolbox, please refer to the *SolidWorks Administration Bible* (Wiley, 2009).

Tutorial: Gaining Experience with the Hole Wizard and Toolbox

Figure 17.21 shows a section view of the assembly used for this tutorial. Notice that there is a gasket under the Sensor part.
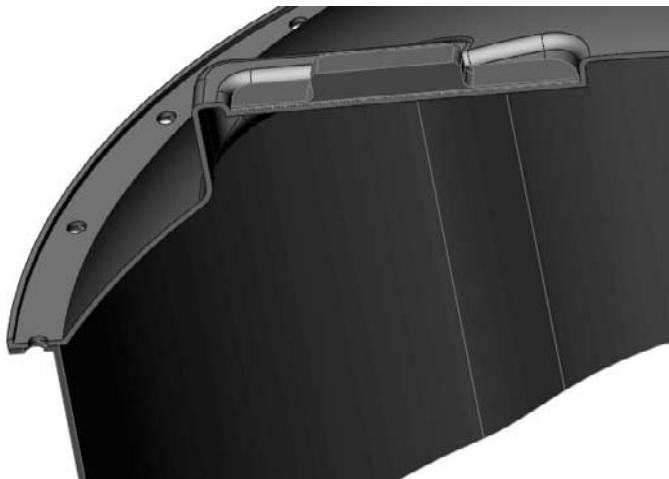
This tutorial assumes that you have a working copy of Toolbox running on your computer. If you do not have Toolbox, then you can proceed to the next chapter. This tutorial also assumes that your Toolbox is using the default Create Configurations setting, although it can also work with the Create Parts setting. To get some experience using this tool, follow these steps:

FIGURE 17.21

A section view of the tutorial assembly
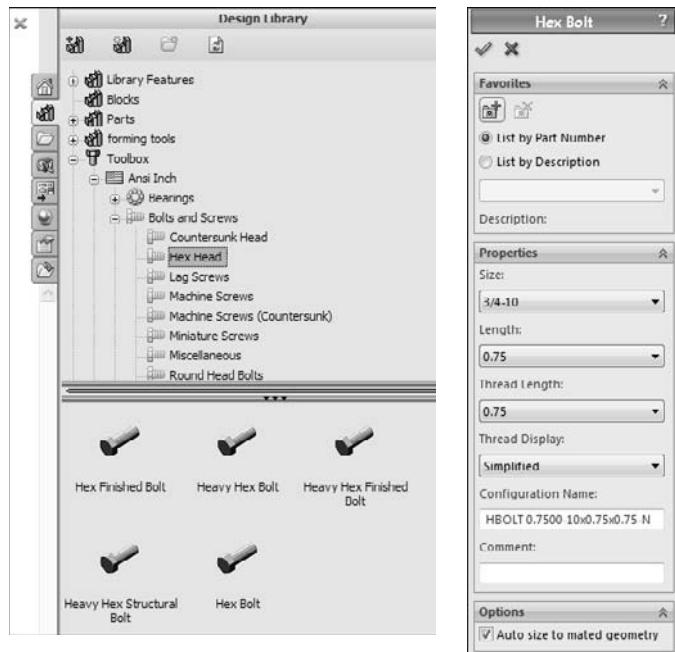
1. **Open the assembly from the CD-ROM called** `Chapter 17 Tutorialstart. sldasm`.

2. **Make sure that the Toolbox Browser is turned on by choosing Tools ➪ Add-Ins ➪ SolidWorks Toolbox and ➪ SolidWorks Toolbox Browser.**

3. **Expand the Task pane, found on the right side of the graphics window, and display the Design Library panel, which contains the Toolbox icon.** Expand the ANSI Inch standard, and the Bolts and Screws folder, and finally click the Hex Head bolt, as shown on the left in Figure 17.22. Drag-and-drop the Hex bolt into any hole. It snaps into place because of the Mate Reference that is used on the Toolbox part. Toolbox should automatically size the bolt correctly. Click the green check when you are done, then click the red X to finish placing bolts.

Select and place a fastener.
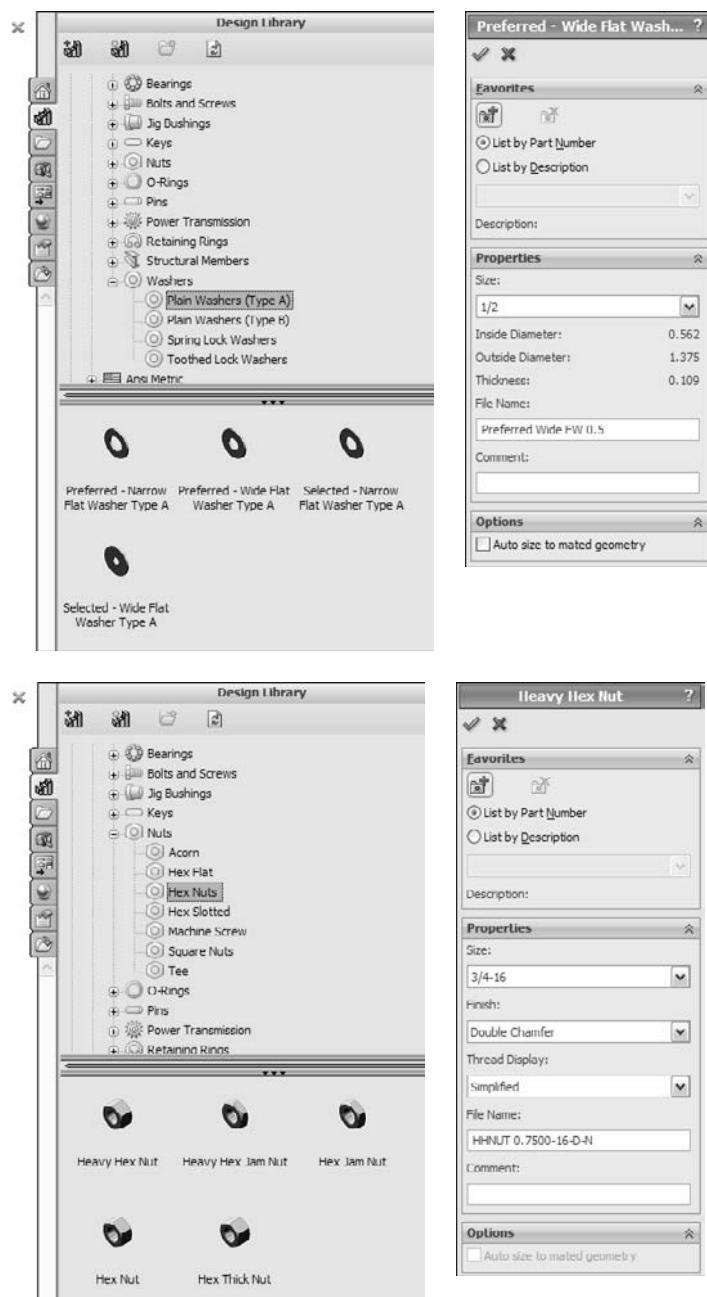
4. **Add a flat washer and nut to the bolt, as shown in Figure 17.23.** The washer is Plain Washer Type A, Preferred - Wide Flat Washer. The nut used is Hex Nut, Heavy Hex Nut.

FIGURE 17.23

Specifying the washer and nut
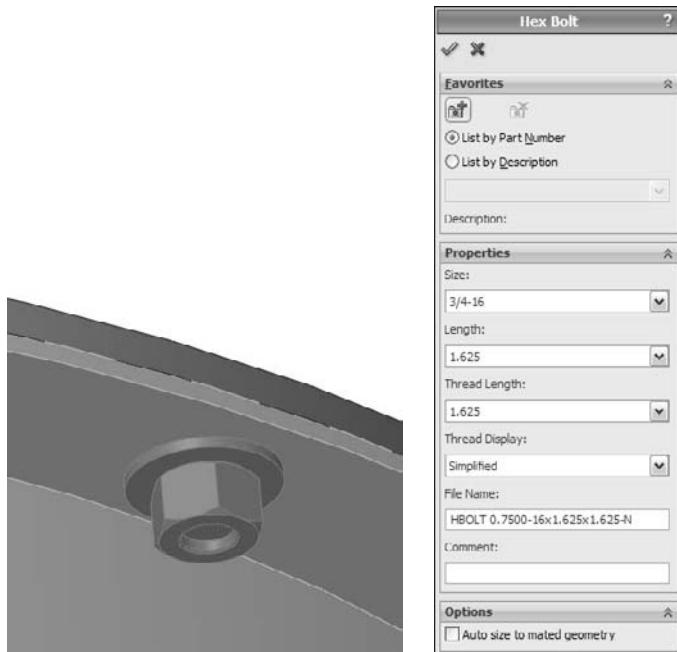
5. **Right-click the bolt, either in the graphics window or in the FeatureManager, and select Edit Toolbox Definition, toward the bottom of the menu.** Notice that the bolt is too short, as shown in Figure 17.24. Change the length of the fastener to 1.625 inches. You can change this in the PropertyManager or by dragging the green arrow placed on the bolt.

FIGURE 17.24

The bolt is too short.

Note
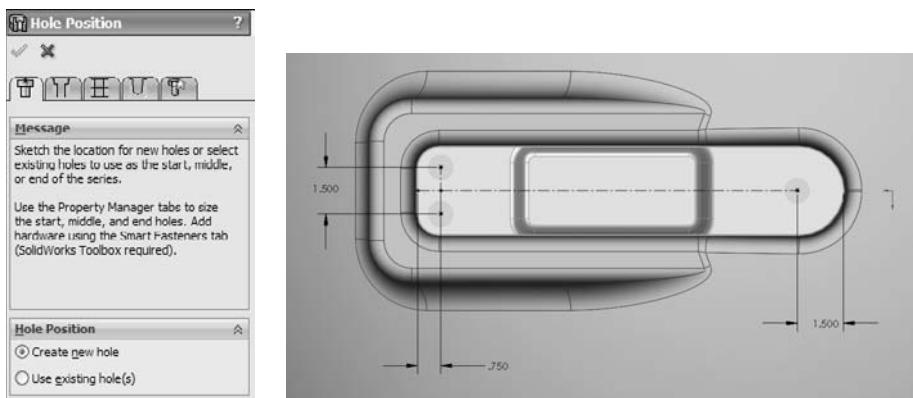
If you try to apply Smart Fasteners to the hole, you will notice that the fastener is placed incorrectly, and changing fastener types is not simple. ■

6. **Create a Feature Driven component pattern (Insert ➪ Component Pattern ➪ Feature Driven) using the circular pattern of holes on either the Top or Base parts.** Pattern the bolt, washer, and nut all in the same component pattern.

7. **Zoom in on the sensor on the top of the assembly.** There is a gray gasket between the orange sensor and the blue top parts. Click one of the flat ends of the sensor part and then click the Hole Series toolbar button, or choose Insert ➪ Assembly Feature ➪ Hole ➪ Hole Series.

8. **Make sure that you select the Place Fastener option on the final tab when you get there, as well as the Create New Hole option.** This workflow is different from previous versions. Be aware also that Toolbox may delay several seconds as you initiate these features.

9. **Make three sketch points and use construction geometry and dimensions to locate the holes, as shown in Figure 17.25.** The size and types of holes are determined in a later step. (This is the reverse of the normal Hole Wizard, where you first determine the type and size of hole, and then you establish the positions.)

The positions of holes in Step 9
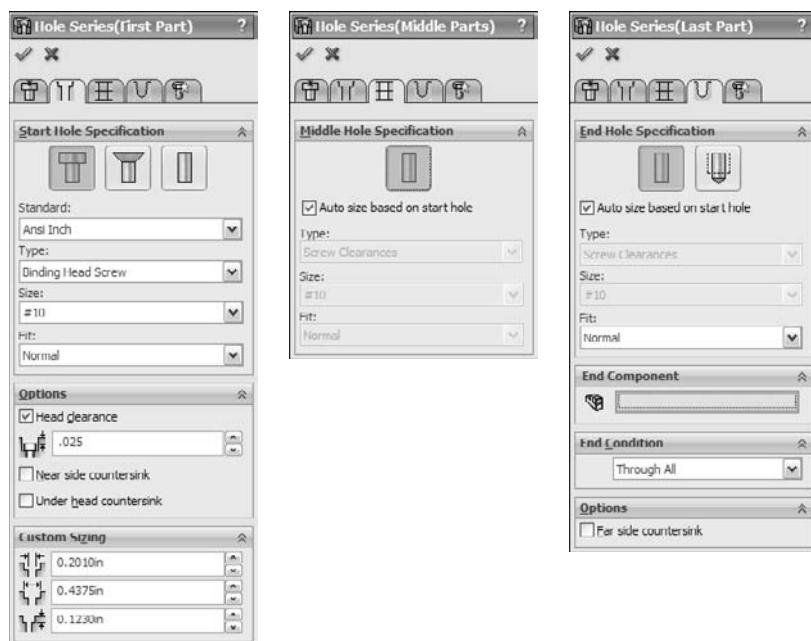
10. **Click the Next button (the blue arrow pointing right) to move to the First Part hole specification.** Set it to a counterbored hole, for a #10 binding head screw, with a head clearance of .025 inches, as shown in Figure 17.26 in the image to the left. Click the Next button to advance to the Middle Parts hole sizing.

11. **In the Middle Parts PropertyManager, make sure that the Auto size based on start hole option is selected, as shown the middle image in Figure 17.26.** This creates a normal fit clearance hole for the gasket part. Click Next to advance to the hole definition for the Last Part.

12. **In the End Hole Specification panel, make sure that you select the Hole rather than the Tap option, as well as the Auto size based on start hole option.** This is shown to the right in Figure 17.27.

13. **Proceed to the Smart Fasteners tab.** Make sure the Place Fastener option is selected, along with the Auto size option.
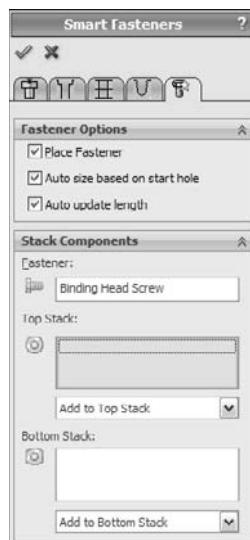
FIGURE 17.26

Sizing the holes

FIGURE 17.27
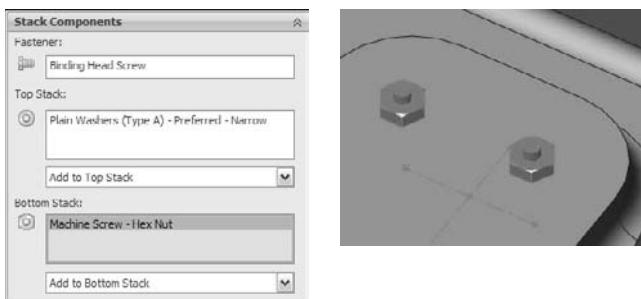
The Smart Fastener PropertyManager

14. **Add a washer and a nut to the bottom stack of the binding head screws.** Using the Stack Components panel of the final tab of the Hole Series/Smart Fastener interface, add a washer and a nut to the bottom stack.

15. **A dialog box appears, enabling you to add a washer and a nut, as shown in Figure 17.28.** You may want to roll the model over so that you can see the components being added to the underside of the screw. You can add other properties to the parts by clicking the Properties button. Notice that the screw has been lengthened to accommodate the added components.

FIGURE 17.28

Adding washers and nuts

Note

If you add a washer to the top stack, the hole does not automatically become larger, and it may cause interference. Be careful about your choice of top-stack washers. ■

Note

You may have noticed that this time, Smart Fasteners worked almost flawlessly and certainly saved you some time. Although this tool is not applicable to other purposes, when used with the Hole Series, it is quite useful. ■

Caution

The final version of the assembly on the CD-ROM may open up on your computer with Huge Screws if you open it before completing the tutorial. This is because the configurations used in the assembly are on my computer. Although you have the same parts, before doing this tutorial, you may not have the same configurations; therefore, they cannot be found and come in Huge instead. This was intentional; it is a practical reminder of this problem and how easily it can happen to you. ■
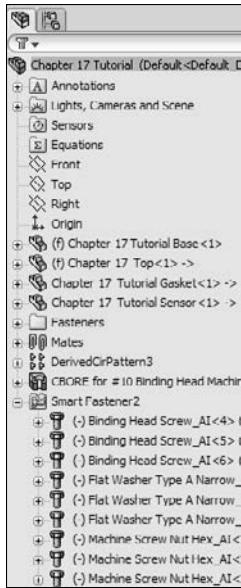
16. **You may want to group the fasteners and even the fasteners' mates into folders, as shown in Figure 17.29.**

The finished Assembly FeatureManager interface

Summary

The Hole Wizard can make holes based on 2D or 3D sketches. The type of hole that you create is up to you. Two-dimensional sketches are far easier to use than 3D sketches.

I have met people who claim to have had good success with Toolbox even in a shared environment, but given that the problems with the tool are so easy to demonstrate, these people are either extremely disciplined or extremely lucky. For all users except those who work alone and do not share files with other Toolbox users, Toolbox can cause a number of major problems. You can develop techniques to prevent you from experiencing Huge Screws; for example, either not sharing assemblies with other Toolbox users or pre-populating all your configurable parts with all possible configurations. Further, Smart Fasteners that you use in conjunction with Hole Series violate any best practice guidelines that you could name when it comes to assembly performance and circular references; however, if you can work with that, then it is a really sophisticated technique.

Working with Library Features

L ibrary features are features that you create once and re-use many times. They are intended to be parametrically flexible to fit into many types of geometry, but they can also be of a fixed size and shape. You will use all the information that you have learned in previous chapters about designing for change, and design intent in this chapter, as well as learn how to create, use, and store library features.

Using Library Features

Library features reside in the Design Library, which is located in the Task Pane to the right of the graphics window.

Tip
You can detach the Task Pane from its docking location and move it wherever you want, leave it undocked, or move it to a second monitor. ■

You can use library features for snap rings, grooves for o-rings, custom holes, mounting bosses for plastics, mounting hole patterns, electrical connector holes, and much more.

One very useful aspect of library features is that they can be driven by configurations and design tables. Once the feature is in the part, the configurations are still available, and so you can change the config of an applied library feature at any time.

You can also link a library feature to an external file. This enables you to change a feature, or a set of features in several parts at once if they are all externally linked to the file.

Getting started with library features

Library features are simple to use and only slightly less simple to set up. For that reason, in this chapter I discuss using them first, so that you know what kind of behavior you are trying to create when you go to make your own features. As a result, setting them up should make a little more sense.

To use a library feature, you just drag-and-drop it from the Design Library onto the appropriate geometry. You are then prompted to select references in the new part that match the base geometry that the library feature is attached to. You can be fairly creative with references, but one of the goals when creating the feature is to make it work with as few references as possible, in order to make it easy, fast, and reliable to use.

SolidWorks software installs with several sample library features in the Design Library. The following demonstration uses some of these standard library features. Later, you can add library features from the CD-ROM to your Design Library.

Applying the Library Feature interface

Library features work best if they go from a certain type of geometry to a similar type of geometry; for example, from rectangular to rectangular or from circular to circular. This is because the relations or dimensions that link the feature to the rest of the part tend to be dimensions from straight edges or concentric sketch relations. Of course, there are other ways of applying library features, but these are the most prevalent. Library features can be applied unconstrained and then constrained or moved later, but the process is cleanest when it all just falls together correctly the first time.

Task Pane

You do not have to save the part or do anything special before applying a library feature. All you need to do is find the Task Pane. The Task Pane is the window that flies out from the right when you open SolidWorks. You may have turned it off and forgotten about it, in which case you can turn it back on by choosing View➪Task Pane from the menu.

The Task Pane automatically closes when you click outside of it unless you pin it open using the pushpin icon in the upper-right corner of the window. When you do this, any toolbars that appeared on the right side of the Task Pane control tabs are moved out and positioned between the graphics window and the Task Pane, which now remains open by default.
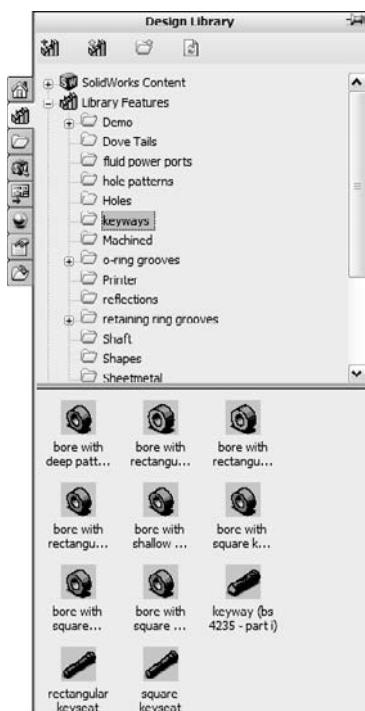
You can also detach the Task Pane by dragging the bar at the top of the pane. Figure 18.1 shows the Task Pane docked to the right side of the SolidWorks window.

Tip

If you are using dual monitors, you can drag the detached Task Pane onto the second monitor, which enables you to use the Task Pane and at the same time gives you more room in the graphics area. You must do this for each session; the Task Pane does not remember positions on a second monitor. ■

FIGURE 18.1

The Task Pane docked to right side of the SolidWorks window

Design Library

The Design Library tab displays an image of a stack of books. It is the overall library area for all sorts of elements in SolidWorks, which I discuss later in this chapter. The only part of the Design Library of concern right now is the Features folder. If you expand this folder, you can see that it is populated with some sample features.

Open a new part and create a cylinder using any method you want (for example, extrude, revolve). Make the diameter three inches and the length a little more than one inch.

In the Inch features folder, click the folder called o-ring grooves. The first feature in the list is called *face static – gas*. Drag-and-drop this feature onto the end flat face of the cylinder. As shown in the image to the left in Figure 18.2, the PropertyManager displays a yellow information panel explaining the process.
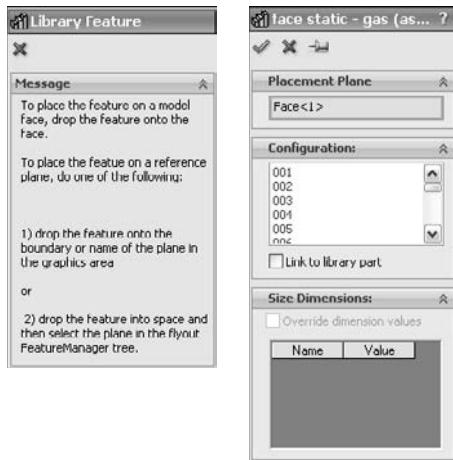
The next step is to select the configuration, as shown in the image to the right in Figure 18.2. Not all library features have multiple size configurations, but these do. The configs in this case are driven by design tables. Select configuration 330.

FIGURE 18.2

Placing the feature and selecting the configuration
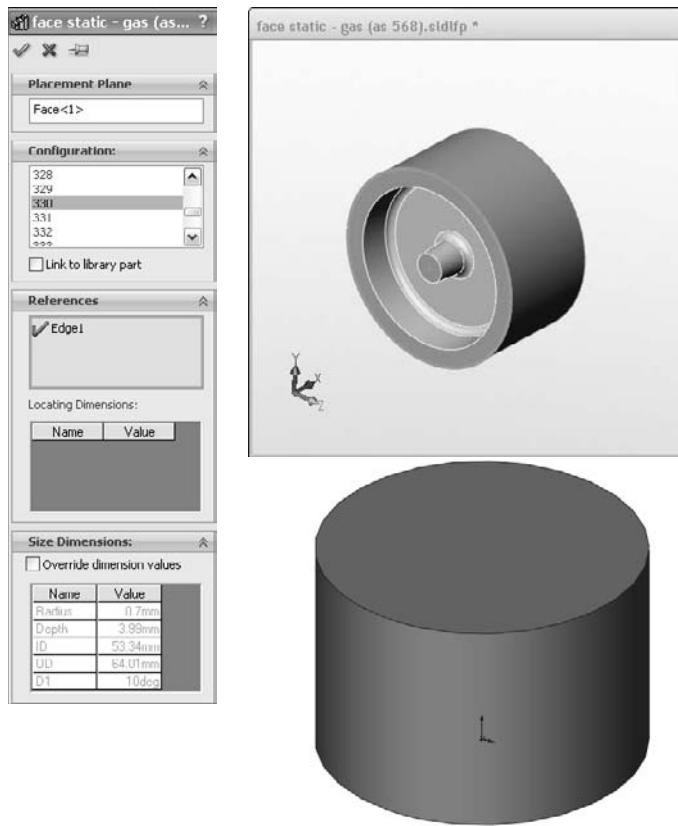
When you select the configuration, the interface changes. In this case, only a sketch relation locates the feature; it is not located by dimensions. Notice that in the References panel shown in Figure 18.3, there is an Edge entry with a question mark, which becomes a check mark after you select the edge. This means that the library feature needs a circular edge to locate it. Notice that a small window appears, displaying the library feature. You may not be able to distinguish it in Figure 18.3, but the circular edge around the face that the library feature is on is highlighted in green. This indicates that you need to select an edge that has the same relation to the library feature as the highlighted edge. Pick the circular edge of the part on the end of the cylinder where you want to place the library feature.

Next, use a rectangular part where the library feature is located by using dimensions rather than sketch relations. Create a rectangle 1.5 inches by 2 inches and extrude it to about 2 inches in depth.

FIGURE 18.3

Locating the library feature

In the Design Library, browse to features, then inch, and then the fluid power ports folder and drag the sae j1926-1 feature onto the end of the extruded rectangle. Select the 38-24 size from the configurations list. A window appears, prompting you for reference selections, as shown in Figure 18.4.
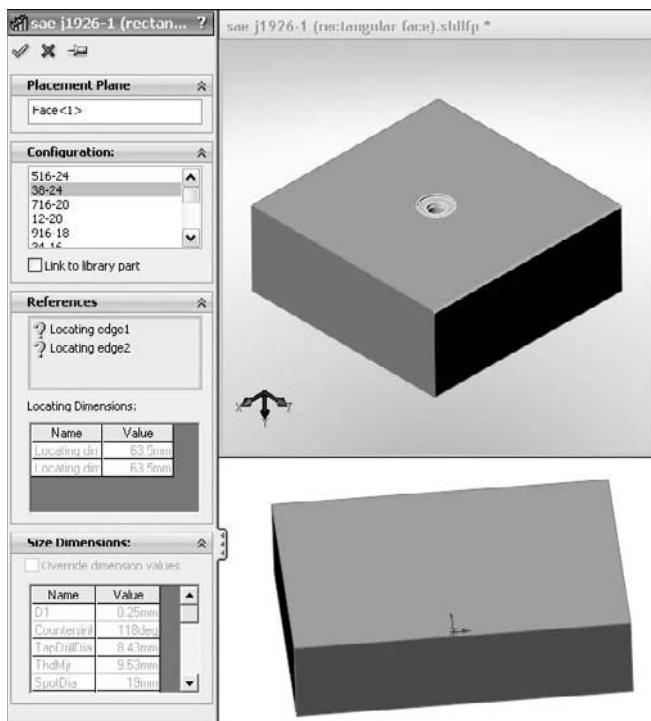
Tip

It is often helpful to orient the part that is receiving the library feature in the same way as the part shown in the preview window. This helps you to visualize which edges to select. ∎

FIGURE 18.4

Placing a library feature with dimensions

After the locating edges have been identified, the Locating Dimensions box becomes active and you can change the values of the dimensions to locate the feature. Further, in the Size Dimensions pane at the bottom of the PropertyManager, selecting the Override dimension values option enables you to change dimensions of the feature itself.

When you use a library feature with a design table, the design table is not brought into the part with the library feature. If the part already had a design table, this would cause multiple tables, which is not currently possible in SolidWorks. The configurations in the design table are brought forward, however.

If you override the feature dimensions using the Override Dimension Values option in the Size Dimensions panel of the Library Feature PropertyManager when feature configurations already exist, then a new configuration called Custom Configuration is created in the list of feature configurations. It appears that multiple custom configurations are not allowed, and so if you have to make changes, then you must ensure that they are right before you use the library feature in a part.

Exploring other Design Library functions

The Design Library has other functions besides library features. For example, you can use it as a repository for other items that you use frequently.
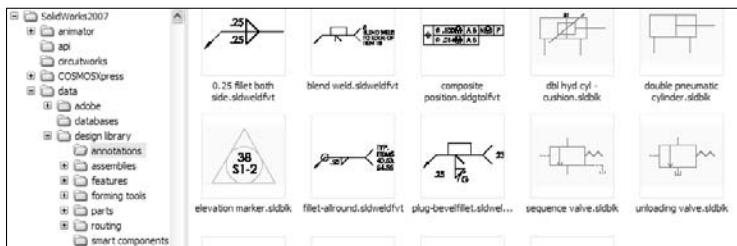
Annotations

You can store commonly used annotations in the Design Library. If you look at the Annotations folder with the default sample annotations, you see a combination of symbols and blocks. You can use symbols and notes in 3D models, but you can only use blocks in sketches or 2D drawings. Keep in mind that not all annotation types can be used in all places.

Annotations can be stored in the library as favorites or blocks. Many file extensions are used for different types of favorites, but they typically begin with `*.sld` and end with `fvt`, as in `*.sldweldfvt`. Figure 18.5 shows the default location of the Design Library and the Thumbnail view of the favorites and blocks in the Annotations folder.

FIGURE 18.5

The Annotations folder in Windows Explorer

Location of the Design Library folder

If you frequently work with different types of annotations, then you should organize the library into subfolders to separate symbols, annotations, and blocks, and move these folders to a different location. By default, the Design Library folders are found at `C:\ProgramData\SolidWorks\ SolidWorks 2010\design library\`. You should store them in another location, not in the SolidWorks installation directory, but in an area that you have selected to maintain SolidWorks data between releases. For example, I have a folder at `D:\Library` that contains folders for macros, templates, library features, library parts, favorites, and so on. You can easily back up or copy these files from one computer to another, although you must quit SolidWorks before making these changes. Keeping the data off the operating system drive is also a good idea because if you ever have to reinstall the OS, you will not lose your SolidWorks library data.

After moving the library, you have to point SolidWorks to the new location. To do this, choose Tools⇨Options⇨File Locations⇨Design Library. Delete the old location and browse to the new location. You should move other items in this list and redirect any items that you use, such as the templates and any other items you use frequently. Once you have specified the settings, they should be retained when you install service pack upgrades or future versions.

Library parts

The Design Library can also store commonly used library parts. One of the advantages of using the library for parts is that on placement into the assembly, if configurations are available in a part, then a window pops up, enabling you to select which configuration to place into the assembly.
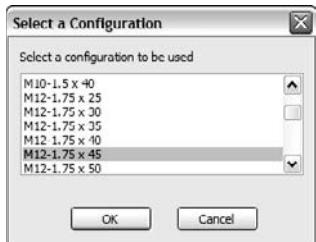
Note

In many cases, using the Design Library for library parts is considered an acceptable replacement for the automated function of Toolbox. If you use Toolbox to make the parts and populate them with configurations, and then save the parts out of Toolbox and into the Design Library, many options, including naming conventions, and a more flexible use of custom properties that are not available through Toolbox become available. ■

Figure 18.6 shows the configuration selection window. Note that the configurations are alphabetically listed and you can type in the box to go to the configuration that you want.

FIGURE 18.6

Inserting library parts with configurations

Parts inserted from the library parts folder can also take advantage of the Mate References functionality in the same way as Toolbox, by allowing parts to snap into place.

Sheet metal forming tools

I only mention sheet metal forming tools here as a part of the library. They work much like other library features, but they do so within the specialized functions of sheet metal parts in SolidWorks. I discuss sheet metal forming tools in Chapter 29. Forming tools folders have special properties. If you want to use the parts in a folder as forming tools, you must right-click the folder in the Design Library and choose Forming Tool Folder. The only other library type that needs special folders is library assemblies.

Assemblies

You can use library assemblies in SolidWorks in the same ways that you use library parts because they are inserted into the top-level assembly as a subassembly. For subassemblies that require motion, such as universal joint subassemblies, you can set the subassembly to solve as flexible or simply dissolve the subassembly into the upper-level assembly, through a RMB option.

Tip

When saving assemblies to the library, it is recommended that you put the parts in a separate folder to segregate the parts of different assemblies. ■

Routing

Routing is a separately purchased add-in that is included with SolidWorks Office Premium. It includes piping, tubing (rigid and flexible), and wiring. Routing makes extensive use of libraries and automation, but is not part of the scope of this book. The documentation on Routing at this time is rather sparse, but SolidWorks offers a reseller training class, and at this time, it's your best bet for information on this add-in.

Smart Components

Smart Components are components that resize by automatically selecting configurations, depending on the size of the geometry onto which they are being dropped. They do this from a design table–like interface that you set up to enable it to automatically select part configurations based on mating part sizes. For example, a clamp with many sizes driven by configurations would select the correct config when dropped onto different sizes of cylinders. This is a very useful tool. Smart Components are discussed in depth in Chapter 19.
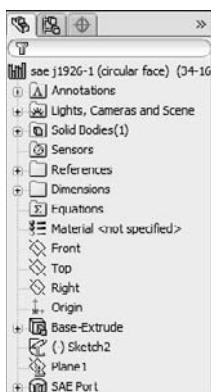
Creating Library Features

When you save library features to the library, they use the file extension `*.sldlfp` (library feature part). They must contain some base geometry, which simulates the part onto which the feature will be dropped. The base geometry is not transferred to the new part; only features that are marked with the "L" in the FeatureManager (for Library) are transferred to the new part. Figure 18.7 shows the FeatureManager of a library feature part.

FIGURE 18.7

The FeatureManager of a library feature part

Creating a library feature

When creating a library feature, the first problem that you need to solve is how the feature will be located on a new part. Does it need to be placed on cylindrical parts, rectangular parts, other types of shapes, or does this matter at all? Will the feature be located by using dimensions or sketch relations, or will it just be placed underdefined and later fully defined manually rather than automatically?

You may have noticed in one of the earlier examples that the sample fluid power ports had two versions of the same feature. One version is intended for the feature to be placed on the flat end of a cylinder, and the other version is intended for the feature to be placed on a rectangular face.

A few limitations

Library features can contain multiple features of different types. They may add and remove material, even within a single library feature. However, a few limitations exist. For example, they require a base feature, and multi-body features and external references are not allowed, nor are surfaces, sheet metal, weldments, or molds related features. In addition, you cannot add a scale feature (a feature that affects the entire body) to the library, nor can you apply library features to an assembly.

Creating a new library feature

To start a new feature, the first decision that you need to make is what shape to make the base. Is the feature a type that is usually going to go onto a single shape or multiple shapes? Regardless of your decision, you or whoever ends up using the feature will have the flexibility to change, or simply not use, the relations when you place the feature.

For this example, I use a rectangular base. The library feature that I want to create consists of two boss extrudes, a cut extrude, and several fillets. Here is how it works:
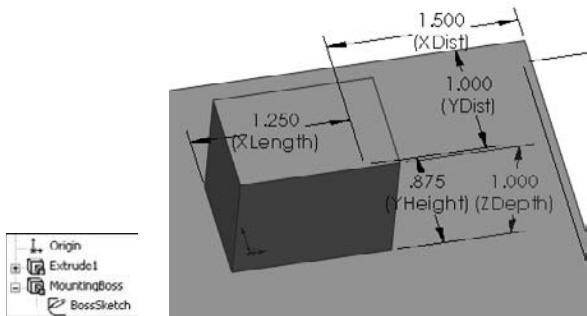
First, you need to create a rectangular extrusion. The size should be bigger than the feature that goes on it and representative of the face of the end part onto which this feature will typically be placed.

Next, in beginning to create the features that you want to reuse, it is very important that you pay attention to any references outside of the sketch; these include absolutely *anything* outside the active sketch, such as the sketch plane, references to edges, the Origin, other planes, other sketches, and axes. Although these references are allowed, each reference to anything that is not already part of the library feature must be reconnected when you place the feature on a new part. The ideal situation is obviously a single drag-and-drop, but generally speaking, at least one other step is usually needed. The initial drag-and-drop determines the face for the feature to start from, and from there, you usually need to locate features, either by using relations or dimensions. A concentric relation locates the feature in a single reference selection of a circular edge (although it may also need to be rotated), and dimensions typically require one dimension in the X dimension and another one in the Y dimension.

Figure 18.8 shows the base feature and the first feature of the library feature. The only relations between the sketch of the library feature and the base feature are the sketch plane and the two dimensions.

Creating a library feature

Note

Notice that names have been assigned to all the dimensions, sketches, and features. This is because the dimension names all display in the interface. If you look back to Figure 18.4, in the Size Dimensions pane, dimension names make it easy to know which dimension to change, whereas the D1 dimension leaves you guessing as to what it applies. ∎

You should ensure that subsequent features after the first one reference only the first feature of the library feature (which is the second feature in the part). This is not a mandatory requirement, but a helpful guideline. You can make additional references, but they should be limited to the same items that were already referenced if possible. Users who model carelessly or do not pay attention to what they are doing typically have trouble making library features that function and are easy to use. Successful modeling of library features is like planning a strategy in a game of chess.
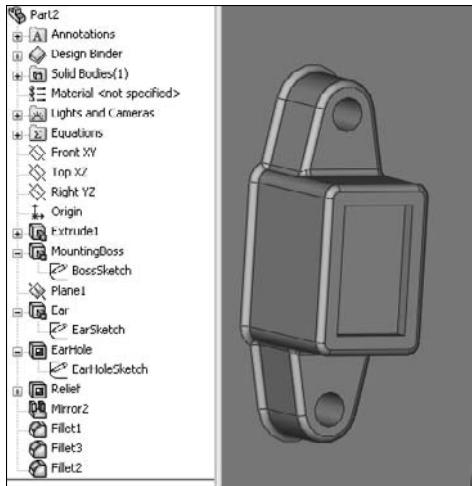
Now you can add the second extruded feature, being careful to reference only geometry that is going to move with the library feature. Figure 18.9 shows the newly added feature. If you would like to follow along as I detail how this feature is built, you can open the part from the CD-ROM under the filename `Chapter 18 First Library Feature.sldlfp`.

FIGURE 18.9

Adding the next feature to the library feature

Notice that a plane has been added. The plane is made to only reference geometry that is internal to the library feature; it is perpendicular to an edge at the midpoint, which simultaneously locates and orients the plane correctly to enable it to be used to mirror the Ear feature.

Also, notice that the EarSketch uses the same face reference from the base feature. This will appear in the Reference list as a single reference.

Saving the library feature

You can use two methods to save a library feature. You can either drag-and-drop it into a Design Library folder or use the Save As method. Because Save As is a little more common, I describe it first.

The first step in saving the library feature is to select all the features in the FeatureManager that are intended to be a part of the library feature. Collapse the features first so that the sketches belonging to features are not selected. If the sketches are selected, you may get a warning message saying that no selected features can be used in the library feature. Do not worry; the sketches still will be included.

Tip
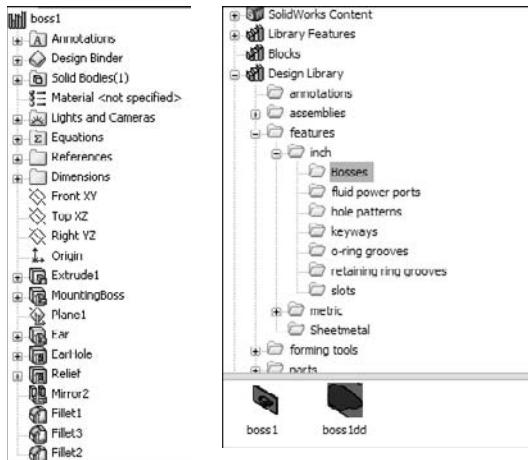
Remember that you can Ctrl+select individual features, Shift+select a range, or click-and-drag a selection box in the FeatureManager to select multiple features. Also keep in mind that if you do not select a feature (other than the base feature), then it will not be placed into the part when you insert the library feature. If there were any relations to the omitted feature, they may display as errors or warnings when you place the feature. ■

With the features selected, click File, click Save As, and under Files of Type drop-down list, select the `*.sldlfp` file type. Browse to the Design Library folder and save the part. Figure 18.10 shows the FeatureManager of the finished library feature.

The finished library feature part

Display of the Library Feature icon

During the Save As process, a new folder was added to the Design Library named Bosses, as shown in Figure 18.10. Notice the new icon for the library feature in the lower window. You may notice that some of the default library features saved in the Design Library have a bluish background. This occurs because of the SolidWorks viewport background color, which you can set by choosing Tools ⇨ Options ⇨ Colors. Even if you never see that color because you are using a gradient background or a scene, SolidWorks still uses the color specified by that setting as the background when saving thumbnails and previews. I always set this color to white for this reason, so that document backgrounds in previews do not have the blue color.

You may want to orient and zoom the library feature before saving it so that it displays clearly in the panel. One of the techniques that I like to use is to make the base feature a different color than the library feature itself; this helps you to more easily determine what is the geometry that will be transferred and what is just dummy material.
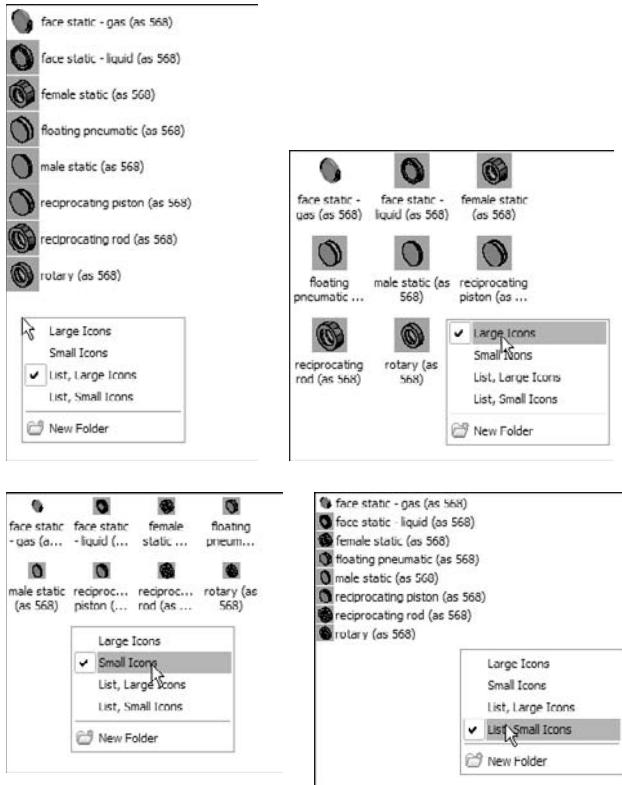
Tip

Figure 18.11 shows various settings for displaying the icons in the Design Library. Which one you select will depend on your screen resolution, the number of icons that you want to display, and the quality of the preview images. ∎

FIGURE 18.11

Display modes for the Design Library
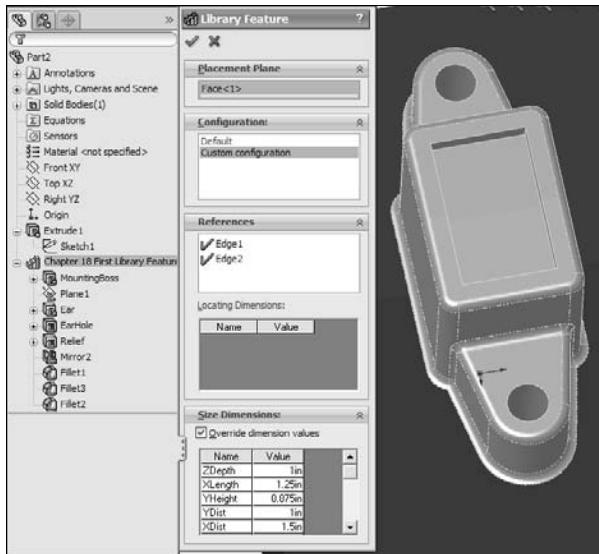
The real test for a library feature comes when you actually use it. This feature is re-created perfectly on the new part, but I have noticed one problem. When the feature was placed, it was 90 degrees away from the orientation that I wanted it to be in. It seems that the only way to make the feature rotatable is to create it with parallel and perpendicular relations rather than horizontal and vertical ones, so that one of the references can act as a rotation reference. Figure 18.12 shows the completed library feature placed on a part.

Once you place a library feature on a part, it can be edited unless you select the Link to library part option in the Configuration pane, in which case the feature is driven externally from the `*.sldlfp` file. The Link to library part option is only available when the feature is first placed; it is not available when you edit an existing library feature in a part.

One of the available RMB menu options is to dissolve the library feature so that all the constituent features become regular features in the main part. Doing this on a configured library feature will destroy the configurations.

The completed library feature placed on a part

Creating a library feature from an existing part

When creating a library feature from an existing part, you use essentially the same process, but it is somewhat more difficult to achieve the correct results. It is best to remove all the features that do not either form the base feature or go into the library feature itself. This can cause many broken references. Most parts are not modeled with creating a library feature in mind, and creating library features with effective references, as I mentioned earlier, is like a chess match and requires some forethought. It may be better to use a different technique, such as creating a new part with only the base feature. You can then Ctrl+drag the desired features from the existing part to the new part, set up the rest of the library feature, and save it with a `*.sldlfp` extension.
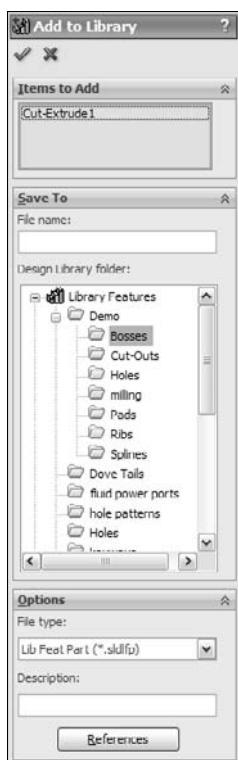
You can also create a library feature by dragging and dropping, although there are some limitations with this technique that seem to override the convenience. However, there is a workaround for the biggest limitation. If you select faces from features and drag them into the lower Design Library window, then an Add to Library PropertyManager interface appears to enable you to start creating a library feature. The Add to Library PropertyManager interface is shown in Figure 18.13. You must select the features from the flyout FeatureManager. This is the source of one of the limitations. In the example, the plane cannot be selected by this method. The work-around for this is to complete the feature without the plane, right-click the icon in the Design Library window, and then select Open. With the library feature open in its own window, right-click the plane feature and select Add to Library; that individual feature is then added.

The Add to Library PropertyManager interface

Adding folders to the library

You can add folders to the library in two ways, either by right-clicking in the Design Library window and selecting New Folder or by using the Windows Explorer interface. Another RMB menu option is Add Existing Folder, which enables you to add a folder from another location to the library. The folder is not moved or copied, but a shortcut is added to the Design Library and the contents appear in the lower pane.

Tip

After a library feature has been edited, or folders or documents have been added to the library using Windows Explorer, you can press F5 in either the lower or the upper window to update the display for that window, or use the Refresh icon located at the top of the Task Pane. ■

Locating and Internal dimensions
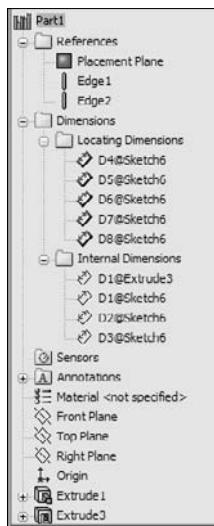
When you create a library feature, SolidWorks adds two folders to the FeatureManager: the References folder and the Dimensions folder. In turn, the Dimensions folder has two more folders: Locating Dimensions and Internal Dimensions. All these folders are shown in Figure 18.14.

The References folder shows the entities you used to reference the library feature. These are the features you are prompted to select upon placing the library feature in a new part. No additional work on your part is required with this folder.

The Dimensions folder lists all the dimensions in the library feature. Some of these dimensions are used to locate the feature to the dummy block and some are meant to size the feature itself. When you make a library feature, it is recommended that you separate the locating features from internal features by dragging the dimensions into the separate folders. The Locating Dimensions can be changed while placing the library feature, but the Internal Dimensions cannot be accessed. You might want to limit user access to some dimensions if you have standard tooling for the library feature that you are creating.

FIGURE 18.14

References, Locating dimensions, and Internal dimensions in a library feature

A function exists within SolidWorks parts called Edit Dimension Access. You can access it by right-clicking the part icon at the top of the FeatureManager. This option enables an administrator to limit the dimensions that users can change. Unfortunately, this option is not available in library features, although it would be very helpful there.

Understanding Dissection

Dissection is a process that SolidWorks goes through by which it examines all the parts on your computer and makes Design Clipart of the sketches and features. People who might get some use from this function are those who have not modeled a lot of parts and will tend to reuse the same features over and over again. The types of data it will try to recycle for you are extrudes and cuts, sketches, blocks, and tables from drawings.

The first interaction most people have with this function is learning how to turn it off. If you notice your computer starting to run small SolidWorks windows in the background starting at about 11 pm daily, you may have Dissection turned on. To select or deselect Dissection, choose Tools ➪ Options ➪ Search. Dissection is connected to SolidWorks Search, another function many users prefer to deselect.

Tutorial: Working with Library Features

This tutorial guides you through customizing a Hole Wizard hole to use as a specialty library feature, then storing it in the library, editing it, and placing it in a part. Follow these steps:
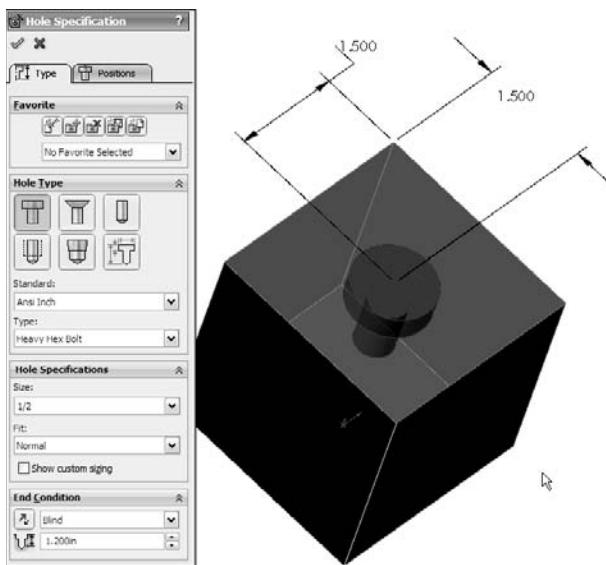
1. **Open a new part, and create a rectangular base feature, about three inches high by three inches wide and three inches deep.**

2. **Pre-select a flat face and start the Hole Wizard.**

3. **Create a counterbored hole for a Heavy Hex Bolt, ½-inch, Normal Fit, Blind, 1.2 inches deep.** Locate the hole with dimensions from two perpendicular edges, as shown in Figure 18.15. Click the green check mark icon twice to accept the hole settings.

4. **Turn on the setting at View ➪ Dimension Names.**

5. **Double-click the counterbored hole feature in the FeatureManager to show the dimensions.** Make sure Instant3D is unselected for the next step.

6. **Click one of the dimensions that you created to locate the center of the hole and rename the dimension in the Dimension PropertyManager (Primary Value panel) using names that will have meaning when you place the dimension, such as XDir, or YDir.** Do this for both dimensions.

7. **Edit the second sketch of the hole.** Figure 18.16 shows what the sketch should look like before and after the edit.

Caution

Do not delete any of the named dimensions in a normal or revised Hole Wizard hole. SolidWorks has a checking mechanism that looks for these names and will display an error if any of the named dimensions is not there. If there is no use for the dimension, it still has to be there, although it does not need to be used for its original use. You could rename another dimension with the name or simply dimension the length of the centerline or an otherwise unused construction line. It does not matter about the function of the dimension, as long as there is a dimension with that name in the sketch. ■

FIGURE 18.15

Placing a hole
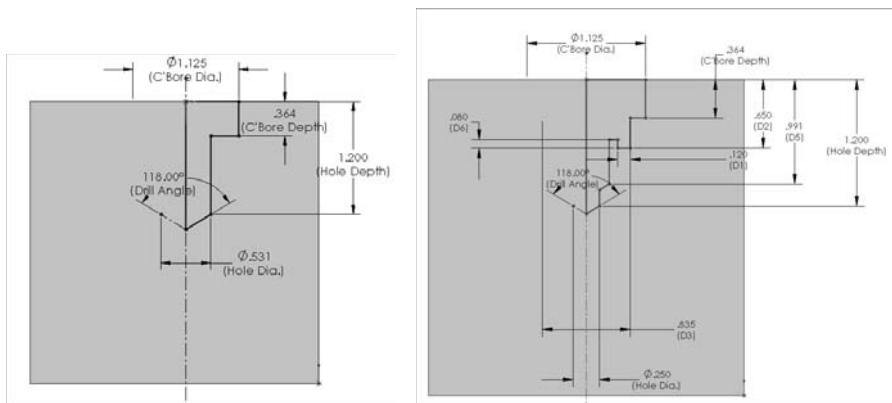
Tip

You should also name any new dimensions that you may want to change. These dimensions will have more meaning when you are placing the feature if they have names. Be aware that the two angled lines are parallel and equal length. This will enable you to get a fully defined sketch. ■

FIGURE 18.16

Reconfiguring the hole

Tip
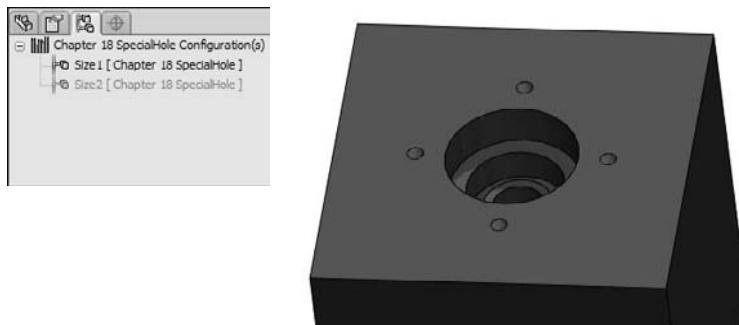
Remember that to get the diameter dimensions shown in Figure 18.15 (instead of radius dimensions), you must use the dimension tool to select the centerline (construction line) and the line or endpoint on one side, and then move the cursor to the other side of the centerline to place the dimension (the order of selection does not matter). When the cursor crosses the centerline, the dimension will display as a diameter instead of a radius. ■

8. When you are done editing the sketch and renaming dimensions, exit the sketch.

9. Click the CBORE feature twice, or click it once and press F2 to rename it as SpecialHole.

10. Pre-select the same flat face that the first hole feature was placed on and then start the Hole Wizard again.

11. Place a #8-32 tapped hole, accept the default depth, and specify a center-to-center distance of .75 inches between it and the SpecialHole in the Horizontal or Vertical direction. Rename the radial dimension as **MountRad**.

12. Using a cylindrical face of the SpecialHole, make a circular pattern of the new tapped hole, creating a total of four instances of the tapped hole. Make the SpecialHole Feature red and the tapped hole and pattern yellow.

13. Split the FeatureManager window into two by using the splitter bar at the top. Change the lower panel to the ConfigurationManager.

14. Rename the Default configuration Size1.

15. Create a new configuration called Size2. Double-click the SpecialHole feature and change the dimension named C'Bore Dia to 1.5 inches. Be sure to change to This Configuration Only using the drop-down menu.

16. Make a dimension change for the MountRad dimension to 1 inch. The results to this point are shown in Figure 18.17.

FIGURE 18.17

The results after Step 16

17. Auto-create a design table by choosing Insert ⇨ Design Table and then selecting the Auto-Create option. Edit the design table to look like Figure 18.18. Hide or delete extra columns that don't match the data shown.

The design table for the SpecialHole feature

Tip

Remember that to fill the first two columns up to Size5, you can make the two-by-two selection of the Size1 and Size2 entries in the first two columns and pull down the handle in the lower-right corner of the selection box until the appropriate boxes are filled. ■

18. Manually fill in the C'Bore Dia values, but in cell D3, type the equation =C3/2+.25. Then use the same Fill technique to populate cells D3 to D7. Click outside of the Design Table to close it.

19. Test the configurations to make sure that they all work.

20. In the Design Library, browse to a folder where you would like to put this library feature. (Make sure that it is not used by assemblies or sheet metal forming tools.) Click a face created by the SpecialHole feature and drag the feature into the lower pane of the Design Library. The Add to Library PropertyManager should appear on the left.

21. Although you selected a feature and dragged it into the library, the Items to Add field appears blank. Select the SpecialHole, tapped hole, and circular pattern features using Ctrl+select for multiple selections, either through the detachable PropertyManager, the split FeatureManager, or the flyout FeatureManager. Selecting the features from the graphics window does not work.

22. Position and zoom the view of the part so that when it is saved, you see a good pre-view of the library feature. Also, if you have not changed your background color from blue to white, this would be a good time to do so.

23. **In the Save To pane, make sure that you select the correct folder, then fill in a file-name and click OK.** Figure 18.19 shows the completed PropertyManager interface for this function.
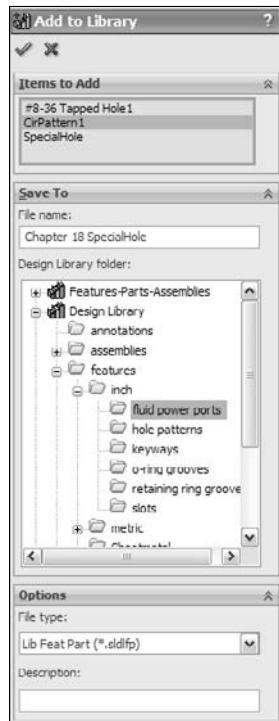
Tip

You may notice that there are two library entries in the window. This is because an additional path has been added in Tools ⇨ Options ⇨ File Locations ⇨ Design Library. ■

24. **If the new library feature does not appear in the Design Library, then click in the Design Library and press F5.** If you do not like the way that it displays, then right-click in a blank space inside the lower library window and select one of the other three options.

25. **To edit the preview image of the feature, right-click the feature in the Design Library window, select Open, reposition or zoom the view, and save it.** Click in the Design Library and press F5 again.

FIGURE 18.19

Saving the library feature

Tip

It is recommended that when placing a library feature, you should close the original library feature window. The workflow proceeds much more smoothly if the part is closed before you use it. ∎

> **26.** **Open the part from the CD-ROM called** `Chapter 18 Tutorial Blank.sldprt`. If you would like to examine the version of the SpecialHole part that I created, it is saved as Chapter 18 – SpecialHole.sldlfp. Notice that this is a library feature part file, not a regular SolidWorks part file.
>
> **27.** **Drag the SpecialHole library feature from the Design Library onto the face of the blank part.** Place the feature near the squared-off end. Select a configuration from the list in the PropertyManager.
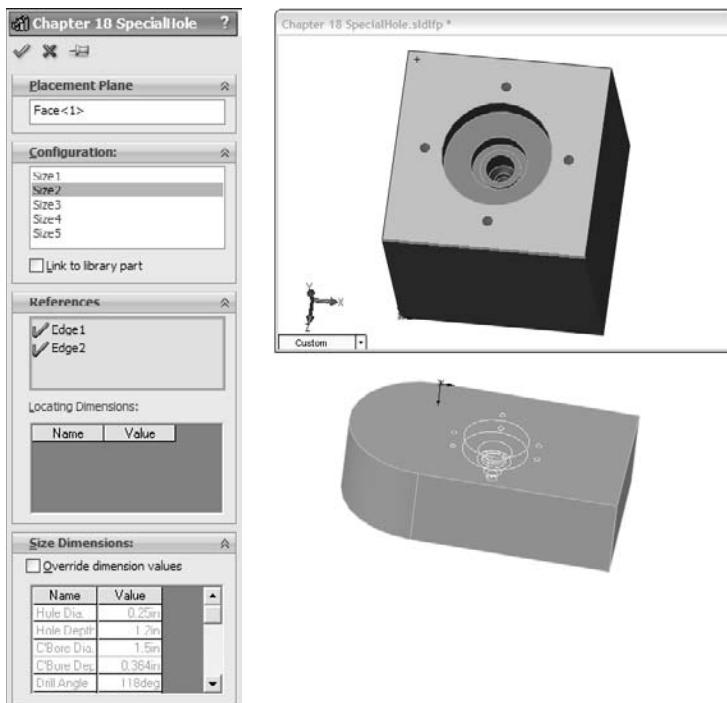
Tip

Although there is no prompt, when the Library Feature interface hesitates and there are configurations in the library feature, it is waiting for you to select a configuration. A prompt actually does exist, but it appears in the lower-left corner of the screen on the status bar in a tiny font and most users probably do not notice it. ∎

> **28.** **Try to orient the part in the same way that it appears in the preview window, as shown in Figure 18.20.**

FIGURE 18.20

Orienting the part and selecting references

29. Select edges on the Blank part that correspond to the preview window. Click OK to accept the placement of the feature.

30. Double-click the SpecialHole feature and change the X and Y placement dimensions to place it one inch from the edges in both directions.
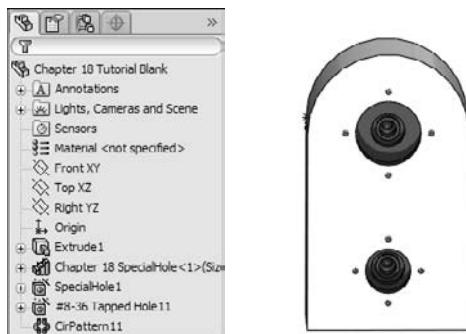
Note
You may remember from Chapter 10 that library feature configurations cannot be controlled by part configurations. In order to show different library feature configurations in different part configurations, you need to suppress one library feature and insert another. This is the best available work-around. It may be time to visit that enhancement request site again. ■

31. Place another library feature onto the blank part. Select a configuration, and click the green check mark icon without selecting edges for the references.

32. Notice that the feature in the FeatureManager appears with an exclamation mark. If you investigate the cause of this, then you can see that the two dimensions that should be attached to edges are dangling because you did not select the references while placing the library feature. This was done on purpose to show a different technique.

33. Expand the library feature and the SpecialHole feature, and edit the first sketch in the Special Hole. This is the placement sketch. Delete the two dimensions that appear in dangling colors.

34. Add a concentric sketch relation between the placement point and the arc edge of the Blank part. Exit the sketch. The error message should now be gone and the hole should now be placed in the center of the arc.

35. Right-click the second library feature and select Dissolve Library Feature. Figure 18.21 shows the finished part and FeatureManager. Good job!

FIGURE 18.21

The finished part

Note

When you dissolve a library feature, you lose any access to any configurations. Some users insist on dissolving every library feature so that they can see regular features in the FeatureManager. This technique may also be useful if you would like to reorder some of the individual features within the library feature. ∎

Summary

Library features are very useful in automating frequent design tasks. They are easy to create, store, apply, and automate. Setting up the features for the most flexibility often takes careful planning and attention to the detail of the references that you use. The more data you reuse, the more time you will save by automating and centralizing your libraries.

Using Smart Components

S mart Components is functionality that can save you a lot of time; the more standard parts or assemblies with associated part-level features that you insert into your assemblies, the more time it can save you. Like anything else of much automation value, it does require some setup, but the setup is a one-time thing.

Smart Components are parts or assemblies that you can place into an upper-level assembly and that carry with them mounting features and hardware (cut-outs, mounting holes, and even fasteners). Smart Components are configurable and can automatically size themselves on cylindrical parts.

Understanding Smart Components

A Smart Component can comprise several elements:

- A single part or an assembly that may use size configurations
- A configurable library feature that usually serves as mounting holes or a viewing window for the Smart Component
- Associated hardware that may also be driven by size configurations
- A training assembly that is used to define the Smart Component

Some minor limitations exist, as you might expect:

- A Smart Component part cannot have references that are external to the Smart Component group of which it is a member

- When placed in the assembly, the associated library feature can only affect one component
- The associated library feature is limited to one of several feature types:
  - Extruded or revolved cuts or bosses
  - Hole Wizard holes
  - Simple hole features

The setup time for Smart Components can be significant for the first one or two that you create, especially if you choose to make use of the auto-size option. The complexity of the setup depends mainly on the number of configurations and configured parts that you use. The auto-sizing function takes the most time to set up because it requires matching configurations, and the auto-size table takes a while to manage, especially for multiple parts. Still, if you end up placing a given part with associated features and other components many times manually, or you have others in your group that do it, this technique can save you and your team a lot of time.

Using Smart Components
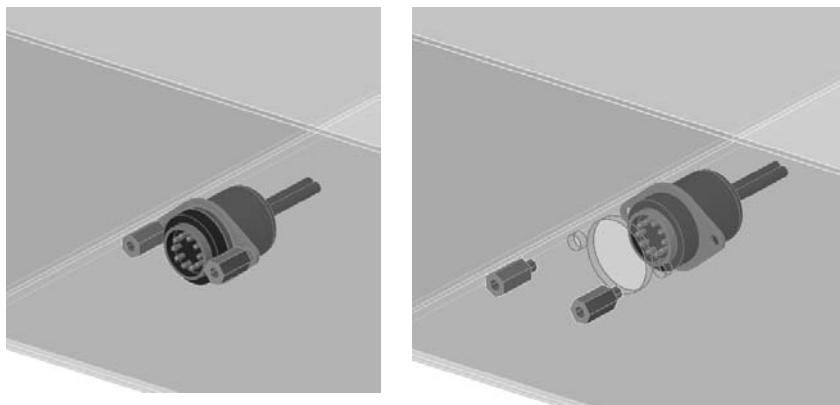
Figure 19.1 shows a simple assembly. It took approximately 20 minutes to model all the parts, set up the Smart Component, and test it in an assembly. This example does not use auto-sizing, but it does use a library part, an in-context feature, and two instances of a single hardware piece. This is an excellent example of Smart Component functionality because it is fast to create and fast to apply and saves you some time whenever you use it.

FIGURE 19.1

A simple Smart Component

Getting started with a simple Smart Component

In this assembly, you first place the electrical connector part in the assembly, mate it in place, and then apply Smart Components. You can apply Smart Components by clicking the Smart Component icon that appears on the part in the graphics window when you select it. SolidWorks then prompts you to select the inside and outside faces of the sheet metal part (the hardware references the outside and the cut-out feature references the inside). SolidWorks then creates the cut-out as an in-context feature that it places in the sheet metal part.

When you create the Smart Component, a new folder is added to the FeatureManager of the component. This folder contains all the required information about the other elements, such as the in-context feature, any other parts that go with the Smart Component, the "training assembly" location, and the face references to locate everything. Figure 19.2 in the image to the left shows this folder in the connector part that is used in this example. The image to the right shows what is added to the assembly FeatureManager when you add a Smart Component. The only thing that existed in the design tree shown in Figure 19.2 before the Smart Component was the Test Box sheet metal part.

A star appears on the part symbol at the top of the FeatureManager, indicating that the part is a Smart Component. You can place this Smart Component by following these steps:

1. **Create an assembly, and add the target part to it.** The target part is the one that the Smart Component will be mated to and the one that will have the in-context cut-out inserted into it. In this case, the target part is a sheet metal box.

Caution

It is a good idea to save the assembly before you add the Smart Component to it. If the Smart Component is placed before the assembly is saved, the assembly has a tendency to forget that it has not been saved, and bumps the in-context feature to out-of-context when the name is changed from whatever the default name is (for example, `Assem1.sldasm`) to the name that you assign to it. ∎

2. **Put the Smart Component into the assembly.** You can do this in the same way that you would add any normal part, including from the Design Library. If you use a part frequently enough to make it into a Smart Component, then you are probably going to want to have it in the Design Library for quick access. In fact, you can add a Smart Component to an assembly without using any of the Smart Component options.

3. **Mate the Smart Component in the assembly.** In this case, it was done with a face-to-face coincident mate and a pair of distance mates.

4. **Next, apply Smart Components by clicking the Smart Component symbol on the part.** If this symbol does not appear, then select the part in the FeatureManager. Figure 19.3 shows the Smart Component symbol on the part. If you have inserted many instances of a Smart Component, then each instance has the option to apply the Smart Component features and associated components.

FIGURE 19.2

The Smart Component folder in the connector part

Feature tree
of Smart Component

Feature tree of assembly where a
Smart Component has been used
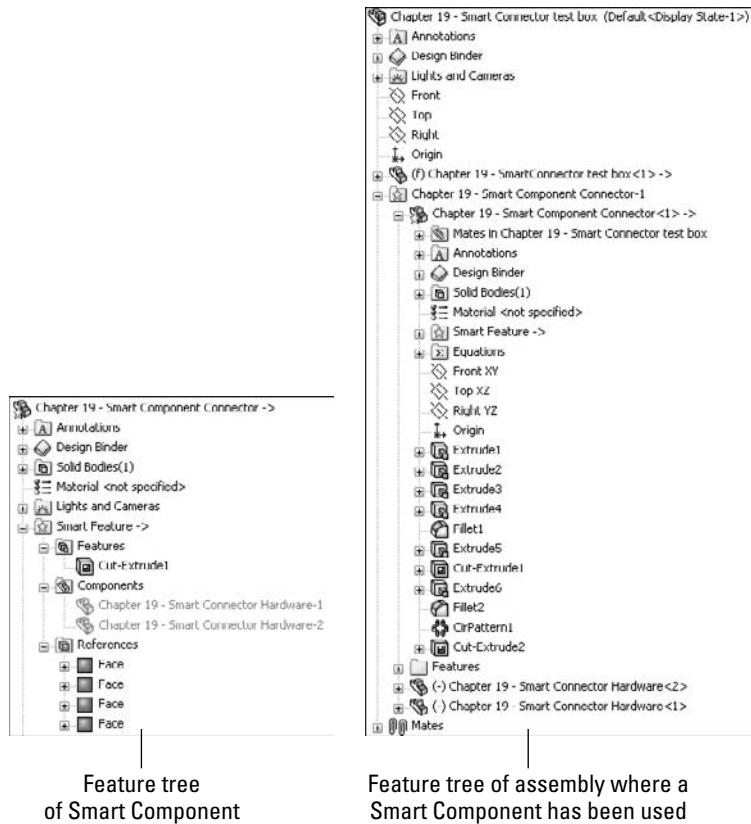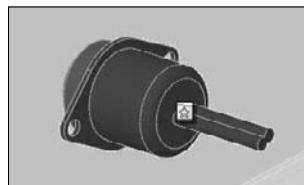
FIGURE 19.3

The Smart Component symbol on a part
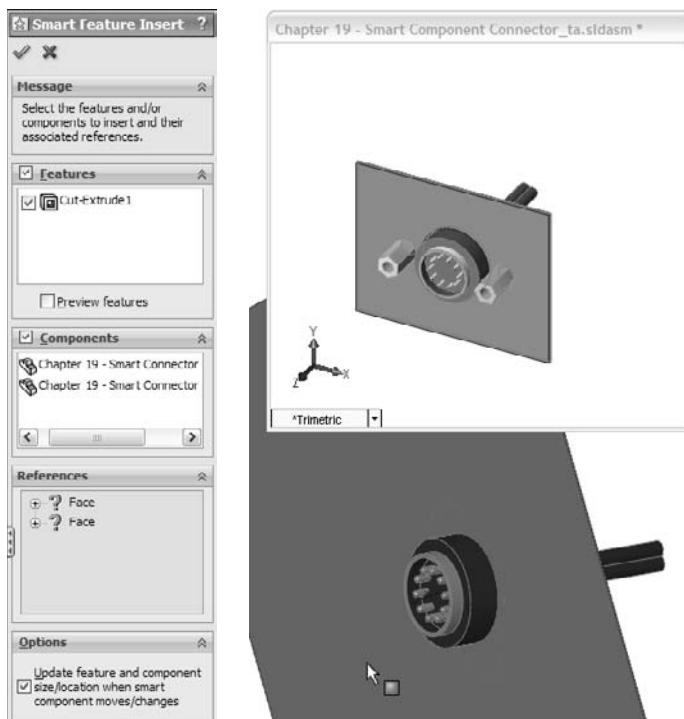
   5. At this point, an interface similar to that of the Library Feature interface appears, with the small prompt window and a box for selecting references, as shown in Figure 19.4. After you select the references and click the green check mark icon, the Smart Component, as well as the Smart Feature (in-context feature) and associated hardware components, are placed, and the job is done. You can rotate the part in the small preview window to get a better look at the part.

Auto-sizing Smart Components

Auto-sizing is the capability of a Smart Component to automatically select a size from a list of configurations based on the size of geometry onto which it is being dropped. At this time, the only shape that can be auto-sized is the cylindrical shape.

FIGURE 19.4

The interface for adding the Smart Feature and additional components of the Smart Component

Figure 19.5 shows the effects of auto-sizing. Notice the two shaft holders. These are two instances of the same part, using different size configurations. When you drag the Smart Component over the small end of the stepped shaft, the configuration corresponding to that shaft size appears. As you drag the part along the shaft and the shaft diameter increases, the next-larger Smart Component configuration appears. This is part of the functionality of Smart Components. Each configuration of the Smart Component is set up to fit onto a range of shaft diameters. If the diameter of the shaft is outside of the range or between sizes, then the Smart Component is not applied.

FIGURE 19.5

A Smart Component with auto-sizing

Sizes are governed by a configurator table, which looks similar to a design table, but works somewhat differently. The configurator table relates the configurations of the Smart Component to configurations of the individual parts, which may also change size with the Smart Component. This serves as a subset of the function of a design table in an assembly, assigning part configurations to assembly configurations. Figure 19.6 shows a sample configurator table made for the assembly shown in Figure 19.5.

FIGURE 19.6

A configurator table

When you look at this table, you begin to understand why creating auto-sizing Smart Components is much more involved than the first example in this chapter. The configurations of the Smart Component are listed to the left, and the individual part configs can be selected in each cell from a drop-down list of all available configs for that part. There is no way to set configs for multiple components at once, nor is there a copy-and-paste function. These shortcomings combine to make this format somewhat less user-friendly than an Excel-based design table.

Most notable are the Minimum and Maximum Diameter columns to the right. These columns supply the parameters that make the auto-size function work. While the range of sizes used here is too large for real-world design (+/– .050 inches), it serves to convey the idea. More important, SolidWorks understands that mating sizes are not always exactly equal, and the ability to use a range rather than exact values accommodates this very nicely, although it can be tedious to set up.

Another aspect of the setup shown here is that it uses Toolbox parts. If you want to use the auto-size functionality, then you need to be using configurations for Toolbox parts. You should prebuild all the needed configurations and ensure that they are always available.

Making Smart Components

The most important point to remember about Smart Component setup is that you need to do it only once for each Smart Component. The second most important point is that the first setup is the most difficult. After that, subsequent setups become much easier to create. Adding components to the Smart Component is not so time-consuming unless the additional components are also configured and also auto-sized.

Smart Components must contain at least one associated component and one in-context feature, or have the configurator table filled out and functional. If you try to create a Smart Component from a stand-alone part, then nothing happens; the Smart Component interface simply closes because there is nothing for it to do. You may combine all three elements (associated component, in-context feature, and auto-size), but you must have at least one element.

Getting started with a simple Smart Component

Because I have already used the electrical connector shown in Figure 19.1 to demonstrate the insertion of a Smart Component, I will use it here to demonstrate how to create one.

All that you need to make a Smart Component with an associated Smart Feature (in this case, a cut-out and mounting holes) and mounting hardware (in this case, two stand-off screws) is the part itself. The part can even be an imported part (a Smart Component made from dumb geometry).

From the CD-ROM for Chapter 19, open the file named `Chapter 19 – Connector Start. sldprt`. This part is shown in Figure 19.7.

An electrical connector part
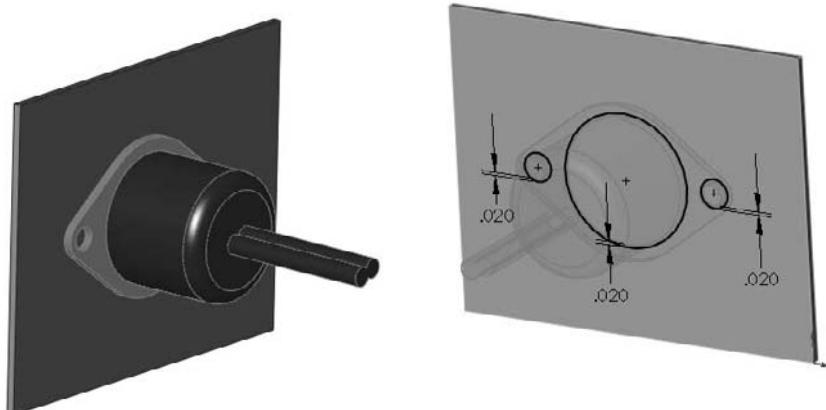
There is nothing special about this part. I modeled it in SolidWorks using standard features, and there are no configurations or special features. It could have been downloaded from 3D Content Central. It represents an electrical connector that may be mounted in a sheet metal electrical enclosure.

The first step in setting it up is to create a mock assembly with a dummy part representing the sheet metal box. The part does not need to be complex or even sheet metal for that matter; it just needs to be close to the thickness that you would expect the Smart Component to be mounted to. The assembly is called a *training* assembly, not because you are learning how to make a Smart Component, but because you are training the Smart Component to be *smart*.

1. **Make a simple rectangular part, approximately 4 inches square and about .06 inches thick.** Save the part to your hard drive. Give a name to the part so that it is clear that it belongs to this training assembly.

2. **Place the rectangular dummy part into a new assembly, with a name that is both unique and identifiable.**

3. **Put the connector into the assembly.** Mate the part so that the flange is flush with the rectangular piece. Also use distance mates to locate the connector planes from the edges of the part, similar to Figure 19.8 in the image to the left.

4. **Edit the dummy part in context (right-click the dummy part in the assembly and select Edit Part), and offset edges of the connector part to extrude a cut, as shown in the image to the right in Figure 19.8.** Offset the two mounting holes and the area around where the connector will stick through the sheet metal by about .02 inches.

FIGURE 19.8

Placing the connector on the dummy part

5. **Exit Edit Component mode and add two instances of the part named** `Chapter 19 – Smart Connector Hardware.sldprt` **to the assembly.** Mate the hardware part to the in-context hole, making sure that it goes to the outside thickness of the dummy sheet metal part.

Note

I have spent a fair amount of time trying to convince you that it is a best practice to avoid mating parts to in-context features, and yet here I tell you to do exactly that. Another thing I have tried to stress is that best practice suggestions are more like guidelines. If you are having performance problems with an assembly, then this may not be a great technique to use. However, sometimes there is a price to pay for sophisticated functionality, and if you think that your design can afford the price and will benefit from this functionality, then you should use it. ■
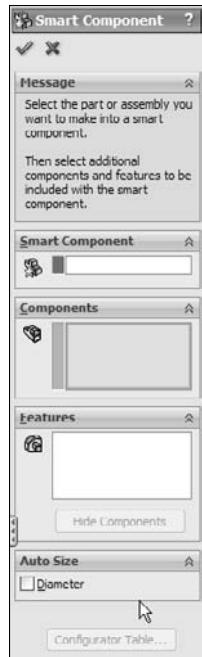
6. **Now that everything is in place, click the Make Smart Component tool on the assembly toolbar.** If the button is not there, you can add it to the assembly toolbar by choosing Tools ⇨ Customize menu, or by choosing Tools ⇨ Make Smart Component. The resulting interface is shown in Figure 19.9.

7. **In the Smart Component selection box, select the connector part.**

8. **In the Components selection box, select the two hardware components.**

9. **In the Features selection box, select the in-context feature from the dummy part.**

10. **You are now finished setting up the Smart Component.** Click the green check mark icon to accept the changes and exit out of the PropertyManager, and save the file.

The Smart Component PropertyManager interface

Creating an auto-sizing Smart Component

The simple Smart Component took about 20 minutes to model and set up. That is not too bad for a feature that you might use a lot. The benefits are somewhat modest, placing three components and a feature.

However, when it comes to the auto-sizing example that is shown next, the benefits are more extensive. A total of seven individual parts are placed (including Toolbox parts) — three of which are automatically sized, depending on the geometry into which the Smart Component is dropped — and an in-context feature is added.

To begin, open the part from the CD-ROM named `Chapter 19 – Clamp Start.sldprt`. Notice that this is a multi-body part. There is no special knowledge that you need to have about multi-body parts to complete this task. Multiple bodies are discussed in detail in Chapter 26.

1. **With the Clamp Start part open, notice that several configurations already exist.** If you click through the configurations or examine the design table in the part, you can see that various dimensions change. The primary dimension that changes is the diameter of the hole, and this change drives the diameter of, and distance between, the mounting holes.

Note

You can only drive auto-sizing by cylindrical geometry. ■

2. **Part of the Smart Component definition includes applying a Mate Reference to the part so that the big hole automatically snaps to cylindrical geometry.** Another aspect is that it adds in-context holes that match the mounting holes to the plate. Figure 19.10 shows the assembly that this part is meant to go into. The clamp snaps onto the stepped shaft and adds holes to the plate.

FIGURE 19.10

The assembly where the Smart Component will be used

3. **Open the file named** `Chapter 19 Autosize Training Assembly.sldasm`. This has been prepared to help you get started with the Smart Component training.

Note
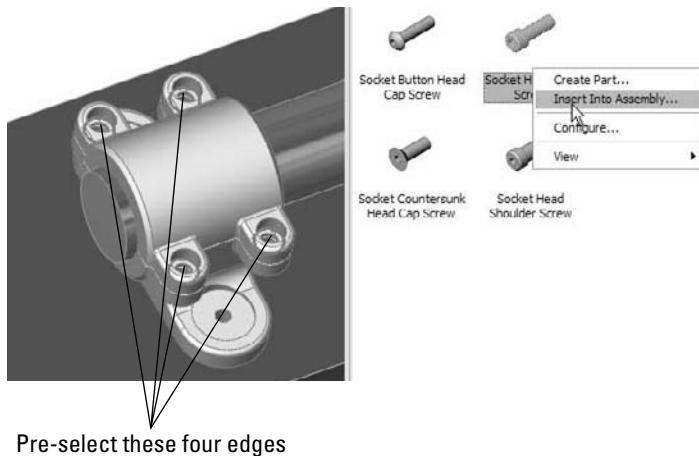
The shaft is not necessary in the training assembly. The training assembly is intended to create the in-context Smart Feature and to create the configurator table. The shaft part has been added here for visualization only. ■

4. **Insert the clamp part into the assembly and mate it concentric to the shaft and coincident to the blue plate.** It does not matter where the clamp sits along the shaft, but it should be fully mated into the location so that it does not slide back and forth. A distance mate from a plane or planar face would be a good choice.

5. **Edit the plate in the context of the assembly, and convert entities from the mounting holes in the clamp to create holes in the plate that align with the holes in the clamp.**

6. **Exit the Edit Component mode.**

7. **Activate Toolbox, select the four holes, as shown in Figure 19.11, and insert Socket Head Cap Screws, ⅜ by 24 by ⅝ inches.** If you do not have Toolbox or choose not to use it, then a part with the correct name and sized configurations is provided on the CD-ROM.

8. **Use the same fastener to place in the mounting holes, using the correct size for the hole.** The length will be set later. Use a default length of 2.25 inches for both mounting holes.

Inserting four screws at once using Toolbox

Pre-select these four edges

Note

Working with the length of the fasteners is not a clean operation in Smart Components. The length is dependent on the thickness of the plate, which is not controlled by the Smart Fastener, nor can the Smart Fastener account for it, except through mates. (Remember that auto-sizing is driven only by a diameter.) Later in this chapter, you will see how the washers and nuts are put in place on the underside of the plate, but the screw length cannot be automatically calculated (unless the screw itself had an in-context relation to the nut). ■

9. **Place washers and nuts on the screws on the backside of the plate.**

10. **When the shaft diameter changes, the hole in the clamp changes to match (within the ranges that will be established).** As the clamp becomes bigger, bigger screws are needed to secure the clamp and the holes grow further apart. Bigger screws mean additional configurations for the screw, washer, and nut parts. Remember that the configurations do not necessarily exist. I would not count on a Smart Component working if this meant that Toolbox had to create new configurations.
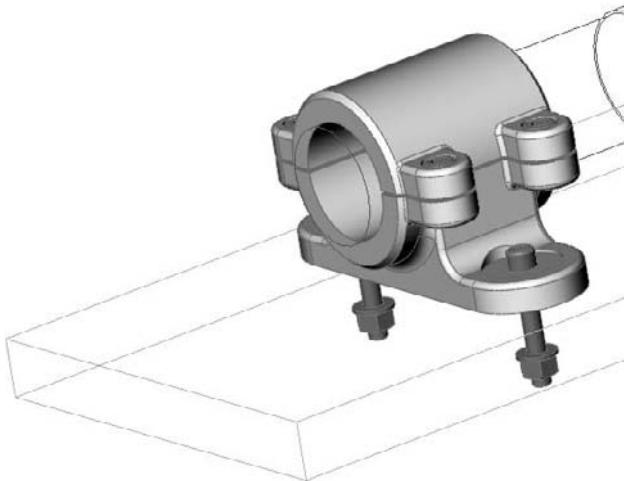
    To create this example, I have pre-populated the Toolbox parts with all the configurations needed for the range of sizes involved with this Smart Component. When you make your own Smart Components, you will have to do the same thing if you intend to use auto-sizing with Toolbox parts. The difficulty here is that the configurations include the diameter size of the screw as well as the length, which is unknown until you place the part.

    All you have to worry about in this step is to make sure that the configurations are available and that the screws are placed properly.

11. **Up to this step, you have just assembled the parts as if this were the only time you were going to do it.** The automation of the process comes next. Figure 19.12 shows the training assembly to this point. The shaft and plate are shown in wireframe because they are external to the Smart Component.

The training assembly up to Step 11

12. **This is the point in the previous example where the Make Smart Component command was used, and it is no different here.** Click the Make Smart Component tool on the assembly toolbar.

13. **Figure 19.13 shows the filled in Smart Component PropertyManager.** Activate the Smart Component selection box and pick the clamp part.

    In the Components selection box, select the six screw instances, the two washers, and the two nuts.

    In the Features selection box, select the in-context feature or features that are associated with the Smart Component.

Note
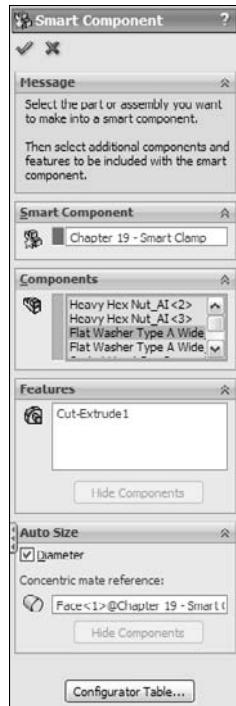
Although the in-context feature can affect only a single part, this does not mean that you are limited to a single in-context feature. In most cases, only one feature is needed, but I am sure that there are situations where more than one would be useful. Also, remember that the in-context features are limited to extruded and revolved bosses and cuts, Hole Wizard holes, and simple hole features. ■

14. **The configurator table is simply a table that enables you to select which component configurations are to be used with which Smart Component configuration.** It looks and works very much like an assembly design table, but it is not Excel-based and every cell must be set explicitly rather than using techniques for mass population or assigning properties to a range of configs, such as you can do with a real design table.

FIGURE 19.13

The Smart Component PropertyManager

Each cell has a drop-down list of all the available configurations for that component. If you have four instances of a single component, then you have to set each instance of each component. Figure 19.14 shows the configurator table for this example.

FIGURE 19.14

The configurator table for the Clamp Smart component

Note

If the configurator table were to ever be as easy to use as, for example, an Excel design table, then Smart Component complexity could increase significantly. The configurator table could even ideally be created from an assembly design table. Instead of a single component with its associated hardware and mounting features, think of larger-scale subassembly attachments. This sort of work is possible now, but it is difficult to do more than a handful of parts. ■

> **15.** **Click OK.** You are now done creating the auto-sizing Smart Component!

Managing file with Smart Components
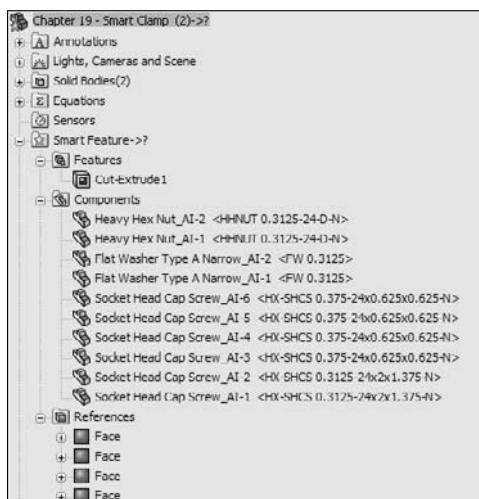
You may expect that with the training assembly, there is an extra burden of file management with Smart Components. This may seem counterintuitive, but in fact, the only file that you need to worry about is the Smart Component itself.

This is not explained very well in any of the documentation, but the Help and every reseller demonstration that I have seen on the topic all recommend that you simply *delete* the training assembly once you are done with it because it is not needed any more. This seems like saying that you should delete all the mates in an assembly or the sketch relations in a part. How do you edit the Smart Component if you delete the assembly in which it is created?

It turns out that all the information to re-create the training assembly is stored in the Smart Component. This includes the in-context feature (which is stored as a library feature) and the locations of any associated components, as well as the configurator table. Figure 19.15 shows a part of the FeatureManager of a Smart Component. As you can see, the in-context feature, the associated components, and the face references are all listed there.

FIGURE 19.15

Part of the FeatureManager of a Smart Component

I was skeptical of this originally and had to test it thoroughly to ensure that deleting the training assembly would not cause any data to be lost, so I can assure you that it works. Go ahead and delete the assembly.
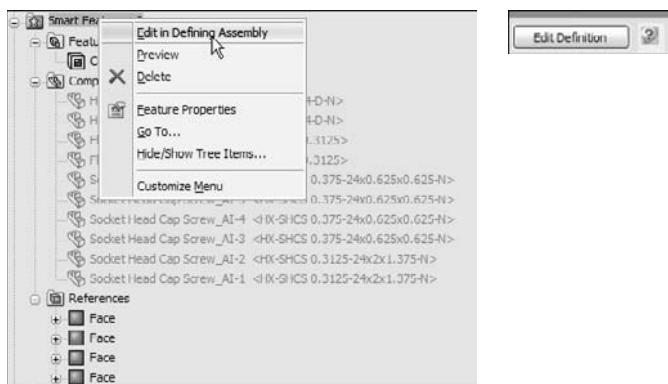
Editing Smart Components

Expanding somewhat on the discussion about whether or not to keep the training assembly file, here is a little exercise that you can try. Make a Smart Component by going through the preceding steps, using the following tutorial or creating one of your own. Just make a simple one with perhaps one associated component and an in-context feature. Then go ahead and delete the training assembly.

With the defining assembly gone, there appears to be no way to edit the setup of the Smart Component. Right mouse button (RMB)-click the Smart Feature folder in the FeatureManager of the Smart Component and select Edit in Defining Assembly, as shown in Figure 19.16. What happens next is that SolidWorks re-creates the defining assembly from the data that is stored in the Smart Component. This assembly is saved in a system temp folder using the name `<Smart Component name>_ta.sldasm`. If the Smart Component uses an in-context feature, it is saved to the temp directory as a library feature file using the name of the dummy part and appending `_lf` to the file-name; for example, `Dummy_lf.sldlfp`.

FIGURE 19.16

Selecting the Edit in Defining Assembly command

The Edit Definition button appears in the upper-right corner of the graphics window. The Edit Definition button is shown to the right in Figure 19.16. If you click this button, the Smart Component PropertyManager interface appears again, enabling you to change the selection of associated components and in-context features, to change the auto-size setting or edit the configurator table.

Thus, all the settings are preserved, and the training assembly exists only as a phantom in a temp directory. Although this appears to be counterintuitive, it works.
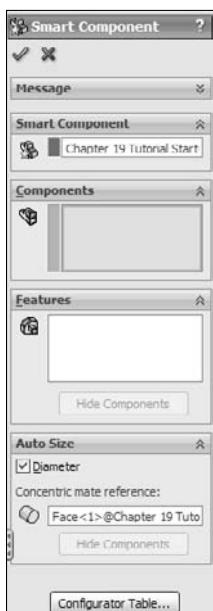
Tutorial: Working with Smart Components

This tutorial guides you through creating a Smart Component that only uses the auto-sizing feature. This enables you to manually create parts that snap to size like Toolbox parts, but without using Toolbox functionality. Follow these steps:

1. **Open the part from the CD-ROM that has the filename** `Chapter 19 – Tutorial Start.sldprt.` This part originally came from Toolbox and already contains a few configurations.

2. **Make an assembly that contains *only* this part.**

3. **Make the part into a Smart Component (Tools ➪ Make Smart Component), and turn on the option to auto-size.**

4. **Select the small diameter of the part as the concentric mate reference.** Figure 19.17 shows the selection.

5. **Click the Configurator Table button, and fill in the table so that it looks like Figure 19.18.** Some configurations are blank. This is because only the rows that have minimum and maximum values are used by the auto-size function. The rest are overlooked.

Selecting the concentric mate reference face

FIGURE 19.18
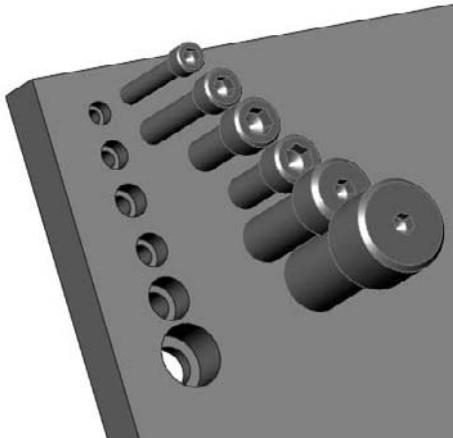
Filling in the configurator table

Chapter 19 Tutor Default	Minimum Diameter	Maximum Diameter
HX-SHCS 0.138-3		
HX-SHCS 0.19-32		
IIX-SIICS 0.19-32		
HX-SHCS 0.19-32	0.1	0.199
HX-SHCS 0.25-20		
IIX-SIICS 0.25-20		
HX-SHCS 0.25-28	0.2	0.249
HX-SHCS 0.3125-		
IIX-SIICS 0.3125-		
HX-SHCS 0.3125-		
HX-SHCS 0.3125-		
IIX-SIICS 0.375-2		
HX-SHCS 0.375-2		
HX-SHCS 0.375-2		
IIX-SIICS 0.375-2		
HX-SHCS 0.375-2	0.25	0.449
HX-SHCS 0.4375-	0.45	0.469
IIX-SIICS 0.4375-	0.47	0.479
HX-SHCS 0.4375-	0.48	0.499
HX-SHCS 0.5-20x	0.5	0.529
IIX-SIICS 0.5-20x	0.53	0.549
HX-SHCS 0.5-20x	0.55	0.579
HX-SHCS 0.5-20x	0.58	0.599
IIX-SIICS 0.625-1	0.6	0.629
HX-SHCS 0.625-1	0.63	0.659
HX-SHCS 0.625-1	0.66	0.699
IIX-SIICS 0.75-16	0.7	0.8
HX-SHCS 0.875-1	0.801	1.109
HX-SHCS 1.25-12	1.11	2
PreviewCfg	0.801	1

6. **Close the configurator table, click the green check to exit the feature, and save the assembly.**

7. **Exit the assembly and, in the part file, save it to a folder in your Design Library.** If you do not know where your Design Library is located, then choose Tools ⇨ Options ⇨ File Locations ⇨ Design Library.

8. **Display the part in the Design Library panel of the Task pane.**

9. **Open the part from the CD-ROM with the filename** `Chapter 19 Tutorial Plate. sldprt.` Place this part into a new assembly.

10. **Drag the Tutorial Start (Smart Component) from the Design Library into the assembly, and move the part over the holes in the plate.** As you drag the part up and down the row of holes, the part changes sizes to match each hole. Figure 19.19 shows all the holes that are populated with the matching Smart Component sizes, as driven by the configurator table. Place the part over one of the diameters.

11. **To edit the configurator table, open the Smart Component part in its own window.** Then right-click the Smart Feature folder and select Open In Defining Assembly.

12. **An assembly that was created from the data stored in the part opens.** Click the Edit Definition button that appears in the upper-right corner of the graphics window.

FIGURE 19.19

Smart Component parts match holes in the part.

13. **Reassign the minimum and maximum diameter values for the ³⁄₁₆-inch and ¹⁄₄-inch configurations to the shortest lengths.** For example, the chart shows the ³⁄₁₆-32x.75-inch configuration to be assigned to a minimum of .1 and a maximum of .199. Move the .1 and .199 values up two cells to the ³⁄₁₆-32x¹⁄₄-inch configuration. Do something similar for the ¹⁄₄-28x1-inch configuration. The edited part of the chart now looks like Figure 19.20.

FIGURE 19.20

The edited configurator table

Chapter 19 Tutor	Minimum Diameter	Maximum Diameter
Default		
HX-SHCS 0.138-32x0.5x0.5-N	0.1	.199
HX-SHCS 0.19-32x0.25x0.25-N		
HX-SHCS 0.19-32x0.4375x 0.4375-N		
HX-SHCS 0.19-32x0.75x0.75-N		
HX-SHCS 0.25-20x0.625x0.625-N	.2	.249
HX-SHCS 0.25-20x1.375x1.375-N		
HX-SHCS		

Tip

You may have difficulty expanding the width of the column that contains the configuration names, thus making it difficult or impossible to read the ends of the long config names. However, like Excel, you can expand the height of the rows, which causes the config names to wrap, as shown in Figure 19.20. ■

Summary

Smart Components can automate the placement of a main component, as well as associated mounting features and components. It can also offer automatic resizing options, depending on the geometry to which it is mated. The setup for Smart Components varies from simple to complex, with auto-sizing causing most of the complexity. Libraries, and in particular smart library parts like Smart Components, can vastly simplify repetitive tasks.

Part V

Creating Drawings

Part V, "Creating Drawings," walks you through the tools you need to make drawings work for you. The main focus is on task automation where possible, and where it supports the end goal. SolidWorks enables you to create a wide range of view types that are all automatically updated with model changes.

Automating Drawings: The Basics

Engineering drawings include a lot of repetitious information from one drawing to the next. The information is not always exactly the same, but it is usually in the same format and of the same type. For example, part drawings always include information about who made the drawing, when the person made it, what the material and surface finish of the part are, and some basic notes that depend on the use of the drawing (manufacturing, assembly, or inspection).

All this information needs to appear consistently on each drawing, every time. However, humans are not always good at following dull routines, which is why we have computers to help with these boring or difficult tasks.

SolidWorks drawing templates and formats enable you to automate some of these tasks. SolidWorks can insert information on a drawing's creator and the materials to be used, and start similar types of drawings from a consistent starting point. Drawing templates and formats also save settings that you may want to reuse.

Comparing Templates and Formats

Simply put, *templates* are collections of document-specific settings and default views saved in the `*.prtdot` (part template), `*.asmdot` (assembly template), and `*.drwdot` (drawing template) file type. In this chapter, I cover the `*.drwdot` file type.

Formats, more formally called "sheet formats," are exclusive to drawing documents, and contain the sheet size, the drawing border-line geometry, and the text/custom property definitions that go with the text in the drawing border. Formats can also include company logo images.

You can save formats in drawing templates; in fact, this is the method that I use and recommend. Using SolidWorks' default drawing templates, the templates and formats are initially kept separate. You specify the size and the format when creating a new drawing from a blank template. However, when the format is already in the template, the size has already been determined, and so the templates end up being saved as sizes. Of course, you can change formats later if you need to use a larger drawing sheet.

Changing existing templates

Can you change templates on existing documents? No. This is one of the most common questions from new users. Perhaps if SolidWorks received enough enhancement requests on this topic, they would be willing to change the software to enable the user to transfer the settings from an existing template to one or more existing documents.

Currently, once you create any kind of document from whatever kind of template, you cannot change the underlying template. However, you can change all the settings, which is for the most part equivalent.

SolidWorks offers custom drafting standards, which provide some of the functionality the ability to swap templates would achieve. You can take a drafting standard such as ISO (International Organization for Standardization) or ANSI (American National Standards Institute), make adjustments to it, and save the standard out to a file that you can distribute to other users. You can change the standard by choosing Tools ➪ Options ➪ Document Properties ➪ Drafting Standard from the menus. You can load and save standards from the same location. More details on what you can actually change within the drafting standard comes later in this chapter.

While *templates* cannot be reloaded, *formats* can be. You might want to reload a format (drawing border and associated annotations) if you have made changes to the information or line geometry.

Maintaining different templates or formats

Different formats must be maintained for different sheet sizes. If you do contract design or detailing work, then you may need to maintain separate formats for different customers. Some people also choose to have different formats for the first sheet of a drawing and a simplified format for the following sheets.
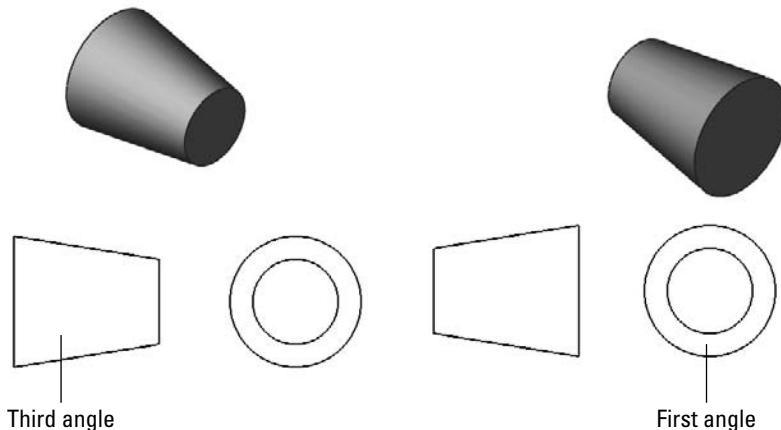
If you put formats on the templates, then you are making separate templates for various sized drawings. Also, separate templates are frequently created for different units or standards because templates contain document-specific settings. I also keep a blank drawing template with a blank format on it just to do conceptual scribbles or to make an informal, scalable, and printable drawing without the baggage that typically accompanies formal drawings.

Caution

SolidWorks can install with default document templates that use different standards. Be careful of the difference between drawings with ANSI and ISO standards, or more importantly, the use of Third Angle Projection versus First Angle Projection. Figure 20.1 shows the difference between a Third Angle and First Angle Projection. Third Angle is part of the ANSI standard used in the United States, while First Angle is part of the ISO standard used in Europe. ■

FIGURE 20.1

Third Angle versus First Angle Projection

Third angle

First angle

If you work for a company that does a lot of work for manufacturing in Europe, then you may have to deal this issue more frequently. The setting that controls the projection angle is not in Tools ➪ Options (where you might expect it to be), but in the Sheet Properties, which you can access by right mouse button (RMB)+clicking anywhere on the blank drawing sheet and selecting Properties.

Creating custom drafting standards

In my experience, in companies that work in the real world, very few companies follow any of the single drafting standards perfectly. Each company seems to have its own interpretation of, or exceptions to, the standards. SolidWorks is coming to grips with this in a practical way. In SolidWorks, you can create your own custom drafting standards, equivalent to the established ISO and ANSI standards. These standards allow you to save all the settings found in Tools ➪ Options ➪ Document Properties to a single standard that you can then transfer to other users.
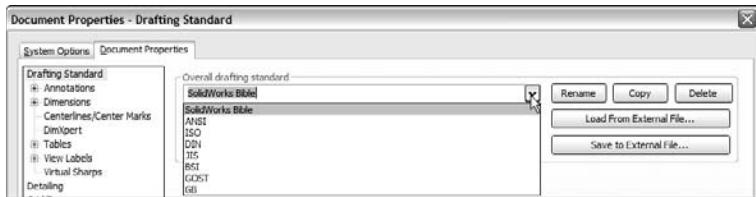
To make your own custom standard, make changes to the various settings for annotations, symbols, dimensions, and so forth, and then go back to the Drafting Standard page of the Document Properties tab, rename the Overall Drafting Standard, and save the standard to a file. I have created a new standard, which is shown in Figure 20.2.

Creating a new customized drafting standard

The drafting standard file type has the extension of `*.sldstd`. If someone else has sent you a standard file, you can read it in to your drawing and assign it, and your drawing will assume all the customized properties.

On the CD-ROM

I have saved a custom standard file and put it on the CD-ROM for Chapter 20. You can load this file into an open drawing by choosing Tools ⇨ Options ⇨ Document Properties ⇨ Drafting Standard and using the interface. ∎

Creating Drawing Formats

Creating drawing formats can be either simple or difficult. Generally, copying existing drawing borders from other drawings imported through DXF (Data eXchange Format) or DWG format is the easiest way to go. Trying to edit an existing border into a different size is usually much more difficult. Adding all the automated annotation information is much easier than editing the lines in the border and title block.

Customizing an existing format

The simple solution is to customize an existing format of the size or sizes you require for your own use. This generally works well, and you can usually finish the task in a few minutes, depending on your requirements. The easiest option is to take the existing SolidWorks sample formats and add a few things such as a company name, logo, and tolerance block to them. You can also use formats from other drawings, editing and saving them out as your own.

Sample formats

The sample formats installed with SolidWorks include ANSI sizes A through E, and ISO sizes A0 through A4. You can probably find enough space on the formats to place a company logo and some standard notes. These templates are located in different directories in Windows XP, Windows Vista, and Windows 7. Choose Tools ⇨ Options ⇨ File Locations to locate the path for your templates.
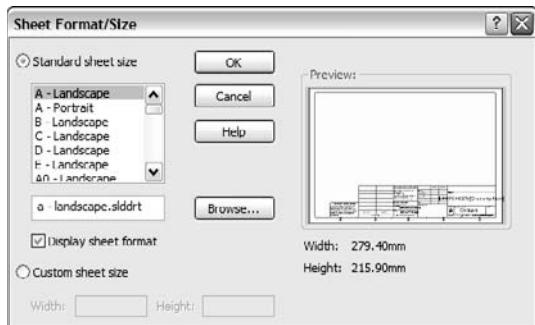
You cannot open a format directly — it must be on a drawing — and, so, to get a closer look at the format, you must make a new drawing using the format.

Note

Templates that have been saved with a format already on them skip the step of prompting you to select a format. This enables you to create new drawings more quickly. If you select one of the default SolidWorks templates, these do not have formats on them, so you are prompted to select a format immediately. Figure 20.3 shows the interface for selecting a format that displays after you have selected the template for a drawing. ∎

Selecting a format

Editing a format

In the drawing, you are either editing the sheet or editing the format. You can think of the sheet as being a piece of transparent Mylar over the top of the drawing border format. In order to get to the format, you have to peel back the Mylar layer. Drawing views go onto the sheet, so when you edit the format, any drawing views that may be there disappear.

To peel back the sheet and gain access to the format, right-click a blank area of the sheet and select Edit Sheet Format. Alternatively, you can also access the sheet format by right clicking on the sheet tab in the lower-left corner of the SolidWorks window. This RMB menu is shown in Figure 20.4. Be careful of the terms here, which include Sheet and Sheet Format. The sketch lines of the format light up like a sketch becoming active, and the "Editing Sheet Format" message appears at the lower-right- corner on the status bar.
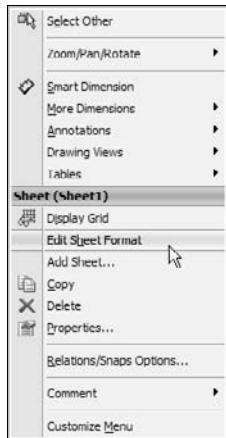
The lines in the format border are regular SolidWorks sketch entities, but they display a little differently. Also, sketch relations are sometimes not used in formats because solving the relations causes the software to be a bit sluggish. Typically, Trim, Extend, and Stretch functions are the best sketch tools for editing lines.

You can use most common image types to insert logo or other image data onto your drawing or format by choosing Insert ➪ Picture. Not all compression styles are supported, however. I have had difficulty with compressed TIFF (Tagged Image File Format) images. Be aware of the file size of the image when you put it into the format, as images can be large and all that extra information will travel around with each drawing that you create from the format. Figure 20.5 shows a bitmap placed in the format.
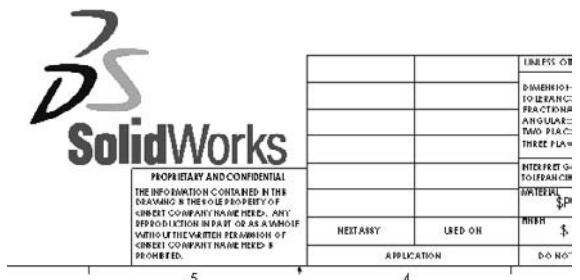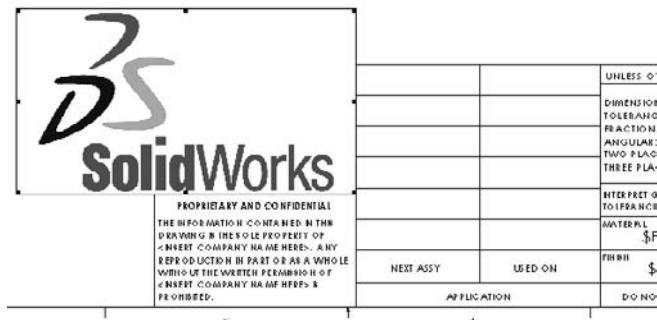
Selecting the edit sheet format

Placing an image

You can resize the image by dragging the handles in the corners and move it by simply dragging it. The bottom image in Figure 20.5 shows the Print Preview window. I included it here to show that the outline around the image that displays while you are working in SolidWorks does not print out.

Managing text

SolidWorks allows you to make a text box of a specific size that causes text to wrap. This is particularly useful in drawings. The upper image in Figure 20.6 shows a new annotation being added. The lower image shows the same text box after the corner has been dragged.

FIGURE 20.6

Adding an annotation and wrapping the text

Tip

When dragging the text box, it may seem intuitive to drag the middle handle on the end, thinking that shortening the box will cause it to wrap. However, that only works if the box has some space on the bottom to wrap. SolidWorks does not automatically reduce the text box down the way PowerPoint does. You are better off dragging the lower-right corner handle of the text box to get the wrap to work. ■

Using custom properties

The most important part of the drawing format is the custom properties. While the rest of the format is just for display, custom properties use automation to fill out the title block using matching custom properties in either the model or the drawing document. Custom properties can pull items such as filenames, descriptions, materials, and other properties from the model associated with the sheet, or they can pull data from the drawing itself, such as the sheet scale, filename, sheet number, and total sheets. If you are seriously looking to automate drawings, you cannot overlook custom properties.

Custom property data entry

Custom property data entry happens at the part or assembly level. This information is then reused in the drawing format and in tables such as BOMs (Bills of Materials) and revision tables, as well as
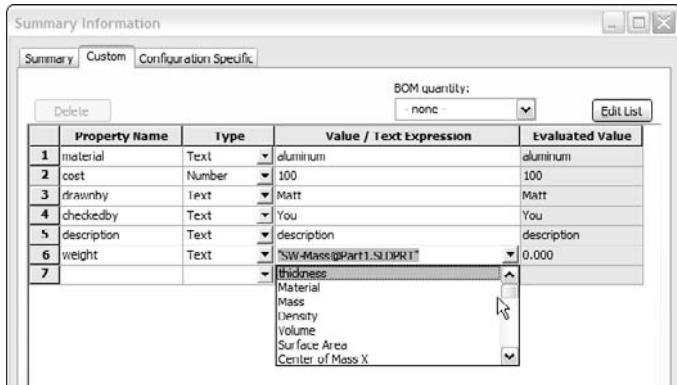
searches using the FeatureManager filter, and all PDM (Product Data Management) systems make use of SolidWorks custom properties. You can enter the data several ways, but the two most prominent ways are through the Summary Information dialog box and through the Custom Properties Tab in the Task Pane.

Summary Information

Figure 20.7 shows the Summary Information dialog box. This functionality has existed in SolidWorks for several releases. You access this dialog box by choosing File ⇨ Properties from the menus. You can select Property Names from a drop-down list or type in your own, assign types of data, and enter in a specific value for the property. The Value/Text Expression column also has a drop-down list from which you can select several preset variables, such as mass, density, and even link values used in the part.

FIGURE 20.7

The Summary Information dialog box

This is a perfectly functional way of entering data, but the fact that it is somewhat out of the way, hidden in the menus, means that it does not get used as much as it should. So SolidWorks came up with another way of entering data.

The Custom Properties Tab
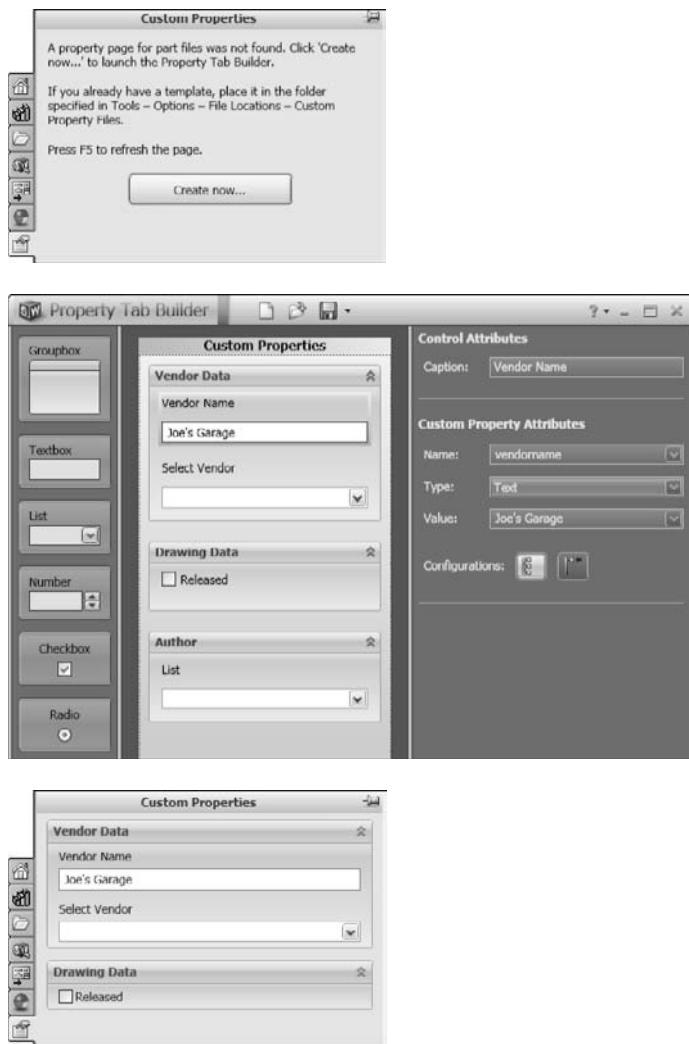
The Custom Properties Tab of the Task Pane enables you to quickly and easily access and assign custom properties within a document. Figure 20.8 shows the process of building your own Custom Properties Tab. You can start the Custom Property Tab Builder by either clicking the Create button on the Custom Properties Tab or choosing Start ⇨ Programs ⇨ SolidWorks ⇨ SolidWorks Tools ⇨ Property Tab Builder from the menus.

The interface enables you to add drop-down lists, toggles, and text entry boxes. This gives you a lot of flexibility with custom property data entry, and is a very nice addition to the software.

FIGURE 20.8

Using the Custom Properties Builder and Custom Properties Tab
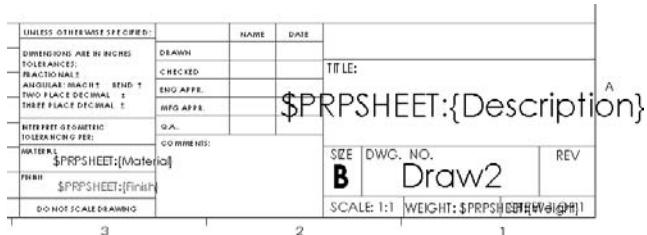
Property link display

Figure 20.9 shows the existing custom property formatting in the default format being used for this example.

FIGURE 20.9

Custom property formatting in the title block
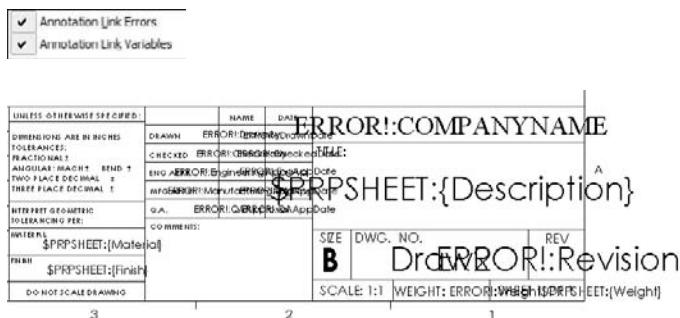
The syntax $PRP or $PRPSHEET indicates that the property that follows the syntax is to be pulled from either the current document (drawing) or from the model specified in the Sheet Properties, respectively. This is an important distinction to make. Most of the time, you can type custom properties in at the part or assembly level so that you can reuse the data by drawing properties, BOM, or even design tables.

Notice that all the notes in the format that are showing raw syntax are pulling data from the model. "Draw2" and the Scale notes are driven by the drawing. When no value exists for the property to display, you have an option of what to show. The top portion of Figure 20.10 shows the settings in the View menu that control the display of syntax of the custom property links. In general, it is common to deselect the error display and to show the link variables.

FIGURE 20.10

The link variable's display options and effects

Errors and link variables
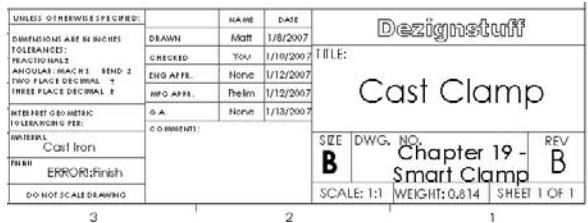
The errors in Figure 20.10 are caused by links to the local document for which there is no corresponding property. For example, the "ERROR!: COMPANYNAME" message is linked to "$PRP: COMPANYNAME," but the local custom property COMPANYNAME does not exist. If it existed but had a null or space value, the error would disappear.

Likewise, with the option to display link variables selected, the syntax that calls model custom properties displays until there is some value for it to pull from. If a part is put onto the drawing, then some of the properties are filled in because properties and values exist to pull from, and the rest of the properties simply disappear to make space. Notice in Figure 20.11 that the Material property has been filled in, but the Finish property has not. This is because either there is no Finish property in the part on the drawing or there is a null value in the Finish property.

FIGURE 20.11

Custom properties filled in by a part

Tip

When initially setting up the format, it can be useful to have a dummy model already on the drawing. The dummy model should have all the custom properties in it that you intend to use in your drawings. This prevents the blank fields or error messages from appearing during setup. ■

Note

If you drag-and-drop a part onto a drawing while editing in the Sheet Format, the views may appear for a split second and then disappear again. This is because you cannot display drawing views while editing the Sheet Format. Once you exit the Sheet Format and go back to editing the sheet, the views can display once more. ■

Creating linked properties

It is easy to create annotations that are linked to properties. Begin as if you are creating a note:
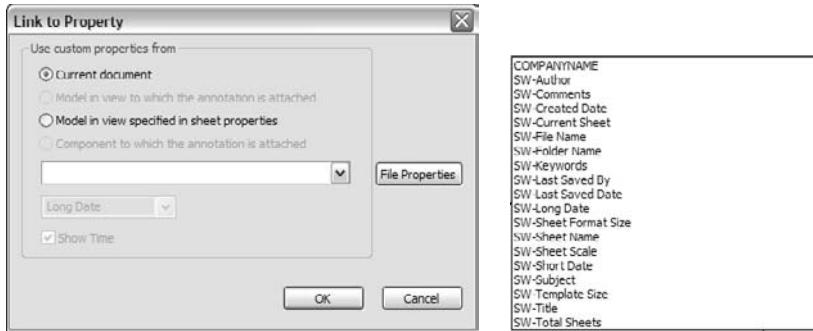
1. **Click the Note toolbar button on the Annotations toolbar, or choose Insert ➪ Annotations ➪ Note.**

2. **Place the note on the drawing.** The Formatting toolbar appears.

3. **Click the Link to Property button in the Text Format pane of the Note PropertyManager.** This displays the Link to Property dialog box, as shown in Figure 20.12, which gives you the option of linking to a custom property in the current (drawing) document or in the model (part or assembly) that is on the drawing.

FIGURE 20.12

The Link to Property dialog box

4. **If the desired custom property is not in the drop-down list shown to the right, then you can type it into the text box or click the File Properties button to edit the properties.** If the property is added to the part file or a part file with that property is used on the drawing, this linked annotation will pick it up. This button is not available for the model if there is no model on the drawing, in which case you must type in the name of the property manually.

Using the Title Block function
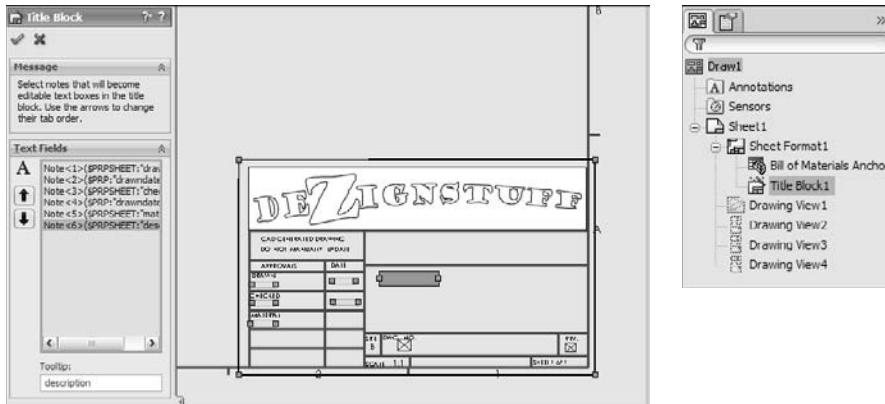
The Title Block enables the person who sets up the sheet format to specify an area that contains notes that are easy to access without editing the format. (Many CAD administrators prefer that the users not have to deal with the details inside the Sheet Format.) You can even cycle through these notes in a specific order by pressing Enter or Tab. Figure 20.13 shows the resizable black border of the Title Block, the Title Block PropertyManager, and where the Title Block sits in the drawing FeatureManager.

You can access the Title Block to edit or define it by right-clicking in the Sheet Format (while editing the Sheet Format, not the sheet) and selecting either Define Title Block or Edit Title Block, as the situation requires.

The Title Block can be any size you like, but it must remain rectangular, and you can only create one Title Block area per sheet format. The area bounded by the Title Block box is used to zoom the display to make it easier to fill in the text boxes. If you want to include areas in different corners of the drawing in the Title Block area, you will need to make the Title Block box as big as the entire sheet and the user will have to manually zoom to each corner.

FIGURE 20.13

Using the Title Block function

Select each Note item to add it to the list in the PropertyManager selection box. Use the arrows to the left of the box to assign the order in which the user cycles through the boxes. The idea is that the user clicks in a box within the Title Block area, fills it in, then presses Enter or Tab to get to the next box. The order will loop if the user does not start on the first box listed in the PropertyManager.

On the CD-ROM

You will find a sample template with a format with a Title Block definition added to it on the CD-ROM. The file is called `title block.drwdot`**. Add it to your template library folder and try it out.** ∎

Creating a format from a blank screen

SolidWorks is not good at manipulating a lot of 2D sketch-line data, such as what you find when drawing title blocks. I have gone through the process of making my own formats, as well as the process of importing DWG data from which to create them. If you choose to custom build one size and then use it to create the rest of the sizes, you need to be patient. SolidWorks typically select the most useful parametric sketch functions when working with a format (what SolidWorks considers a large sketch) because of speed problems. If you would like to select these settings, you can find them by choosing Tools ➪ Sketch Settings.

If you insist on creating your own borders and title blocks, set aside some time for it and have an idea of what you are trying to achieve, maybe sketched out by hand or in a printout of a title block that you would like to replicate. You can also use the DWG Editor, which is much better suited to this kind of work.
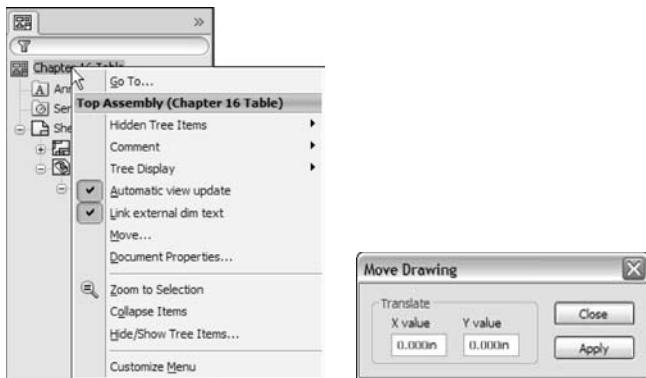
The Modify Sketch tool may be useful for moving entities around the screen, and even scaling them. You can also access a useful hidden command by right-clicking the name of the drawing in the FeatureManager and selecting Move from the menu, as shown in Figure 20.14. A small dialog box appears that enables you to move the entire format by a specified distance.

FIGURE 20.14

The Move Drawing dialog box

Tip

If you need to use construction geometry to help you size or locate objects or text while manually creating your Sheet Format, then you do not need to delete the geometry when you are done. You can put all the construction geometry on a specially created drawing layer and select the layer. ∎

Creating a format from an imported DWG/DXF file
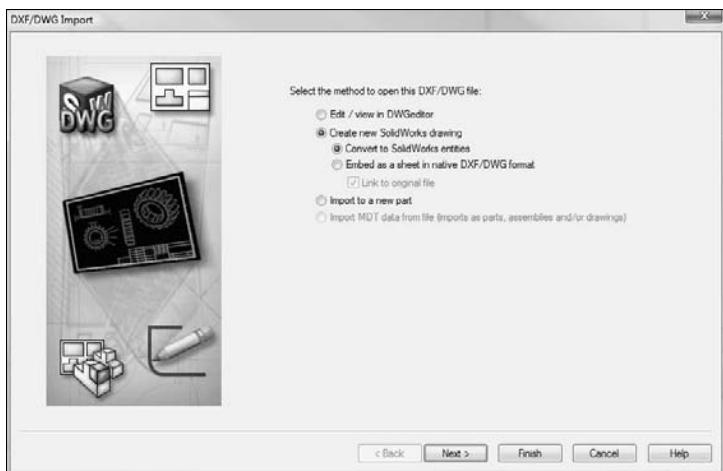
If you want to create your format from an imported DWG or DXF file, choose File ⇨ Open to locate the file that you would like to import and then click to open it. The DXF/DWG Import screen appears, as shown in Figure 20.15.

You can find the sample files used for this example on the CD-ROM for Chapter 20 if you are interested in following along. You will find five *.dwg format files. You can use any of them to create a format, but I suggest either the A or B size. To make a drawing format, you can select the Create New SolidWorks drawing and Convert to SolidWorks entities options. Although one of the other options contains the word *format*, it is not being used in the same sense, so do not be misled. When this selection is complete, click Next. Figure 20.16 shows the next screen.
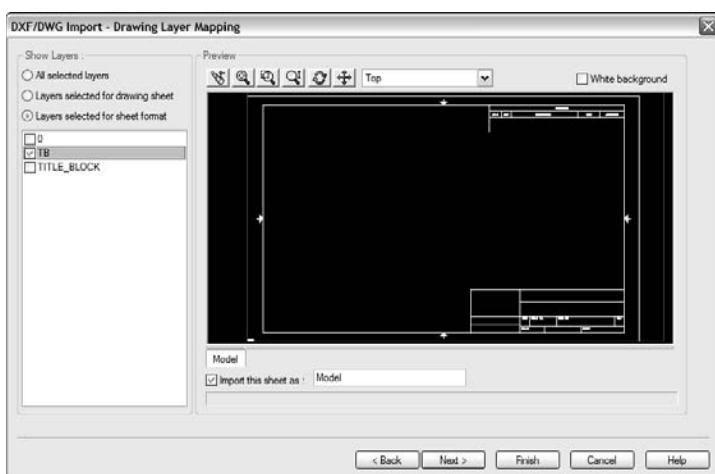
The DXF/DWG Import screen

The Drawing Layer Mapping screen
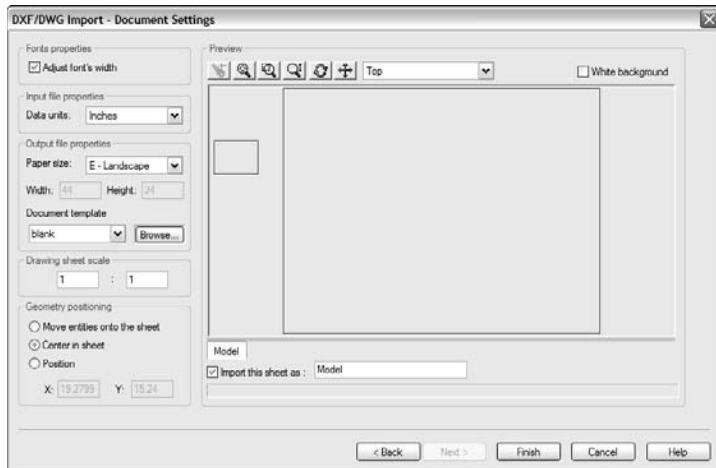
Select the Layers selected for sheet format option. Select the TB layer, leaving the other layers unselected. Every imported file will be different in this respect, because layers used by title blocks vary widely. Click Next when you have made these selections. Figure 20.17 shows the Document Settings screen.

FIGURE 20.17

The Document Settings screen

The important features in the Document Settings screen are the Document template selection and the Geometry positioning options.

Document template selection is only important if you plan to save the format with a template. Be sure to select a template that does not already have a format saved in it. In the Geometry positioning section, if you can get the software to center the title block for you, definitely take advantage of this functionality and use the Center in sheet option. Once you are happy with these settings, click Finish. The resulting format is shown in Figure 20.18.

From here, you can add the links to custom properties as described earlier, as well as logo images, loading favorites, and blocks. You can now save the format as described in the next section.

Tip
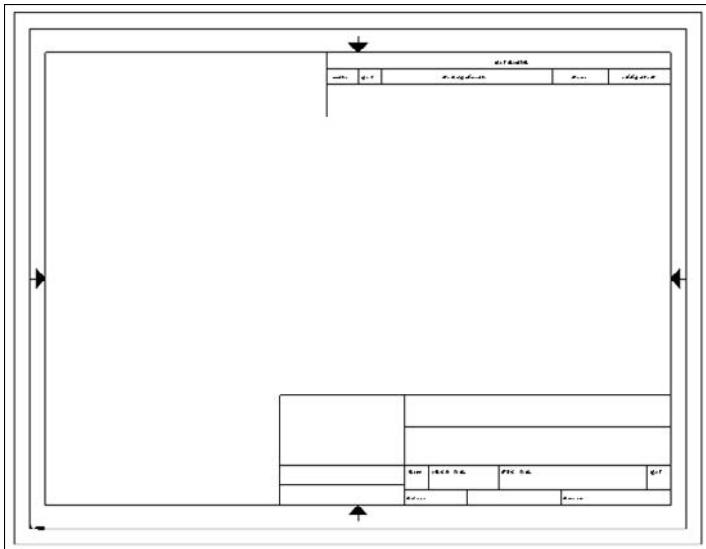
The Color Display Mode button on the Line Format toolbar toggles the display between using the layer color to using the SolidWorks sketch colors. Another setting that affects sketch display in drawings is found at Tools ⇨ Options ⇨ System Options ⇨ Drawings ⇨ Display Sketch Entity Points, which shows endpoints and arc center points in the same way that they are shown in feature sketches. ■

Saving the format

You can save drawing formats in two ways: either with the template or separate from the template. You cannot edit formats separate from a template, but they do have their own file type, *.slddrt.

FIGURE 20.18

The finished imported format

Note

If you are wondering how the extension `*.slddrt` relates to a sheet format, what is now known as sheet format used to be called a drawing template (thus, the drt of slddrt). What is now called a template did not exist in 1997. The shift in architecture and, more importantly for users, the shift in terminology still leaves many people a bit confused. ■

Saving templates is covered in the next section. To save a format, choose File ➪ Save Sheet Format. You can do this with or without the format being active. Save the format into a location with other formats and give it a descriptive but unique name. If you have not yet done so, this is a good opportunity to create a separate folder, outside of your SolidWorks installation folder, that contains your most frequently used files. Remember also to tell SolidWorks where this library location by choosing Tools ➪ Options ➪ System Options ➪ File Locations ➪ Sheet Formats.

Even if you have saved a format with a template, it is a good idea to also save the format on its own. This is because you might want to use that format on an existing drawing that has a different format on it or use it on a second sheet.
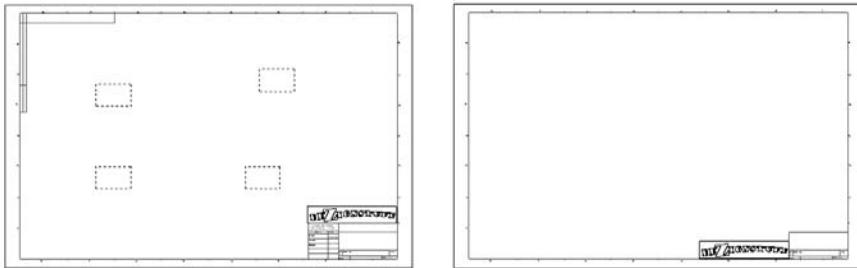
Second sheet formats

When you have multi-sheet drawings, it is often important to have a simplified or specialized format for the second sheet. Figure 20.19 shows sample page-one and page-two formats side by side.

First and second sheet formats

Adding new sheets
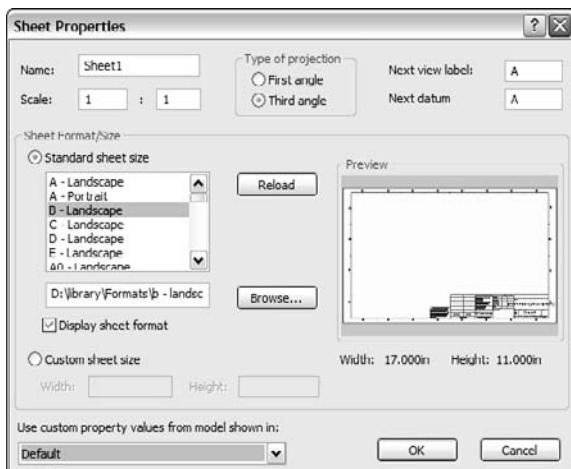
You can add sheets to a drawing by clicking the Add Sheets icon to the right of the sheet tabs at the bottom of the SolidWorks window or through the RMB menu of the sheet tab at the lower-left corner of the drawing window. If you right-click the first sheet tab, the sheet that is added gets the format that is used on the first sheet. If you right-click the second sheet tab, the added sheet gets the second sheet format.

Reloading formats

If a format has been changed, and you would like to update a drawing to the new format, this option is available in the Sheet Properties, as shown in Figure 20.20.

Updating a format through the Sheet Properties

Creating Drawing Templates

Document-specific settings are an important part of the template, and it is probably best to get one drawing size completely set up the way you want it, and then create the other sizes from this drawing. This helps to ensure that the settings, such as bent leader length, font, and line weight, are the same for all the templates. Uniform settings on drawings give them a consistent look and make them easier to read. Drafting standards are also controlled by drawing templates.

Using Pre-defined Views in drawing templates

When I use drawing templates, one of my favorite techniques to get to a multi-view drawing quickly is to put one Pre-defined View on the template along with appropriate views projected from the Pre-defined View. A Pre-defined View establishes an orientation and location on the drawing sheet. You can add multiple Pre-defined Views and align them with one another on the drawing sheet so that a drawing is automatically populated by the model, but this is not recommended because if you decide to change the orientation of the drawing, you have to change each Pre-defined View independently. If you set up a single Pre-defined View and make the rest of the views with Projected Views, changing the orientation of the Pre-defined View causes all the Projected Views to update associatively. You cannot directly change the orientation of a Projected View. Pre-defined Views and views projected from Pre-defined Views appear blank until they are populated with model geometry. The *pre-defined* part of a Pre-defined View is the orientation and placement of the view.
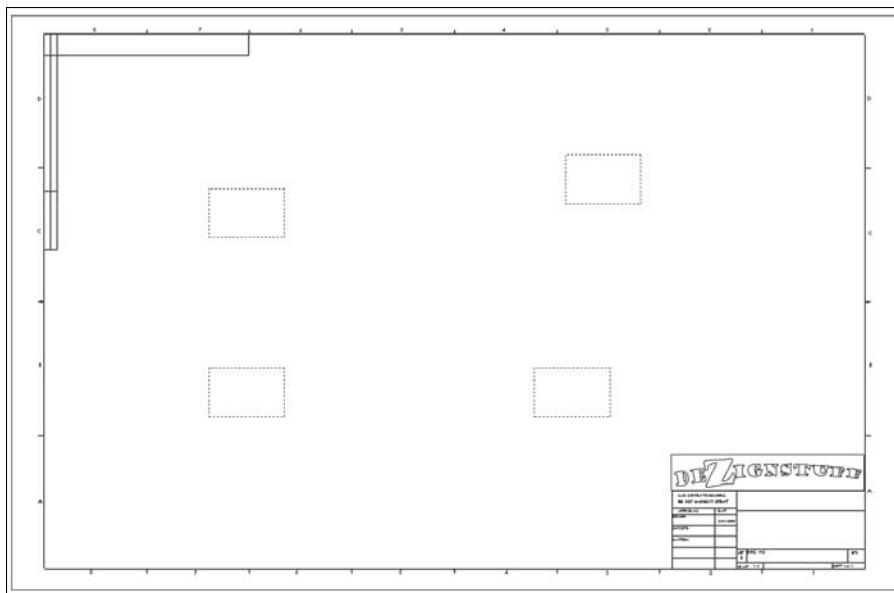
Figure 20.21 shows a template using Pre-defined and Projected Views. You can access Pre-defined Views on the Drawings toolbar; although it is not there by default, you can place it on the toolbar by choosing Tools ➪ Customize ➪ Commands and using the interface. You can also access Pre-defined Views by choosing Insert ➪ Drawing Views ➪ Pre-defined. Projected Views are also accessed from the Drawings toolbar.

Once a Pre-defined View has been placed, you can select an orientation for it from the PropertyManager. Figure 20.22 shows the Drawing View PropertyManager. The orientation for a view is set in the top Orientation panel. In addition to orthogonal views, you can also create isometric and other custom views as Pre-defined Views.
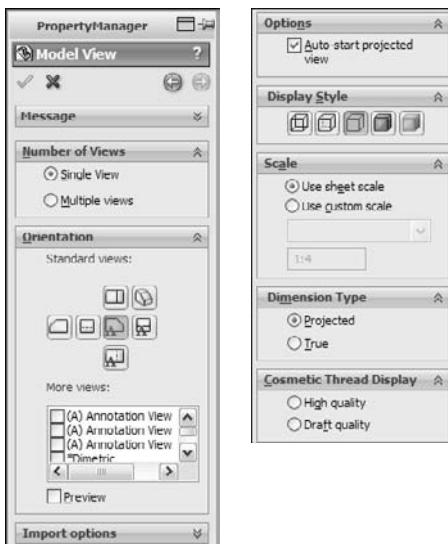
Pre-defined Views on a template

The Drawing View PropertyManager
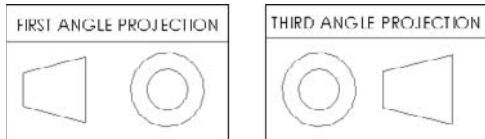
After the view has been oriented, you may want to create more views on the drawing that also become populated by model geometry. This is where the Projected Views are used. Make sure that the drawing properties are set to the correct projection angle.

Because the rest of the views have been created relative to the Front view, none of the views needs to be rotated as it would if, for example, the Top view was placed above the Back or Right view.

Although it is not on this drawing, many drawing templates include a Third Angle Projection symbol as a part of the Title Block, which is in the format. Figure 20.23 shows First and Third Angle Projection symbols. These are included as blocks with the sample data in the SolidWorks installation. Blocks are discussed in more detail in Chapter 22.

FIGURE 20.23

Projection angle symbol blocks

Aligning Pre-defined Views

You can align views to one another through a view's RMB menu, as shown in Figure 20.24. Projected Views are aligned to one another automatically, but if you chose to use a Pre-defined View rather than a Projected View to one side of the original Pre-defined View, you can use the Align Vertical by Origin or the Align Horizontal by Origin command. This ensures that the parts in each view are aligned. Aligning by center should not be used for Projected Views on an engineering drawing, because it is not guaranteed to line up edges in adjacent views.

Populating a drawing with Pre-defined Views
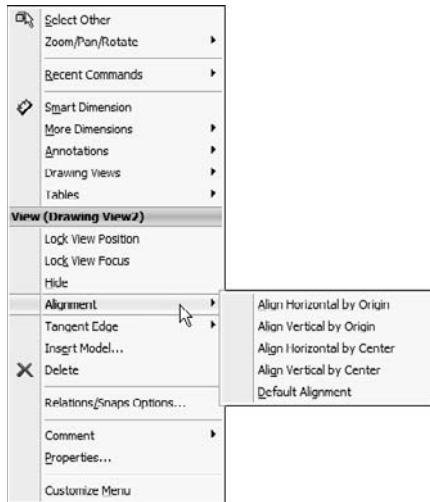
Four methods exist to populate a drawing with Pre-defined Views:

- **Drag-and-drop.** Drag a part or assembly from the FeatureManager and drop it in the drawing window. All Pre-defined Views are automatically populated.

- **Insert Model.** Right-click a view and select Insert Model. From the interface, browse for the model to be displayed in all the related (projected) views.

- **PropertyManager.** Select a pre-defined view, and from the PropertyManager, select Browse in the Insert Model panel.

- **Make Drawing from Part/Assembly.** Click the Make Drawing From Part/Assembly button in the Standard toolbar and select a template that uses Pre-defined Views.

Alignment options

Pre-defined Views and sheet scale

When Pre-defined Views are created, they are set to follow the sheet scale by default; however, you can manually set them to have a custom scale. If you are using the automatic scaling option (found at Tools ⇨ Options ⇨ System Options ⇨ Drawings ⇨ Automatically Scale New Drawing Views), the sheet scale is automatically changed when the drawing views are populated to make a nice fit of the model geometry on the drawing. The scales used by the automatic feature are all standard multiples of two, so you do not have to worry about odd scale factors on your drawings.

Pre-defined View limitations

The function and expectations of Pre-defined Views are fairly straightforward, although there are a few things that could be improved. For example, SolidWorks does not allow you to create pre-defined section or detail views. Also, the View Palette does not preview the populated Pre-defined Views.

Using styles and blocks in templates

Starting in SolidWorks 2009, the functionality formerly known as *favorites* is now known as *styles*. In SolidWorks, styles function like styles and formatting in Microsoft Word, or other word-processing software, by adding underlines, bold formatting, and even items such as tolerances and symbols. Hole Wizard Styles are described in Chapter 17, and work similarly to Dimension and Note Styles (described in Chapter 22). This chapter addresses the fact that styles can be saved to files and loaded to documents. In particular, they can be loaded to documents that can be saved as

templates, thus maintaining the loaded styles. Several types of styles can be loaded into and saved with drawing templates, including dimension, note, GD&T, weld, and surface finish symbols.

When a style is loaded into a template, any document that you create from that template can use any of the loaded styles. The many file types for styles exist mainly to transfer styles from one document to another, but they are not needed once the style is loaded. As a result, before saving a template, you should gather together your styles into your library folder and load them into the template.

You can load styles by going to the interface for the type of favorite; for example, dimensions or notes. Figure 20.25 shows the top of the Note PropertyManager interface, which contains the Styles panel.

FIGURE 20.25

The Styles panel for the Note PropertyManager

The buttons in the Styles panel of the Note PropertyManager interface have the following functions, from left to right:

- Apply the default attributes to the selected notes

- Add or update a style

- Delete a style

- Save a style

- Load a style

This section is concerned with the last function, Load a style. After clicking this button, you can load multiple styles at once by Shift+selecting them through the Open dialog box that appears.

Even symbol types that can be applied by dragging-and-dropping from the Design Library can also be loaded as styles. However, I prefer dragging from the Design Library because you get a preview of the symbol; with the styles, you just see a text tag.

Blocks can also be loaded into a template or used from the Design Library as drag-and-drop items.
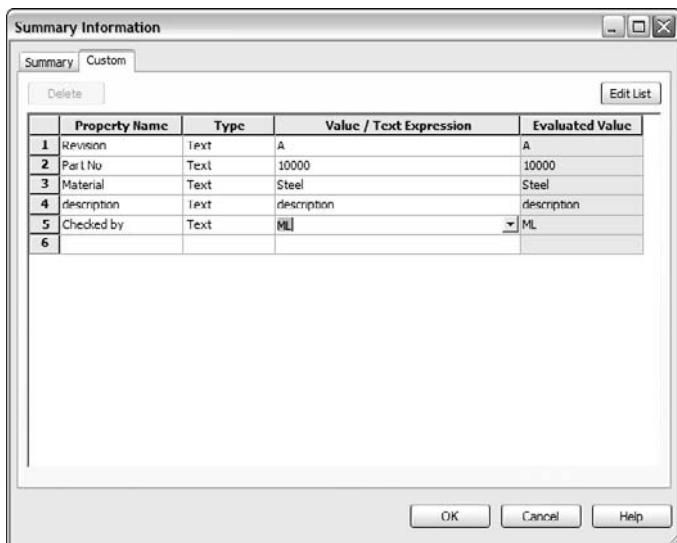
Custom properties in templates

Part of the usefulness of templates is that you can do work once and have it replicated many times. This is an excellent example of process automation. One of the ways that you can take advantage of this feature is by putting default custom properties in your templates. In many cases, simply having a default value for something is better than no value, and a default value may even prompt you to put a value with real significance in the property. For example, the Description of a document is extremely important, especially if you are using sequential part numbers for your filenames. A custom property named Description can be added to your template, and the default value is used unless it is changed when the template is used in a document.

You have already seen how custom properties used in parts can be instrumental in filling out a title block on a drawing. Custom properties in part and assembly documents work exactly the same as they do in drawings. The custom properties interface is shown in Figure 20.26.

FIGURE 20.26

The custom properties Interface

Saving a template

To save drawing templates, choose Save As ⇨ Files of Type and then select Drawing Templates from the drop-down list. This automatically takes you to the folder for the templates, as specified in Tools ⇨ Options ⇨ File Locations ⇨ Templates.

In the case where a template and format have been saved together and are being saved together, but the format also needs to be saved to its own file, saving the template with the changed format only changes the format for documents that are made from that point forward with that template.

You may also save out the format to its own file from the edited template. Formats are needed in their own file (in addition to existing within a template) for situations when you have an existing drawing and want to change the size of the sheet, and then need a format to put on the sheet. Another situation is when a drawing may come in to your organization from an outside contractor, and they have not used your format; in this case, you can simply replace their format with yours, or you can send them your format (and template, for that matter), from which the contractor can create all drawings for you.

Separate formats are important for when you have multi-sheet drawings. When adding a sheet, you also need to add a format. You can save multi-sheet drawing templates in which the first and second sheets have different formats on them.

Creating Blocks

Blocks are an important aspect of automating drawing creation. They enable you to combine text and sketch geometry and to annotate common features on drawings. Blocks are discussed in Chapter 4 and also in Chapter 22 (creation, editing, and placement). Blocks can be used for many purposes, including the following:

- Tolerance blocks on drawings that might change with the process (if you do not have separate formats that already contain this information)
- Electrical or pneumatic schematic symbols that can be snapped together
- Flowchart type symbols
- Fluid flow-direction arrows
- Special markers calling attention to a specific detail
- Sheet formats that can be created as a block, enabling you to move it around as a single entity much more easily

You can create blocks by selecting a group of sketch entities, annotations, or symbols and then choosing Tools ⇨ Block ⇨ Make.

Cross-Reference

For more information on creating, editing, managing, and placing blocks, see Chapter 22. For more information on general CAD Administration and specific recommendations for templates and formats, please refer to the SolidWorks Administration Bible (Wiley, 2009). ■

Summary

Getting your templates and formats correct creates an excellent opportunity to save some time with drawings by automating many of the common tasks using templates, Pre-defined Views, multiple formats, blocks, favorites, and linked custom properties. Setup becomes more important when you are administering a larger installation, but is also important if it is just for yourself. One of the most important things that you can do is to establish a file library and direct your Tools ➪ Options ➪ File Locations paths to the files. There is nothing quite as productive as having something that works right the first time, and every time.

Working with Drawing Views

I n SolidWorks drawings you do not create lines to create views. The drawing view is a snapshot of the 3D model from a particular point of view. To change the lines on the view, you have to change the 3D model, you do not just move lines around the view.

If you are coming from AutoCAD, this might seem a little bit confining. However, this method will become liberating rather than confining. It means that you do not have to worry about the drawing views being inconsistent or incorrect. All you have to worry about is the 3D model being correct.

SolidWorks automatically maintains the views better than you could do it manually. It can update any type of view from any point of view of even the most complex model or assembly geometry perfectly.

Creating Common View Types

The previous chapter discussed Pre-defined Views in templates. Pre-defined Views make it faster to automatically create drawings with consistently placed, simple views. However, sometimes you may need to create views on templates that do not have Pre-defined Views, or you may need a special arrangement of views. SolidWorks has a good assortment of view types to make practically any type of view that you may need.

Note
When creating or changing either the geometry or the settings that control how a view is displayed, the view may become cross-hatched, indicating that the model needs to be rebuilt. To resolve this problem, press Ctrl+B to rebuild the drawing. ■

View palette

The View palette is shown in Figure 21.1. It is activated automatically if you use the Make Drawing From Part tool, unless the drawing template that you select has Pre-defined Views on it. In this case, the Pre-defined Views are populated and the View palette is not activated.

The View palette contains all the standard named views, the current view of the model, custom named views, and any annotation views (views that the model was in when annotations were added to it). You can drag-and-drop these views on the drawing.
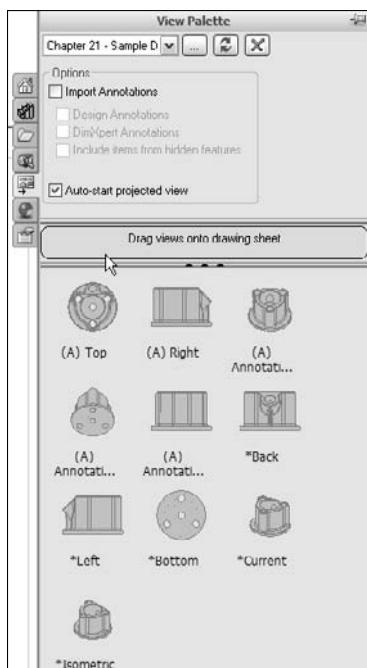
If multiple parts are available, they are listed in the drop-down list at the top of the panel. You can also browse, refresh, or cancel out of the view from this same area.

To activate the View palette without using the Make Drawing From Part tool, simply create a new drawing document, ensure that the Task Pane is available, and click the View Palette tab in the Task Pane. Then use the ellipsis button (...) to browse to a part. After you select a part, the palette window is populated with views of the model. This method has the advantage of enabling you to see the views before you put them down. It does not link views in the same way that the Predefined and projected views are linked, however. I find this interface somewhat difficult to use, and prefer to set up the Pre-defined Views in templates or to use the Multiple Views option in the Model Views PropertyManager, which is shown later in this chapter.

FIGURE 21.1

The View palette

Model View
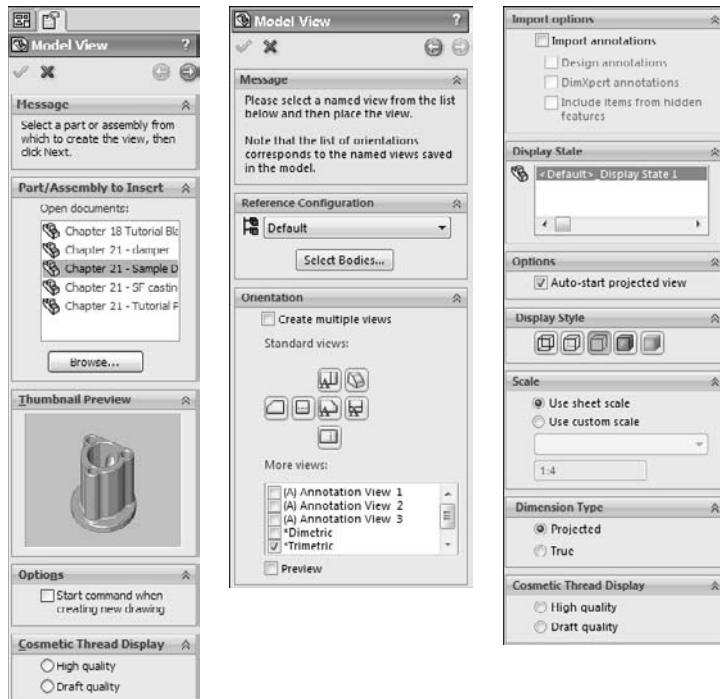
Model Views are one of the few types of views that are not dependent on another view. Everything has to start from somewhere, and most drawings have to start with either a named or Pre-defined View. A Model View starts from a named view in the model.

You can place named views by clicking the Model View button on the Drawings toolbar or by choosing Insert ➪ Drawing View ➪ Model. Using the Model View PropertyManager is a two-step process, and is shown in Figure 21.2. In the first step, you select the model, and in the second step, you set the options for the view. Views dragged from the View Palette are also Model Views.

The Model View PropertyManager has seen several changes in SolidWorks 2010, some of which are shared with the Drawing View PropertyManager and other view creation PropertyManagers. This has to do with Display States and annotations to import to the drawing view.

FIGURE 21.2

The Model View PropertyManager

Open documents

The large selection box in the Part/Assembly to Insert panel displays any models that are open in SolidWorks at the moment. If the model that you are looking for is not in the list, then you can use the Browse button to look for it.

I typically use Create Drawing From This Part/Assembly if the part is open, and if not, I drag-and-drop the part onto a new drawing created from a template with Predefined and projected views on it. This combination saves a lot of extra steps.

If you click in the drawing window for some reason (for example, if you are expecting it to simply place a view), then a prompt appears, stating that you have selected a drawing document, and that only parts and assemblies can be inserted into drawings.

Thumbnail Preview

This is a nice option that shows the part that you selected in the Open Documents window. It is a useful feature, but because it is collapsed by default, it is easy to miss. After it is used the first time, it remembers the expanded setting.

Start Command When Creating New Drawing Option

The Start Command When Creating New Drawing option causes this PropertyManager to open immediately when a new drawing is created. If you click in the drawing window, then the prompt appears, telling you that you are not paying attention.

Reference Configuration

The Reference Configuration list enables you to select which configuration of the part to show in the view. This shows up not only when creating new views, but also in the generic Drawing View PropertyManager that shows up when you select any view.

Select Bodies

When a part has multiple bodies, a button called Select Bodies also shows up in this panel. If the part does not have multiple bodies, you will not see this button, When you click the button, it immediately takes you out to another PropertyManager, the smaller one shown in Figure 21.3 called Drawing View Bodies, where you are sent back to the model window to select a body. Clicking the green check after selecting a solid body in Drawing View Bodies then sends you back to the drawing to place the view. It does not send you back to the Model View PropertyManager.

If you click the red X in the Drawing View Bodies PropertyManager, SolidWorks leaves you in the part window, and you will have to press Ctrl+Tab to get back to the drawing window.

Cosmetic thread display

Many people see the High and Draft quality options and assume that the option refers to the quality of the view, while in fact it refers to the quality of the cosmetic thread display. Cosmetic threads can display in either high or draft quality. The distinction is made for performance reasons. The
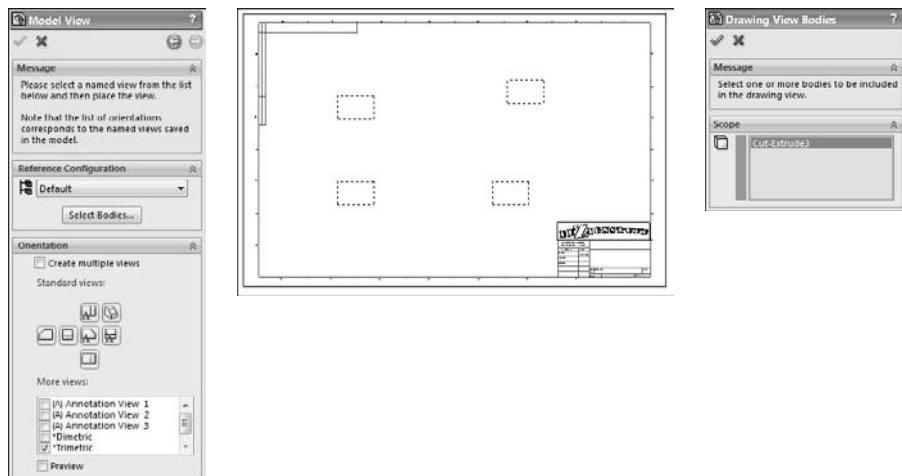
difference in terms of display is that in high-quality mode, hidden cosmetic threads (cosmetic threads that are behind a face) do not display in shaded mode.

Number of Views and Orientation

The way you go about placing multiple views with the Model View tool is slightly different in 2010 than previous versions. In SolidWorks 2010, you now must toggle multiple view creation on or off. Previous versions asked you to select either single view or multiple views. When you have the Create Multiple Views turned on, you select all views that you want to be displayed, including choices such as Current Model View and any named or annotation views that are created in the model interface. These views are indicated on the drawing as boxes (representing view borders), as shown in Figure 21.3.

FIGURE 21.3

Placing multiple views

This is really useful functionality. It makes view selection and placement very easy and is visually clear. Unfortunately, the Single View setting is the default setting, and the PropertyManager does not remember the last setting that was used. Still, the combination of Multiple Views and Orientation is far better, in my opinion, than the View Palette.

Import Options

Also new in 2010 are the Import Options for bringing annotations into the drawing view being created. You should find it an advantage being able to select these items quickly from the PropertyManager while the view is being created.

Display State

On the drawing you can now select the Display State. This is probably meant for situations like using Display States for hide and show operations, but remember that there is a lot more to Display States than just hide and show. You can also change display styles and colors, and transparency, which you might think do not have any place on a technical drawing.

Display Style

You can set the default display style by choosing Tools ➪ Options ➪ System Options ➪ Drawings ➪ Display Style. This panel provides an override for views being placed. This panel also enables you to control High or Draft quality views, which are described later in this chapter.

Scale

SolidWorks drawings always default to showing views at the overall sheet scale unless the System Option on the Drawings page called Automatically Scale New Drawing Views is selected. If this setting is selected, the sheet scale saved with the drawing template is overridden. For example, a 1:1 sheet scale can be changed automatically by the setting to 1:4.

You can change the sheet scale through the sheet properties, which were discussed in Chapter 20. Controlling views with the sheet scale makes it much easier to change the size of a drawing and to scale all the views together. Individual views can be displayed at the view scale, and detail views are typically created at a different scale automatically. To locate the scale setting, choose Tools ➪ Options ➪ System Options ➪ Drawings ➪ Detail View Scaling. Detail Views, covered later in this chapter, automatically get a note showing the custom scale for the view. You can manually create a note that functions in the same way as the automatically created "Scale" text from a note and a link to a custom property for views where the link to the sheet scale has been broken manually.

Tip

You could consider creating a note style or block for a note that automatically links to the scale of a drawing view. ■

Dimension Type

Even in non-orthogonal (isometric) views, true dimensions should be used for most drawing views. Projected dimensions depend on the angle of the edge to the view plane.

Cosmetic Thread Display

If something is worth having, it is worth having twice. This panel appears in both steps, just in case you missed it in the first step.

Projected View
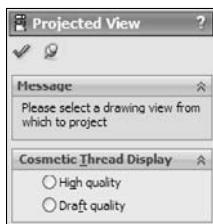
The Projected View type simply makes a view that is projected in the direction that you dragged the cursor from the selected view. Be aware that first-angle and third-angle projections result in views

that are opposite from one another. For example, if you drag at a 45-degree angle, the result is an isometric view. When placing an isometric view that you have created in this way, SolidWorks constrains the new view to a 45-degree-angle line through the Origins of the two views. To place the view somewhere other than along this line, press the Ctrl key while placing the view to break the alignment. The PropertyManager for the Projected View is shown in Figure 21.4.

The Projected View PropertyManager

When you use the pushpin on the Projected View PropertyManager, you can place multiple projected views from the originally selected view or select a new view to project views from. Display properties and scale of the projected views are taken from the parent view.

Standard 3 View

You can access the Standard 3 View tool on the Drawings toolbar by choosing Insert ⇨ Drawing View ⇨ Standard 3 View. This places a Front view, and projects Top and Right views for third-angle projection drawings.

Detail View

You activate the Detail View from the Drawings toolbar or by choosing Insert ⇨ Drawing View ⇨ Detail. Either way, you can use the function in two different ways: one that is fast and easy and the other that gives you more control but is not quite as fast.
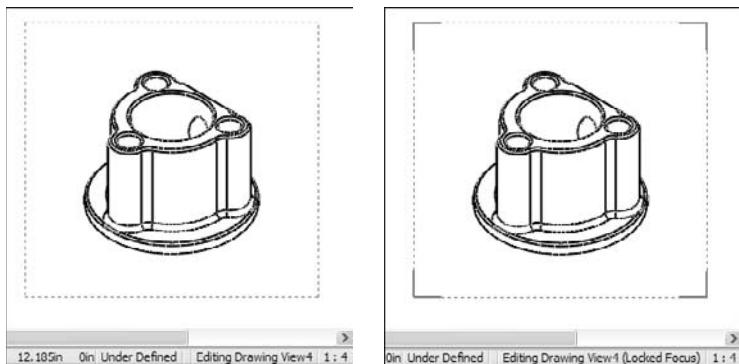
Pre-drawn detail circle

You can draw the detail "circle" before you initiate the Detail View command. When you pre-draw a detail circle, you must ensure that you are sketching in the view and not on the sheet. To draw in the view, the view must be activated. You can activate a view by clicking in the view or by bringing a sketch cursor inside the boundary of the view. When you activate a view, the status bar in the lower-right corner of the SolidWorks window displays the message, Editing Drawing View, as shown in Figure 21.5.

FIGURE 21.5

Activated drawing views

12.185in   0in Under Defined   Editing Drawing View4   1 : 4   0in Under Defined   Editing Drawing View4 (Locked Focus)   1 : 4

The dotted border in the image to the left shows that the view is selected, and the status bar shows that it is activated. The image to the right with the solid corners indicates that the view has Locked Focus. You can lock focus on a drawing view by double-clicking it or by right-clicking and selecting Lock View Focus from the menu.
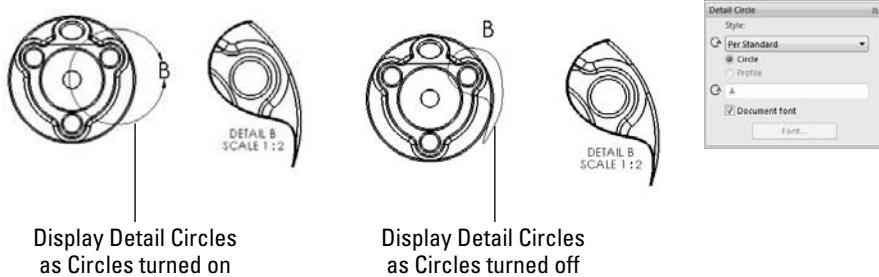
If a view is not activated or the focus is not locked on the view, then any sketch elements that you draw will be placed on the drawing sheet. While sketching in a drawing view, it is a good practice to watch the status bar.

The point of all of this is to sketch a closed loop in the view so that it can be used for a Detail View. The closed loop can be a circle, ellipse, spline, series of lines, or any other shape, as long as it is a closed loop. SolidWorks refers to a closed loop as a *circle* with the Detail View.

A setting controls how the circle displays, in particular whether it displays as drawn or as an actual circle. This setting is found in the Detail Circle PropertyManager. If the setting is grayed out, the Style option may be set *per standard*, and the standard you are using does not allow for non-circular detail circles. You could choose With Leader instead or change the drafting standard you are using. The different results are shown in Figure 21.6.

FIGURE 21.6

Drawing a closed loop with the Display Detail Circle as Circles option both on and off

Display Detail Circles
as Circles turned on

Display Detail Circles
as Circles turned off

Once you create the loop, you can click the Detail View toolbar button and place the view. The view is automatically scaled by the factor set at Tools ➪ Options ➪ Drawings ➪ Detail View Scaling. By default, this scale is set to twice the parent view scale, but you can reset the default to whatever you like.

Detail circle drawn in-line

A faster way to complete the Detail View is to simply click the Detail View toolbar button without pre-selecting or pre-drawing the loop. This activates the Circle sketch tool immediately, which activates the view as soon as you bring the cursor over the view, so that when you draw the circle, it is sure to be in the view rather than on the sheet.

Alternatively, you could swap the circle tool for an ellipse or spline; this works just as well, but offers more flexibility. Regardless of the sketch tool, when you close the loop, SolidWorks prompts you to place the view. The workflow for this in-line method is better than the old-school pre-drawn loop technique.

Editing a Detail View

You can edit a Detail View by dragging the circumference of the detail circle to a new diameter, dragging the center of the detail circle to a new location, or by selecting Edit Sketch from the detail circle right mouse button (RMB) menu. This method enables you to edit sketch relations or otherwise edit the sketch that you used for the detail. When you are done with the sketch, you can use the Confirmation Corner to click OK.

You can delete Detail Views by selecting and deleting the detail circle. Deleting the detail circle gives you the option to delete the resulting view as well as the original sketch. Also, deleting the Detail View gives you the option to delete the detail circle and the original sketch.

Section View

Section Views in SolidWorks offer many options, such as Default Section View, Partial Section View, Aligned Section View, and Editing a Section View.

Default Section View
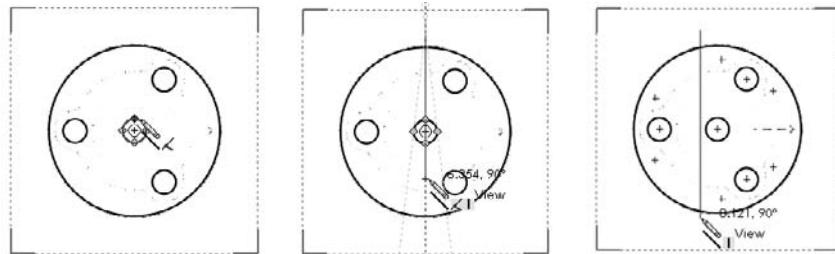
The Default Section View has the same in-line and pre-drawn optional techniques as the Detail View, as well as the same advantages and disadvantages.

Section Views may have a straight line that may go through the center of a cylindrical feature. Even though you are in a drawing and not in a model sketch, you may still benefit from model sketching techniques. For example, to draw a straight vertical line through the model shown in Figure 21.7, in the images on the left, you can hover the cursor over a circular edge to wake up the center, and then pick up the inference lines to the center. Another technique is to show the temporary axes (using the View menu), just sketch the line, and then assign a sketch relation in the same way that you would in a feature sketch. This technique is shown in the images on the right.

FIGURE 21.7

Aligning a line in a Section View sketch
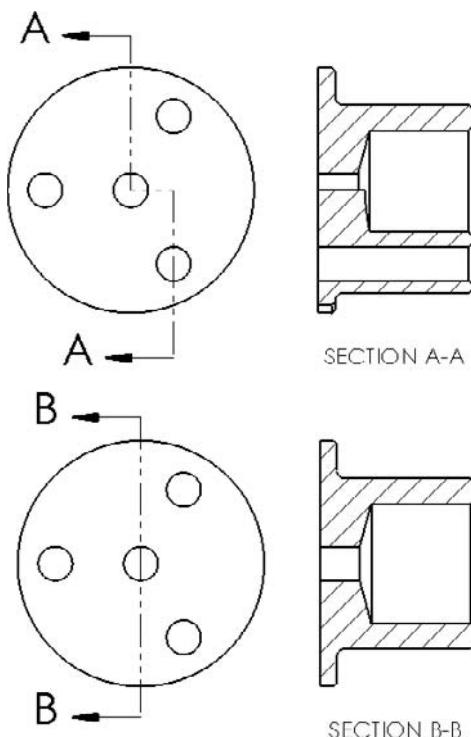
You can also use jogged section lines with the default section tool. Similar sketch relation techniques are more common in jogged sections because there are more sketched lines. The results are shown in Figure 21.8.

You can also tell a Section View how deep you want the section view to see into the part. In the PropertyManager of the Section View, you can set a depth with a number or you can select a face, edge, or vertex to determine the depth. When you select the check box at the top of the Section Depth panel, a graphic handle becomes available on the drawing view, which enables you to visually drag the depth as well. This functionality is shown in Figure 21.9.

FIGURE 21.8

Default and jogged Section Views

SECTION A-A

SECTION B-B

FIGURE 21.9

Setting a depth for a Section View

Partial Section View

A Partial Section View is created when the section line does not cut all the way across the model. In Figure 21.10, the line that is drawn to create the Partial Section View was the vertical section line. The prompt that appears enables you to confirm that you intended to create a partial section cut. If you answer No to the prompt, the result is an error, with the new view displayed in the dangling color.

FIGURE 21.10
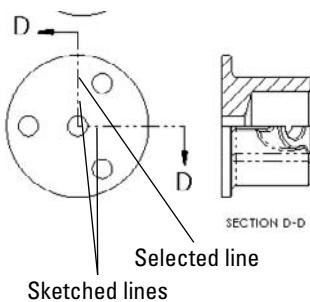
A Partial Section View requiring only a single sketched line

Sketched line

You can use another technique to create a section that looks like a partial section, but is not considered a partial section by SolidWorks. You can create it by drawing perpendicular lines, and selecting the line to be used as the projection direction for the section before clicking the Section View tool. This differs from a true partial section in that it shows half of the model unsectioned. It is also similar to the Aligned Section View, but it does not unfold the second sectioned side. When the prompt shown in Figure 21.10 displays, clicking Yes causes the resulting view to look like the view on the left in Figure 21.10. Clicking No causes the view shown in Figure 21.11 to appear. Creating the view shown in Figure 21.10 requires only a single sketched line, while Figure 21.11 requires perpendicular sketched lines.

FIGURE 21.11

A Section View requiring perpendicular sketched lines

Selected line

Sketched lines
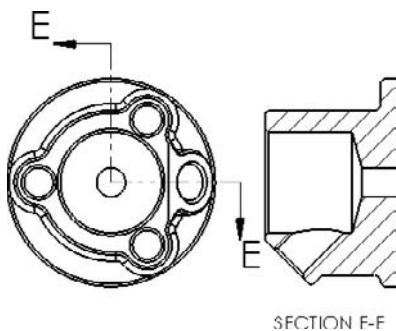
Aligned Section View

The Aligned Section View takes two separate sections at right angles to one another and lays them out flat on the page. It is essentially two partial sections that display side by side. The section lines look identical to those shown in Figure 21.11, but the resulting view is different, as shown in Figure 21.12. The finished view aligns with the selected sketch element.

FIGURE 21.12

The Aligned Section View

SECTION F-F

Editing a Section View

Section Views are edited in the same way as Detail Views. You can edit the section lines directly by dragging, or the section line sketch through the RMB menu. You can click the RMB menu and select the Edit Sketch command to edit sketch relations, or to add or remove sketch elements to the sketch.

Section Views are also deleted in the same way as detail views, with the option to also delete the underlying sketch for the section. When you delete one segment of the section line, the resulting view, as well as the underlying sketch, is also deleted.

Creating Other View Types

SolidWorks is able to create any type of view that you need. If you can define what the view should look like, then SolidWorks can create it. The only limit is your creativity.

Crop View
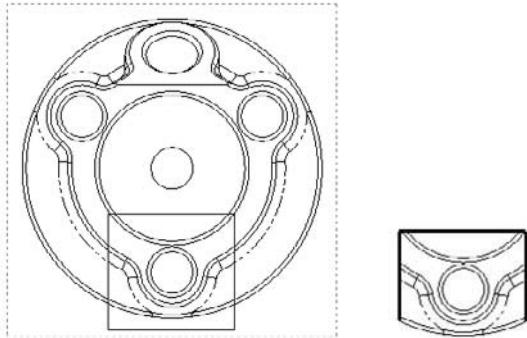
The Crop View is simply a view that looks like a Detail View without requiring a parent view. This feature enables you to reduce the number of views on a sheet and save some room. However, a cropped view may be confusing if it is not clear which area is being detailed in the cropped view.

Unlike Detail Views, in Crop Views, the closed loop must be sketched in the view before you invoke the command. To make the Crop View, draw the closed loop as shown in Figure 21.13 in the image to the left, and then click the Crop View button on the drawing toolbar or access the command by choosing Insert ⇨ Drawing View ⇨ Crop.

FIGURE 21.13

A sketch loop and a Crop View

To edit a Crop View, right-click the view, expand the arrow next to Crop View, and select either Edit Crop or Remove Crop. Removing the crop does not delete the sketch that the crop was created from.

Broken-out Section View

The Broken-out Section View is another view type that alters an existing view rather than creating a new view. It also requires a closed loop sketch. The Broken-out Section View is very useful in assembly views where parts are obscured by other parts, in particular when a set of parts is inside a housing and you want to show the inside parts without hiding the housing. Of course, you can also use Broken-out Section Views on parts with internal detail.

Broken-out Section Views act like a cut that is created from the drawing view. Any faces created by the cut are hatched. Figure 21.14 shows a simple assembly view using a Broken-out Section View. On the left is the view with the driving sketch (in this case, a closed loop spline), and on the right is the finished view. You cannot create Broken-out Section Views using existing Detail, Section, or Alternate Position Views.
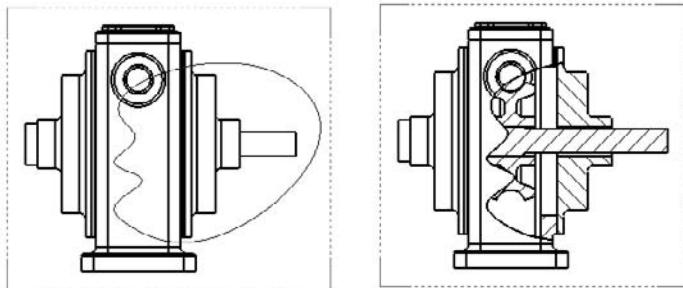
Broken-out Section Views require you to specify a depth for the break. You can use an edge selected from a different view or a distance to specify the depth. In the case of the broken-out section, the depth is into the screen, while with the regular section the section depth is measured as a distance perpendicular to the section line.

Drawing the closed loop

Broken-out Section Views are initiated from an existing view either with or without a pre-drawn closed loop. If the loop is pre-drawn, then you must select it before clicking the Broken-out Section toolbar button on the Drawings toolbar or accessing the command by choosing Insert ⇨ Drawing View ⇨ Broken-out Section.

FIGURE 21.14

A Broken-out Section View

If the view has no pre-drawn, pre-selected loop, then initiating the function activates the spline sketch tool. It is not necessary to use a spline as the closed loop for this view type, but Broken-out Section Views are traditionally created with a freehand sort of boundary, even when drawn manually.

If the loop is closed in an uninterrupted workflow, then after the last spline point is drawn, joining the spline back to itself, the Section Scope dialog box appears. This enables you to select any parts that are not to be sectioned. It is customary to avoid sectioning shafts, screws, or other cylindrical components. Using the Section Scope, the image on the right in Figure 21.14 would be altered to look like the image on the right in Figure 21.15.

The recommended workflow is to initiate the function from the toolbar, use the spline to create the closed loop, and to not pre-draw a loop. This makes everything flow more smoothly, and you create the view surprisingly quickly. If you must use a sketch tool other than the spline, then you must pre-draw it. Even if you simply change sketch tools when the Broken-out Section View automatically activates the spline, because the workflow has been broken, creating the closed loop does not automatically display the Section Scope interface.
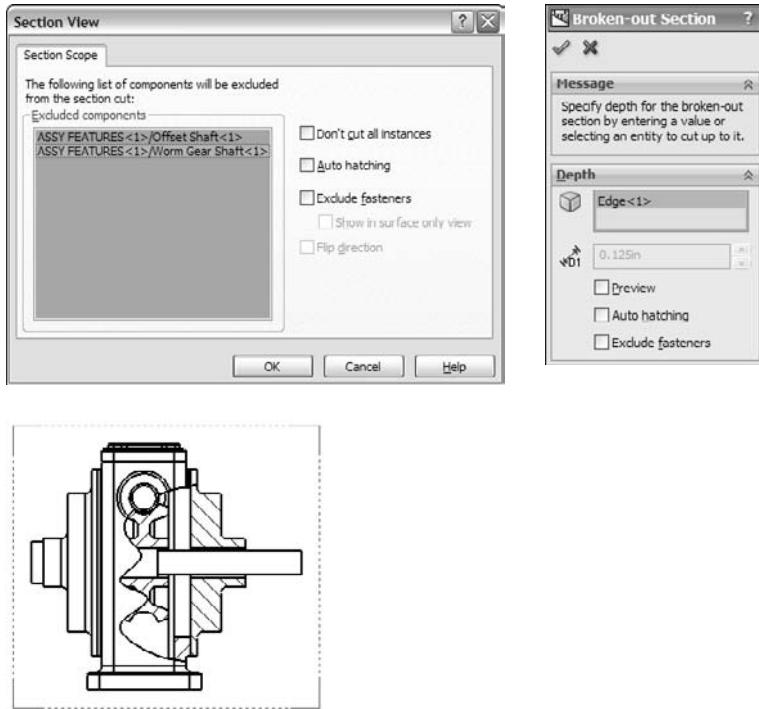
Selecting the depth

After you make the Section Scope selections, the next step is to set the depth of the cut. You can do this in one of several ways. Broken-out Section Views are usually applied to the center of a hole if available, or in other ways that show the view as cleanly as possible. If you know the depth of the cut that you want to make, then you can type it in as a distance value. Of course, that raises the question "Distance from *what*?" to which the answer seems to be "from the geometry in the view that would come the farthest out of the screen toward the user." Users most often choose the distance when it does not matter *exactly* how deep the cut goes or exactly *where* it cuts, but it gives a relative position.

FIGURE 21.15

Using the Section Scope

In situations when you want to cut to the center of a particular feature or up to an edge, it is far easier and less bothersome to simply select the geometry from a drawing view. For example, Figure 21.16 shows the PropertyManager interface where the depth of the cut is set. In this example, the edge of the shaft in the view to the right has been selected. This tells SolidWorks that the cut should go to the center of the shaft. Another possibility is to show the temporary axes, as shown in Figure 21.16, and to select an axis through the center of the shaft.
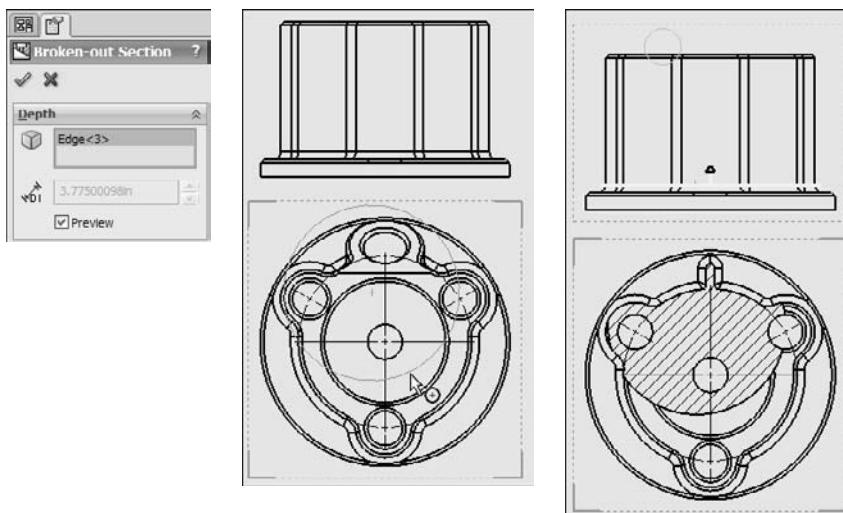
Editing the view

At this point the view is finished. Now you may choose to edit the view in some way, such as by changing the sketch, the depth, the section scope, and so on. Figure 21.17 shows how the Broken-out Section View is positioned in the Drawing FeatureManager. It is listed as a modification to an existing drawing view. The Broken-out Section RMB menu is also shown. Selecting Edit Definition displays the PropertyManager, shown in Figure 21.16. Selecting Edit Sketch enables you to change the section spline shape. Selecting Properties displays the dialog box shown to the right in Figure 21.17. This contains options for the underlying original view as well as the Broken-out section modification to the original view. Only the Section Scope tab is added by the Broken-out Section View. The rest of the options are for normal view properties.
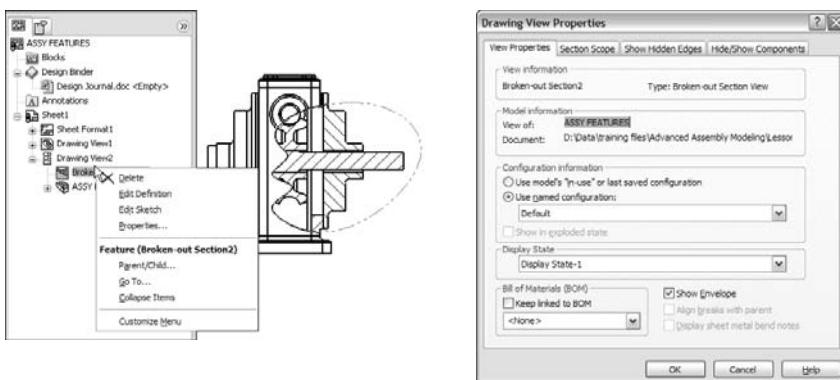
Setting the depth of the Broken-out Section View

Editing the Broken-out Section View

Break View
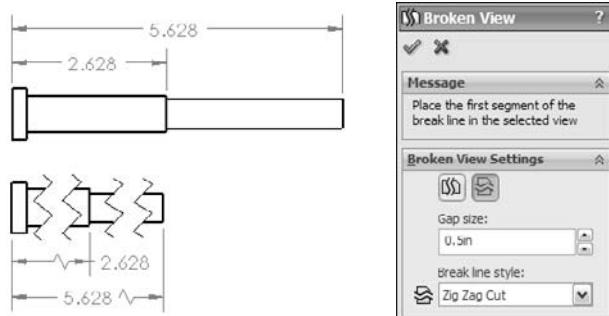
Break Views are typically used to display parts that are very long in one dimension on a drawing in such a way that you can see both ends or other important features. You can break views more than once in the same direction, or even in opposite directions. Figure 21.18 shows the full view of a part and a view that was broken twice. Notice that the dimensions are correct, and any dimension that includes a broken length has a special dimension line.

Dimensions on a Break View
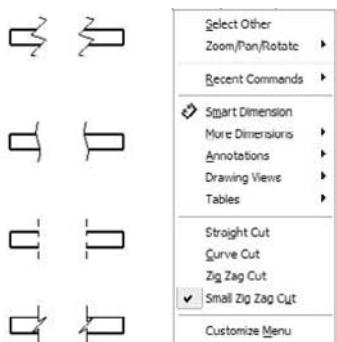
The PropertyManager in Figure 21.18 shows a bit of a discrepancy in terminology. Using "Break" as the name of the view seems a little awkward, and I much prefer it as "Broken View," as it is shown here in the PropertyManager. It is inconsistent, but you get the idea anyway.

To create a Break View, click the Break toolbar button on the Drawings toolbar, or choose it at Insert ➪ Drawing View ➪ Break. You need to place break lines in pairs, and you can choose from one of four break symbol styles, as shown in Figure 21.19. You can change the style from the RMB menu or from the PropertyManager.

Selecting the break symbol

The Broken View PropertyManager also enables you to set the gap size and the style of the break. The gap refers to the gap between the break lines in the finished broken view. The setting here overrides the default for this view only. The default setting is a template setting found in Tools ⇨ Options ⇨ Document Properties ⇨ Detailing. Other options that you can set in this location are the break line extension (the distance the lines extend past the model edges) and the break line font (on the Line Font page of the Document Properties tab). The setting that enables the broken symbol on a dimension is found on the Dimensions page, and is named Show Dimensions As Broken In Broken Views.

You can remove individual breaks in a broken view by selecting one of the break lines and pressing Delete. You can add breaks by applying the Break command and adding more breaks to a view. You can alter breaks by simply dragging the break lines. In past versions, it was possible to get the view very confused by dragging one set of breaks to interfere with another set of breaks. That problem has been fixed by not allowing break lines to be dragged past one another.

Starting with SolidWorks 2009, broken views now enable you to dimension the break lines themselves so that when the model changes, you can control the location of the break lines relative to part geometry.

Consider using the Unbreak option from the RMB menu to temporarily unbreak a view to make dimensioning more convenient.

Auxiliary View

An Auxiliary View is a view that is projected from a non-orthogonal edge. This type of view is often necessary to view features (such as holes drilled at an angle) square on, so that they appear circular in the view rather than foreshortened and elliptical. An Auxiliary View is shown in Figure 21.20 in the image to the left. If the edge that the view was created from is updated, then the Auxiliary View will reorient itself. The image to the right shows an Auxiliary View projected from an arbitrarily drawn sketch line. The line or edge used to project an Auxiliary View cannot be reselected; however, if a sketch is used to project the view, then the Edit Sketch option is available through the view arrow RMB menu.

Alternate Position View
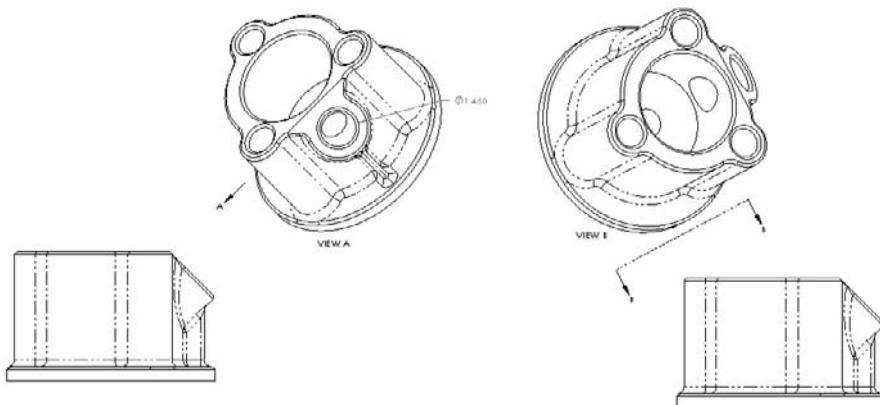
The Alternate Position View is only available for views of an assembly and shows the assembly in two different positions (not from different viewpoints; this requires an assembly that moves). This is another view type that does not create a new view, but alters an existing view. Figure 21.21 shows the PropertyManager interface for the Alternate Position View, a sample view that it creates, and the way that it is represented in the drawing FeatureManager.
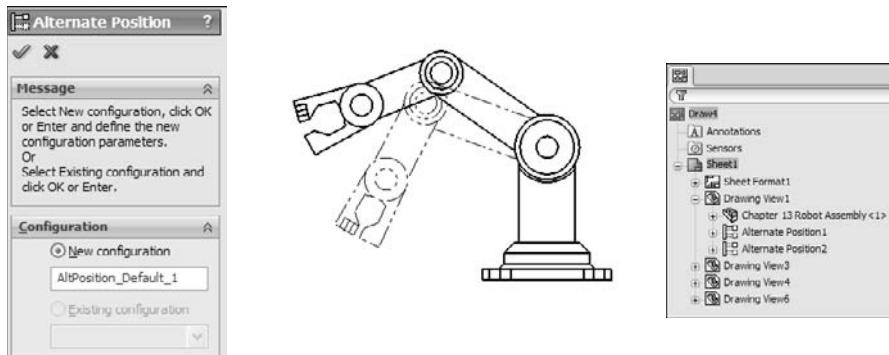
FIGURE 21.20

Two Auxiliary Views

FIGURE 21.21

The Alternate Position View

To create an Alternate Position View, ensure that you have an assembly on the active drawing that can have multiple positions, and click the Alternate Position View button from the Drawings toolbar, or choose Insert ⇨ Drawing View ⇨ Alternate Position and then select the Alternate Position View.

Next, click in the drawing view to which you want to add the alternate position. The PropertyManager shown in Figure 21.21 prompts you to select an existing configuration for the alternate position or to create a new configuration. If you choose to create a new config, then the model window appears, a new config is created, and you are required to reposition the assembly. The alternate position is shown in a different line font on the same view, from the same orientation as the original.

Tip

The best way to create this view is to either create two configurations used exclusively for the Alternate Position View or to have two configurations where you know that parts will not be moved, suppressed, or hidden. The main idea is that you need to ensure that these configurations remain in the same position or are changed intentionally, knowing that it will alter this drawing view. ∎

To delete an Alternate Position View, select it in the drawing FeatureManager, and press Delete.

Pre-defined View

Pre-defined Views are discussed in depth in Chapter 20, and are primarily used as views on drawing templates.

Empty View

Empty Views are just that — empty. The reasons for creating an Empty View can include making a view from a sketch, making a schematic from blocks, or combining several elements — such as blocks, sketches, imported drawing geometry, annotations, and symbols — into an entity that can be moved as a group on a drawing.

Custom View

You can create Custom Views by orienting the view in the model document and saving the view. Remember that views can be saved in the View Orientation window, which you can access by pressing the spacebar. Custom Views are placed on the drawing using the Named View functionality.

While not appropriate for showing dimensions, views using perspective are most useful for pictorial or illustrative views. The only way to get a perspective view on a drawing is to save a custom view in the model with perspective selected. You can access the Perspective option by choosing View➪Display➪Perspective, and you can edit the amount of perspective by choosing View➪Modify➪Perspective.

Relative View

The Relative View enables you to create a view that does not necessarily correspond to any of the standard orthogonal views or named views. This type of view is very similar to using the Normal To tool. First select the face that is to be presented square to the view, and then select the face that represents the top of the view. When this view type is initiated, SolidWorks opens the 3D model window to allow you to select the faces needed to define the view.

This type of view is particularly useful when a part has a face that is at an odd angle to the standard planes of the part. It is in some ways similar to the Auxiliary View, except that in the Auxiliary View you cannot select which face is the top.
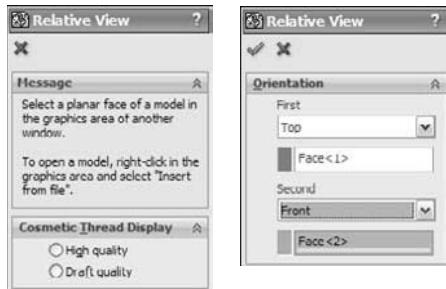
The Relative View has a special function that is important for drawings of multi-body parts. If both faces used to establish the view are from the same body, then all the rest of the bodies in the part can be hidden with an option in the Relative View PropertyManager, which is shown in Figure 21.22. This functionality is particularly useful in Weldments and is covered in more detail in Chapter 31. Multi-body modeling is covered in Chapter 26.

The Relative View PropertyManager

3D Drawing View Mode
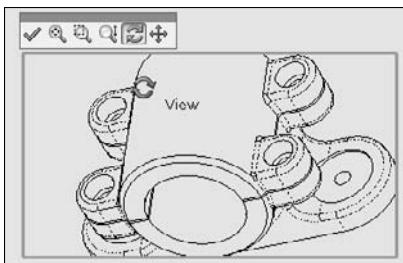
3D Drawing View Mode is not technically a drawing view type. It is a mode that enables you to select faces or edges of the model that may need to be selected for some purpose, but cannot be seen from the orientation of the drawing view. You can invoke the 3D Drawing View Mode from the 3D Drawing View toolbar button, which is on the View toolbar and can be accessed by choosing View ⇨ Modify ⇨ 3D Drawing View from the menus.

Ironically, this mode does not work for the Relative View, which would be a perfect application for it. Instead, Relative View makes you go to the model window. 3D Drawing View Mode is intended for views such as the Broken-out Section View where a depth must be selected for the cut.

In Figure 21.23, notice the small toolbar above the drawing view. This toolbar is available while the 3D Drawing View Mode is turned on. Clicking OK on the small toolbar turns off the mode and returns the view to its previous state.

3D Drawing View Mode
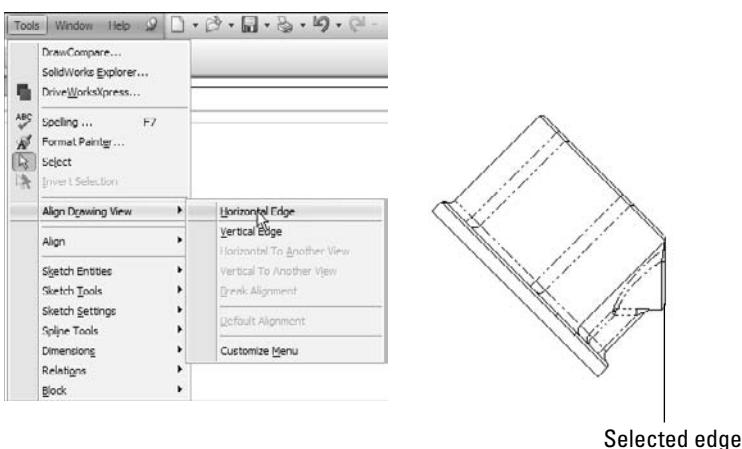
View orientation and alignment

Although you may have selected the Top view, and it displays the correct geometry, you may want to spin the view in the plane of the paper, or orient it in a particular way. You can do this using two methods. The easiest way to reorient the view is to use the Rotate View tool on the View toolbar. This rotates the view in the plane of the paper much like it rotates the model in 3D.

Another option is to select an edge in the view and assign the edge to be either a horizontal or vertical edge. Figure 21.24 shows how a view can be reoriented using this tool, which you can locate by choosing Tools ⇨ Align Drawing View ⇨ Horizontal or Vertical Edge.

Rotating a drawing view to align an edge

Selected edge

Another option for view alignment is to align it relative to another view; this involves stacking one view on top of another or placing them side by side. You can do this by selecting the second pair of options in the menu shown in Figure 21.24, Horizontal to Another View and Vertical to Another View. These are grayed out in the figure, but preselecting a linear edge before selecting this option in the menus will activate them.

Situations may arise where a view is locked into a particular relationship to another view, and you need to disassociate the views. The Break Alignment option, which is grayed out in the menu in Figure 21.24, serves that purpose.

Using Display Options in Views

Some important display options and settings are not listed in the Tools ➪ Options menu; they are only available through the menus, and in particular the View menu. You can effectively deal with most items in the View menu by assigning a hotkey that can be toggled on or off. For example, Axes and Temporary Axes are things you often want to be visible when you're sketching something, but not visible when printing a drawing. You can easily assign the display for Axes and Temporary Axes to hotkeys, making them ready at your fingertips. You can assign hotkeys by choosing Tools ➪ Customize ➪ Keyboard.

Display States

You can use Display States in drawing views, but unless you are only hiding and showing parts with the Display States (that is, you are not changing colors or display styles with Display States), they only have an effect when a drawing view is set to Shaded Display style. You can control Display States for drawing views in the View PropertyManager. The Drawing View Properties dialog box appears, as shown in Figure 21.25.
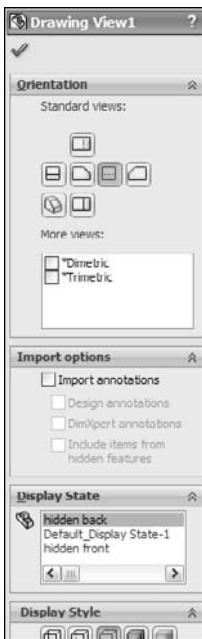
One of the limitations of the Display States functionality in drawing views is that when wireframe display is used, the drawing edges appear in black rather than using the color settings to show wireframe in the same color as shaded. The necessary color settings are found in two places, and you need to set both. The System Options setting is on the Colors page and is called Use Specified color for shaded with edges mode. The second setting is in the part Document Properties (not assembly), again on the Colors page, and is called *Apply Same Color to Wireframe, HLR (Hidden Lines Removed), and Shaded*.

Display styles

The 2D drawing world is becoming less and less black-and-white and SolidWorks has the capability to apply shaded views to drawings. This is probably most useful in isometric, perspective, or pictorial views on the drawings. The shading and color may be distracting for dimensioned and detailed views, but it can also be indispensable when you need to show what a part actually looks like in 3D. Not everyone can read engineering prints, and even for those who can, nothing communicates quite like a couple of shaded isometric views.

FIGURE 21.25

The View PropertyManager

The more standard 2D drawing display modes are Wireframe, HLR, and HLV (Hidden Lines Visible), which work in the same way as they do in the model environment. Unless you override it on a per part basis, the Display mode is set for all the components in the view.
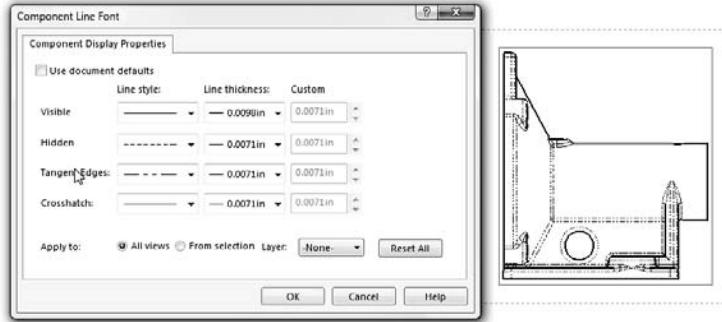
Component Line Font

Individual components within an assembly can be shown in different fonts, similar to the display in the Alternate Position View. You can access this function by right-clicking the component to access the RMB menu and selecting Component Line Font. Figure 21.26 shows the Component Line Font dialog box, along with a drawing view in which a couple of part line fonts have been changed. The part can only be changed in the view where it was selected, or it can be changed across the board in all views in the active drawing where it appears. This is useful if you want to emphasize or de-emphasize certain parts in the assembly view.

The Component Line Font dialog box
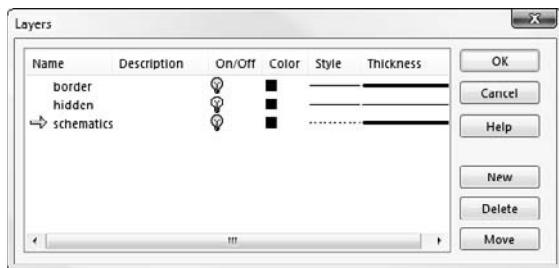
Layers

Yes, SolidWorks drawings *can* use layers. You can place individual parts onto layers, and the layers can have different colors and line fonts. Most entities can be put into layers, including edges, annotations, and sketch items. Hidden layers are often used for reference information or construction entities on a drawing. Figure 21.27 shows the Layers dialog box on a SolidWorks drawing.

Using the Layers dialog box in SolidWorks
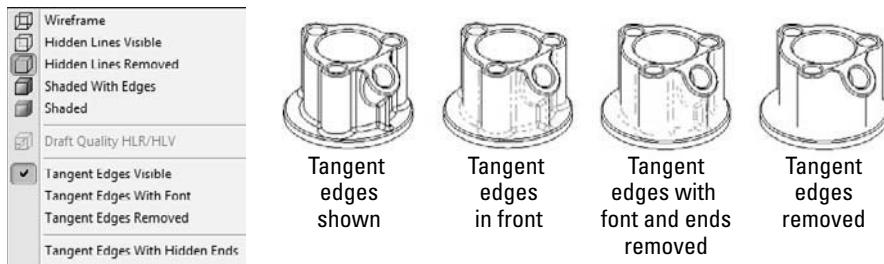
Tangent edge display options

SolidWorks drawings and models offer some options for displaying tangent edges. Many users find it distracting when tangent edges (which in a physical part are not edges at all) are given as much visible weight as the sharp edges of, say, a chamfer. To find the settings, shown in Figure 21.28, choose View⇨Display, or select them from the RMB menu. The Tangent Edges Removed option may be appropriate for parts with few fillets, but it causes a part to look oversimplified and makes details of the shape difficult to distinguish.

FIGURE 21.28

Edge display options

Tangent edges shown

Tangent edges in front

Tangent edges with font and ends removed

Tangent edges removed
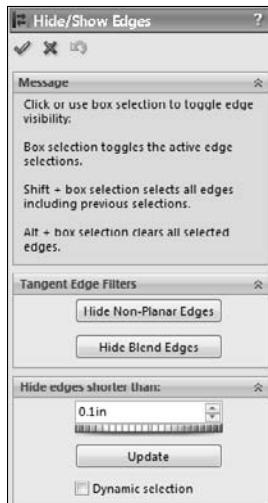
Additional options are available, including applying a color to the tangent edges by choosing Tools ⇨ Options ⇨ Color list for Tangent Edge display, and the Hide/Show Edges options.

SolidWorks 2010 replaces the individual Hide Edge and Show Edge toolbar icons with a single icon called Hide/Show Edges. You can use this tool for specific selected edges, or just click the tool with nothing selected to activate the Hide/Show Edges PropertyManager, shown in Figure 21.29. Hide/Show Edges functionality is only available with high quality views. View quality is addressed in the next section.

FIGURE 21.29

The new Hide/Show Edges PropertyManager offers additional tangent edge display options.

The PropertyManager indicates that Hide/Show Edges works from a selection, although it has no list box to show the edges selected. The tools offered here may not have a wide general appeal, but they will be most useful probably to people documenting complex shapes or plastic parts. Hide Non-Planar Edges is meant to simplify the display of complex parts with a lot of curvature. Hide Blend Edges limits the filtering to edges with curvature continuity across the edge, which is usually found only in more sophisticated surfaced parts. Hide Edges Shorter Than gives you the ability to filter the edge display again. These settings apply only to the current view, not to the entire drawing.

Take note of the options in the Hide/Show Edges PropertyManager message box shown in Figure 21.29:

- Box selection toggles the active edge selections
- Shift+box selection selects all edges, including previous selections
- Alt+box selection clears all selected edges

These mass selection options can be important if you have used one of the Tangent Edge Filters, Hide Non-Planar Edges, Hide Blend Edges, or edges shorter than a given value.

View quality settings

You have the choice between two options for drawing view quality: High quality and Draft quality. The quality that you choose influences the performance of the software. Draft quality views are noticeably rough when viewed closely, but from a distance, they are at least recognizable. However, Draft quality is becoming less accessible, and so I would not recommend relying on this option. Although new Draft quality views can be created, once they are set to High quality, they cannot be set back to Draft quality.
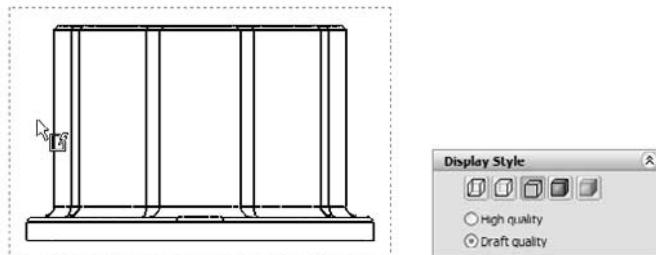
All views are created as High quality unless the view quality setting is overridden. To find this set-ting, choose Tools ➪ Options ➪ System Options ➪ Drawings ➪ Display Style ➪ Display Quality For New Views. The only other way that you can create Draft quality views is if you open a drawing from an older version of SolidWorks that used Draft quality views.

In Figure 21.2 earlier in the chapter, the image to the right shows the Display Style pane. This PropertyManager has been taken from a High quality view. A Draft quality view enables you to toggle between Draft and High quality, as shown in Figure 21.30. This means that you can switch a view from Draft to High, but not from High to Draft. Also notice in Figure 21.30 that the cursor over a Draft quality view displays a lightning bolt symbol, indicating draft quality.

You can access the Cosmetic Thread Display setting in both the Step 1 PropertyManager and the Step2 PropertyManager. However, you need to be careful not to misread the interface by thinking that either of these interfaces controls the View Quality.

The Draft quality options and cursor

Distinguishing Views from Sheets

It is sometimes difficult for new users to understand the difference between being *in* a sketch and being *out of* a sketch, or the difference between editing the *sheet* as opposed to the *sheet format*. In the same way, confusion frequently surrounds the difference between sketching in a view and sketching on a sheet. The easiest way to determine if a sketch will be associated with a view or with the drawing sheet is to look at the prompt in the lower-right corner of the SolidWorks window, on the status bar, which displays the message, Editing Sheet, Editing Sheet Format, or Editing View.

This issue becomes especially important when you want to do something with a sketch entity, but it is grayed out and unavailable. This means that whatever entity is active is *not* the one that the sketch entity is on. Drawing views expand to contain all the sketch entities that are associated with the view, and so if you see a view that is extended on one side, larger than it should be, then it could be extended to contain the grayed-out sketch entity. Activate the sheet and the suspected views; when the sketch entity turns from gray to black, you have found the place where it resides.

Tutorial: Working with View Types, Settings, and Options

This tutorial is intended to familiarize you with many of the view types, settings, and options that are involved in creating views. To begin, follow these steps:

1. **From the CD-ROM, open the part called** `Chapter 21 – Tutorial Part.sldprt`.

2. **Move the drawing template named** `Inch B Bible Template.drwdot`, **also found on the CD-ROM, to your templates folder.** If you do not know where your templates are located, choose Tools ➪ Options ➪ System Options ➪ File Locations ➪ Document Templates.
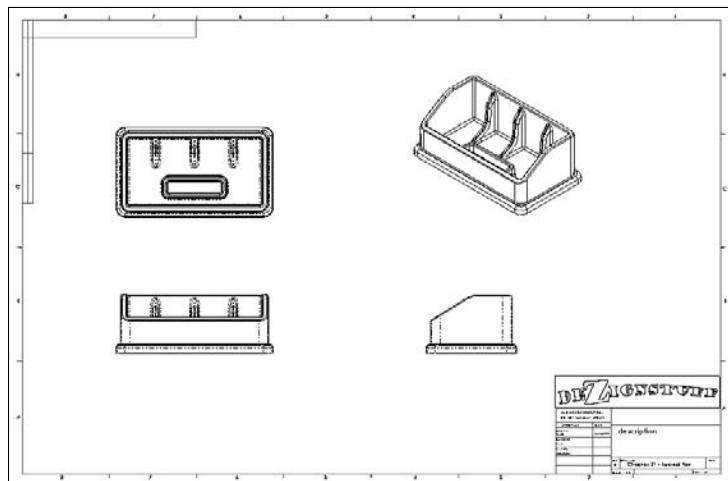
3. **From the window with the open part, click the Make Drawing from Part button from the toolbar.** The drawing becomes populated with three standard views and an iso-metric view, as shown in Figure 21.31.
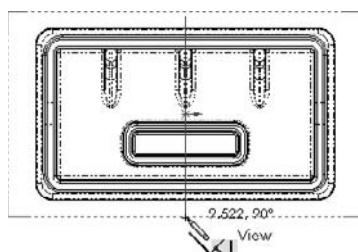
Using a template with Pre-defined Views

4. **In the drawing document, select the display of the Origins.** This will help you to align a section view. You can display Origins through the menus by choosing View⇨Origin.

5. **Click the Section View tool on the Drawings toolbar.** This activates the Line sketch tool.

6. **In the Top view (in the upper-left section of the drawing), draw a vertical line that picks up the inference from the Origin.** You may have to run the cursor over the Origin to activate the inference lines. Make sure that the line goes all the way through the model geometry in the view, as shown in Figure 21.32. When you finish the line, the sec-tion view is ready to be placed. Place it to the right of the parent view.

Creating a section view

When sketching, remember to make sure that you are sketching in the view rather than on the sheet. A section view cannot be created from a sketch entity if it is not in a view. A glance at the status bar in the lower-right corner of the window lets you know if you are in Editing View or Editing Sheet.

To change the letter label on the drawing, click the section line and change the label in the top panel of the Section View PropertyManager.
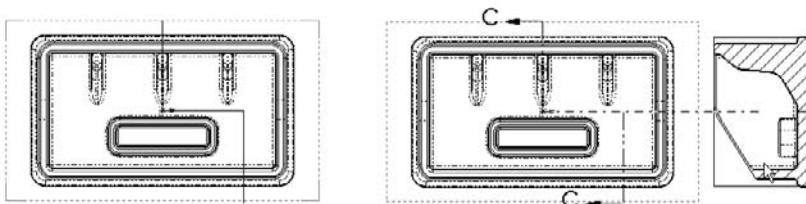
Note that you may see a Section Scope dialog box asking you to omit rib features. Ribs are usually not hatched in section views.

7. **Bring the cursor over the corner in the section line until the cursor looks like the image to the left.** Double-click the cursor; the section arrows flip to the other direction, and the drawing view becomes cross-hatched. The cross-hatching indicates that the view needs to be updated.

8. **Press Ctrl+Q; the view updates, removing the cross-hatching.**

9. **Click the section line and press Delete.** Answer Yes to the prompt. You may also need to separately delete the sketched section line.

10. **Create a new section view using a jogged section line, as shown in Figure 21.33.** In order to do this, you must pre-draw the jogged section line, and press the Section View button with the part of the line that you want to use to project the new view.

FIGURE 21.33

Creating a jogged section view
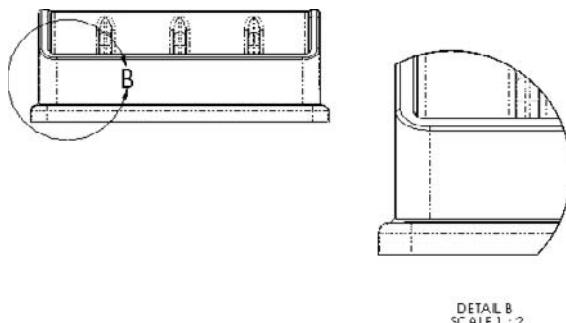
11. **Next, click the Detail View button on the Drawings toolbar.** This activates the Circle sketch tool.

12. **Sketch a circle in the Front view, located in the lower-left section of the drawing.** Try not to pick up any automatic relations to the center of the circle. One way to prevent this is to hold down the Ctrl key when creating the sketch.

13. **Place the view when the circle is complete.** Note that the view was created at a scale of 1:2. The sheet scale is 1:4, and so the detail is two times the sheet scale. The Detail View is shown in Figure 21.34.
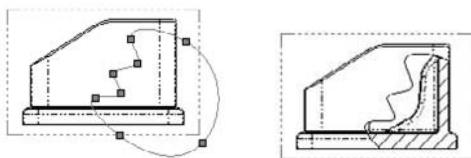
Creating a Detail View

DETAIL B
SCALE 1 : 2

14. **Drag the circumference of the circle and watch the view dynamically resize.**

15. **Leave the Detail circle selected so that the center of the circle is highlighted.** Drag the center of the circle around the view. The effect is like moving a magnifying glass over the part. If you drag the center with the Ctrl key pressed, then you will not pick up any automatic sketch relations when you drop it somewhere.

16. **Click the Broken-out Section View tool on the Drawings toolbar.** Draw a spline on the Right View similar to the one shown in the image to the left in Figure 21.35. Use a section depth of 1 inch, Splines take a little practice.

Creating a Broken-out Section View
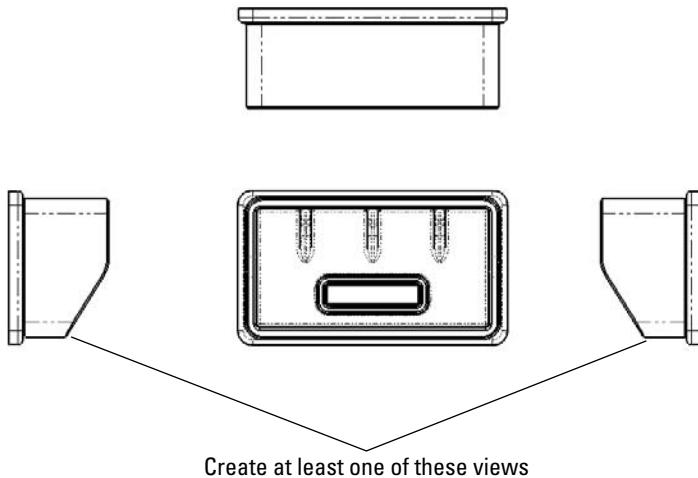
17. **Click inside the view border but outside of the part in the Top view (in the upper-left section of the drawing).** Press Ctrl+C.

18. **Click the Add Sheet icon to the right of the sheet tab in the lower-left corner of the drawing that says Sheet1, and select Add New Sheet.** If you used the template that I provided, a message may appear, saying that SolidWorks cannot find the format. This is because I only supplied you with the template file, not the format as a separate file. In any case, switch to the B size format and accept.

19. **Click any spot inside the sheet and press Ctrl+V.** SolidWorks pastes the copied view from the other sheet. Delete the section line. You may also need to delete the sketch lines separately.

20. **Click the Projected View tool from the Drawings toolbar, and then click the pasted view.** Practice making a couple of projected views, including dragging one off at a 45-degree angle to make an isometric. Make sure that one of the views is a side view showing the angled edge, as shown in Figure 21.36. Once you create the views, click model edges in the views and drag them around to a better location.

FIGURE 21.36

Projecting views

Create at least one of these views

21. **Select the angled edge from one of the side views and click the Auxiliary View toolbar button.** While placing the view, press and hold the Ctrl key to break the alignment. You can resize the view arrow by selecting the corners and dragging. If you drag the line itself, then you can move it between the views. Alternatively, with the view arrow selected and the PropertyManager displayed, you can deselect the green check mark icon in the Arrow panel at the top of the window to turn off the arrow.

22. **Create a new drawing from the New dialog box.** Select a template without predefined views on it, so the Inch B Bible Template (no Views).drwdot will work. If you select a default SolidWorks template, you need to verify that the template uses thrid-angle rather than first-angle projection. An easy way to do this is to switch the drafting standard from ISO to ANSI in Tools➪Options➪Document Properties➪Drafting Standard. If the automatic Model View interface appears in the PropertyManager, click the red X icon to cancel out of it.

23. **Expand the Task pane and activate the View Palette (the tab that looks like a drawing icon).** Click the ellipsis button (...) and browse for the assembly named `Chapter 21. SF casting assembly.sldasm`. This is shown in Figure 21.37.
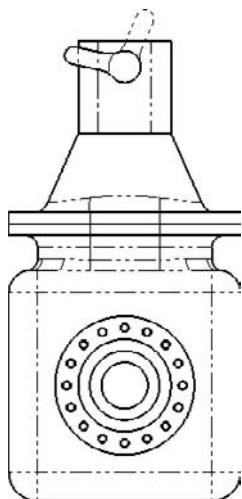
The View Palette

24. **Drag the Back view onto the drawing.** Notice that when you use this technique, the views do not resize automatically, regardless of the setting at Tools ➪ Options ➪ Drawings ➪ Automatically Scale New Drawing Views.

25. **Delete any view that you have created using this method.** Open Windows Explorer, browse to the assembly, and drag it into the drawing. The views that you create using this method are equivalent to the Standard 3 View tool. This time, the views auto-size.

26. **Select the Front view and change it to the Back view.** Notice that the rest of the views change to reflect the new parent view. You will get a warning about this change.

27. **Zoom in on the Back view.** Change the view to show Tangent Edges With Font through View ➪ Display. You can also change this from the view RMB menu.

28. **Click the Alternate Position View toolbar button.** Type a name in the PropertyManager for a new configuration and click the green check mark icon. SolidWorks opens the assembly model window.

29. **Rotate the handle 90 degrees and click the green check mark icon.** SolidWorks returns to the drawing and shows the new position in a dashed font, as shown in Figure 21.38.
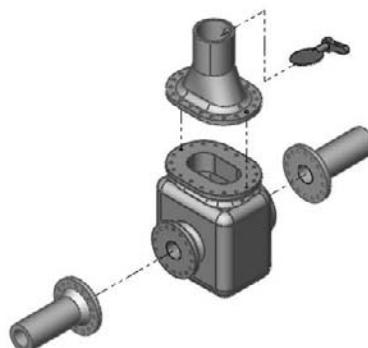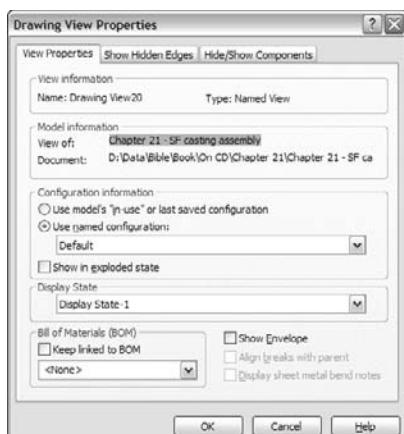
Creating an Alternate Position View

30. **Place an isometric view on the drawing.** Change the Display Mode to make it a shaded view.

31. **Right-click inside the view but away from the parts, and select Properties.** The dialog box appears, as shown in Figure 21.39. Make sure that the view is set to use the Default configuration, and also select the Show in exploded state option.

The Drawing View Properties dialog box

Summary

SolidWorks has the capacity to make many different types of views of parts and assemblies. In addition to the tools for projecting views, custom views saved in the model document can be saved and used on the drawing. The associative nature of the drawing to the model helps ensure that drawing views, regardless of how unusual the section angle or view orientation, are displayed in the correct size, location, and geometry.

It is sometimes better to create some of the views that require sketches by pre-sketching. Make use of workflow enhancements when possible; for example, the Broken-out Section automated work-flow works well, but forcing it to be a manual process makes it awkward to use.

Using Annotations and Symbols

Annotations and symbols are a major component of communicating a design through a drawing. SolidWorks has several tools available to help you manage these entities to make engineering drawings look good and communicate effectively.

Using Notes

A Notes are the workhorse of SolidWorks annotations. You can use notes in many different configurations and mix them with links to custom properties, hyperlinks, and text wrapping boxes. You can also use them with styles, leaders, symbols, and balloons; and you can even embed balloons into notes.

Setting up a workflow for placing notes

Sometimes users have difficulty working through some of the interfaces in SolidWorks. This is not necessarily the fault of the software, but is often because users may not fully understand how the workflow of a particular feature is supposed to function. The Model View interface from the last chapter is one that can be confusing until you have been through it a few times and gain a more intuitive feel for how it works.

Understanding the workflow is paramount to being able to use the software efficiently. I sometimes find myself using the annotations clumsily, and sometimes wind up with blank notes, double notes, or extra lines at the ends of notes. After using the tool a few times, I get back in the groove.

For these reasons, I have added some step-by-step suggestions here to help you create an efficient workflow with annotations.

Follow these steps to create a note:

1. **Click the Note toolbar button on the Annotations toolbar.**

2. **Click in the graphics window where you want to create the note *or* click an entity that you want the note leader to point to, and *then* click where you want the note.**

3. **Type the note.** Press Enter at the end of a line, or, if you intend to force the note to wrap later, just allow the line of text to be as long as it wants to be. While you create the note, the text box expands to the right until you press Enter, and it expands down every time a line is added.

   At the end of the last line of the note, *do not* press Enter again (this creates extra lines), but you may press Esc. Esc gets you out of the note and ready to place a new note. When you press Esc twice, you get out of the note you were typing, and then get out of the Note command altogether.

4. **Another way to finish the note is to click the mouse outside of the text box. After that, if you are done, press Esc.** If you want to continue with another note, click again to place it, and start typing. If you want to place the same note as the first one again, the text is already there, so click a second time.

Making use of fonts

SolidWorks can make use of any TrueType fonts that Windows will accept. This includes symbol, non-English, and Wingding fonts. SolidWorks does not use true monofonts like AutoCAD because they do not have width information. Some AutoCAD monofont look-alike fonts are installed with SolidWorks that do have a very narrow width, and are shaped like some of the monofonts.

In the Customize dialog box (Tools ➪ Customize), the Fonts toolbar displays as the Formatting toolbar. The Formatting toolbar also appears in the graphics area immediately over your text every time you either insert a new note or edit an existing note, unless the toolbar is already docked somewhere. The Formatting toolbar is shown in Figure 22.1.

FIGURE 22.1

The Formatting toolbar

Using text boxes and wrapping

Text boxes enable you to limit the size, particularly the width, that a note can occupy. This enables notes to wrap in tight spaces on title or revision blocks, as well as other places.

You can size text boxes immediately after placement, even while they are blank; the text then wraps as you type it. The text box expands downward automatically. Blank text boxes can be left

on the drawing to provide a placeholder for future text. The blank text box has a rectangular border that contains an X, both of which are removed when you add text. If spaces are added to a text box, the text box becomes invisible, although you can select it if you know where it is. When you move the cursor over the text box, the cursor displays the note symbol.

While typing a note, it is not possible to make the note box smaller using the middle handle on the right end of the box; you can only stretch it larger. Using any corner handle, as shown in Figure 22.2, you can make the box taller, wider, or narrower. The note box will not resize smaller if the text string it contains does not contain spaces.

FIGURE 22.2

Resizing a text box using the lower-right corner handle

Do not resize with middle handle

Use corner handle to resize

If a custom property value is used to populate a note and you select the Annotation Link Variables option in the Views dialog box, when you activate the text box to resize it, the text value will go away and the link variable will be displayed. This makes it difficult to dynamically resize the box to fit the note, so it might be best to deselect the Annotation Link Variable option before placing and sizing notes.

Placing notes and leaders

When you start to place a note, a preview shows the text box with or without a leader depending on the position of the cursor. If the cursor is over a blank section of the drawing, the note is placed without any leader. If the cursor is over a face, edge, or vertex, then a leader is added using the arrow found in the Attachment dialog box at Tools ➪ Options ➪ Document Properties ➪ Arrows ➪ Attachments. By default, a leader attached to a face uses a dot as an arrow, and a leader attached to an edge, sketch entity, or nothing at all uses a regular arrow. You can change these defaults at the options location mentioned previously, and you can change individual note leaders in the PropertyManager that becomes available when you select a note. These settings can also become part of a custom drafting standard.
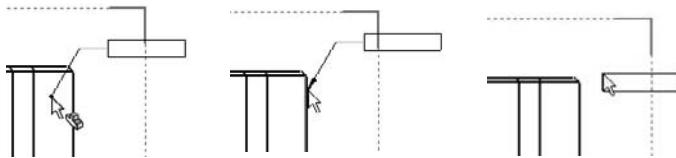
Figure 22.3 shows the preview that is displayed by the cursor when you place a note over a face, an edge, and blank space on the drawing.

FIGURE 22.3

Placing a note with a leader

You can also change settings for bent leaders by choosing Tools➪Options➪Document Properties. It is recommended that you use the same bent leader lengths for all annotations, and save them in the templates that you use.

Single-clicking inside an active text box places the cursor between letters, as expected. Double-clicking inside an active text box selects the entire word that you click, again as expected. Ctrl+A selects all the text inside a text box. If you double-click an existing note to activate it, the entire contents are highlighted immediately. You also cannot drag-and-drop selected text to move it within a text box. However, you can Ctrl+C, Ctrl+X, and Ctrl+V the text.

To format the entire note, do not activate the text box; instead, only select the note, and apply the setting to the entire note rather than to selected text within the note.

Add a leader to a note

To add a leader to a note that was created without a leader, click the note and select the leader options in the Leader panel of the PropertyManager, as shown in Figure 22.4. After you add the leader, you can reposition the handle at the end of the leader to attach it to an entity on the drawing.

Multiple leaders
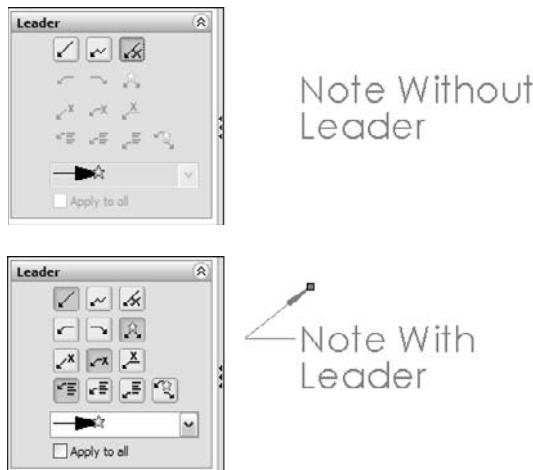
You can also attach multiple leaders to notes. To create a new note with multiple leaders, pre-select the entities that the leaders are to be attached to, and then click the Note toolbar button.

To add a leader to an existing note, first click the note, and then Ctrl+drag the handle or small dot on the end of a leader to the second location. A note with multiple leaders is shown in Figure 22.5. To remove one of multiple leaders from a note, click the handle at the end of the arrow and press Delete.
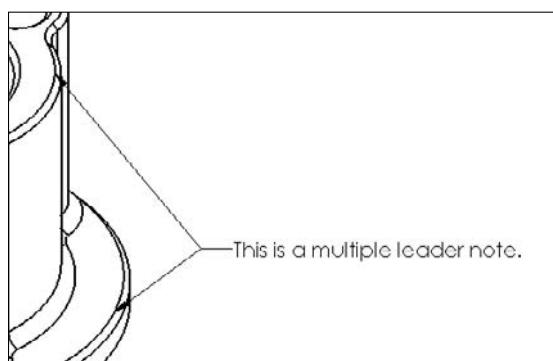
FIGURE 22.4

Adding a leader to a note

FIGURE 22.5

A note with multiple leaders

Jogged leaders

Jogged leaders have come a long way since their introduction many releases ago. You can switch a regular leader to a jogged leader by selecting an option in the PropertyManager. In Figure 22.4, the middle icon in the top row is the Jogged Leader icon. The icon to the left simply turns on the default leader, and the icon to the right turns off leaders altogether.
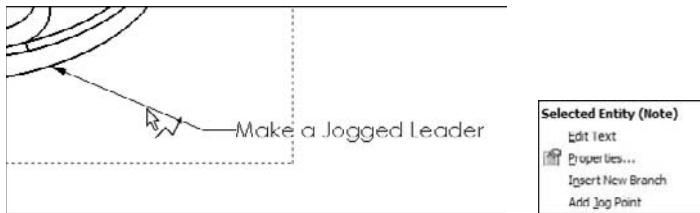
Once you activate the jogged leader option, you can add a jog from the leader right mouse button (RMB) menu. Notice in Figure 22.6 that two options give you control over the jogged leader — Insert New Branch and Add Jog Point.

Jogging a leader

Add Jog Point
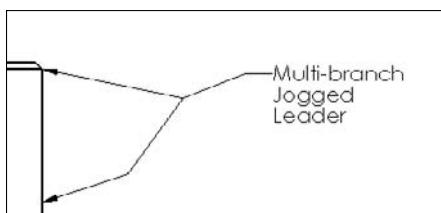
Selecting the Add Jog Point command adds a new handle to the leader that you can move around. You can add multiple jog points to the leader.

Insert New Branch

The Insert New Branch command enables you to create a new jogged leader that ends in another arrow from the selected point. This arrangement with multiple branches in a jogged leader is shown in Figure 22.7.

The results of adding a new branch to a jogged leader

Adding styles

For notes, a style can apply a font, an underline, bold formatting, or any other setting from the Formatting (Fonts) toolbar to a note.

To create a note that uses the style setting from another existing note, pre-select the existing note with the style to be copied before starting the Note command; SolidWorks applies the style to the new note.

Caution

Sometimes adding a style to a note can make other changes that you may not expect, such as deselecting the leaders if a note has multiple leaders. In particular, if the style is made from a note with a jogged leader, then it deselects leaders for regular multiple leaders. Styles that are created from regular leader notes do not deselect jogged leaders.

Making a change to the leader of a note after you apply the style removes the style from the note, although the formatting remains. This does not apply to adding multiple leaders, only to changing the type of leader.

Applying a style may also remove the ability of the text to wrap, as well as any changes to the text box shape. You cannot move the corner of a text box of a note to which you have applied a style. ■

Styles exist only in the document in which they were created, but they can be shared to other documents by saving the style out as a separate file. Note styles use the extension `*.sldnotestl`. Once you save the style, you can load it into other documents. The Style panel of the Note PropertyManager interface is shown in Figure 22.8.

Annotation types that can use styles are

- Note
- Dimension
- Weld Symbol
- Surface Finish
- Datum Feature
- Datum Target
- Balloon
- Auto Balloon
- Stacked Balloon
- Center Mark

FIGURE 22.8

The Style panel of the Note PropertyManager interface

Note

Styles before the 2009 release were called Favorites. The file type for note favorites was `*.sldnotefvt`. If you run across any of these legacy file types, you will at least know what they are. When loading styles, the Open dialog box will recognize and use both the new file type and the legacy file type. ■

The Style panel contains the following buttons:

- **Apply Defaults/No Styles.** Removes style settings from the current interface, setting all values back to the defaults.

- **Add or Update Styles.** Either adds a new style to the database or changes the name or other settings for an existing style.

- **Delete Style.** Removes a style from the database.

- **Save Style.** Saves a style to an external file (`*.sldhwfvt`), which can be loaded by other users and added to their databases.

- **Load Style.** Loads a saved style file.

Styles can be loaded into document templates so that for every document created from the template, those Styles will be available.
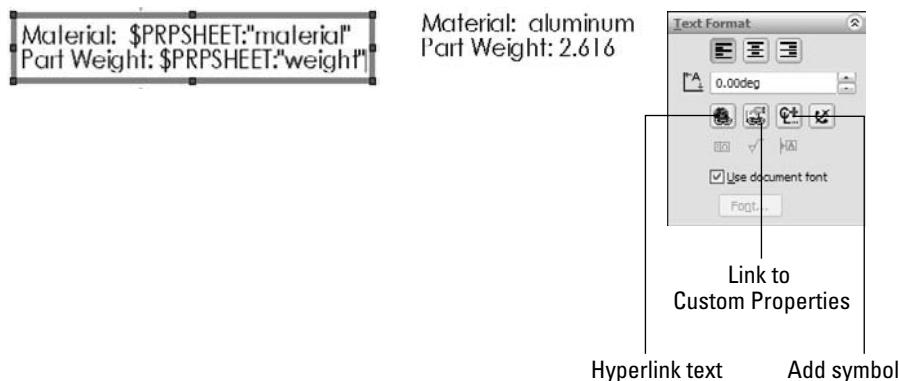
Linking notes to custom properties

You can link notes to custom properties. The custom properties can be from the drawing, or from the model that is referenced by the drawing. I mention this kind of link briefly in Chapter 20, but discuss it more thoroughly here.

Figure 22.9 shows a note on a drawing with custom property links pulling data from the model shown on the drawing. To add these links, driven by the syntax `$PRPSHEET:"material"`, click the icon indicated in the image to the right in Figure 22.9.

FIGURE 22.9

Linking notes to custom properties

In this case, text has been combined with custom properties, but custom properties can also appear alone. To access the Custom Properties interface, choose Tools➪Properties.

When you activate the note, you may want to see the syntax, or you may want to see the actual text value of the custom property. You can find the setting that controls which one is displayed by choosing View➪Annotation Link Variable.

Hyperlinking text

Hyperlinking text is sometimes useful on drawings to provide a link to reference documentation, specification, test results, and so on. The first button in the third row of the Text Format panel of the Note PropertyManager enables you to add a hyperlink to text in the note. Figure 22.9 shows this panel. Either copy the URL to the hyperlink dialog box that appears, or browse to it from the dialog box.

Adding Notes and symbols

Notes and symbols are regularly combined in SolidWorks. Symbols are discussed more fully later in this chapter, but are mentioned here because of the frequency with which they are used with notes. In reference to Figure 22.9, the image to the right shows the Text Format panel, which contains a button to the interface where you can add symbols.

Using Blocks in Drawings

Blocks in SolidWorks can contain sketch elements and notes. When used in drawings, blocks have several common uses, including the following:

- You can use standard note blocks for tolerances, disclaimers, or default requirements.
- You can put together a mechanism in 2D where each block represents a part.
- You can use flow direction for fluid systems.
- You can use drawing stamps such as "Not For Release," "Preliminary," "Obsolete," and so on.
- You can use symbols for schematics that can be snapped together.
- You can save drawing formats as blocks to make them easier to place as a single entity.

Like styles, blocks reside in the document in which they are created, but you can save them out to a `*.sldblk` file, load them into other documents, and save them as a part of a document template.
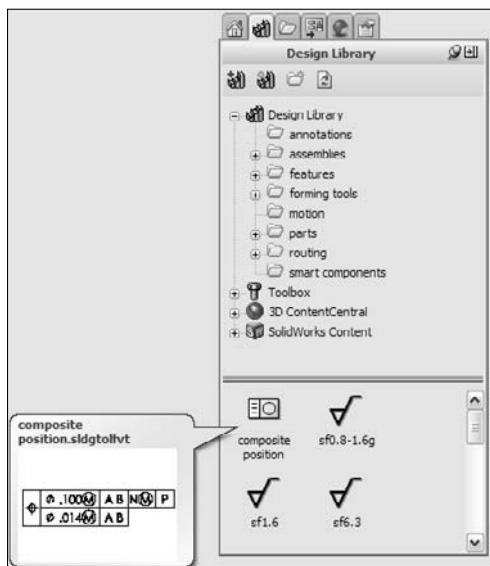
Inserting blocks

You can apply blocks in several ways, including by dragging from Windows Explorer and by using the Block menus (Insert ⇨ Annotations ⇨ Block). However, the most efficient way is to access them from the Design Library. Library folders can be established specifically for blocks. Check the setting by choosing Tools ⇨ Options ⇨ File Locations ⇨ Blocks, and redirect this setting to a library area outside of the SolidWorks installation directory. Figure 22.10 shows the Design Library with a folder containing blocks that are selected. The blocks do not show previews in the window, but the tool tip displays large previews. You can drag blocks from the Design Library onto the drawing sheet.

FIGURE 22.10

Blocks in the Design Library

Each block has an insertion point, which snaps to any sketch entity point, even if it is in another block. This makes schematics easy to snap together. If the default insertion point is not the point that you need to snap to the other geometry, then you can place the block anywhere on the drawing and drag the point that needs to snap.

Once blocks are snapped together, to detach them from one another, you can click the point at which they touch; a Coincident sketch relation displays in the PropertyManager. Deleting the sketch relation enables you to drag the block away from the other geometry.

When blocks are inserted, you can control several options in the PropertyManager. This function may be somewhat hidden because it does not appear automatically when you place the block. After you place the block, SolidWorks wants you to place another copy of the block. If you press Esc to cancel out of placing additional blocks, the first placed block is not selected, so the PropertyManager does not display. Figure 22.11 shows the Block PropertyManager.

The Block PropertyManager

Following is a list of sketch relations linked to the block.

- **Existing Relations.** This panel lists the sketch relations that are linked to the block. These may cause the block to not move properly when you drag it. This feature is most helpful when the block is being used as a representation of a part in a simulated 2D mechanism.

- **Add Relations.** This panel enables you to quickly select sketch relations to apply when placing blocks.

- **Definition.** Blocks can be linked to an external file, which enables all linked instances of a block to be updated at once, even if they are being used in other drawing documents. The path box for the Link to File option only displays if you select the check box.

  The Edit button refers to editing the block. A toolbar button also exists for editing blocks. Clicking the Leader & Insertion Points button enables you to edit both of the controls. You can select the For Construction option to change the entire block to construction entities.

- **Parameters.** The top field with the two circles to the left controls the scale of the block. This number affects the entire block, including the text. You also have the option to scale dimensions, so that the dimension text size (not the dimension value) increases with the overall block scale.

  The Lock Angle option refers to the rotation of the block. If the Lock Angle option is not selected, then you can rotate the block if one point on it is coincident to a stationary object, such as a vertex in a drawing view.

- **Leader.** You will recognize these options from the Notes leaders. The leader is attached to the block where the angled black handle was placed when you created the block. You can edit the leader connection and insertion points by clicking the Leader & Insertion Points button on the Definition panel.

- **Text/Dimension Display.** The Display Dimensions option controls whether or not any notes and dimensions in the block are displayed or hidden.

- **Layer.** You can assign most entities on drawings to layers, which in turn have controls for items, such as line type, color, and visibility.

Creating blocks
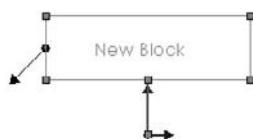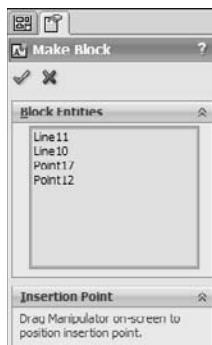
You can create blocks by selecting the sketch and annotation elements and clicking the Make Block toolbar button in the Blocks toolbar, or by accessing the command by choosing Tools ➪ Block ➪ Make from the menus.

By default, when you create a block, the Insertion Point panel of the PropertyManager does not expand. If you expand this panel, the blue Origin symbol represents the insertion point that is attached to the cursor during block insertion, as shown in Figure 22.12. The angled line hanging off the left side of the block is the leader attachment point for the block. You can also drag this line around the block and snap it to sketch geometry. By default, this block does not use a leader, but if one is required, then you can select when you place the block.

FIGURE 22.12

Creating a block

Cross-Reference

Sketch blocks are covered in some detail in Chapter 3. This chapter is limited to a discussion of blocks that may be found on drawings rather than those used in model sketches or Layouts. ∎

Editing blocks

Although you do not have many options when creating blocks, many more options become available when you edit them. You can access editing options for a block from four locations:

- The Edit Block toolbar button on the Blocks toolbar
- The Edit button in the Block PropertyManager
- Choosing Tools ➪ Block ➪ Edit from the menus
- From the RMB menu of the block in the Blocks folder in the drawing FeatureManager

The standard edit function gives you access to the sketch and note elements that make up the block.

- **Add/Remove Entities.** While you are editing the block, the Add/Remove Entities button on the Blocks toolbar becomes available. This enables you to add or remove sketch or note entities from the block definition.

- **Rebuild.** Rebuild Blocks reapplies sketch relations within the block, without exiting Edit Block mode.

- **Explode.** Explode is available when you are not editing the block, but when it is selected. Explode returns the contents of that particular instance of the block to the drawing, removing them from the block. This removes any leaders that are attached to the block, as well as sketch relations. This is not technically an edit option, but it certainly does change things.

Using Symbols

SolidWorks symbols are different from symbols that are a part of a font family. SolidWorks symbols fall into several categories, including weld, surface finish, hole, modifying symbols, GD&T (geometric dimensioning and tolerancing), and several flag symbols. You can also construct custom symbols.

Using symbols in notes and dimensions

You can use symbols in notes and dimensions. They also are an intrinsic part of weld symbols and surface finish symbols. Hole Callouts use symbols extensively, as do GD&T frames.
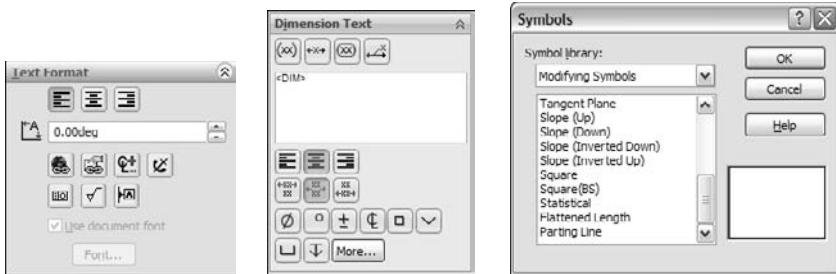
Figure 22.13 shows the Text Format panel from the Note PropertyManager and the Dimension Text panel from the Dimension PropertyManager. Both of these interfaces give you access to the symbol library.

FIGURE 22.13

Accessing symbols and the symbol library

Creating custom symbols

You can create custom symbols in SolidWorks, but creating them may not be as simple as you expect. In the `lang\english` subfolder of the SolidWorks installation directory is a file called `Gtol.sym`. This file stores the representations of all the SolidWorks symbols. It is also where you can create symbols of your own. You can edit the file in Notepad.

Be warned, however. Unless you enjoy writing scripts for the command line, or you are a fan of DOS 5.0, you may not want to create custom symbol projects. The format for creating symbols is simple enough, but it is what you might call somewhat arcane. It is effective at creating line-art symbols that can be used with text and can even be used to contain text. If you are a little inventive with this, you can create interesting shapes that integrate with your notes and dimensions.

Keep in mind that this topic does not appear in the Help files, but all the instructions you need are inside the file itself. You may have to experiment a little to discover what the rules are in terms of making shapes outside of the limits of the 1X1 matrix. It is probably easier to create the geometry using Blocks functionality, but blocks cannot be inserted into text notes as easily as symbols.

Using Center Marks and Centerlines
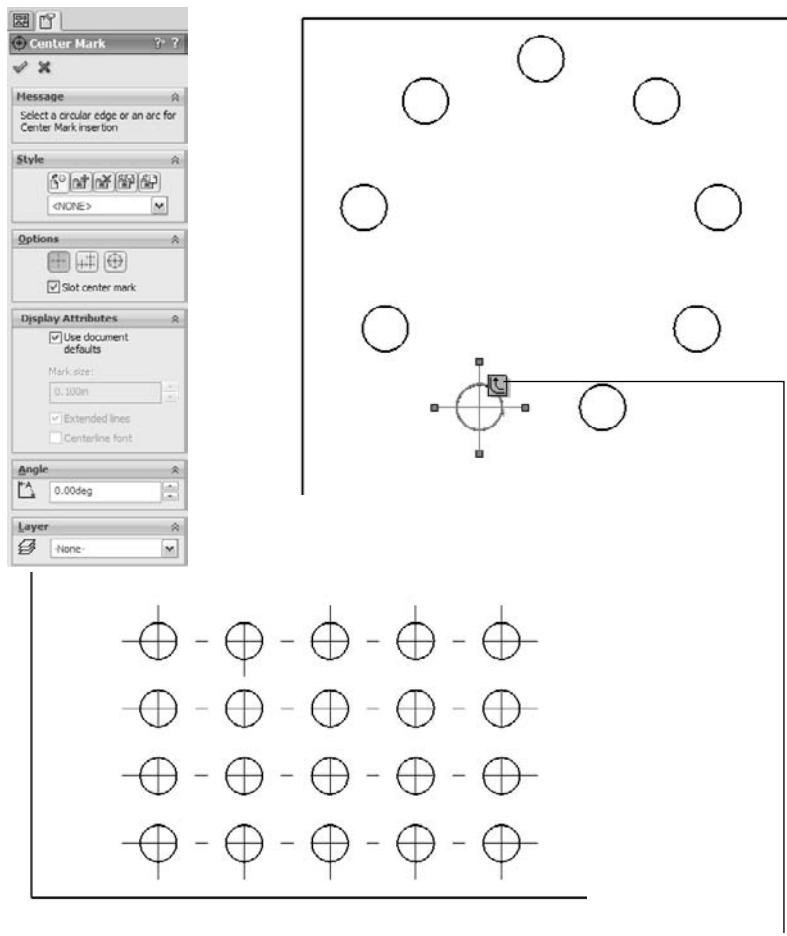
You can apply center marks either manually or automatically to edges that project as circular in the drawing view. To find the settings that control automatic insertion, choose Tools ➪ Options ➪ Document Properties ➪ Drafting Standard ➪ Centerlines/Center Marks. The size of the mark at the center and the use of lines extending to the actual circular edge are also controlled on this tab, in the Center Marks section. Figure 22.14 shows some of the options available for center marks.

Center marks propagate well to patterns, and you can dimension to them individually. You can rotate center marks in views where they need to be referenced from an edge that is not horizontal. You can also place center marks into layers.

FIGURE 22.14

Options available for center marks

Select symbol to propagate
center mark to pattern

You can apply centerlines to any geometry that has a temporary axis that is perpendicular to the view. Centerlines can also be placed automatically when you place the part into the drawing. You can create centerlines by selecting a face or a pair of parallel lines or concentric arcs. Centerlines may be displayed improperly on parts that are created by mirroring, as shown in Figure 22.15. This bug was originally printed here in the 2007 version and is still active for 2010.

FIGURE 22.15

Centerlines can display improperly on a mirrored part.

Tutorial: Using Annotations
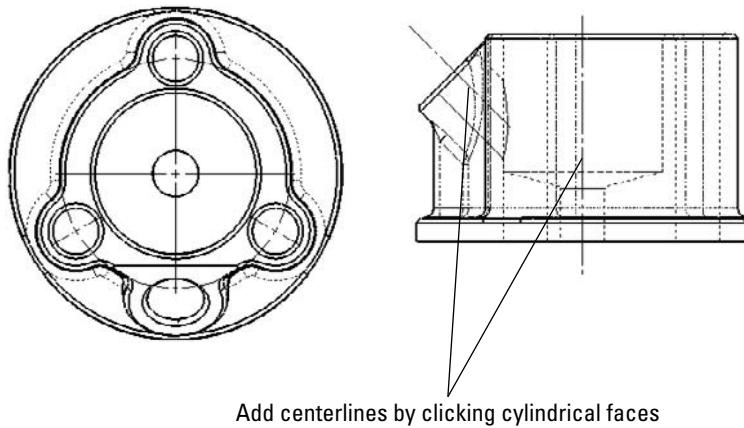
This tutorial shows you how to use some of the tools that were discussed in this chapter. It does not cover every feature, so you should explore a little on your own and not necessarily follow the instructions exactly. Start here:

1.   **From the CD-ROM, open the file named** `Chapter 22 – Tutorial.slddrw.` This is a drawing file with views of the part from Chapter 21, but it does not contain dimensions or annotations.

2.   **Click the Center Mark tool on the Annotations toolbar.** (If the button is not there, then choose Tools ⇨ Customize ⇨ Commands to place it on the toolbar, along with the Centerline tool.) Click one of the holes in the pattern of three, and click the Propagate symbol to propagate the center marks to all three holes in the pattern. The view should look like Figure 22.16 when you are done.

FIGURE 22.16
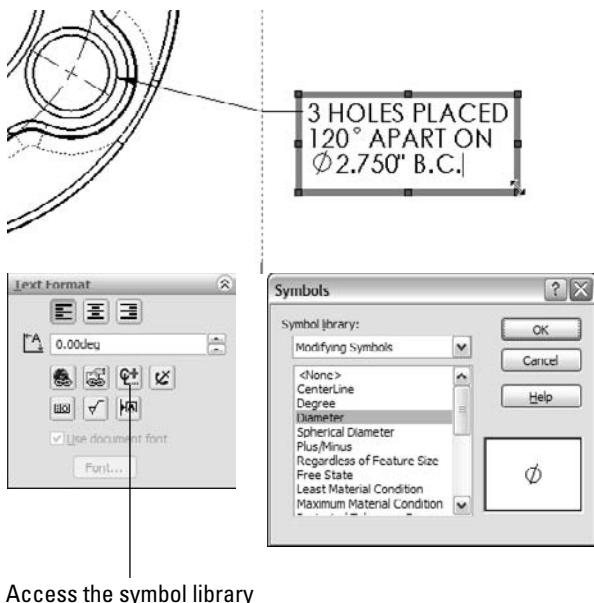
Center marks and centerlines on a part

Add centerlines by clicking cylindrical faces

3. **Activate the Centerline tool to add two centerlines to the right view, in the lower-left area.** Change the view display style to Hidden Lines Visible. Select the cylindrical faces for each feature to place the centerlines. Click the vertical centerline and drag the ends past the edges of the part.

4. **Select the edge that is indicated in Figure 22.17, and initiate a note from the Annotations toolbar.** Type the text shown, all in one line. You can place the degree and diameter symbols from the symbol library, which you can access using the indicated button in the PropertyManager. Both symbols are in the Modifying Symbols library, also shown in Figure 22.17. Drag the lower-right corner of the text box to make the text wrap as shown.

FIGURE 22.17

Placing symbols in an annotation

Access the symbol library
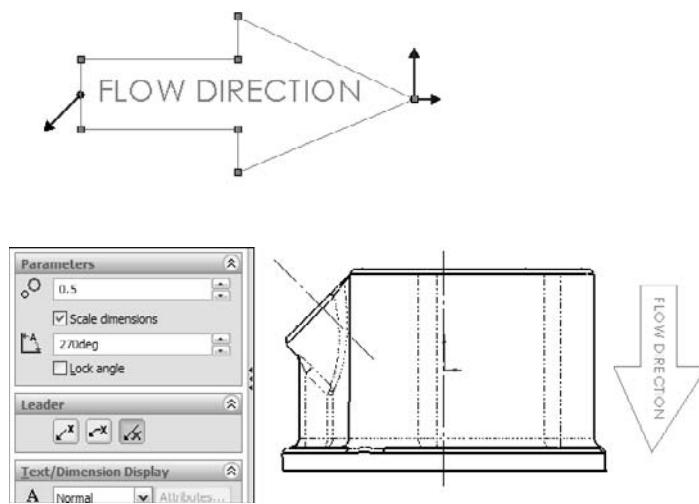
5. **Draw an arrow with a text note inside it, as shown in Figure 22.18.** Make the sketch and text into a block by window-selecting all of it and clicking Make Block from the Blocks toolbar, or by choosing Tools ➪ Block ➪ Make. Make sure that the end of the arrow is its insertion point. You have to expand the Insertion Point panel in the PropertyManager to access this option. When the block is set up, accept it by clicking the green check mark icon. When the block is created, delete it from the drawing.

Creating a block

6. **Place the block using the Insert Block function, so that the block is to the right of the right view.** Once you place it, press Esc to cancel the placement of more blocks. Then select the block to activate the PropertyManager. Deselect the Lock Angle option, and set the angle to 270 degrees.

Tutorial: Using Balloons on Assembly Drawings

Balloons support mainly assembly drawing type functions. This tutorial steps you through some of these to give you familiarity with the tools.
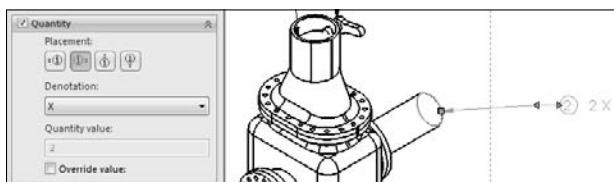
1. **Open the** `Ch 22` **folder on the CD-ROM, and copy the file named** `inch B.drwdot` **to your template folder.** If you don't know where your templates are, check for them by choosing Tools ⇨ Options ⇨ File Locations ⇨ Document Templates.

2. **Open the assembly named** `Chapter 22 – SF casting assembly.SLDASM`, **and click the Make Drawing From Part/Assembly toolbar button (under the New icon in the title bar toolbar).** As the template, use the inch B.DRWDOT drawing template that you copied into your template path in Step 1. This should automatically create a four-view drawing of the small assembly.

3. **Click inside the isometric view; then on the Annotation tab of the Commend Manager, click the Auto Balloon icon.** This places balloons on all the parts. Drag the cursor to place them a comfortable distance from the parts, but still inside the view. Accept the Auto Balloon by clicking the green check mark icon.

4. **Click on one of the balloons, and try to reconnect the arrow to a different location on a different part.** Notice the number in the balloon changes. If the balloon won't relocate to the face of a part, drag it to an edge. You can even drag the attachment point to another part in another view. Leave the balloon attached to one of the pipes coming out of the assembly in the isometric view. The balloon should have the number 2 in it.

5. **With the balloon used in the last step still selected, use the Balloon PropertyManager, shown in Figure 22.19.** Click the box next to Quantity, and notice "2 X" shows up next to the balloon. Try some of the Placement and Denotation options to see what you prefer.

Displaying a quantity next to a balloon
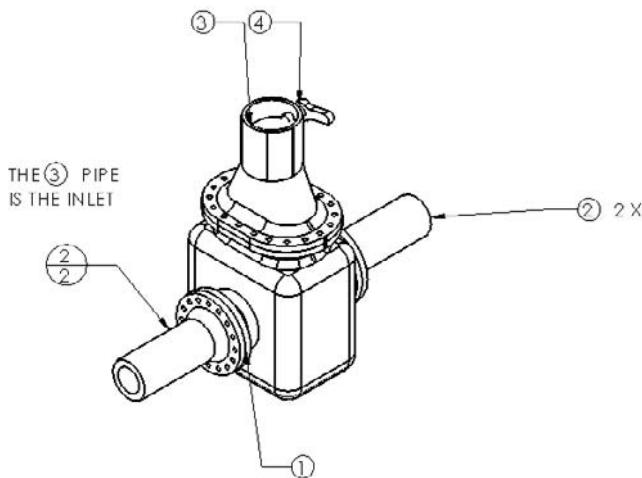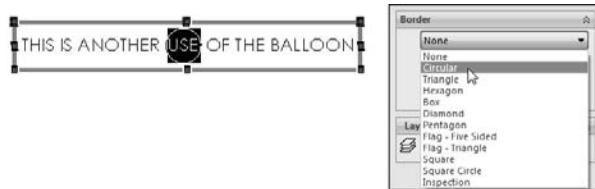
6. **Click the Note icon on the Annotation tab of the CommandManager.** Click the cursor in a white area of the drawing near the isometric view to place a note. Type **THE PIPE IS THE INLET**. Use the lower-right handle of the text box to resize the note so that it is displayed in two lines.

7. **Place the cursor between the words THE PIPE, and click the balloon with the 3 in it.** This places the balloon inside the text. See Figure 22.20.

9. **Type another note in empty space on the drawing that says, for example, "This is another use of the balloon."** With the text box still active, clear the highlighting and then re-highlight only the word "use." In the PropertyManager, expand the Border panel, and select Circular from the top drop-down list. This puts a balloon around the word "use."

FIGURE 22.20

Using balloons in more situations

Summary

Annotations and symbols in SolidWorks have many options for connection, creation, and display. Recent releases have brought major improvements to text box–driven annotations. Custom properties and hyperlinks enable the user to populate drawing annotations with content and links to content. Sharing styles in templates is a great idea for readily available note styles.

Blocks have several flexible uses and can be updated from external files across many documents. Their use for simulating mechanisms, piecing together schematics, and annotating drawings, in addition to the Belts and Chains functionality discussed in Chapter 13, make blocks one of the most flexible functions available.

Dimensioning and Tolerancing

Dimensioning and tolerancing is an art form as much as a science. People become very passionate when discussing the right way to perform these tasks. In truth, the techniques are not so black and white, but are highly dependent on the industry, the means of manufacture, and the purpose of the drawing. Drawings might be used for quotes, manufacturing, inspection, assembly, testing, and so on; and the drawings, as well as the dimensioning and tolerancing used, for each purpose need to be somewhat different.

While it is important to follow standards and use drawing conventions properly, this is not an argument that I want to reignite here. In this chapter, I will focus on how the available tools work. You will need to decide for yourself how to apply them in each situation.

Putting Dimensions on Drawings

The debate on how to get the dimensions from the model to the drawing is much like the "tastes great/less filling" debate. Each side of the issue has valid points, and the question is not likely to be resolved any time soon.

At the center of this debate is whether you should place the dimensions that you use to create the model directly on the drawing, or whether you should use reference dimensions created on the drawing. In the following sections, I examine each method for its benefits and drawbacks.

Using Insert Model Items
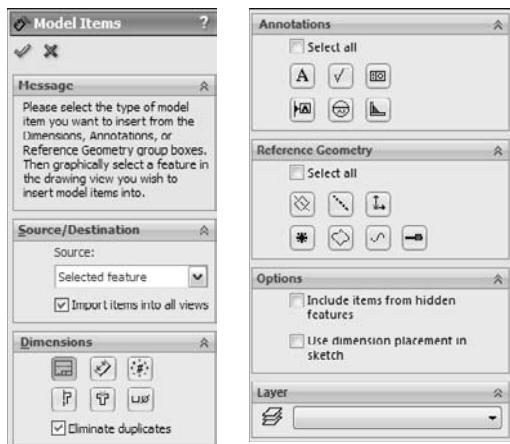
Insert Model Items takes all the dimensions, symbols, annotations, and other elements that are used to create the model and puts them onto the drawing. Because these dimensions come directly from the sketches and features of the model, they are *driving* dimensions. This means that you can double-click and change them from the drawing the same way you can change sketch and feature dimensions, and with the same effect. As a result, changing these dimensions even from the drawing causes the parts and assemblies in which they are used to be changed.

You can insert the model items on a per-feature basis, either only bringing the items that are appropriate into the current view, or bringing items into all views. Insertion can be further broken down by type of item, and it can become as specific as pattern counts, Hole Wizard items, specific symbol types, and reference geometry types. To use Insert Model Items, you can choose Insert ⇨ Model Items or can access this command from the Annotations toolbar. The Model Items PropertyManager interface is shown in Figure 23.1.

FIGURE 23.1

The Model Items PropertyManager interface
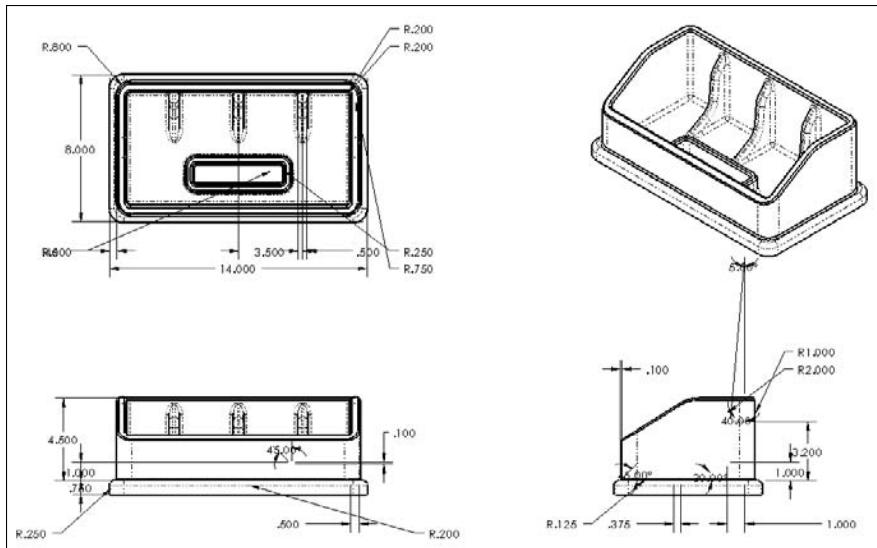
Usually, the dimensions need to be rearranged, although SolidWorks does try to arrange them so that they do not overlap. Figure 23.2 shows the result of bringing dimensions into all views for the part. The part is on the CD-ROM in the Chapter 21 materials.

Figure 23.2 contains duplicate dimensions, overlapping dimensions, unnecessarily long leaders, radius dimensions pointing to the wrong side of the arc, and a lot of awkward placement. This is what you can expect from using the automatic functions. At best, these dimensions require rearranging, and at worst, they probably require that you delete and replace some of them or move them to new views where they make more sense.

FIGURE 23.2

The default placement of dimensions into all views

To move a dimension to another view, you can Shift+drag it from one view to the other (make sure that the dimension is appropriate in the destination view). To copy a dimension, you can Ctrl+drag it. If you cannot place the dimension in the view that you have dragged it to, then the cursor will indicate this with a special cursor symbol.

If you approach this task by placing dimensions on a per-feature or per-view basis, it does not change the number of dimensions that you will have to move; it just means that you will insert fewer dimensions several times. Keep in mind that if you choose this method, there is a significant amount of cleanup and checking that you must do. The convenience of having the dimensions put into the views for you, and the ability to actually change the model from the drawing are quite useful, but you may not save very much time or effort by doing things this way.

Using reference dimensions

One alternative to automatically inserting all model dimensions is to manually place reference dimensions. You create reference dimensions by using the regular Smart Dimension tool. At first, this appears to be simply re-creating work that you have already done, and this is somewhat true, but there is more to the story.

These dimensions are not duplicates of the model items. In fact, the reference dimensions that you manually place on the drawing are completely different from the dimensions that are used in the model. The dimensions serve completely different purposes in the two settings, and could only be the same through some odd coincidence.

When modeling, I tend to dimension symmetrically, but only on one side, which would not be shown on a manufacturing or inspection drawing. I frequently use workarounds to avoid some special problem that forces a different model dimensioning scheme than I would prefer to use. Often, a feature is located from the midpoint of an edge, which involves no dimensions whatsoever. Sketch entities may have Equal relations, which also leave sketch elements undimensioned. Dimensions may lead to faces or edges that are not in the final model or to faces that are later changed by scale, draft, or fillets. Beyond that, when draft is involved, as is the case with plastic or cast parts, the dimensions of the sketch that you used to create the feature often have little to do with the geometry that is dimensioned on a print for inspection or mold building. Dimension schemes in models reflect the need for the model to react to change, while dimension schemes in drawings reflect the manufacturing or inspection methods, in order to minimize tolerance stack-up, and to reflect the usage of the actual part.

Although there are strictly technical reasons for dimensioning drawings independently from the way the model was dimensioned, there are other factors such as time, and the neat and orderly placement of dimensions. Time is an issue because by the time you finish rearranging dimensions that were inserted automatically from the model — checking and eliminating duplicates and then manually adding dimensions that were left out or that had to be eliminated because they were inappropriate for some reason, as well as ensuring that all the necessary dimensions are on the drawing — it would have been much quicker to manually dimension the drawing correctly the first time using reference dimensions.

In most cases, inserting model dimensions into the drawing is impractical for manufacturing or inspection drawings unless you have simple plates with machined holes. This is because of the amount of time required to rearrange and check the dimensions, the need to ensure that you have placed the necessary dimensions and taken geometric tolerancing into account, and the simple fact that the dimensioning and sketch relations needed for efficient modeling are usually very different from the dimensioning needed for manufacturing or inspection.

I recommend that you use the manual dimension placement option, which works much in the same way as when dimensions are added to sketches. Dimensions that you place in the drawing in this way are called *driven*, or *reference*, dimensions. In drafting lingo, reference dimensions are "extra" dimensions that you place to ease calculations, and you usually create these dimensions with parentheses around them; in SolidWorks lingo, reference dimensions are simply driven rather than driving dimensions. To find the setting that controls the parentheses around reference dimensions, choose Tools ➪ Options ➪ Document Properties ➪ Dimensions ➪ Add Parentheses By Default.
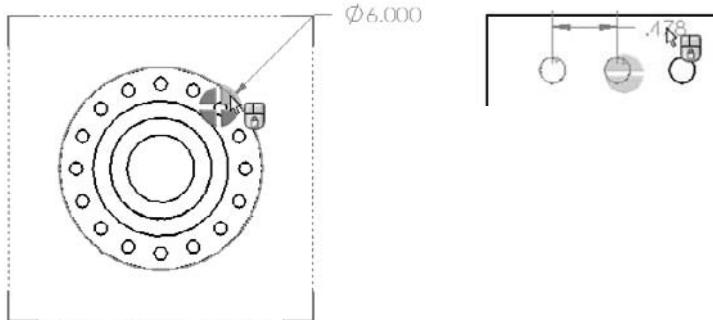
Rapid Dimension

A new feature in SolidWorks 2010 is Rapid Dimension, which offers a manipulator wheel that shows the possible locations of a dimension you are trying to place.

Rapid Dimension works for dimensions inside drawing views, not on dimensions on the sheet. It enables you to choose from either two or four options, and you can move between the options with the Tab key, making your selection with the Spacebar. You can also left click on the manipulator to select an option The Rapid Dimension manipulator wheel is shown in Figure 23.3 in the placement of diameter and linear dimensions.

FIGURE 23.3

Placing dimensions with the Rapid Dimension manipulator wheel

The tool does have some limitations, however. It does not seem to be capable of handling combined aligned dimensions and horizontal and vertical dimensions. For example, if you dimension an angled line, Rapid Dimension would only allow a dimension that is aligned to the angled line. If you dimension between the diagonal corners of a rectangle, it would not allow you to place the diagonal dimension, only the horizontal and vertical dims.

At this time, there does not seem to be any way to deselect the Rapid Dimension tool. For some situations, it appears that simply placing dimensions with the mouse is both more accurate and faster, in addition to being more flexible. There is probably an element of personal taste with this feature, but as of SolidWorks 2010 sp1.0, there is no way to get rid of it if you don't like it.

Reference dimensions and the DimXpert

You can apply reference dimensions to the 3D model or to the drawing. In this chapter, I talk mainly about adding them to the 2D drawing, but I do want to take a moment to talk about how reference dimensions in the model relate to the DimXpert functionality, which you will find later in this chapter.

Reference dimensions on the solid model
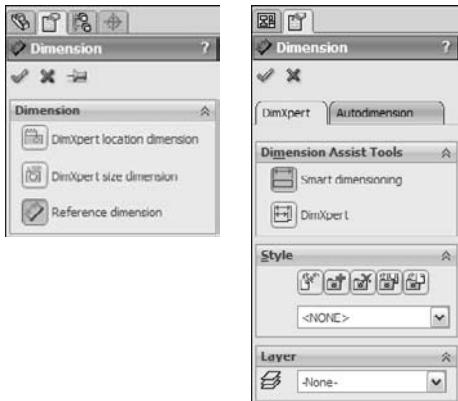
By default, when you go to add new reference dimensions to a solid model, you may see some error messages you aren't accustomed to seeing and some odd toolbars. If you are not expecting it, the DimXpert can interfere with reference dimension functionality.

When you activate the Smart Dimension tool, a PropertyManager appears, giving you the option to use dimensions to drive the DimXpert (the default) or use it to place reference dimensions. Figure 23.4 shows this Smart Dimension PropertyManager for parts on the left and for drawings on the right.

FIGURE 23.4

The Dimension PropertyManager for choosing DimXpert or Reference dimensions

Reference dimensions on the drawing

I've already made the case for why I think it is better to use reference dimensions on the drawing than model dimensions. This is opinion, of course, and I realize that for many simple parts, you actually *can* model them the way you would detail them, so the model items make more sense. Many respectable power users use model items on drawings. I mostly model \ complex plastic parts. Reference dimensions on the drawing are simply driven dimensions, and they update when the model updates.

Using the DimXpert

The DimXpert is a tool to apply reference (driven) dimensions with tolerances to models and drawings. DimXpert employs feature and topology recognition so it can work on either native data or imported data. Use the DimXpert tab at the top of the part Feature manager, and click the Autodimension Scheme button (the first button on the left) to apply dimensions to the entire model based on selected datums. The goal of DimXpert is to comply with ANSI Y14.41, which deals with 3D annotated models. The general consensus on this is that neither the standard nor the DimXpert tools are up to the rigors of daily production use at this time. The concept seems compelling, but the implementation is not yet practical.
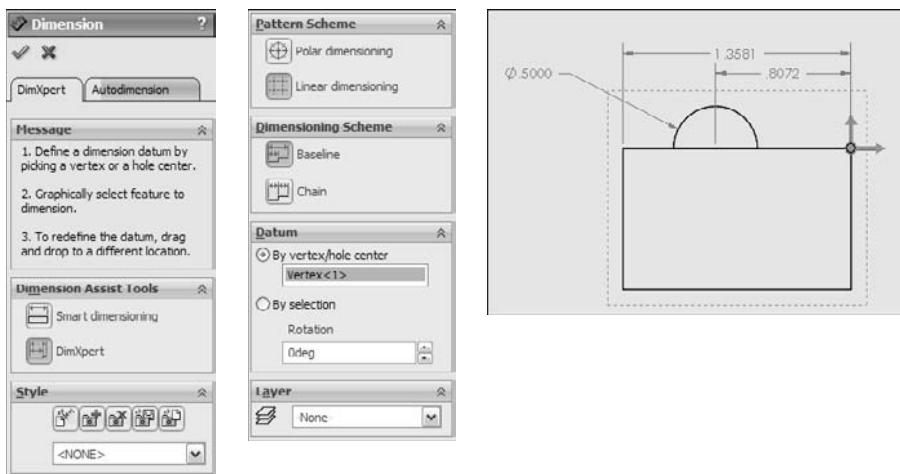
When you use the dimensions and tolerances created with the DimXpert in conjunction with the TolAnalyst, you are able to do simple stack-up analysis. TolAnalyst is outside the scope of this book, because it is part of the Premium package and I confined this book to SolidWorks Standard. A limitation of this system is that you can only apply location or size dimensions; you cannot apply non-dimensional geometric form tolerancing such as parallelism, cylindricity, or flatness. All controls must drive size or location, and have associated dimensions.

When you use the DimXpert on a drawing, it first places a datum at a vertex or centerpoint. After that, it automatically dimensions the entire feature in the view that is the parent of the edge you select. Figure 23.5 shows the Dimension PropertyManager when DimXpert is activated. The image

on the right shows a few dimensions applied by the DimXpert, along with the datum used for the dimensions in the view.

Dimension PropertyManager for DimXpert in drawing

You can choose to place the DimXpert dimensions on the drawing when placing the views either through the second page of the Model View PropertyManager, or on the Import Options panel (which is closed by default). No, it's not your imagination, this is about as obscure as SolidWorks could possibly make this functionality. Apparently they didn't really expect anyone to use it. Both pages of the Model View (Insert ⇨ Drawing View ⇨ Model, or click Model View from the View Layout tab on the Command Manager) are shown in Figure 23.6. The Import Options panel is shown at the bottom of the second page, although I have cut the second page off about half way down.

You can find this functionality in one other place: when you drag views from the View Palette in the Task Pane. This interface appears in the image on the right in Figure 23.6.

Consensus on this functionality is that it is a work in progress, and while it may offer some interesting functionality, you may not find that it is ready to save you a lot of time when you are dimensioning and tolerancing parts on a drawing. It seems that it has particular difficulty with molded or cast parts, which typically don't have parallel faces.
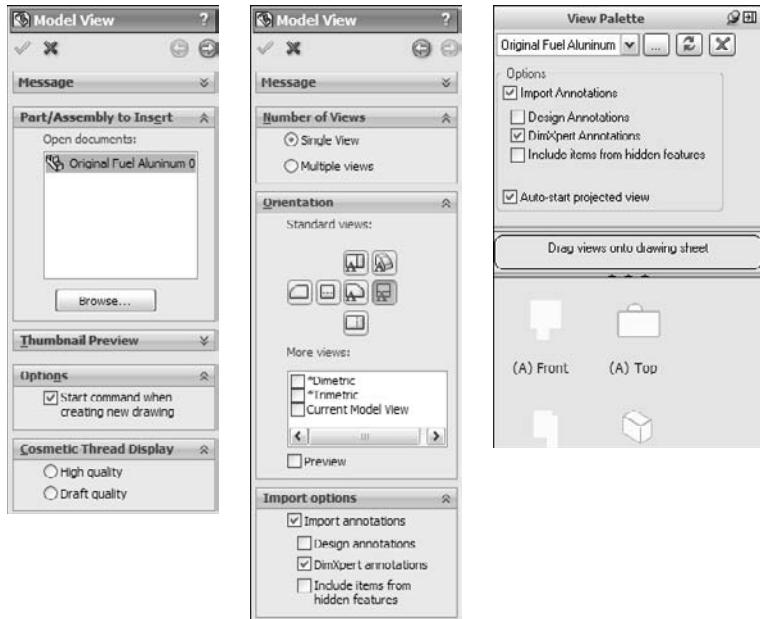
Annotation views

Annotation views are views in the model in which annotations have been added. You can access annotation views from the Annotations folder in the model FeatureManager. They are created automatically when you add dimensions or notes to the part. You can use the annotation view in the model to show the note or dimension in the view in which it was created or on the drawing to help parse the dimensions into views where they are easily read.

The setting to import DimXpert annotations is buried well.

Annotation views can be inserted manually or automatically. You can access the settings for annotation views through the right mouse button (RMB) menu of the Annotations folder of the model, shown in Figure 23.7. The image on the right shows part of the PropertyManager you get when inserting a named view on a drawing. It shows that the Front and Top views of the model have annotations associated with them (indicated by the A on the view symbol).
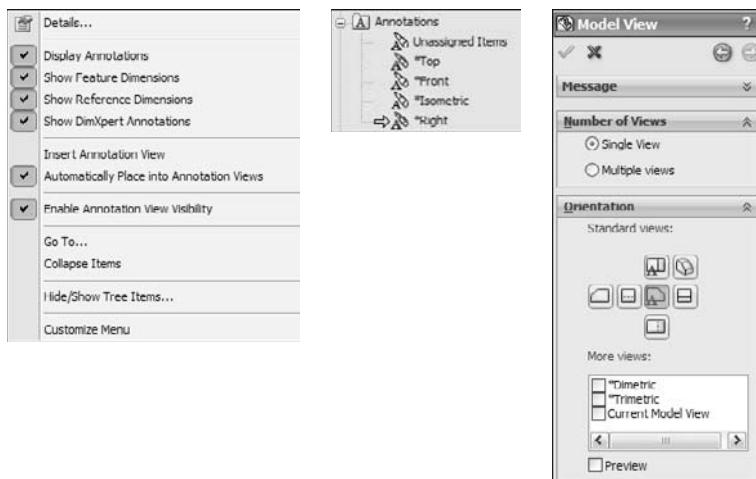
Driven dimension color

Driven dimensions on the drawing display in gray, and this can be a problem when the drawing is printed out. There are two methods that you can use to deal with this printing problem. The first method is to set the Page Properties of the drawing to force it to print in black and white. To find the Page Properties, choose File ⇨ Page Setup. The Page Setup dialog box is shown in Figure 23.8.
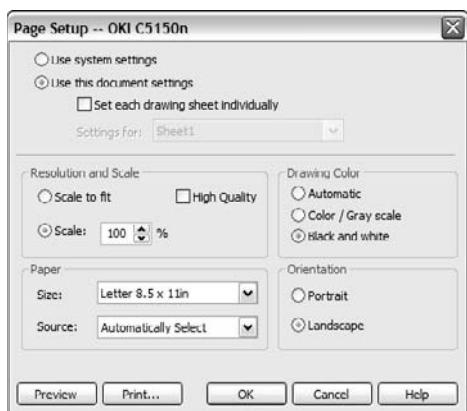
FIGURE 23.7

The Annotations folder RMB menu and the Model View orientation panel

FIGURE 23.8

The Page Setup dialog box

The second method is to set the color for driven dimensions to black rather than gray. You find this color setting by choosing Tools ➪ Options ➪ Color ➪ Dimensions Non-Imported (Driven).

Ordinate and baseline dimensions

Ordinate and baseline dimensions are appropriate for collections of linear dimensions when you have a number of items that can all be dimensioned from the same reference. Flat patterns of sheet metal parts often fall into this category. When you apply ordinate dimensions, a zero location is selected first, followed by each entity for which you want a dimension. When dimensions become too tightly packed, SolidWorks automatically jogs the witness lines to space out the dimensions adequately. You can create jogs manually by using the RMB menu. Once you create a set of ordinate dimensions, you can add to the set by selecting Add To Ordinate from the RMB menu.
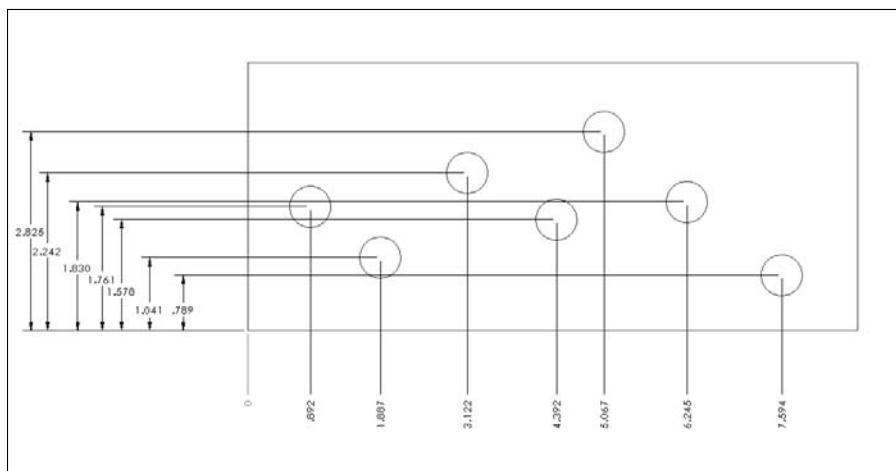
Baseline dimensions are normal linear dimensions that all come from the same reference and are stacked together at a defined spacing. To find the default settings for baseline dimensions, choose Tools ⇨ Options ⇨ Dimensions ⇨ Offset Distances.

Tip

Baseline dimensions work best either when they are horizontal or when the dimension text is aligned with the dimension line (as is the default situation with the International Organization for Standardization, or ISO, standard dimensioning). Vertical dimensions where the text is horizontal do not usually stack as neatly because the dimension text runs over the dimension line of the adjacent dimensions. Figure 23.9 shows ordinate and baseline dimensions in the same view. ■

FIGURE 23.9

Ordinate and baseline dimensions in the same view

You can access ordinate and baseline dimensions from the Dimensions/Relations toolbar or by right-clicking in a blank space, selecting More Dimensions, and then selecting the type of dimension that you want to use.
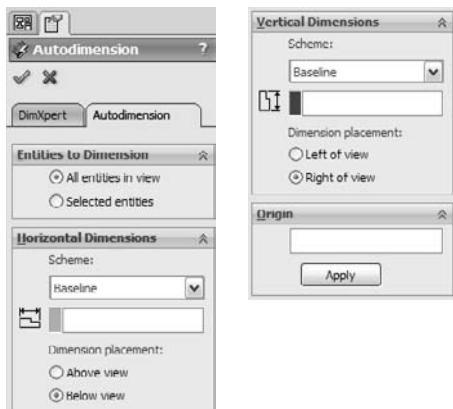
Autodimensioning

If the Insert Model Items feature is not likely to produce dimensions that are usable in a manufacturing drawing, then the Autodimension feature is even less likely to do so. However, if you use autodimensioning in a controlled way, in the right situations, it can be a valid way to create selected dimensions. The Autodimension PropertyManager is shown in Figure 23.10. Autodimension is only available in the drawing environment. In the part environment, similar functionality is part of the Fully Define Sketch tool. To access Autodimension, click the Smart Dimension toolbar icon and click the Autodimension tab in the PropertyManager.

FIGURE 23.10

The Autodimension PropertyManager interface

The Autodimension function can fully dimension the geometry in a drawing view. This is best for ordinate or baseline dimensioning where many dimensions are derived from a common reference, as is often the case with sheet metal parts or a plate with many holes drilled in it. You should limit the use of this option to cases where that type of dimensioning is what you would choose, having the choice of all available types of dimensions — do not allow the software to dictate the dimensioning scheme for your drawing.

Reference sketches

For some types of dimensions, you may need to create additional reference sketch entities. For example, with angle dimensions, it may be desirable to add construction lines to help define the angle. You can add centerlines as separate axis-like entities, as discussed in Chapter 22, but you can also sketch in centerlines manually if needed. This type of sketch is most often attached to the view rather than the drawing sheet.

Tip

Remember that, if necessary, you can create angle dimensions by selecting three points (vertex of the angle first) instead of two lines. When you do this, sketch lines are typically drawn to indicate the vertex of the angle. ■

Understanding Dimension Options

The Dimension PropertyManager contains settings, default overrides, tolerances, styles, and several other important settings for use with dimensions. The PropertyManager for driven dimensions is shown in Figure 23.11. I cover styles and tolerances specifically later in this chapter; the other panels of the Dimension PropertyManager are as follows.

FIGURE 23.11

The Dimension PropertyManager interface

Dimension Text

The Dimension Text panel enables you to add text to the dimension. You can add lines of text both above and below the dimension value itself, and you can also add text before and after the DIM value on the same line. The DIM field is what places the actual value; if this syntax is somehow deleted, you can type it back in and the dimension will still work.
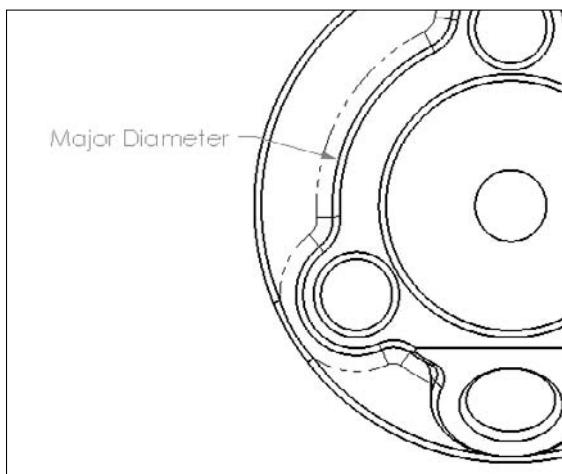
The Dimension Text panel includes some formatting tools, such as justification settings and a setting for the position of the dimension line. The last two rows of buttons include the more commonly used symbols, with access to the complete library, such as any custom symbols that you may have made for the library.

Primary Value Override

The most famous bad habit former AutoCAD users use is overriding dimension values. Apparently due to popular demand, the Primary Value Override is now available in SolidWorks, in the Dimension PropertyManager, as shown in Figure 23.11. This option was added to the software mainly to enable the creation of dimensions with words instead of numbers, as shown in Figure 23.12.

FIGURE 23.12

Using the Override Dimension value

Display Options

You can control the default setting for parentheses around driven (reference) dimensions in the Add Parenthesis By Default dialog box by selecting the choosing Tools⇨Options⇨Document Properties⇨Dimensions⇨Add Parentheses By Default.

Although you can also control dual dimension defaults in the Options dialog box by choosing Tools⇨Options, you can turn them on and off from this interface for individual dimensions. When you enable the Dual Dimension option, SolidWorks uses the settings from the Tools⇨Options menu.
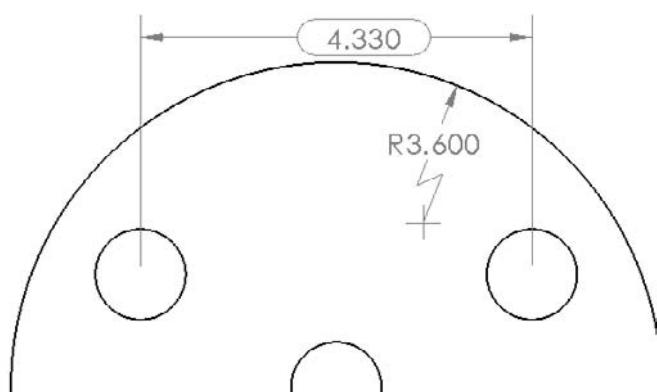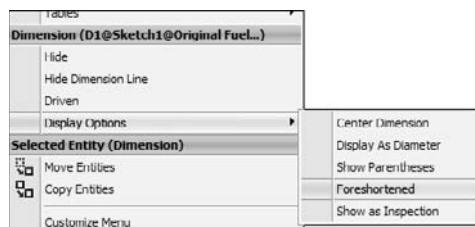
Note

The Display Options now appear on the RMB menu. The options shown in Figure 23.11 are different depending on what type of dimension you have selected. In the images in this chapter, a diameter dimension was used. ∎

The foreshortened radius is only valid for individual radial dimensions. A foreshortened radius is shown in Figure 23.13. Foreshortened radius dimensions are typically used for large radii when dimensions to the centerpoint are not important. The Foreshortened radius function does not work on diameter dimensions. The inspection dimension is shown in Figure 23.13 with an oval around the dimension.

FIGURE 23.13

A foreshortened radius

Witness/Leader Display
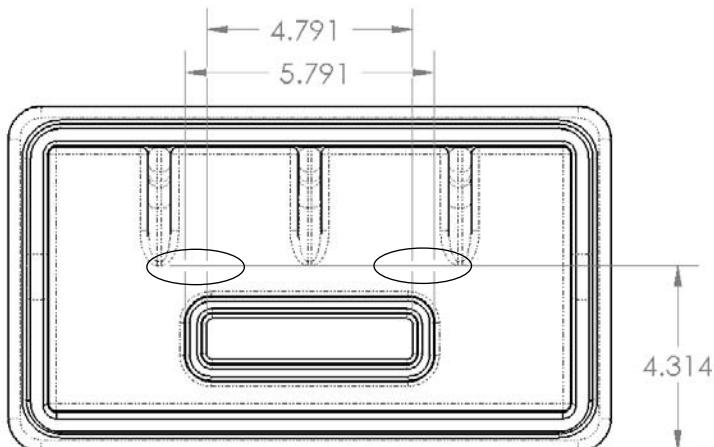
This panel enables you to set the arrows and dimension lines to be placed inside the witness lines. You can perform this function more easily by using the handles on the arrowheads. From this panel, you can also change the display type of individual arrowheads.

Break Lines

When you select the Use document gap option in this panel, the witness, or extension, lines of the selected dimension are broken by other crossing dimension lines, witness lines, or arrows. This is shown in Figure 23.14.

FIGURE 23.14

Broken witness lines

Cross-Reference

Layers are discussed in detail in Chapter 25. ■

Formatting dimensions with the Dimension Palette

The Dimension Palette is a box that appears when you select a dimension on a drawing. The dimension can be a model item or a reference dimension. Dimension Palette enables you to:

- Add tolerances
- Specify dimension precision
- Assign styles (favorites)
- Apply parentheses
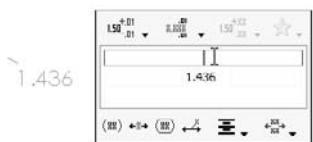
- Center the dimension between the witness lines
- Apply inspection dimension
- Offset the text with a leader
- Establish horizontal and vertical justification
- Add text on top, before, after and below the dimension value

The Dimension Palette appears when you select a dimension, and disappears when you move the cursor away from it. The Dimension PropertyManager still appears, but the Dimension Palette pops up right next to the dimension, making it very easy to use. Figure 23.15 shows the Dimension Palette.

Adding text and tolerances to dimensions using the Dimension Palette

The Dimension Palette seems to be the most convenient place to make these alterations to the basic dimension itself. If you don't like the Dimension Palette, there is currently no way to deselect it.

Adding Tolerances

You can add dimension tolerances in the Dimension PropertyManager, which you can activate by selecting the dimension that you want to modify. Available tolerance types include:

- Basic
- Bilateral
- Limit
- Symmetric
- MIN
- MAX
- Fit
- Fit with tolerance
- Fit (tolerance only)
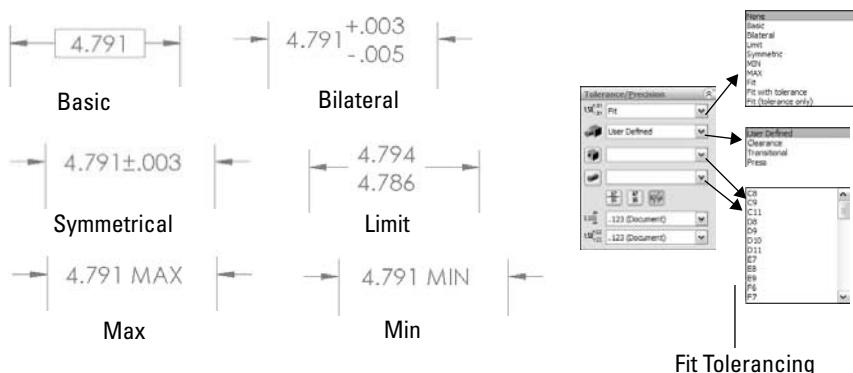
Note

You can also add tolerances to dimensions in models; the tolerance is brought in with the dimension if you use the Insert Model Items feature. ■

Refer to the Tolerance/Precision panel shown in Figure 23.11. The appropriate number entry fields are activated when you assign the corresponding tolerance type to the dimension. The tolerance types that are available in SolidWorks are shown in Figure 23.16.

FIGURE 23.16

The available tolerance types in SolidWorks

Fit Tolerancing

Changing precision values

In SolidWorks, *precision* means the number of decimal places with which dimensions are displayed. Typically, SolidWorks works to eight places with meters as the default units. You can create templates that use up to eight places as the default setting, and then change the number of places for individual dimensions as necessary. The first of the two boxes under Precision is used for the dimension precision, and the second is used for tolerance precision.

You can change Precision values for individual dimensions in the PropertyManager for the dimension as well as the entire document by choosing Tools ➪ Options ➪ Document Properties ➪ Units.

Using Geometric Tolerancing symbols
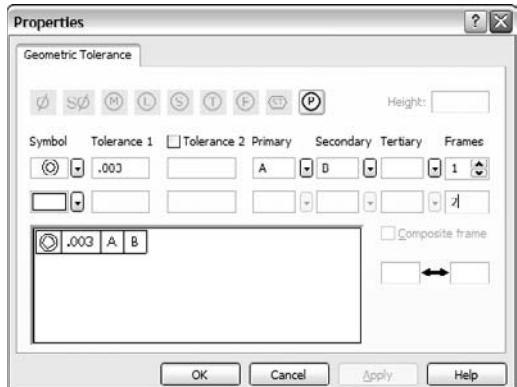
The full range of Geometric Tolerancing symbols is available for control frames, datums, datum targets, and so on. You can use the Geometric Tolerance dialog box to build control frames. This dialog box is shown in Figure 23.17. For commonly used Geometric Tolerance symbols, you may want to create and use styles.

FIGURE 23.17

The Geometric Tolerance settings

Using Dimension Styles

You can use dimension styles to apply many items to dimensions. Styles were formerly known as *favorites* in SolidWorks. Unlike notes, this is not limited to fonts and formatting. Some of the most common uses of dimension styles are

- To add standard tolerances to dimensions

- To set precision values for dimensions

- To add text, such as TYP, to a dimension

- To add a commonly used GD&T (geometric dimensioning and tolerancing) reference

You can save styles from one document and load them into another document, even between document types. For example, you can load part dimension styles into a drawing.

When a style is updated from an external file, any document that it is linked to also updates. In addition, you can break links to external styles (with the appropriate button on the Styles panel). Otherwise, dimension styles have very similar functions to the other types of styles; the functions of all the buttons on the Styles panel are the same.

Tutorial: Working with Dimensions and Tolerances
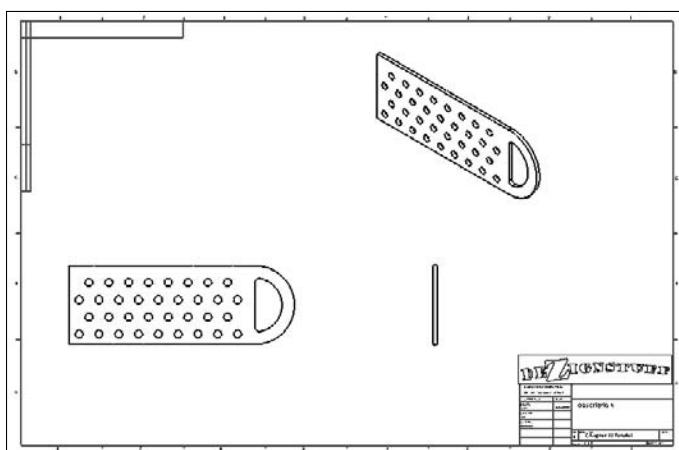
In this tutorial, you can use a single part in several different ways to demonstrate different dimensioning and tolerance functions. Follow these steps to learn more about these topics:

1. **Open the part from the CD-ROM called** `Chapter 23 Tutorial.sldprt`.

2. **Open the drawing from the CD-ROM called** `Chapter 23 Drawing.slddrw`.

3. **Tile the windows by choosing Window ⇨ Tile Vertically, and drag the part from the top level of the FeatureManager into the drawing window.** This automatically populates the four drawing views.

4. **Delete the Top view, leaving the views as shown in Figure 23.18.**

FIGURE 23.18

The drawing after Step 4
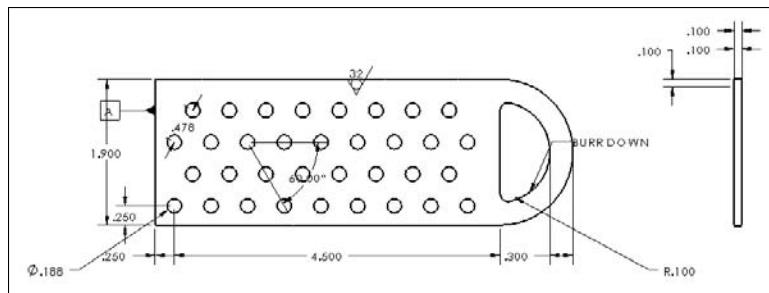
5. **Choose Insert ⇨ Model Items, and ensure that the Select All option is selected for Annotations and the Marked for drawing option is selected for Dimensions.** Also make sure that the Source/Destination drop-down list is set to Entire Model. Click the green check mark icon and watch the drawing populate.

6. **The resultant drawing is quite cluttered.** Delete and move dimensions so that the drawing looks like Figure 23.19.

7. **Shift+drag the surface finish symbol to the Right view, and do the same with the 1.900-inch dimension.** You may have to first Shift+drag it into the other view, and then drag it again to correctly attach or position it.

8. **Create a set of horizontal ordinate dimensions from the left end of the part, and dimension the X position of each column of holes.** Do the same for rows of holes, using the bottom edge of the Front view as the zero reference. Remember that you can create ordinate dimensions by starting a normal Smart Dimension, right-clicking to display the More Dimensions list, and then selecting your choice.

9. **If necessary, add center marks and centerlines to the view for clarity.**
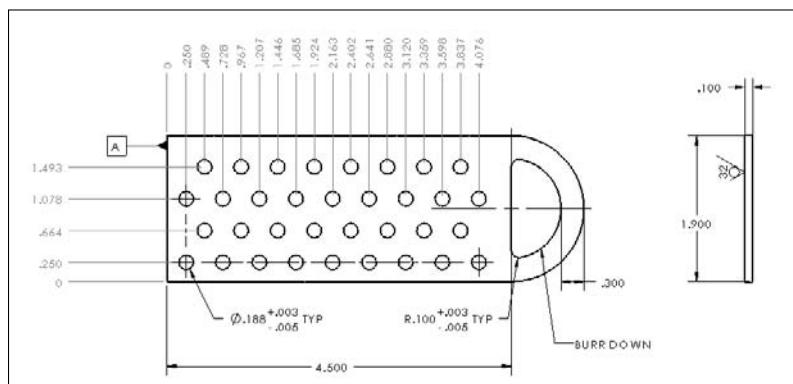
The drawing after dimensions have been deleted and moved

10. **Select the.188 diameter dimension, and in the Dimension Text box, type** TYP **after the <DIM> text, and add a bilateral tolerance of +.003, −.005.** Save this as a style by clicking the Add Style icon.

11. **Apply the newly created dimension style to the R.100 dimension.** The results up to this step are shown in Figure 23.20.
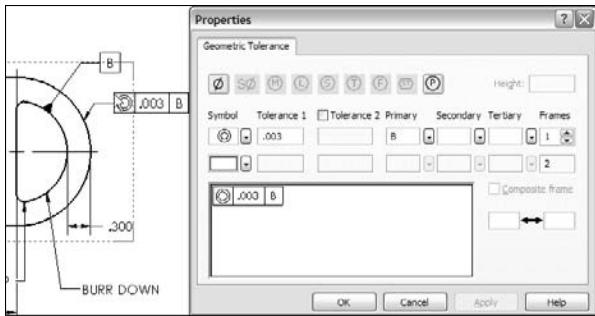
Dimensions and tolerances after Step 11

12. **Make one of the dimension leaders for either the.188 or the R.100 dimensions cross the extension lines of the 4.500 dimension.** Then select the 4.500 dimension and in its PropertyManager, select the Use Document Gap option in the Break Lines panel, on the Leaders tab.

13. **Place a B datum marker on the circumference of the smaller arc on the left end of the part.** Create a Geometric Tolerance control frame, as shown in Figure 23.21.

FIGURE 23.21
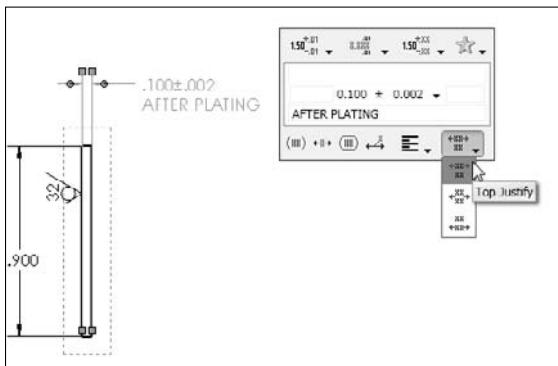
Creating a Geometric Tolerance control frame

14. **In the Right view that shows the thickness, select the.100 dimension, and add a tolerance and a note below the text using the Dimension Palette, as shown in Figure 23.22.**

FIGURE 23.22

Using the Dimension Palette
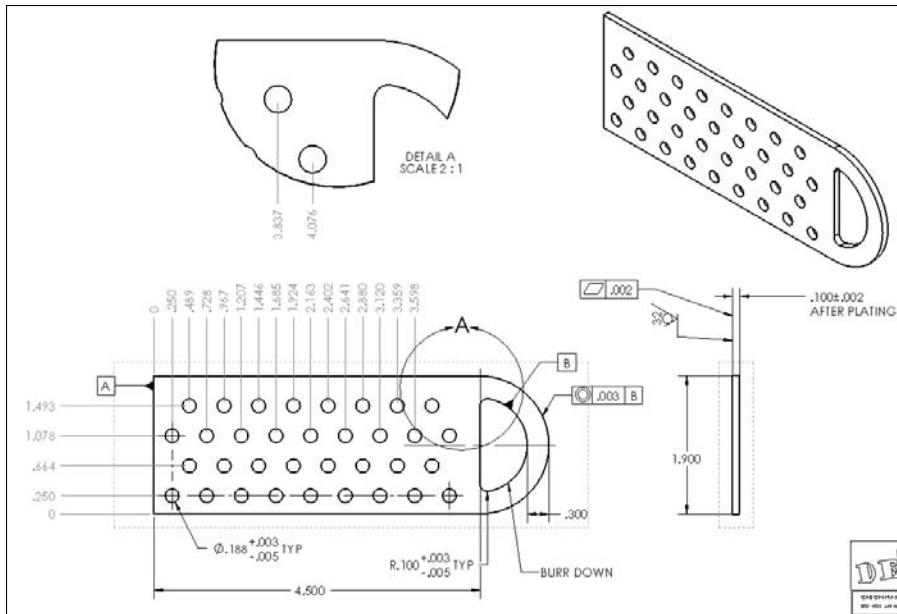
15. **Delete the last two dimensions of the horizontal ordinate group (3.837 and 4.076).** Start a new Detail view that includes the holes those dimensions called out, as shown in Figure 23.23. Place the detail view above the main view.

16. **Right-click the zero for the horizontal ordinates and select Add To Ordinate.** Place the two ordinate dimensions in the detail view.

17. **Use the Geometric Tolerance tool to make a Flatness call out of.002.** Add a leader to the control using the PropertyManager, and attach the leader to the .100 thickness dimension in the Right view.

Adding more dimensions and annotations

Summary

The argument about how to set up and use dimensions on drawings is as old as the process of creating geometrical plans from which objects are built. It is often difficult to separate fact and best practice from opinion. Although I leave it up to you to decide these issues for yourself, this chapter is intended to help you understand how to create the type of drawing that you want.

The biggest conflict in this subject arises over whether to place live model dimensions on the drawing or to allow the requirements of the drawing to specify which dimensions are placed where. I am by no means impartial when it comes to this question, but again, you must make the choice for yourself.

Working with Tables and Drawings

SolidWorks enables you to create several types of tables on drawings, such as the Bill of Materials, or BOM. Design Tables that are used in parts and assemblies can also be shown on the drawing to create a tabulated type drawing. Hole Tables enable you to chart the center locations and sizes of holes for easy access to manufacturing data. Revision Tables can work with Workgroup PDM (Product Data Management) or by themselves to help you document the revision history of a drawing. General Tables are also available for any specialized items that are not covered by the other table types.

Driving the Bill of Materials

The Bill of Materials (BOM) is one of the most frequently used types of tables that are available in SolidWorks. BOMs are intended for use with assemblies, but can also be used with individual parts for specialized applications. The information that you can expect to see on a BOM includes item number, file-name, quantity used, description, and any other custom property that you would like to add to it. A typical BOM is shown in Figure 24.1.

A sample BOM

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Saddle	Vice Base Casting	1
2	lower plate	Tie Plate	2
3	upper plate	Upper Tie Plate	2
4	cap screw	1/4x.80" Round Head Screw	5
5	tool holder	Vice Tool Holder	1
6	handle shaft	Locking Handle Shaft	4
7	locking handle	Locking Handle	4
8	compound center member	Multipivot Member	1
9	upper compound member	Upper Half Pivot	1
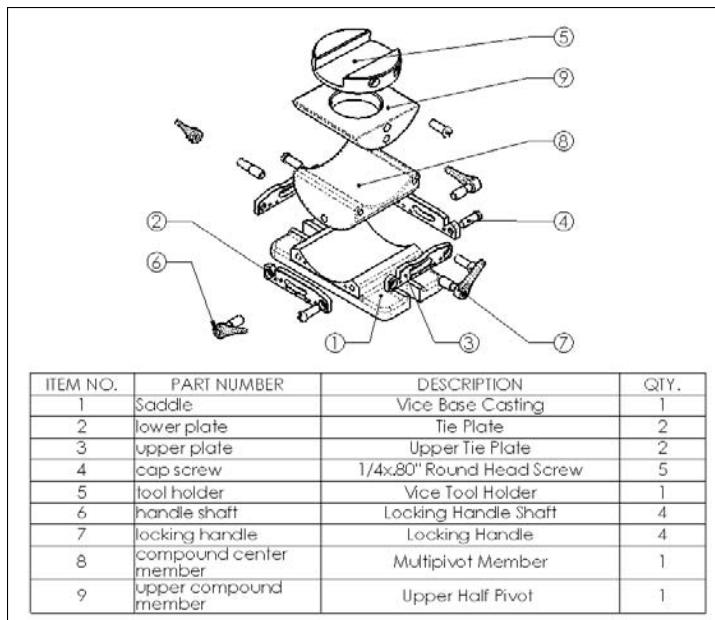
BOMs are made in one of two ways. The default BOM is made from a special SolidWorks table, while an Excel-based BOM is driven by Excel. While Excel has advantages and disadvantages, many users appear to prefer the default BOM. Excel and SolidWorks table-based BOMs are not interchangeable, so if you plan to customize the default templates, you need to make a decision in which type of BOM you will want to develop the BOMs.
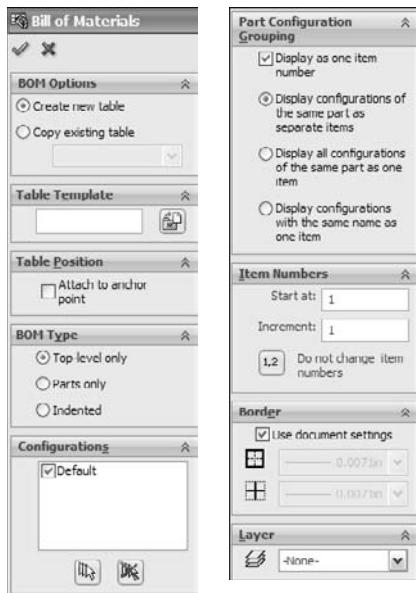
Note
BOMs can also be placed directly in the assembly and even in multi-body part files. ∎

Examining the SolidWorks table-based BOM

The BOM shown in Figure 24.1 is a default SolidWorks table-based BOM. The differences between the displays of the two types of BOM are mainly cosmetic; the bigger difference is in the functionality. The PropertyManager interface for the SolidWorks Bill of Materials is shown in Figure 24.2.

FIGURE 24.2

The PropertyManager for a table-driven BOM

Table-based BOM templates

Like other types of data, the SolidWorks table-driven BOM starts from a template. The BOM in Figure 24.1 was created from the default BOM template. When a BOM is initiated, you can select the template in the Table Template panel near the top of the PropertyManager, as shown in Figure 24.2.

Table-based BOM templates are created in much the same way that other templates are created:

1. Specify the settings
2. Delete the document-specific data
3. Save the template
4. Access the template from a library location

To save the template, right-click the BOM and select Save As. In the Files of Type drop-down list, select Template (*.sldbomtbt, which stands for SolidWorks Bill of Materials Table Template). Any of the settings, additional columns, links to properties, and so on are saved to the template and reused when you create a new template from it.

Best Practice

Put the BOM template in your library area outside of the SolidWorks installation folder. Then put the path in the Tools ⇨ Options ⇨ File Locations area. ∎

Table anchor

A table anchor locks a corner of the table to a selected point on the drawing sheet format. If no point is selected in the format, then the table is placed at a corner of the sheet. To specify a point in the format to act as the anchor, you must be editing the format. Right- click the sheet and select Edit Sheet Format). Then right-click a sketch endpoint in the format, select Set As Anchor, and specify which type of table the anchor is for. You can set different anchor locations for different types of tables. Figure 24.3 shows the selection and menus for this option.

FIGURE 24.3

Setting a BOM table anchor

Tip

You should save the format and drawing template with these table anchors specified so that you do not need to re-specify them for each new document. If you want to check a Sheet Format to see what anchors exist, you can expand the Sheet in the FeatureManager of the drawing. ■

BOM type

You can use one of three BOM types in SolidWorks: Top-level only, Parts only, or indented.

As the name suggests, the Top-level only BOM only shows components on the top level. It treats subassemblies as a single entry. As a result, if the top-level assembly shown on the drawing is made up of five subassemblies and two individual parts, and you select the Top-level only option, then only seven items are shown in the BOM.

The Parts only BOM ignores subassembly structure and only displays parts in an un-indented list.

The Indented BOM shows the parts of subassemblies in an indented list under the name of the subassembly. This is the most complete list of SolidWorks documents used because it includes all parts and assemblies.
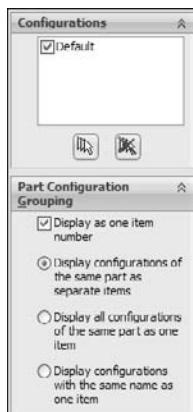
The Show Numbering option for indented assemblies is only activated after the Indented option is selected and you have placed the table. When you use this option, it causes subassembly parts to be numbered with an X.Y number system. For example, if item number 4 is a subassembly and it has three parts, those parts are numbered 4.1, 4.2, and 4.3.

Configurations

The Configurations panel of the BOM PropertyManager displays slightly differently for Top-level only BOMs compared to the other types. The Top-level only BOM type enables the option to show multiple assembly configs and display the quantities for top-level components in separate columns, as shown in Figure 24.4. This figure shows that the configuration named "D" has some suppressed parts, including some parts that are now not used in the "D" configuration, and that therefore have a zero quantity. Notice the available options for dealing with zero-quantity parts.

FIGURE 24.4

Configuration options with the BOM

ITEM NO.	PART NUMBER	DESCRIPTION	Default/ QTY.	d/QTY.
1	Saddle	Vice Base Casting	1	1
2	lower plate	Tie Plate	2	2
3	upper plate	Upper Tie Plate	2	2
4	cap screw	1/4x.80" Round Head Screw	4	4
5	Assem1	inch assembly description	1	1
6	handle shaft	Locking Handle Shaft	4	2
7	locking handle	Locking Handle	4	2
8	compound center member	Multipivot Member	1	-

Keep Missing Items

When you are making changes to a model, parts are often either suppressed or deleted altogether. Some company documentation standards require that parts that are removed from a BOM remain on the bill and appear with strikethrough formatting, although this may be a relic from days gone by when it was more difficult to remove items from hand drawings.

Keep Missing Items and Zero Quantity Display have both been moved to Tools⇨Options⇨ Document Properties⇨Detailing⇨Tables screen.

Zero Quantity Display

The Zero Quantity Display settings are only used for configurations where some components are not used in some configurations. The three options that are available are

- **Quantity Of Dash.** Substitutes a dash for the quantity value.
- **Quantity Of Zero.** Uses a zero for the quantity value.
- **Blank.** Quantity value is blank.

Item numbers

Item numbers for components listed in the BOM can start at a specific number and be given a particular interval. The Do not change item numbers option means that even when rows are reordered, item numbers stay with their original components.

The Follow Assembly Order option, which is also available through the right mouse button (RMB) menu, means that the order of the components in the BOM follows the order of the components in the Assembly FeatureManager. If the order is changed in the assembly, it also updates in the drawing.

BOM Contents
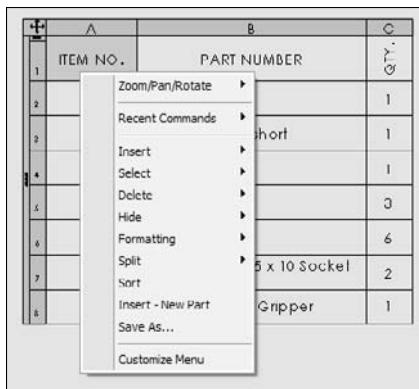
The BOM Contents are handled by functionality available right on the BOM or through the RMB menu. Figure 24.5 shows a simple BOM with the RMB menu. For example, you can drag the row numbers to reorder BOM items, and right-click to hide them. Row numbers are only displayed after you select the BOM table.

You can add columns or rows to the BOM for additional properties or manually added parts (such as items you wouldn't model, like paint or glue). To change the property displayed in a column, double-click in the column header. In previous releases, many of the settings and options now found on the RMB menu were found in a more complex Bill of Materials Properties window. The newer arrangement is more intuitive. Most SolidWorks users know to try the RMB menu if they select something and don't get the option they are looking for in the PropertyManager.

FIGURE 24.5

The BOM Contents interface
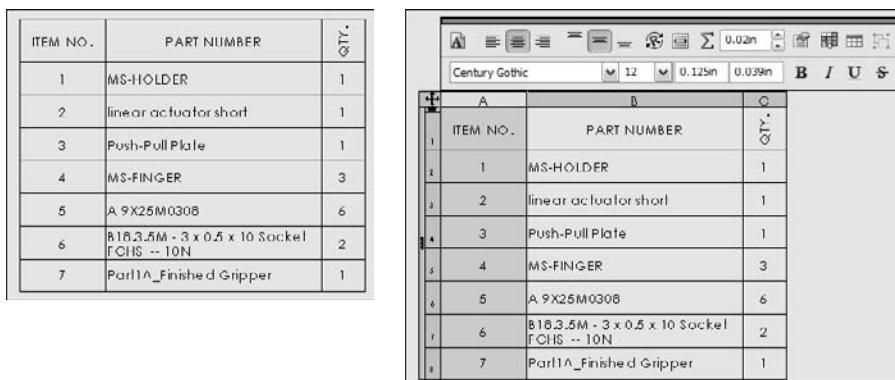
Controlling the appearance of the table-based BOM

If you are already familiar with formatting an Excel-based BOM, then you will quickly get used to formatting the SolidWorks table-based BOM. Figure 24.6 shows the table unselected on the left and selected on the right. While it's selected, you have access to a full range of appearance and organization options through the RMB menu.

FIGURE 24.6

Selecting a column, row, cell, and table

To move the table, click outside the upper-left corner and drag the table to a new location. When the table is selected, a border appears around it that is not normally visible, and is not printed with the drawing. You can change the properties of a row or a column by selecting just outside of the row or column to the top or the left.

You can establish spacing and width of rows or columns by dragging the border on the left side of the column with the split cursor or by accessing the column width setting through Formatting in the RMB menu.

While it is activated, you can also expand a panel to the left and another to the top by clicking the three small arrows in the selected BOM border. Figure 24.7 shows the left panel, called the Assembly Structure panel, expanded with the cursor pointing to the three small arrows. In addition to showing the assembly structure, this panel can also show which parts are ballooned on the drawing.

FIGURE 24.7

An expanded BOM border

Notice item 7 on the BOM. It has a different symbol and no name. This is a virtual component: a component created in context but not saved to its own document file; it exists only within the assembly.

Dissolving, combining, numbering, and restructuring for indented BOMS

You can dissolve an assembly in the BOM. To do this, the BOM has to show an indented list, then access Dissolve from the RMB menu on the assembly icon. Any restructuring done to the BOM can be deleted by RMB+clicking the assembly icon with red arrows and selecting Restore restructured components.

If the BOM shows several parts that are identical, and you would like to combine them, again, access the option from the RMB menu.
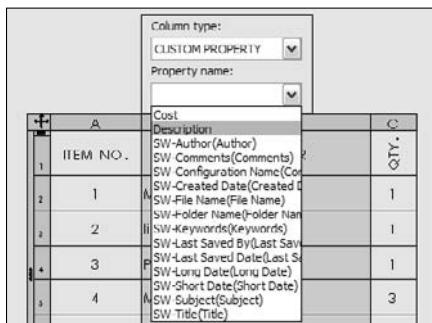
Item numbers in indented BOMs can be flat like 1, 2, 3, or they can be detailed like 1.1, 1.2, 1.3, 2, 2.1, 2.2 to reflect parts as members of subassemblies. This formatting option is available in the BOM Type panel in the BOM PropertyManager, under the Indented option.

Adding rows or columns

To add a column, right-click near where you want to add the column, and choose Insert ⇨ Column Right or Column Left. Inserting rows is exactly the same. The next thing you will want to do with a column is to assign what kind of data goes into it. You can use a custom property such as Part Weight or Vendor, as shown in Figure 24.8. Access this interface by double-clicking a column header and selecting Custom Property from the drop-down list.

Establishing the property driving the column content

One of the really beautiful aspects of custom property management in the BOM is that if you just type text in a column set up to be driven by a part property, SolidWorks automatically updates the part with the property. If the property didn't exist in the part previously, SolidWorks also creates the property. This is another very nice addition to the software.

Note

If you create a BOM with the columns and properties that you like, then you can save it to a template as described earlier in this chapter. ■

Editing BOMs

When you need to manually enter text in a BOM, for example in General Tables or custom properties in BOMs, starting in SolidWorks 2010, you can use the tab and arrow keys to move the cursor between cells. In fact, most of the Excel-like functionality has been added back into the SolidWorks native table format.

A summary of the added Navigation functionality is as follows:

- **Enter.** Move to next cell down
- **Tab.** Move to next cell to the right
- **Arrow.** Move cursor in any direction
- **Shift+Arrow.** Increases selection size
- **Shift+Tab.** Move cursor backward
- **Shift+Enter.** Move cursor up
- **Home.** Move cursor to first column
- **Ctrl+Home.** Move to upper left of table
- **End.** In combination with arrow keys goes to end indicated
- **Ctrl+End.** Move cursor to bottom right of table

A summary of the added Editing functionality is as follows:

- **F2.** Edit contents of cell
- **Double-click.** Edit contents of cell
- **Alt+Enter.** Add multiple rows to cell
- **Delete.** Delete contents of cell without activating cell
- **Backspace.** Delete contents of cell and activate cell
- **Ctrl+Delete.** Delete one word at a time
- **Edit.** Edit multiple rows heights simultaneously
- **Lock.** Lock row height and column width
- **Copy.** Copy cells from Excel to SolidWorks tables

Retiring the Excel-based BOM
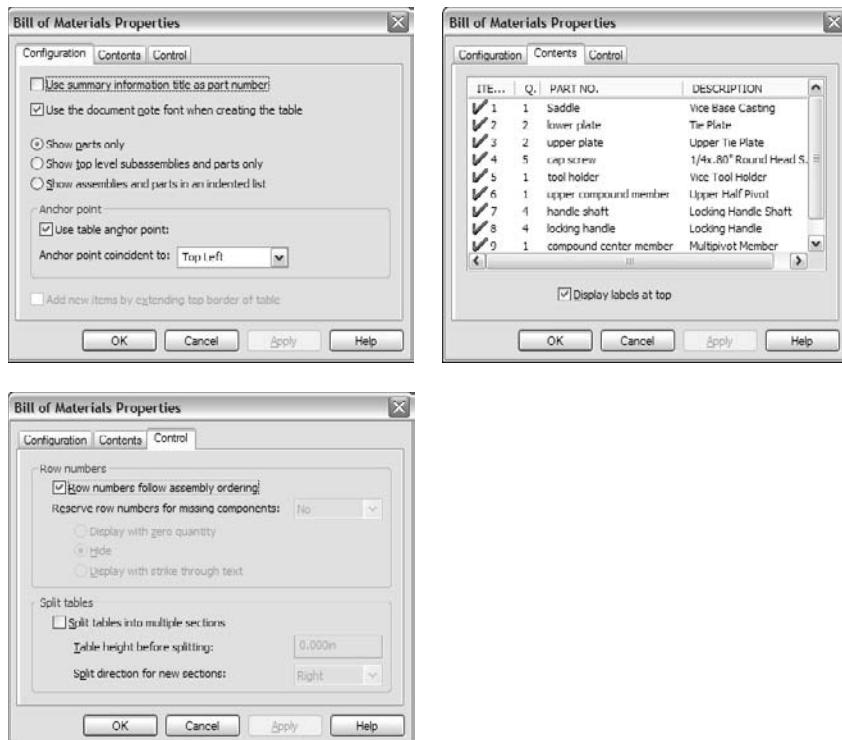
In previous releases, the Excel-based BOM was the only way to add a BOM to a drawing. This feature has been replaced in most respects by the SolidWorks native table-driven BOM, but many people still use the Excel-based BOM either out of habit or to comply with legacy standards. Figure 24.9 shows the interface for the Excel-based BOM.

Best Practice
Unless you have a compelling reason to do otherwise, I recommend using the SolidWorks table-based BOM because it is the function that will be best supported in future versions of SolidWorks software. ∎

The interface for Excel-based BOMs

Using Design Tables

Design Tables that are used to drive configurations of parts and assemblies can be shown on the drawing. This is often called a *tabulated* drawing and is typical of parts that have a basic shape that is common among several sizes or versions of the part. The sizes are shown by a symbol on the drawing, with a column headed by that symbol showing the available dimensions and the corresponding size (configuration) names.

You can insert a Design Table into a drawing by choosing Insert ➪ Tables ➪ Design Table or by clicking the Design Table button on the Tables toolbar. In either case, you must pre-select a drawing view of a part or assembly that contains a Design Table before the menu selection or toolbar button become activated.

Design Tables that are displayed in this way are often formatted visually to some extent. It is necessary to hide columns and rows unless you want the dimension or feature name syntax to display on the drawing as well as the values. Extra columns and rows are often added to make the Design Table readable. The image to the left in Figure 24.10 shows a Design Table that is formatted to be placed on a drawing. The image to the right shows the same Design Table with all the information visible. The first column and the first row are hidden to make the table more readable on the drawing, and the second column and second row use the $user_notes header to format the names.

Figure 24.11 shows the drawing with the table inserted. To display the table properly, you have to edit the table in the window of the parent document and adjust the border of the table to be exactly how you want it to appear on the drawing. The adjusted table is shown in Figure 24.11.

The labeled dimensions were created by simply making reference dimensions and overwriting the <DIM> value in the Dimension Text panel of the Dimension PropertyManager. If you would like to examine this data more closely, the drawing and part are included on the CD-ROM. The drawing is named `Chapter 24-DT.slddrw`.
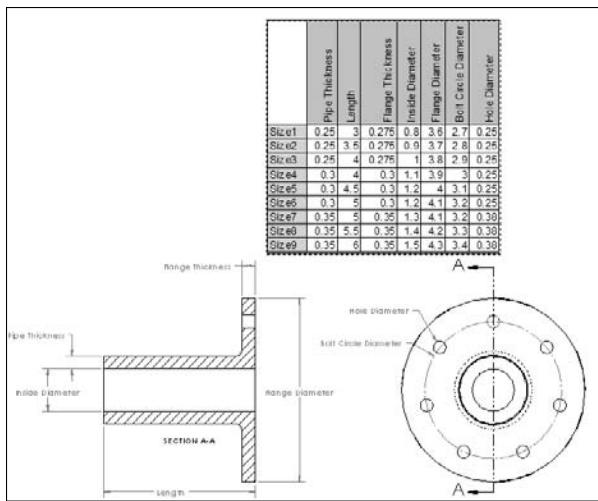
FIGURE 24.10

A Design Table prepared to be placed on a drawing

FIGURE 24.11

A drawing with the Design Table inserted

This drawing uses a part Design Table, but you can also place assembly Design Tables onto the drawing. This type of drawing is often called a *tabulated* drawing.

If you need to place something on your drawing such as a Design Table, but it does not appear that the Design Table is going to meet your needs, you may want to simply copy the data out of the Design Table and re-create it in a static Excel spreadsheet. The Design Table that you place on the drawing updates if it is changed in the part or assembly, just like the drawing geometry, but you must manually update an Excel spreadsheet that is created from copied data. Again, you must answer the question about whether the automatic functions make up for the cost of setting them up to work for you. In many cases they do, but in other cases they require more work than they save.

Placing Hole Tables on Drawings

You can place Hole Tables on drawings to include information such as the size, position, and number of holes or slots of a given size on a drawing. Only circular holes and through slots are recognized. You do not have to use the Hole Wizard or simple hole features to make the holes. The Hole Table will not recognize counterbored slots or even slots with a chamfer edgebreak. The position is given relative to a selected reference position, and the holes are labeled. I do not recommend using a Hole Table for slots unless you first test to make sure that you are getting correct data.

Like other table types, Hole Tables can use templates. As with other templates, you should locate Hole Table templates in a library area outside of your local SolidWorks installation folder. You can
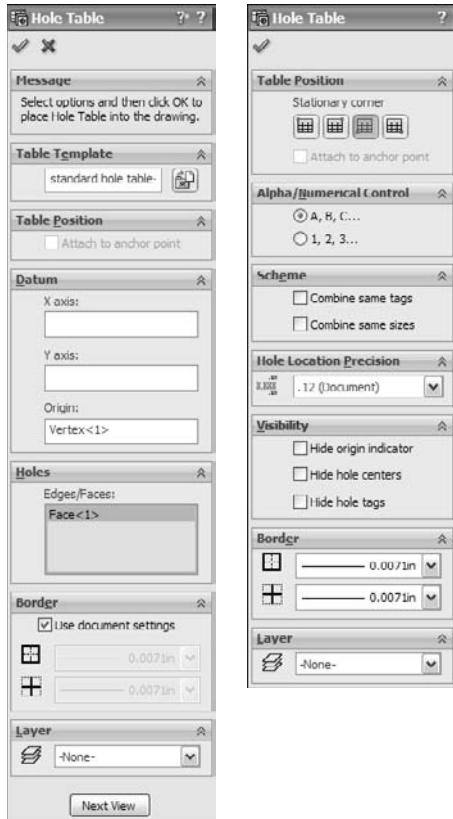
then direct SolidWorks to this location by choosing Tools ➪ Options ➪ File Locations and using the path settings.

Hole Tables use anchors in exactly the same way as BOMs. For more information, see the section on table anchors earlier in this chapter.

Figure 24.12 shows the PropertyManager for a Hole Table. The image on the left is the PropertyManager you get when you create the table, and the one on the right is the one you get when you edit the table. Figure 24.13 shows the resulting Hole Table on a drawing with a part that contains holes. The table incorporates holes from multiple views, using a different zero reference for each view.
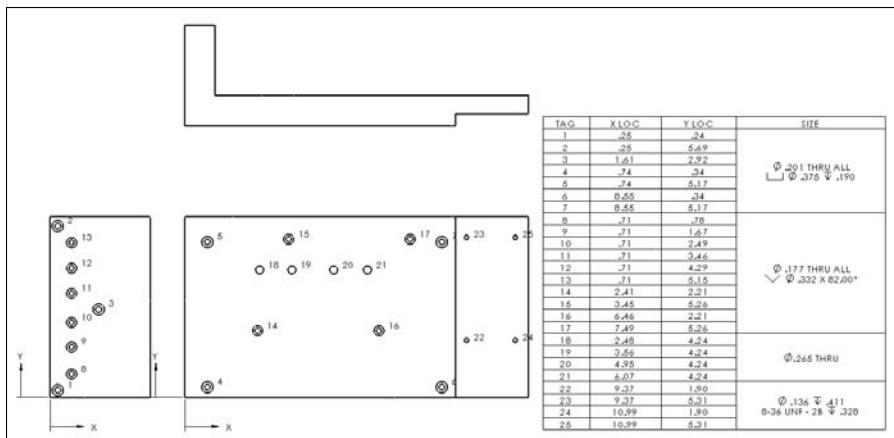
FIGURE 24.12

The PropertyManager for the Hole Table

A Hole Table combining holes in different views

To initiate the Hole Table function, you must first select a view. You can access the Hole Table function by choosing Insert ➪ Table ➪ Hole Table.

To specify the datum, either select an edge in each direction to serve as the zero mark for the X and Y directions or select a vertex or point to serve as the Origin in both directions.

To select the holes to be included in the table, activate the Holes panel selection box, and either select the hole edges directly or select the faces on which the holes are located. Once you place the table, you can add holes or change the datum information. To do this, click on the RMB menu of the Hole Table entry in the Drawing FeatureManager, and select Edit Feature. The RMB menu is shown in the middle image of Figure 24.12. If you simply select a Hole Table that has already been created, the Properties interface displays, as shown in the top-right image. You can resize columns and rows in the same way as for BOM tables.
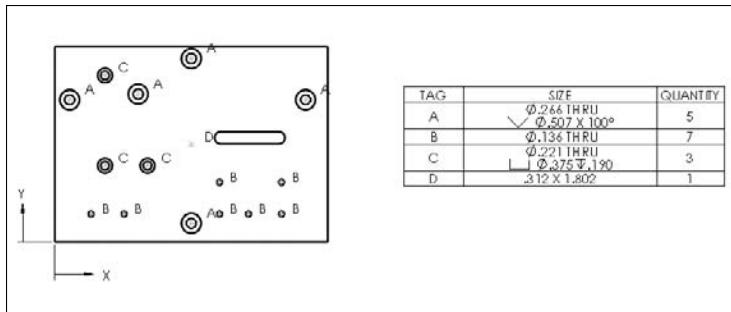
In the table in Figure 24.13, the Combine Same Sizes option is used, which causes several of the cells of the table to merge. If you use the Combine Same Tags option, then the hole locations are not displayed — only the hole callout description and the quantity appear. Figure 24.14 shows this arrangement.

You can control the hole callout description used in Figure 24.14 by using the file named `call-outformat.txt`, which is found in the `lang\english` subdirectory of the SolidWorks installation directory. Again, if you customize this file, then you should keep it in a library external to the installation directory and list it in the Tools ➪ Options ➪ File Locations area. This text file enables you to specify how hole callouts are specified for different types of holes.

FIGURE 24.14

The Combine Same Tags option used with a Hole Table that includes a slot

Using Revision Tables

You can use Revision Tables in SolidWorks in conjunction with SolidWorks Workgroup PDM, but this integration goes beyond the scope of this book. The Revision Table uses a table anchor, which is used in exactly the same way as the BOM table. Revision Tables also use templates in the same way as the other table types, and it is recommended to move customized templates to a library location and specify the location in Tools ⇨ Options ⇨ File Locations.

Figure 24.15 shows the Revision Table PropertyManager interface where you can create and control the settings for the table. You can find the default settings for Revision Tables by choosing Tools ⇨ Options ⇨ Document Properties ⇨ Drafting Standard ⇨ Tables ⇨ Revision.

The settings are now contained in a single PropertyManager; a toolbar and a RMB menu were formerly contained in five PropertyManager pages.
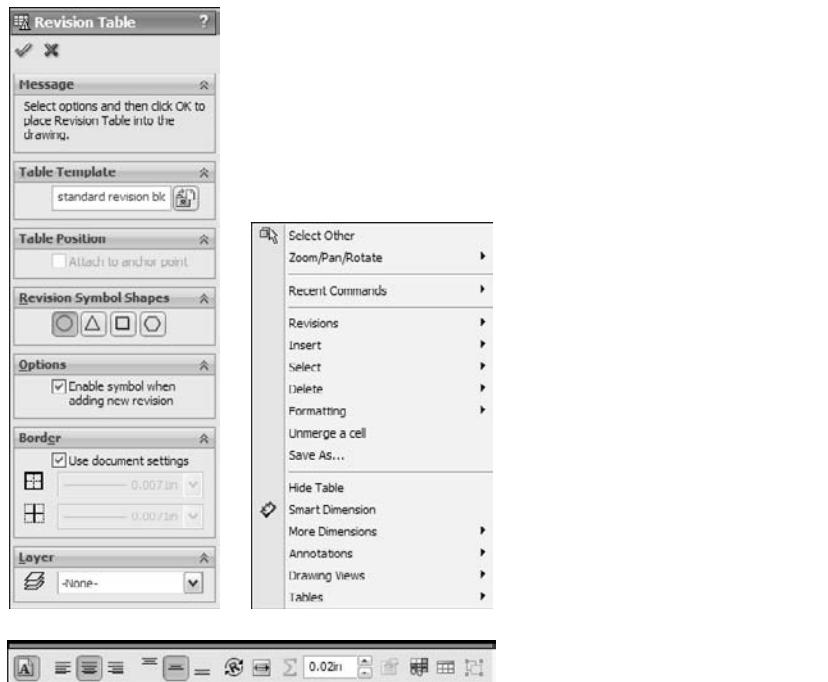
The image in the upper left of Figure 24.15 is the PropertyManager interface that displays when you initially create the Revision Table. The upper-right image is the RMB menu for the Revision Table, and the bottom toolbar is the formatting toolbar that displays when you select the Revision Table.

You can initiate the Revision Table function through the menus or the Tables toolbar. However, this function simply creates the table; it does not populate it. You must set the table anchor in the format in order for the Table Anchor to work. Additional columns may be added or formatted to accept other data. Once you have created the columns or formatting, you can save the changes to a template, which is also available through the RMB menu.

You can add a revision to the table by right-clicking the table and choosing Revisions ⇨ Add Revision. This includes control over whether the revision uses numerical or alphabetical revision levels, but does not provide for more complex revisioning schemes.

FIGURE 24.15

The Revision Table PropertyManager interface

Immediately after you have created the revision, if the option is enabled, you are prompted to place a balloon that contains the revision level to identify what has been changed. To finish placing symbols, you can press Esc. When you are finished placing the balloons, you can fill in the description of the revision by double-clicking in the Description cell where you want to add text. Figure 24.16 shows a Revision Table with balloon symbols placed on the drawing.

Revision Tables work by creating a Revision custom property in the drawing document, and by incrementing this revision each time a revision is added to the table. Additional columns linked to custom properties can be added to Revision Tables and Revision Table templates.
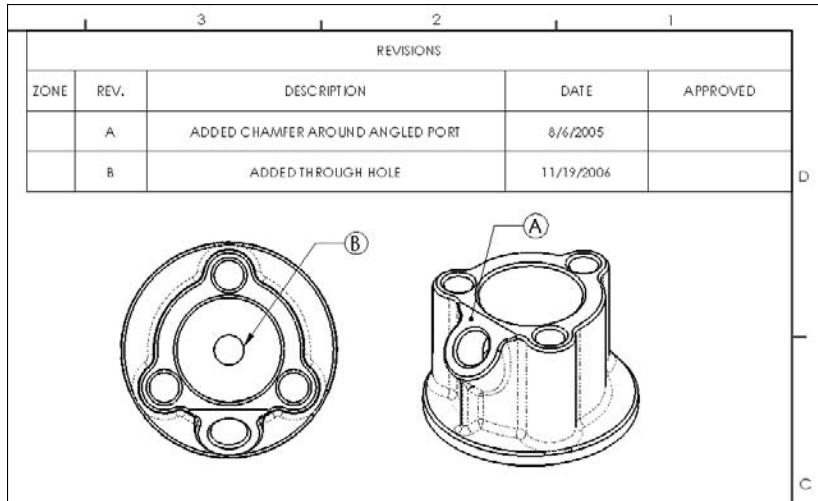
Cross-Reference

Gauge Tables and Bend Tables are specific to sheet metal parts and are covered in detail in Chapter 29. Weldment Cut Lists are a special type of table that closely resembles a BOM table in many ways. These are discussed in Chapter 31, which covers Weldments. ■

FIGURE 24.16

A Revision Table with balloon symbols

Using General Tables

General Tables can be used for any type of tabulated data. Column headers can be filled with either text labels or custom property links. Regular Excel OLE objects can also be used for the same purpose, and depending on the application, you may prefer this.

The General Table uses the filename extension `*.sldtbt`. You can create it without a template, as a simple block of four empty cells, or you can use a template that has a set of pre-created headers.

Using Tables in Models

Proponents of solid modeling have been saying for years that 2D drawings are going to disappear. I'm not convinced. Paper drawings will continue to be useful until all old manufacturing methods are abandoned, and I don't see that happening in my lifetime. People who use modern manufacturing methods have already eliminated drawings, but it may never happen across the board.
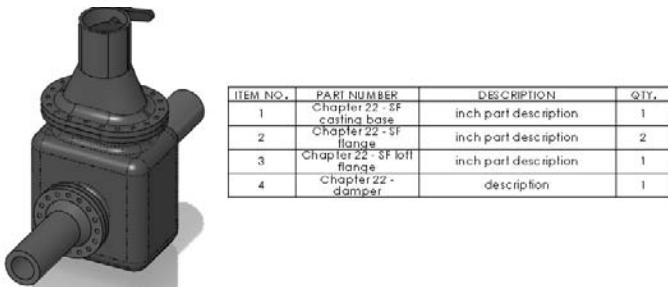
But because some companies rely on 2D and paper drawings less, the industry is developing new ways to create 2D-type documentation inside a 3D document. The ANSI Y14.41 standard is primarily about this transition.

SolidWorks is responding to this type of requirement by adding features that enable you to document the 3D data. Placing BOMs in assembly files is one way of doing this. Placing 2D type data

into 3D model documents can reduce the need for paper or even electronic 2D documentation. Figure 24.17 shows a BOM inside an assembly model document.

Displaying BOM data inside an assembly document

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Chapter 22 - SF casting base	inch part description	1
2	Chapter 22 - SF flange	inch part description	2
3	Chapter 22 - SF loft flange	inch part description	1
4	Chapter 22 - damper	description	1

It is a little tricky to get the relative scale correct between the table and the model. To do this, you have to adjust the zoom state of the model until it is fairly small within the screen, then place the table. After the table is placed, the assembly and the table zoom together. Most users get around this issue by viewing the table in a separate window.

Another type of table that you can use within a 3D model document is the Title Block table. You can use Title Block tables inside parts and assemblies. You can use them in the drawing to fill in information about the part or assembly, while avoiding creating a full 2D drawing.

Tutorial: Using BOMs

Rather than having tutorials for every table type, this chapter has tutorials only for the BOM, Hole Table, and Revision Table. You can transfer the skills you use with these types to the other types.

This tutorial guides you through the steps that are necessary to prepare an assembly for the drawing and BOM. Configurations and custom properties are used in this example. Remember that if a drawing view is cross-hatched and you cannot see the geometry, then you may have to press Ctrl+Q to rebuild it. Follow these steps:

1. **Begin this tutorial with SolidWorks closed and Windows Explorer open.**

2. **If you have not already done so, create a folder for a library that is not in your SolidWorks installation folder.** Call it `D:\Library\` or something similar. Make a folder inside this folder called Drawing Templates. Copy the files from the CD-ROM named `inch B.drwdot` and `inch B (no views).drwdot` to this new folder.
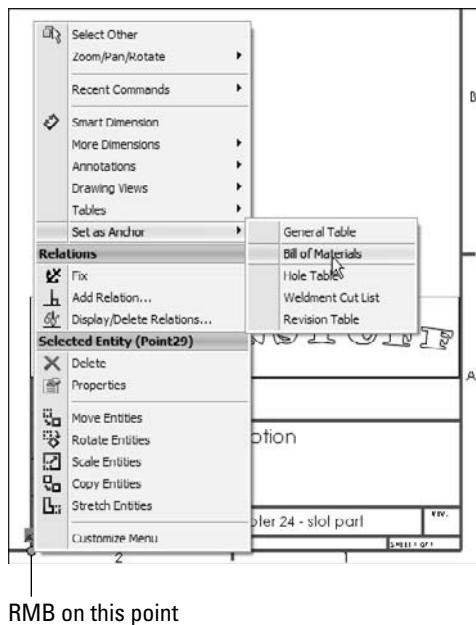
3. **Launch SolidWorks and choose Tools ⇨ Options ⇨ File Locations ⇨ Document Template.** Click the Add button and add the new library path to the list. Shut down SolidWorks and restart it.

4. **Open the assembly** `Chapter 24 – BOM Assy.sldasm` **from the CD-ROM.**

5. **Click the Make Drawing From Part/Assembly button, make a new drawing of the assembly from the drawing template in the folder created in Steps 2 and 3.**

6. **Delete the isometric view, and in its place make a new drawing view using the named model view "exploded."** If prompted to use true dimensions in an isometric view, click Accept.

7. **Edit the sheet format.** Right-click the sketch point at the location indicated in Figure 24.18. In the popup menu that appears, select Set as Anchor and then select Bill of Materials.

8. **Exit Edit Sheet Format mode by selecting Edit Sheet from the RMB menu.**

9. **Select the new view and show it in the exploded state (right-click, Properties, Show in Exploded State).** Then choose Insert ⇨ Table ⇨ Bill of Materials or click the Bill of Materials button in the Tables toolbar. Use the default selections, except in the panels shown in Figure 24.19.
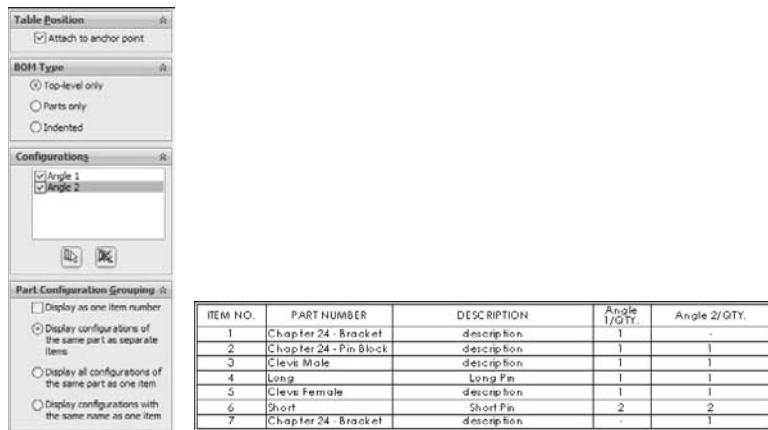
FIGURE 24.18

Setting the Table Anchor

RMB on this point

FIGURE 24.19

Creating the Bill of Materials
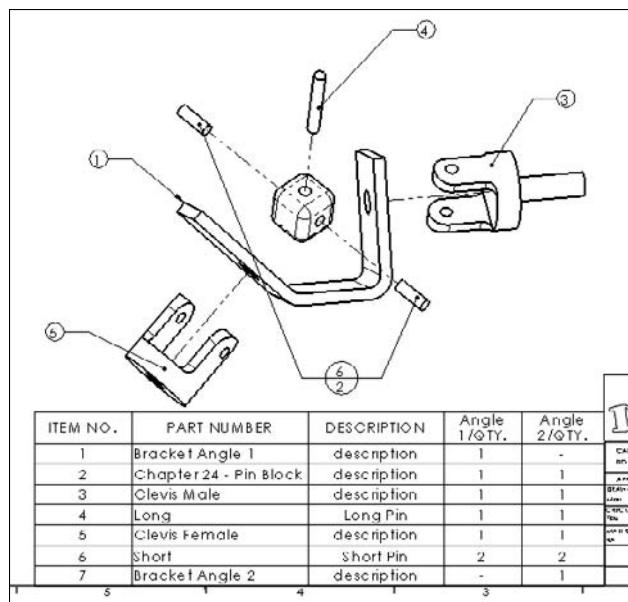
10. **Click inside the exploded view, but not on any part geometry, and then select the Autoballoon tool from the Annotations toolbar.** Toggle through the available options to see whether any of the possible autoballoon configurations meets your needs. If not, use the standard Balloon tool to select the part and place the balloon. This gives you more control over the attachment points and placement of the balloons.

11. **Change the balloon for the short pin to be a circular split-line balloon (do this by clicking the balloon and then switching the style in the PropertyManager).** Notice that the quantity appears in the bottom of the balloon. The drawing view and the BOM should now look like Figure 24.20.

    Add a second leader to the balloon for the short pin by Ctrl-dragging the attachment point for the first leader from one pin to the other.

12. **Notice that several of the parts use a default description of "description."** Edit each of these parts by right-clicking the part's row in the BOM table and selecting Open *<file-name>* from the menu. Change the Description custom property in each part. Keep in mind that this may be handled differently for configured parts.

13. **The Bracket part is listed twice using the configuration name because of the way the configurations are set up for the parts.** To list the bracket only once using the filename, open the bracket, RMB+click one of the configuration names in the ConfigurationManager, and select Properties. In the Bill of Materials Options panel, select Document Name from the drop-down list. Do this for the other configuration, as well.

    Notice also that the Description field holds the configuration-specific custom property for Description, which is used in the BOM.

FIGURE 24.20
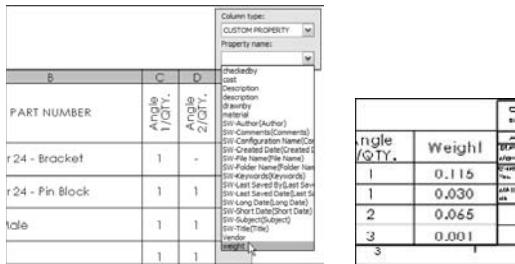
The drawing view and the BOM after Step 11

ITEM NO.	PART NUMBER	DESCRIPTION	Angle 1/QTY.	Angle 2/QTY.
1	Bracket Angle 1	description	1	-
2	Chapter 24 - Pin Block	description	1	1
3	Clevis Male	description	1	1
4	Long	Long Pin	1	1
5	Clevis Female	description	1	1
6	Short	Short Pin	2	2
7	Bracket Angle 2	description	-	1

14. **Toggle back to the drawing (pressing Ctrl+Tab), select anywhere on the BOM table, and then select Table Properties from the PropertyManager.** Expand the Part Configuration Grouping panel, and select the Display all configurations of the same part as one item option. This changes how the bracket displays, as well as the pins.

15. **Now add a column to the BOM that calls on an existing custom property that is already in all the parts.** Place the cursor over the last column on the right and RMB+click it. Choose Insert ⇨ Column Right. This places a new column to the right of the last one and displays a pop-up menu that enables you to set the column to be driven by a custom property , as shown in Figure 24.21.

16. **In the first drop-down selection box, select the Weight custom property.** Click the green check mark icon to accept the changes. If the popup menu disappears and you need to get it back, double-click the column header, and it will reappear.

17. **You can save the BOM with the additional column as a BOM template by right-clicking anywhere in the BOM and selecting Save As.** You can then set the type to a BOM template and the directory to the library location for BOM templates.

Adding a column to the BOM

If you would like to compare your results against mine, the finished drawing is called `Chapter 24 – BOM Tutorial Finished.slddrw`.

Tutorial: Using Hole Tables

This tutorial guides you through creating and using setting changes that are common in SolidWorks Hole Tables. Follow these steps:
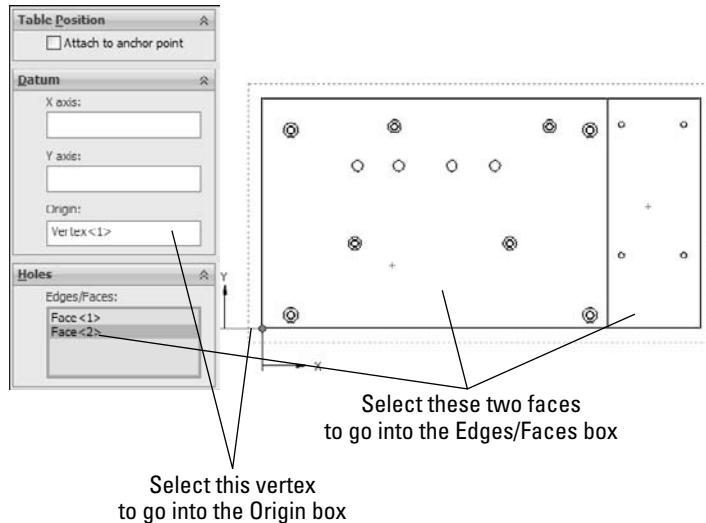
1. **Create a new drawing from the inch B (no views).drwdot template.** If you have not done the BOM tutorial, then move the drawing template named `inchB.drwdot` from the Chapter 24 materials on the CD-ROM to your library location for drawing templates. Then create the drawing from the template.

2. **Click the Model View button on the Drawings toolbar, and browse to the part named** `Chapter 24 – Hole Table Part.sldprt`.

3. **Place a Front view and project a Left view and an isometric view.** Then press Esc to quit the command. Finally, delete the four pre-defined views.

4. **There is not an anchor in this template for a Hole Table.** If you would like to create one, this would be a good time to do so. Follow the steps in the BOM tutorial for specifying the anchor point.

5. **Click the Hole Table button in the Tables toolbar.** Figure 24.22 shows a section of the Hole Table PropertyManager with the selections that you need to make for this Hole Table.

6. **Once you have completed the selections, click the Next View button at the bottom of the PropertyManager, and make similar selections in the Left view.** The holes for both views are added to a single Hole Table.

FIGURE 24.22

The Hole Table PropertyManager and selections

Select these two faces
to go into the Edges/Faces box

Select this vertex
to go into the Origin box

7. **The table is created using the default settings established in Tools ⇨ Options ⇨ Document Properties ⇨ Tables, but you can change them here for this specific table.** Click anywhere in the table, and then select Table Properties at the bottom of the PropertyManager. Changing from numerical to alphabetical assigns a letter to each hole type and a number to each instance of the type. Make this change and update the table. Figure 24.23 shows the table before and after the changes.

FIGURE 24.23

Using numerical and alphabetical hole tag identification

8. **Change the number of decimal places used in the Hole Table from two places to three.** You can do this in the PropertyManager.

9. **Deselect the Hide Hole Centers option in the Visibility panel.**

10. **Select the Combine same sizes option in the PropertyManager.** This demonstrates a much cleaner table and would be a good thing to have in the tutorial.

11. **Save the drawing.**

Tutorial: Using Revision Tables

In this tutorial, you create a basic Revision Table and make a template. Follow these steps:

1. **Using a drawing that you completed in one of the previous tutorials, make sure that a Revision Table Anchor has been placed in the upper-right corner of the Sheet Format.** You must edit the Sheet Format to do this by RMB+clicking the point that you want to use for the anchor. Remember to select Edit Sheet from the RMB menu to exit Edit Sheet Format mode.

Note
Ideally, the anchors for all table types should be set in templates and formats, but it is set up here to give you some practice creating the anchors. ∎

2. **Click the Revision Table button on the Tables toolbar.** Select the Attach to Anchor option in the PropertyManager. Click the green check mark icon to accept the table. Figure 24.24 shows the initial stub of the Revision Table.

FIGURE 24.24

The initial stub of the Revision Table

Tip
Drawing templates can be saved with the Revision Table stub if it also has a format. The Revision Table is not saved with the format because it has to go on the drawing sheet. ∎

3. **To initiate a new revision level in the Revision Table, RMB+click the table and choose Revisions ⇨ Add Revision.**

4. **Depending on the default settings in Tools ⇨ Options ⇨ Document Properties ⇨ Drafting Standard ⇨ Tables ⇨ Revision, the first revision will be either A or 1.** If you are using PDMWorks Workgroup, then you may have other options.

5. **Depending on your options settings, you may immediately be prompted to place a balloon that contains the new revision level.** You can place balloons with or without leaders. The balloons are meant to indicate areas of the drawing that are affected by the revision. Press Esc when you are finished placing the balloons.

Note

Be careful when using balloons on assembly drawings or other drawings that already have balloons on them for other purposes. It may be a good idea to use a distinctively shaped balloon for Revision Tables. ■

6. **To add text to the Description field, simply click in the field and start typing.** The text automatically wraps to fit the box.

7. **Practice by adding a couple of revisions, balloons, and descriptions.**

8. **After you have added a couple of revisions, check the custom properties by choosing File ⇨ Properties ⇨ Custom.** Notice that a revision property has been added, and the latest revision is represented by the value of the custom property.

Note

The number of revisions kept in the Revision Table is no longer an option as it was in previous releases, but in its place you can now control how Revision Tables interact with multiple sheets. ■

9. **You can add columns in the same way that they were added to the BOM.** You can merge and unmerge cells, and link properties to cells. With the cursor over the last column (Approved), RMB+click and choose Insert ⇨ Column Left. In the Column Properties, select Custom, and from the Properties drop-down menu, select DrawnBy. If DrawnBy is not a property in this template, create it through the interface at File ⇨ Properties. Accept the changes by clicking the green check mark icon.

10. **Save the template by right-clicking anywhere in the Revision Table and choosing Save As ⇨ Rev Table Templates.** Then save it to the appropriate location outside the SolidWorks installation directory.

Summary

SolidWorks enables you to work with both tables that are highly specialized for particular uses, and General Tables, which are available for any type of tabulated data. The most frequently used types are BOMs, Hole Tables, and Revision Tables. Design Tables that drive part and assembly configurations can also be placed on a 2D drawing, but in these cases, some formatting is usually necessary to make the Design Table presentable and the information on it easy to read.

Other types of tables, such as Gauge Tables and Bend Tables, and Weldment Cut Lists are specialized for sheet metal and weldment parts, respectively, and are discussed in chapters that cover those topics.

Using Layers, Line Fonts, and Colors

AutoCAD has left its mark on CAD users of all kinds in the form of the default expectations users have about CAD software. A few common expectations are that layers, the Command Line, and paper space/ model space need to exist in order for graphical software to be considered CAD, and printing should be really difficult.

When former AutoCAD users make the switch to SolidWorks, the questions start: Where is the Command Line? How do I put parts on layers? How do I change the background color to black? And my personal favorite, Where is the zero-radius trim?

This chapter addresses AutoCAD-like functions in the SolidWorks drawing environment. The goal is not to make the functions look or work or compare in any way to AutoCAD, but to simply to make them useful in the context of the SolidWorks software. It is never productive to try to use SolidWorks as if it were AutoCAD. If you are making the transition, you will be much further ahead if you just embrace SolidWorks for what it is, and accept that it does not work like AutoCAD. You will be even further ahead if you do not assume that AutoCAD functionality is universal.

Controlling Layers

Layers are only available in SolidWorks drawing documents, not in 3D modeling or even sketching at all. Even in drawings, layers do not see a lot of use. This is not to say that they serve no purpose, just that the software does not depend on them.

Layers in imported 2D data

When you import data through DXF (Data eXchange Format) or DWG format files, the layers that exist in the original data are brought forward into SolidWorks, and you can use them in a similar way to the original AutoCAD usage. For example, you can select and deselect layers (visible or hidden), and you can change layer names, descriptions, color, line thickness, and line style.

The way you intend to use the imported data determines how you should open the file. If you only intend to view and print the drawing, then I would suggest using DWG Editor, which is installed with SolidWorks and enables you to do almost anything you can do with basic AutoCAD. It also has the advantage of having a familiar interface for the AutoCAD user. DWG editor is available from the Start menu, by choosing Programs ➪ SolidWorks ➪ DWGeditor.

If you need to integrate data from the imported document into a native SolidWorks drawing, you can open the DWG file from the normal Open dialog box in SolidWorks.

Tip

If you want to make a 3D part from the 2D data in the DWG file, you may want to import the drawing into the part sketch environment. This usually leads to some speed issues. If you prefer, sketch entities can also be copied from the drawing to the model sketch. You can even copy entities from DWG Editor to the SolidWorks sketch. The sketch needs to be open in order to paste the sketch entities. In the case where imported 2D data is brought into the model sketch, you lose all the layer information because part and assembly documents do not allow layers. ■

The colors assigned to layers in data coming from AutoCAD are often based on a black background, and so they can be difficult to see on a white background. The two ways of dealing with this are to change the SolidWorks drawing sheet color to something dark or to change the individual layer colors to something dark. Either method is easy, although if you have to send the 2D data back to its source, it may be best to temporarily change the drawing sheet color.

Figure 25.1 shows the layer interface with an imported drawing in the background. To open the Layers dialog box, click the Layer Properties button, which is found on both the Layer and Line Format toolbars.

Be aware that many items in an imported drawing may come into SolidWorks as blocks. These items may need to be exploded before you can work with them. This is often the case with the drawing border, title block, or format.
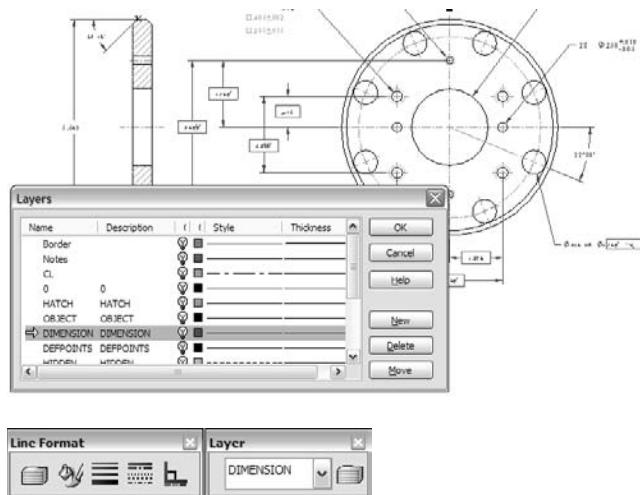
Layers on the sheet format

One of the most obvious uses of layers is for the drawing border sketch lines on the sheet format. The sketch lines used to create the border often have a heavier line weight and a different color that easily distinguishes them from model geometry.

FIGURE 25.1

The Layers dialog box and the Layer toolbar

You can assign layers in one of three ways:

- Select existing items, and then select a layer from the drop-down list on the Layer toolbar.

- Set the active layer and create new items.

- While creating items such as sketch entities and annotations, select the layer for the new entity directly from the PropertyManager.

To set a layer to the active layer, double-click it from the Layers dialog box, as shown in Figure 25.1, or change it from the drop-down list on the Layer toolbar. When you assign an active layer, other newly created entities are also placed on the layer, not just sketch entities. Symbols, annotations, blocks, and other elements can also be put onto layers. If you are not particular about the layering scheme on a drawing, then it may be advisable to set the active layer to None, which is a valid option in the Layer toolbar drop-down list.

When you create a new layer in SolidWorks, the new layer becomes the active layer, and any new items that are added are automatically placed on that layer.

Another option when building a sheet format, or any other drawing function that requires sketching, is to use a special layer for construction geometry. This enables you to hide the layer when it is not being used, but it still maintains its relations. Hidden layers can be used in several other ways (for example, as standard notes on the drawing) and they can be easily turned on or off.

Dimensions and notes on layers

SolidWorks drawings have a tendency to be drab black-and-white drawings in contrast to AutoCAD drawings, which often seem to take on a plethora of contrasting colors. Still, drawings are often a little easier to comprehend when different types of items are colored differently, but in order for to do this, you must apply the coloring scheme consistently. Dimensions and annotations can also be placed on layers in the three ways described in the previous section (active layer, from the PropertyManager during creation, and through the drop-down list on the Layer toolbar). However, the line styles do not affect dimensions and notes, only the color and visibility settings.

Components on layers

Assembly drawings probably suffer the most from the monochromatic nature of most SolidWorks drawings because individual components can be difficult to identify when everything is the same color. This is why SolidWorks users typically color parts in the shaded model assembly window. It only makes sense that they would want to do the same thing on the drawing.

An intuitive and easy workaround for this problem would seem to be to simply turn the drawing view to a wireframe mode just as you can change a drawing view to a shaded mode to show the parts in color. Unfortunately, wireframe on drawings always defaults to black edges. Even if you set a Display State using some wireframe parts where the wireframe displays in the same color as the shaded part in the assembly window, this still appears in black and white on the drawing.
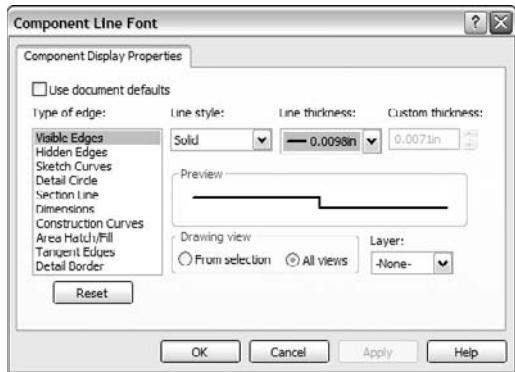
Your only option to display the components of an assembly in different colors while using a wireframe display mode is to set the Component Line Font options. (Line Fonts are covered in the next section.) The Component Line Font dialog box contains a Layer setting, which you can use to put a part on a layer. If the layer is set up with a color, then the part displays with that color in all views of the drawing or in just the current view, depending on your settings. While it does take a little time to set up the individual layers for each part and then to set the parts to the layers, it is better than the alternative, which is to do nothing.

You can access the Component Line Font dialog box by right-clicking a component in a drawing view. The Component Line Font dialog box is shown in Figure 25.2.

In normal use, the Use document defaults option is selected and all the settings in the dialog box are grayed out. To gain access to these settings, you must deselect the Use document defaults option, as shown in Figure 25.2.

The Component Line Font dialog box

Controlling Line Format

The Line Format toolbar contains the Layer tool and four additional tools that control lines: Line Color, Line Thickness, Line Style, and Color Display Mode. These settings can be controlled separately from layers; therefore, they can be used in model sketches as well as on drawings. In the model, the line font can only be displayed for inactive sketches. Any sketch that is both closed and shown can be displayed with the Line Format settings.

Cross-Reference
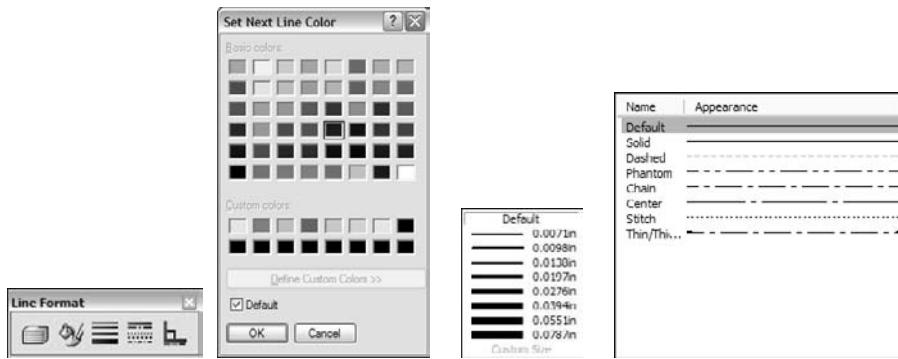
For more information on using line styles in the model, see Chapter 6. ∎

Figure 25.3 shows the Line Format toolbar along with the interfaces for Line Color, Line Thickness, and Line Style.

FIGURE 25.3

The Line Format toolbar and related interface options

Note

The term line font refers to a combination of style, end cap, and thickness. To set line fonts, choose Tools ⇨ Options ⇨ Document Properties ⇨ Line Font and use the document-specific settings. ■

Using the Line Format settings

You can specify the Line Format settings using two different methods. In the first method, you can set them with nothing selected, in which case they function like System Options (the new setting takes effect for all documents that are opened on the current computer). In the second method, if they are set with sketch entities or edges selected, then the settings apply only to the selected entities.

Caution

If you change these settings with nothing selected, then the Line Format settings for color, thickness, and style function as system options. ■

End Cap Style

Another option for the Line Font settings is the End Cap Style. This offers an important option, especially for thick lines. The three options are flat, round, and square. Of these, the square style is usually most appropriate. In the past, flat was the default style. To find this setting, choose Tools ⇨ Options ⇨ Document Properties ⇨ Line Font. You may want to change this setting and update your drawing template files.

Figure 25.4 shows the difference between the three options of End Cap Style.

FIGURE 25.4

The End Cap Style setting options
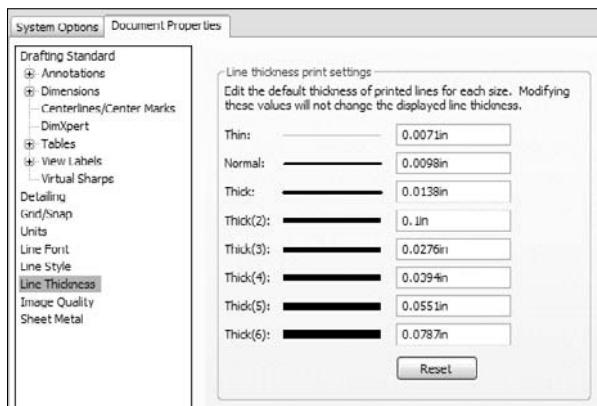
Line Thickness settings

The Line Thickness settings are Default, Custom, and eight width settings. Interestingly, the different thicknesses are named in the interface where you set the actual thicknesses, but not in the interface where you set lines to thicknesses. Figure 25.5 shows the Line Thickness page (Tools ⇨ Options ⇨ Document Properties ⇨ Line Thickness).

FIGURE 25.5

The Line Thickness settings in Tools ⇨ Options

The way the line thickness is shown in the drawing does not have anything to do with the numerical width that is assigned to it. For example, in Figure 25.5, notice that Thick(2) is set to 0.1 inch, which is much wider than Thick(3). Changing the numbers only affects printed line thickness; it does not affect the display at all.

Caution

The Line Thickness settings are document options, not system options. As a result, two drawings with the same line type assignments may have different numerical widths; thus, the two drawings would print differently on the same computer. ∎

Line Style setting

You can create custom line styles using the syntax shown on the Line Style page (Tools ➪ Options ➪ Document Properties ➪ Line Style). This is a document-specific setting; therefore, if you make a custom line style and want to use it in another document, you have to save it out (as a `*.sldlin` file) and load it into the other document. Also, if you save your templates with this line style loaded, then you will not have to load the styles for any document made from that template.

Color Display mode

Color Display mode toggles between the display of assigned colors and standard sketch state colors. This is primarily used in drawings when you are making sketches where sketch relations are important.

Hiding and Showing Edges

Sometimes, for illustrative purposes, it is desirable to hide certain edges in drawing views. The Hide/Show Edge toolbar button is on the Line Format toolbar, although it may not be on the toolbar by default. You can choose Tools ➪ Customize to put it on a toolbar.

To use the Hide Edge tool, simply select the edges that you would like to hide, and click the Hide Edge toolbar button. To show the edges, click the Show Edge toolbar button; the cursor will now be able to select the hidden edges.

Be aware that if your view is in Draft Mode, edges that you hide will still be shown until the view is made into a High Quality view.

Note
Hide/Show Edges was formerly two separate toolbar buttons. In SolidWorks 2010, they are in a single button. ∎

Tutorial: Using Drawing Display Tools

Some of the functions described in this chapter are difficult to understand until you actually use them. This tutorial guides you through the functions step by step so that you can see them in action. Start here:

1. **From the CD-ROM, open the drawing called** `Chapter 25 – Tutorial.slddrw.` Make sure that the Layer and Line Format toolbars are active and that the Hide/Show Edges buttons are available on the Line Format toolbar.

2. **Right-click a blank space and select Edit Sheet Format from the menu.**

3. **Window+select everything on the format and use the drop-down list on the Layers toolbar to assign the selection to the Border layer.** Notice that this changes the color and the thickness of the sketch lines.

4. **Right-click a blank space and select Edit Sheet.**

5. **Click the Layer Properties button on either the Layer or Line Format toolbar.** Add new layers for each of the part groups, bracket, clevis, pins, and blocks, assigning different colors to each layer. Figure 25.6 shows the Layers dialog box with these layers created.

FIGURE 25.6

The Layers dialog box

Caution

Be aware that creating new layers leaves the last layer that you created active, as indicated by the yellow arrow in Figure 25.6. There is no way to set the active layer to None from the Layers dialog box; you have to do this using the drop-down list in the Layer toolbar. ∎

6. **Set the active layer to None in the Layer toolbar drop-down list.**

7. **Right-click the Bracket part in one of the views and select Component Line Font.** Deselect the Use Document Defaults option, and select the Bracket layer from the drop-down list in the lower-right corner of the dialog box, as shown in Figure 25.7. Make sure that the Drawing View option is set to All views.

8. **Repeat Step 7 for all the components, assigning each component to its own layer.** Notice how this makes the parts easier to identify.

Note
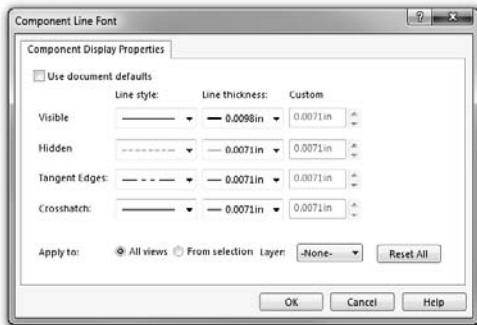
Alternatively, you could simply change the line style and thickness for each component. This saves you creating the layers, but you lose the color settings. The way SolidWorks handles line thickness and thickness values has changed significantly in SolidWorks 2010. The line thickness assignments in the Print dialog are still the old format. ∎
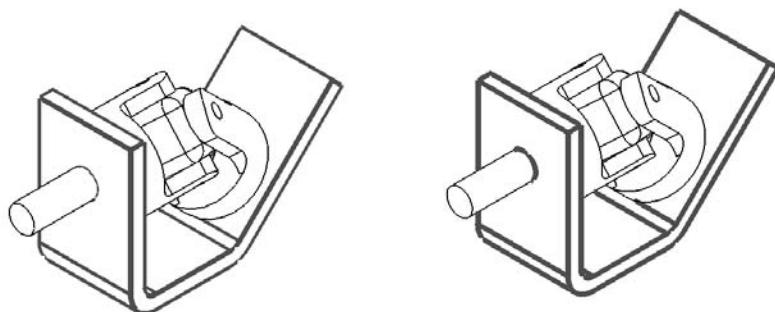
FIGURE 25.7

The Component Line Font dialog box

9. **Open the Component Line Font dialog box for the Bracket part again.** This time, set the Line thickness to 0.0787, and click OK. You may have to rebuild the drawing to show the change (Ctrl+B or Ctrl+Q). Figure 25.8 shows a detail of the corners that are created by the thick lines. Notice the notches created at the corners.

FIGURE 25.8

Applying thick edges
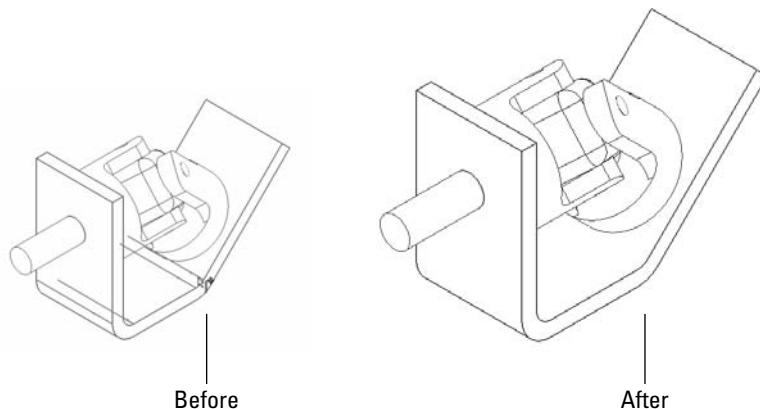
10. These notches are supposed to be fixed using the End Cap setting at Tools ➪ Options ➪ Document Properties ➪ Line Font. Set the End Cap Style to Square. Click OK to exit the Document Properties. In the drawing, select inside the view where you are working and make sure that it is set to High Quality. (The setting is found in the PropertyManager for the view in the Display Style panel. If it is already set to High Quality, then there will be no other view option; if it is not, then there will be an option that is set to Draft Quality.)

The image to the left in Figure 25.8 is the old setting with the draft quality view, and the image to the right is the new setting with the high quality view.

11. **In the Component Line Font dialog box, set the Line Weight setting back to Default for the Bracket part, but keep it on the Bracket layer.**

12. **In the isometric view, Ctrl+click all the tangent edges on the Bracket part, as shown in Figure 25.9.** Click the Hide/Show Edges toolbar button on the Line Format toolbar.

FIGURE 25.9

Hiding edges

Before   After

13. **Click the Hide/Show Edges toolbar button.** The PropertyManager message changes to indicate that you can now select hidden edges, and the hidden edges are shown. Ctrl+select the hidden edges and right-click when you are done.

Summary

While SolidWorks is not primarily built around the strength of its 2D drawing functionality, it offers more capabilities than most users take advantage of. Layers in SolidWorks offer adequate functionality, but could be improved by some automation to put parts on layers automatically; this would enable SolidWorks to show the parts in wireframe with the same colors assigned to the solid.

Other line and edge display functionality is sometimes difficult to find or access, or may have obscure functions. Creating drawings that use color to make them easier to read should be easier in SolidWorks than it is in AutoCAD.

Part VI

Using Advanced Techniques

The techniques on display in Part VI certainly fall into the advanced category. Not everyone needs these tools, but for those who do, advice on them can be difficult to find from other sources. I encourage you to explore the tools available here. All three chapters go together, so it is most beneficial to read them all in order.

Multi-body modeling is a technique that can open up new methods for you and help you realize the full potential of the software. Surfacing enables you to unlock the complete potential of SolidWorks and is as much for reference geometry as it is for complex shape creation. Master Model techniques enable you to drive several parts from a single part without using in-context assemblies.

Modeling Multi-bodies

You get multiple bodies in SolidWorks when you have simultaneous separate enclosed volumes within a part. It can be useful or problematic. Multi-body parts are not inherently good or bad, but depending on how you use them, they can mean either good news or bad news. This chapter will show you how to use them carefully to get the most benefit and avoid typical problems.

You could work with SolidWorks in such a way that you would never need to use multiple bodies inside a single part, ever. Almost everything that average users normally do can be done with a single solid body and without any knowledge of multi-body functionality whatsoever.

However, to access some more powerful functionality, and options that offer more flexibility, multi-body modeling is necessary. In fact, if you want to move on to surface modeling, multi-body knowledge is mandatory because in surface modeling, multi-body is the default.

Multi-body modeling is the gateway from basic solid modeling mainly described in the book up until now into the more advanced functionality that follows. The gateway can lead in two directions: it can lead to more power, more flexibility, more options, more advanced functionality, or it can lead to sloppy, bad habits that could get you or those who work with your data into some modeling hot water down the road. This chapter will help you tell the difference and avoid the pitfalls.

IN THIS CHAPTER

Creating models responsibly

Leveraging multi-body techniques

Learning how to use multi-bodies

Understanding how to manage bodies

Working with multi-body data

Working with multi-bodies tutorial

Using Powerful Tools Responsibly

I find myself giving the same responsible-modeling advice when discussing multi-bodies as I tend to give for in-context modeling and virtual components. This is because multi-bodies present some of the same issues as in-context modeling. The first similarity is that on the surface, both techniques appear to be all-day-every-day types of design tools. The second similarity is that there is more to the story than what initially appears.

The responsibility part arises when you are thinking about the users who will use the multi-body parts once they are created, including yourself. It is not always easy to remember how you executed a particular project 6 months and 100 models ago. Other users may have to edit your work, and if errors happen, then you have to be able to navigate the design intent without destroying the relationships in the FeatureManager or completely rebuilding it. This is the reason for trying to standardize best-practice issues, particularly with advanced functionality and particularly in larger organizations where more users may work with the data.

If you are an independent contractor and do not share your models with other SolidWorks users, then you have more flexibility to model how you like. As long as you can come back to the model and change it when you need to, more power to you.

Comparing multi-body modeling with assembly modeling

This concept is important, and so I will repeat it: Multi-body modeling is not assembly modeling. Many times when new users are introduced to the capabilities of multi-body modeling, the first thought that comes to mind is, "This is far easier than making assemblies." However, multi-body modeling should not be treated as a replacement for assembly modeling.

Several assembly type functions are missing or more difficult to obtain from multi-bodies. They include the following:

- Interference detection
- Dynamic assembly motion
- Exploded views
- Configs for separate parts
- Drawings for individual parts
- Center-of-gravity calculations for individual parts
- Mass property calculations for individual parts

To say that these functions are missing from or difficult in multi-bodies does not imply that they should or will be there someday. In fact, I believe that the distinction between multi-body and assembly modeling techniques should be kept as clear as possible. Simply because a technique is easier does not make it *better*. Above all, remember that modeling multi-body parts puts *all* the data for *all* the bodies in a *single* part file, in a *single* FeatureManager; there is no easy way to separate out the parametric features into individual parts later on, regardless of how complex the part becomes.

You may find parallels between making multi-body parts and making virtual components (parts that are saved within an assembly file). While both these techniques offer shortcuts or make some basic tasks easier, good reasons exist for being mindful of the "one part, one file" mentality, including:

- Segmenting rebuild times (the ability to rebuild one part instead of several)
- Segmenting large data sets (being able to work on one part at a time)
- Switching out parts
- Reusing parts
- Bills of Materials (BOMs)

Further, creating drawings of individual bodies of a multi-body part is more difficult than creating drawings of individual parts, not that it cannot be done (remember that starting in SolidWorks 2010 you can now specify bodies from a part to be used in a drawing view), just that it is more difficult. Also, editing the features of individual bodies is not as easy as if the individual body were an individual part. When you create several bodies in a single part, you constantly have to carry the feature and design intent overhead of *all* the features used to create *all* the bodies to edit any individual body.

Using multi-body techniques appropriately

You need to have a healthy respect for the problems that you can create for yourself and others by using multi-body modeling in inefficient or inappropriate ways. Still, appropriate uses for multi-body modeling do exist. You may hear people recommend that at the end of the FeatureManager, only a single solid body should remain, with the rest of the bodies either absorbed or deleted. On the other extreme, for some people, anything they can create is allowable. I recommend that if you decide to use multi-bodies, then you should be at least able to articulate *why* you have chosen to do so in a way that does not sound like you are making excuses for careless work.

Appropriate uses for multi-body modeling include (but are not limited to):

- As an intermediate step on the way to a single-body solid.
- As multiple or inserted bodies for reference (reference bodies may be deleted at the bottom of the FeatureManager).
- As over-molded parts.
- As parts that need to be assembled into a single, smooth shape, such as a computer mouse or an automobile body where the shape is impossible (or at least far more difficult) if done in-context.
- When the end shape of the finished product is known, but the part breaks due to manufacturing methods, and materials have not been decided yet, multi-body techniques can save a lot of time compared to modeling an assembly.
- As captive fasteners and purchased inseparable subassemblies.
- When SolidWorks weldments result in a single multi-body part.

- When features require tool bodies, such as the Indent feature.
- When the Mold Tools result in a single multi-body part representing the plastic part and the major mold components.

If you are administering a SolidWorks installation of multiple users, then you may be looking for a "bright line" test to clearly define for users which types of multi-body modeling are allowable and which are not. So many possibilities exist that it is difficult to say definitively what really *should not* be done, but here is a short list that you can modify for your needs:

- Do not use multi-body modeling simply to avoid making an assembly — you must be able to cite a specific reason for using the technique.
- Do not leave a part in a multi-body state that should be joined together into a single body.
- Hiding a body is sometimes appropriate, and deleting a body is sometimes appropriate — understand the difference.

Okay, the lecture is over. The message that you should take from all this is not to use multi-body techniques just because you can; use them only when you have a solid reason to do so. I do not say this because I am the *design police*; I say this because it is the criterion that I use for my own modeling, what I would like to see in models that I inherit from other SolidWorks users, and a philosophy that will serve you well if you are conscientious about it.

Multi-body modeling is powerful, and for complex parts can even increase rebuild speed compared with single body modeling or assembly modeling. You can develop and use many powerful techniques based on multi-bodies, but as I mentioned earlier, as can be true for in-context and virtual components, sometimes you pay a price for the short cut.

Understanding Multi-body Techniques

To complicate the issue somewhat, nearly all surface modeling is also multi-body modeling. In this chapter, I am referring to solids unless I specifically state otherwise. Still, most solid body techniques have some sort of equivalent in surface body techniques. Surface bodies are discussed in Chapter 27.

Multi-body techniques cover a wide range of functionality, and as soon as someone creates a list of what you can do with them, someone else will come up with a new technique. Still, here is a short list of techniques where multi-body functionality makes things either easier or simply possible:

- Complex shapes across multiple parts
- Tool bodies/Boolean operations
- Local operations
- Patterning

- Simplifying very complex parts
- As a bridge between solids
- Undetermined manufacturing methods
- Manipulating imported geometry

In the remainder of this chapter, I illustrate each technique using an example model, and discuss the positives and negatives of each technique. As you get deeper into SolidWorks, you will find that the more complex functions tend to come at a price. A given feature might be the only way to accomplish a particular task, but it rebuilds slowly, might only work in special conditions, might crash from time to time, or might not make sense to other users if they have to edit it.

Creating complex shapes across bodies

When creating a part such as a computer mouse, you encounter complex shapes that span several parts. It makes the most sense to model the entire shape as a single part, and then to break it up into separate bodies, making parts from the bodies, adding detail to individual piece parts, and then bringing the parts back together as an assembly.

Cross-Reference

This method also uses the Master Model techniques discussed in Chapter 28. ◼

A part that uses this technique is shown in Figure 26.1. This part seems to contradict what I said earlier about not being able to use exploded views with multi-body parts, but this part uses the Move/Copy Bodies feature to move bodies within the part. This function remains in the part as a history-based feature in the FeatureManager and is much more labor-intensive to create than an assembly exploded view because each body is moved by a separate feature.

The part shown in Figure 26.1 is not complete, but the starting point for each part has been formed. This part was created from surface features that are discussed in detail in Chapter 27. The part is named `Chapter 26 – Mouse Base Part.sldprt` and is located on the CD-ROM. You may find it interesting to open the part to see how it has been modeled.

From here, each body is saved out to individual parts to complete the detailing, and then the parts are brought back together to create an assembly. The separate bodies in this case were created using the Split feature, which enables you to use surfaces or sketches to split a single body into multiple bodies. This is described in more detail later in this chapter.
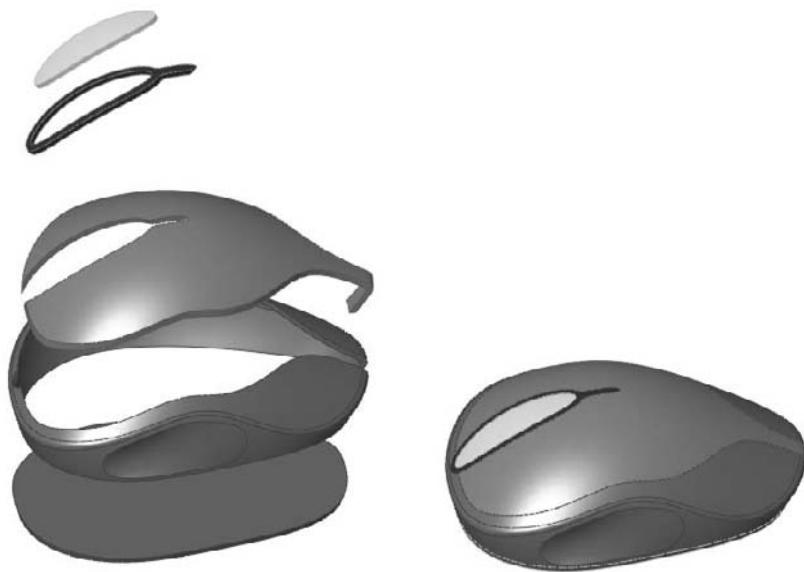
The entire process for creating a finished assembly of finished parts is detailed in Figure 26.2. This flow chart shows conceptually how the overall shape created as a single part has moved from a single part/single body to a single part/multiple body to individual parts to an assembly of individual parts.
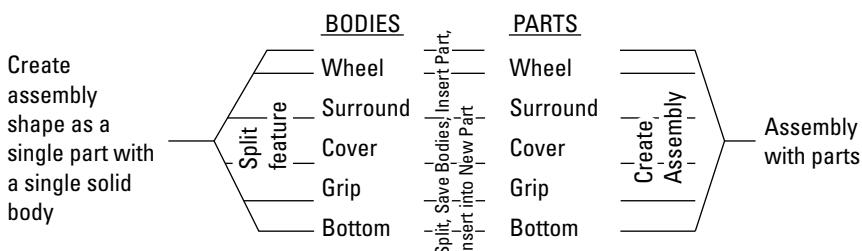
FIGURE 26.1

A multi-body part with a complex shape across bodies

FIGURE 26.2

A Master Model workflow
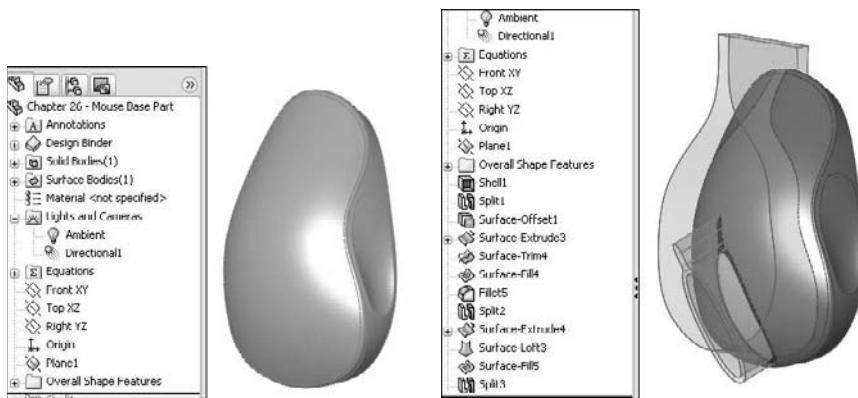
The image to the left in Figure 26.3 depicts how this part was modeled. The first step was to create the shape as a single body within the part. As shown in the FeatureManager, this is all contained inside the Overall Shape Features folder. This folder is presented here as a *black box* because surface features were used to create the part. It really doesn't matter at this point *how* the part was created, and these features are not discussed until Chapter 27.

The image to the right in Figure 26.3 shows transparent surface bodies that were used to split the model into separate bodies using the Split features shown in the tree. Using this technique, you can create the overall shape as a single piece and then split it into separate parts. It is also possible to apply this technique in the context of an assembly, but this method is far more direct.

FIGURE 26.3

Splitting the part into bodies

To go from the multi-body part created here to a set of separate parts uses a Master Model function, which is described in Chapter 28.

Using Tool bodies and Boolean operations

Some features require multiple bodies within a part, such as the Indent and Combine features, among others. Using one body to create a shape in another is a common use for bodies within a part.

Indent feature
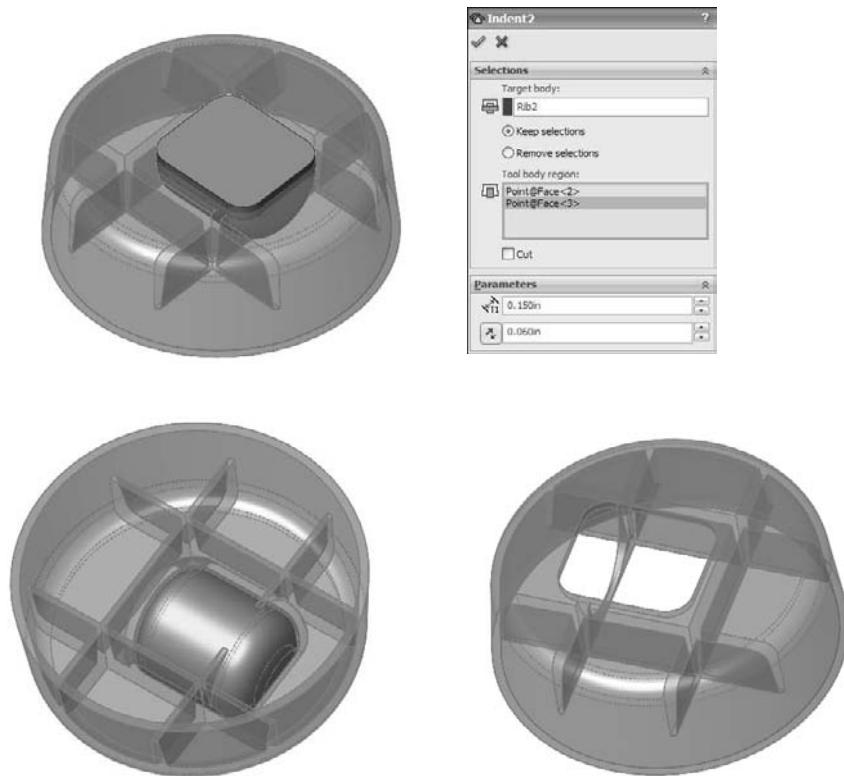
The Indent feature is covered briefly in Chapter 7 before multi-bodies are introduced, so it is fitting that I revisit it here so that you can better understand the multi-body aspect of its use. The Indent feature *indents* the *target* body with the *tool* body. It can also use another part in the context of an assembly as the tool. The indentation can exactly fit the form of the tool, or there can be a gap around the tool. You can also control the thickness of the material around the indent. A further option is to simply cut the target with the tool instead of indenting.

Figure 26.4 shows the target part as transparent, and the tool as opaque, before and after the Indent feature has been applied. The Indent PropertyManager is also shown.

FIGURE 26.4

The Indent feature using a tool body

On the CD-ROM
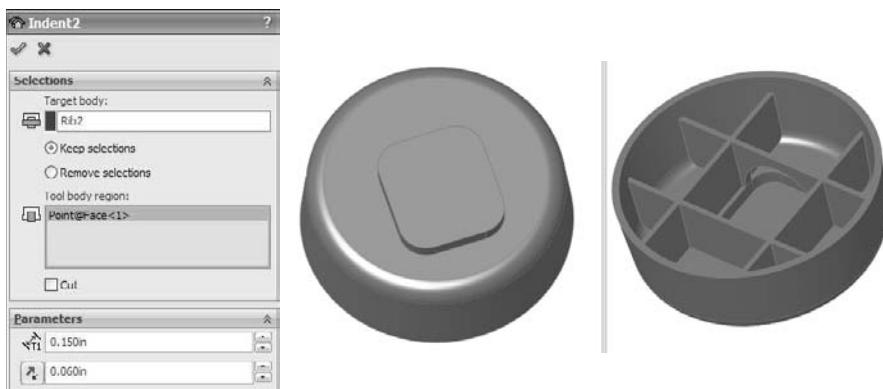
To take a closer look at this part and the Indent feature, look at the part on the CD-ROM named `Chapter 26 – Indent Part.sldprt.` ∎

The Indent feature can be problematic if it breaks into multiple areas as it does in this part, due to the ribbing on the underside of the target body. Notice that in the PropertyManager in Figure 26.4, two selections were made in the Tool body region selection box. The tool body is selected on either side of the rib that bisects the tool. This concept is not very intuitive, and you may have to play with the part and the options to understand what it is doing.

The Keep selections and Remove selections options are equally unintuitive, but they determine which side of the target body is indented. For example, if the part of the tool body that is outside of the target body (flat side) were selected instead of the two inside regions, then the resulting part would look as it does in Figure 26.5, where the tool body has been hidden. You can achieve the same result by toggling the Keep selections and Remove selections options. These options exist because sometimes it is difficult or impossible to select the correct areas of a body that is embedded in another body.

FIGURE 26.5

Using the Keep and Remove selections options

Tip

Also notice that toggling the Keep and Remove selections options means that only one region of the tool body needs to be selected to create the original result shown in Figure 26.4. ∎

Move/Copy Bodies and Combine features
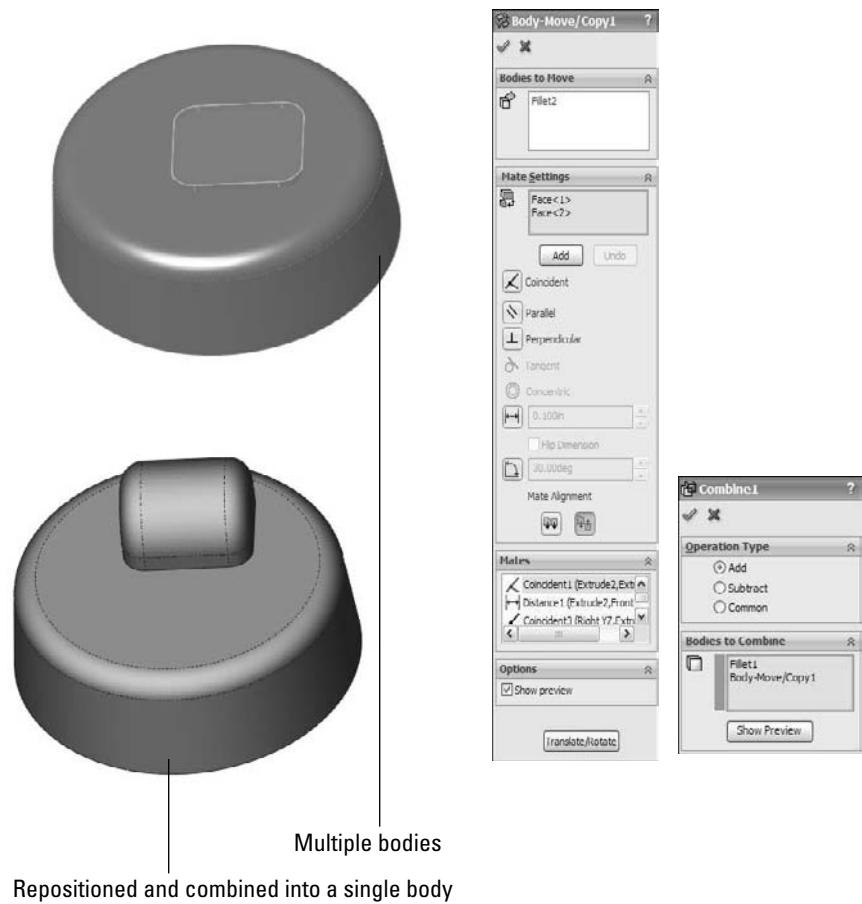
The Move/Copy Bodies and Combine features can be demonstrated using the same part. The body that was used in the previous example to indent the main body is moved and then added to the main body in this example.

Figure 26.6 shows the starting and ending points of the process, as well as the PropertyManagers of the two features used to get from one point to the other. Keep in mind that both the Move/Copy Bodies and the Combine features are history-based features listed in the FeatureManager.

FIGURE 26.6

Using the Move/Copy Bodies and Combine features

Multiple bodies

Repositioned and combined into a single body

In this case, the Move/Copy Bodies feature uses mates. These mates enable you to locate bodies in a way similar to the way they are used in assemblies. One important difference is that with bodies, you must use the actual body geometry of the body that is moving; you cannot use reference geometry such as planes. By clicking the Translate/Rotate button at the bottom of the PropertyManager, you can also position bodies using distances and angles.

In the Combine PropertyManager, you will notice that common Boolean operations, such as union (add), difference (subtract), and intersection (common), are available through this interface.
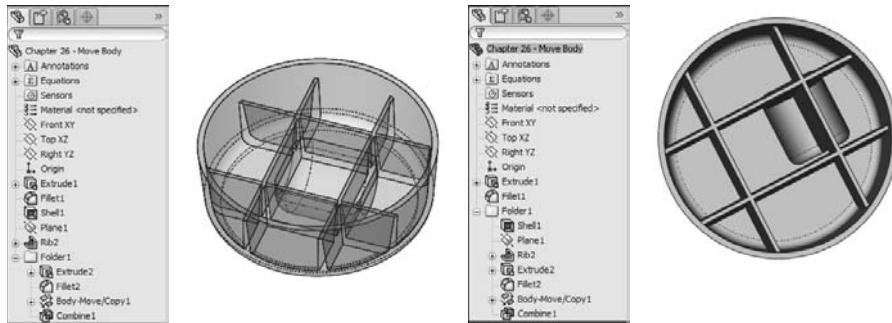
Tip

You can use an interesting technique in this part. The features creating the smaller tool body and the Move/ Copy Bodies and Combine features can be put together into a folder, and the folder itself reordered before the Shell feature. This means that the combined body is also shelled out, and the rib goes down inside of it. This produces an odd error message and unexpectedly places several features into the folder, but it does work.

You may want to open this part in SolidWorks to see exactly how all this was done instead of relying on the figure illustrations. The part used for Figure 26.7 is on the CD-ROM and is named `Chapter 26 – Move Body. sldprt.` ■

FIGURE 26.7

Reordering features

Using local operations

If you have ever had a modeling situation where you needed to shell out a portion of a part but not the entire part, or you had a fillet that would work if only certain geometry were not there, then you may have been able to benefit from multi-body techniques to accomplish these tasks.

Flex feature
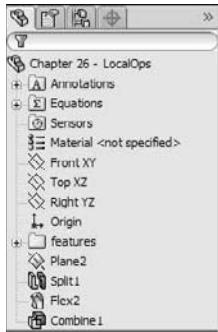
The part shown in Figure 26.8 first appears in Chapter 7, where I demonstrate the Flex feature. This is a rubber plug for an electronic device. In order to make one side of the part flex without flexing the other side, multiple bodies were used. The part was split into two bodies using the Split feature and a plane. One side of the part was then twisted, and the two bodies were combined back together. The Features folder contains the features that were used to build the original part geometry, which could just as easily have been either native or imported.

Splitting a part to perform a local operation

Shell feature

The Shell feature hollows out one solid body at a time. If there are multiple solid bodies, then you must select one to be shelled. Any face of the solid that you select will be removed during the shelling. You can select a body without selecting a face by using the small Solid Body selection box under the larger Faces to Remove selection box in the PropertyManager. If you do not select any faces to be removed, then the body will be hollowed out with no external indication that the part is hollow unless you view it in section view, transparency view, or wireframe view. Single or multiple faces can be removed. This feature works by offsetting the faces of the outside of the model, and the feature may fail if this causes problems with the internal geometry.
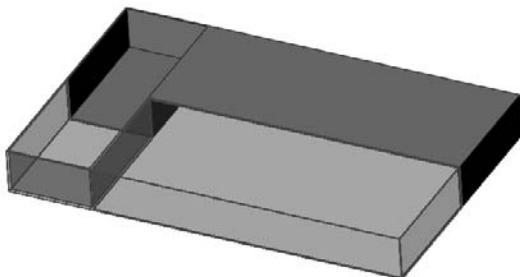
The Multi-thickness Shell option enables you to select faces that will have a different thickness from the overall shell thickness. This is one method that you can sometimes use to limit the scope of the Shell feature to a certain area of a body, but it is somewhat limited. Faces with different thicknesses cannot be tangent to one another.

Because the Shell feature only works on one body at a time, splitting a part into multiple bodies can be an effective way to limit the scope of the feature. The part shown in Figure 26.9 has been split in half, and one-half has been made transparent for visualization purposes; as a result, you can see that the part is shelled on the bottom on one end and on the top on the other end. The Shell feature has no option for doing this with existing geometry. The only ways that you can do this are either through feature order or by using multi-bodies. You can find the part shown in Figure 26.9 on the CD-ROM with the name `Chapter 26 – LocalOps Shell.sldprt`.

To shell the part this way with feature order, you create one block and shell it, and then create the other block and shell that. In order for this technique to work, the second shell needs to be as big as, or bigger than, the first shell. If it is smaller, then it will (or may) hollow out areas that are not intended to be hollow.

FIGURE 26.9

Shelling locally

To shell the part with multi-bodies, you can use two methods. One method is to build the first block, and then build the second block but turn off the Merge option. This creates bodies that are side by side. You then shell one block on the bottom and the other on the top. To avoid a double-thickness wall between them, the end face can be removed along with either the top or bottom face. If you edit the part, then you may notice that one of the Shell features has two faces removed.

The second method is to build a single block, then split it using a sketch line, a plane, or a surface, and then proceed in the same way as the first method.

Patterning

Patterns of bodies are fast, powerful, and commonly used alternatives to patterning features. Chapter 8 discusses feature patterns and mirroring, and examines, at least in part, how different types of patterns affect model rebuild speeds. When appropriate, patterning bodies can also be a big rebuild time-saver. When patterning a body, none of the parametrics or intelligence is patterned with it, but you must pattern the entire body. Another odd thing about patterning bodies in SolidWorks is that there is no option to join the bodies either to one another or to a main body. This requires an extra step that involves adding a Combine feature. Mirroring is the same, except that it has an option to merge bodies, but it only merges the original body to the mirrored body. It will not merge either the original or the mirrored body to a central main body.

In this example, an imported part has a "feature" that needs to be reused around the part. The technique used here is to split away the feature as a separate body and then pattern the body around the part and join it all back together. This function can be used with native geometry as well as imported. This process is shown in Figure 26.10. This function does use a simple planar surface. A plane could have been used to split off the body to be patterned, but the plane would have also split off a part of the globe at the top, so a planar surface (which can be limited in extent where a plane can not) was used.
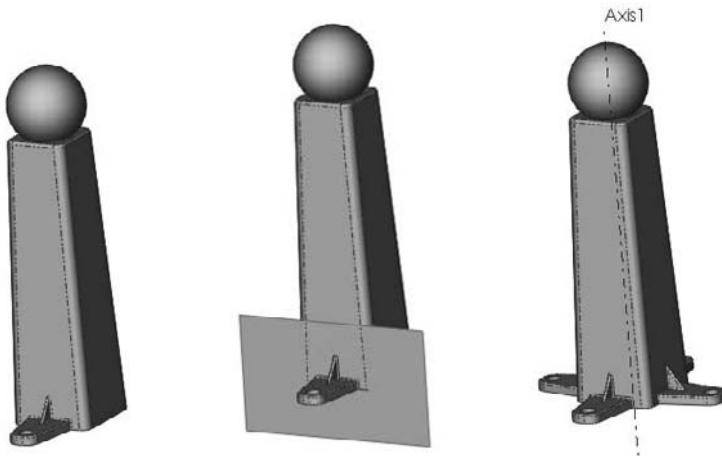
The image on the left in Figure 26.10 is the raw imported part. The middle image shows a planar surface created on the face of the part, where the planar surface has been used with the Split feature to cut the leg off the part. The image on the right shows the split leg patterned around an axis that was created from the intersection of two planes.

If you would like to practice with this part, it is on the CD-ROM for Chapter 26; the imported Parasolid file is named `Chapter 26 – Pattern import.x_t`.

Splitting away a body and patterning it

Performance

In some situations, patterning bodies is a performance advantage, and in some situations it is not. You get an advantage from patterning bodies when the geometry used to create the pattern seed is complex, uses many features, or does not work well or at all for a feature pattern.
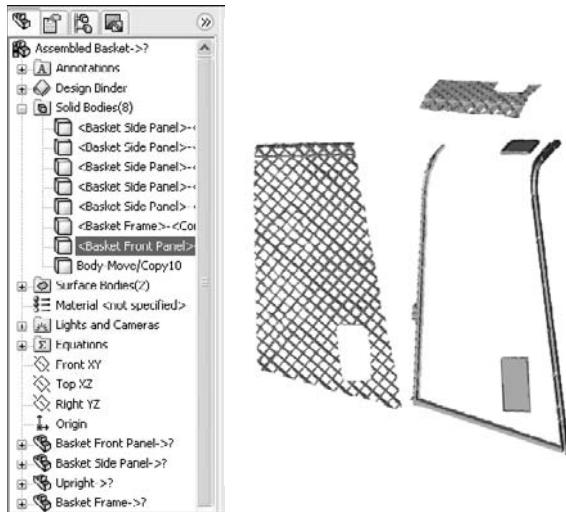
On the other hand, if you repeat the experiments from Chapter 8 using a small body with a hole in it instead of patterning a hole feature, you find that the body pattern is far slower than the feature patterning because of the necessary step of combining bodies. ■

Simplifying very complex parts

Certain types of parts work better when they're built in sections as separate parts than when they are built as a single feature tree. For very complex parts with a lot of features, this sometimes makes sense from the point of view of segmenting the rebuild times for parts with hundreds of features. The example used to demonstrate this technique is a large plastic part built entirely from ribs, and making use of literally hundreds of solid bodies, and is shown in Figure 26.11 and Figure 26.12.
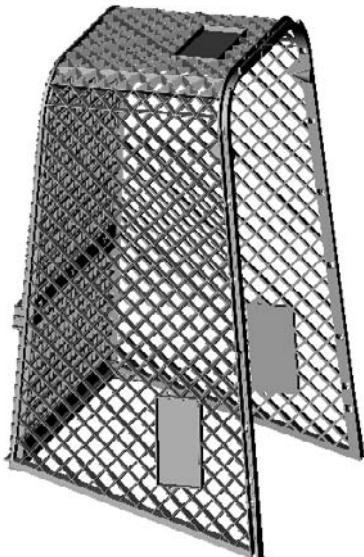
FIGURE 26.11

A complex model created as separate parts and brought together as bodies in a single part

FIGURE 26.12

Bodies all joined together as a single body

This part is molded using tooling pulls in five directions. Two of these directions are symmetrical, and the core block pulls in a single direction; as a result, in the end, the modeling has to account for three directions.

The rebuild time for a model like this can easily reach several minutes, and the feature count can be in the hundreds, or in this case, well over one thousand. To minimize the rebuild time, a different workflow was established for this part. First, the major inside and outside faces were created with surfaces. Next, the surfaces were saved into several other parts (using Master Model techniques that are discussed in Chapter 28). Each of these parts represents the part geometry that will pull in a particular direction from the mold. Enough information exists in the Master Model to align the features in each part.

The ribs on this part were created by making a single extrusion (the Rib feature could not be used because there was no geometry to serve as a boundary for the ribs), and then the extrusion was patterned and the pattern was mirrored. After all the ribs were created, they had to be shaped, and so the surfaces from the Master Model were used to cut the ribs to shape.

The ribs could not be extruded with a draft or with fillets because the outer and inner surfaces were non-planar. The draft had to be built as a Parting Line draft for the same reason, and the fillets had to be applied after the draft. Further, draft and fillets can only be applied to a single body at a time; as a result, a separate draft feature and a separate fillet feature had to be applied to each body, and each rib was a separate body. Once the draft and fillets were applied, the bodies were joined into a single body.

I recognize that this description of how I made the part is a lot to follow. The point is not to show in detail how the parts were built, but to demonstrate how you can get to a part with 1,200 features or more. It is precisely on parts with this level of complexity that you need to think about modeling the part in this modular fashion — build each part separately and bring each separate section of the part together as individual bodies.

Figure 26.11 shows two of the separate pull direction parts being separated from one another in the same way that the mouse part was shown exploded in the previous example. Here the frame is also modeled as a separate part, again because it was not so intimately related to the other parts and was easily separated out.

Once this was complete for each direction, the separate parts were put together as bodies into a single part and again joined together using the Combine feature. Having all those features in separate parts enables you to segment the rebuild time. This is the opposite of building all the parts of an assembly in a single part, where you are simply compounding your rebuild time. Figure 26.12 shows bodies joined together as a single body.

This is probably a technique that you will not use very often, but when you do, it can save you a lot of rebuild time. I use it whenever I have a model that takes more than 20 to 30 seconds to rebuild and I know that I am going to be working on it a lot; it must also lend itself to segmenting in the way that this one did, with easily definable areas.
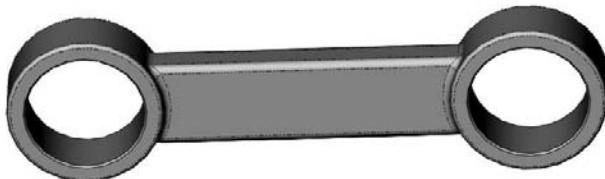
Bridging between solids

Often when modeling, you "build what you know" and "fill in as you go." An example of this would be modeling a duct between end connections that are well defined. The duct in between is defined only by the ends, which must exist first. Another example is a connecting rod where you know the diameter of each end and the distance between the ends, and the connection between them is of secondary importance.

Figure 26.13 shows a connecting rod made in this way. In this case, the bearing seat at one end was created, and the other end was created by copying the body of the first one. From there, the link between bearing seats was created, which joined the separate bodies together into a single solid body.

Connecting disjoint bodies

This part contains some interesting features. First is the Thin Feature extrude that is used to make the first bearing seat, which is combined with a Mid-plane extrude to make it symmetrical at the same time. Then comes the Move/Copy Bodies feature, which copies the body in the same way that the feature in previous examples has moved bodies. Next is the use of the Extrude From option, which extrudes from a face, and then the use of the end condition Up To Next, which ends the feature neatly. The part also incorporates fillets that use faces and features to form the selection.

If you are not familiar with these options, I recommend that you open up the part from the CD-ROM and have a look at it. It is a simple part that takes advantage of nice but simple productivity-enhancing options that have been available for some time in the SolidWorks software. The part filename is `Chapter 26 – Bridge.sldprt`.

By default, Solid features have the Merge option selected, and they automatically combine with any bodies that they touch. At the same time, they do not display errors if the Merge option is selected but the new body does not touch any existing bodies.

Modeling for undetermined manufacturing methods

Sometimes you must start a design before you know exactly how the product will be manufactured. This is an example of where the geometry of the finished product exists first and is then broken up into manufacturable parts. The initial model, shown in the image at the top in Figure 26.14,
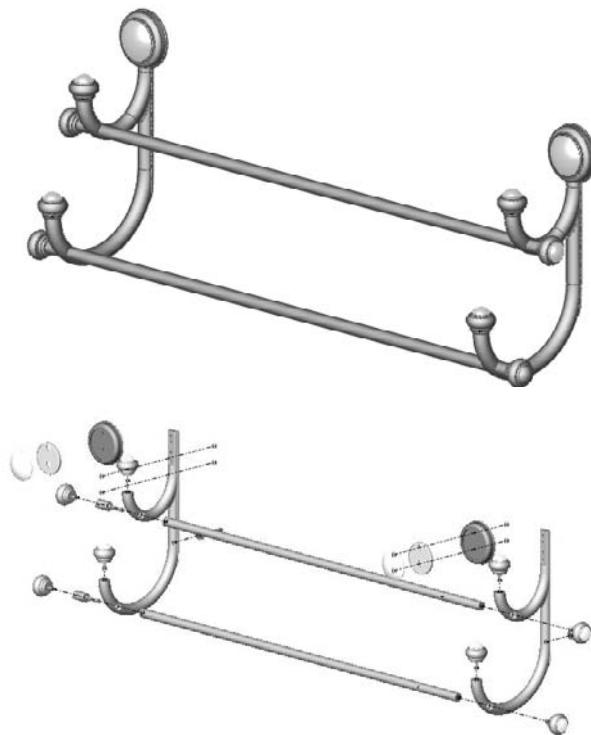
is created as a single part as a result of input from marketing, but when it comes time for manufacturing input, the part count and processes keep changing. Where the parts break from one another keeps changing as well. When that kind of change is happening, having the parts created as individual parts is a big liability because it is difficult to change. Changing which bodies are merged together is much easier.

A towel rack, modeled as a single part and broken into individual parts in an assembly

It is worth mentioning two potential difficulties that you may run into with methods like this. The first is that if you have people making drawings from parts that have been derived from bodies in a single part, then they are forced into the Reference Dimension scheme of dimensioning parts because the feature dimensions do not survive being moved from the multi-body part. This may or may not be an issue, depending on how the people doing the drawings are accustomed to working.

The second potential issue is what you do in situations where there are multiple instances of a part that has been modeled this way. Notice in the towel rack in Figure 26.14 that there are several finials, spacers, rails, and other parts that are duplicated. This requires some manual assembly modeling. You can make the assembly directly from the multi-body part, but if you need to make multiple instances of particular parts, you need to do this manually rather than automatically.

Creating Multi-bodies

In the first section in this chapter, I raised the questions of *if* or *when* multi-bodies should be used, and in the second section, I raised the question of *why* multi-bodies should be used. In this section, I simply ask, or rather answer, *how* they should be used.

Using disjoint sketches

The easiest way to create multiple bodies is to simply create what SolidWorks classifies as multiple disjoint closed contours. What that means is simply two circles or rectangles that do not touch or overlap. If these are created in the same sketch, then when extruded, they will create as many bodies as there are closed loops in the sketch.

If the part has an existing solid, then creating a sketch that does not touch the solid can also create a separate body. You cannot make Multiple Thin features in a single sketch. This is presumably because the interface has no way to identify different thickness directions for different open profiles. This holds true whether or not the Multiple Thin features create multiple bodies.

If a solid feature other than a mirror or pattern touches a solid body, then the new and the old bodies will be merged into a single body.

Turning off the Merge Result option

You can prevent a feature from automatically combining with other bodies by deselecting the Merge Result option. This holds true between features, but not across all bodies in a part. For example, if an extrude feature uses the Merge result option, all the bodies that it touches become merged together, but if the original extrude feature does not touch a body, it will not be merged. This option is shown in Figure 26.15 and is found on all features that create new solid bodies except for the Patterns, Rib, and Move/Copy Bodies features.

FIGURE 26.15

The Merge result option
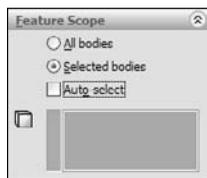
Applying the Feature Scope to bodies

The Feature Scope used for multi-bodies is not the same as the Feature Scope used for assembly features, but it does function in a similar way. In assemblies, the Feature Scope identifies which parts are affected by the current assembly feature. In parts, it only applies to bodies and can be used for features that add material as well as features that remove material (assembly features can only remove material).

The Feature Scope is a way to make the Merge result option more selective. The Merge result option by default does not discriminate; it causes the feature to merge with any other solid body that it touches. However, the Feature Scope enables the user to select which bodies to merge with or otherwise affect. Feature Scope also applies to additional feature types such as cuts. The Feature Scope becomes available in the PropertyManager whenever there are multiple bodies in the part and an eligible feature is used. The Feature Scope panel is shown in Figure 26.16.

FIGURE 26.16

The Feature Scope panel

The default setting for the Feature Scope is to use the Selected bodies option with the Auto-select option. The All bodies option is essentially the same as using the Merge result option. When the Selected bodies option is selected and Auto-select is unselected, as is shown in Figure 26.16, you must select bodies for the current feature to affect them. New bodies that are added to the model are not automatically added to the list; you need to manually edit the feature and add additional bodies to the list as appropriate.

Rib feature

The Rib feature is hypersensitive to changes to the number of bodies. A rib only automatically merges with a body if it is the only body in the part. If a rib is created in a single body part and then the model is rolled back and an additional body is created before the rib, then the Rib feature will fail when the tree is unrolled. The error that this causes reads, "The rib is not bounded properly. The extension of the rib does not intersect the part model." Technically speaking, this is true, but like other SolidWorks error messages, that does not make it helpful. The cause of the error is that suddenly the body that the rib is supposed to merge with is no longer identified. The Rib feature does not have an Auto-select option. You can fix this problem by going to the Feature Scope in the Rib PropertyManager to select a body for it to merge with. The Rib feature does not use the

normal Feature Scope, because the Feature Scope is intended to select multiple bodies, and the Rib feature requires a single body. There is a simple Selected Body selection box in the Rib PropertyManager.

After you have selected the target body, deleting the new bodies that caused the problem in the first place, thankfully, does not make the problem reoccur. However, if the body that the rib is merged with is split using the Split feature, then that *does* cause a problem. As a result, the two things that cause the Rib feature problem are rolling back and either adding bodies or splitting the body to be ribbed.

Caution

As you encounter more specialized situations with multi-bodies and dependencies, you may notice more quirks in the SolidWorks internal body management. The next section on managing bodies addresses some of these quirks directly. ■

Delete Solid/Surface feature

If you have created many ribs in a casting or plastic part, then it may be tedious to go through and repair them all every time the body count changes. This sort of thing happens even if the other body is just a reference body or an unused leftover.

In cases like this, you can use the Delete Solid/Surface feature. This is alternatively called Delete Body, depending on where you look. Delete Solid/Surface removes the body from the body folder (discussed in the section on body folders). This is a *history-based delete*, which means that before the Delete Body feature in the tree, the body exists, and after the Delete Body feature in the tree, the body does not exist. This feature has no effect on file size, because the data for the body must still exist, and it has little, if any, effect on rebuild time. What is happening is that the body is still there; you just cannot see it and have no access to it.

Delete Solid/Surface is often used for other purposes as well, primarily to clean up a model at the end of the tree. The reasoning is that multiple bodies in a part confuse people. My recommendation here is to remove bodies if they are getting in the way, either for a hyper-sensitive feature like the Rib feature or if they are causing visualization problems.

Creating multi-bodies with the Cut feature
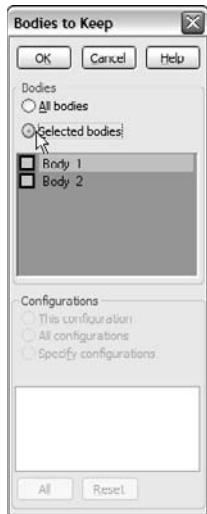
A Cut feature may create multi-bodies, either intentionally or unintentionally. When it does happen, the Bodies to Keep dialog box appears, enabling you to select which bodies you intend to keep. The Bodies to Keep dialog box is shown in Figure 26.17. This dialog box was formerly called Resolve Ambiguity, which was not as descriptive as Bodies to Keep.

Notice that the Bodies to Keep settings are also configurable; therefore, different bodies can be kept in different configurations, which is very useful.

FIGURE 26.17

The Bodies to Keep dialog box

Managing bodies with Split feature

The Split feature has essentially three functions:

- To split a single solid body into multiple solid bodies using planes, sketches, or surface bodies
- To save individual solid bodies out to individual part files
- To reassemble individual part files that are saved out into an assembly where the parts are all positioned in the same relative position as their corresponding bodies

The last two functions of the Split feature are addressed in Chapter 28. The part of the Split feature that concerns this chapter is the first function mentioned, which is splitting a single solid body into multiple bodies using a sketch, a plane, or a surface body.

The Split feature cannot be used to split surface bodies. In fact, nothing in SolidWorks can split surface bodies. Only solid bodies can be split. Surface bodies can only be trimmed, so the effective workaround for not being able to split surface bodies is to copy the body, then trim and keep one side of the copy and the other side of the original. This seems like a functionality oversight, and would make a great enhancement request if you have ever come across the need for this functionality.
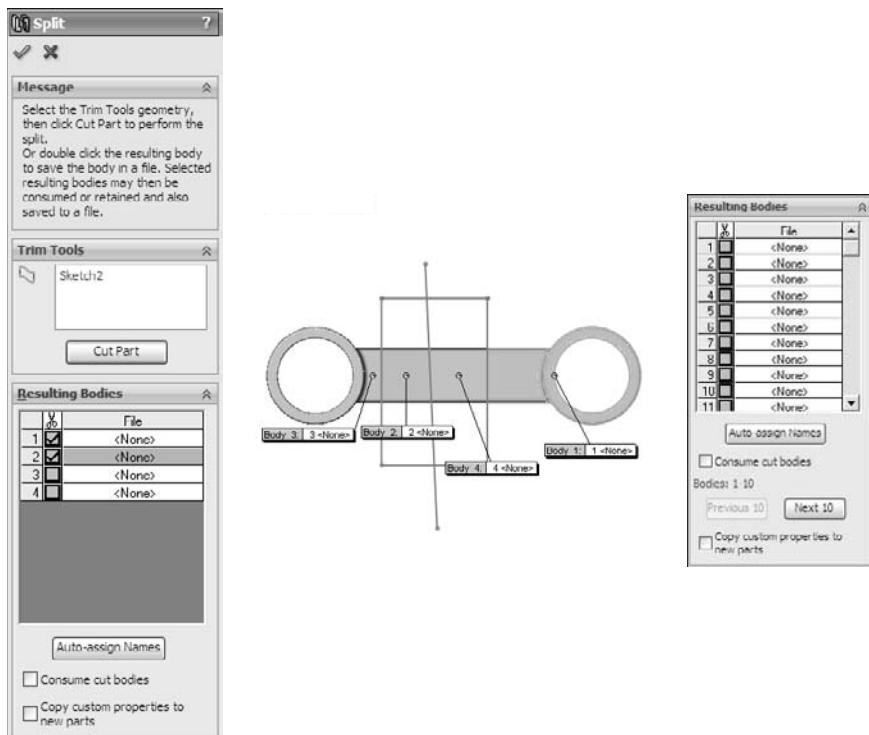
Splitting with a sketch

When you are using a sketch to split a single solid body into multiple bodies, the Split process works like this:

1. **Create a sketch with an open or closed loop; even a mixture of open and closed profiles will work.** If it is open, then the endpoints have to either be on an exterior edge or hanging off into space; they cannot actually be inside the boundaries of the solid.

2. **Initiate the Split feature from the Features toolbar or from the menus by choosing Insert ⇨ Features ⇨ Split.** You can do this with the sketch active, with the sketch inactive but selected, or with nothing selected at all.

3. **Click the Cut Part button.** This does not actually cut anything; it only previews the split. When this is done, the resulting bodies appear in the window below and callout flags are placed on the part in the graphics window. These flags are often useless because they tend to point to the borders between two different bodies in such a way that it is completely ambiguous as to which body they are indicating. However, in the example shown in Figure 26.18, the result is very clear.

FIGURE 26.18

Using the Split feature

Check marks next to the body in the list indicate that the body will be split out. The lack of a check mark does not necessarily mean anything. For example, in Figure 26.18, notice that two boxes are checked, but this will result in a total of four bodies. If only Body 1 were selected, then the result would be only two bodies.

The callout flags and the bodies list where <None> is shown are looking for a path and filename to save the body out to a file. Again, this functionality is covered in Chapter 28 with the Master Model information.

Clicking the Save All Bodies button simply puts check marks in all the boxes. If the Resulting Bodies box contains more than ten bodies, then the interface changes slightly, as shown in the image to the right in Figure 26.18. The Consume cut bodies option removes, or consumes, any of the bodies that have a check mark.

Splitting with a plane

Splitting with a plane provides the same type of results, and uses the same options, as splitting with a sketch. However, you never have to worry about the plane being extended far enough, because the cut is made from the infinite extension of the plane. The only thing you have to worry about with a plane is whether it intersects the part.

Splitting with a surface body

Surface bodies are used to split solid bodies for a couple of reasons. In the part shown in Figure 26.10, a surface body was used to make the split instead of a sketch or a plane, because both of those entities split everything in an infinite distance either normal to the sketch plane or in the selected plane. A surface body only splits to the extents of the splitting surface body. If you look closely at the part, you will notice a plane or sketch would lop off one side of the sphere on top of the object, but the small planar surface is limited enough in size to split what is necessary.

Another advantage to using a surface body is that it is not limited to a two-dimensional cut. The surface itself can be any type of surface, such as planar, extruded, revolved, lofted, or imported. Taking this a step further, the surface is not limited to being a single face, or a body resulting from a single feature; it could be made from several features that are put together as long as it is a single body and all the outer edges of the surface body are outside the solid body. If you examine the mouse part shown in Figure 26.1, you will notice that it has splits made from multi-feature surface bodies.

I mention splitting with surface bodies here because this is where I discuss the Split function, even though I haven't covered the surfacing functions yet. It may be useful to read parts of this book out of order; given how the topics interrelate, it is impossible to order them in such a way that some sections will not refer to a topic that has not yet been covered.

Cross-Reference

For more information about surface bodies, see Chapter 27. ∎

Adding bodies using the Insert Part feature
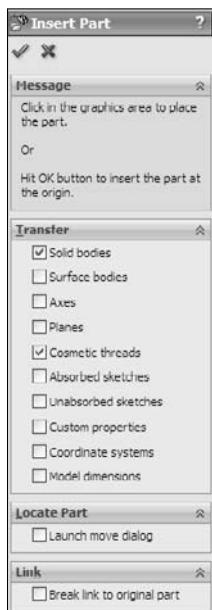
The Insert Part button can be found on the Features toolbar, or you can access this feature by choosing Insert ➪ Part from the menus.

Insert Part enables you to insert one part into another part. When inserting the part, you have the option to insert solid bodies, axes, planes, cosmetic threads, surface bodies, and several other types of entities, including sketches and features. The PropertyManager interface for the Insert Part feature is shown in Figure 26.19.

FIGURE 26.19

The Insert Part PropertyManager

This feature has two major functions: inserting a body as the starting point for a new part, and inserting a body to be used as a tool to modify an existing part. Notice that the basket part shown in Figure 26.11 and Figure 26.12 also uses Insert Part to put together bodies to form a finished part.

When you use Insert Part, there is no Insert Part feature that becomes part of the tree. Instead, a part icon is shown with the name of the part being inserted as a feature.

Also notice in Figure 26.19 that the Launch move dialog option appears near the bottom, and is selected by default. This option launches the Move dialog box after you insert the part. This Move feature is the same as the Move/Copy Bodies feature, with the same options (translate or rotate by distance or angles, or use assembly-like mates to position bodies).

Insert Part is used in many situations, some of which are covered in Chapters 11 and 28 in the sections on skeleton techniques and Master Model.

Working with Secondary operations

One commonly used technique has to do with secondary operations. For example, you may have designed a casting that needs several machining operations after it comes from the foundry. The foundry needs a drawing to produce the raw casting, and the machine shop needs a different drawing to tap holes, spot face areas, trim flash, and so on.

You can use configurations to do this by using Insert Part is another way. This has nothing to do with multiple body techniques, but this is the only place where Insert Part is covered in much detail. One of the advantages of using Insert Part is that you no longer carry around the overhead of all the features in the parent part. It is as if the inserted part were imported. The configurations method forces you to carry around much more feature overhead. Of course, the downside is that now there is an additional file to manage, but this can be an advantage because many companies assign different part numbers to parts before and after secondary operations.

Starting point

Looking back to the mouse shown in Figure 26.1, the main part has been split into several bodies. You can use Insert Part to insert the whole mouse into a new part where all the bodies except one are deleted, and then the remaining body serves as the starting point for a new part. Many additional features are needed on all the bodies that make up the mouse, such as assembly features, cosmetic features, functional features, and manufacturing features.

Managing Bodies

Managing bodies in SolidWorks is not as clean a task as managing parts in an assembly. As you work with bodies, you may discover some surprises in how bodies are managed. This section prepares you for the challenges involved in managing bodies in SolidWorks.
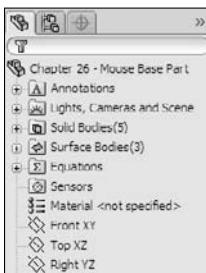
Using Body folders

The top of the FeatureManager includes a pair of folders: one called Solid Bodies and the other called Surface Bodies. These folders are only there if you have solids or surfaces in the model, and they reflect the state of the model at the current position of the Rollback bar. As a result, the folders can change and even disappear as you roll the tree back and forth in history. Figure 26.20 shows the top of a FeatureManager that has both solid and surface body folders. Notice that the number in parentheses after the name of the folder shows how many bodies are in that particular folder.

An odd fact about these folders is that you are allowed to rename the folders, but the name changes never remain. If you go back to rename the folder again, the name that you previously assigned will display.

FIGURE 26.20
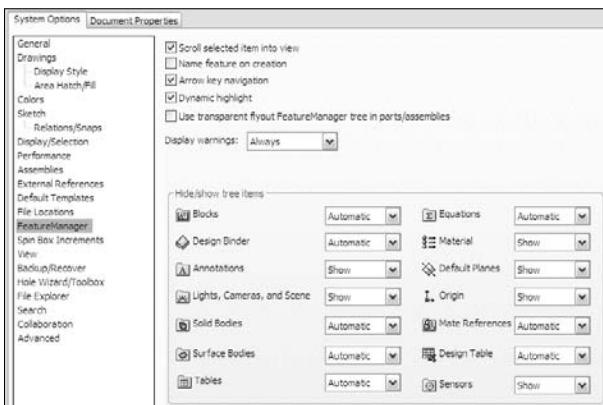
Body folders in the FeatureManager

You may encounter another problem with the display of FeatureManager header items in general when they are set to Automatic display (display only when they contain something). This does not guarantee that the folder is going to display when it should. A more direct way of saying this is that the Automatic setting works incorrectly from time to time. For this reason, I suggest using the Show option to display important folders. Figure 26.21 shows the Options page (Tools ➪ Options) that controls the visibility of folders.

FIGURE 26.21

Control the visibility of FeatureManager items

By right-clicking either of the bodies folders, you can select the Show Feature History option, which shows the features that have combined to create the bodies in an indented list under the body name. This view of the FeatureManager is shown in Figure 26.22. This option is very useful when you are editing or troubleshooting bodies.

Figure 26.22 also shows the other options in the right mouse button (RMB) menu. All the bodies in the folder can be alternately shown or hidden from this menu, as well as deleted. While the Hide or Show state of a body does not create a history-based feature in the tree, the Delete feature does, as discussed previously. The Insert into New Part feature and the Save Bodies feature shown in this menu are discussed in Chapter 28.

You can expand the Display pane in parts to show display information for bodies. In Figure 26.23, the Display pane shows the colors assigned to the solid bodies, as well as the fact that several surface bodies exist but are hidden.

FIGURE 26.22

Using the Show Feature History option
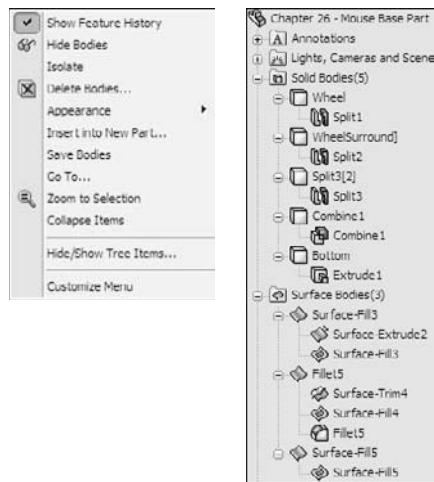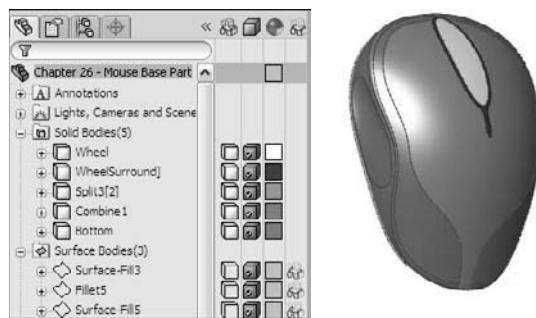
FIGURE 26.23

The Display pane showing information about solid and surface bodies

The folders also make bodies easier to identify, especially when combined with the setting found at Tools➪Options➪Display/Selection➪Dynamic Highlight From Graphics View. This setting quickly turns the body outline red if you move the mouse over the body in the body folder.

Hiding or showing bodies

You can hide or show bodies in one of several ways. I have already described the method of using the bodies folders to hide or show all the bodies at once, but you can also right-click individual bodies in the folders to hide or show them using the RMB menu. Remember that with the context bars, you have the option to use them with the RMB menu as well as with left-click selections. I include all context bar options in the RMB menu generically.

If you can see a body in the graphics area, then you can right-click the body and select Hide under the Body heading. This works for both solids and surfaces. The Display Pane, shown to the right of the FeatureManager in Figure 26.23, can also be used to hide or show bodies, change body transparency and appearance, as well as change the display mode of bodies. Display Pane is a handy tool for visualization options.

When you are hiding or showing bodies from the FeatureManager, but not using the bodies folders and using the features themselves instead, things get a little complicated. If you want to hide or show a solid body, then you can use any feature that is a parent of the body to hide or show the body. For example, you can use the Shell feature in the mouse model to hide or show all the bodies of which it is a parent.

Other facts that you need to know about bodies and their hide or show states are that the Hide or Show feature is both configurable and dependent on the rollback state. As a result, if you hide a body, and then roll back, it may appear again and you will have to hide it. Then, if you roll forward, the state changes again. Also, a body can be hidden in one configuration, and then when you switch configurations, it remains hidden. This makes it rather frustrating to work with bodies. To me, it would be nice if bodies had simple on/off switches.

SolidWorks 2010 adds the Display State functionality from assemblies to multi-body parts. This is extremely handy, and a very fast way to change color, transparency, or display modes for individual bodies. The best way to handle this is to expand the Display Pane and control Hide/Show, Display Mode, Appearance, and Transparency directly from the Display Pane. Figure 26.24 shows the Display Pane in action on a multi-body part.

In Figure 26.24, the FeatureManager panel is split so you can see the configurations and Display State information alongside the bodies folder and display pane. If you are familiar with Display States in assemblies, it is the same as for bodies.

Another change in SolidWorks 2010 that is related to the Display States, and in particular the Appearances, is that you are now able to assign different materials to each body. I'm not a big fan of this change because I think it starts to encroach too much on the realm of the assembly. If different parts are made of different materials, the only situations in which I would model this way would be overmolds, where multiple materials are molded onto one another, and inseparable subassemblies, like purchased components such as screws with captive washers or a circuit board with rivets. If you need to manufacture something that has different materials, you need to have each

material in a separate file for manufacturing purposes. Computer Numerical Control (CNC) software operators generally do not want parts that need to be machined separately given to them as multi-body single parts. To apply materials to bodies, right-click the body in the Solid Bodies folder and select Material.

Using the Display Pane to control multi-body Display States

Caution

Some features exclude bodies if the bodies are hidden when you edit the feature. Be careful of this, and be sure to show all the bodies that are used in a particular function before you edit it. For example, if a body is hidden and you create a new extrude that touches the hidden body, then the new body does not merge with the hidden one even if the Merge option is on. If the hidden body is then shown and you edit the second body, then the bodies will merge upon the closing of the second body. ■

Deleting bodies

I have already mentioned that you can delete bodies using the Delete Solid/Surface feature, and that this feature exists in the tree of the part. This feature used to be called Delete Bodies.

Delete Solid/Surface does not affect file size or rebuild speed. In fact, I find it difficult to come up with examples of when you should use it, other than the situation already mentioned with the Rib feature, or if a throwaway body somehow remains in the part. Some people use this feature to clean up the organization of the tree, which could be useful if there are many bodies in the part. Other users insist on keeping the tree free of extraneous bodies and immediately delete bodies that have been used. To me, this technique replaces one kind of clutter with another, and means that tools that should be available to you (solid or surface bodies) are not available unless you reorder the Delete Body feature down the tree and/or roll back. In any case, this is really a matter of personal working style and not of any great importance.

Renaming bodies

Notice that the bodies that you see in the folders have been named for the last feature that touched a given body. That naming scheme is as good as any, except that it means that the body keeps changing names. If you deliberately rename a body, it will retain the name through future changes. You should follow the same rules of thumb for naming bodies as you do for naming features. It is not necessary to rename every body, but if you will use one body frequently and need to select it from the FeatureManager, renaming it is very useful.

Using multiple Bodies with Sheet Metal, Weldments, and Molds

Various types of special SolidWorks documents require that you work with multi-body data. Those are sheet metal, weldments, and molds. Sheet metal and Weldments are SolidWorks files with special properties, and the molds are simply created from a separate set of specialized Mold Tools.

Introducing multi-body functionality in Weldments

Weldments are a special type of SolidWorks document that require the use of multiple bodies. The Weldment tools are covered in detail in Chapter 31, but I mention them here because they fall under the topic of multiple bodies. Each structural member in a weldment frame is created as a separate solid body. So if you are getting involved in using the Weldment tools in SolidWorks, you will need to also master the multi-body tools.

Introducing multi-body functionality in sheet metal

New in SolidWorks 2010 is the capability to have multiple bodies in sheet metal parts, and put sheet metal parts into weldments. Sheet metal components are frequently used in welded frames, so SolidWorks added the multi-body functionality to sheet metal. Sheet metal is covered in Chapter 29, and the details of multi-body modeling as it relates to sheet metal are covered there. Again, I include the multi-body capabilities of sheet metal in this chapter as an introduction to the idea.

Introducing multi-body functionality in molds

Creating mold geometry using SolidWorks Mold Tools is another multi-body function. Other mold creation software that works within SolidWorks uses assemblies to do this, but SolidWorks has chosen to use bodies. You can also do the same kind of work without using the specialized tools if you choose, and can even work exclusively in solids or in a hybrid solid and surface workflow. Unless you choose to create geometry for various types of molding or casting manufacturing methods using assemblies, you are probably going to need to do it using multi-body parts. SolidWorks Mold Tools are discussed briefly in Chapter 32.

Tutorials: Working with Multi-bodies

This tutorial contains various short examples of multi-body techniques in order from easy to more difficult.

Merging and local operations

This tutorial gives you some experience using the Merge Result option and using features on individual bodies to demonstrate the local operations functionality of multi-body modeling. Try these steps:

1. **Start a new part, and sketch a rectangle centered on the origin on the Top plane.** Size is not important for this exercise.

2. **Extrude the rectangle to roughly one-third of its smaller dimension.**

3. **Open a second sketch on the Top plane.** Hide the first solid body by right-clicking it in either the FeatureManager or the graphics window.

4. **Show the sketch for the first feature, and draw a second rectangle on the far side of the rectangle from the Origin.** Make sure that the second rectangle gets two coincident relations to the first sketch at two corners so that the rectangles are the same width. When the sketch is complete, hide the sketch that was shown.

5. **Extrude the second rectangle to about two-thirds of the depth of the first rectangle.**
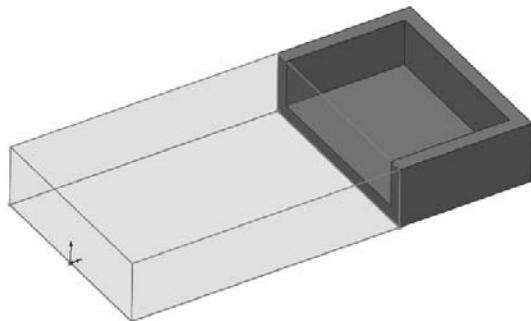
Note

Notice that the Merge option was not changed from the default setting (On) for the second extrude, but because the first extrude was hidden, the second extrude did not merge with it. Be careful of subsequent edits to either of the features if the first body is shown, because this may cause the bodies to merge unexpectedly. In this tutorial, the bodies are later merged intentionally. Ideally, what you should do is deselect the Merge option of the second extrude. ∎

6. **Shell out the second extrusion by removing two adjacent sides, as shown in Figure 26.25.** One of the sides is the top and the other is the shared side with the hidden body. The body that should be hidden at this point is shown as transparent in the image for reference only. The body was made transparent to make it easier to select the face of the second body.

7. **Show the first body either from the Solid Bodies folder at the top of the tree or from the RMB menu of the first solid feature in the tree.**

8. **Shell the bottom side of the first body, so that the cavities in the two bodies are on opposite sides.**

9. **Combine the two bodies using the Combine tool you can find by choosing Insert ⇨ Features ⇨ Combine.** This feature is also available via the RMB menu in the solid body folder. Select the Add option and select the two bodies. Click OK to finish the feature. Figure 26.26 shows the finished part.
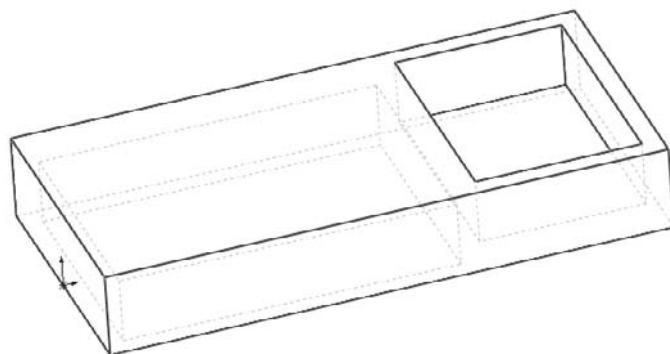
FIGURE 26.25

Shelling two sides of a block

FIGURE 26.26

The finished part

Splitting and patterning bodies

This tutorial guides you through the steps to delete a pattern of *features* from an imported body, separate one of the features, and then pattern it with a different number of features. This introduces some simple surface functions in preparation for Chapter 27. Follow these steps:
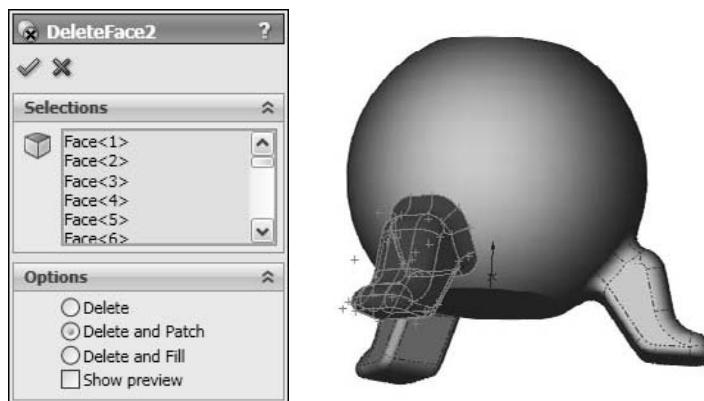
1. **Open the Parasolid file from the CD-ROM called** Chapter 26 – Bonita Tutorial.x_t.

2. **Using the Selection Filter set to filter Face selection (the default hotkey for this is X), select all the faces of the leg.** You can use window selection techniques to avoid clicking each face.

3. **Click the Delete Face button on the Surfaces toolbar, or access the command by choosing Insert ⇨ Face ⇨ Delete from the menus.** Make sure that the Delete and Patch option is selected. The selected faces and the Delete Face PropertyManager should look like Figure 26.27. Click OK to accept the feature.

The Delete Face PropertyManager

4. **Repeat the process for a second leg, leaving the third leg to be separated from the rest of the part and patterned.**

5. **After the two legs have been removed, click the outer main spherical surface, and then choose Insert ⇨ Surface ⇨ Offset from the menus.** Set the offset distance to zero. Notice that a Surface Bodies folder is now added to the tree, near the top.
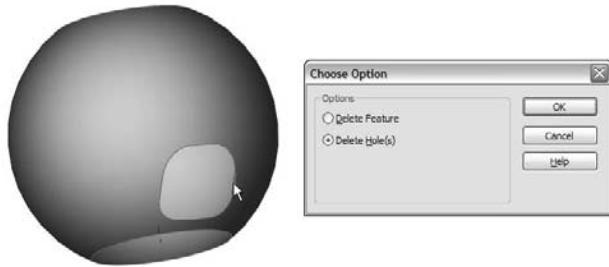
Tip

A zero distance offset surface is frequently used to copy faces. ■

6. **Hide the solid body.** You can do this from the Solid Bodies folder, from the FeatureManager, or from the graphics window.

7. **Hiding the solid leaves the offset surface, and there should be three holes in it.** Select one of the edges of the hole indicated in Figure 26.28 and press the Delete key. The Choose Option dialog box appears. Select the Delete Hole option rather than the Delete Feature option. The Delete Hole operation becomes a history-based feature in the model tree. Before moving on to the next step, remember that you may need to turn off the Selection Filter for faces.

Using the Delete Hole option

Note

Delete Hole is really a surface feature called Untrim. Untrim is discussed more in Chapter 27, but you can use it to restore original boundaries to a surface. ∎

8. Once you delete the hole from the surface body, change the color of the surface body the same way you changed the colors of parts, faces, and features.

9. **Click the surface body in the Surface Bodies folder and either press the Delete key, or select Delete Body from the RMB menu.** Then click OK to accept the feature. This places a Delete Body feature in the tree. It keeps the body from getting in the way when it is not needed. This is not a necessary step, but many people choose to use it. (Suppress the Delete Body feature, since this body is needed again later.)

Tip

If you delete a body in this way and then need it later down the tree, you can delete, suppress, or reorder the Delete Body feature later in the tree. ∎
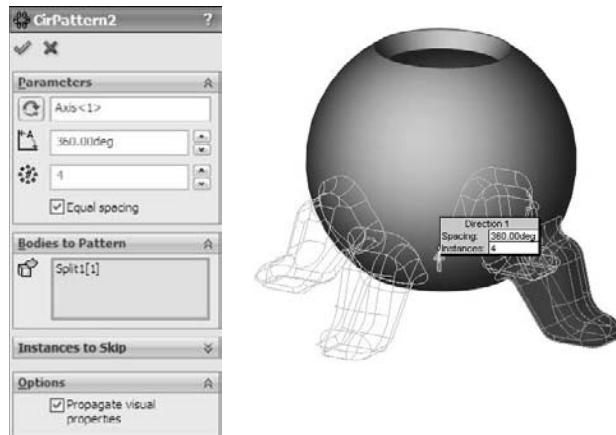
10. **Now show the solid body.** You will notice the color of the surface conflicting with the color of the solid. This mottled appearance is due to the small approximations made by the rendering and display algorithms.

11. **Initiate the Split feature by choosing Insert ➪ Features ➪ Split from the menus, or clicking the Features toolbar.** Use the surface body to split the solid body. Click the Cut Part button, and select the check boxes in front of both bodies in the list. Click OK to accept the feature. Notice now that the Solid Bodies folder indicates that there are two solid bodies.

12. **From the View menu, select the display of Temporary Axes.** Initiate a Circular Pattern feature, selecting the temporary axis as the axis, and the split-off leg in the Bodies to Pattern selection box. Set it to four instances, as shown in Figure 26.29.

FIGURE 26.29

Patterning a body

13. **Use the Combine feature to add together all five bodies.** You can access this feature by choosing Insert ➪ Features ➪ Combine from the menus.

Summary

Beginning to understand how to work with multiple bodies in SolidWorks opens a gateway to a new world of modeling possibilities. Like in-context design, multi-body modeling is definitely something that you have to go into with your eyes open. You will experience difficulties when using this technique, but you will also find new possibilities that were not available with other techniques. The key to success with multi-body techniques is understanding the capabilities and limitations of the tools.

When using a model with the multi-body approach, make sure that you can identify a reason for doing it this way rather than using a more conventional approach. Also keep in mind the list of applications or uses for multi-body modeling mentioned in this chapter. For some types of work, you cannot avoid multi-body modeling, such as using the SolidWorks Mold Tools and Weldments.

Working with Surfaces

W ith *surface modeling* you build a shape face by face. Faces made by surface features can be knit together to enclose a volume, which can become a solid. With solid modeling, you build all the faces to make the volume at the same time. In fact, solid modeling is really just highly automated surface modeling. Obviously there is more detail to it than that, but this definition will get you started.

You can drive a car without knowing how the engine works, but you cannot get the most power possible out of the car by only pressing harder on the gas pedal; you have to get under the hood and make adjustments. In a way, that is what working with surfaces is really all about — getting under the hood and tinkering with the underlying functionality.

The goal of most surface modeling is to finish with a solid. Some surface features make faces that will become faces of the solid, and some surface features only act as reference geometry. Surface modeling is inherently multi-body modeling because most surface features do not merge bodies automatically.

Introducing Surfaces

In the end, you may never really *need* surfaces. It is possible to perform workarounds using solids to do most of the things that most users need to do. However, many of these workarounds are inefficient, cumbersome, and raise as many difficulties as they solve. Although you may not view some of the typical things you now do as inefficient and cumbersome, once you see the alternatives, you may change your mind. The goal for this chapter is to

introduce surfacing functions to those of you who do not typically use surfaces. I am not showing how surfaces are used in the context of creating complex shapes, just how you can use them for various general 3D modeling tasks.

The word *surfacing* has often been used (and confused) to signify complex shapes. Not all surface work is done to create complex shapes, and many complex shapes can be made directly from solids. Many users think that because they do not make complex shapes, they never need to use surface features. This chapter shows mainly examples that do not require complex shapes, in situations where surfaces make it easier, more efficient, or simply possible to do the necessary tasks.

While some of the uses of surfaces may not be immediately obvious, by the end of this chapter, you should have enough information and applications that you can start experimenting to increase your confidence in surfacing techniques.

Understanding Surfacing Terminology

When dealing with surfaces, you may hear different terminology than the terminology typically used with solid modeling. This special terminology also often exists for surfaces because of important conceptual differences between how solids and surfaces are handled.

These terms are fairly universal among all surfacing software. The underlying surface and solid construction concepts are generally uniform between the major solid and surface modeling packages. What varies from software to software is how the user interacts with the geometry through the software interface. You may never see some of these terms in the SolidWorks menus, Help files, training books, or elsewhere, but it becomes obvious as you use the software that the concepts are relevant.

Knit

Knit is analogous to the solid feature Combine in that it joins multiple surface bodies into a single surface body. Unlike Combine, Knit does not perform the subtract or intersect Boolean operations. It also has an option to create a solid if the resulting surface body meets the requirements (a fully enclosed volume without gaps or overlaps). However, unlike the solid bodies in Combine, which may overlap volumetrically, surface bodies must only intersect edge to edge, more like sketch entities that can only touch end to end.

Knit is also sometimes used in the same way that the zero-distance offset is used, to copy a set of solid faces to become a new surface body.

You can find one nice option that enables you to quickly see where the boundaries of a surface body lie by choosing Tools ➪ Options ➪ Display/Selection ➪ Show Open Edges Of Surfaces In Different Color. By default, this color is a medium blue, and you can change it by choosing Tools ➪ Options ➪ Colors ➪ Surfaces ➪ Open Edges.

Trim

The Trim function is analogous to the solid Cut. You can use sketches, planes, or other surface bodies to trim a surface body. The underlying surface is defined by a two-dimensional mesh, and for this reason, it is usually four-sided, but may be other shapes. When the underlying surface is trimmed, the software still remembers the underlying shape, but combines it with the new boundary, which is typically how face shapes (especially non-four-sided shapes) are created.
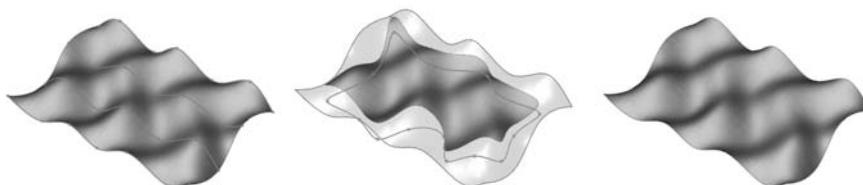
Untrim

Untrim is predictably the opposite of Trim. It removes the boundary from a surface. It can remove the boundary selectively (one edge at a time, interior edges only, and so on) or remove all the edges at once. Untrim even works on imported geometry, as described in the tutorial in Chapter 26. Figure 27.1 shows how Untrim works.

FIGURE 27.1

Untrimming a surface

Surface created native
in SolidWorks by
lofting

Surface trimmed by a
sketch, trimmed away
portion shown as
transparent

Trimmed surface with
an Untrim feature
applied to it

Untrim works on native and imported geometry. It is not truly like feature history in imported geometry, but it does help to uncover the underlying original shape of the face.

Hybrid modeling

Modeling software has long divided itself along Solid/Surface lines with products such as Rhino (strictly surface modeling) and early versions of SolidWorks (strictly solid modeling). However, in the last several years, modelers increasingly enabled both methods and allowed them to interact. This hybrid modeling is a combination of solid and surface modeling. These days, it is much more common to mix methods than it was even five years ago. Surface modeling is slow because you model each face individually, and then manually trim and knit. Cutting a hole in a surface model is much more involved than cutting a hole in a solid. Solid modeling is faster because it is essentially highly automated surface modeling; however, as any software user knows, automation almost always comes at the expense of flexibility, and this situation is no different. Surface modeling puts the compromised power back into your hands.

Solid modeling tends to limit you to a type of parts with square ends or a flat bottom because solids are creating all sides of an object at once. For example, think about an extrusion: regardless of the shape of the sketch, you have two flat ends. Even lofts and sweeps typically end up with one or two flat ends because the section sketches are often planar. Surfaces enable you to create one side at a time. Another way of looking at it is that using surfaces *requires* you to create one side at a time.

You will find times when, even with prismatic modeling, surfacing functions are extremely useful, if not completely indispensable. I do not propose that you dive into pure surface modeling just to benefit from a few of the advantages, but I do recommend that you consider using surface techniques to help define your solids. This hybrid approach is sensible and opens up a whole new world of capabilities. I have heard people say after taking a SolidWorks surfacing class that they would never look at the software in the same way again.
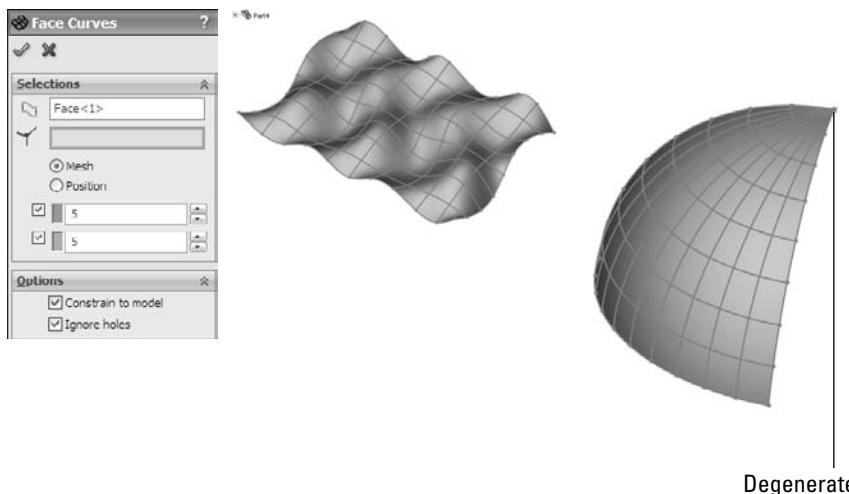
Non Uniform Rational B Spline

NURBS stands for Non Uniform Rational B Spline. NURBS is the technology that most modern mechanical design modelers use to create 3D geometry. NURBS surfaces are defined by curves in perpendicular directions, referred to as U and V directions, which form a mesh. The fact that perpendicular directions are used means that the surfaces have a tendency to be four-sided. Of course exceptions exist, such as three-sided or even two-sided patches. Geometry of this kind is referred to as *degenerate* because one or more of the sides has been reduced to zero length. Degenerate geometry is often, but not always, the source of geometrical errors in SolidWorks and other CAD packages.

Figure 27.2 shows some surfaces with the mesh displayed on them. You can create the mesh with the Face Curves sketch tool.

FIGURE 27.2

Meshes created with the Face Curves sketch tool

Degenerate point

An example of a competitive system to NURBS surface modeling is point mesh data. This comes from systems such as 3DSMax, which create a set of points that are joined together in triangular facets, and can be represented in SolidWorks as an STL (stereolithography) or VRML (virtual reality markup language) file. When displayed in SolidWorks, this data looks very facetted or tessellated into small, flat triangles, but when viewed in software that is meant to work with these kinds of meshes, it looks smooth. Many advantages come with this type of data, especially when it comes to applying colors and motion. However, the main disadvantage is that the geometrical accuracy is not very good. Point mesh data is typically used by 3D graphic artists, animators, and game developers.

By using a SolidWorks add-in such as ScanTo3D (available only in SolidWorks Premium), it is possible to take point mesh data and create a NURBS mesh over it. This is not a push-button solution, but it offers capabilities where none previously existed. ScanTo3D is beyond the scope of this book, but you should find it useful if you are interested enough to read about NURBS and point meshes.

Developable surface

Developable surfaces are surfaces that can be flattened without stretching the material. They are also surfaces that you can extend easily in one or both directions. These include planar, cylindrical, and conical shapes. It is not a coincidence that these are the types of shapes that can be flattened by the Sheet Metal tools.

Ruled surface

Developable surfaces are a special type of a broader range of surface called ruled surfaces. SolidWorks has a special tool for the creation of ruled surfaces that is described in detail in the next section. Ruled surfaces are defined as surfaces on which a straight line can be drawn at every point. A corollary to this is that ruled surfaces may have curvature in only one direction. Ruled surfaces are far less limited than developable surfaces, but are not as easily flattened.

Ruled surfaces are used frequently in plastic parts and plastic mold design where draft and parting surfaces from 3D parting lines are needed.

Gaussian curvature

Gaussian curvature is not referred to directly in SolidWorks software, but you may hear the term used in more general CAD or engineering discussions. It can be defined simply as curvature in two directions. As a result, a sphere would have Gaussian curvature, but a cylinder would not.

Existing Surface Tools

Surface feature equivalents are available for most solid features such as extrude, revolve, sweep, loft, fillet, and so on. Some solid features do not have an equivalent, such as the Hole Wizard, shell, and others. Several surface functions do not have solid equivalents, such as trim, Untrim, Extend, Thicken, Offset, Radiate, Ruled, and Fill.

This is not a comprehensive guide to complex shape modeling, but it should serve as an introduction to each feature type and some of the details about how it operates.
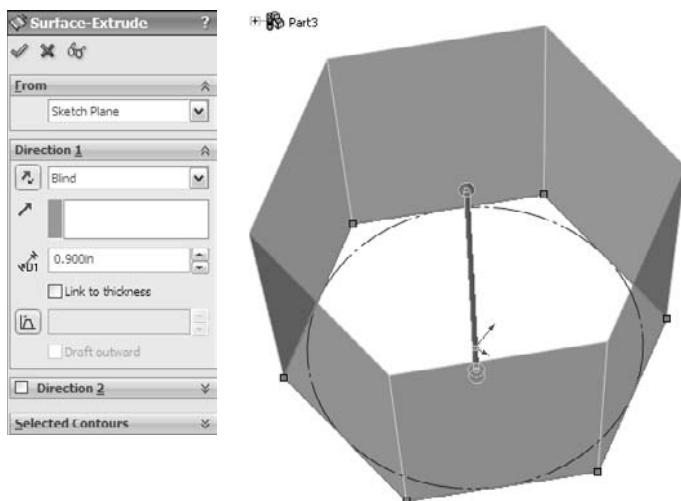
Extruded Surface

The Extruded Surface works exactly like an extruded solid, except that the ends of the surface are not capped. It includes all the same end conditions, draft, contour selection, sketch rules, and so on that you are already familiar with. Figure 27.3 shows the PropertyManager for the Extruded Surface.

The Extruded Surface PropertyManager

You can also create extruded surfaces from open sketches, and, in fact, that is probably a more common situation than creating a surface with a closed sketch.

When two non-parallel sketch lines are joined end to end, the result of extruding the sketch is a single surface body that is made of two faces with a hard edge between them. If the sketch lines were disjointed, then the extrude would result in disjoint surface bodies. If the sketch lines were again made end to end, but done in separate sketches, then the resulting surface bodies would be separate bodies; the second body would not be automatically knit to the first one as happens with solid features. This is an important quality of surfaces to keep in mind. If you create surfaces in different features and want them knitted into a single body, then you will have to do that manually.

Revolved Surface

The Revolved Surface functions like its solid counterpart, right down to the rules for how it handles entities that are touching the axis of revolution; nothing can cross the axis. A single sketch entity is allowed to touch it at a single point, but multiple sketch entities cannot touch it at the same point.

Swept Surface

Swept Surfaces work much like their solid counterpart, and the sketch rules and available entities are the same. The main difference here is going to be that Swept Surfaces usually use an open contour for the profile, while swept solids use closed contours.

Lofted Surface

The main difference between Lofted Surfaces and lofted solids is that the surfaces can use edges and curve features as profiles, rather than simply sketches and faces.

Boundary Surface

The Boundary Surface was created as a higher quality replacement for the Loft feature, but certain limitations mean that Loft has not been removed from the feature list. The Boundary Surface most resembles a loft, but has elements of the Fill Surface, and at times can also function like a sweep. It requires you to select edges or sketch elements in two different directions, directly relating to the NURBS scheme I discussed earlier in this chapter. This feature can work with only one set of edges selected.

If several edge or sketch segments combine to form one side of a direction, then you must use the SelectionManager to form the edge segments into a group. SelectionManager enables you to select portions of a single sketch or to combine elements such as sketch, edge, and curve into a single selection for use as a profile or guide curve for Boundary or Loft features.

The interface for the Boundary Surface is shown in Figure 27.4.

The types of models where you end up using the Boundary Surface are highly curvy models that are modeled mainly with surface features and require a four-sided patch.

The main advantage of Boundary Surface over Loft is that Boundary Surface can apply a Curvature boundary condition all the way around, while Loft cannot apply curvature on the guide curves. Fill Surfaces also can apply a Curvature boundary condition all the way around.
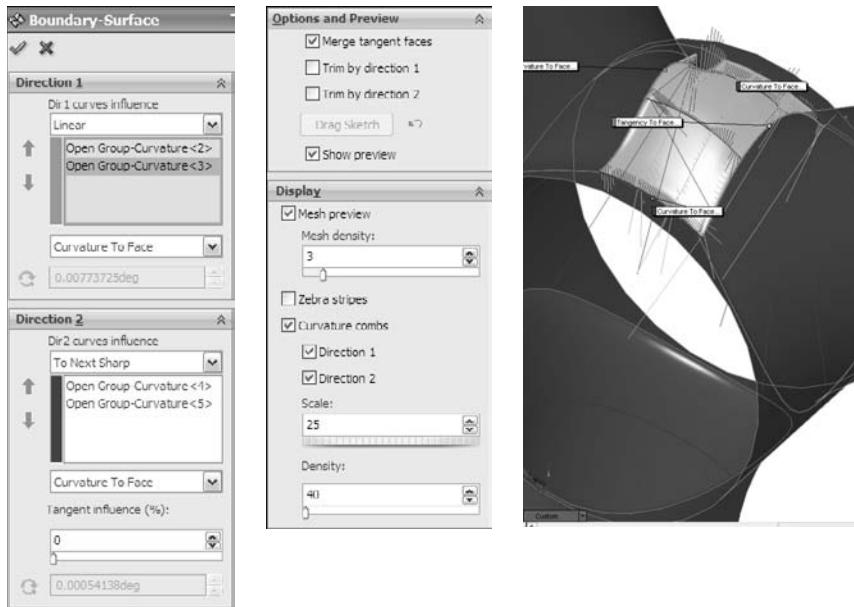
Boundary Surface can be a rather nuanced feature, but when working on the type of model that suits it well, I default to Boundary Surfaces when possible. Boundary solid features are now also available, and I expect these will also take a little bit of a learning curve to understand where they are best applied.

For a more detailed look at the primary shape creation tools (Sweep, Loft, Boundary, and Fill) and surface modeling in general, please refer to the *SolidWorks Surfacing and Complex Shape Modeling Bible* (Wiley, 2008).

FIGURE 27.4

The Boundary Surface PropertyManager

Offset Surface

The Offset Surface has no solid feature counterpart, but it does in 3D what the Offset Sketch function does in 2D; it may also fail for the same reasons. For example, if you offset a.25-inch radius arc by.3 inches to the inside, it fails because it cannot be offset up to or past a zero radius. The same is true of offsetting surfaces. Complex surfaces do not have a constant curvature, but are more like a spline by having a constantly changing curvature. If the offset is going in the direction of decreasing radius, and is more than the minimum radius on the face or faces being offset, then the Offset Surface feature will fail.

One of the ways to troubleshoot a failing Offset Surface is to use the Check tool to check for minimum radius. Remember that the area with the minimum radius is only a problem if the curvature is in the same direction as the offset. If a small radius will increase when it is offset, then that small radius is not the problem. The problem comes from the other direction where you are offsetting to the inside of a small radius.

Unlike the Sketch Offset function — and as was shown in Chapter 26 — you can offset surfaces by a zero distance. This is usually done to copy either solid or surface faces to make a new surface body. Zero-distance offset and Knit are sometimes used interchangeably, although Knit causes a problem if you are selecting a surface body that is composed of a single face. Knit assumes that you are trying to knit one body to another, and so, by default, it selects the body, and then fails with the message that you cannot knit a body to itself. Because Knit has this limitation, and Offset does not, I prefer the Offset tool when copying faces to make a new surface body. You may also notice that when you enter a zero for the offset distance, the Offset PropertyManager name changes automatically to Copy Surface.

Knit does have two functions that Sketch Offset does not. One of these is the option to create a solid from the knit body if it forms a closed body. The second option is somewhat more obscure, offering the ability to select all faces on one side of a Radiate Surface. I discuss this option in more depth later in this chapter in the Knit Surface section.

When talking about copying surface bodies, you must also consider the Move/Copy Bodies feature, which is described in Chapter 26. When simply copying a body without also moving it, this feature issues a warning that asks whether you really intend to copy the body without moving it. This is an annoying and pointless message. Also, the Move/Copy Bodies feature does not enable you to copy only a part of a body (selected faces) or to merge multiple bodies into one like the Knit and Offset Surface features.

All things considered, I recommend using the zero-distance Surface Offset feature to copy bodies or parts of bodies unless your goal is to immediately make a solid out of it (in which case you should use the Knit feature) or when using a Radiated surface (typically in a mold-building application).

Radiate Surface

The Radiate Surface is not one of the more commonly used surface features. It has been largely superseded by the Ruled Surface. This is because Ruled Surface does the same sort of thing that Radiate Surface does, as well as a lot more, and is also more reliable. Radiate works from an edge selection, a reference plane, and a distance. The newly created surface is perpendicular to the selected edge and parallel to the selected plane, and the set distance is wide. It is probably most commonly used in creating molds or other net shape tooling such as dies for stamping and forging, blanks for thermoforming, and so on. Figure 27.5 shows the PropertyManager and selection for creating a Radiate Surface.

Tip
The Radiate Surface feature does not give you a preview of the finished surface, only the small arrows that indicate the direction in which the surface will radiate. At times, you may need to switch the arrows to the other side, which you can do by using the arrow button next to the plane selection. ■
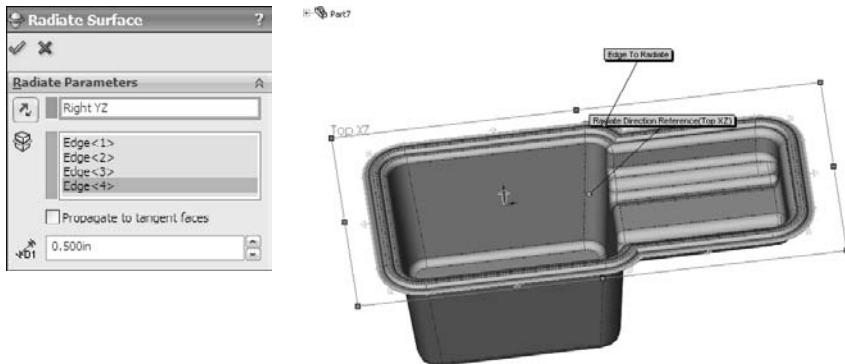
Caution
When creating a Radiate Surface, the use of a loop in the edge selection always results in an incorrect result, because the feature only uses the initial edge that was selected for the loop. As long as individual edges are listed in the selection box, you should be okay. ■
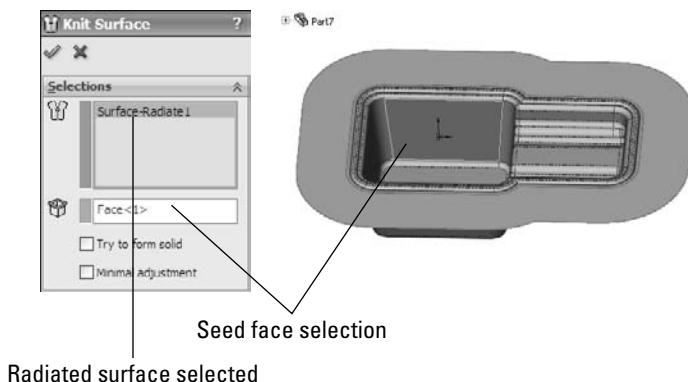
The Radiate Surface PropertyManager

The one application where the Radiate Surface has a very interesting usage is when you combine it with the Knit function, as mentioned earlier. Figure 27.6 shows a part surrounded by a Radiate Surface in which the Knit feature is being used to select all the faces to one side of the radiated surface. The second smaller selection box in the PropertyManager that contains Face<1> is called a *seed face* and causes the Knit to automatically select all the faces on the same side of the model as the selected seed face. The requirement here is that the Radiate goes completely around the model and separates the faces into faces on one side of the Radiate and faces on the other side of the Radiate. The use of the Radiate with the Seed Face selection is extremely useful for mold creation.

Using Radiate Surface with Knit

Seed face selection

Radiated surface selected
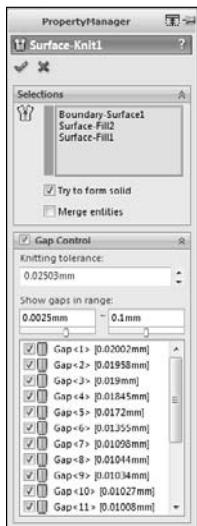
Knit Surface

The Knit Surface functionality is discussed previously in the terminology section as well as in the Radiate Surface section.

If the knit operation results in a watertight volume, the Try to form solid option turns the volume into a solid. You can also make a solid from a surface using two other functions. The Fill Surface has an option to merge the fill with a solid or to knit it into a surface body; if the knit surface body is closed, then it gives you the option to make it a solid. This is very nice, complete interface design, with options that save you many steps. The Fill Surface feature is described in more detail later in this chapter.

In SolidWorks 2010 the Knit feature has some important new functionality. The Gap Control panel, shown in Figure 27.7 shows the gap between the edges of surfaces to be knit, and enables you to see the gaps in a certain range and force gaps of less than a specified tolerance value, called the Knitting Tolerance, to knit. This replaces the old Minimal Adjustment option in Knit, which was SolidWorks did not go to any great length to help users understand. The Knitting Tolerance tools are easier to understand and more powerful when it comes to forcing knit features to work.

FIGURE 27.7

Knitting Surfaces with gaps using tolerances

With all powerful tools the possibility of misusing the tool always exists. The ability to play with tolerances is a double-edged sword. On one hand it gives you the ability to force surfaces to knit which may have otherwise not knit at all. On the other hand, if you allow a larger tolerance, SolidWorks may force together surfaces or edges that have problems that should be solved in other

ways. such as removal and remodeling. Gap tolerances can cause problems during import operations to other software that doesn't have the capability to adjust tolerances. Certainly use the Gap Control options, but also be aware of the potential problems you might see with data downstream.
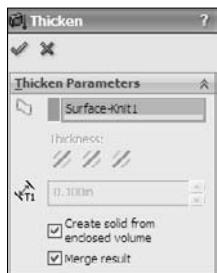
Thicken Surface

The other function that also creates a solid from a surface is the Thicken feature. If a surface body that encloses a volume is selected, then the option Create solid from enclosed volume appears on the Thicken PropertyManager, as shown in Figure 27.8. You can access the Thicken feature by choosing Insert ⇨ Boss/Base ⇨ Thicken from the menus.

FIGURE 27.8

The Thicken PropertyManager

Planar Surface

Planar surfaces can be created quickly and are useful in many situations, not just for surfacing work. Because they are by definition *planar*, you can use them to sketch on and for other purposes that you may use a plane for, such as mirroring.

However, more commonly, Planar Surfaces are created from a closed sketch such as a rectangle. You can create multiple Planar Surfaces at once, and the surfaces do not need to all be on the same plane or even parallel. This is commonly done to close up holes in a surface model, such as at the bottom of cylindrical bosses on a plastic part, using a planar circular edge. A good example of this is the bike frame part in the material for Chapter 27 on the CD-ROM, named `Chapter 27 – bike frame.sldprt`.

Remember that a Planar Surface was used in Chapter 26 with the Split feature to split the leg off of an imported part. This was more effective than a sketch or a plane because the split was limited to the bounds of the Planar Surface, not infinite like the sketch or the plane.

The Planar Surface does not knit itself into the rest of the surface bodies around it automatically; you have to use the Knit feature to do this.
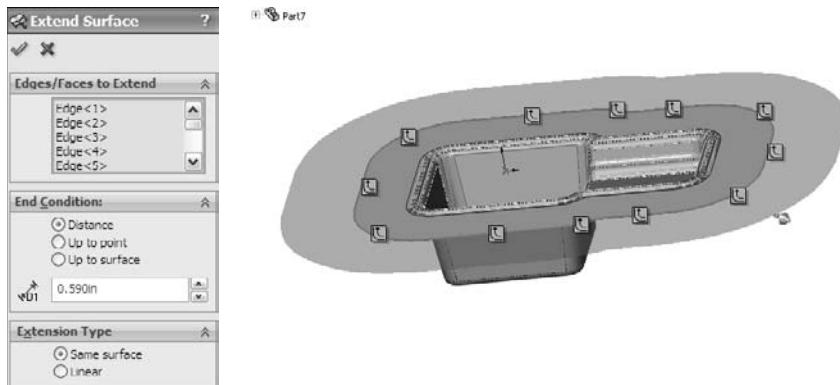
Extend Surface

The Extend Surface feature functions much in the same way that the Extend function works in sketches. Figure 27.9 shows the PropertyManager interface and an example of the feature at work.

FIGURE 27.9

The Extend Surface PropertyManager

The only item here that requires explanation is the Extension Type panel. The Same surface option means that the extended surface will simply be extrapolated in the selected direction. A Planar Surface is the easiest to extend because it can go on indefinitely without running into problems. A cylindrical surface can only be extended until it runs into itself. Complex lofted or Swept Surfaces are often difficult to extend. Extrapolating a complex surface is not easy to do and often results in self-intersecting faces, which cause the feature to fail.

When the Same surface setting works, it creates a nice result because it does not create an edge where the extension begins; it smoothly extends the existing face.

The Linear option is more reliable than the Same surface option because it starts tangent to the existing surface and keeps going in that direction, working much like a Ruled surface, which is covered later in this chapter. It does not rely on extending the existing surface. This option creates an edge at the starting point of the new geometry.

Trim Surface

The Trim Surface feature is described briefly earlier in this chapter, but it warrants a more complete description here. Surfaces can be trimmed by three different types of entities:

- Sketches
- Planes
- Other surfaces

855

When you use surface bodies to trim one another, you must select one of two options: Standard or Mutual Trim. The Standard option causes one surface to act as the Trim tool and the other surface to be trimmed by the Trim tool. When you select the Mutual Trim option, both surfaces act as the Trim tool, and both surfaces are trimmed.

For an example of trimmed surfaces, open the mouse example from Chapter 26 and step through the tree. This shows examples of a couple of types of trimmed surfaces, as well as extended surfaces and others.

Many people overlook the ability to trim a surface with a plane, which can be very handy sometimes. Planes are infinite, which means you have less to worry about when it comes to changes that affect features rebuilding correctly.

Finally, trimming with 2D sketches is well known, but trimming with 3D sketches is less known. There is a 3D sketch tool called Spline on Surface that enables you to draw a spline directly on any surface body. An option exists in the Trim Surface PropertyManager to trim a surface with this type of sketch. This is very useful in many situations if you can remember that it is available.

Fill Surface

The Fill Surface is one of my favorite tools in SolidWorks. I often refer to it as the "magic wand" because it is sometimes amazing what it can do. It is alternately referred to by the SolidWorks interface and documentation as either Fill or Filled, depending on where the reference is made. You will find it listed as both in the SolidWorks interface.

The Fill Surface is intended to fill in gaps in surface bodies. It can do this either smoothly or by leaving sharp corners. You can use constraint curves to drive the shape of the fill between the existing boundaries. It can even knit a surface body together into a solid, all-in-one step. Beyond this, you can use the Fill Surface directly on solid models and integrate it directly into the solid automatically (much like the Replace Face function, which is described later in this chapter).
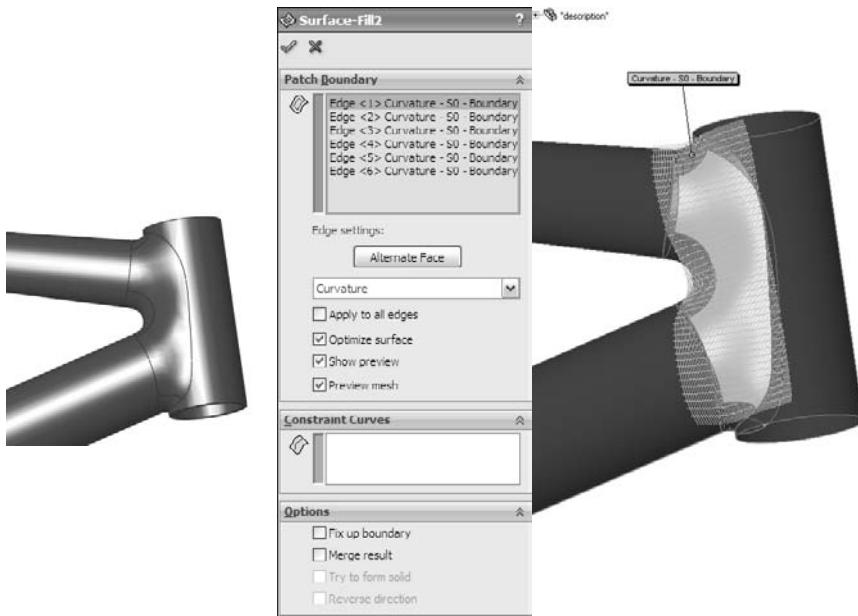
Several rather complex examples of the Fill Surface are found in the bike frame example that was originally shown in Chapter 12. One of these fills is shown in Figure 27.10.

The first thing you should notice about the Fill Surface is that it is creating an oversized, four-sided patch and trimming it to fit into the available space. This is one of the reasons why I consider this to be such a magical tool. The four-sided patch I referred to earlier in the section on NURBS is shown very clearly in this feature preview. Also, the trimmed surface concept is illustrated nicely by this feature. Not surprising, if you untrim the Fill Surface, then you return to the surface that is previewed here. In this one function, SolidWorks gives you some useful insight about what is going on behind the scenes.

When using the Fill Surface, it is best to have a patch completely bounded by other surfaces, as shown in Figure 27.9. Fill Surface can work with a boundary that is not enclosed, but it works better with a closed boundary.

FIGURE 27.10

The Fill Surface PropertyManager and the results of applying it

You can set boundary conditions as Contact, Tangent, or Curvature. Contact simply means that the faces touch at an edge. Tangent means that the slopes of the faces on either side of the edge match at all points along the edge. Curvature means curvature continuous (or C2), where the Fill Surface matches not only tangency, but also the curvature of the face on the other side of the boundary edge. This results in a smoother transition than a transition that is simply tangent.
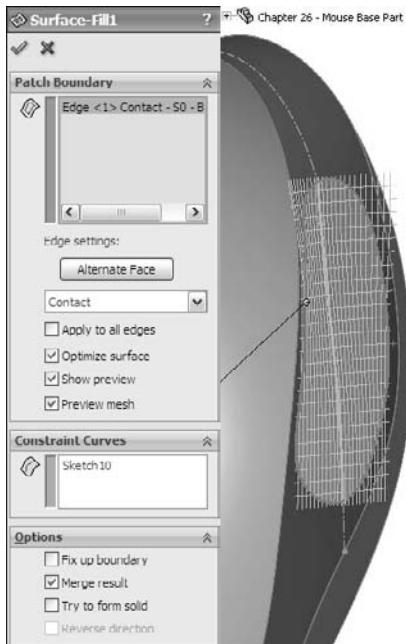
When you select the Optimize Surface option, SolidWorks tries to fit the four-sided patch into the boundary. Notice that on this part, even though the Optimize surface option is selected, it is clearly being ignored because the boundary is a six-sided gap and cannot be patched smoothly with a four-sided patch. It is not necessarily an improvement to make a Fill Surface optimized, even when it works.

Constraint curves can influence the shape of the Fill Surface. An example of this is shown in Figure 27.11. The construction splines shown on the faces of the part were created by the Intersection Curve tool and enabled the spline used for the constraint curve to be made tangent to the surface.

FIGURE 27.11

The Fill Surface feature with constraint curves

MidSurface

The MidSurface feature is not used very often. It was probably originally intended to be used in conjunction with analysis tools to create plate elements for thin walled structures. It works on parallel faces of a solid, creating a surface midway between the faces. If the faces have opposing draft (such that a wall is wider at the bottom than at the top), then the MidSurface will not work. It works on linear walls and cylindrical walls, but not on elliptical or spline-based shapes. The PropertyManager for the MidSurface is shown in Figure 27.12.

Similar to the Planar Surface, you can also use the MidSurface to create a surface that can be used like a plane. No plane type can create a symmetrical plane, but using a MidSurface, you can create a symmetrical Planar Surface between parallel walls.
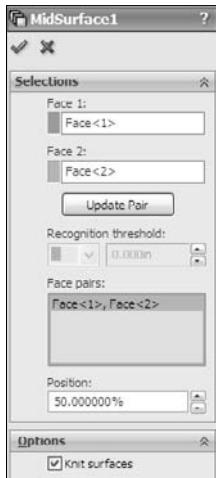
Replace Face

The Replace Face feature does not create new surface geometry, but it does integrate existing surface geometry into the solid. It replaces selected faces of a solid or surface body with a selected surface body. Replace Face is one of the few tools that can add and remove material at the same time with a single feature.

FIGURE 27.12

The MidSurface PropertyManager

If you were to manually perform the functions that are done by Replace Face, then you would start by deleting several faces of the solid, then extending faces, and then trimming surface bodies, and finish by knitting all the trimmed and extended faces back into a single solid body.

This is a very powerful and useful tool, although it is sometimes difficult to tell which situations it will work in. Figure 27.13 shows a part before and after a Replace Face feature has been added. The surface used to replace the flat face of the solid has been turned transparent. The first selection box is for the original face or faces, and the second selection box is for the surface body with the new faces. The tool tips for each of the boxes are Target Faces For Replacement and Replacement Surface(s), which seem a little ambiguous. I like to think of them as Old (top) and New (bottom).

Untrim Surface

The Untrim Surface is discussed in the terminology section of this chapter. You can use it either selectively on edges or on the entire surface body.

Parting Surface
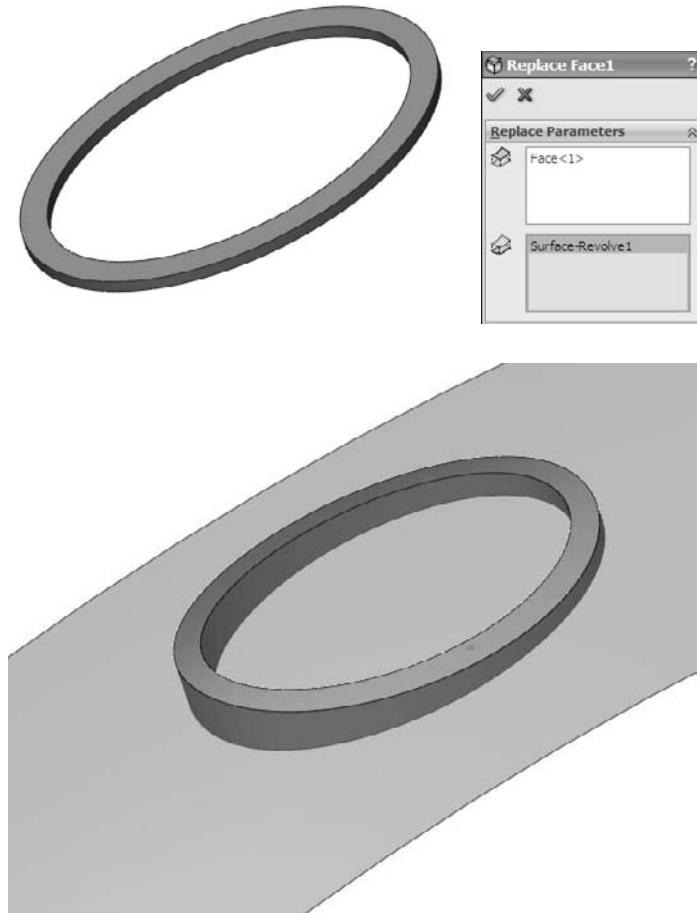
The Parting Surface is part of the SolidWorks Mold Tools. The Mold Tools are addressed in Chapter 32.

FIGURE 27.13

Using Replace Face

Ruled Surface
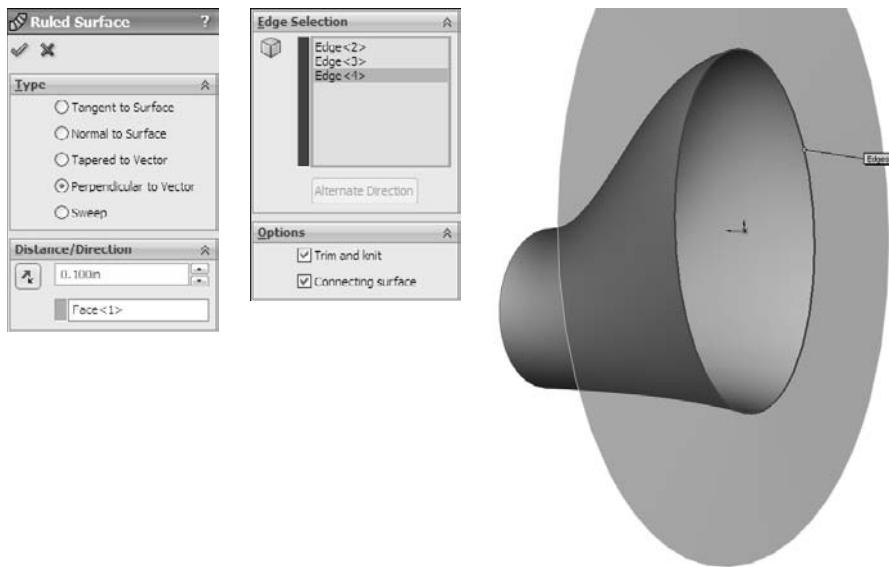
Ruled Surfaces are discussed in general in the section on terminology. Here I discuss the topic in more detail, and specifically with regard to the SolidWorks interface for creating Ruled Surfaces.

The Ruled Surface feature in SolidWorks is one of those features that you may never have missed until you see it in action. It is extremely useful for constructing faces with draft, extending faces tangent to a direction, making Radiate Surface types, building molds, and many other applications.

Figure 27.14 shows the PropertyManager interface for the Ruled Surface.

FIGURE 27.14

The Ruled Surface PropertyManager interface

The Ruled Surface works from the edge of a solid or surface body. The feature has five basic types of operations that it can perform:

- Tangent to Surface
- Normal to Surface
- Tapered to Vector
- Perpendicular to Vector
- Sweep

The Tangent to Surface setting is self-explanatory. The Alternate Face option would be available if the base shape had been a solid, with a face filling the big elliptical hole. This would make the Ruled Surface tangent to the bottom face instead of the side.

With the Normal to Surface setting, it tilts up five degrees from the horizontal because the surface is lofted with a five-degree draft angle at the big end, making a Ruled Surface that is normal. Be careful of using this setting because it looks close to what you may be hoping for, but is slightly off. One of the other options may be a better choice, depending on what you are looking for.

The Tapered to Vector setting needs a plane or axis selection to establish a direction, and then the Ruled Surface is created from that reference at the angle that you set. With a combination of the Alternate Side button and the arrow direction toggle button next to the plane selection, you can adjust the cone created by this setting. The interface to make the changes is not exactly clear unless you use this function often, but it does work.

The Perpendicular to Vector setting is a better option than the Normal to Surface setting when the surface has been created with some sort of built-in draft angle. This is also the setting that looks most like the Radiate Surface feature, although it works much better than Radiate Surface.

The Sweep setting makes a face that is perpendicular to the surface created by Perpendicular to Vector. It is as if a straight line were swept around the edge. This is actually a great way to offset an edge or 3D sketch: using the edge of the surface as the offset of the original.

Using Surfacing Techniques

This section serves as a brief overview of some of the techniques commonly used among people who model using surfaces. This section could be the topic for an entire book on its own. In fact, it is the topic of an entire book that goes into far greater detail. You may want to use the *SolidWorks Surfacing and Complex Shape Modeling Bible* (Wiley, 2008) to continue your SolidWorks education in far greater detail and depth.

Up To Surface/Up To Body

Cross-Reference

Chapter 7 contains more information on end conditions such as Up To Surface and Up To Body. ∎

Some modeling situations seem to require elaborate workarounds until you think of doing them with a combination of solid and surface features, such as the part shown in Figure 27.15. This geometry could be made completely with solids, but it would be more difficult. In this case, a surface is revolved, representing the shape at the bottom of the hole, and the cut is extruded up to it. You can follow along with the part from the CD-ROM at `Chapter 27 – square hole.sldprt`.

Another familiar situation is when you have a feature to place and you want to use an Offset from Surface end condition, but the feature spans two faces. In that situation, you can knit the necessary faces together (or use offset), and then extrude offset from that surface body.
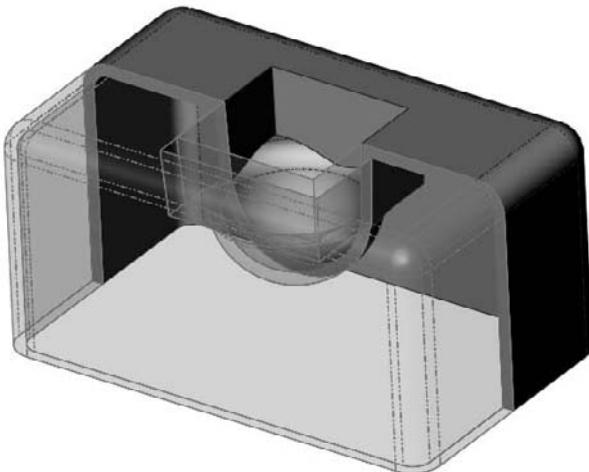
Tip

Using Up To or Offset From Body rather than Face often avoids the common error message, "The end face cannot terminate the extruded feature," especially if the feature that is extruded spans more than one face. ∎

Figure 27.16 shows a part using an Offset Surface to extrude text up to where the text spans more than a single surface. This is a very common application, even if it is not text that is being extruded. The part that was used in Figure 27.15 is on the CD-ROM in the materials for Chapter 26, and is called `Chapter 26 – Up To Body.SLDPRT`.
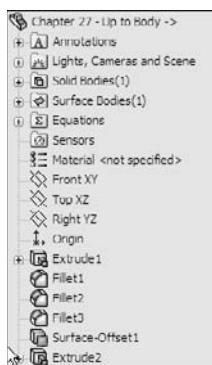
FIGURE 27.15

Using the Up To Body setting

FIGURE 27.16

Extruding text

Cut With Surface
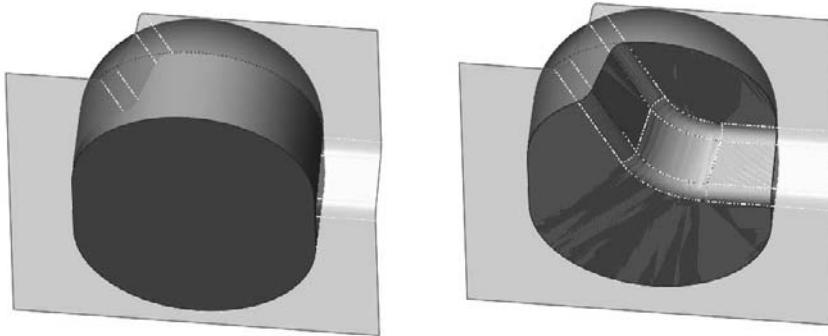
Sometimes you may need to make a cut that is more complex than what a simple extrude can do. For example, the cut may need to have shape in multiple directions. You could make the cut with multiple cut features, or even with a surface. Figure 27.17 shows a part that is cut with a surface.

FIGURE 27.17

Using the Cut With Surface feature on a part

When cutting with a surface, the edges of the surface must be outside of the body that is being cut. With sketches, it is advisable to have more sketch than you need so that you are not trying to cut line-on-line. The same applies to cutting with a surface, where it is advisable to have more surface than you need to make the cut.

Replace Face

The Replace Face feature can be used on imported or native geometry. You can use it to add or remove material from a part. When it adds material, it must be able to extend faces adjacent to those that are being replaced, which can be a limitation. A face or faces do not need to be replaced with the same kind or same number of faces, but the entire face that is being replaced must be removed. If you only want to replace a part of a face, then you can use a Split line to scribe the face, and then replace the part you want.
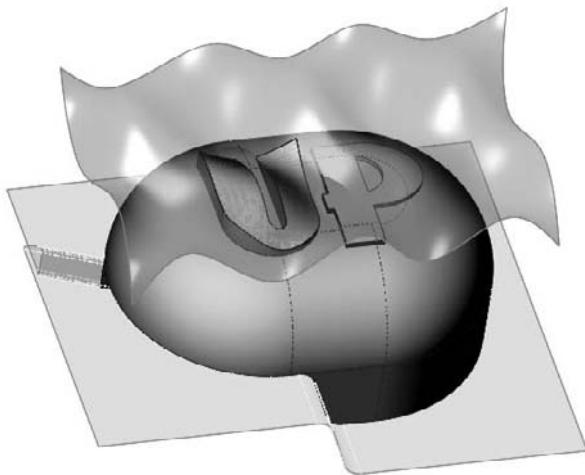
Figure 27.18 shows that the multiple faces of the letter U on this part have been replaced with a surface from an inserted part. Replace Face is a fantastic tool that you can use in a number of situations, although it is a little particular sometimes and you cannot always predict when it will or will not work.

Fill Surface

The Fill Surface is my favorite tool in the SolidWorks software. It can get you out of modeling binds easily, and is often used to cover over nasty modeling mistakes or areas you just can't get right by any other method. In addition to its duty in the complex shapes department, it can also be used as a fast way to create a Planar Surface in some situations. If you do much surface modeling, the Fill feature will become a staple of your diet.

FIGURE 27.18

Using Replace Face

Note

The Fill Surface is an advanced surfacing function. Sometimes, when talking about advanced surfacing functions, or indeed any software function, users have a tendency to sound a little cynical. This is because the tool is often expected to work on very complex geometry. It is not always the software's fault when it cannot perform a particular task, or does not do what you imagine you want it to do. Sometimes, the tool is simply not meant to perform certain tasks, there may be an unseen flaw in the geometry that prevents it from working, or the user does not understand the settings completely. The more complex the work, the more frequently you need to find workarounds to get something done. Avoiding problems does not make them go away. In this book, I have chosen to take a realistic look at most of the features. ■

Figure 27.19 shows the Fill Surface blending an intersection between tubes. The image to the left shows the before condition with the tubes coming together at an edge. The center image shows the edge trimmed out using the Trim feature, and the right image shows the hole blended over by the Fill Surface feature.
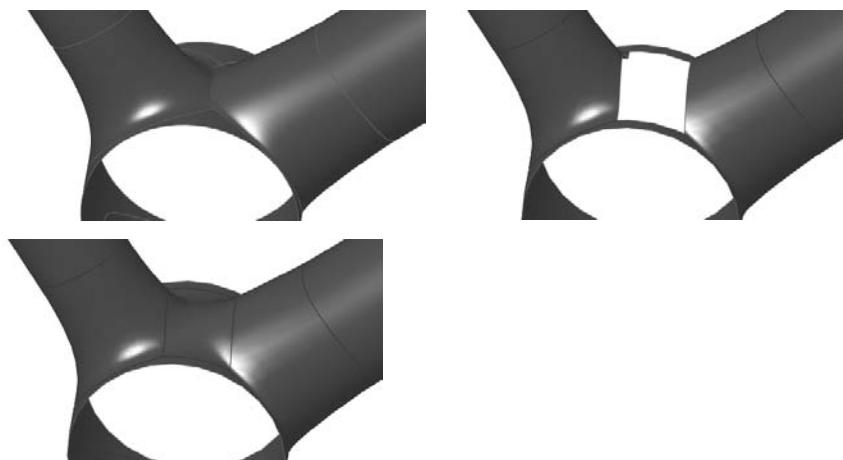
In Figure 27.20, a solid starts with a Split line on the surface. A sketch is then added, and a Fill Surface is created using the sketch as a constraint and the Split line as the boundary. The Merge Result option in the Fill PropertyManager has a different significance than it does in a solid feature PropertyManager, but the end result is the same. Remember that this is a surface function, and if it does not merge, then it is left as a surface feature.

FIGURE 27.19

Blending with the Fill Surface

FIGURE 27.20

The Fill PropertyManager for merging a Fill Surface directly into a solid
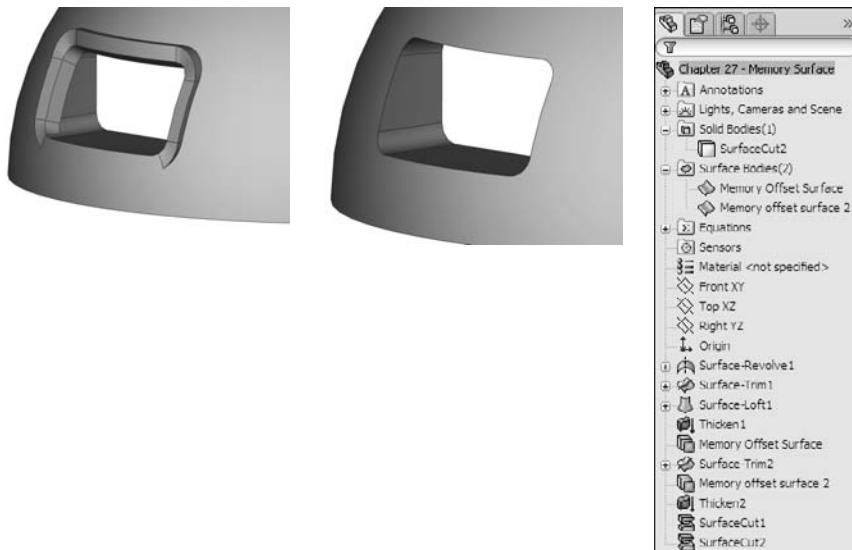
If you had to go through these steps manually, then you would use the Replace Face feature to integrate the surface into the solid. The key to integrating the Fill Surface directly into the solid without any additional features is the Merge Result option in the Fill PropertyManager.

Memory surface

A *memory surface* is not another new type of feature that you can select from the menu or a toolbar; it is just the name that I gave to a technique that I use from time to time. A memory surface is just a Knit or Offset Surface that is made at one point in the feature tree when a particular face is whole, and reused later when the face has been broken up, but you still need to reference the entire original face. An example of this technique is shown in Figure 27.21. In this case, extra material is created around the opening, and a surface that was created in a Rollback state is used to remove it.

FIGURE 27.21

Using the memory surfaces technique to cut away unwanted geometry

Tutorial: Working with Surfaces

This is another chapter that contains many important ideas, and yet there is only so much space for tutorials. The best way to learn is to experiment. I recommend that you closely follow the tutorial steps once, and then, when you understand the concepts involved, that you can go back and experiment.
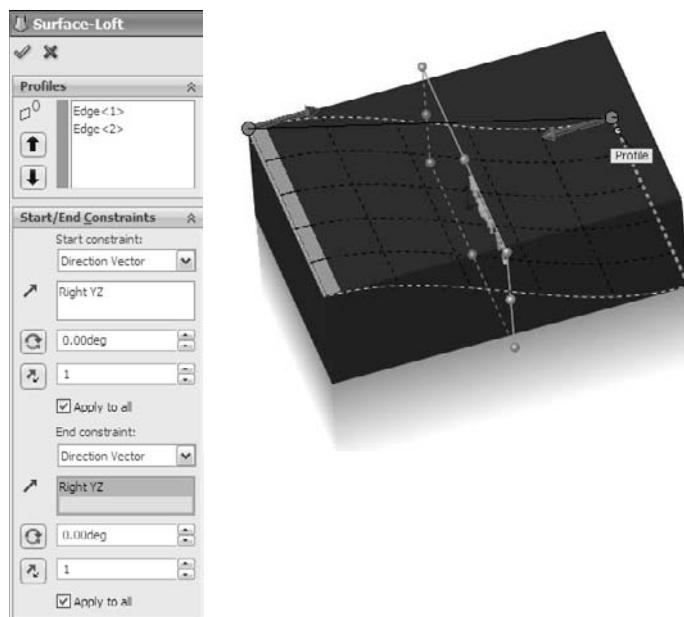
Using Cut With Surface

Follow these steps to gain some experience with the Cut With Surface feature:

1. **Start by creating a new part and drawing a rectangle on the Top plane, centered on the Origin, about 4 inches by 6 inches, with the 4-inch dimension in the vertical direction.**

2. **Extrude the rectangle Mid-plane, by 2 inches.**

3. **From the Surface toolbar, select Lofted Surface, and select one 4-inch edge as a loft profile.** Then select a second 4-inch edge diagonal from the first one. This is shown in Figure 27.22.

4. **Expand the Start/End Constraints panel, and set both ends to use the Direction Vector setting, selecting the plane that is in the middle of the long direction in each case.** In the part shown, the Right plane is used. Click OK to accept the feature. This is shown in Figure 27.22.

FIGURE 27.22

Lofting a surface from the edges of a solid

5. **From the menus, choose Insert ⇨ Cut ⇨ With Surface.** Select the surface from the fly-out FeatureManager, and toggle the arrow direction so that the top is cut off. (The arrow points to the side that is cut off.)

Using Offset Surface

Follow these steps to gain some experience with the Offset Surface:

1. **Open the part from the CD-ROM called** `Chapter 27 – Offset Tutorial.sldprt`.

2. **Right-click a curved face of the part and click Select Tangency in the menu.**

3. **With the faces still selected, from the Surfaces toolbar, click Offset Surface, and set the surface to offset to the outside of the part by .060 inches.** You can tell when the surface is offsetting to the outside when the transparent preview appears. If you do not see the transparent preview, toggle the Flip Offset Direction arrow button. Click OK to accept the feature when you are satisfied.

4. **Look in the Surface Bodies folder at the top of the FeatureManager tree, expand the folder, and select the Offset Surface.** Then use the Appearances toolbar button to change the transparency of the surface body to about.75. You can also do this through the Display Pane, by clicking in the column following the surface body in the bodies folder that is farthest to the right, as shown in Figure 27.23. This is done so that you can see the part underneath the surface, without mistaking the surface for the actual part.

FIGURE 27.23

Using the Display Pane to change transparency

Tip

It is a common practice to change surface colors to something that contrasts with the part color. I usually use a color like yellow, which suggests temporary status or construction. Some users take this a step further and set the template colors for surface types by choosing Tools ➪ Options ➪ Document Properties ➪ Colors. These settings do not always work; in some cases, they turn surface features to a different color, and in other situations, they do not. ∎

5. **Select Sketch2, and select Extruded Boss/Base from the Features toolbar.** Do not mistake the Extruded Surface for an Extruded Solid. Set the end condition to Up To Body, activate the body selection box, and select the Offset Surface body from the Surface Bodies folder. The result is shown in Figure 27.24.

FIGURE 27.24

Extruding with the Up To Body setting

Tip

It is preferable to select the surface from the Surface Bodies folder, rather than the feature list or the graphics window. In this case, you want to extrude up to a body. If you make the selection from the feature list, then you are likely to select a feature (which is okay in this situation, but not in all situations). If you make the selection from the graphics window, then the selection is likely to be interpreted as a face. It is best to be as explicit as possible when making selections because SolidWorks may interpret your selection literally. In this case, it is probably a better idea to use Up To Body for the end condition than Up To Surface, because the goal is really to use the surface body as the end of the feature. ■

> 6. **To invert the lettering so that it sits below the surface rather than above the surface, you can make a few simple changes.** First, edit the Offset Surface feature and flip the direction of the offset so that the surface is now inside the solid rather than outside the solid. You will not be able to see it unless the solid is either transparent or in wireframe mode.
>
> 7. **Next, delete the extrude that you created to extrude the text.** There is no way to change an extrude into a cut in this context.
>
> 8. **Re-create the extrude as an extruded cut.** Use the From settings at the top of the PropertyManager window. The settings and results are shown in Figure 27.25.
>
> Another way to accomplish this would be to use the Move Face tool, select the faces of the letters, and move them .120 of an inch into the solid.

FIGURE 27.25

An extruded cut

Using Fill Surface blend

Sometimes fillets do not meet your needs. Blends, such as those shown in the bike frame example, are smoother and can blend just about anything. However, the technique is not exactly straightforward. Follow these steps to gain familiarity with this technique:
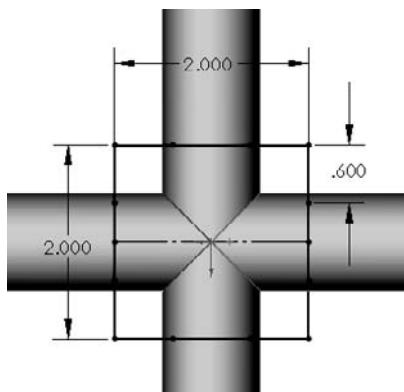
1. **Open the part from the CD-ROM for Chapter 27 called** `Chapter 27 – Blend. SLDPRT`**. Box select all the features from the DeleteFace1 to the Shell, and suppress them.**

2. **On the Top plane, draw a square 2 inches on a side and centered on the Origin.**

3. **Use the Split Entities tool found on the Sketch toolbar or choose Tools ⇨ Sketch Tools ⇨ Split Entities from the menus.** Divide each line of the rectangle into three pieces, with the two outer pieces of each line being .6 inches (use an Equal sketch relation). The sketch should be fully defined when you are done. This arrangement is shown in Figure 27.26. This is done because the edges of the tubes need to be broken into sections.

4. **Use Delete Face to delete the ends of the four tubes.** Set the option to Delete, not the default option of Delete and Patch. This converts the solid into a surface body.

5. **Use the sketch with the split entities to trim out the center section of the tubes, keeping the outer section and leaving four surface bodies.** This divides each tube end into four segments where they have been trimmed, as shown in Figure 27.27.
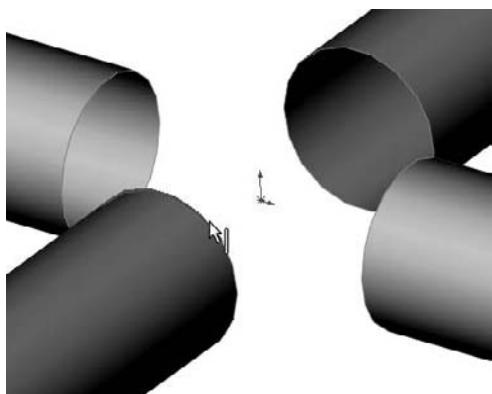
FIGURE 27.26

Using split entities to split lines

FIGURE 27.27

Split ends after trimming
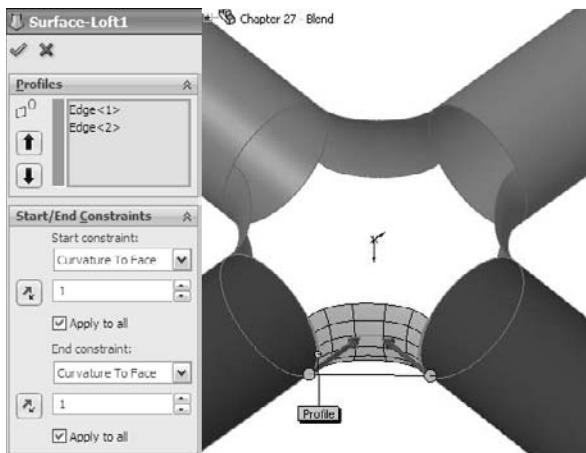
6. **Initiate the Lofted Surface feature, and select the nearest edge segments from adjacent tubes.** If the loft preview twists, use the light-blue handles to straighten it out or deselect and reselect one of the edges in approximately the same location as the other edge was selected. Expand the End Conditions panel and set each edge to use the Curvature setting. You may adjust the End Tangent Length option if you want, but keep in mind that this may make the part asymmetrical.

As a note, you may choose to use the Boundary Surface in the place of the loft. For this function, the two are similar enough.

7. **Create Lofted Surfaces all the way around the part, linking all the tubes.**
    Figure 27.28 shows the part with three of the lofts already completed and the last one in
    progress.

Adding Lofted Surfaces

8. **Start a Planar Surface feature, and select the open ends of each tube where the faces
    were deleted in Step 4.**

Note

Not all features allow you to operate from multiple bodies, but the Loft and Planar Surface features do.
Features such as Fillet and Draft restrict you to creating features that are associated with one body at a time. ∎

9. **Start a Knit Surface feature, and Shift+select all the bodies in the Surface Bodies
    folder (select the first body in the list and Shift+select the last body).** When you
    click OK to accept the feature, notice that the number of surface bodies changes to one.
    Selecting bodies in this way is much faster for large numbers of bodies than selecting
    them one at a time from the graphics window.

Note

Notice that the open edges of the surface body are shown in a different color. At this point, there are two open
edges around the holes at the intersection of the tubes. ∎
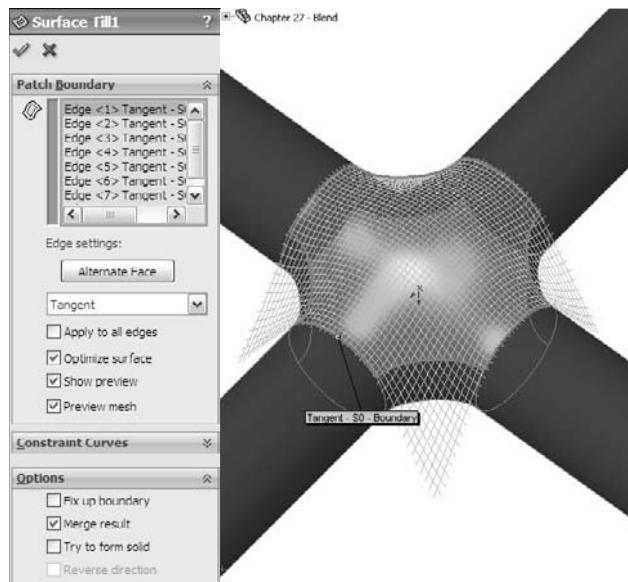
10. **This is a situation that the Fill Surface is really meant for.** In fact, this technique was
    created specifically to take advantage of the Fill Surface capabilities. Right-click any of the
    open edges and choose Select Open Loop. Initiate the Fill Surface. Change the Edge set-
    ting option to Tangent, and make sure that the Apply to all edges option is selected.

Select the Merge result option, but leave the Try to form solid option unselected. The model at this point is shown in Figure 27.29, along with the PropertyManager settings that are used.

Creating a Fill Surface patch
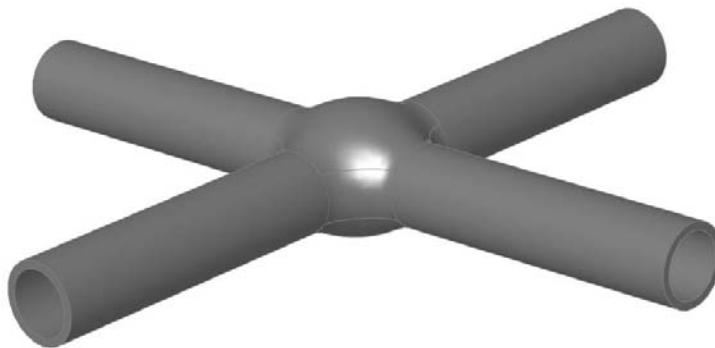
Note

The Optimize surface option is ignored for this part because the opening is eight-sided rather than four-sided. Also note that you may have to change the resolution control slider to get the surface to remain convex instead of going concave in the center. ■

11. **Click OK to accept the feature.**

12. **Start another Fill Surface, turning the part over to use the same selection on the back and the same settings as the first fill.** However, on this one, also use the Try to form solid option. Click OK when the selections and settings are complete.

13. **For the last feature, apply a Shell feature, selecting the flat ends of the tubes and shelling to .100 inches.** The final state of the model is shown in Figure 27.30.

The finished model

Summary

Surface functions have a wide range of uses other than for complex shape parts, but thinking about your models in terms of surface features requires a slightly different approach. Becoming comfortable with the terminology, and the similarities and differences between solids and surfaces, is the first step toward embracing surfacing tools for everyday work.

You can think of surfaces as being reference geometry — stand-alone faces that you can use to complete various tasks.

Employing Master Model Techniques

I n this book, the term *master model* is used to refer to a technique where an entire assembly is laid out or has its major faces constructed in a single part, and that part is then placed into other files from which the individual parts are created. Master model techniques are usually used in situations that in-context design cannot deal with, or where in-context design is cumbersome.

Master model techniques are a product of four separate features or functions that have some similarities and some differences, and rely heavily on the knowledge of parent/child concepts, multiple bodies, and surface functions. The four features are Split, Save Bodies, Insert Part, and Insert Into New Part. In turn, these four features can be categorized into Push and Pull type functions.

As an example of a master model technique, consider the mouse model shown in Figure 28.1, which should be familiar by now. The overall shape is modeled as a single part, and is split into several bodies using multibody methods. Then, using these master model techniques, the individual bodies are used to create individual part files where detail features are added.

SolidWorks has four distinct features or functions that do essentially the same thing with subtle but important differences. The functionality of the four tools overlaps significantly, but none is an exact copy of another.

Understanding the concepts of *parent* and *child* documents is key to understanding the concepts behind master model techniques. A parent document is always the driving document — the one that existed first — so changes to the parent propagate down to the child. The child document is always dependent upon the parent. In these master model schemes, it is not always possible to find the child document from the parent, but you can always find the parent from the child.

FIGURE 28.1

A mouse master model

The concepts of *Push* and *Pull* type functions are ones that I developed and you may not find them in other documentation. I found classifying the techniques helpful in understanding which tool was best for various situations. Push simply means that data from the parent document is pushed out to the child and the relationship is defined in the parent document. Pull means that the child document pulls data from the parent and the relationship is defined in the child document.

Here is a quick summary of the four tools that this chapter covers as master model tools:

- **Insert Part.** Enables you to pull all the solid and surface bodies, sketches, and even features from an existing part into the current part. Available as a toolbar icon and from the Insert menu.

- **Insert Into New Part.** Enables you to insert a selection of solid and surface bodies from the current part into a brand new part. Even though it is initiated from the parent document, it is classified as a Pull function because it doesn't leave a feature in the parent, but does leave one in the child.

- **Split.** Enables you to split a single solid body into multiple solid bodies and save (push) each body out to a separate part file. Available as a toolbar icon and a menu entry in the Insert ➪ Features menu. Creates a feature in the FeatureManager of the originating (parent) part file.

- **Save Bodies.** Enables you to save (push) all the solid bodies from a part out to separate part files. Available only via the right mouse button (RMB) menu on the solid bodies folder. Does not create a feature in the FeatureManager of the parent part.

The one common weakness of all these tools is the file management side, or more precisely, the body management side. It boils down to a question of what happens to the child document if you rearrange the bodies in the parent document. There are a number of ways body management issues can come up. The Insert Part feature is the one that has received the most attention from SolidWorks when it comes to the robustness of file and body management issues, but Insert Part still does not cover all the functionality. (You cannot insert selective bodies; you must insert all solids or all surface bodies.)

Using Pull Functions

Pull functions are initiated from the child document and pull data from the master model (parent document) into the child document. These functions insert a feature into the child that points to the parent, but do not insert a feature into the parent that points to the child.

The features that fall into this category are Insert Part and Insert Into New Part.

Insert Part

You initiate Insert Part from the child document by choosing Insert ⇨ Part from the menus or clicking the Insert Part button in the Features toolbar (which may not be on your toolbar by default). As the name suggests, this feature pulls one part into another. Insert Part gives you the option to bring forward all solid and surface bodies, planes, axes, and sketches in addition to other options. You can even break the link between the inserted part and the parent data. This simply copies all the sketch and feature data into the current part. The Mirror Part feature also uses this same PropertyManager with the same options. The Insert Part PropertyManager interface is shown in Figure 28.2.
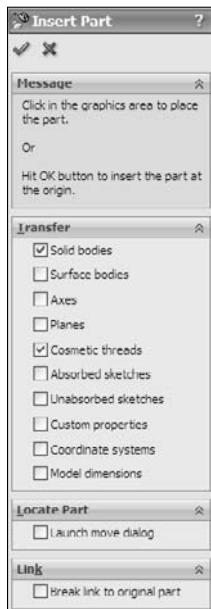
Figure 28.3 shows the FeatureManager of a part where the only feature is an Insert Part feature. All the solid bodies are listed under the normal Solid Bodies folder as well as under a second Solid Bodies folder under the inserted part icon. Other inserted items, such as surface bodies, planes, sketches, and axes, are also listed in folders under the inserted part icon.

FIGURE 28.2

The Insert Part PropertyManager
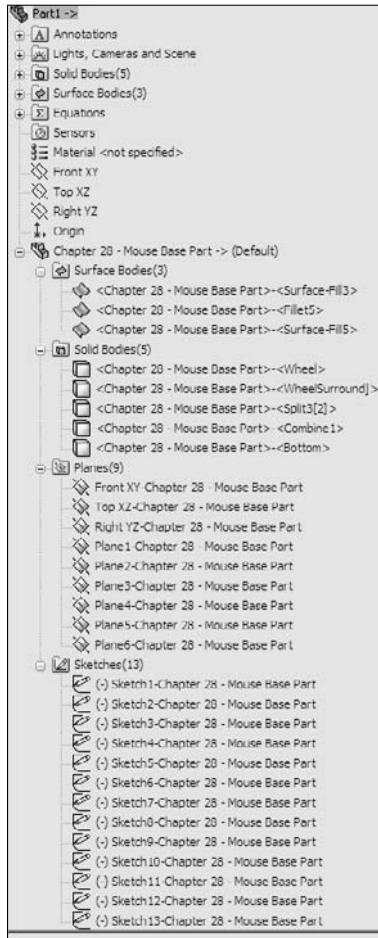
You cannot be selective about which bodies are pulled forward, but you can delete unwanted bodies once they have all been brought in (using Delete Solid/Surface). If you are trying to handle data efficiently, this may not be the best option for you. Because you have to first bring forward all the bodies and then delete those you don't want, the body data is still stored inside the part. Remember that the Delete Bodies feature does not actually delete anything. It simply makes it inaccessible after a certain point in the part history. If you are inserting a part with many complex bodies, you may want to use a more selective method such as Insert Into New Part or Save Bodies, each of which I describe in more detail later in this chapter.

FIGURE 28.3

The FeatureManager showing items inserted with an inserted part
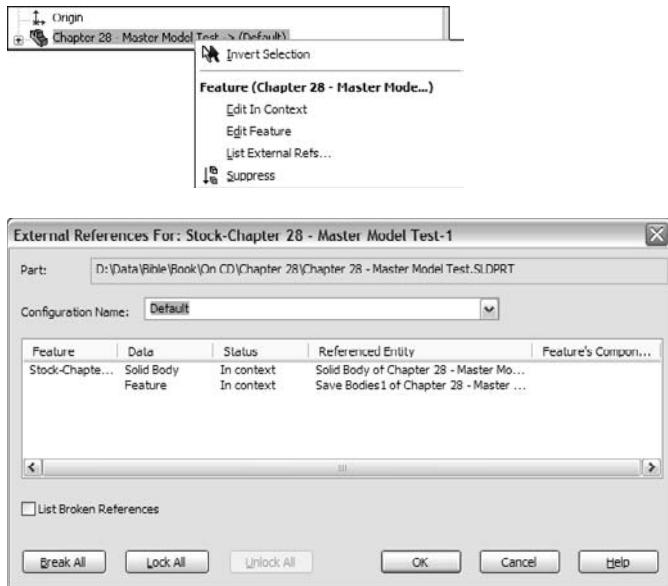
For the inserted part, you can set configurations of the parent document in the External References dialog box, which is available through the RMB menu of the feature that is inserted into the child document FeatureManager. The External References dialog box is shown in Figure 28.4. If the overhead of bringing many bodies forward only to be deleted is an issue for you, then you can use parent part configurations to delete the bodies first; then, from the child using List External References, you can select which configuration to insert.

FIGURE 28.4

The External References dialog box

As the original inserted bodies are modified by additional features in the child document, the names change, and they are removed from the folder under the inserted part and only appear in the body folders under the top level.

File management is a real issue with all these master model functions; in fact, it may not be an exaggeration to say that it is the *biggest* problem that arises with them, although you could say the same thing about overall body management. It is safe to say that you should be careful and follow file management best practice recommendations when performing name changes for documents with external references, especially if they use any of these features.

Insert Into New Part

Insert Into New Part qualifies as a Pull function because it does not create a feature in the FeatureManager of the parent file, even though it is actually initiated from the parent rather than from the child. This function does not have a drop-down menu location, nor does it have a toolbar button. You can initiate it from the RMB menu from either the Solid or Surface Bodies folders or from the individual bodies within the folders.
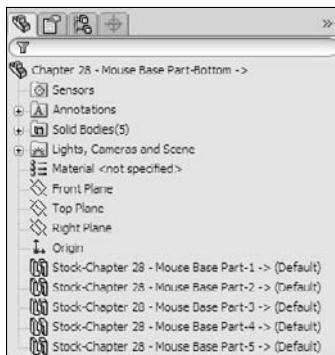
This gives Insert Into New Part both advantages and disadvantages when compared to Insert Part. The advantages are that it can *selectively* insert *either* solid or surface bodies, *or even selections of both types*. You can Ctrl+select multiple bodies (solid and/or surface) to bring only the bodies forward

that you need. However, it cannot bring forward planes or axes, or change the selection of what is brought forward after the feature is created. It also cannot be used to add a body to an existing part file; it can only be used to create *new* documents. This is definitely a good news/bad news situation, but with this information, you can make a more informed decision about which function to use.

When Insert Into New Part is used to place bodies into a part, the bodies are not shown in the same way that the Insert Part function shows them. Figure 28.5 shows that the Stock feature symbols are used rather than the Inserted Part symbol given individual bodies are being placed rather than entire parts.

Bodies placed in a part using Insert Into New Part

One note about this feature is that if it loses its referenced bodies, they cannot be reattached. An implication is that you cannot intentionally replace a body. Neither situation (lost references or the need to replace bodies) should come up frequently, if at all, for most users.

Using Push Functions

Push functions are initiated from the master model (parent document) and push data from the parent part out to a child part. A feature in the tree of the parent identifies the point at which the model is pushed out to the child, and the child file can be found from the parent.

The first feature in the child part is a Stock feature, and contains a reference back to the parent, so that the parent can be found from the child. The features that fall into this category are Split and Save Bodies. The bidirectional identification of the source and target of the feature holds a distinct advantage over the Pull functions, which do not allow you to identify the child from the parent document.
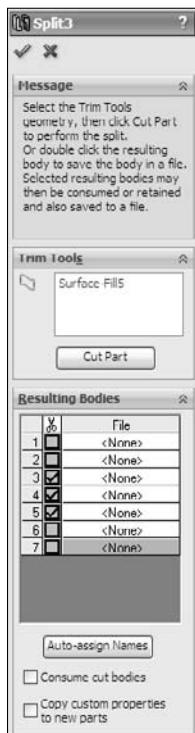
Split feature

I have already mentioned that the Split feature has three functions, two of which are plainly visible and one that is hidden. The Split PropertyManager is shown in Figure 28.6.

The Split PropertyManager

Split

The primary function of the Split feature is to split a single solid body into multiple bodies. You do this with sketches, planes, or surface bodies, and I discuss it in depth in Chapter 26. The Split feature can save both the preexisting bodies and any bodies that result from the split as individual part files using the Stock feature as the initial feature in the part.

Auto-assign names

The ability to save solid bodies out to part files directly from the lower half of the Split PropertyManager has caused some serious file management problems in versions previous. In previous versions, if you add features to a part created by saving a body out from the Split feature,

making changes to the Split feature could wind up overwriting the file with the additional features. Also changing the number of bodies going into the Split feature usually had bad effects on down-stream data. Still some changes to the number of bodies (such as reducing the number of bodies, and then increasing it manually, undoing the original change) can still result in unpredictable behavior.

Create Assembly

After you create the Split feature and you save bodies out of the Split feature, the RMB menu displays the Create Assembly option, which puts all the parts from the bodies in the original part back together in the correct positions. Parts within the assemblies created this way are fixed in space with the same relationship to the assembly origin that they originally had as bodies to the part origin.

Save Bodies

To access the Save Bodies feature, either right-click on the bodies folders and select it from the RMB menu or choose Insert ⇨ Features ⇨ Save Bodies from the menus. Save Bodies works by giv-ing path and filename information in the Resulting Bodies selection box of the Split PropertyManager. However, Save Bodies enables you to create an assembly right in the PropertyManager for the feature, rather than as a hidden feature with no record of the assembly name created by the feature. The PropertyManager for Save Bodies is shown in Figure 28.7.

FIGURE 28.7

The Save Bodies PropertyManager

Tutorial: Working with Master Model Techniques

Some of the concepts presented in this chapter may not make much sense until you apply them to a specific situation. The goal of this tutorial is to demonstrate the strengths and weaknesses of the various functions, as well as to give you some practical experience with the file management issues that you will encounter. You will use each of the four tools with the mouse multi-body part to become familiar with the differences in functionality. To get some experience with these techniques, follow these steps:

Insert Part
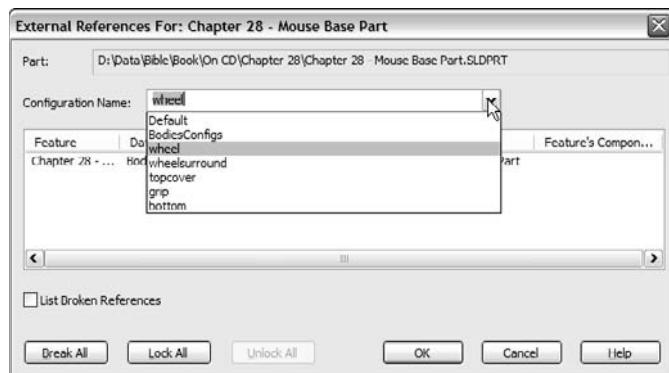
To work with the Insert Part function, follow these steps:

1. **Make sure that you have access to the material from the CD-ROM for Chapter 28.** Create a new part, and insert the part named `Chapter 28 – Mouse Base Part.sldprt`. You can access Insert Part by choosing Insert ➪ Part from the menus. After issuing the command, SolidWorks attaches the part to your cursor and prompts you to specify a location for the inserted part in the PropertyManager. Drop the part at the Origin of the child part, or simply click the green check mark to accept the part. There is no need to use the Move dialog box; if it appears, then deselect the option that enables it. For this part, do not transfer any of the optional items, only the solid bodies.

2. **After the feature is accepted and in the tree, right-click it and select List External Refs from the menu.** The External References dialog box is shown in Figure 28.8, with the list of configurations displayed.

FIGURE 28.8

The External References dialog box

3. Select the Wheel configuration from the list.

4. Save and name the part file in such a way that it has the name of the technique used to create it (Insert Part) and the name of the body that it represents (such as Wheel).

Note

I have done little bit of preparation work to make this tutorial flow more smoothly. If you ever choose to do modeling in this way, then you will need to know what this preparation work entails. In the mouse master model, I made a separate configuration for each body, and in that configuration, I created a Delete Body feature that deleted all the bodies except one. The alternative to this approach is to bring all the bodies into each new part, and use a Delete Body feature in each child part that deletes all but the one body that is needed. The advantage to using configurations is that bringing in a single body theoretically decreases the overhead for the individual part files. ∎

5. Repeat Steps 1 through 4 for each of the five bodies in the master model.

6. **If the mouse master model (`Chapter 28 – Mouse Base Part.sldprt`) is open, then close it.** In any of the child parts, the inserted part feature shown in the tree should have the Out Of Context symbol on it (- >?). Right-click the inserted part feature and select Edit In Context, which opens the master model.

   Notice that from the master model, you have no way of knowing where the child parts are or even *if* any child parts exist. Notice also that there is no easy way to create an assembly.

7. Create a new assembly document.

8. **Drop all the individually created parts into the assembly by selecting them in Windows Explorer and dragging them onto the assembly Origin.** This is probably the easiest way to create an assembly using the Insert Part feature.

Note

There is no link from the parent to the child; if the child part is renamed, the parent will not lose track of it. However, there is a link from the child to the parent; if the parent is renamed without the child being open at the same time, the child loses track of the parent. If the parent is changed, the child does not update unless the symbol is showing In-Context (->). If it is out of context, broken, or locked, the child does not update with the parent. Both documents need to be open at the same time to make the update happen (although they do not both need to be open when the original edit happens to the parent master model). ∎

9. Save and close all the parts and assemblies.

Insert Into New Part

To work with the Insert Into New Part function, follow these steps:

1. **For this feature, start from the master model. Open the part** `Chapter 28 – Mouse Base Part.sldprt`. Make sure that the part is set to the Default configuration. If it is set to a different configuration, inserting bodies will not go as smoothly as it could.

2. **Expand the Solid Bodies folder in the FeatureManager.** Right-click the first body in the list (Wheel), and select Insert Into New Part from the menu.

Note

You could select multiple bodies and even combine solid and surface bodies to insert using this technique. ∎

3. **When prompted, name the new part using the same convention used in the previous tutorial, which was to use the name of the technique (Insert Into New Part) and the name of the body.** In this part, leave the configuration setting in the External References dialog box to the Default configuration.

4. **Repeat Steps 1 through 3 for each of the bodies.**

5. **Right-click the Stock feature in the tree, and select Edit In Context.** SolidWorks opens the master model part.

Note

Once again, there is no way back to the child document from the master model using the Insert Into New Part feature. ∎

6. **Create a new assembly document and use the same technique from the previous tutorial to put all the parts in the assembly located from the Origin.** Again, no automated assembly creation tool exists for this method.

7. **Save all documents and close them.**

Split

To work with the Split function, follow these steps:

1. **This time, start from a copy of the master model part.** The Split feature makes additions to the model, and because you have already created assemblies based on the original, any additional features should be created using a copy of the part rather than the original. Copy it using the Copy and Paste feature in Windows Explorer, and rename the copy as `Chapter 28 – Split Tutorial`.

Note

It is best to copy and rename this document before continuing with the rest of the tutorial. Otherwise, you may encounter problems with the file references, from which it is difficult to recover. ∎

2. **With the newly copied and renamed document open, initiate the Split feature by choosing Insert ⇨ Features ⇨ Split.**

3. **Because the bodies already exist, there is no need for the Trim tools or Cut Part functions in the Split feature, only for the resulting bodies.** To save the bodies to individual files, you must give each one a unique name. You can click the Auto-assign Names button to automatically name them with the existing names of the bodies. It might be difficult to discern where the callout flags are pointing. Once the names are all satisfactory, click OK to accept the feature.

Note

The Consume Cut Bodies option deletes the bodies of any bodies involved in the Split feature. For most purposes, you should deselect this option. Deselecting the option makes sure that the bodies are still available after the Split feature. If what you are really looking for is to eliminate the bodies once they are saved out, then you should select the Consume Cut Bodies option. ■

4. **To automatically create an assembly with all the components located in the proper location, right-click the Split feature in the Master Model FeatureManager, and select Create Assembly.** Multiple Split features can be included in this command if bodies have been created by multiple Split features. Click the Browse button to locate and name the new assembly. Click OK when you are done. Completing this step opens the assembly that you just named and located. When you create the assembly, the parts will show up visually but may not be displayed in the FeatureManager until you have saved and reopened the assembly file. You should still have access to the data through the RMB menu from the graphics window.

5. **Right-click one of the parts in the assembly to open it.** Notice that a Stock feature is used in the tree, so it is possible to access the parent part and change the parent part configuration used in the current part. Right-click the Stock feature and select Edit In Context.

6. **With the master model open, right-click the Split feature and select Edit Feature.** From here, it is possible to see where each of the child parts is located.

7. **If you rename any of the documents, then you should do this either by using SolidWorks Explorer or the Save As command with the other documents open as well.** If you want to rename the parent part (master model), then make sure that all the child parts are open as well. (You can easily do this by opening the assembly; although the assembly was created from the master model, there is no direct link between the Split feature and the assembly.)

8. **Save and close all the files before proceeding.**

Save Bodies

To work with the Save Bodies function, follow these steps:

1. **As before, create a copy of the original master model part and rename the copy** `Chapter 28 – Save Bodies Tutorial`.

2. **Open the renamed copy, and right-click the Solid Bodies folder.** Select Save Bodies from the menu. (Save Bodies has its own icon, which looks like the Split icon and is used to denote the placeholder feature in the FeatureManager.)

3. **The Save Bodies PropertyManager is nearly identical to the lower section of the Split PropertyManager. Use it to save the solid bodies out to separate files.** The major addition in the Save Bodies dialog box is that of the Create Assembly function directly within the PropertyManager. The primary benefit of this addition is that it retains the name and path of the assembly in this interface so you can look it up later if necessary, remedying one of the weaknesses in the Split feature.

Note

In both the Split/Create Assembly and Save Bodies features, when an assembly is created, SolidWorks may rebuild the tree of the part as many times as you have bodies to save out. This may take some time for a complex model with a lot of bodies. ■

4. **After finishing Step 3, open the reconstructed assembly.** Right-click one of the parts and select Open Part to open it in its own window. Notice that the Stock feature has again been used to push a single body into the part.

5. **Right-click the Stock feature and select Edit In Context, which takes you back to the master model.**

6. **Save and close all of the files.**

Summary

Each of these four functions has strengths and weaknesses. Because of this, there is no one feature that is clearly superior to the others in all respects. Most of the weaknesses have to do with the types of data that they can work with, the ease of creating an assembly, and access to children from the parents. Unfortunately, I cannot offer a single solution that addresses all the problems and retains each of the strengths.

The most important strengths, in my view, are the Insert Part and Insert Into New Part functions' capabilities to deal with both solids and surfaces. If you are working with surface bodies, you are forced into using one of these two functions. In particular, the Insert Into New Part function enables you to be selective about which bodies to pull into the child document. The other most important strength belongs to the Save Bodies feature, which makes the child accessible from the parent and identifies the assembly in the parent.

Why the various strengths of these tools cannot be consolidated into one or at most two different features or even how it's gotten to the point where so many different functions do approximately the same thing is a mystery to me.

Part VII

Working with Specialized Functionality

Specialized functionality in SolidWorks makes certain tasks easier by grouping tools and making tools that apply only to a limited range of techniques. Sheet metal and Weldments are highly useful techniques in SolidWorks due to the specialized tools developed around the design styles. Chapter 30 is new in this edition and addresses the direct editing tools that you can use in SolidWorks to edit native and imported files. Chapter 32 covers plastic functionality, which is very powerful and robust. Mold Tools enable you to add a draft to plastic parts and split it into cavity and core faces. MotionManager is an ever-growing area of the software and is covered in Chapter 33.

Using SolidWorks Sheet Metal Tools

S olidWorks contains two completely separate methods for working in sheet metal. In one method you can use dedicated sheet metal features from the start, and in the other method you build a part using thin features and other generic modeling tools, and then tell SolidWorks it is sheet metal so you can flatten it.

The reason for two methods is that the generic modeling method came first, and then SolidWorks introduced a more powerful set of dedicated sheet metal features. You can use these tools together or separately, and either way you get an accurately flattened part at the end.

Sheet metal tools do not always represent real-world sheet metal manufacturing processes 100 percent accurately because some shapes that result from bending processes are too complex to easily represent in a CAD model. So there are times when you still have to use your imagination a little bit. The main point is that the Flat Patterns are always accurate because sheet metal is usually fabricated using 2D data.

Using the Base Flange Features

The features used in the Base Flange method are easy to grasp conceptually, and they have many individual controls. These are the tools that represent the newer method of building sheet metal parts from dedicated sheet metal features. You can edit many of the features by pulling handles, by using spin arrows, or by typing in specific numbers or dimensions. Maybe best of all, SolidWorks knows to change the thickness for the entire part at once.

You can access the Sheet Metal features by clicking the tool you need in the Sheet Metal toolbar or by choosing Insert ⇨ Sheet Metal from the menus, and selecting the appropriate tool.

Base Flange/Tab feature
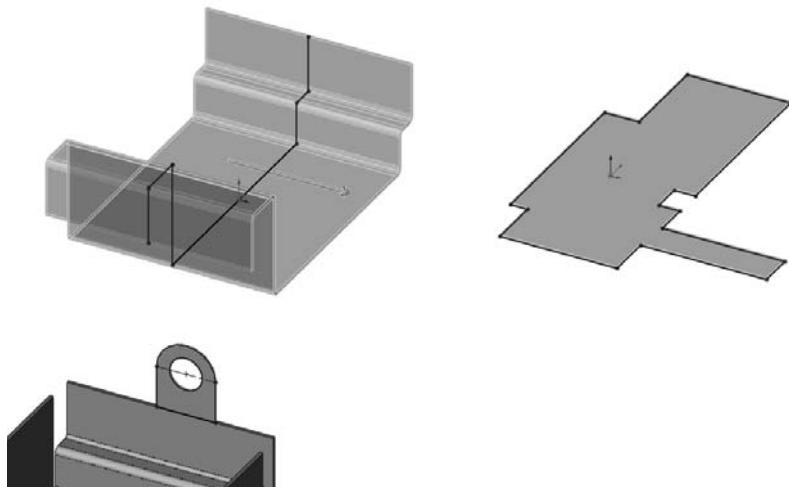
In addition to letting SolidWorks know that the part is a dedicated sheet metal part, the Base Flange/Tab tool has three functions:

- By drawing an open contour in the first feature, the Base Flange creates a thin feature-like extrusion that includes the rounded corners of the bends.

- By drawing a closed contour in the first feature, the Base Flange creates a flat sheet that is shaped like your sketch for you to start from.

- When the Base Flange is used at any time other than the first feature, it functions as a tab.

Figure 29.1 shows these three functions of the Base Flange/Tab feature.

FIGURE 29.1

The three functions of the Base Flange/Tab feature

Notice that the sketch of the part shown in preview in Figure 29.1 has all sharp corners, and that the bend radius is automatically added to each corner by the software. SolidWorks automatically adjusts when bend directions are combined to make sure that the inside radius is always the same, regardless of bend direction.

The bends are shown as BaseBend features in the FeatureManager. You can change individual bend radii from the default setting by editing the BaseBend feature, as well as by assigning custom bend allowances on a per-bend basis. You cannot change the bend angle for these particular bends because the angle is controlled through the sketch. However, for other types of bends (such as those created by Edge Flanges), you can adjust the bend angle through the feature PropertyManager.

If you need to, you can reorder all the bends from a list that you can access from the right mouse button (RMB) menu selection Reorder Bends on the Flat Pattern. This dialog box is shown in Figure 29.2.

FIGURE 29.2

The Reorder Bends dialog box

The BaseBend features can be suppressed, but the only effect that this has is to prevent the associated bend from flattening when the Flat Pattern feature is unsuppressed.

Sheet Metal feature

The FeatureManager is shown for the Base Flange with all the bends in Figure 29.3. The Sheet-Metal1 feature is automatically added to sheet metal parts as a placeholder for default sheet metal settings such as material thickness, default bend allowance settings, and Auto Relief options, as well as the default inside bend radius.

Gauge Table

Gauge Tables are a legacy table type, which is simply an Excel spreadsheet. In SolidWorks 2009, the data from gauge tables was consolidated with data from bend tables. However, you can still use the legacy gauge tables. The point of consolidating gauge and bend tables is so that you don't need a separate gauge table for each K-Factor (or bend allowance or bend deduction).
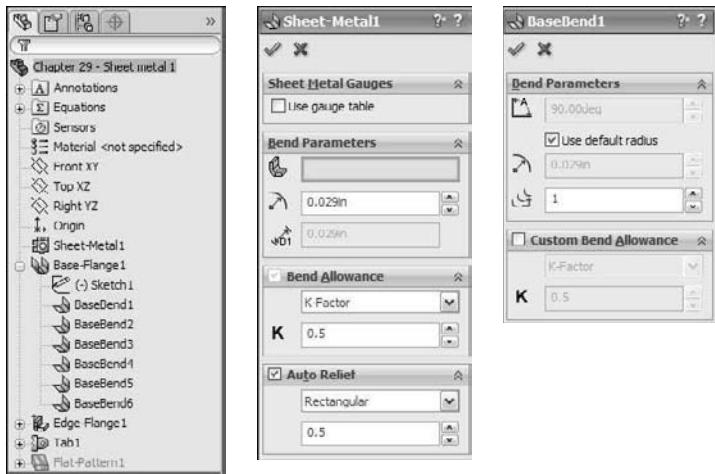
Cross-Reference

Bend tables are described in more detail later in this chapter. ∎

FIGURE 29.3

The FeatureManager after the Base Flange is added

Gauge tables enable you to assign a thickness and available inside-bend radii, which limits the choices that the user has for those settings in the table. Each K-Factor has a separate table, and the choices listed in the table appear in the drop-down lists in the Sheet Metal PropertyManager. Figure 29.4 shows the top few lines of a sample Gauge Table and a Sheet Metal PropertyManager when a Gauge Table is used.

If necessary, you can override the values that are used in the Gauge Table by using the override options in the thickness, bend radius, and K-Factor fields of the PropertyManager.

The Bend Allowance options (Allowance, Deduction, and K-Factor) are explained in more detail later in this chapter.

On the CD-ROM

Several sample tables with both gauge and bend data are provided on the CD-ROM that accompanies this book. ∎

Bend Radius

This option specifies the default inside bend radius for all bends in the part. You can override values for individual bends or individual features.

FIGURE 29.4

A sample Gauge Table and Sheet Metal PropertyManager

Thickness

The part thickness is grayed out in the Sheet Metal PropertyManager. You can change the value by double-clicking any face of the model. The thickness displays as a blue dimension rather than a black dimension. It is easier to identify if you have dimension names selected, because it is assigned the link value name Thickness.

All features in sheet metal parts that use the thickness value use a link value to link all the feature thicknesses. This makes it easy to globally change the thickness of every feature in the entire sheet metal part.

To save these settings to a template file, you can create a Sheet Metal feature, specify the settings, delete the Sheet Metal features, and then save the file to a template with a special name that represents the settings that you used.

Tip

When a link value is named Thickness, the Extrude dialog box always shows a Link To Thickness option to link the depth of an extrusion to the Thickness link value. If you save a template where Thickness has been created as a link value, then the option is always available to you, regardless of whether or not you are making sheet metal parts. ∎

Bend Allowance

You can control the Bend Allowance by using one of four options:

- Bend Table
- K-Factor
- Bend Allowance
- Bend Deduction

Bend Table

Two general types of Bend Tables are available, text-based and Excel-based. The first few rows of each type of table are shown in Figure 29.5. Each table can use K-Factor, Bend Allowance, or Bend Deduction.

FIGURE 29.5

Sample text- and Excel-based Bend Tables

Sample Bend Tables can be found in the `lang\english\Sheetmetal Bend Tables` subdirectory of the SolidWorks installation directory. While the values may not be what you need, the syntax and organization is correct. You may want to contact your sheet metal fabrication shop to see what they are using for a table or equations.

Note

Data from gauge tables and bend tables have been consolidated, but both legacy types can still be read. ∎

K-Factor

When sheet metal is formed from a flat sheet, bending the metal causes it to stretch slightly on the outside part of the bend, and to compress slightly on the inside part of the bend. Somewhere across the thickness of the sheet is the Neutral Plane, where there is no stretching or compression.

This Neutral Plane can be at various places across the thickness, depending on the material, tooling, and process. The ratio of the distance from the inside bend surface, to the Neutral Plane, to the thickness is identified as the K-Factor, where .5 means halfway, 0 means on the inside face, and 1 means on the outside face. Typically, you can expect values between .5 and .3.
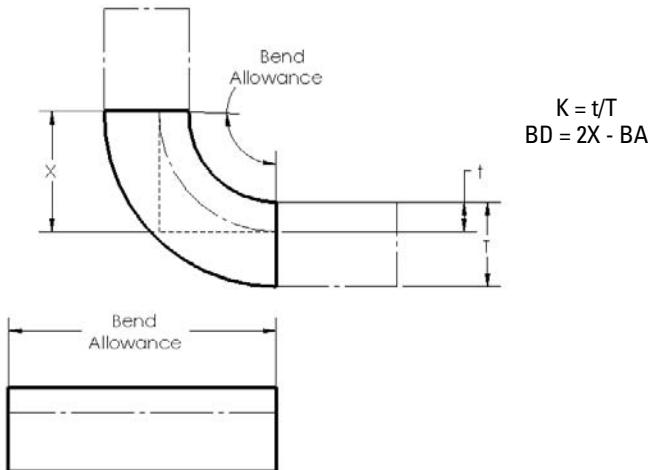
Bend Allowance and Bend Deduction

Bend Allowance and Bend Deduction are specific length values, not a ratio like the K-Factor. The Bend Allowance is essentially the arclength of the Neutral Plane through the bend region. The Bend Deduction is the length difference between a sharp corner and the radius corner, as expressed by the formula in Figure 29.6.

The three values are related, as shown in Figure 29.6. The dark rectangle represents the bend area. Material outside of the bend area really does not matter, although it is usually shown and used in the generally accepted formulas about bend calculations for sheet metal.

FIGURE 29.6

Calculating the Bend Deduction from the Bend Allowance and K-Factor

You usually use a ratio t/T (the K-Factor) from a published table or by asking your sheet metal vendor what values they typically use. The values from the tables have been developed experimentally by bending a piece of metal of known length, and then measuring the arclength of the inside of the bend and the arclength of the outside of the bend. By comparing these numbers to the original linear length of the bent area, you can find the *t* value and thus the *K* value. From the *K* value, the *BA* (Bend Allowance) value can be calculated and from that, the *BD* (Bend Deduction) value is easy to find.
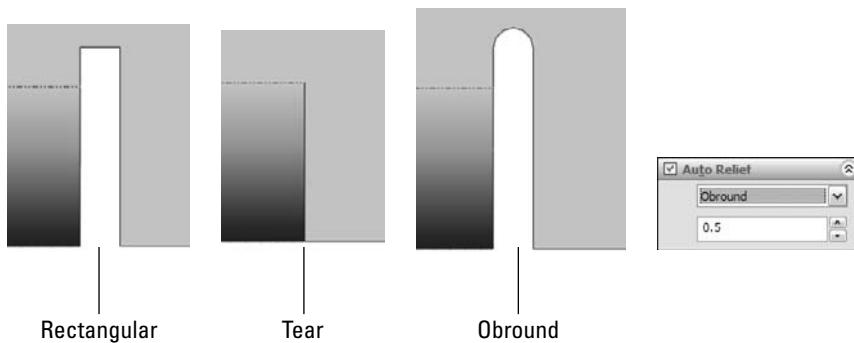
The specific formulas for finding these numbers are not as important as an intuitive grasp of what the numbers mean and how they are used, at least in relation to using SolidWorks to model sheet metal parts. The numbers used to fill out Bend Tables using K, BA, or BD values are typically taken from experimentally developed tables.

Auto Relief

Auto reliefs were formerly called Bend reliefs. You can specify three different Auto relief options to be applied automatically to bends that end in the middle of material. These options are illustrated in Figure 29.7.

FIGURE 29.7

The three Auto relief configurations: Rectangular, Tear, and Obround

Rectangular   Tear   Obround

For the Rectangular and Obround types, you can control the width and the distance past the tangent line of the bend through the Relief Ratio selection box, which is immediately below the type selection box in the Sheet Metal PropertyManager. This ratio is the width of the relief divided by the part thickness. For the Rectangular relief, a ratio of .5 and a thickness of .050 inches means that the relief is .025 inches wide and that it goes .025 inches deeper into the part beyond the tangent line of the bend. The Obround relief goes slightly deeper because it has a full radius after the distance past the tangent line of the bend, and so it essentially goes a total of one full material thickness past the tangent line.

The Tear relief is simply a face-to-face shear of the material with no gap.

Flat Pattern feature

The Flat Pattern feature is added automatically to the end of the tree when the Base Flange feature is added. This feature is used to flatten the sheet metal part when the feature is unsuppressed. The Flatten toolbar button acts as a toggle to unsuppress or suppress the Flat Pattern feature in the tree. It may be a little confusing, but the Flatten toolbar button and the flat-pattern feature in the FeatureManager refer to the same functionality. As mentioned earlier, the Flat Pattern has a couple
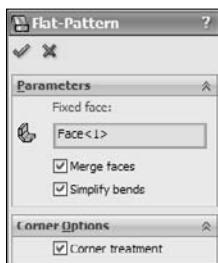
of special properties that are not seen in other features. The first is that it remains at the *bottom* of the FeatureManager when other Sheet Metal features are added.

The second property of the Flat Pattern feature is that it is added in the *suppressed* state. When it is unsuppressed, it flattens out the sheet metal bends.

By editing the Flat Pattern feature, you can set a few options. The Flat Pattern PropertyManager is shown in Figure 29.8.

The Flat Pattern PropertyManager

The Fixed face parameter determines which face remains stationary when the part is flattened out. Generally, the largest face available is selected automatically, but if you want to specify a different face to remain stationary, you can do that here.

When the Merge faces option is selected, it causes the Flat Pattern to form a single face rather than being broken up by the tangent lines around the bends. This does a few things. First, selecting the face of the flattened part and clicking Convert Entities (found on the Sketch toolbar) makes an outline of the entire flattened part, which is easier to use for certain programming applications. Second, the edges around the outside are not broken up. Third, the tangent edges around the bends are not shown. The differences between Flat Patterns with this option selected and unselected are shown in Figure 29.9.

Bend lines are shown in both examples in Figure 29.9.

When you turn select the Simplify Bends option, it simplifies curved edges that are caused by flattening bends to straight lines from arcs or splines. When the option is unselected, the complex edges remain complex. Simple edges can be cut by standard punches, and do not require Computer Numerical Control (CNC) controlled lasers or abrasive water jets.
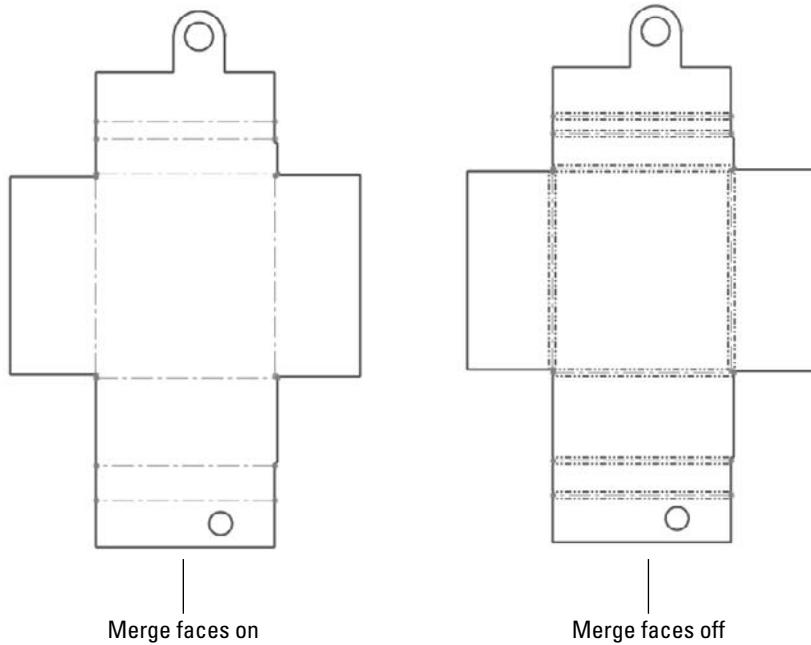
The Corner Treatment option controls whether or not a corner treatment is applied to the Flat Pattern of a part. The corner treatment is illustrated in Figure 29.10. The model used to create this corner used a Miter Flange around the edges of a rectangular sheet.
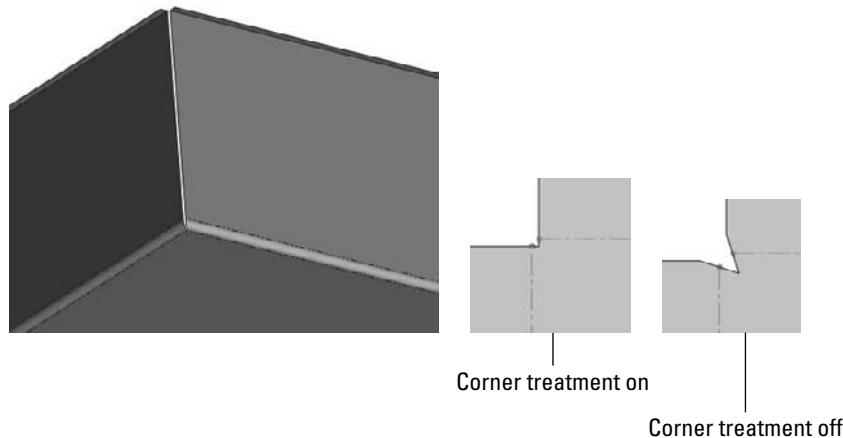
FIGURE 29.9

The Merge Faces option showing on (selected) and off (unselected)

Merge faces on   Merge faces off

FIGURE 29.10

Using the Corner Treatment setting in the Flat Pattern PropertyManager

Corner treatment on

Corner treatment off

Note

You can export a `*.dxf` file of the Flat Pattern directly from the model without creating a drawing. ∎

Edge Flange feature
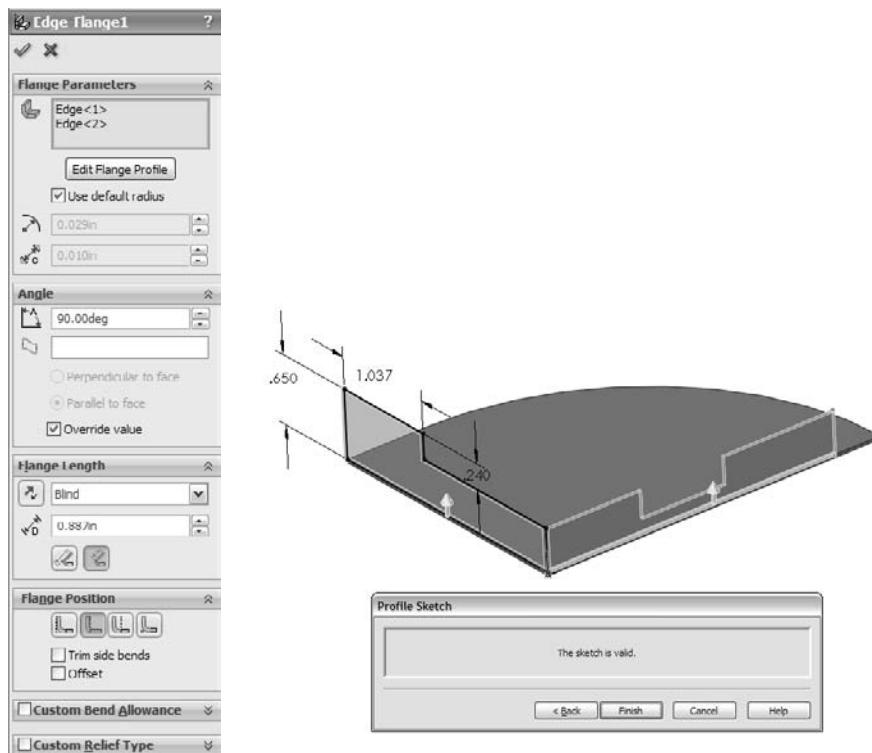
The Edge Flange feature is very flexible and can be changed in several ways. If you have not kept up with the changes to Edge Flange for the last couple of releases, then you may find some surprises.

Edge Flange is intended to turn a 90-degree flange from a selected straight edge in the direction and distance specified using the default thickness for the part. The default process for this feature is that you select the tool, select the edge, and then drag the distance, clicking a distance reference such as a vertex at the end of another flange of equal length or typing a distance value manually. You can select multiple edges from a part that do not necessarily need to touch one another. That is all there is to a simple default flange, although several options give you some additional options for angle, length, and so on. Figure 29.11 shows the Edge Flange PropertyManager, as well as a simple flange.

FIGURE 29.11

The Edge Flange PropertyManager and a simple flange

Edit Flange Profile

The Edit Flange Profile button in the Edge Flange PropertyManager enables you to edit a sketch to shape the flange in some way other than rectangular, or to otherwise edit the shape of the flange. Notice in Figure 29.11 that both of the flanges made by a singe flange feature have been edited. You can do this by selecting the flange for which you want to edit the profile before clicking the Edit Flange Profile button.

Note

If you have added dimensions to the sketch, as shown in Figure 29.11, then you will no longer be able to use the arrow to drag the length of the flange. To edit the length, you will need to edit the sketch or double-click the feature, and then double-click the dimensions that you want to change. ■

You can add holes to the flange profile as nested loops. This enables you to avoid creating additional hole features, but does not enable you to control suppression state independently from the flange feature.

You can make flanges go only part of the way along an edge by pulling one of the end lines back from the edge. This works even though the end lines appear black and fully defined. A situation where the sketch has been edited this way is shown in the image to the right in Figure 29.11.

Use default radius

This option enables you to override the default inside bend radius that is set for the entire part for this feature. The bend radii for individual bends within an Edge Flange that has multiple flanges cannot be set; the only override is at the feature level. If you need individual bends to have different bend radii, then you need to do this using multiple Edge Flange features.

Gap distance
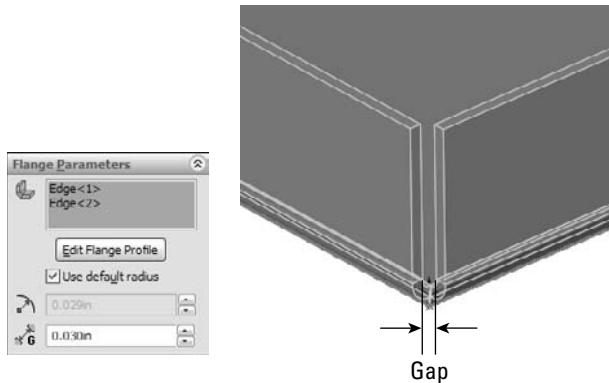
The gap distance is illustrated in Figure 29.12. The Gap Distance selection box is only active when you have selected multiple edges in the main selection box for this feature. The gap refers to the space between the inside corners of the perpendicular flanges.

Angle

Because the Edge Flange is not dependent on a sketch for its angle like the Base Flange is, you can set the angle in the Angle panel of the PropertyManager. The values that this selection box can accept range from any value larger than zero to any value smaller than 180. Of course, each flange has practical limits. In the flange shown in Figure 29.13, the limitation is reached when the bend radius runs into the rectangular notch in the middle of the flange to the right, at about 158 degrees. The angle affects all the flanges that are made with the feature. To create a situation where different flanges have different angles, you need to create separate flange features.
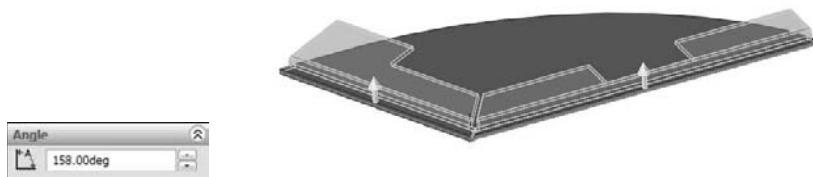
FIGURE 29.12

Specifying the gap distance

Gap

FIGURE 29.13

Establishing the limit of the flange angle

Flange Length

As mentioned earlier, if you have edited the Flange Profile sketch and a flange length dimension is applied in the sketch, then the flange length is taken from that sketch dimension. If this dimension has not been added to the profile sketch, then the options for this setting in the PropertyManager Flange Length panel are Blind and Up To Vertex. Using Up To Vertex is a nice way to link the lengths of several flanges.

Flange Position

The small icons for Flange Position should be fairly self-explanatory, with the dotted lines indicating the existing end of the material. The names for these options, in order from left to right, are

- Material Inside
- Material Outside
- Bend Outside
- Bend From Virtual Sharp (for use when an angle is involved)
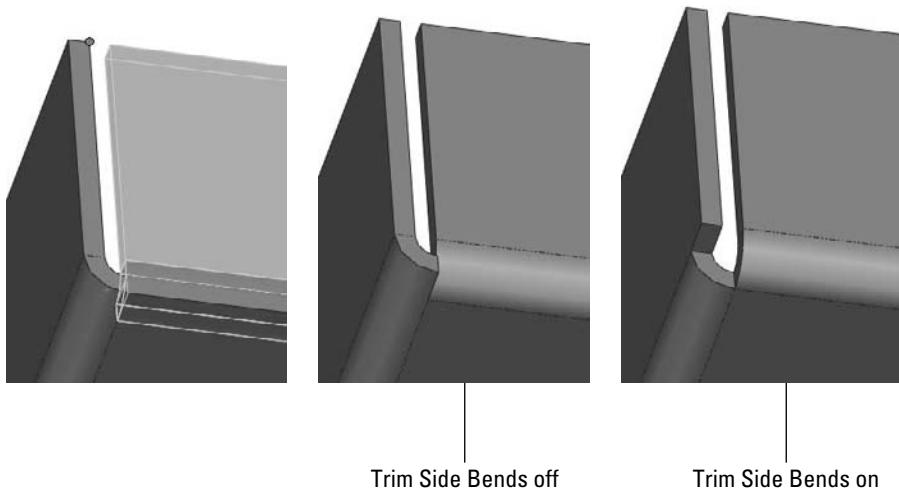
Trim side bends

In situations where a new flange is created next to an existing flange, and a relief must be made in the existing flange to accommodate the new flange, you can select the Trim side bends option to trim back the existing flange. Leaving this option unselected simply creates a relief cut, as shown in Figure 29.14. This is functionality that requires some imagination from the user. A real sheet metal part manufactured like this would have an area at the corner where the deformation from the bends in different directions overlaps. This overlapping bend geometry is too complex for SolidWorks to create automatically, so it offers you a couple of options for how you would like to visually represent the corner. The Flat Pattern is correct, but the formed model requires some imagination.

FIGURE 29.14

Using the Trim side bends option

Trim Side Bends off   Trim Side Bends on
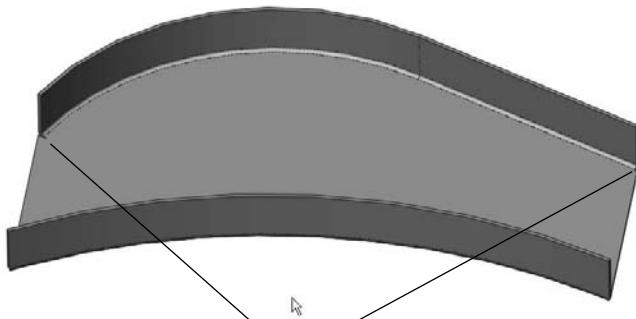
Curved edges

Edge Flanges can be created on curved edges, but the curved edge must be on a planar face. For example, if the part were the top of a mailbox, then an Edge Flange could not be put on the curve on the top of the mailbox. The flange would have to be made as a part of the flat end of the mail-box, instead.

Figure 29.15 shows Edge Flanges used on a part. Notice that reliefs are added to the ends of the bends, although they are not really needed.

FIGURE 29.15

Curved Edge Flanges on a part

Notice bend reliefs where they are not needed

All the edges that you select to be used with a curved Edge Flange must be tangent. This means that in Figure 29.15, neither of the Edge Flanges could have been extended around the ends of the part. You would need to create separate Edge Flange features for those edges.

Because these Edge Flanges are made in such a way that they are developable surfaces, they can be (and are) flattened in such a way that they do not stretch the material of the flange when the flat is compared to the formed shape. Doubtless there is some deformation in between the two states in the actual forming of this flange, and so its manufacturing accuracy may not be completely reliable.

Miter Flange feature

The Miter Flange feature can create picture frame–like miters around corners of parts, and correctly recognizes the difference between mitered inside corners and mitered outside corners. The PropertyManager and a sample Miter Flange are shown in Figure 29.16.

A Miter Flange feature starts off with a sketch that is perpendicular to the starting edge of the Miter Flange feature.

Tip
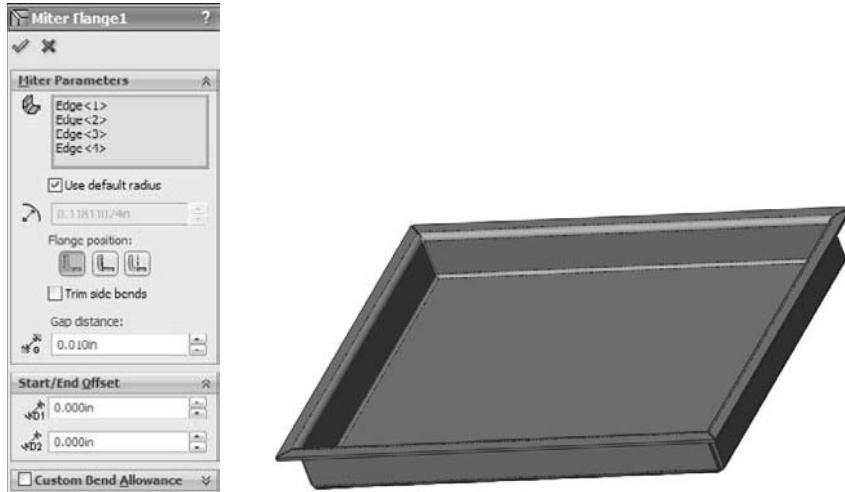
A quick way to start a sketch for a Miter Flange that is on a plane perpendicular to a selected edge is to select the edge, and then click a sketch tool. This automatically creates a plane perpendicular to the edge at the nearest endpoint. ∎

Miter Flange sketches can have single lines or multiple lines. They can even have arcs. Still, remember that just because you can make it in SolidWorks does not mean that the manufacturer can make it. It is often a good idea to check with the manufacturer to ensure that the part can be made. Also, you usually learn something from the experience.

FIGURE 29.16

The Miter Flange PropertyManager and a sample part

Tip

When selecting edges for the Miter Flange to go on, be sure to remain consistent in your selection. If you start by selecting an edge on the top of the part, then you should continue selecting edges on the top of the part. If you do not, then SolidWorks prompts you with a warning message in a tool tip that says that the edge is on the wrong face. ■

Some of the controls in the Miter Flange PropertyManager should be familiar by now, such as Use default radius, Flange Position, Trim side bends, and Gap Distance. You have seen these controls before in the Edge Flange PropertyManager.

The Start/End Offset panel enables you to pull a Miter Flange back from an edge without using a cut. If you need an intermittent flange, then you may need to use cuts or multiple Miter Flange features, as shown in Figure 29.17.

Hem feature
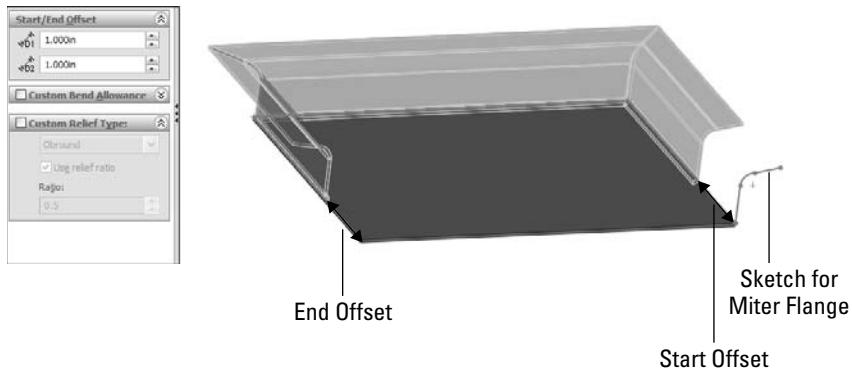
The Hem feature is used to roll over the edge of a sheet metal part. This feature is often used to smooth over a sharp edge or to add strength to the edge. You can also use it for other purposes, such as to capture a pin for a hinge. SolidWorks offers four different hem styles — Closed, Open, Tear Drop, and Rolled — which are shown as icons on the Hem PropertyManager. The PropertyManager for the Hem feature is shown in Figure 29.18.
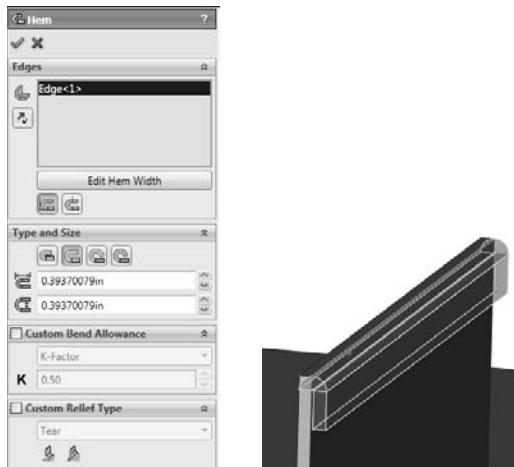
FIGURE 29.17

The Start/End Offset settings for a Miter Flange

End Offset

Start Offset

Sketch for
Miter Flange

FIGURE 29.18

The Hem PropertyManager and a sample hem

One of the limitations to keep in mind with regard to hems is that SolidWorks cannot fold over a part so that the faces touch perfectly line on line. Doing this would cause the two sections of the part to merge into a larger piece, thus removing the coincident faces. SolidWorks, computers, and mathematics in general do not always handle the number zero very well. In reality, you can often see light through these hems, and so a perfectly flush hem may not be as accurate as it seems.

You can edit the profile of the Hem, like an Edge Flange, to control the length of the edge that is hemmed. To do this, click the Edit Hem Width button below the Edges selection box in the Hem PropertyManager, shown in Figure 29.18.
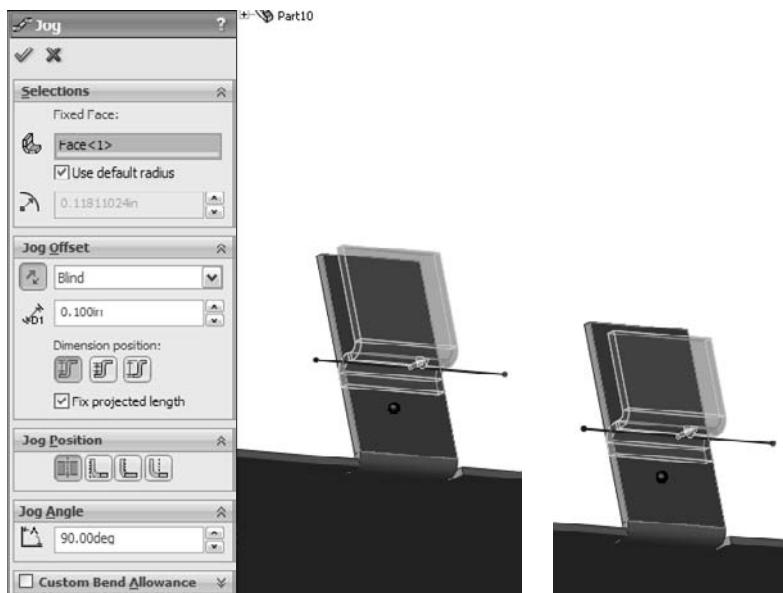
Jog feature

The Jog feature puts a pair of opposing bends on a flange so that the end of the flange is parallel to, but offset from, the face where the jog started. The Jog PropertyManager and a sample jog are shown in Figure 29.19.

The Jog PropertyManager and a sample jog

The Jog feature is created from a single sketch line on the face of a sheet metal part. The geometry to be jogged should not have any side bends; it should be a simple tab-like flange, as shown in Figure 29.19. The line to create the jog can be drawn at an angle, causing the jog to also be angled.

The three icons on the Jog Offset panel illustrate what dimension is being controlled by that setting.

Fixed Face

Like most sheet metal features, the Jog feature bends faces on the part, and when it does so, although it may be obvious to you as the user, it is not obvious to the software which face should remain stationary and which faces should be moved by the bend. The Fixed Face selection box enables you to select a face, or in this case, a part of a face, that you want to remain stationary as the rest of the faces move. The black dot on the face identifies it as stationary.

Tip

Problems can sometimes arise when you are using configurations that change sizes, because these markers for fixed faces can be pushed onto other faces. This can cause problems with assemblies and drawings, and in general makes visualization difficult. In cases like this, it may be advisable to select a larger face or one that has fewer changes, if possible, to be used as the fixed face. ■

Jog Offset

You can control the direction of the jog by using the arrow button to the left of the end condition selection box. You can control the jog distance by selecting the end conditions Up To Surface, Up To Vertex, or Offset From Surface. The default setting is Blind, in which you simply enter a distance for the offset, in exactly the same way that end conditions are controlled for features such as extrudes.

Fix projected length

One setting that may not be obvious is the Fix projected length. This refers to the length of the flange that the jog is altering. In Figure 29.19, you can see that the height of the jogged feature is the same as the height of the original feature. The jog obviously requires more material than the original, but the Fix projected length option is selected, and so the height is maintained. If you deselected this option, then the finished height of the flange after the jog is added would be shorter, because the material is used by the jog and additional material would not be added. For comparison, the image to the right in Figure 29.19 shows this situation.

Jog Position

The Jog Position selection establishes the relationship between the sketched line and the first bend tangent line. The Jog Position icons have tool tips with the following names, from left to right: Bend Centerline, Material Inside, Material Outside, and Bend Outside.

Jog Angle

The Jog Angle enables you to change the angle of the short perpendicular section of the jog. You can angle it to smooth out the jog (angles of less than 90 degrees) or to curl back on itself (angles of more than 90 degrees). Again, be careful to check with your manufacturer's capabilities.

Sketched Bend feature

Sketched Bend works in some respects like half of a jog. It requires the sketch line and the Fixed Face selection. You define a bend position with the same set of icons that you used in the jog, and you assign a bend angle in the same way.

Tip

You can use the Sketched Bend feature to *dog ear* **corners. You do this by drawing a line across the corner at an angle and setting the angle to 180 degrees and then overriding the default radius with a much smaller one, such as .001 inches.** ■
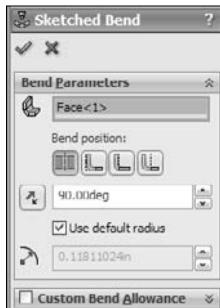
Unlike Jog, the Sketched Bend feature does not show you a preview. The Sketched Bend PropertyManager is shown in Figure 29.20.

The Sketched Bend PropertyManager
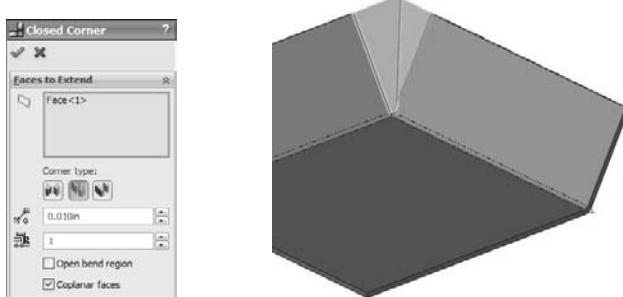
Closed Corner feature

The Closed Corner feature extends flanges on the sides to meet with other flanges. It is typically used when corners leave big open gaps in order to create a corner that is more easily welded shut (although welds cannot be created in sheet metal parts in SolidWorks). Figure 29.21 shows a part where angled flanges have been applied. This creates big gaps in the corners. Although a Miter Flange may have been better, these were created using regular Edge Flanges.

Applying the Closed Corner feature

Faces to Extend

You must select the thickness face of one of the flanges in order to extend it. Selecting one face automatically selects the matching face from the other flange that you also want to extend. The Corner Type selection icons depict the selected face as red, and the three icons display tooltips: Butt, Overlap, and Underlap.

Faces to Match

The faces selected in the Faces to Match selection box act as an "up to" end condition for the faces to extend. Prior to SolidWorks 2010, the Closed Corner feature always automatically selected a matching face to extend for each face selected, when appropriate. SolidWorks 2010 enables you to manually select matching faces in the Faces to Match selection box for those times when the automatic selection does not work.

Note

If you deselect faces in either the Faces to Extend or Faces to Match selection boxes, the Auto Propagation option toggles off to enable you to make selections manually. ■

Gap

The Gap setting enables you to specify how close you want the closed corner to be. Keep in mind that you cannot use the number zero in this field. If you do, then SolidWorks reminds you to "Please enter a number greater than or equal to 0.00003937 and less than or equal to 0.86388126." It is good to know your limits.

Overlap/Underlap ratio
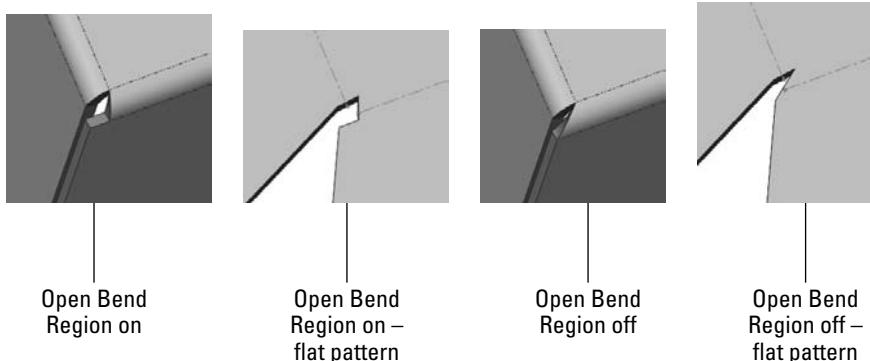
The Overlap/Underlap ratio setting controls how far across the overlapped face the overlapping flange reaches. Full overlap is a ratio of 1, and a Butt condition is (roughly) a ratio of zero. This ratio is only available when you have specified Overlap or Underlap for the corner type.

Open bend region

The Open bend region option affects how the finished corner looks in the bend area. If Open bend region is selected, then a small gap is created at the end of the bend. If the option is deselected, then SolidWorks fills this area with geometry. Figure 29.22 shows the finished model with this option selected and unselected, as well as the resulting Flat Patterns for each setting.
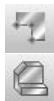
FIGURE 29.22

The Open bend region option, both selected and unselected, and the resulting Flat Patterns

Open Bend Region on

Open Bend Region on – flat pattern

Open Bend Region off

Open Bend Region off – flat pattern
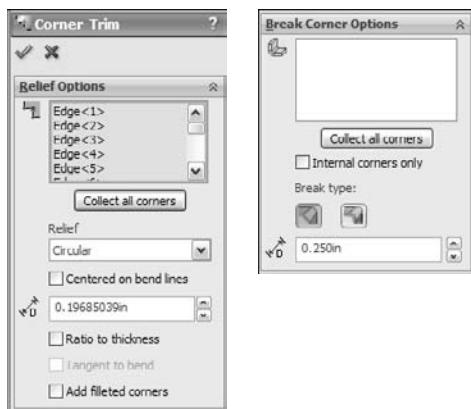
Coplanar faces

When the Coplanar option is selected, any faces that are coplanar with any selected faces are also selected Corner Trim and Break Corner features. The Corner Trim feature is available only when the sheet metal part is in its flattened state. The Corner Trim PropertyManager also has the Break Corner Options interface built right into it. However, the Break Corner feature is only available when the sheet metal part is in its folded state. Figure 29.23 shows the combined interface. Both functions are included here, and SolidWorks treats them as if they are part of a single function.

The Corner Trim PropertyManager, including the Break Corner Options panel
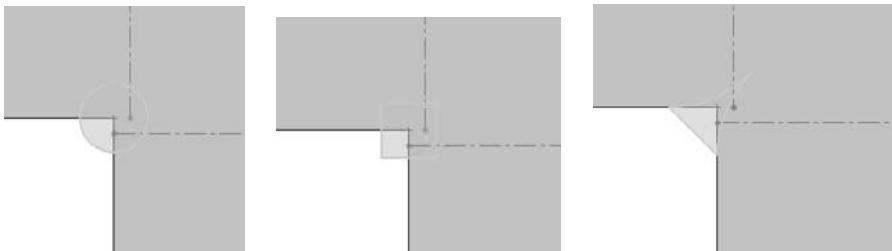
When finished, the Corner Trim feature places itself after the Flat Pattern feature in the FeatureManager. It similarly follows the suppress/unsuppress state of the Flat Pattern feature. When the Break Corner feature is used on its own, it is placed before the Flat Pattern feature. With this in mind, it seems best to use Break Corner as a separate feature unless it is being used specifically to alter the Flat Pattern in a way that cannot be done from the folded state.

Break Corner on its own is primarily used to remove sharp corners using either a chamfer or a rounded corner. This tool is set up to filter edges on the thickness of sheet metal parts, which is useful, because these edges are otherwise difficult to select without a lot of zooming. Break Corner can also break interior corners.

One of the main functions of the Corner Trim feature is to apply bend relief geometry to the Flat Pattern. The three available options are Circular, Square, and Bend Waist. These options are shown in Figure 29.24.

FIGURE 29.24

Applying the Corner Trim Relief options

Forming Tool feature

Forming tools in SolidWorks enable you to place features that are not formed on a brake press. These are features that are not straight-line bends, but rather punched, drawn, formed, lanced, sheared, or otherwise deformed material.

One of the important things to understand about forming tools is that they do not stretch the material in the SolidWorks part in the same way that happens in a real-life forming operation. In real life, material is thinned when it is punched, stamped, or drawn. In SolidWorks, the thickness of a sheet metal part remains the same, regardless of what happens to it. For this reason, you need to be careful when using mass properties of sheet metal parts or doing stress analysis of parts that have formed features. You might consider taking your part weight from the Flat Pattern rather than from the formed sheet metal.

SolidWorks installs with a library of fairly simple forming tools that you can use as a starting point for your own personal customized library. You can also examine some of these tools to see how they create particular effects. You find this library in your Design Library in the Task pane. Some of the more interesting forming tools are the lances and louvers.

Creating forming tools

Forming tools are essentially a part that is used as a tool to form another part. One flat face of the forming tool part is designated as a Stopping Face, which is placed flush with the top face of the sheet metal part. You can move and rotate the tool with the Modify Sketch tool, and you can use dimensions or sketch relations to locate it.

Creating forming tools is far easier than it used to be. This section of the chapter gives you the information that you need to effectively create useful forming tools, addresses the limitations and unintended uses of forming tools, and provides a couple of hints for more complex forming tool creation.

To create a forming tool, click the Forming Tool button on the Sheet Metal toolbar. Figure 29.25 shows the PropertyManager interface for this tool.

FIGURE 29.25

The Form Tool PropertyManager and a sample tool with orientation sketch

The Stopping Face turns a special color, and so do any faces that are selected in the Faces to Remove selection box. Faces to Remove means that those faces will be cutouts in the sheet metal part.

Another aspect of the forming tool is the orientation sketch. Create the orientation sketch by using Convert Entities on the Stopping Face. If you have used this function in any of its previous versions, then you know that this latest iteration is far easier to create than before. However, to me, it looks like the orientation sketch has taken a step backward. The orientation sketch cannot be manually edited, and so for forming tools where *footprints* are symmetrical, but other features in the tool are not, you cannot tell from the sketch which direction the forming tool should face. Orientation could be managed more easily in earlier versions of forming tools because the placement sketch was just a manually created sketch.

When creating a forming tool, you must remember to build in generous draft and fillets, and not to build undercuts into the tool. Also keep in mind that when you have a concave fillet face on the tool, the radius becomes smaller by the thickness of the sheet metal; as a result, you must be careful about minimum radius values on forming tools. If there is a concave face on the tool that has a .060-inch radius and the tool is applied to a part with a .060-inch thickness, then the tool will cause an error because it forms a zero radius fillet, which is not allowed. Errors in applied forming tool features cannot be edited or repaired, except by changing forming tool dimensions.
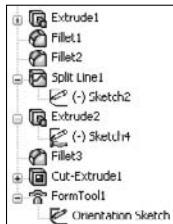
Once the forming tool is created, special colors are used for every face on the part. For example, the Stopping Face is a light blue color, Faces to Remove are red, and all the other faces are yellow. Figure 29.26 shows the small addition that is made to the FeatureManager when you make a part into a forming tool. This feature did not exist in older versions of the tool.

Forming Tool Library

The folder that the forming tools are placed into in the Design Library must be designated as a Forming Tool folder. To do this, right-click the folder that contains the forming tools and select Forming Tool Folder (a check mark appears next to this option).

FIGURE 29.26

The FeatureManager of a forming tool part

Placing a forming tool

To place a forming tool on a sheet metal part (forming tools are only allowed to be used on parts with sheet metal features), you can drag the tool from the library and drop it on the face of the sheet metal part. Forming tools are limited to being used on flat faces.

From there, you can use the Modify Sketch tool or horizontal and vertical sketch relations to move and rotate the forming tool. It may be difficult to orient it properly without first placing it, seeing what orientation it ends up in, and then reorienting it if necessary because of the limitation mentioned earlier with not being able to edit the orientation sketch to give it some sort of direction identifier.

Configurations cannot be used with forming tools like they can with library features, although you can change dimensions by double-clicking the Forming Tool icon in the sheet metal part FeatureManager. Forming tools are suppressed when the part is flattened.

Special techniques with forming tools

One application of forming tools that is asked for frequently is the cross break to stiffen a large, flat sheet metal face. SolidWorks has a cosmetic cross break which I discuss next. Cross breaks are clearly not something that SolidWorks can do using straight bends, but a forming tool can do it.

You can create the forming tool by lofting a rectangle to a sketch point on a plane slightly offset from the plane of the rectangle. This creates a shallow pyramid shape. Open the part from the material on the CD-ROM for Chapter 29 called `Chapter 29 – Cross Break Sheet Metal. sldprt` to examine how this part was made. Figure 29.27 shows the Cross Break forming tool applied to a sheet metal part.

Cross Breaks
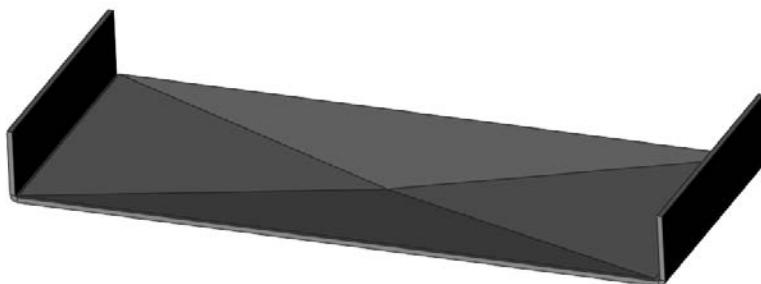
Using a forming tool to create a cross break is overkill for most situations. You may need to do it if you need to actually show the indented geometry. The Cross Break feature is essentially a *cosmetic* cross break, and it enables you to specify the radius, angle, and direction used to create the cross break. It does not actually change the part geometry at all, but it does add two curve-like display entities.

The Cross Break forming tool applied to a part
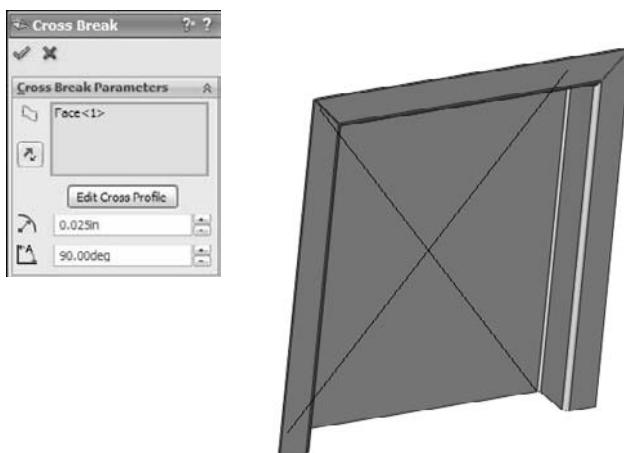
When you place a Cross Break feature, you have the option to edit the sketch profile that creates the cross. This sketch has two intersecting lines. You cannot add more lines; the feature will fail if you have more than two lines in the sketch. (For example, if you wanted to put three breaks across a hexagonal face, the software will not allow this.) The lines do not have to end at a corner, but they do have to end at an edge. If the lines extend past or fall short of an edge, the feature will display a red X error icon, but it still creates the break lines where the sketch lines are.

Figure 29.28 shows the Cross Break PropertyManager and a part to which a Cross Break was applied. Notice that you can see the break lines through the solid, much like curves or cosmetic threads.

The Cross Break feature shows up in the FeatureManager just like any other feature, not like a cosmetic thread, which is the only other entity in the software that the Cross Break much resembles.

Creating a Cross Break

Form across bends
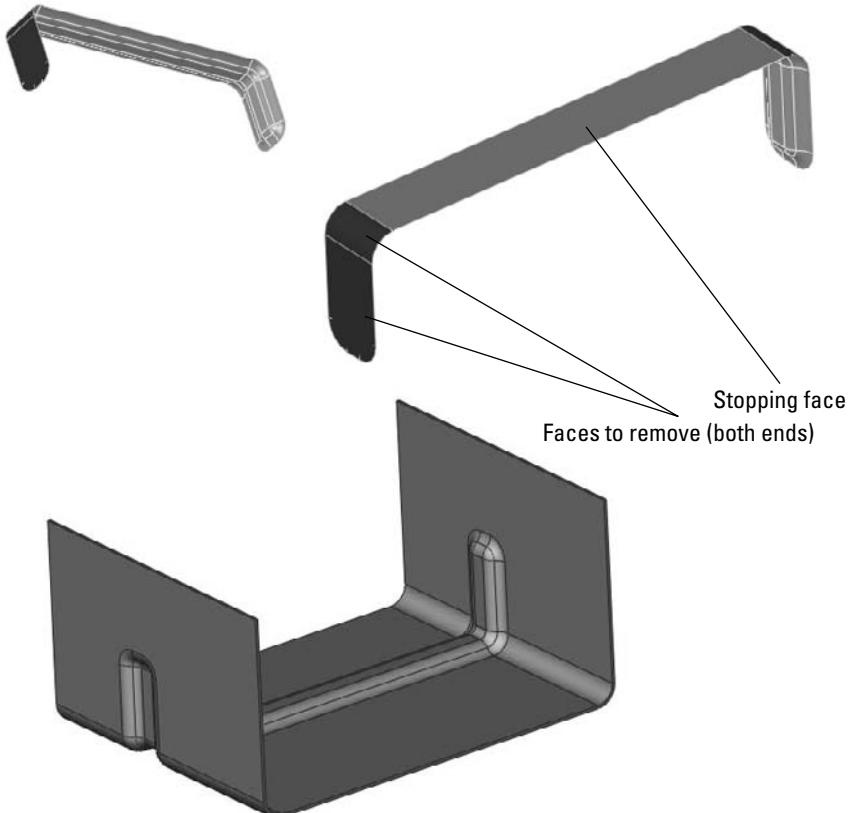
A second special technique is a gusset or a form that goes across bends. This can be adapted in many ways, but it is shown here going across two bends. I cannot confirm the practicality of actually manufacturing something like this, but I have seen it done.

The technique used here is to call the single long flat face of the forming tool the Stopping Face. The vertical faces on the ends and the fillet faces must be selected in the Faces to Remove selection box. The fillets of the outside of the forming tool also have to match the bends of the sheet metal part exactly. You may need to edit this part each time you use it, unless you apply it to parts with bends of the same size and separated by the same distance.

When you place the tool on the sheet metal part, you must place it accurately from side to side to get everything to work out properly. This part is in the same location as the Cross Break file, and is called `Chapter 29 – Form Across Bends Sheet Metal.sldprt`. Figure 29.29 shows the tool and a part to which it has been applied.

FIGURE 29.29

Forming across bends

Stopping face

Faces to remove (both ends)

Lofted Bends feature
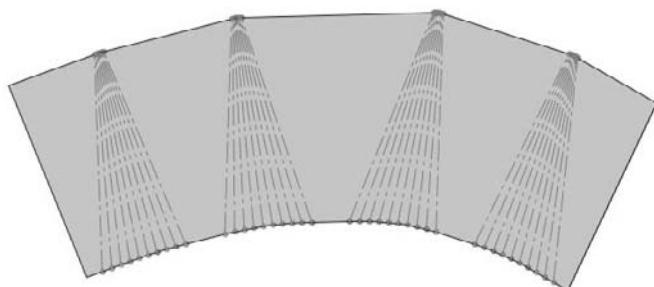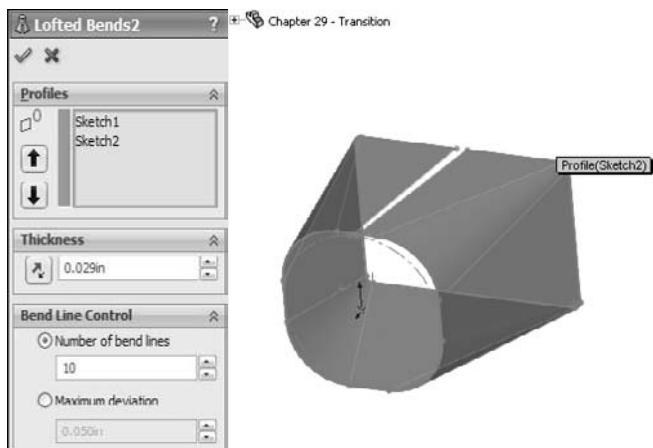
The Lofted Bends feature enables you to create transitions between two profiles. The range of functionality available through the Loft feature is not available with Lofted Bends; it is limited to two profiles with no end conditions or guide curves. Both profiles also need to be open contours in order to allow the sheet metal to unfold.

Lofted Bends is not part of the Base Flange method, but it is part of the newer set of sheet metal tools available in SolidWorks. Figure 29.30 shows what is probably the most common application of this feature. The bend lines shown must be established in the PropertyManager when you create or edit the feature. Bend Lines are only an option if both profiles have the same number of straight lines. For example, if one of the profiles is a circle instead of a rectangle with very large fillets, then the Bend Lines options are not available in the PropertyManager.

FIGURE 29.30

The Lofted Bends PropertyManager, a sample, and a Flat Pattern with bend lines
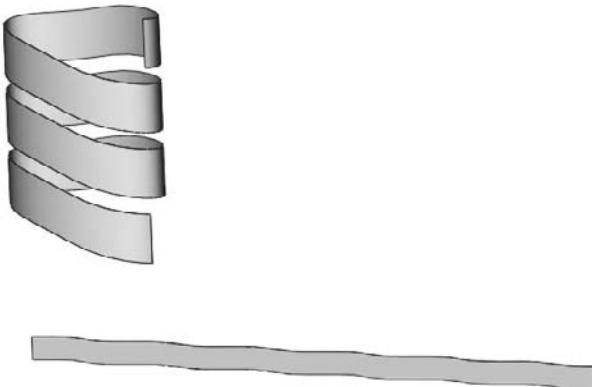
Like the forming tools, you can also use Lofted Bends in situations for which they were probably not intended. Figure 29.31 shows how lofting between 3D curves can also create shapes that can be flattened in SolidWorks. In this case, a couple of intermediate steps were required to get to the 3D curves, which involve surface features.

Note
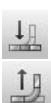
This part is included on the CD-ROM with the name `Chapter 29 – wrap.sldprt.`

FIGURE 29.31

Using 3D curves with Lofted Bends to create flatten complex shapes

Unfold and Fold features

Unfold is a feature that unfolds selected bends temporarily. It is typically used in conjunction with a Fold feature to re-fold the bends. This combination is used to apply a feature that must be applied to the Flat Pattern; for example, a hole that spans across a bend.
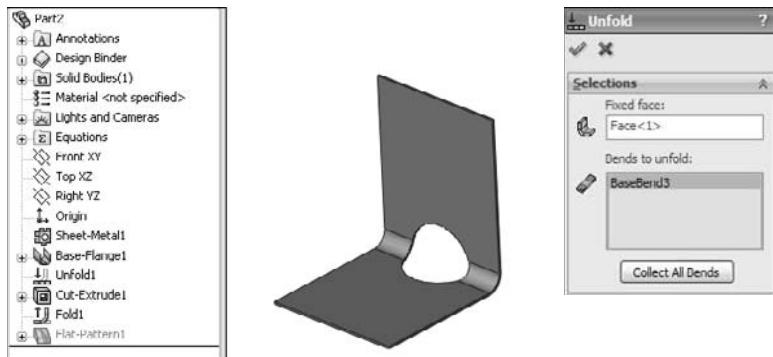
Figure 29.32 shows the FeatureManager of a part where this combination has been applied, as well as the part itself, showing the bend across a hole, and the PropertyManager, which is the same for both features.

Both the Unfold and Fold features make it easy to select the bends without zooming in, even for small bends. A filter is placed on the cursor when the command is active, which allows only bends to be selected. The Collect All Bends option also becomes available. This feature also requires that you select a stationary face to hold still while the rest of the model moves during the unfolding and folding process.

Applying the Unfold and Fold features

Making Sheet Metal Parts from Generic Models

SolidWorks can also convert generic constant thickness models into sheet metal parts that flatten, and on which any of the dedicated sheet metal features can be used. You can make models from thin feature extrudes or regular extrudes with Shell features, and then use the Insert Bends feature to make them sheet metal parts. The structure of parts created with the Insert Bends feature is somewhat different. Figure 29.33 shows a comparison of the two methods' FeatureManagers for simple parts.

The most notable difference is that the Insert Bends part starts off with non-sheet metal features. The Rip feature also stands out, but the Rip feature is not exclusive to sheet metal. Although you can use Rip on any model, it is found only on the Sheet Metal toolbar.
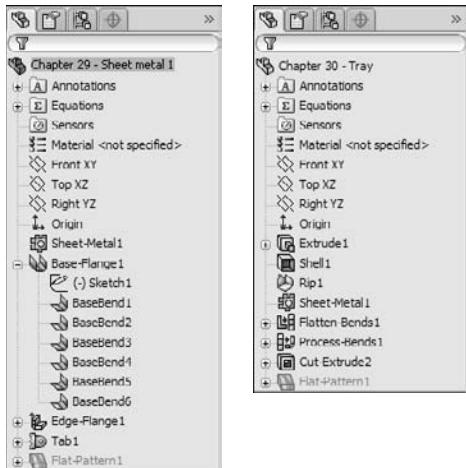
The Sheet Metal feature is found in both the Base Flange and Insert Bends methods, and has the same PropertyManager function in both methods.

The new features in the Insert Bends method are the Flatten Bends and Process Bends features. The way the Insert Bends method works is that the model that is built with the sharp-cornered non-sheet metal feature is flattened by the Flatten Bends feature. The model is then reconstructed with bends by the Process Bends feature.

The main rule that SolidWorks enforces on sheet metal models regardless of how they came to be sheet metal is that the parts should have a consistent wall thickness. When all the geometry is made from the beginning as a sheet metal part (using the Base Flange method), there is never a problem with this. However, when the part is modeled from thin features, cuts, shells, and so on, there is no telling what may happen to the model.

FIGURE 29.33

A comparison between default features for Base Flange and Insert Bends

If you perform an Insert Bends operation on a model that does not have a consistent wall thickness, then the Flatten Bends and Process Bends features fail. If a thickness face is not perpendicular to the main face of the part, then the software simply forces the situation, making the face perpendicular to the main face.

Normal cut feature

If a Cut feature is placed before the Sheet Metal feature, then as far as SolidWorks is concerned, the part is not a sheet metal part. However, if the cut feature is created after the Sheet Metal feature, then the model has to follow a different set of rules. The "normal shear" mentioned previously is one of those rules. In Figure 29.2, the sketch for a cut is on a plane that is not perpendicular to the face that the cut is going into. Under a normal modeling situation, the cut just goes through the part at an angle. However, in SolidWorks sheet metal, a new option is added to the PropertyManager for the cut. This is the Normal cut option, and it is selected by default. You could be modeling and never even notice this option, but it is important because it affects the geometrical results of the feature.

As shown in Figure 29.34, when the Normal cut option is selected, the thickness faces of the cut are turned perpendicular (or *normal*) to the face of the sheet metal. This is also important because if the angle between the angled face and the sketch changes, the geometry of the cutout can also change. This setting becomes more important as the material becomes thicker and as the angle between the sketch and the sheet metal face becomes shallower.
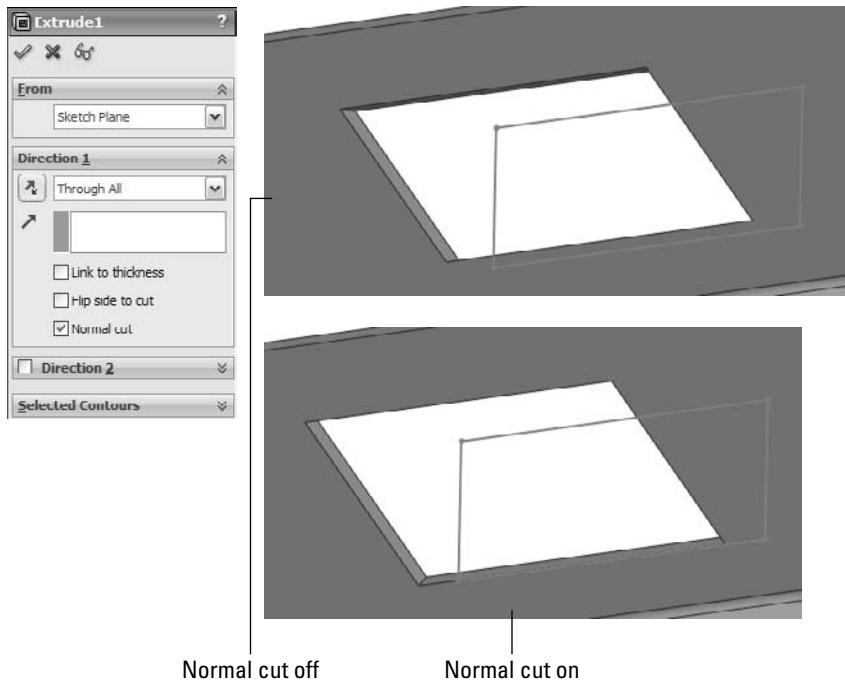
SolidWorks allows you to have angled faces on side edges, and will maintain the angle when it flattens the part. In previous versions, angles on side faces cause the Flat Pattern feature to fail. Even a

cut that does not use the Normal cut option and creates faces that are not perpendicular to the main face of the part will not cause the Flat Pattern to fail.

Using the Normal cut option

Normal cut off   Normal cut on

Rip feature
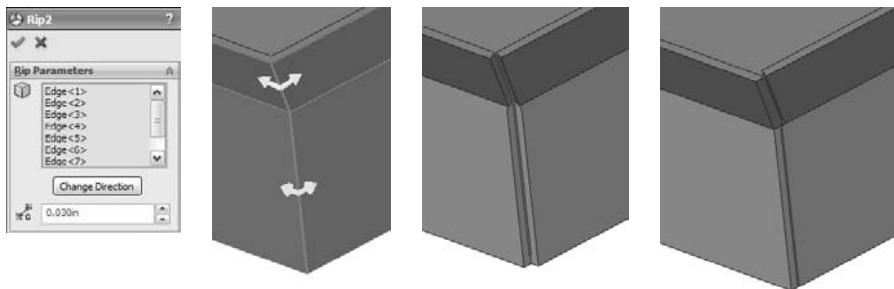
When building a sheet metal part from a generic model, a common technique used to achieve consistent wall thicknesses is to build the outer shape as a solid and then shell the part. The only problem with this method is that it leaves corners joined in a way that cannot be flattened. You can solve this problem by using the Rip feature. Rip breaks out the corner in one or both directions in such a way that it can be unfolded. Bend reliefs are later added automatically by the Process Bends feature.

Figure 29.35 shows the Rip PropertyManager and the results of using this feature. The model was created to look like a Miter Flange part.

Notice also in Figure 29.3 that after the Rip, the edges of the material are still sheared at an angle. Because the top of the part was shelled, the thickness of the part is not normal to the main face of the sheet metal. You can fix this by using the Flatten Bends feature, which lays the entire part out flat, calculates the bend areas, and corrects any discrepancies at the edges of the part.

Using the Rip feature

Note

Rip functionality is included in the Insert Bends Sheet Metal PropertyManager when it is first initiated, although it is no longer there when you edit the part later. If you use it, the Rip data becomes a feature of its own and is placed before the Sheet Metal feature in the FeatureManager. Be aware that there are slight differences between using the Rip function as an independent feature and using it as a part of the Insert Bends feature. You may want to check this on a part you are working with to verify which method best suits your needs. ■

Sheet Metal feature

The Sheet Metal feature used in the Insert Bends method is very similar to the one used in the Base Flange method. However, two main differences exist: Insert Bends Sheet Metal requires the user to select a Fixed Face, and Base Flange Sheet Metal allows the use of Gauge Tables. Both features function as placeholders and otherwise contain the same information, use the same name and icon, and are inserted automatically when a different feature is created.

Flatten Bends feature
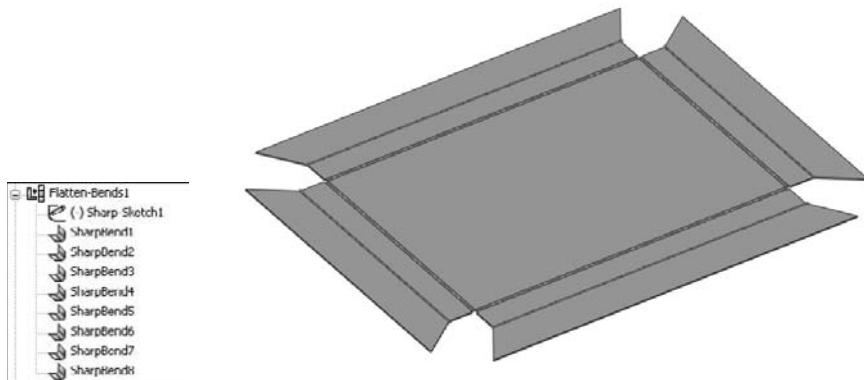
The Flatten Bends feature is added automatically by the Insert Bends tool. As mentioned earlier, it takes the model with sharp corners and lays it flat, adjusting the material in the bend area and normalizing the thickness faces around the Flat Pattern. The Merge Faces option is available in the Flat Pattern feature in the Insert Bends method ; therefore, the Flat Pattern created by the Flatten Bends feature always has edges created by the tangent lines of the bends.

Notice in Figure 29.36 that the Flatten Bends feature has a sketch and several Sharp Bend features under it. The Sharp Sketch is simply an account of the bend lines, and you cannot edit it manually. The Sharp Bend features can be suppressed, in which case they are not re-formed in the Process Bends feature. You can also edit Sharp Bend features to change the default radius, bend allowance, and relief type.

FIGURE 29.36

Using the Flatten Bends feature

Process Bends feature

The Process Bends feature takes all the Flat Pattern information, the bend information, and entities in the Flat Sketch, and rebuilds the model with the formed bends. The Flat Sketch under the Process Bends feature is the Insert Bends method version of a sketched bend. You can add sketch lines here to bend panels of the part. After you add lines to this sketch, exiting the sketch causes the part to be created with a default 90-degree bend corresponding to the line. Of course, all the Sketched Bend rules exist, such as that the line has to extend at least up to the edges of the part, the lines cannot extend across multiple faces, and construction lines are ignored.

For every bend created by a sketch line in the Process Bends Flat Sketch, a Flat Bend feature is added to the list under Process Bends. You can control the angle and radius of each of these Flat Bends by editing the Flat Bend feature. This is all illustrated in Figure 29.37.

No Bends feature
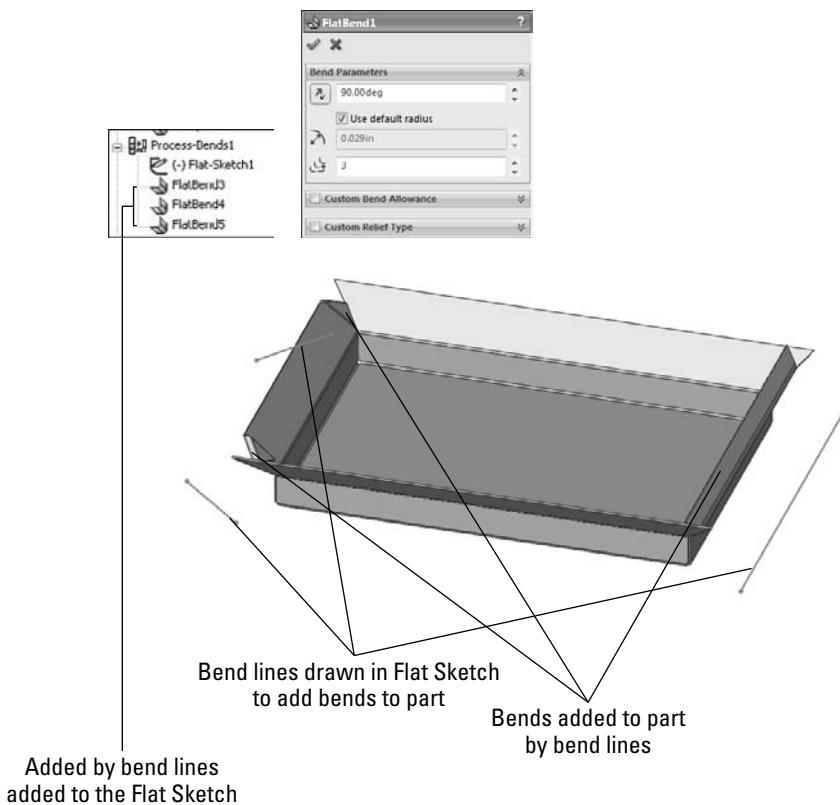
You use the No Bends tool on the Sheet Metal toolbar to roll back the model before the Flatten Bends feature in the tree with a single button click. This is primarily to add new geometry that is turned into bends through the Flatten and Process Bends features.

Flat Pattern feature

The Insert Bends method uses the Flat Pattern feature as well as the Base Flange method. However, it was not part of the original scheme, and was added at some point after the new tools had proved their value. This enables you to make use of the new features as well, as discussed later in this chapter in the section on mixing methods.

FIGURE 29.37

Using the Process Bends feature

Bend lines drawn in Flat Sketch
to add bends to part

Bends added to part
by bend lines

Added by bend lines
added to the Flat Sketch

Convert to Sheet Metal feature

The Convert to Sheet Metal feature can use either SolidWorks native data or imported data. It can also use solids as well as surfaces. The model can be shelled or not shelled, and have filleted edges or not. This feature enables you to identify which edges will become bends and automatically identifies the edges to rip. See Figure 29.38.
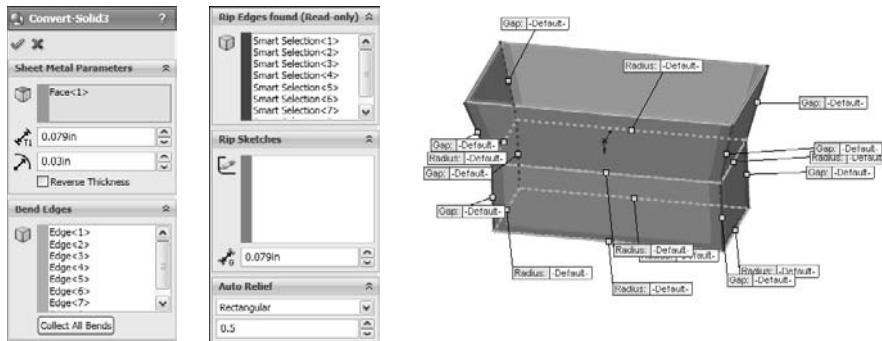
This tool is very useful for imported geometry and for parts with tricky shapes. Although the PropertyManager interface looks busy, it is fairly straightforward to use. Your first selection in the top Fixed Entity box should be a stable face, preferably an outer face on the bottom or the top. Inner faces generally do not work.

Note that you can reverse the thickness of the sheet metal, so that the solid that you start with can be treated as the volume inside the sheet metal enclosure, or the outer faces of the initial solid turn out to be the inner faces of the sheet metal part. Use the Reverse Thickness option to accomplish this.

FIGURE 29.38

Using Convert to Sheet Metal

Selecting Bend Edges is the next step, with the implication that any edge that is not a bend will be ripped. Also note that three bend edges cannot intersect at a point or one bend edge cannot intersect at the middle of another edge.

Setting default bend radius, thickness, and Auto Relief options are the same as in other sheet metal functions.

Using Other Methods

The sheet metal tools have been available in SolidWorks for quite some time, and have had some time to mature and for users to become well acquainted with them and develop effective techniques using them.

Working with imported geometry

Working with imported geometry starts at the point where you use the Rip feature. While imported geometry can be geometrically manipulated to some extent in SolidWorks, this is beyond the scope of this chapter. The need for a model with walls of constant thickness still exists, even if the imported model has filleted edges showing bend geometry already in the model.

FeatureWorks may be used to recognize sheet metal features or to fully or partially deconstruct the model by removing bend faces as fillets. While FeatureWorks is not covered in this book, the technique may be useful when editing imported parts with overall prismatic geometry that is common to sheet metal parts.

When a sheet metal part is imported, whether it meets the requirements immediately or must be edited in one way or another to make a sheet metal part of it, you can simply use the Insert Bends feature or even the Convert to Sheet Metal feature.

Making rolled conical parts
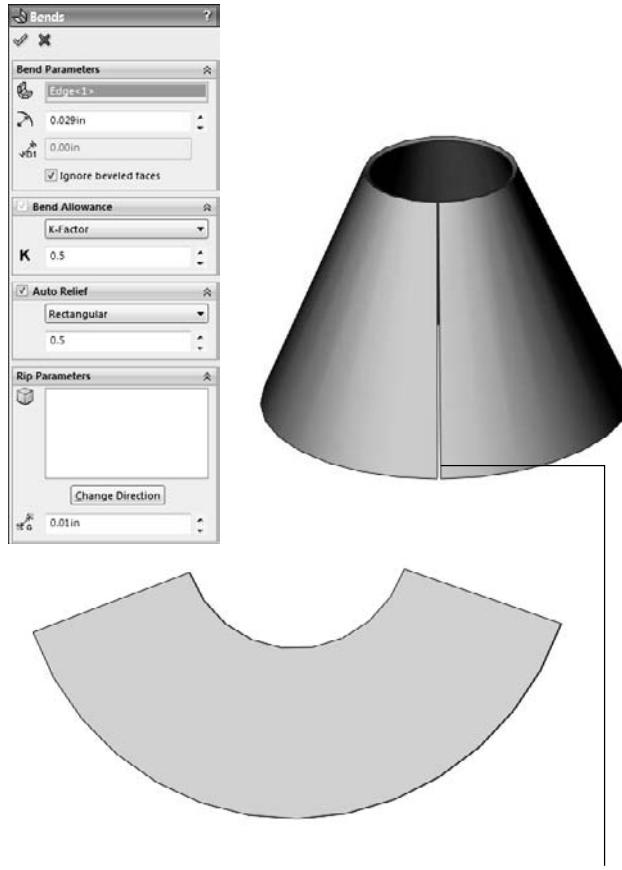
One of the reasons for maintaining the legacy Insert Bends method is to have a way of creating rolled conical parts. You can create cylindrical sheet metal parts by drawing an arc that almost closes to an entire circle, and creating a Base Flange from it. However, no equivalent technique for creating tapered cones exists with the Base Flange method.

With the Insert Bends method, a revolved thin feature does the job nicely. You simply revolve a straight line at an angle to the centerline, so that the straight line does not touch or cross the centerline; the revolve cannot go around the full 360 degrees because there must be a gap. Sheet metal parts are not created by stretching the material (except for Forming Tools).

When creating a rolled sheet metal part, you cannot select a flat face to remain fixed when the part is flattened. Instead, you can use a straight edge along the revolve gap, as shown in Figure 29.39.

FIGURE 29.39

Selecting a straight edge for a conical part

Select one of these edges in the
Fixed Face/edge selection box

Note

When a conical sheet metal part is created, it does not receive the Flat Pattern feature at the end of the FeatureManager. This is because none of the new Base Flange method features are allowed on this type of part. ∎

Mixing methods

If you use the Insert Bend tool on a part, you can still use the more advanced tools available through the Base Flange method, unless it is a cylindrical or conical part. A Flat Pattern feature is added to the bottom of most feature trees, and the presence of this feature is what signifies that the current part has now become a sheet metal part to the Base Flange features.

However, it is recommended that you avoid mixing the different techniques to flatten parts; for example, suppressing bends under Flatten and Process Bends, as well as using the Flat Pattern.

Using Multi-body Techniques with Sheet Metal

As of SolidWorks 2010, you can now use multi-body techniques with sheet metal models. For many of the same reasons you might want to make any other kind of model using multi-body techniques, you may also want to make sheet metal parts using similar techniques. The new rules have several implications for old limitations of sheet metal parts such as:

- You can now have multiple Base Flange features.
- With multiple Base Flange features you also get multiple Flat Pattern features.
- If you have multiple bent bodies, you can only show one body flattened at a time.
- Merging sheet metal bodies eliminates one Flat Pattern feature.
- You can use the Split feature to create multiple sheet metal bodies.
- The commands that can create new bodies in sheet metal parts are as follows:
  - Convert to Sheet Metal
  - Lofted Bend
  - Insert Bends
  - Base Flange
  - Insert Part
  - Split
- The commands that can merge bodies in sheet metal parts are as follows:
  - Edge Flange
  - Combine

The Mirror function enables you to mirror bodies, but the new bodies have to be merged manually with the existing body.

Using Insert Part

Using the Insert Part feature inserts an existing part as a new body inside a sheet metal part, but even if the inserted part was a sheet metal part initially, it does not show up as sheet metal after being inserted in the other part.
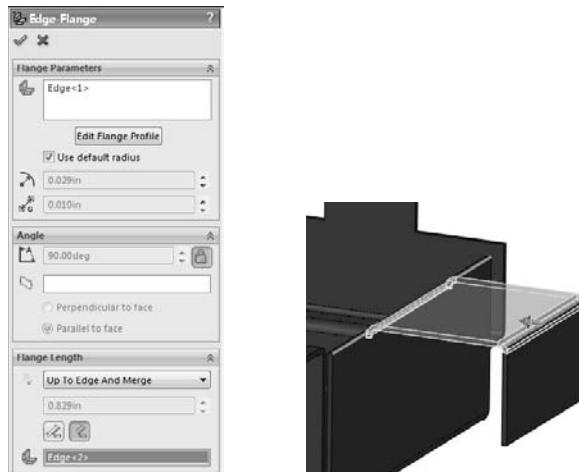
You can join the inserted part to the local sheet metal body by using the Combine feature, but not by using the merge option in an Edge Flange as you can to merge two bodies modeled within a single part. When the Combine feature is used, any sharp intersection between the parts is left sharp, and will not flatten unless you use the Insert Bends feature to convert the sharp into a bend. This is an odd twist on combining the old (Insert Bends) method with the new (Base Flange) method.

Using multiple Base Flanges

Another method to get multiple bodies inside a sheet metal part is to start from disjoint Base Flange features. You can build flanges toward one another until flanges touch. Figure 29.40 illustrates a situation where a disjoint flange created by a base flange feature is connected to the main part using an Edge Flange feature with the Up To Edge And Merge flange length setting.

FIGURE 29.40

Using an Edge Flange to connect disjoint bodies in a sheet metal part

Tutorial: Working with the Insert Bends Method for Sheet Metal Parts

The Insert Bends method has been relegated to duty mainly for specialty functions. To gain an understanding of how this method works, follows these steps:
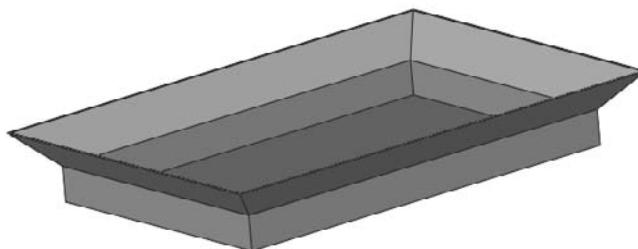
1. **Create a new blank part.**

2. **On the Top plane, open a sketch and sketch a rectangle centered on the Origin 12 inches in the Horizontal direction and 8 inches in the Vertical direction.**

3. **Extrude the rectangle 1 inch with 45 degrees of draft, Draft Outward, in Direction 1, and extrude 1 inch with no draft in Direction 2.** The two directions should be opposite from one another.

4. **Shell out the part to .050 inches, selecting the large face on the side where the draft has been applied.** The part should now look like Figure 29.41.
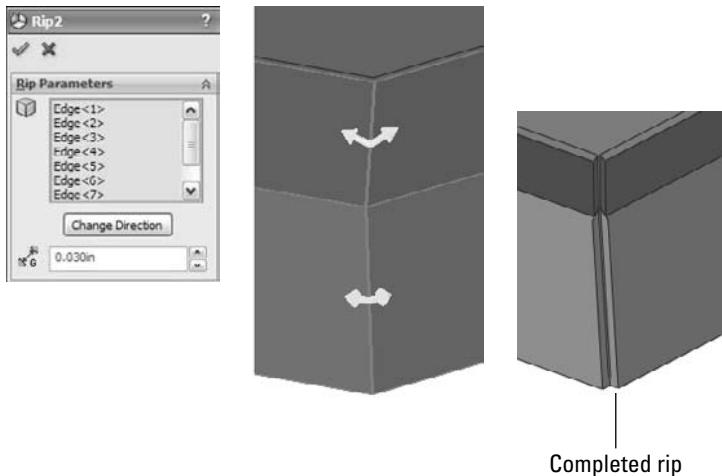
The part as of Step 4

5. **Use the Rip feature to rip out the four corners.** Allow the Rip to rip all corners in both directions. The part should now look like Figure 29.42.

6. **Create an Insert Bends feature, accepting the default values, and picking the middle of the base of the part for the fixed face.**

7. **Draw a rectangle on one of the vertical faces of the part, as shown in Figure 29.43.**
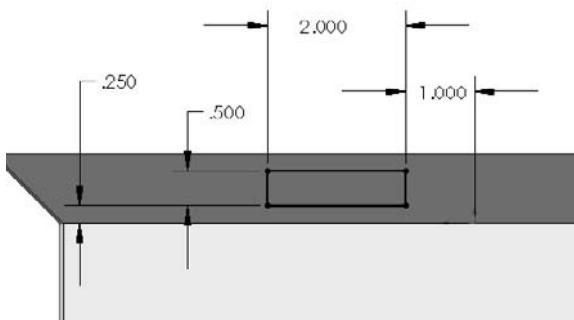
FIGURE 29.42

Ripping the corners

Completed rip

FIGURE 29.43

Adding a sketch for the cut
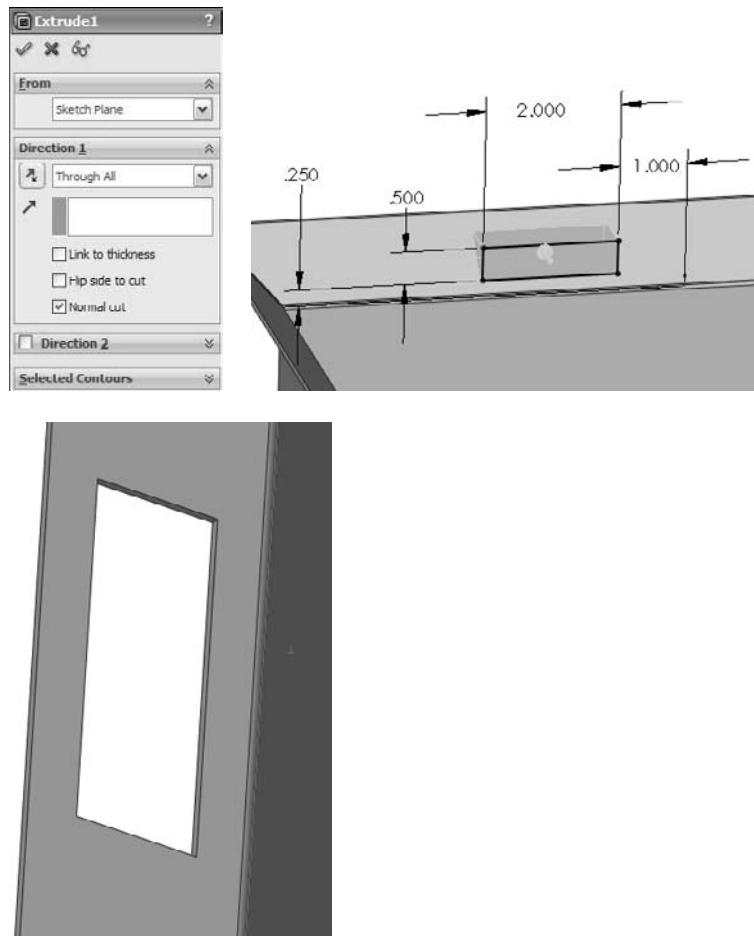
8. **Use the sketch to create a Through All cut in one direction.** Notice that the Normal cut option is on by default. Examine the finished cut closely; notice that it is different from the default type of cut because it is not made in a direction normal to the sketch, but rather in a direction normal to the face of the part. Details of this are shown in Figure 29.44.
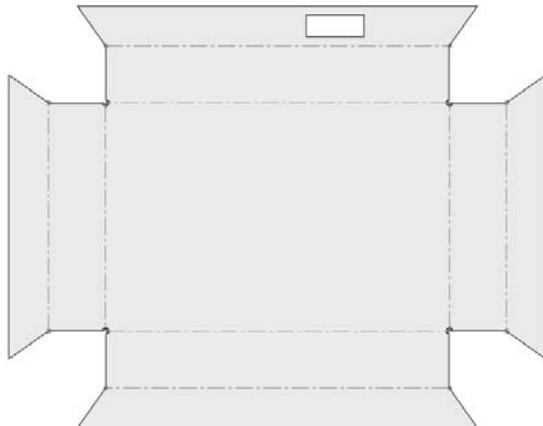
Using the Normal cut option

9. **Click the Flatten button on the Sheet Metal toolbar.** Notice that the Flat Pattern feature becomes unsuppressed and that the Bend Lines sketch under it is shown. This works just like it did in the Base Flange method. The finished part is shown in Figure 29.45.

FIGURE 29.45

The finished part with the Flat Pattern feature unsuppressed

Tutorial: Using the Base Flange Sheet Metal Method
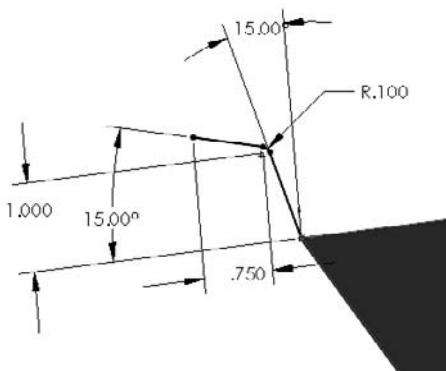
SolidWorks Base Flange method for sheet metal is fun and easy to use as you will see in this tutorial:

1. **Open a new part using a special sheet metal template if one is available.**

2. **On the Top plane, draw a rectangle centered on the Origin, 14 inches in X by 12 inches in Y (or Z).**

3. **Initiate the Base Flange tool, set the thickness to .029 inches, and change the K-Factor to .43.** Notice that the default inside bend radius is not shown. This setting is made in the Sheet Metal feature that is placed before the Base Flange feature in the FeatureManager.

4. **After the Base Flange has been created, edit the Sheet Metal feature, and change the default bend radius to .050 inches.**

5. **Click one of the 14-inch edges and then select the Line tool from the Sketch toolbar.** This is a shortcut to creating a plane perpendicular to the end of the edge and opening a new sketch on the plane. This is useful in other situations in addition to working with sheet metal. Draw a sketch similar to that shown in Figure 29.46. The arc overrides the default inside bend radius setting, and directly controls that particular bend.
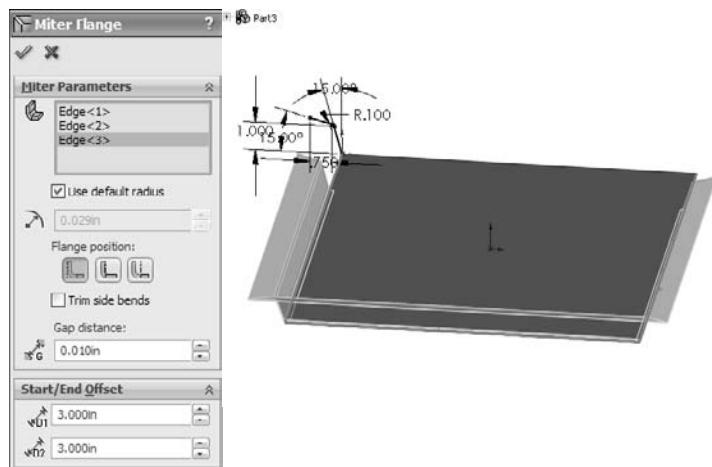
FIGURE 29.46

The sketch to start a Miter Flange

6. **With the sketch still active, click the Miter Flange button on the Sheet Metal tool-bar.** Use the settings shown in the image to the right in Figure 29.47. Select three edges as shown. Remember to select the edges on the same side of the Base Flange. In particular, notice the Start/End Offset settings. Click OK when you are satisfied with the settings.

FIGURE 29.47

Specifying the Miter Flange settings
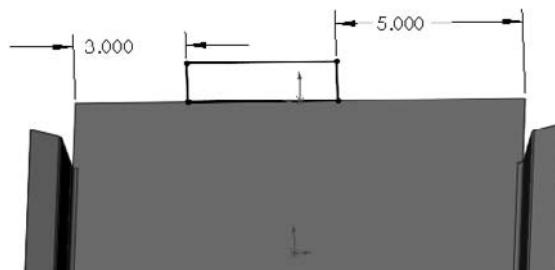
7. **Select the remaining edge that is not touched by the Miter Flange, and click the Edge Flange tool on the Sheet Metal toolbar.** Click the top point of one end of the Miter Flange to establish the flange length using the Up To Vertex end condition.

8. **Click the Edit Flange Profile button in the PropertyManager, and manually pull the sketch back from the ends of the flange.** Add dimensions to make the flange 3 inches from the corner on the left side, and 5 inches from the corner on the right side, as shown in Figure 29.48; otherwise, use the default settings for the flange. Click OK to accept the feature when you are satisfied with the settings.

FIGURE 29.48
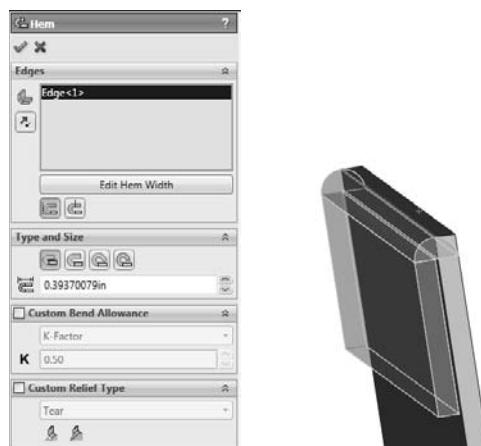
Creating an Edge Flange

9. **Select the inside edge of the top of the Edge Flange that you have just created, and initiate a Hem feature.** Use the settings Material Inside, Closed Hem, with a length of .25 inches, and make the material go toward the inside of the box. The settings and preview of the feature are shown in Figure 29.49.

FIGURE 29.49

Creating a hem

937
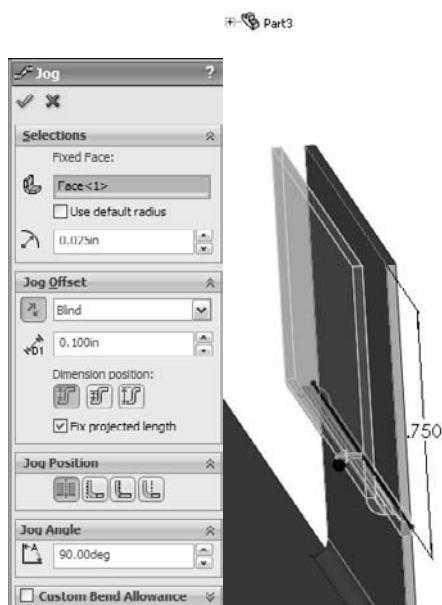
10. **Create a second Edge Flange the same height as the first, just to the right of the first flange, as seen from the point of view used in Figure 29.48.** Edit the flange profile and pull the new flange away from the existing flange. Add a dimension to make the new flange 2 inches wide. Click OK when you are satisfied with the settings.

11. **Open a sketch on the inside face of the new Edge Flange and draw a line across the flange .75 inches from the end.**

12. **Create a Jog feature with the settings shown in Figure 29.50.** Make sure to set a custom bend radius by deselecting the Use default radius option and entering **.025 inches**. If you do not set the custom radius, you may get a warning that the jog distance is less than a minimum jog value. Be careful when selecting the fixed face to select the side of the line with the largest area, or the face you want to remain where it is while the rest of the part bends and moves around it.

FIGURE 29.50

Creating a jog
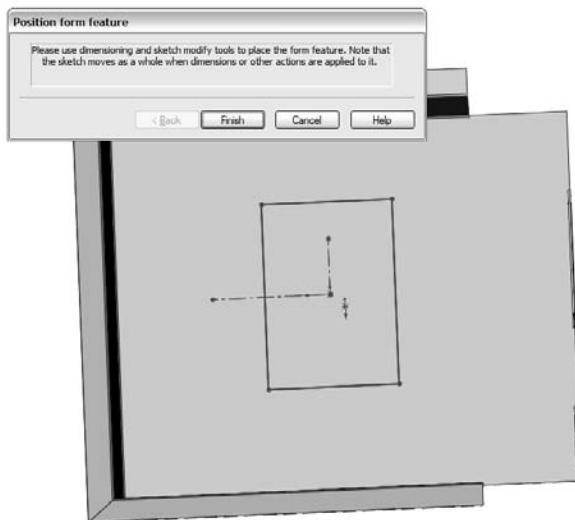
13. **From the CD-ROM, in the folder for Chapter 29, find the part named** `Chapter 29 – Cross Break.sldprt.` Copy this file to a folder in the library that you have established outside of your SolidWorks installation folder, called Forming Tools.

14. **Make sure that this folder appears in the Design Library.** You may have to press F5 or click the Refresh button at the top of the Task pane. When the folder appears, right-click the folder and select the check mark next to Forming Tools Folder.

15. **When the file has been copied and the folder has been assigned as a Forming Tool folder, drag the Chapter 29 – Cross Break part from the folder and onto the big flat face of the sheet metal part.** You will be put into a sketch that looks like Figure 29.51.

Placing a forming tool
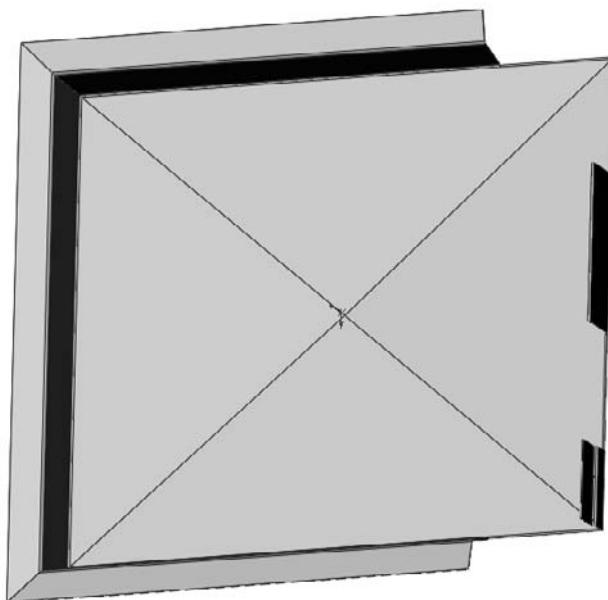
16. **Once you have dropped the feature into the sketch, drag the Origin of the sketch onto the Origin of the part, and then click Finish.** Notice that the cross break is in the middle of the part, but is too small.

17. **Double-click the new feature in the FeatureManager; a set of dimensions appear on the screen.** Change the 4-inch dimension to 13.9 inches, and the 6-inch dimension to 11.9 inches. The cross break should now look like Figure 29.52.

FIGURE 29.52

Resizing the cross break to 13.9

18. **Create a new configuration named Flat**. In this configuration, suppress the forming tool that you just placed, and unsuppress the Flat Pattern feature at the bottom of the tree.

Summary

SolidWorks offers a broad range of sheet metal tools to tackle most of your modeling situations. Some of the tools still require a little imagination because the complex shapes created in the real world where bends intersect are problems for such highly automated software. The tools are able to deal with imported or generically modeled geometry as well as parts created using the dedicated sheet metal tools.

Using Imported Geometry and Direct Editing Techniques

The direct editing set of tools in SolidWorks enables you to change a part without the knowledge of how it was originally created. Direct editing works on both native and imported geometry by simply moving faces. There are two things to consider with direct editing: First is how direct edit tools work in SolidWorks, which is the main focus of this chapter; and second is how direct edit tools compare to the tools in history-free CAD software, which is a side issue but addressed as well.

This chapter exists partially because of recent direct editing developments in the CAD world and partially in honor of the new Direct Editing tab SolidWorks added to the CommandManager in the 2010 release, which is shown in Figure 30.1.

The new tab has the following tools on it, listed from left to right from Figure 30.1:

- Move Face
- Move/Copy Bodies
- Fillet
- Chamfer
- Pattern
- Delete Solid/Surface
- Delete Face
- Split
- Combine

SolidWorks 2010 has a new Direct Editing tab on the CommandManager.

Additional direct edit tools exist in SolidWorks that should possibly be added to the Direct Edit tab:

- Flex
- Deform
- Freeform

Combining these tools with some of the new Instant 3D functionality gives direct editing tools in SolidWorks many of the advantages of the direct edit–only CAD software. Before a discussion of direct editing will make sense, you need to know a little bit about imported geometry.

Understanding the Basics of Imported Geometry

Geometry that is transferred between CAD packages is called imported geometry. The transfer usually happens through IGES (Integrated Geometry Exchange Standard; pronounced *eye-jess*), STEP (Standard for the Exchange of Product), Parasolid, or ACIS (named for the initials of three people from one company who created the standard: Alan, Charles, Ian, and Spatial) formats. SolidWorks also reads some native CAD data directly. For example, SolidWorks can read data directly from versions of Pro/ENGINEER, Unigraphics, Inventor, Solid Edge, CADKEY, and Rhino, as Figure 30.2 shows. In almost all cases, features are not transferred between CAD packages. The geometry that you wind up with is called "dumb" geometry because the smart parametrics and design intent that the model had in its parent software is no longer there.
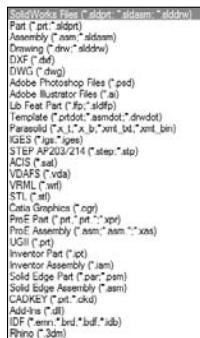
You can bring in imported data in one of two ways. The most common way is to use the Open dialog box and switch the Files Of Type to an imported file format. The other way is to use the Imported Geometry feature by choosing Insert ➪ Features ➪ Imported Geometry from the menus. The Open dialog box creates a new part with the imported feature at the top of the FeatureManager, as shown in Figure 30.3. Using the Imported Geometry feature enables you to insert imported geometry anywhere you like within the FeatureManager, even after other features have been added.
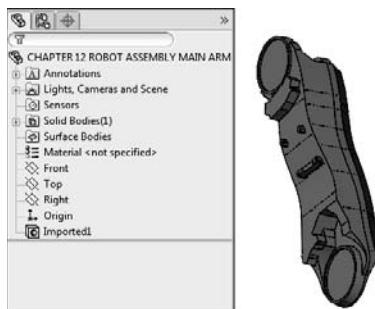
FIGURE 30.2

SolidWorks opens neutral format files as well as several native formats.

FIGURE 30.3

Imported geometry comes in without any feature history.

You can use the new Data Migration tab in the CommandManager to find tasks that support imported geometry. The tools on this tab are shown in Figure 30.4.

FIGURE 30.4

SolidWorks 2010's new Data Migration tab on the CommandManager help you find import tools.

The tools on the tab, listed in order from left to right are

- Open
- Imported Geometry
- Import Diagnosis
- Check
- Draft Analysis
- Recognize Features (FeatureWorks)
- Heal Edges
- Knit
- Move/Copy Bodies
- Move Face
- Delete Face
- Replace Face
- Split
- Combine
- Rip (sheet metal)
- Insert Bends (sheet metal)

Many of these tools are not directly related to imports, but may be frequently used with imports. FeatureWorks is also a part of the SolidWorks Professional bundle, and as such is beyond the scope of this book; however, it is mentioned as part of one possible workflow for editing imported geometry toward the end of this chapter.

Gaining experience with imports

When SolidWorks imports data from another CAD program, the result is an Imported feature in the FeatureManager. The example shown in Figure 30.3 is the situation you are typically looking for: the result as a single solid body. Frequently, imports do not come in this clean. When imports start giving you trouble, you will see errors on a single body, or possibly multiple bodies, or even surface bodies. SolidWorks can address some types of errors automatically, you can address some manually, and you will probably want to avoid doing anything for some altogether.

The best way to start to feel comfortable with imported data is to be exposed to a wide range of files, some that work and some that don't. This chapter is not intended to be a short course on import repair, but repair is certainly part of the reality of working with imported data. When imports fail, it is not often because of SolidWorks; it is often because the parent software fails. SolidWorks import tools are very good and have improved over time.

Understanding the results of imports

When you import geometrical data into SolidWorks, you can get a number of different types of results:

- Single solid body in a part file
- Assembly of multiple parts
- Multiple solid bodies in a part file
- Surface bodies in a part file
- Combination of solid and surface bodies in a part file

When you get an assembly of parts, SolidWorks uses the default template that you have designated in Tools ➪ Options ➪ Default Templates, creates new parts, and saves them to your hard drive automatically.

Some imports also create a report file with the extension `*.rpt` or `*.err`. This file includes statistics about the entities and precision of the data, filename, units, the originating system, and also some information about errors that occurred during the import.

Figure 30.5 shows the first section of a report written for the import of an IGES file.

FIGURE 30.5

Report files can help you understand the contents of the imported file.

```
***********************************************************
         Solidworks Corporation - IGES Processor Status Report
***********************************************************
===========================================================
General Information
===========================================================
Sending System Product I.D.  : cap for test cut.SLDPRT
Receiving System Product I.D.: cap for test cut.SLDPRT
File Name                    : E:\ARE\cap for test cut.IGS
Sending System               : Solidworks 2004 by Solidworks Corporation
Preprocessor Version         : Version 5.3
File Creation Date           : 040430.111404
Model Creation Date          :
Units                        : IN
Model Space Scale            : 1
Author                       : matt lombard
Organization                 :


===========================================================
Entity Processing Information
===========================================================
```

Demonstrating some data import

I am going to import a few parts that don't come in perfectly and ask you to follow along with the files on your CD-ROM. This is not so much a click-by-click tutorial as a "watch over my shoulder" demonstration with commentary.

Starting with a Parasolid import because these are the fastest and easiest, open the part called `Chapter 20 Robot Arm.x_t` from the material on the CD-ROM for Chapter 30. You can open translated format files in a couple of different ways. Many people look for an Import or Translate option in the File menu, but it's not there. You can use the Open command, and select Parasolid

from the Files Of Type drop-down list. That is one way to do it, but it is slow. I prefer to open a translated file using Windows Explorer, and drag-and-drop the file onto the SolidWorks window.

After you open the file, you will notice a couple of things. The first thing that stands out to me is that the model displays in Shaded mode, regardless of how you have the display set. For example, I like to use Shaded With Edges, but imports always set it back to Shaded.
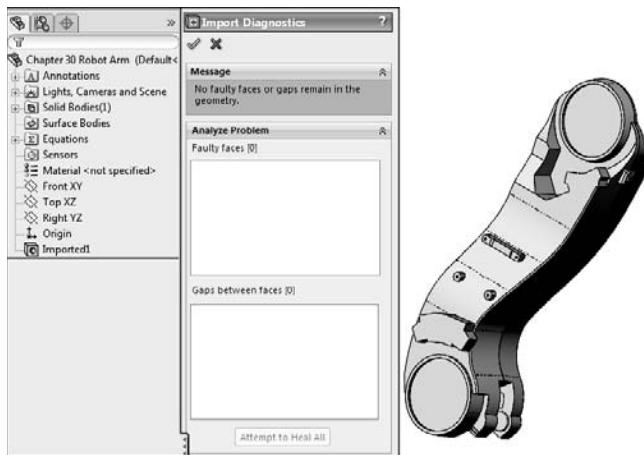
The next thing to notice is the Imported1 feature in the FeatureManager. In this case, the import was clean, so there are no warnings (yellow triangles) or errors (red circles). This is not always a good indication of the state of the part, though, because some errors that SolidWorks knows about are not displayed on the Imported feature icon. To investigate closer, right-click the Imported1 feature and select Import Diagnosis, or click Import Diagnosis from the Evaluate tab of the CommandManager.

In this case, the model really is clean. Running Import Diagnosis is the only way you can really know this. Figure 30.6 shows the FeatureManager, the Import Diagnosis, and the part itself.

FIGURE 30.6

Clean imports from Parasolid have the tendency to be fast and trouble-free.

Next, open the Parasolid file called bad `face.x_t`. This one also imports without an error or warning on the Import feature, but there is clearly a missing face. It is easier to visualize the separate faces of the part if you change the display mode to Shaded With Edges. If you examine the part closely, you can see that several faces are not lined up square with the rest of the part. Notice also that there are small sliver faces. This may be intentional, or it may be part of the problem. Triangular faces and sliver faces (with very sharp corners, usually long and narrow) are often the source of errors in translated parts.
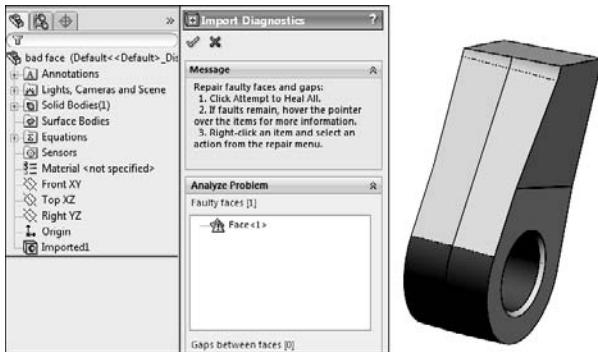
Right-click the Import feature and run the Import Diagnosis; you will see the faulty face listed. Click the Attempt to Heal All button to fix the faulty face. Figure 30.7 shows the part with the faulty face.

Three-sided and sliver faces are often the source of errors with imported parts.
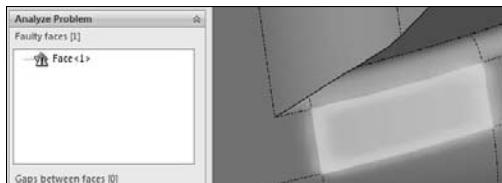
Now open the Pro/ENGINEER file called `bad face 2.prt.29`. Pro/ENGINEER files often end with version numbers as the extension of the file. You need to be careful if you see a file that looks like a Pro/ENGINEER file with no version extension. SolidWorks used the `*.prt` extension for the first couple of years, and you may still find some of those parts in circulation.

Open the file using the Import Geometry Directly option, and switch the display to Shaded With Edges, and run an Import Diagnosis. The part has a bad face. If you click on the face in the Faulty Faces list, you can see that the face is next to a small, pointy triangular face, shown in Figure 30.8. The Attempt to Heal All option takes care of it.

Importing the same data in different ways can give you different results.

Import the same file, but this time choose the Analyze the Model Completely option. It should tell you that it recognizes 13 out of 13 features. Click the Features button and watch the part rebuild.

Notice that this time the part comes up with a feature error. The error is on a fillet. It is not an accident that the previous error was on a face next to a fillet face.

Notice that the Parasolid parts came up almost immediately, but the Pro/ENGINEER parts take several seconds to process the data. As the imported files become larger and larger and branch into assemblies, this difference becomes more and more pronounced.
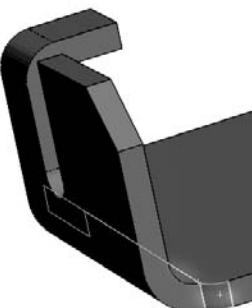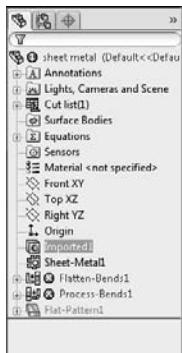
Next is a sheet metal part. Open sheet metal.x_t. Change the display to Shaded With Edges. The part looks good, and Import Diagnosis says that it is okay. Because this is a sheet metal part, and it appears to have a consistent thickness, you will try to flatten it. Click the Sheet Metal CommandManager tab, and click the Insert Bends icon (on the far right side). Select the big flat face as the face to remain stationary (the top selection box), and click the green check mark button to accept the result.

The sheet metal features are added to the part, but notice in Figure 30.9 that they have failed. A close inspection of the part reveals that there is a small ledge between the big flat face and the inside bend on both ends. This was probably a modeling error rather than a translation error. You might be able to fix this to make it usable as a sheet metal part, but for imported geometry editing, you'll need to go to the end of this chapter. Here I cover the results of attempting to repair this model.

FIGURE 30.9

Very small errors can cause special functionality in SolidWorks not to work, and repairing it is not always straightforward.
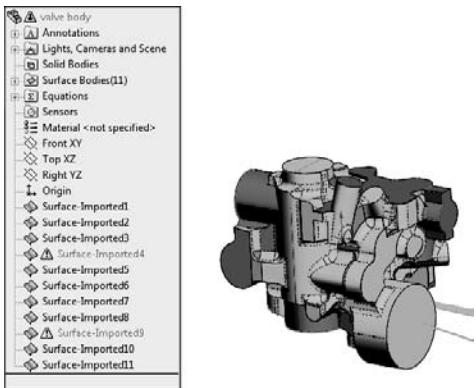
The last part I want to show in this "watch over the shoulder" demonstration is a part that I consider to be a complete loss. This is an IGES file that came from an Autodesk Mechanical product. In the CD-ROM, open the file called valve body.igs. This part takes several seconds to import. When it does import, it has 11 surface bodies, 2 of those with errors, and some obvious problems with a couple of faces that somehow became out of control. This is one of the reasons so many users do not recommend using IGES files. This type of error is more prevalent with IGES files than

other formats. Figure 30.10 shows the FeatureManager and the part on the screen. Again notice that the location where the huge problem faces come off the model are pointy triangular faces.

FIGURE 30.10

Some parts are simply beyond repair.

Note

If you get an error that says "There are no points that exist in this file" when importing an IGES file, it is probably because you have Scan To 3D selected in your add-ins list. IGES is one of the accepted file types for bringing in point cloud data, and Scan To 3D assumes you are trying to use the IGES for that. Scan To 3D is an add-in that comes with SolidWorks Premium, not with SolidWorks Standard, so it is beyond the scope of this book. To disable it, choose Tools ⇨ Add-ins and deselect the check mark in the box next to Scan To 3D. ■

If you get a part that is this bad, your first move should be to try to get better data from the source. If that isn't available, you are going to have an uphill battle trying to make this part work. Automatic healing with the Import Diagnosis isn't going to touch this part. Repair is going to be an exercise in manual surface modeling. If you don't have the patience for that, you could try solid modeling from the reference faces on the part. If you look closely at the interaction of some of the fillets, it is no surprise that the translation failed so badly. Many of the fillets are badly hacked together. In addition, if this is a casting, someone is going to need to apply draft to the part, and with all the fillets on it, this part is not going to lend itself to that very well.

Using direct converters

SolidWorks contains some direct converters that either extract the kernel data (Parasolid information) or read the actual feature data and rebuild feature-based parts. The direct converter for Pro/ENGINEER data is probably the most widely used. Figure 30.11 shows the dialog box for the Pro/ENGINEER to SolidWorks Converter.

Converting Pro/ENGINEER files to SolidWorks gives you some options.

If you use the Import geometry directly option, you just get the dumb solid. Pro/ENGINEER uses its own "kernel," or underlying geometry engine, called Granite. SolidWorks shares its kernel with NX and Solid Edge (a Parasolid). To read the geometry directly, SolidWorks has to read the Granite data stored inside the Pro/ENGINEER file and translate it to Parasolid geometry.

If you choose the Analyze Model Completely option, you get a dialog box similar to the one shown in Figure 30.12. In this case, the Pro/ENGINEER part had 42 features, and the SolidWorks converter can read them all. Here too, you are given the option to take the features or the body. It is not clear that the Body option is the same as the Import geometry directly option, but they both result in the import of the dumb solid.

You would want to click the Body option if the percentage of recognized features was low, or if you didn't really need the parametric features that much. Sometimes, even if it doesn't recognize all the features, you might still get all the sketches you need to re-create the part.

Note

When parts are re-created in this way, sketches are not made dependent on faces of the model; instead every sketch will lie on a reference plane. This is very similar to the Wide Tree approach discussed in Chapter 11. ■

If you choose the Features option, SolidWorks rebuilds each sketch and feature to build a history-based model. This is not usually what people mean when they talk about imported geometry. Many CAD neophytes assume that features just automatically transfer from one CAD system to another, but this is by far the exception rather than the norm.
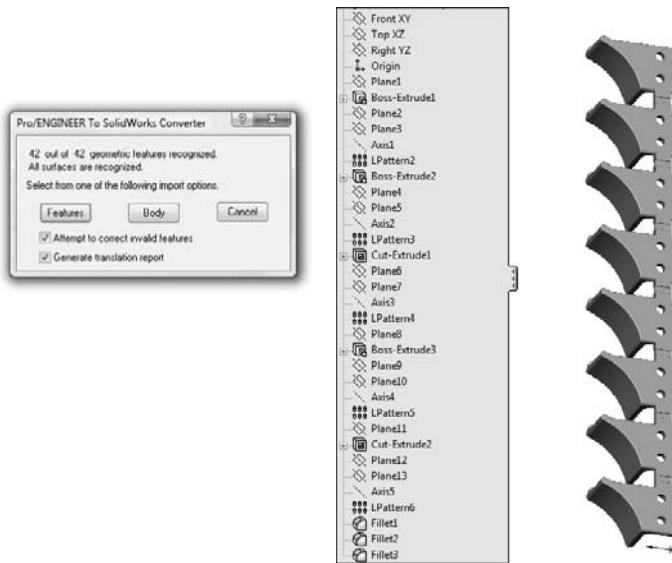
Figure 30.12 shows the Pro/ENGINEER to SolidWorks Converter dialog box and the finished imported model.

This translation depends on the version of the Pro/ENGINEER file. Check the SolidWorks documentation to see what versions of Pro/ENGINEER are supported.

FIGURE 30.12

A Pro/ENGINEER part can be translated with a complete feature tree.

Handling import errors

Import errors are usually caused by differences between the exporting and importing CAD software vendors. Some imports are more prone to errors than others. For example, the IGES format is interpreted different ways by different software vendors, and is, therefore, very prone to errors.

Another type of error is the error due to tolerance or accuracy issues. Catia is notorious for having very large tolerances on exported data. This is to enable Catia to work with large data sets more easily, but it means that when the geometry is read into SolidWorks, which typically requires more accurate data, you can see a lot of errors. The fact that the Catia to SolidWorks translation is one of the most problematic in the CAD industry seems odd because SolidWorks and Catia are both owned by Dassault Systemmes, but the difficulties have persisted for more than a decade, so it is not a technological difficulty, it is a business decision.
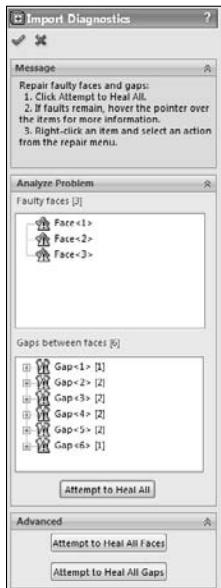
Repairing import errors automatically

SolidWorks has a tool called Import Diagnostics. Import Diagnostics can run automatically or manually to find the errors in imported data and to make repairs if possible. You can access Import Diagnostics by right-clicking an imported feature in the FeatureManager, as long as there are no native SolidWorks features that follow the imported feature. Figure 30.13 shows typical results from a faulty import.

FIGURE 30.13

Import Diagnostics helps you find and repair errors in imported data.

Sometimes the errors Import Diagnostics finds are things you can see, like missing faces, and sometimes they are things you can't see, like the edges of a face that don't intersect or inconsistent face normals. When errors can be found and repaired by Import Diagnostics, it is the best way to go. In fact, whenever I import a model, I always run Import Diagnostics on it to make sure everything is good.

Repairing import errors manually

Some errors are so big that the Import Diagnostics cannot fix them. An example of this type of error would be when a face is missing altogether from the imported data. When something like this happens, the only thing you can do is resort to surface modeling. If the import leaves you with surface bodies, and it cannot repair them automatically, you have to be able to remove the bad faces and replace them with new faces that you construct. This is all about using surface tools to create the new face, extending and possibly trimming the new face to fit into the gap caused by the bad face or faces, and then knitting everything together back into a solid. This may be an oversimplification of the workflow for manual import repair, but it is essentially the big-picture steps that you have to go through to get the task done.

There is such a thing as models that are so bad that you can't fix them, or that would not be worth your time to fix. If automatic repairs don't work, and simple manual repairs don't work, the next thing to do would be to go back to the source of the file and ask for better data.

Best Practice

When ever you export data to another CAD system, it is considered best practice (and professional courtesy) to "round trip" (save out, then read back in) the data to make sure that you can accurately read the data that you wrote out. You should not send someone else data that you cannot read back into the CAD system that created it. This may be of little comfort to you when you are the one receiving the bad data, but if you receive bad data, you have every right to go back to the creator and ask if he can round trip his data. It might be that he can adjust tolerance or accuracy settings to give you better data, or possibly that his model had some problems that he didn't know about when he exported it out to you. ∎

If you get bad data and you do not have access to the source, and automatic and manual errors prevent you from using the data, then the next best thing is to rebuild the part using the error filled data as a reference. This is never a pretty thing, but if you really need the data to be clean, and there is no other way, this is what you need to do.

You can take measurements in one file and build a new part in another file, build one file in the context of an assembly directly over the problem file, or even rebuild it as a multi-body part.

Tricking data into working

Occasionally you can employ tricks to heal problem imports. Simply saving out of SolidWorks as Parasolid and reading back in repeatedly can sometimes heal troublesome imported geometry. I frequently use Rhino to import problem files, then export from Rhino as a Parasolid. Rhino is an inexpensive surfacing application. You can read more about it at `www.rhino3d.com`. You can download and install a trial version that allows you to save 25 times. Rhino works great as a translator because it reads and writes many file types that SolidWorks does not read. Sometimes when I get a very bad IGES file, I read it into Rhino, and save it out as Parasolid, then read the Parasolid into SolidWorks. Sometimes this will repair the data to the point that SolidWorks can deal with it more effectively.

This is not to say that Rhino is a better file translator than SolidWorks, because this workaround does not always improve things. It is sometimes effective, and because it is free, the only thing it will cost you to try it is time.

You can use the same trick with other CAD packages. For example, if you know that you have an IGES file from VX, and you are having difficulty reading it into SolidWorks, it might pay to download a trial version of VX (`www.vx.com`) and see if it can import the data and re-export. It is best to use the source program to read and re-export when possible.

Ensuring that you get good data

If you can't get a SolidWorks file from someone who needs to send you data, the type of translation file that you get has a huge influence on the likelihood that your translation will be successful. Ask for data in this order:

1. **Parasolid (including native formats that use Parasolid, such as NX, Unigraphics, and Solid Edge).** Parasolid can come in text format (`*.x_t`) or binary format (`*.x_b`). You may also see file extensions such as `.xmttxt` from older versions of Unigraphics. Of

these, the binaries are smaller, but the text files have WordPad readable and editable headers that can be useful in various situations, such as correcting units or part scaling, as well as telling what the parent CAD program was.

2. **STEP (AP 214 or AP 203; Standard for the Exchange of Product model data).** The AP stands for application protocol. Most mechanical CAD programs use these two protocols, which were developed for automotive and configuration controlled design (read more at `www.steptools.com/library/standard/step_2.html`).

3. **ACIS (named for the initials of the three people and one company who created the standard: Alan, Charles, Ian, and Spatial).** ACIS creates `*.sat` files.

4. ***.VDAFS, *.VDA (Verband der Automobilindustrie Flächenschnittstelle).** A German automotive geometry transfer format.

5. **IGES (Integrated Geometry Exchange Standard; pronounced *eye-jess*).** Because of the age and lack of clear definition in the IGES format, there is little that is truly standard about it any more, and many geometry creation software packages export data that SolidWorks cannot read correctly. While this format is an old standby for old timers, it is one you probably want to avoid unless you are getting data from someone you know will give you something usable.

Another advantage of the Parasolid data is that SolidWorks reads it so quickly. A large IGES or STEP file can take minutes to read in, where SolidWorks can read equivalent Parasolid data in a couple of seconds. Once the data is read into SolidWorks, it should all be the same, with no difference between data from Parasolid and any other source, because it is all converted to Parasolid to be stored inside the SolidWorks file; but because it's now all Parasolid, you save time on the initial read.

Whether or not the data you receive is of value to you depends in part on what you want to do with it. If your data only has to be a visual representation, and not a CAD-accurate NURBS (Non Uniform Rational B Spline) model, you may be able to accept a wider range of data types. If you are looking for manufacturing quality data, some formats are simply not worth your time to deal with. These file types are mesh data that SolidWorks can read and are useful for visual data, but useless for clean NURBS data:

- `*.stl.` Stereolithography, typically used for rapid prototypes
- `*.vrml.` Virtual reality markup language, typically used for games, an old format that allowed color to be transmitted with the mesh geometry
- `*.cgr.` Catia graphics

Converting point cloud data

One of the most common import questions is how to import data from file formats such as `*.obj` or `*.3ds`, among others. These file types are mesh files, which means they are simply point cloud data. SolidWorks and most other CAD programs create data that is based on NURBS data, where the surfaces are represented by very accurate mathematics. Mesh data is represented by points in space, which is much faster to work with because it is similar to the data used by graphics cards

and drivers to display curvy shapes. Mesh data is used by Hollywood and video game computer graphics studios, and NURBS data is used by engineering and manufacturing companies. It is easy to convert NURBS to mesh, but more difficult to convert mesh to NURBS. The mesh to NURBS conversion can be done, but complex software and specialized expertise is needed for it to happen correctly.

Note

SolidWorks can make the mesh to NURBS conversion with the Scan To 3D software that is part of the Premium package. Because this is not included in the Standard package, it is beyond the scope of this book and will not be covered here. ■

Understanding the Traditional Role of Direct Edit Tools

What I call "direct editing" is also known by other names. You may hear it referred to as direct modeling explicit modeling, or history-free modeling. I call it direct editing because the way the geometry is created is all approximately the same and the difference is in the way edits are made. In fact, one of the claims to fame of direct editing software tools is that it is the only system that deals effectively with imported geometry, because "geometry is geometry," regardless of its source — native or imported. This is the reason why I cover imported geometry and direct edit tools together in this chapter.

Traditionally, direct editing CAD software has lived by the rule that how you create geometry should not affect how you edit the geometry. Therefore, direct edit CAD packages still use functions like extrude and revolve to build parts, but they would not return to those functions to edit the part; they would instead directly manipulate faces of the part by moving, offsetting, and rotating.

CAD software that depends on direct editing tools has existed for a long time: it has been around longer than parametric history-based CAD software. In the last several years, some new direct edit CAD products (Sketchup and Spaceclaim) have appeared on the scene and have renewed an interest in direct edit techniques. Until this recent resurgence, the direct edit programs were seen as old-fashioned and inferior to parametric history-based programs.

Part of the charm of the direct edit scheme was that the software didn't add any information to the model. The model could be transferred between direct edit CAD packages without losing any information. This is clearly not the case with parametric history-based CAD software, where you basically lose the ability to edit the existing model when you transfer it from one parametric history-based program to another. You can add new features, but you can't change what originally appears.

Defining the role of direct edit tools

But direct edit CAD software has not taken over the market, even with all of the recent hype. History-based modelers still dominate. There must be more to this story: the direct edit scheme does have some weaknesses. While making individual changes is easy and intuitive, making conceptual changes (changes to the design intent) is just like starting over again with direct edit. There are also situations in which certain edits cannot be made, or reversing a set of edits does not result in the original configuration. People who are used to the control and power of history-based systems are not going to relinquish that control for simple ease of use.

It turns out that the simplified capabilities of direct edit tools are more suited to non-CAD specialists who still need to work with geometrical data either upstream or downstream from the main production modeling. This means that a non-specialist can do the equivalent of a 3D napkin sketch in Sketchup and hand it off to a specialist to add the details. It means that an FEA (Finite Element Analysis) analyst can make simple edits to a model given to her by a specialist. A machinist can add stock to a finished detail model without being highly CAD proficient. I don't see CAD power users or production modelers giving up history-based CAD software and changing over to direct edit tools any time soon.

As a part of the direct edit resurgence, CAD companies have figured out how to apply the concept of parametrics (driving dimensions and geometrical relationships) directly to the solid model. Also, the history-based modelers are adding some direct-editing functionality. This cross-pollination helps bridge the gap that is now becoming mainly a discussion about history-based and history-free CAD systems. In this chapter, I do not cover the details of how each CAD company has implemented the direct editing scheme, I just want to help you understand where SolidWorks is today in the direct edit spectrum, and what you might have to look forward to as the comparison between the history-based and history-free worlds continues to take shape.

This background information is important as a part of the overall discussion of the place of these tools within the SolidWorks software. Adding direct edit tools to SolidWorks is not an earth shattering change, but of secondary importance. Notice that the topic of direct edit does not dominate this book, just one chapter out of 33. SolidWorks includes the tools because they are useful, not because they are necessary.

Understanding the strengths and limitations of direct edit tools

Some of the selling points for the direct edit CAD tools are that you don't have to worry about how a part was made (you just make the changes you want to make), and that feature trees with "history" always include feature rebuilds (which users tend to complain about taking a long time, especially for large parts and assemblies).

Without a doubt, grappling with feature order and feature rebuild times are problems that SolidWorks users face daily and often complain about.
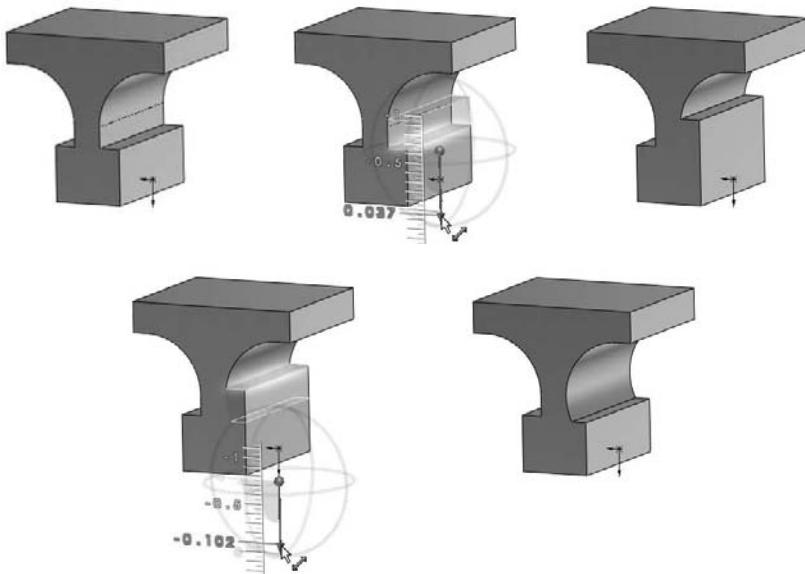
Direct edit strengths include:

- Avoid feature order problems
- Avoid rebuild time
- Changes are more intuitive
- Concept of moving faces directly is very intuitive

Probably the most serious limitation of direct edit tools is that once a face has been eliminated from the model, you cannot bring it back using more direct edit tools. For example, take the part in Figure 30.14.

FIGURE 30.14

Editing limitations with direct editing

If you change the bottom square area such that the flat face between the bottom square and the arc disappears, you cannot get it back. This limitation is less serious when the direct edit tool is just another tool within a history-based modeler, because you can resort to the more powerful history-based tools to compensate, but with a dedicated direct edit–only tool, you will be up against a more difficult task.

Using SolidWorks Direct Edit Tools
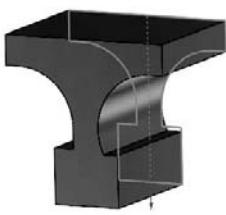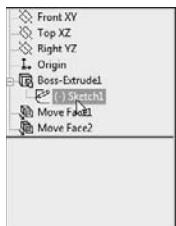
So how does SolidWorks, a history-based system, incorporate the advantages of direct editing, which is *not* history-based? What does it look like when contradictory regimes collide and start sharing ideas? SolidWorks has taken ideas from a history-free scheme and incorporated them into a history-based scheme.

First, I will show you how this works in a simple example. Using the part from Figure 30.14, Figure 30.15 shows the FeatureManager and the original sketch. Notice that the original sketch has not changed, but the part itself has. Rolling back eliminates the Move Face features, and making changes to the original sketch changes the starting points for the Move Face features when they are unrolled.

FIGURE 30.15

Direct edits in a history-based part

You may be able to imagine that in a part with a much longer feature tree, where the relationships between the faces and the features are more complex, overriding that complexity by editing a face directly could have some appeal.

Combining direct edit with history

SolidWorks has several features that do not create new geometry, but edit existing geometry only, whether it is native or imported. These include the following:

- Move Face
- Delete Face
- Freeform
- Flex
- Deform
- Scale
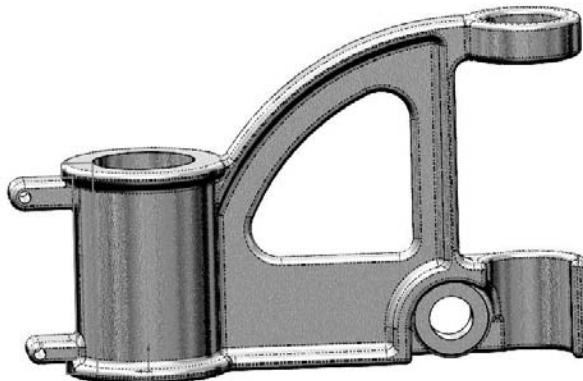
I'll show you a slightly more complex example. In traditional SolidWorks usage, if you wanted to move the face indicated in Figure 30.16, you would go back and edit the sketch or feature that was used to create it. The fillets would update to match the new geometry.

The direct edit salesmen would say that first finding the feature you need to change is difficult, then waiting for the change to rebuild all the features, and last dealing with the downstream features like fillets that might fail due to the changes is frustrating, and he would be right. The problem is that the part, as shown in Figure 30.16, cannot be changed at all in direct edit software. In the direct edit scheme, the fillets are just faces; they are not intelligent features. If you move faces they are attached, you have to also explicitly tell the fillet faces what to do as well.

FIGURE 30.16

Fillets greatly complicate direct editing schemes.
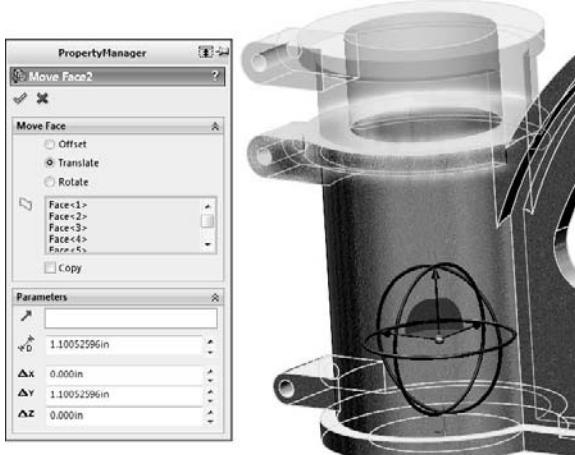
Here's how to simplify the part by suppressing fillets and using the Move Face feature to change the part (see Figure 30.17). Here, I've moved the ring around the top of the barrel as well as the mounting boss. In order to make this move, I had to select seven faces:

1. Top of ring
2. Bottom of ring on left
3. Bottom of ring on right
4. Top of boss
5. Bottom of boss
6. Full round on end of boss
7. Hole in the mounting boss

FIGURE 30.17

Using Move Face to extend the length of the barrel of the cast part after the fillets have been removed

Notice how the Web now extends up the side of the barrel. This is the type of edit that would be clumsy in SolidWorks if it were what you wanted to do. In direct edit, it is the kind of edit that would be clumsy if it were *not* what you wanted to do.

This is one of the limitations of direct edit techniques in general — not a limitation of SolidWorks, but a limitation of the concept. Direct edit cannot make new faces without adding another feature, and has problems with the intelligent type of face creation that you find in history-based systems. Direct edit programs are certainly not more powerful than SolidWorks or any other history-based CAD program. They have weaknesses in different areas, and the weaknesses are more limiting than SolidWorks history-based weaknesses.
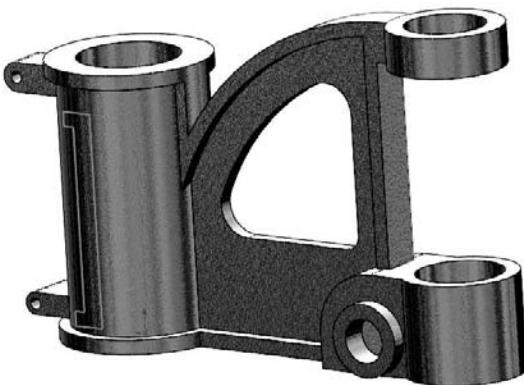
Figure 30.18 shows where the weakness of using the direct edit tools within SolidWorks begins to show up. The first feature in the part is a revolve that uses a sketch. The last feature in the part is a Move Face feature that moves the faces created by the revolve. So if you want to change the geometry of the barrel, it is controlled by two different features. If you make the sketch a specific size, the Move Face feature will change it. Move Face can only change relative to a starting point; it cannot make something a specific dimension.

Some SolidWorks power users consider this double-jeopardy condition sloppy design or bad practice. You can make a strong case that the direct edit tools should not be used on native SolidWorks parts, and that you should edit the feature the way it was created. There is certainly a place for that argument. But I know from my own modeling work that sometimes changing a feature way back in the tree can have unintended repercussions later in the tree, so using a Move Face feature late in the tree avoids fixing a lot of propagated errors. Is that a cheap, sleazy cheat? You will have to decide that for yourself.

Move Face features put some geometry in double jeopardy — it can be changed from two different places.

Combining direct editing with imported geometry

Regardless of any argument against using direct edit tools on native SolidWorks data, there can be no such argument against using them on imported data. Direct edit tools may have their limitations, but when dealing with imported geometry in SolidWorks, your choices are limited as you see from the following:

- Direct edit tools
- Cut and add modeling
- Manual surfacing tools
- FeatureWorks deconstruction

When faced with these options, direct edit tools look like the safe choice. The big problem will be that if the part is covered with fillets, you may not be able to change much. In cases like that, you might use a combination of tools, such as FeatureWorks to remove fillets, and then direct edit to make changes.

Handling imported data with FeatureWorks

FeatureWorks is not part of the scope of this book because it is a part of SolidWorks Professional, and the book is limited to SolidWorks Standard. FeatureWorks is an add-in that rebuilds imported solid models as feature-based models. It can recognize and rebuild features automatically, or semi-automatically with you guiding it through the process.

Evaluating possible future improvements to a SolidWorks history-based system

The current implementation of direct edit tools in SolidWorks does not seem finished. SolidWorks appears to not be chasing the hyped claims of dedicated direct edit CAD programs, but I do not believe that they are ignoring the development of the tools.

The key to answering the valid criticisms coming from the direct edit movement is not to abandon the proven benefits of history-based systems, but rather to make the history-based system work for users instead of against them. Most of the valid criticism could be addressed by giving users more direct control over the FeatureManager and the rebuilds.

The first thing to do is to allow users to display the FeatureManager in a linear history-only order, without indenting or consuming features to show parent-child relations. SolidWorks ordinarily takes a sketch and indents it under the feature that it is used to create, even when the sketch existed before the feature and technically should be shown above the feature. Doing this would allow users to understand and control features better.

The second thing to do is to give the user more direct control over rebuilds. If the user could put a Freeze feature in the tree so that nothing above the Freeze ever rebuilds, this would control time wasted in rebuilds for things that have not changed, but still keep the advantages of history-based models after the Freeze feature.

Direct Editing has advantages over history-based modeling, but it also has weaknesses. The strengths are not so great that users need to abandon history modeling. I believe that history-based modeling as implemented by SolidWorks can be modernized in such a way as to erase the comparative weaknesses and build on the strengths.
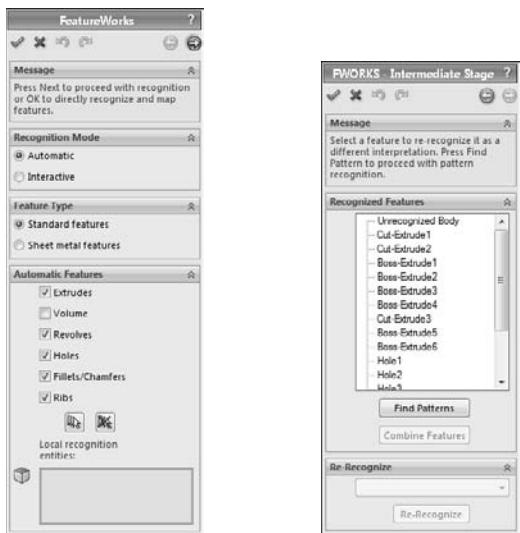
FeatureWorks works by deconstructing, or "unbuilding," the part, removing faces of features that would be applied last. For example, first it will try to recognize fillets, and removes them from the model. Then it recognizes other types of features, including holes, and then finally it recognizes the base extrude or revolve feature.

You can use this as part of your process, but only deconstructing the part partially, leaving the portion of the part that is not assigned to a feature as an imported body. If you remove the fillets and the holes, you might be able to do the edits that remain using direct edit tools. This will leave you with a partially imported part with some parametric features and some direct edit features. Some folks might consider this a sloppy mess, but getting work done on schedule does count for something; if that is what it takes, then it removes the sting to some extent. The FeatureWorks PropertyManager is shown in Figure 30.19.

Use FeatureWorks to fully or partially deconstruct models to allow direct edit tools to work.

Tutorial: Importing and Repairing Solid Geometry

This tutorial walks you through the button-clicks to import CAD data from IGES and make some edits.
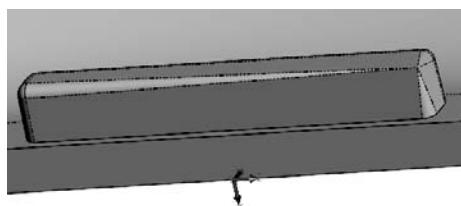
1. **From the CD-ROM, open the file called** `cover.igs`. You can open this file by selecting it in the Open dialog box (File ⇨ Open) or the dragging and dropping it from the Windows Explorer.

2. **Right-click -the Imported1 feature and run the Import Diagnosis.** Notice that Import Diagnosis finds no errors on the part.

3. **Look at the inside of the part, in the area near the part origin. Notice on one end of the tab, all the fillets come to a point, as shown in Figure 30.20.** This is clearly not right, although Import Diagnosis did not identify it as a problem.

FIGURE 30.20
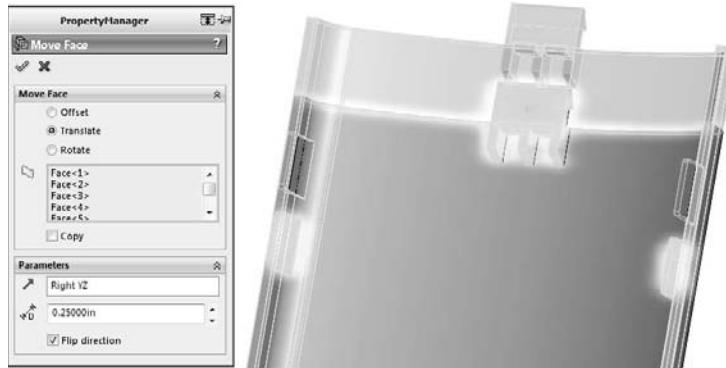
A corner where three fillets come together is not correct.

4. **Open a sketch on the flat face from which the tab protrudes.** Right-click any edge where the tab intersects the sketch face, and select Select Tangency. Then click the Convert Entities toolbar button, and click the green check mark icon to accept the result, and exit the sketch.

5. **Right-click one of the long faces of the tab and select Select Tangency.** This selects all but two faces of the tab. There are 16 faces altogether. Activate the Delete Face feature, and select the Delete and Patch option, select the two remaining faces of the tab, then accept the result. The model should remain a solid model, and the tab should be replaced by a smooth face.

6. **Select the sketch created in Step 4 and extrude it to blind depth of 0.050 inch.** Apply a fillet around the end face with a radius of 0.010 inch.

7. **Activate the Move Face feature, and select all the faces of the two side tabs and the snap feature on the end.** Use the Translate option, and the Right (YZ) plane as the direction, and use a distance of 0.25 inch. You should have no less than 21 faces selected. Faces that are parallel to the translate direction do not need to be selected, because they are automatically extended or trimmed to fit. If you are having trouble selecting the right faces, refer to Figure 30.21, and open the finished part on the CD-ROM called `Tutorial1finished.sldprt`.

8. **Save the part under a different name and exit.**

FIGURE 30.21

Moving tabs and snap in a single Move Face feature

Tutorial: Flex and Freeform

This tutorial steps you through importing and editing geometry using the Move Face, Flex, and Freeform features in SolidWorks.

1. **Open the Parasolid file from the CD-ROM called** `ellipse.x_t`.

2. **Increase the size of the part by using the Move Face feature with the Offset option.** You can use the axis of the triad handle to drag the offset distance or key it in in the PropertyManager. Use an offset distance of about 0.08 inch. Make sure only the elliptical face of the model is selected.

3. **Activate the Flex feature, and use the settings shown in Figure 30.22:**
   - Select the body in the graphics window in the top selection box.
   - Select Bending option in the Flex Input section.
   - Select Hard edges in the Flex Input section.
   - Select 190deg for the angle.
   - In Trim Plane 1, set the distance to 1.9 inch.
   - In Trim Plane 2, set the distance to 4.5 inches.

   Accept the feature when you are done.
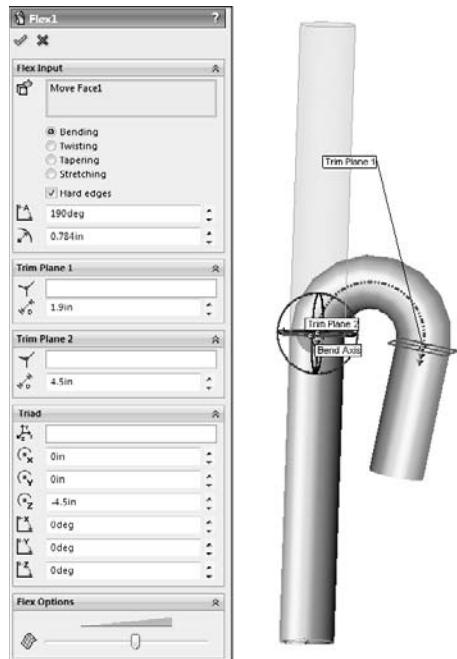
4. **Open a sketch on the Front (XY) plane, orient the view normal to it, and then sketch an ellipse with one end of the major axis at the origin and the rest dimensioned, as shown in Figure 30.23.**

965
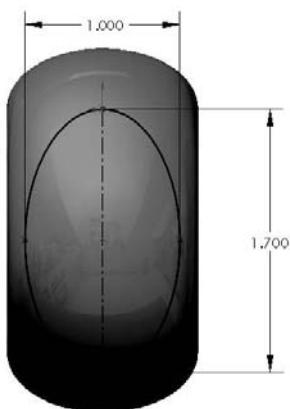
Using Flex Bending to bend an imported part

Sketching an ellipse

5. **Initiate a Split Line (Insert ⇨ Curves ⇨ Split Line) and select the bent face of the part.** Accept the feature when you are done.

Note

The Split Line will split the outside and the inside of the curved face. If you select Shaded With Edges, you will be able to see the split on both sides. ■
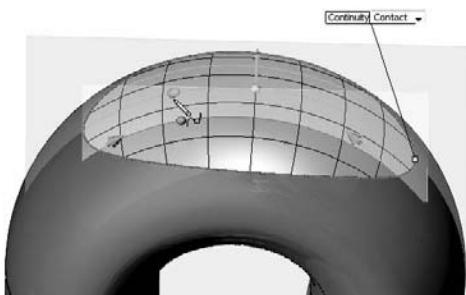
6. **Initiate a Freeform feature, and select the elliptical split face on the outside of the bend created by the Flex feature.**

   Click the Direction 1 Symmetry option, and a gray plane should appear along the Right (YZ) plane.

7. **Click the Add Curves button, and then snap the cursor to the symmetry plane and click to add a curve.** The symmetry plane will highlight orange when it is selected. Add a second curve parallel to the first one about one-third of the way from the symmetry plane to the edge of the split.

8. **Click the Add Points button, and place a point approximately, as shown in Figure 30.24.**

FIGURE 30.24

Locating a point

Place a point on the second curve in approximately the same location as the first point. Figure 30.24 shows one point on each curve.

Click the Add Points button to deselect it when you are done.
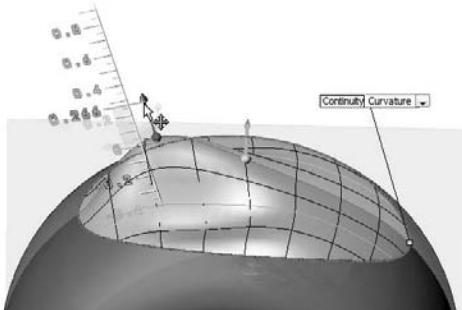
9. **Change the Continuity flag pointing to the edge of the split from Contact to Curvature.**

10. **Click on the curve on the symmetry plane, then click on the point on that curve.** Drag the arrow handle to pull the point away from the part approximately, as shown in Figure 30.25.

FIGURE 30.25

Pulling the point to create a freeform shape from the existing face

11. **Click on the second curve, and then click and move the second point in a way similar to the first point.** When you are done, the part should look similar to Figure 30.26.

FIGURE 30.26

The finished freeform surface

12. **Save the file as a different name.**

Summary

In recent years direct edit tools have received a lot of hyped press and marketing attention. However, CAD tools dedicated to working primarily as direct edit tools are not going to overtake the tools used by professionals who design and model for production. If anything, they will be used primarily by non-CAD specialists for simple editing and simple concept development, and possibly by downstream data consumers like FEA analysts or CNC (Computer Numerical Control) machinists.

The direct edit tools available within SolidWorks are powerful, and are becoming more powerful with each release. While they might be best applied to imported data, they can also be applied to native SolidWorks data. This brings up questions of best practice and duplication of effort. Sometimes the changes involved in editing a feature that is near the top of a long feature tree can be time-consuming compared to simply moving a couple of faces.

Using Weldments

Weldments in SolidWorks are built on driving structural profiles along sketch entities in a multi-body part environment. Weldment members can be curved, you can make them using standard or custom profiles, and you can build them from both 2D and 3D sketches. A Cut list within the part keeps track of the length of each profile that is needed to fabricate the weldment. Weldments are specialized parts that are similar in some ways to sheet metal parts.

You can use weldments for round or rectangular tubular structures, structures made from channels, flanged sections, standard or custom shapes, gussets, and end caps, and they can also represent weld beads in the part. You can also use weldments to create structures that are bolted together, structural aluminum extrusion frames, vinyl window frames, and wooden frames and structures, and you can put them into assemblies with other parts such as castings, sheet metal, and fabricated plate.

Sketching in 3D

The 3D sketch is an important tool for creating weldments (and many other features) in SolidWorks. Structural frames are a large part of the work that is typically done using weldment functionality in SolidWorks, and frames are often represented as 3D wireframes. You can do this with a combination of 2D sketches on different planes, with a single 3D sketch, or with a combination of 2D and 3D sketches. If you have confidence in your ability to use 3D sketches, then that is the best way to go. 3D sketches can be challenging, but they are certainly manageable if you know what to expect from them.
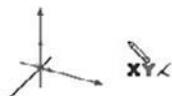
Earlier chapters discuss the tools that are available for 3D sketches; this chapter covers techniques for 3D sketching.

Navigating in space

When drawing a line in a 3D sketch, the cursor and Origin initially look like those shown in Figure 31.1. The large red Origin is called the *space handle*, with the red legs indicating the active sketching plane. Any sketch entities that you draw lie on this plane. The cursor also indicates the plane to which the active sketching plane is parallel. The XY graphic shown in Figure 31.1 does not mean that the sketch is going to be *on* the XY plane, just parallel to it.

FIGURE 31.1

The space handle and the 3D sketch cursor
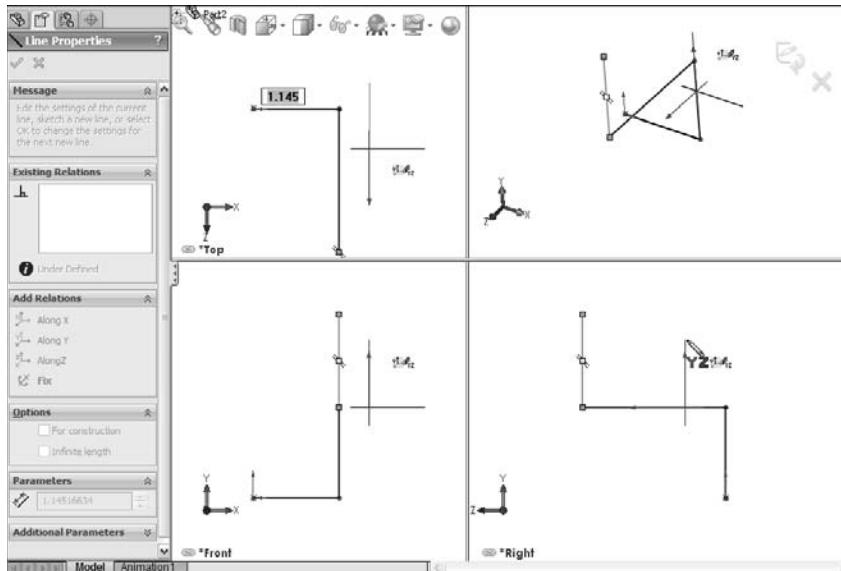
Pressing the Tab key causes the active sketching plane to toggle between XY, YZ, and ZX. The active sketching plane indication does not create any sketch relations; it just lets you know the orientation of the sketch entities that are being placed. If you want to create a skew line that is not parallel to any standard plane, you can do this by sketching to available endpoints, vertices, Origins, and so on. If there are not any entities to snap to, then you need to accept the planar placement, turn off the sketch tool, rotate the view, and move one end of the sketch entity.

An excellent tool to help you visualize what is happening in a 3D sketch is the Four Viewport view. This divides the screen into four quadrants, displaying the Front, Top, and Right views in addition to the trimetric or isometric view. You can sketch in any of the viewports, and the sketch updates live in all the viewports simultaneously. This arrangement is shown in Figure 31.2. You can easily access the divided viewport screen by clicking buttons on the Standard Views toolbar. You can also manually split the screen by using the splitter bars at the lower-left and upper-right ends of the scroll bar areas around the graphics window. These window elements are also described in Chapter 2.

When unconstrained entities in a 3D sketch are moved, they move in the plane of the screen. This can lead to unexpected results when you are viewing something at an angle, moving it, and then rotating the view, which shows that it has shot off into deep interplanetary space. This is another reason for using the Four Viewport view, which enables you to see what is going on from all points of view at once.

FIGURE 31.2

The Four Viewport view

Sketch relations in 3D sketches

Sketch relations in 3D sketches are not exactly the same as in 2D sketches. Improvements have been made in the past several versions, but 3D sketches still lack some important bits of functionality. Pierce is not applicable in a 3D sketch, and is replaced by Coincident, because in 3D sketches, there is no difference between Pierce and Coincident. Relations are not projected into a plane in a 3D sketch the way they are in 2D.

On the other hand, several other relations are available in 3D sketches that are not found in 2D sketches, such as AlongX, AlongY, AlongZ, and OnSurface.

As mentioned earlier, relations in 3D sketches are not projected like they are in 2D sketches. For example, an entity in a 2D sketch can be made coincident to an entity that is out of plane. This is because to make the relation, the out-of-plane entity is projected into the sketch plane, and the relation is made to the projection. In a 3D sketch, Coincident means Coincident, with no projection.
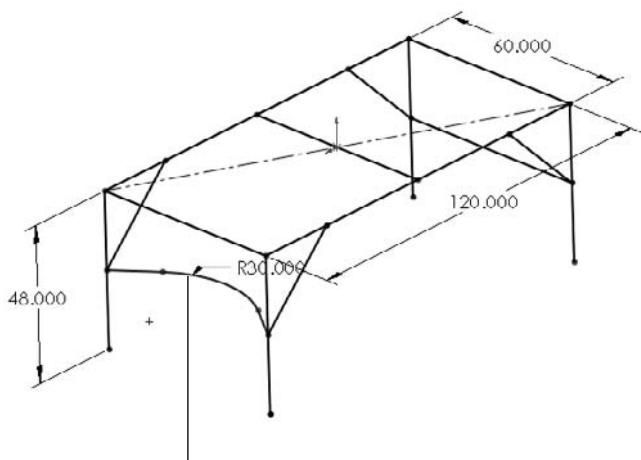
As a general caution, keep in mind that solving sketches in 3D is more difficult than it is in 2D. You will see more situations where sketch relations fail, or flip in the wrong direction. Angle dimensions in particular are notorious in 3D sketches for flipping direction if they change and go across the 180-degree mark. When possible, it is advisable to work with fully defined sketches, and also to be careful (and conservative) with sketch relations.

For example, the sketch shown in Figure 31.3 cannot be fully defined without also overdefining the sketch. The main difficulty is that the combination of the tangent arc and the symmetric legs of the end brace cannot be located rotationally, even using the questionable reliability of 3D planes that are discussed next. The only workable answer to this is to create a separate 2D sketch on a real 2D sketch plane, where the plane is defined by the elements of the 3D sketch.

FIGURE 31.3

Three-dimensional sketches may be difficult to fully define.

This set of sketch entities cannot be located rotationally within the 3D sketch

Planes in space
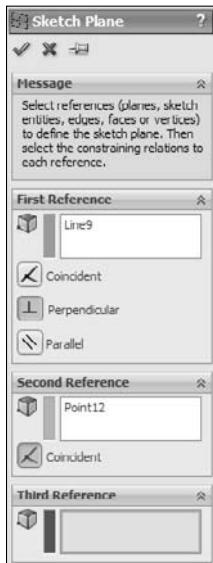
It is possible to create planes directly in 3D sketches. These planes work like regular planes starting in SolidWorks 2010, where they are defined by constraints and selections rather than selecting a type of method to define a plane. Sketches can be created on these planes, and move with the planes. Having planes in the sketch also enables planar sketch entities such as arcs and circles in 3D sketches.

Unfortunately, there is a lot to watch out for with 3D planes, as they are called. The first thing to watch out for is that they do not follow their original definition like normal Reference Geometry type planes. Figure 31.4 shows the PropertyManager interface for creating 3D planes; however, keep in mind that the plane does not maintain the original relation to these initial references. The parent-and-child relations that SolidWorks users are used to are suspended for this one function, or work in the reverse from what you normally expect.

FIGURE 31.4

The 3D planes PropertyManager

3D planes cannot be fully defined unless there is some sketch geometry on the plane that is in turn related to something else. Limited types of sketch relations can be applied directly to the plane itself. Horizontal and Vertical relations cannot be applied directly to the plane to orient it. Horizontal and Vertical relations of entities on the plane are relative only to the plane and not to the rest of the part; therefore, making a line horizontal on the plane does not mean anything when the plane rotates (which it is free to do until it is somehow constrained to prevent this).

Beyond this, when a plane violates a sketch relation, the error is not reported, which severely limits the amount of confidence that you can place in planes that are created in this way. The biggest danger is in the plane rotating, because that is the direction in which it is most difficult to fully lock down. The best recommendation I can make here is to suggest you create reference sketch lines with relations to something stable, preferably outside of the 3D sketch.

If you choose to use these planes to activate the plane for sketching, you can double-click the plane with the cursor. The plane is activated when it displays a grid. You can double-click an empty space to return to regular 3D Sketch mode. The main thing that you give up with abandoning 3D sketch planes is the ability to use the dynamic drag options when all loft or boundary sketches are made in a single 3D sketch (something I have only used once to demonstrate the idea).

Planar path segments

Some path segments that are allowed in 3D sketches can only be used if they are sketched on a plane. These entities include circles and arcs, and can include splines, although splines are not required to be on a plane. I have already been mentioned that to sketch on a 3D Plane (a plane created within the 3D sketch), you can simply double-click the plane.

To sketch on a standard plane or reference geometry plane, you can Ctrl+click the border of the plane with the sketch entity icon active or double-click the plane. The space handle moves, indicating that newly created sketch entities will lie in the selected plane.

Dimensions

Dimensions in 2D sketches can represent the distance between two points, or they can represent the horizontal or vertical distance between objects. In 3D sketches, dimensions between points are *always* the straight-line distance. If you want to get a dimension that is horizontal or vertical, you should create the dimension between a plane and a point (the dimension is always measured normal to the plane) or between a line and a point (the dimension is always measured perpendicular to the line). For this reason, reference sketch geometry is often used freely in 3D sketches, in part to support dimensioning.

Using the Weldment Tools

Like the Sheet Metal tools, the Weldment tools in SolidWorks are specialized to enable you to create weldment-specific features in a specialized environment. Everything starts from a sketch or set of sketches representing the wireframe of the welded Structural Members.

Weldment
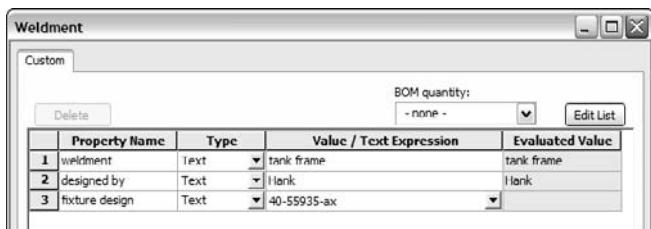
The Weldment button on the Weldment toolbar simply places a Weldment placeholder in the FeatureManager. This placeholder tells SolidWorks that this part is a special weldment part, much in the way that the Sheet Metal feature in sheet metal parts is a placeholder, and denotes a special part type. The Weldment feature moves to the top of the tree, regardless of when it is created in the part history. If you do not create a Weldment feature manually, then one is automatically created for you and placed at the top of the tree when the first Structural Member feature is created. Structural Members are discussed next in this chapter.

This feature offers only a few special default settings: the ability to set custom properties that transfer to all Cut list items that are created in the current part, and the fact that the Merge Result option is deselected by default in Weldment parts. The former is important when multiple weldments go together to make an assembly. To access the custom properties interface, shown in Figure 31.5, select the Properties option on the Weldment feature right mouse button (RMB) menu.

FIGURE 31.5

The Weldment Properties interface

Structural Member

A Structural Member is the basic building unit of weldments in SolidWorks. You can create a Structural Member by extruding a profile along one or more path segments, and it may result in a single body or multiple bodies. The path segments may be in the form of 2D or 3D sketches.

Note

A single Structural Member feature may create multiple bodies, with each body corresponding to a single cut length of stock. In other words, the feature name "Structural Member" does not necessarily refer to a single piece of the weldment, although it may. ∎

One limitation of the use of sketches in Structural Member features is that only two selected sketch entities may intersect at any one location. For example, at each corner of a cube, three path segments intersect, and so you can only select two of those elements at one time to create a Structural Member feature. Because each of the path segments requires a piece of metal, the leftover path segments may be used by a second Structural Member feature.

When creating the sketch for the weldment, it is important to decide what the sketch represents. For example, does it represent the centerline of the structural elements, or does it represent a corner? You can orient and position structural shape profiles relative to the frame sketch in several ways, with positioning at the shape centroid being probably the most intuitive for closed shapes and a corner being most intuitive for angle channels.
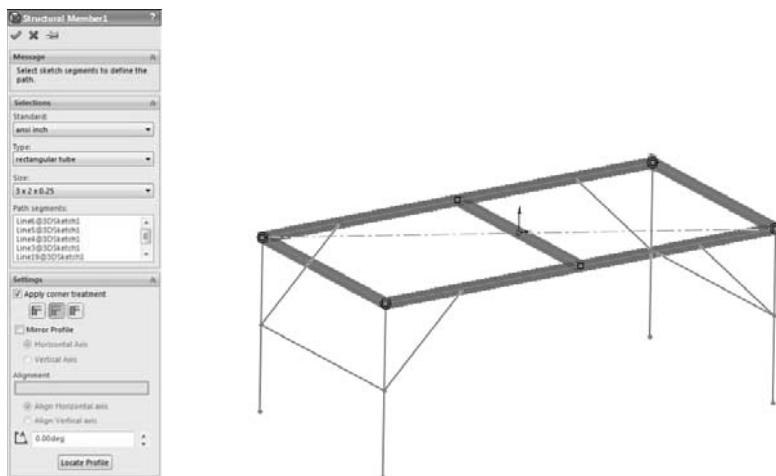
Figure 31.6 shows a single 3D sketch of a simple frame and a Structural Member feature in the process of creation. You must select the standard first, then the type, and finally the size. A limited number of profiles come with the software, and although it is very likely that you will need to create some custom profiles, fortunately they are very easy to create.

To access a large number of weldment profiles in various standards, open the Design Library and click the SolidWorks Content icon. Under that, the Weldments folder has several zip files containing weldment profiles. Ctrl+click an icon to download the file, and then extract the contents of the Zip file to the library location you have established for your weldment profiles.

FIGURE 31.6

A 3D sketch of a frame

Groups

The concept of groups is simple. You can group selected path segments within a structural member feature into two kinds of groups: Parallel or Contiguous. A single structural member may have multiple groups.

Parallel groups contain parallel path segments that do not touch. Parallel groups also have the curious requirement that you have to select the structural profile before you can select more than one path segment.
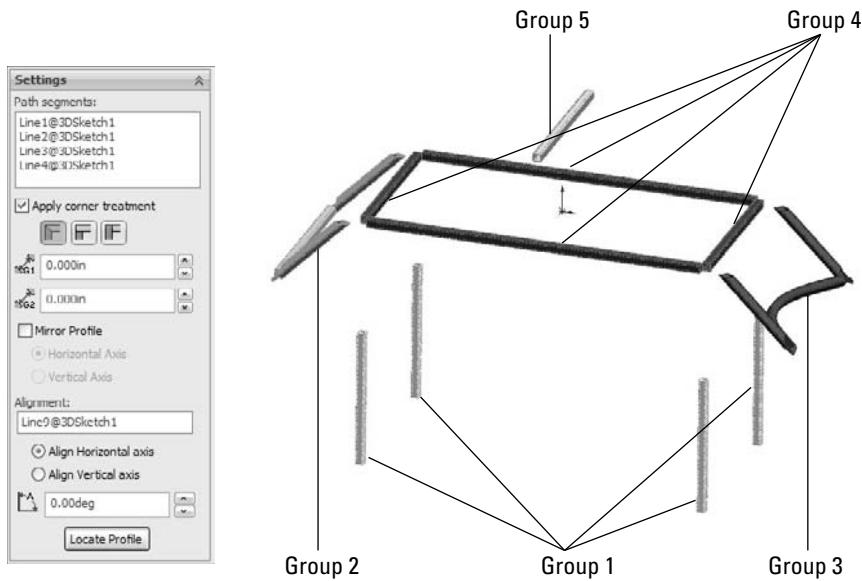
Contiguous groups contain path segments that touch end to end, two segments at a time. A contiguous group cannot have one path segment intersect in the middle of another or more than two path segments intersecting at a corner.

Each group can only have a single orientation of the structural profile. If for example, each frame leg needed to have the profile rotated to a different orientation, you would need to do the legs in four separate groups instead of all in a single group.

Given those requirements, if the frame shown in Figure 31.6 were to be created entirely from the same structural profile, say ANSI (American National Standards Institute) inch, square tube, 3 × 3 × .025, it would require a minimum of five groups, as Figure 31.7 shows, in exploded form. The file used to create this image is included on the CD-ROM under the name `Chapter 31 – Weldment groups.sldprt`.

FIGURE 31.7

Using groups to create the welded frame

The main advantage of the groups functionality is that each member within the group is automatically trimmed to other members of the group and the control of gaps within or between groups. The only trimming you need to take care of separately is the trimming between members of different groups.

Locating and orienting the profile
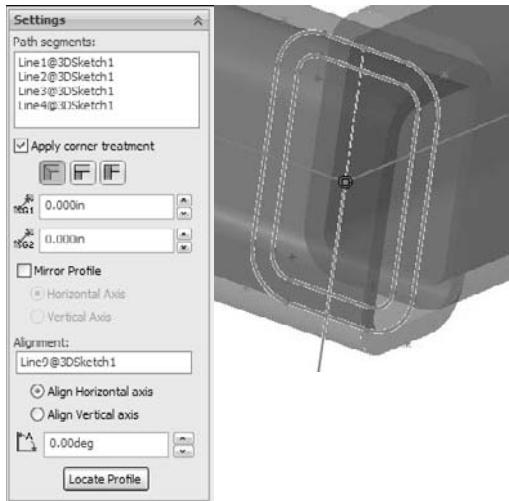
When you apply a profile to a path segment in a Structural Member feature, the profile must have some relationship to the path segment. The default point where the path "pierces" the profile is at the sketch Origin. To change the pierce point, you can click the Locate Profile button at the bottom of the Structural Member PropertyManager, which zooms the view to present the profile sketch so that you can select another sketch point to use as the pierce point. You can select any sketch point on the profile, including endpoints, sketch points, and virtual sharp points if they are present in the sketch.

Profile sketches are generally surrounded by several sketch points, which may seem unnecessary until you consider that you can use any of the points to position the profile. The Settings panel at the bottom of the Structural Member PropertyManager is shown in Figure 31.8 and displays a profile sketch with the interface.

FIGURE 31.8

Locating the profile

In addition to locating the profile sketch, you can also rotate the profile using the Angle field in the Settings panel. This rotates all the bodies that are created by the Structural Member feature at the same time. In the example of the four-legged frame, if the legs are rectangular or circular, they can all be created in the same Structural Member feature because they are all rotated in the same way. However, if the legs are made from an asymmetrical shape such as an angle, then each leg will need to be made using a separate Structural Member feature, with each leg rotated differently.

Disjoint sketch segments

You can select disjoint sketch segments in a single Structural Member feature if they are parallel to the first segment and use the same profile height location and orientation. For example, in Figure 31.6, notice the four angled supports in the corners attaching to the legs. Because they are parallel in pairs, all four of these supports could not be made in a single group. Later in this section, when those path segments are actually used to place Structural Members, the additional requirement of using an angle profile means that the profiles each need to be rotated differently from one another, and thus cannot be used in a single group.
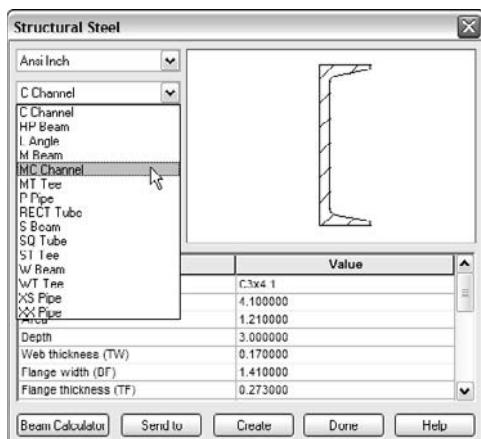
Custom profiles

Most of the custom profiles that you will need may be simply new sizes of existing profiles. You can easily create a custom profile by opening an existing profile, editing it, and saving it under a different name using the Save As command. It is important to note that when creating a weldment profile, a sketch must be selected prior to initiating the Save As command. Weldment profiles are library features, and use a *.sldlfp filename extension. Each size must be saved as a separate library feature in order to appear in the selection list. While library features are configurable, the configurations are not selectable for weldment profiles.

Other sources for custom profiles include 3D Content Central, which has a large number of erector-set aluminum extrusion profiles and the accessory hardware for those systems. Toolbox also has a Structural Steel sketch generator, shown in Figure 31.9, which enables you to generate most standard shapes. If you have Toolbox installed on your system, then you can find this tool in the Toolbox menu.

FIGURE 31.9

The Structural Steel sketch generator interface

As I have said throughout this book, weldment profiles are a great candidate for storing in your special library folder, separate from the SolidWorks installation directory. To establish this library location, you can choose Tools ⇨ Options ⇨ File Locations ⇨ Weldment Profiles. Also keep in mind that if you share design duties with other users, then the library location should either be shared among users on a network, or the libraries should be copied to each user's local library. You can also share library data through a Product Data Management, or PDM, program.

If you are creating completely new custom profiles, then remember that when locating the profile relative to the path segments, you can use any sketch point. As a result, you should provide ample selections for pierce points. Virtual sharps function well around filleted corners, as well as sketch points at the centroid of a shape.

In addition to sketch geometry, the library part files should also contain custom property information about the structural shape, such as part number, supplier, material, and so on. This information propagates to the Cut list.

Corner treatments

Any intersection of sketch lines at mutual endpoints within a single group, except as noted in this section, creates a situation that requires that the corners be cut to match. Figure 31.10 shows an example of the options that are available when lines meet at right angles. Notice that within a group, you have the option to set a weld gap at the intersections.

FIGURE 31.10

Corner treatment options

To access the toolbar with the Corner Treatment options, you can click the pink dot at the intersection of the path segments. Default corner treatment settings are made in the Structural Member PropertyManager, but they may need to be adjusted individually.

Two situations do not require corner treatments. The first situation is when a line intersects another line at some location *other* than an endpoint in the same Structural Member feature; for example, a support meeting the main member in the middle. In this situation, the member that ends in the middle of the other member is trimmed to a butt joint. The second situation is when an intersecting member is created by a later Structural Member feature. You deal with this situation by using the Trim/Extend function, which I describe later in this chapter.

Note

You may encounter a situation where it seems like a good idea to create collinear sketch segments. In a typical extrusion, the faces created from collinear lines are simply merged together as one. However, in a weldment, this does not work when it is done in a single feature. In order to create Structural Members on collinear sketch lines, you must either extend one line to encompass the length of both lines or do the work in two separate Structural Member features. ∎

Arc segments

When arc sketch segments are part of the selection for a Structural Member, a Merge arc segment bodies option displays after the selection box in the Selections panel. This means that any *tangent* arc segment will be joined to the entities to which it is tangent, but any non-tangent entities will create separate bodies.

A tangent arc is illustrated in the curved leg brace shown in Figure 31.11, along with the Merge arc segment bodies option in the PropertyManager.

If the Merge arc segment bodies option is not selected, then a separate body is created for arc segments. The Merge arc segment bodies option applies to the whole feature, and cannot be set selectively for individual arc segments within the selected sketch entities; it is either selected for all or deselected for all. If some arc segment bodies should be merged and others should not, then you should create separate Structural Member features.

It is also a curious limitation that only one arc may be selected if the selected path segments are disjointed. For example, the two arcs for two J shapes that do not touch could not be selected in the same Structural Member feature. The obvious work-around is to create two separate groups.

FIGURE 31.11

A tangent arc segment used in a Structural Member feature

Patterning and symmetry

Bodies created by the Structural Member feature can be patterned and mirrored. Remember that there is a difference between patterning *features* and patterning *bodies*. The Move/Copy Bodies feature is also appropriate for creating bodies to be used in the weldment, although the Structural Member feature does not create them directly.

I mention this to emphasize the point that sketching with symmetry is still important, although it is more difficult with 3D sketches than with conventional 2D sketches. Symmetry in a 3D sketch can only be used when a plane is activated, and you can activate regular reference geometry planes, not just 3D sketch planes. I also mention this because in larger weldments (or when using slower computers), performance may be an issue, and mirroring or patterning bodies is certainly a performance enhancement over building parametric features.

Configurations

When you start creating a weldment, SolidWorks automatically creates a derived configuration. Both configurations are named Default, but they have different descriptions. The parent configuration description is As Machined, and the derived, or indented, configuration description is As Welded.

This arrangement holds true for any additional top-level configurations that you create in the part; they will all get the description As Machined and inherit an identically named derived configuration with the description As Welded. These configurations are meant to help you create drawings where the raw weldment is distinguished from the weldment after it has been machined, ground, and drilled.
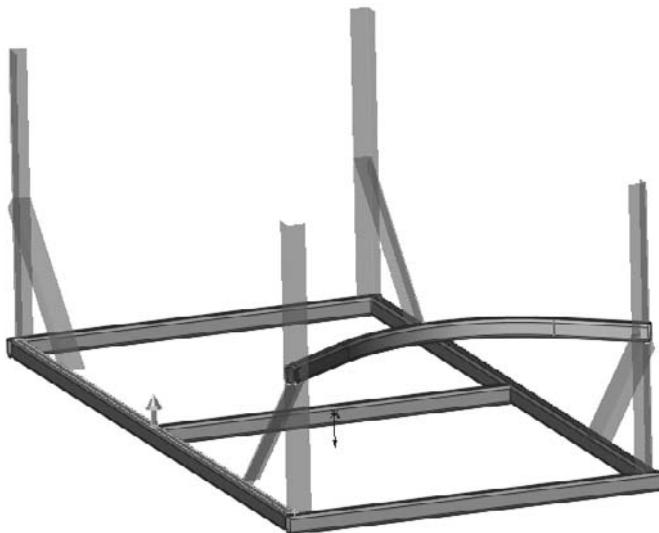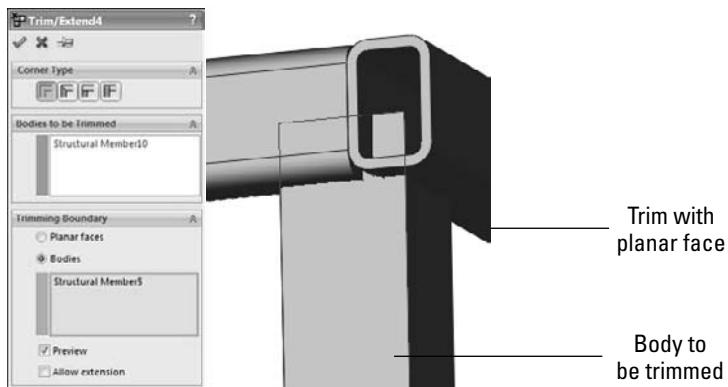
Trim/Extend

In situations where you must create multiple Structural Member features, thus creating intersecting bodies, the interferences must be dealt with using the Trim/Extend feature. An example of this is shown in Figure 31.12. The legs and braces shown are all being trimmed by a single face on the bottom side of the rectangular section of the frame, where the small arrow appears.

FIGURE 31.12

Using the Trim/Extend feature

Trim with
planar face

Body to
be trimmed
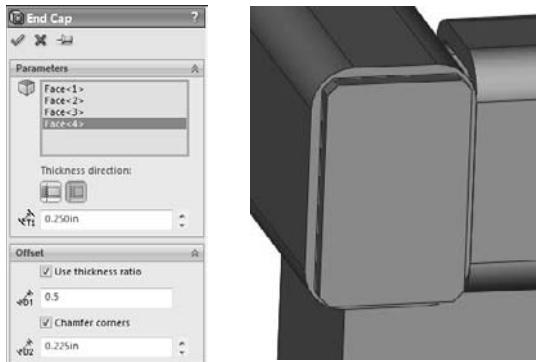
End Cap

The End Cap feature closes off an open-ended Structural Member. You can add multiple end caps in a single End Cap feature. The PropertyManager and the end product are shown in Figure 31.13.

Using the End Cap feature

The end cap using the Outward option sits on the outside face of the member, and overlaps the thickness of the member by the inverse of the Thickness Ratio that is applied in the Offset panel. If the Use thickness ratio option is turned off, then it functions as an offset from the outer faces of the member from which it is created. When this option is turned on, the thickness ratio can range from zero to one. For a value of zero, it is flush with the outer faces of the member, and for a value of one, it is flush with the inner faces of the member. Using the Inward option, the cap fits inside the hole in the member.

Gusset
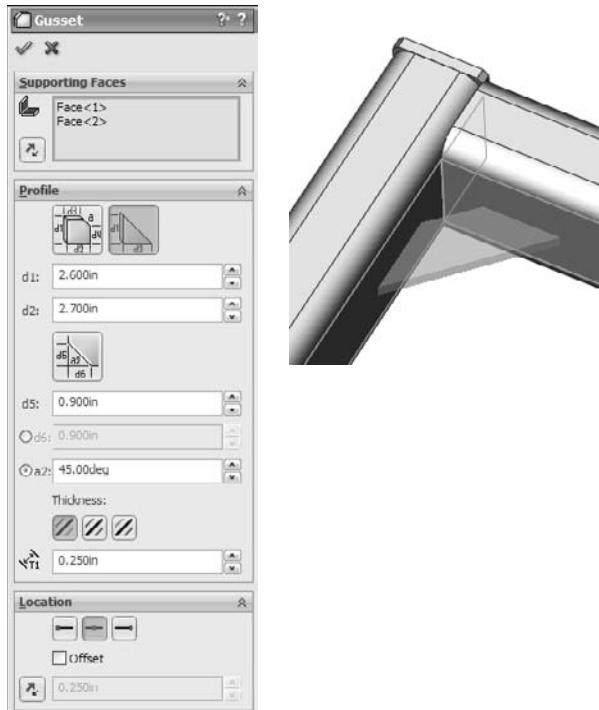
The Gusset feature creates a three-, four-, or five-sided gusset in a corner between Structural Members, as shown in Figure 31.14. You can place the gusset at specific locations along the edge in the corner, or offset it by a specific dimension in a specific direction by using the settings in the Parameters panel. You can control the size and thickness of the gusset in the Profile panel. There is no sketch for this feature type; it is simply created from the parameters that you enter in the PropertyManager interface. Again, if you need to make multiple Gusset features in succession, you can use the pushpin icon to keep the interface displayed until you close it by clicking the red X icon.

FIGURE 31.14

Using the Gusset feature
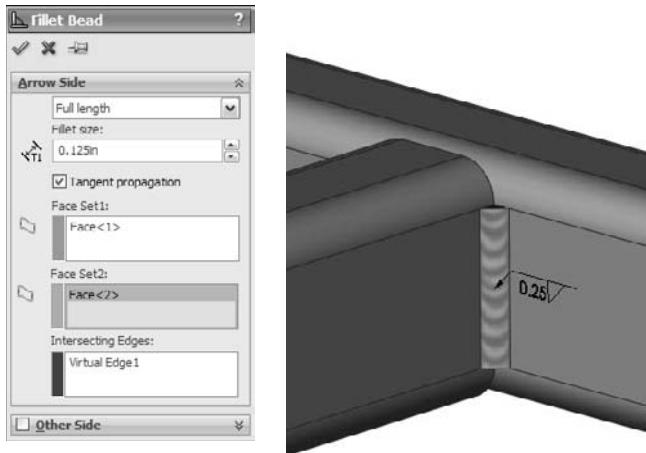
Fillet Bead

The Fillet Bead feature is somewhat limited in its capability to make realistic weld beads on a weldment part. To start with, it is limited to fillet welds, and does not weld around commonly found situations such as that shown in Figure 31.15. It works best where flat faces butt up against one another cleanly, with enough space all the way around them for the weld bead. It is often suggested that for drawings, you are better to leave the Fillet Bead feature off of the model and simply to use weld symbols or caterpillars to represent the weld bead.

FIGURE 31.15

Using the Fillet Bead interface

Using Non-Structural Components

Non-structural components are frequently needed in weldments, and include items such as feet, plates, brackets, mounting pads, castings, and other items. Simpler items that can be easily modeled in place can be placed directly into the weldment part. You can also insert parts into the weldment using the Insert Part feature, and move them into place by using dimensions or mates. In general, if any item is actually welded into the weldment, then you are recommended to place it in the weldment part; however, items that are bolted on should probably be placed into an assembly. Of course, this probably depends more on your company's documentation standards, part-numbering standards, and assembly processes than on software capabilities.
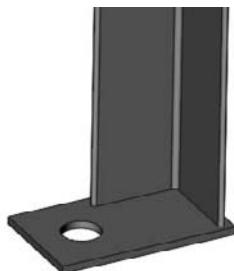
When adding a plate such as the footplate shown in Figure 31.16, the geometry is added using the standard Extrude feature, except that the Merge option is deselected by default. This ensures that non-structural components that are manually modeled, such as this part, are created as separate bodies, and not merged together with the existing structural items.

A foot plate added to the weldment

Using Sub-Weldments

From a modeling point of view, sub-weldments are generally used for either organizational or performance reasons to group together elements of a weldment or to break a larger weldment into more manageable pieces. This is in much the same way that subassemblies are created for the same purposes within larger assemblies. From a fabrication point of view, sub-weldments are also used to break a large weldment into pieces that can be transported or handled.

To create a sub-weldment, you can select several bodies from the Cut list, and then select Create Sub-Weldment from the RMB menu. (You can also select the bodies from the graphics window if you use the Select Bodies selection filter.) This creates a separate folder for the sub-weldment bodies. You can then right-click the sub-weldment folder and select Insert Into New Part.

Using Cut Lists
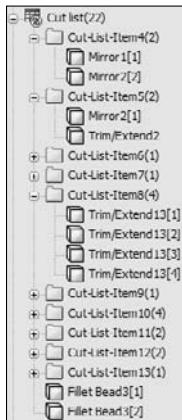
The Cut list that is maintained in the model FeatureManager is simply a replacement for the Solid Bodies folder. It has most of the same functionality as the Solid Bodies folder, as well as a few additional items. The Cut List folder symbol in the FeatureManager can show the folder symbol in one of two potential states. These symbols are shown at the top left of this paragraph. When the Cut list requires an update, the top image is shown, and after the update has been performed, the bottom image is shown. Cut lists are updated automatically when a drawing that uses the Cut list is accessed, but can also be updated manually through a RMB option or by the forced rebuild, Ctrl+Q.

You can access the Update command by right-clicking the Cut List folder and selecting it from the RMB menu. Figure 31.17 shows the result of the update. The weldment solid bodies are broken down further into subfolders that reflect quantities of identical bodies. Notice that the weld beads at the bottom of the list are not in a folder.

FIGURE 31.17

The Cut list in the model FeatureManager

Note

Remember from Chapter 26 that starting in SolidWorks 2010 you can assign different materials to bodies within a part. SolidWorks does not account for weldability of different materials. ■

Using Cut List Properties

In addition to the custom properties for the document, SolidWorks weldments also make use of Cut List Properties. You can access Cut List Properties by right clicking on a Cut List Item folder at the top of the FeatureManager and selecting Properties. The Cut List Properties may not be available for a newly created weldment until you have updated the Cut List (RMB on top level Cut List icon, and select Update).

Figure 31.18 shows the Cut List Properties dialog box, which allows users to enter data for BOMs and Cut Lists such as length and material.
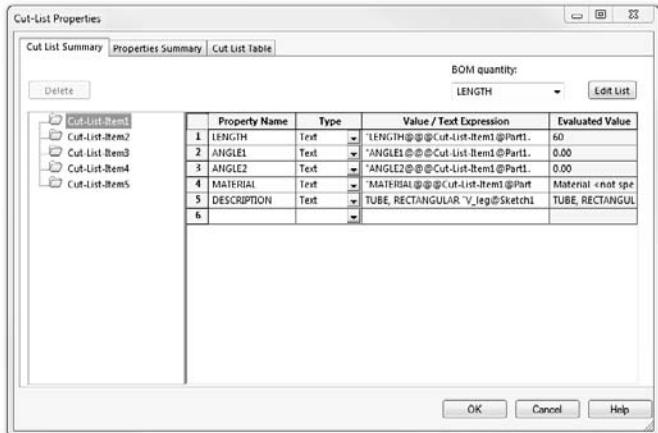
Notice also the Properties Summary tab, which enables you to look at each property, and see what the values for each Cut List item are. You would select a property such as Description, and then assign descriptions for each Cut List item, and then go on to the next property, say Material, and assign values for that property.

The Cut List Table tab shows you a preview of the Cut List, and enables you to use Cut List templates that might have different default columns established. Figure 31.19 shows the Cut List Table tab of the Cut List Properties dialog box.
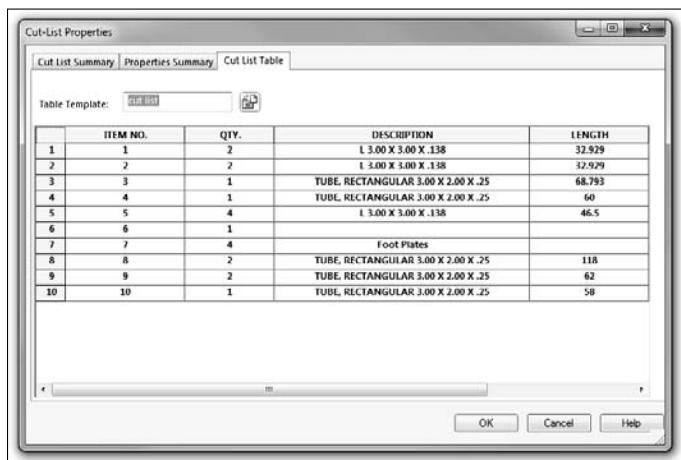
Use the Cut List Properties to enter relevant data for BOMs and Cut Lists

The Cut List Table tab previews the Cut List for you, and gives you the opportunity to edit values in the table

As Figure 31.19 shows, you may need to enter information for items added to the weldment such as end caps or gussets.

Make Weld Bead

To exclude a feature in the FeatureManager from the Cut list, you can select Make Weld Bead from the feature's RMB menu. The next time the Cut list is updated, the members that were created by that feature will be listed at the bottom of the Cut List folder with the weld beads. You can set back a feature to be included in the Cut list by selecting Make Non-Weld Bead from the feature's RMB menu and updating the Cut list again.
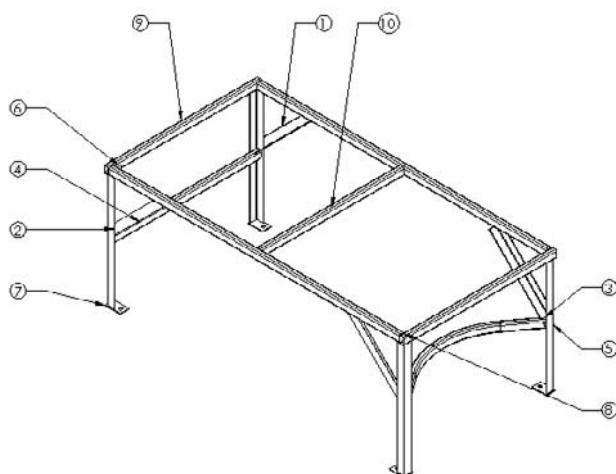
Creating Weldment Drawings

Weldment drawings have a couple of special features that distinguish them from normal part drawings. The first is obviously the Cut list. Like a BOM (Bill of Materials) in an assembly, you can place the weldment Cut list on a drawing by choosing Insert ➪ Table ➪ Weldment Cut List. Figure 31.20 shows a sample Cut list on a drawing. In this case, the blank rows represent non-structural components, being the foot plates and the gusset. You can manually add data for these parts either directly into the table or by adding it to the properties of the corresponding folder in the Cut list in the model document.

FIGURE 31.20

A Cut list on a drawing

ITEM NO.	QTY	DESCRIPTION	LENGTH
1	2	L 3.00 X 3.00 X .138	32.929
2	2	L 3.00 X 3.00 X .138	32.929
3	1	TUBE, RECTANGULAR 3.00 X 2.00 X .25	68.793
4	1	TUBE, RECTANGULAR 3.00 X 2.00 X .25	60
5	4	L 3.00 X 3.00 X .138	46.5
6	1		
7	4		
8	2	TUBE, RECTANGULAR 3.00 X 2.00 X .25	118
9	2	TUBE, RECTANGULAR 3.00 X 2.00 X .25	62
10	1	TUBE, RECTANGULAR 3.00 X 2.00 X .25	58
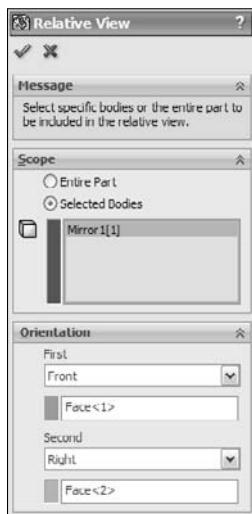
Also shown in Figure 31.19 is an auto-ballooned isometric view of the entire weldment. This works the same way that assembly auto-ballooning works, and it also corresponds to the Cut list in the same way as the assembly corresponds to the BOM.

Weldment drawings can also include views of individual bodies. You can do this by making a Relative view, selecting both faces from the same body, and then using the PropertyManager of the Relative view in the window of the solid model to control whether the view shows the entire part or just selected bodies. The Relative View PropertyManager is shown in Figure 31.21.

FIGURE 31.21

The Relative View PropertyManager

When you place a drawing view of a weldment, you can also link a Cut list to a particular view. This might assist with putting multiple weldments on a drawing or laying out the process for fabricating a weldment, showing the weldment in various stages of completion.

To access the Relative View PropertyManager interface, follow these steps:

1. **Click the Relative View button on the Drawings toolbar or choose Insert ⇨ Drawing View ⇨ Relative To Model.**

2. **Right-click a blank space on the drawing sheet and select Insert From File.** Browse to the part file.

3. **Identify the faces to be shown in the particular orientations, and specify whether the entire part or the selected bodies should be shown in the view.**

Tutorial: Working with Weldments

This tutorial guides you through building a section of a tubular truss support. You can create many different types of weldments, from simple small gauge frames to large architectural designs such as this one. This tutorial also helps you to navigate successfully through some 3D sketch functionality for creating fully defined sketches.

Follow these steps to learn about working with weldments:

1. **Open a new part.** If you have Toolbox, then activate it by choosing Tools⇨Add-Ins⇨SolidWorks Toolbox. If you do not have Toolbox, then simply draw two concentric circles on the Front plane of a new part. The circles should have diameters of 10.02 inches and 10.75 inches. Alternatively, you can copy the library feature from the CD-ROM for Chapter 31 to the location specified at the end of Step 5.

2. **If you have Toolbox, then choose Toolbox⇨Structural Steel.**

3. **Select ANSI Inch, P Pipe, P10.** This profile has an inside diameter of 10.02 inches and an outside diameter of 10.75 inches. Click the Create button, and then click Done.

4. **Use Custom Properties to add any properties that you would like to have automatically added to the Cut list.**

5. **Remembering the technique from Chapter 18 on library features, first close any open sketches, then select the sketch from the FeatureManager, and then save the part as a Library Feature Part file to a path such as** `D:\Library\Weldment Profiles\Custom\Pipe\P-Pipe10in.sldlfp`.

Note

The Custom folder (located in the first level under the Weldment Profiles) will be recognized as the Standard, similar to ANSI or ISO (International Organization for Standardization). The next folder down, Pipe, will be recognized as the Type, and the name of the file will be recognized as the Size, in the same way as shown in Figure 31.6. ■
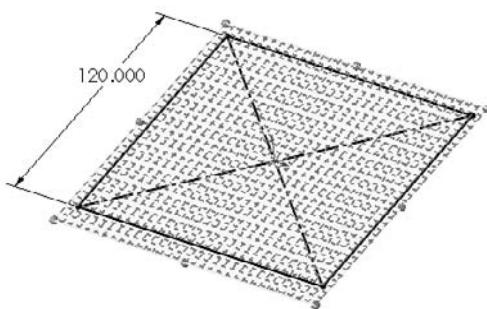
6. **Choose Tools⇨Options⇨File Locations⇨Weldment Profiles, and add your non-installation directory location to the list of folders.** Alternatively, you can remove the Program Files location from the list, and copy the files from that location to your own library location.

7. **Open another new part, and open a new 3D sketch in the part.** Double-click the Top (ZX) plane to activate it, and click the Center Rectangle sketch entity.

8. **Draw a rectangle around the Origin.** The sketch should now look like Figure 31.22. Apply an Equal relation to two adjacent sides of the rectangle, and dimension any of the lines as 120 inches.

9. **Turn off the rectangle, and double-click in blank space to de-activate sketching on the Top plane.**

10. **Activate the Line sketch tool and press Tab until the cursor indicates the XY plane.**

FIGURE 31.22
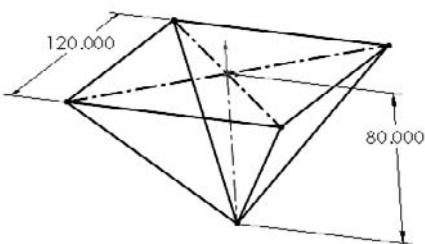
A centered rectangle in a 3D sketch

11. **Draw a line from one corner of the square down, trying to avoid any automatic relations such as coincident relations to other points and any AlongX, Y, or Z relations.** Connect the other three corners of the square with the free endpoint of the new line, as shown in Figure 31.23.

FIGURE 31.23

Adding lines
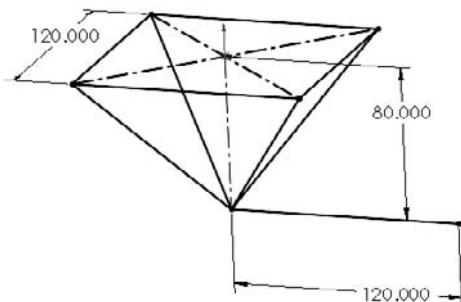
12. **Rotate the view slightly.** Notice that the first line that you drew in Step 10 and one other line are on a plane. Drag a right-to-left selection box around the point where the four lines converge, and assign an Equal relation to all the lines. This makes the shape into an upside-down pyramid shape.

13. **Drag the point.** Notice that it moves up and down, although it seems a little erratic. Place a dimension between the point and the part Origin. Notice that the sketch becomes overdefined and turns red and yellow. Theoretically, this combination should work, but for some reason SolidWorks does not accept it.

14. **Using the Display/Delete Relations tool, delete all the Equal relations that you just added to the part (it may be faster to select Undo from the File menu or press Ctrl+Z).**

15. **Draw a vertical construction line from the part Origin to the point where the four lines meet, and assign this line an AlongY relation.** Notice that the point drags much more smoothly. This is a good reason for using simpler relation schemes when possible. The four equal relations in this case that had to be solved simultaneously are now replaced by a single relation that is easier to solve when you drag the sketch. Apply a dimension of 80 inches to the new construction line.

16. **Draw a new line from the point where the four lines come together AlongX in the positive X direction.** Dimension this new line as 120 inches. The sketch should now look like Figure 31.24.

FIGURE 31.24

The sketch after Step 16
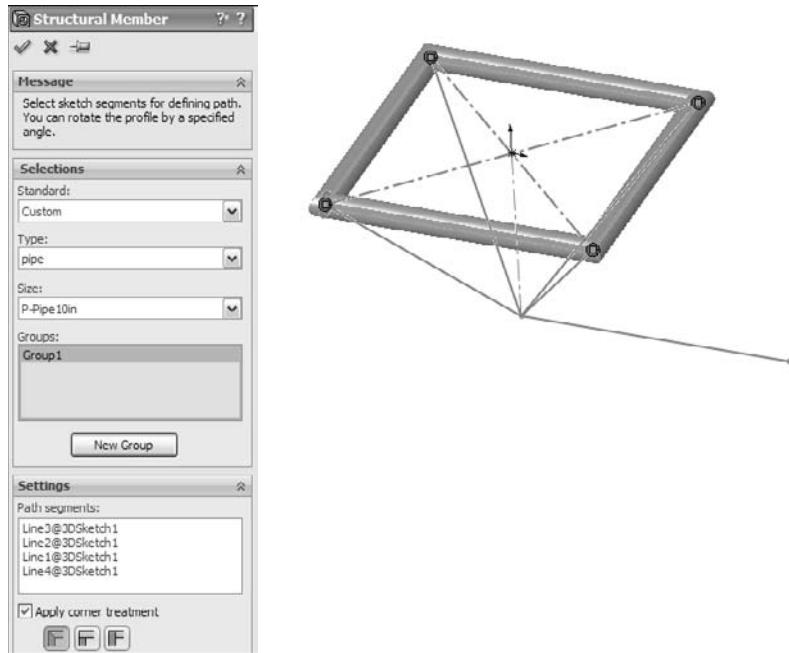
17. **Exit the sketch.** Click the Structural Member toolbar button on the Weldments toolbar. In the Standard drop-down list in the Structural Member PropertyManager, select Custom. In the Type drop-down list, select Pipe. In the Size drop-down list, select P-Pipe10in. This is the name that corresponds to the way the library feature part was saved in Step 5.

18. **In the Path Segments selection box, select the original four sides of the rectangle.** In the Settings panel, make sure that the Apply corner treatment option is turned selected and that the End Miter icon is selected. This is shown in Figure 31.25. Accept the command when you are done.

FIGURE 31.25

The Structural Member PropertyManager and the sketch after Step 18

19. **Expand the Structural Member feature.** Notice that the four bodies are listed under it. Click the Cut List folder to expand it. The bodies should also be listed there.

20. **Open the 10-inch pipe library feature that you created at the beginning of this tutorial.** Edit the two dimensions to subtract 2 inches from each dimension, and add a custom property description called Support Leg. Choose File ➪ Save As to save the library feature to the same location as the original, but with the filename `P-Pipe8in.sldlfp`.

21. **Initiate another Structural Member feature, this time selecting the 8-inch size of pipe from the Custom folder.** In the Path Segments selection box, select two of the angled lines that go to opposite corners. Keep the feature open for the next step.
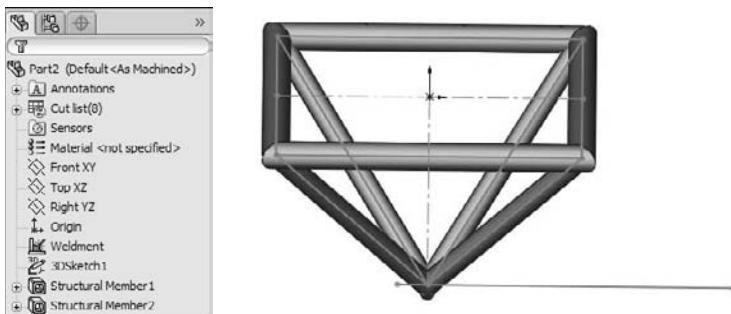
Note

Remember that three intersecting Structural Members cannot be created with a single group. To create material on all four lines, you need two separate groups within the Structural Member feature. ∎

22. **Make a second group with the other pair of angled lines.** Accept the feature when you are satisfied. The model should now look like Figure 31.26.

FIGURE 31.26

The model showing the features accepted in Step 22.

23. **Apply another Structural Member feature to the 10-foot (120-inch) section, again using the 10-inch-diameter pipe.** Notice that this member is not long enough to cut through the peak of the pyramid.

24. **Edit the 3D sketch and draw a 12-inch extension to the original line past the peak of the pyramid.** Use an additional line rather than extending the existing one. Exit the sketch.

25. **Edit the Structural Member feature to add the new line.**

Note
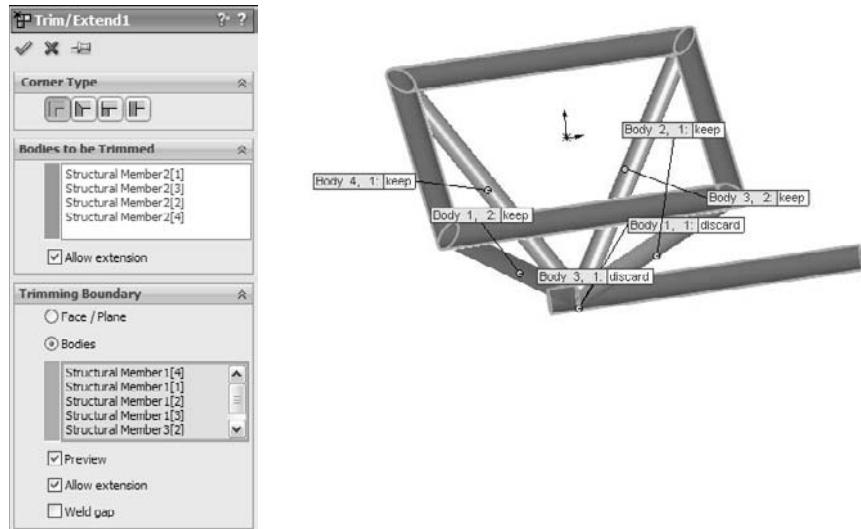
You will have to deselect the Apply corner treatment option to get this technique to work. If this option is selected, SolidWorks tries to miter or otherwise create a corner treatment between the bodies, which fails when the parts are parallel. ■

26. **The four angled members need to be trimmed on both ends because they extend to the ends of the sketch entities rather than stopping at intersecting members.** Initiate the Trim/Extend feature. Select the four angled members in the Bodies to be Trimmed selection box. Select the four members created by the original rectangle as the Trimming Boundary, and make sure that the Bodies option is selected (as opposed to Face/Planar), as shown in Figure 31.27. Accept the feature when you are done.

27. **Create another Trim/Extend feature.** This time, trim off the point end of the four angled members using the 10-inch horizontal pipe and the small segment as the trimming boundary.

28. **Half of the support structure has been modeled to this point.** You can create the rest of it by mirroring the existing bodies. Create a Mirror feature using the free end of the 10-foot-long member as the mirror plane, and selecting all the bodies in the Bodies to Mirror selection box. Do not select the Merge solids option because you will need to merge solids manually. Click OK to accept the feature when it is set up properly. The PropertyManager for the Mirror feature is shown in Figure 31.28.
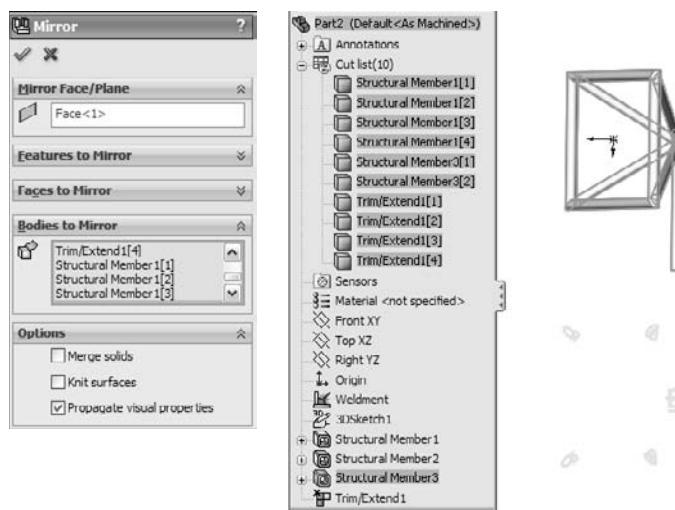
The model after Step 26

The PropertyManager for the Mirror feature in Step 28
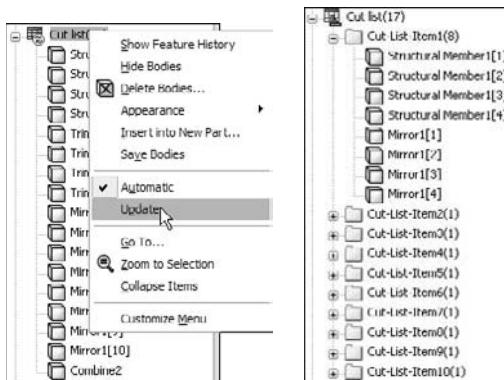
Note

An easy way to select all the bodies is to use the flyout FeatureManager, select the first body in the list, and Shift+select the last body. ∎

29. **Select the Combine feature (Insert ⇨ Feature ⇨ Combine), and set it to Add.** Select the two 10-foot sections and the two smaller 1-foot sections to combine them into a single continuous body. Click OK to accept the feature. Also hide the 3D sketch.

30. **Right-click the Cut List folder, and select Update.** Figure 31.29 shows before and after images of the Cut List folder.

FIGURE 31.29

The Cut List folder in Step 30
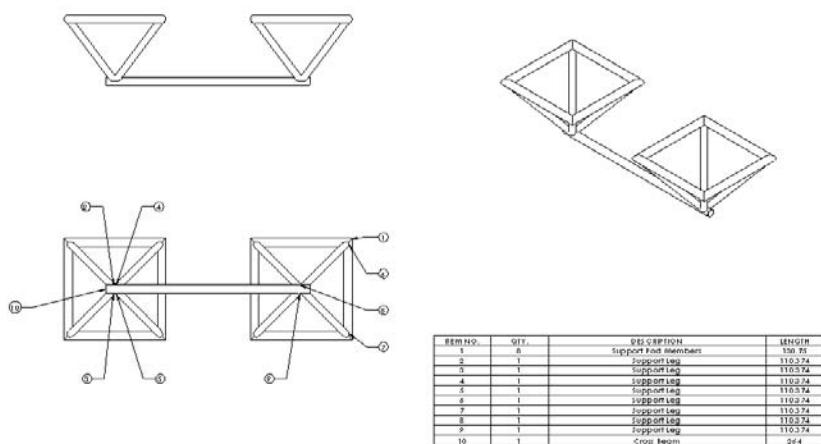
31. **Right-click the folder for the large diameter cross member and select Properties.** Change the Description field to read Support Pod Members.

32. **Use the Create Drawing From Part/Assembly button on the Standard toolbar to make a drawing.** Place Front, Bottom, and isometric views, and then press the Esc key to quit placing views.

33. **Select one of the views and then choose Insert ⇨ Table ⇨ Weldment Cut List.** When the PropertyManager displays, select the options that you want and click OK. Then place the table.

34. **Click inside the Bottom view, and from the Annotations toolbar, click Auto-Balloon.** The finished drawing looks like Figure 31.30.

FIGURE 31.30

The finished drawing

ITEM NO.	QTY.	DESCRIPTION	LENGTH
1	8	Support Foot Members	130.75
2	1	Support Leg	110.374
3	1	Support Leg	110.374
4	1	Support Leg	110.374
5	1	Support Leg	110.374
6	1	Support Leg	110.374
7	1	Support Leg	110.374
8	1	Support Leg	110.374
9	1	Support Leg	110.374
10	1	Cross Beam	264

Note

Relative views are difficult to create with round pipe rather than rectangular tube, although with version 2007 and above, planes can be used as references for relative views. ∎

Summary

Weldments are based on either a single 3D frame sketch or a set of 2D sketches, usually denoting the centerlines or edges of the various structural elements. This creates a special type of part in the same way that the Sheet Metal commands create a special type of part. Structural profiles are placed on the frame sketch to propagate and create individual bodies for the separate pieces of the weldment. Custom profiles are easily created as library features; you can add custom properties to the library features, and the custom properties then propagate to the Cut lists.

Using Plastic Features and Mold Tools

S olidWorks has several tools that are specific to modeling and evaluating plastic parts. These tools can help simplify and standardize some of the complex repetitive tasks involved with plastic part design. Another set of task-specific tools exist for automating mold cavity/core modeling tasks. So whether you are a plastic part designer or an injection mold designer, SolidWorks offers tools tailored to your needs.

You can manually do all the work that these tools automate, and sometimes you need to be able to do the work manually when the automated tools do not provide the needed options or flexibility.

Because of the specific needs of plastic part modeling and mold design, SolidWorks has a set of powerful evaluation tools that help you query your models to check the amount of draft, thickness, and location of undercuts. Finding design-related manufacturability problems in manufacturing is expensive. Finding them in the design office is far less expensive and conserves time. A good grasp of the evaluation tools is an important component of good plastic part and mold modeling practice.

This chapter is written for the user who is already experienced in plastics practice and terminology, but needs to understand the SolidWorks tools used with the plastic and mold features. In this chapter, I assume that the reader already has a grasp of basic plastics and mold design.

IN THIS CHAPTER

Exploring plastic features

Learning about plastic evaluation tools

Understanding SolidWorks' capabilities with mold geometry

Using Mold Tools manually

Working with plastic features tutorial

Using Plastic Features

The plastic features available in SolidWorks are the Mounting Boss, Snap Hook, Snap Groove, Vent, and Lip/Groove. These features offer standardized but flexible geometry to help you make more consistent models more quickly, and with less tiresome repetition. In this chapter I also consider the various functions of the Draft feature in SolidWorks. Other features such as Shell and Indent are frequently used in plastic parts, but I cover them in Chapter 7. You can find all the features in this section on the Fastening Features toolbar.

Some of these features have applications beyond just injection molded plastic parts. Many molding, casting, or "net shape" processes exist in plastic materials, as well as metals, ceramics, and composites.

Using the Mounting Boss

The Mounting Boss feature enables you to place a boss with fins and either a hole or a pin on the end. It does not enable you to place a counterbored hole or a through hole to facilitate screw bosses. It is aimed primarily at press pins.

Figure 32.1 shows the Mounting Boss PropertyManager along with the preview of the boss in progress. The part used in this figure is on the CD-ROM in the data for Chapter 32, and is called `Chapter 32 – right frame.sldprt`.

The workflow for the Mounting Boss feature is as follows:

1. **Select a spot on the part that represents where the boss will attach to the part.** This can be either a flat or curved face. In the example in Figure 32.1, because I selected a curved face, it is necessary to also supply a direction of pull. If you select a flat face, the direction selection box in the next step is not available.

2. **Select a plane, planar face, edge, or axis to establish the axis of the boss.** This is usually the direction of draw. Notice in the part shown in Figure 32.1 that an axis established early in the part is named as the Direction of Draw. In early 2009 releases, the Change Direction arrows do nothing. This step is optional. The default is the direction normal to the face selected in Step 1.

3. **Select an existing circular edge to align the boss.** The new boss will be concentric with the circular edge. This step is also optional. You can choose to use dimensions to locate the boss *after the feature is created*. If you use dimensions, you cannot locate the boss while the PropertyManager is active.

4. **The Boss panel of the Mounting Boss PropertyManager is used to establish the height and other dimensions of the central boss.** Diameter and draft dimensions are obvious. You can establish height by a dimension or by an "up to" face using the Select mating face option. There is no option for an "offset from face" end condition.

FIGURE 32.1

A Mounting Boss in progress

5. **The Fins panel of the Mounting Boss PropertyManager is used to control the alignment, draft, height, width, and patterning of the ribs around the boss.** The first box is for a direction vector such as a plane, edge, or axis to establish the rotational orientation of the ribs. You cannot use a sketch for this. The significance of the dimensional values is obvious. The fin pattern function is also obvious except for the Equally Spaced option, which is only available when the number selected is 2. Then you must select a vector to establish the orientation of the fins, which form a right angle for a corner.

6. **The Mounting Hole/Pin panel enables you to specify a pin or hole boss and the associated sizes.** It is interesting to me that it doesn't enable through holes or counterbore holes from the outside of the part, along with associated screw sizes for clearance.

7. **SolidWorks has renamed Favorites in the rest of the software to Styles, but in the Mounting Boss and the Lip/Groove, it is still called Favorites, and saves settings like other Favorites/Styles functionality.**

8. **After you have successfully accepted the creation of the boss, if you did not use a circular edge to locate the boss (Step 3), you can expand the Mounting Boss feature in the FeatureManager and edit the 3D sketch under the boss.** Inside this sketch is a point, which you can dimension to locate precisely. Remember that dimensions in 3D sketches follow some special rules. To get orthogonal dimensions parallel to X, Y, or Z axes, you will need to dimension from planes. 3D sketch dimensions do not snap to horizontal or vertical orientations like 2D sketches.

   If you have selected a flat face for the initial position of the boss, you get a 2D sketch instead of a 3D sketch.

The features created by this tool are not manually editable. They are not made of extrudes and rib features that are accessible behind the Mounting Boss interface. You have to go through the Mounting Boss PropertyManager to edit the features, and cannot edit sketches used to make the features.

SolidWorks designed this feature primarily as a way to create pin-and-hole press pin bosses. These are certainly needed, but also needed are screw bosses. You will have to size your own screw holes either for threaded inserts or for thread cutting screws. You will also need to manually create some of your own features if you want to make a counterbored clearance hole for a screw, as shown in the cross section in Figure 32.2.

An effective way to pattern a single Mounting Boss feature around a part is to use the Sketch Driven Pattern. This feature uses a sketch with a set of points where each point represents the center of a patterned instance. Refer to the model on the CD-ROM and examine the sketch driven pattern at the bottom of the FeatureManager. Use the `Chapter 32 – right frame.sldprt` file, and change to the "nonmirror" configuration if it is not already there.
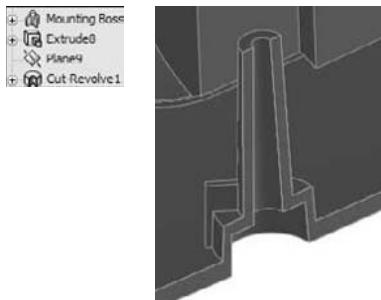
Using the Snap Hook and Snap Hook Groove

Unlike the Lip/Groove feature, which is treated next, the Snap Hook and Snap Hook Groove features are two separate features. Lip/Groove combines both functions into a single PropertyManager to help you get results that work together more easily. Figure 32.3 shows the PropertyManager for the Snap Hook feature, along with a completed hook.
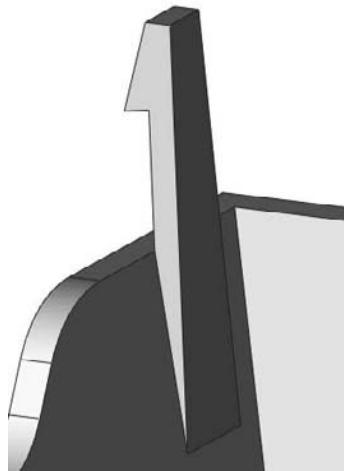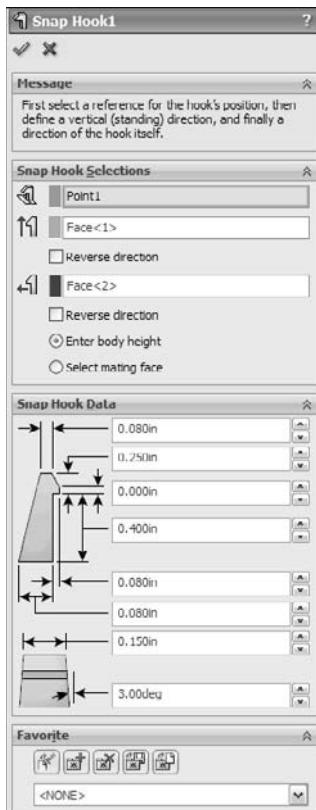
FIGURE 32.2

Manually making a Mounting Boss into a typical screw boss

FIGURE 32.3

The Snap Hook PropertyManager with a completed hook feature

The workflow for the Snap Hook feature goes like this:

1. **Select a spot on the model that will correspond to the center of the undercut edge where the hook intersects the part.** It looks like you can select a face or an edge when you first create the feature, but the software always converts the selection to a point when the feature is accepted.

2. **Select a vector (face, edge, axis, not a sketch) to set the vertical orientation of the hook, or the "top."**

3. **Select another vector to define the "front" of the hook (the undercut side).**

4. **Choose to select a mating face or enter a number to define the height of the hook.**

This feature uses a 3D sketch point where you made the selection in Step 1. You cannot dimension this point while setting up the feature, only by creating the feature and then going back and editing the 3D sketch absorbed under the feature. This is also the arrangement with the Lip/Groove. Remember that you cannot dimension 3D sketches the same way that you dimension 2D sketches. You may need to dimension to planes rather than edges or points to get the dimensions you really intend.

The Snap Groove PropertyManager interface is shown in Figure 32.4, along with a cross section of a finished Snap Hook and a Snap Hook Groove. To use the Snap Hook Groove feature, you must have already created a Snap Hook feature. The interface seems to imply that it requires the body the groove goes into to be in the same part as the body of the hook feature, but this is not the case. You can create this feature in-context between a part with a hook and the part to receive the groove.
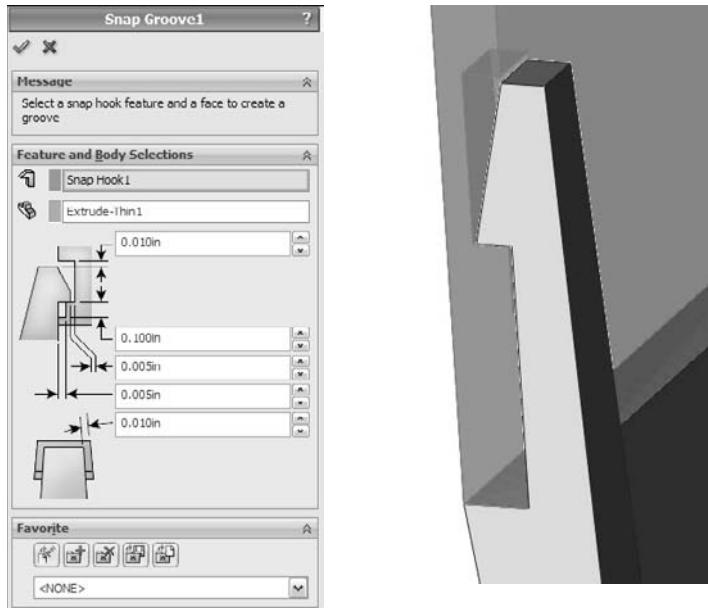
Note
Before designing extensive undercuts into a plastic part, it is advisable to talk to the mold builder if possible. They may have either limited or special capabilities that could impact on the practicality of one approach as opposed to another. I find it is frequently beneficial to work closely with a mold designer or builder on plastic part projects. ■

When I model plastic parts, rarely do situations call for a generic snap feature. Usually situations require more inventiveness due to space restrictions or curvature or material thickness considerations. The Snap Hook and Snap Hook Groove features are reasonably easy to use, but may not have the flexibility for application in all situations.

FIGURE 32.4

The Snap Hook Groove PropertyManager with a completed hook and groove

Using Lip/Groove

The Lip/Groove feature enables you to create a matching lip and groove in either a pair of parts in an assembly or a pair of bodies within a single part. Figure 32.5 shows the Lip/Groove PropertyManager creating a groove in a part. The same interface also creates the lip feature.

The workflow for this feature goes like this:
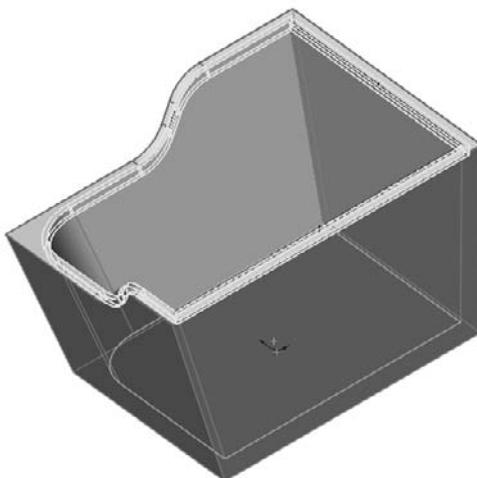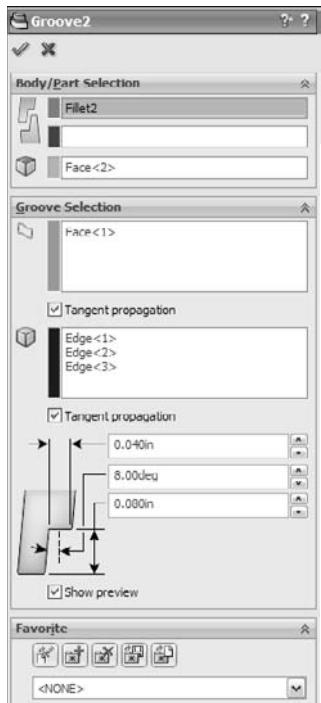
1. **Select the part or body to receive the groove.**
2. **Select the part or body to receive the lip.**
3. **Select a plane, planar face, straight edge, or axis to establish the direction of pull.**
4. **Select faces that represent the parting surface along the area to get the lip/groove.**
5. **Select edges that the lip/groove will affect.**
6. **Set dimensions for both lip and groove features.**

When the Lip/Groove feature works correctly, it saves a lot of time. The work-around techniques are time-consuming and potentially error prone. SolidWorks has put some additional development time into Lip/Groove, so if you have used it before and found it lacking, you might try it again. The 2010 version removes some of the limitations, particularly those involving ribs intersecting the main wall and multiple sections of wall needing lip or groove features.

FIGURE 32.5

Using the Lip/Groove feature

In some cases, the automated tools may not be able to create what you need to create. You can employ one of several manual workarounds to make lips and grooves. Several techniques exist.

- On a planar parting line, you can use sketches offset from the edges and then extrude either a boss or a cut.

- Using a thin feature extrude (boss or cut) can also be effective on planar parting lines.

- Using trimmed and thickened (again, boss or cut) surfaces can be effective but may also be more difficult.

- One of my favorite methods, especially for non-planar parting lines, is to combine a thin feature with an extrude up to an offset surface body.

- Using a sweep to cut or add material can be effective on either planar or non-planar parting lines.

Each of these is really a work-around technique and not a specific plastic modeling tool, I will not go into depth on these techniques here. On the CD-ROM, you will find example parts that demonstrate each technique.

Using the Rib feature

The Rib feature is a flexible tool for creating ribs in a number of different situations. Ribs can be drawn in one of two different orientations, which the SolidWorks interface calls Parallel To Sketch and Normal To Sketch. The names appear only on tool tips when hovering over the icons. To be more precise, what they really mean is that the rib will be created either parallel or normal to the sketch *plane*. If the sketch is a single line, it can be very difficult to tell the difference between parallel and normal.

To me these names are not very descriptive. I call the two orientations *plan view* (view from the top, looking in the direction of draw, normal to the sketch plane) and *skyline* (looking from the side, perpendicular to the direction of draw, rib is parallel to the sketch plane). To me, these names are more intuitively descriptive.

Ribs can incorporate draft, extend, or trim the feature beyond the sketch automatically, and break normal sketch rules (plan view ribs only).

Ribs do have some limitations or opportunities for improvement, which I will discuss briefly later in this section. It is as important to understand what features *can't* do as it is to understand what they *can* do.

Figure 32.6 shows a plan view rib that violates normal sketch rules. Also shown is the Rib PropertyManager. Several models on the CD-ROM show examples of various rib techniques.

The Rib feature workflow should be self-explanatory:

1. **Draw the sketch, either the plan view or skyline.** The sketch represents the top of the Rib. Material can only be added between the sketch and the rest of the part.

2. **Initiate the feature, and set the type of rib and the direction.** Use the Flip Material Side toggle to change the direction of the gray arrow, which should point from the top of the rib toward the part.

3. **Set the thickness and draft amount and direction.**

Using draft in the Rib feature
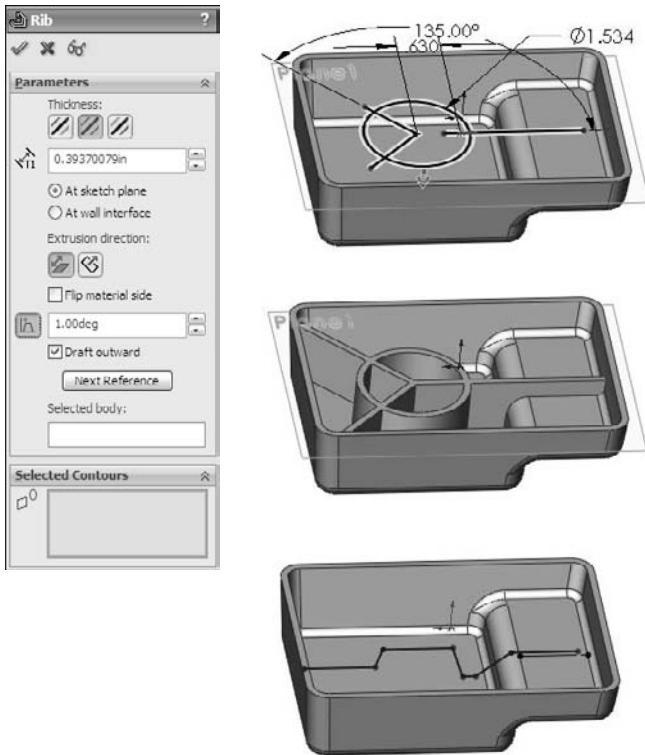
When you create ribs, you almost always apply draft to them. You can apply draft as a separate feature, or you can apply draft as a part of the Rib feature. It is often easier to just do it as part of the Rib feature, but some people like to make all their drafts as separate Draft features to keep the part faces orthogonal for as long as possible, or they just like to organize all the Draft features into a folder at the end of the FeatureManager so that there is never any question about which feature controls the draft.

By default, when you apply draft as a part of the Rib feature, the draft is applied from the sketch end of the rib. This can cause rib thickness problems if you have created a skyline rib where the rib may have various heights. In this case, the top of the rib will vary in thickness, and it may cause the base of the rib to be too thick. When you work this way, sometimes you have to experiment with the proper thickness at the top of the rib in order to get a thickness at the bottom of the rib that will not cause sink marks on the outside face of the part.

The Rib PropertyManager and a Rib feature

A solution to this is to control the thickness of the rib at the base, and then draft away from that. This process is more in line with standard plastics processes anyway, which generally give dimensions at maximum material condition, and then the draft always removes material. Thinking of draft as always removing material helps plastic part modelers and mold designers to remain "steel safe." It is always easier to remove steel from a mold (which adds material to the plastic) than it is to add steel to the mold.

Starting in the SolidWorks 2009 release, you can now specify the thickness and draft such that the base of the rib maintains a specified thickness, and the draft is applied from the base of the rib. The option to do this does not appear until you turn on the draft for the feature.

Ribs and multi-bodies

The Rib feature is sensitive to changes in the number of solid bodies in the part. For example, if you build a part and put a Rib feature in it and it only has one body, then roll back before the rib and do something that makes two bodies when you unroll the FeatureManager, the Rib feature will

fail. The way to fix this error is to edit the rib and use the Feature Scope at the bottom of the PropertyManager to select which body the rib is meant to intersect. The Feature Scope box does not appear unless the part has multiple bodies.

Using Intersection Curves as reference
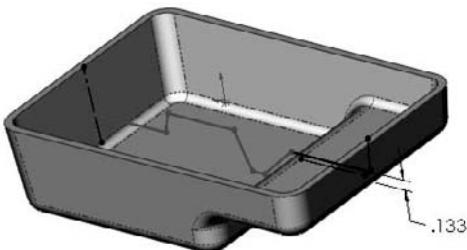
Ribs are features that typically go inside hollowed out parts. For that reason, ribs are often difficult to visualize, especially when they are on a plane that is deep down inside a part. You may find it useful to use some sort of a reference that shows where the current sketch plane intersects the wall of the part. For this I typically use Intersection Curves.

The Intersection Curve is on the Sketch toolbar. While in a 2D sketch, activate the tool, and then select faces that intersect the current sketch plane. Deactivate the tool when you are done. You may want to select all the lines selected by the Intersection Curve tool and turn them into construction geometry. This provides a good reference for the rib sketch.

Figure 32.7 shows an example of using an Intersection Curve as a reference for setting up a rib sketch. The construction lines at the ends and below the right end of the skyline rib sketch are Intersection Curves.

FIGURE 32.7

Using Intersection Curves as references

At the ends of the shell, the intersection curves serve to give the sketch a reference point to be fully defined. Under the right end of the rib sketch, the intersection curve gives you a reference to dimension the height of the rib in the shallower section of the part. The part shown in Figure 32.7 is on the CD-ROM as `Chapter 32 – skyline.sldprt`.

Terminating ribs

The Rib feature automatically extends and trims ribs based on your rib sketch. This is a great ease-of-use function, but it tends to lead to sloppy sketching for Rib features. If you sketch a rib, and the sketch does not lead all the way to the wall of the part, SolidWorks extends it. If your sketch line goes past the wall, SolidWorks trims the rib so that it only goes up to the wall of the part.
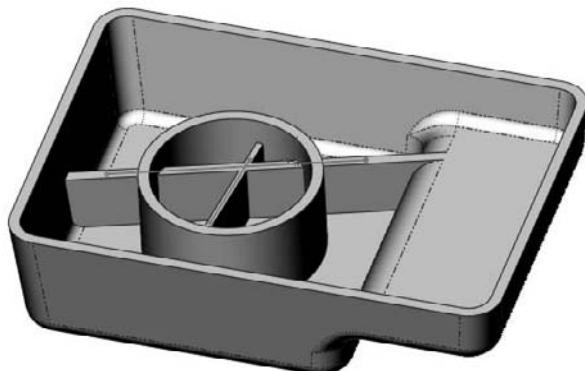
Figure 32.8 shows how the two straight ribs are extended from the existing sketches on a pair of plan view ribs.

FIGURE 32.8

Extending ribs

Sometimes you do not want a rib extended to the wall of the part. You may want to terminate a rib at a specific location in the middle of the part. The way to do this is to use a skyline rib and end the skyline sketch with a vertical (plus or minus draft) line that points to the base of the part. Figure 32.9 shows how to accomplish this. Notice that on the left end of the rib, it is extended straight down to the bottom of the part, and on the right side of the rib, it is extended up to the next wall.

FIGURE 32.9
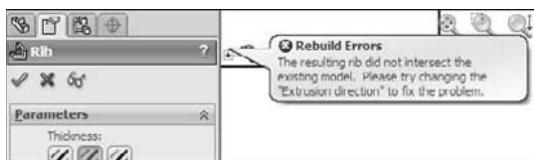
Terminating a skyline rib

A final termination situation I want to mention is one that can sometimes happen at curved edges. If the extension of the rib cannot be contained by the model, the rib will fail. This is not always as obvious a situation as you might think. When a non-horizontal rib intersects a curved edge, it usually forces you to fake something a little. Figure 32.10 shows an example of why this is.

FIGURE 32.10

The part wall does not terminate the rib.

The reason for the error shown in Figure 32.10 is that even though the rib sketch intersects the edge of the part, the width of the top of the rib would go past the edge and not intersect anything. One way to deal with this is to make the sketch intersect the part a little closer to the center of the part from the edge. Another way would be to put a short vertical line at the end of the rib.

Using thin features

Thin feature extrusions are sometimes used in place of ribs. Thin features do not have all the specialized options available with the Rib feature, but they do offer simplicity as the main attraction. Thin features can substitute for Rib features when the rib is a stand-alone rib that doesn't touch the side walls of the part. They can be used to sketch and extrude from the bottom of the rib or from the top. When extruding a thin feature with draft, the end faces get drafted as well, which might cause a problem if you are trying to attach the rib to a wall. Extruding a thin feature down from the top of the rib can replace a plan view rib, but it will not enable you to break those pesky sketch rules that the Rib feature simply dispenses with.

I personally prefer to use Rib features, except for free standing ribs where you do not want the sides extended up to the next wall.

Using Draft

SolidWorks Draft is surprisingly powerful for as simple as it is. All the aspects of working with Draft could take up an entire chapter all on its own. I will try to hit the most important points in this brief synopsis. The Draft feature creates three types of draft:

- **Neutral Plane draft.** Drafts faces from a plane or planar face.
- **Parting Line draft.** Drafts faces from selected edges.
- **Step draft.** Drafts faces from parting line edges, and can create a step at the parting line. (similar to pivot draft in other software)

Figure 32.11 shows the PropertyManager of the draft feature, including the DraftXpert, which is mentioned later in this section.

FIGURE 32.11

Draft and DraftXpert PropertyManagers

Neutral Plane draft

The workflow for Neutral Plane draft is as follows:

1. **Select Neutral Plane — plane or planar face at which the intersection of the drafted face does not move.** The direction of pull is always normal to the Neutral Plane.

2. **Set the draft angle — how many degrees from the direction of pull vector selected faces should be tilted.** This is *not* a cumulative angle, so applying 3 degrees of draft to a face that is already drafted 5 degrees results in 3 degrees rather than 8 degrees.

3. **Select the Direction — one side or the other side of the Neutral Plane.** The arrow points in the direction of decreasing material. If you are drafting a surface body, the "decreasing material" concept does not apply, so you just have to experiment to see which direction is the one you intend. The Draft feature does not have a preview option.

4. **Select faces to draft — use face propagation options.** Inner/Outer faces refer to inside or outside loops around the Neutral Plane face. All means all faces that have an edge on the Neutral Plane face.

Parting Line draft

The workflow for Parting Line draft is as follows:

1. **Select a direction of pull — this can be an edge, axis, sketch line, plane, or planar face.** Set the direction to positive or negative along the selected direction.

2. **Select the parting lines — these are edges of the faces you want to draft.** The parting line edges remain stationary while the rest of the face tilts. Along with the parting line selection, you may also need to use the Other Face option. Every edge is adjacent to two different faces. The Draft feature automatically selects the face it thinks you want to draft, but it does not always get it right. Other Face enables you to intervene when the automatic selection is incorrect.

3. **Set the draft angle — remember that you can use the Allow Reduced Angle option if you need to.**

Step draft

The most complex of the types of draft that SolidWorks creates is the Step draft. Step draft is used on non-planar parting lines when Parting Line draft would cause the drafted faces to be split into multiple faces.

The word "step" can be said to refer to two different aspects of this feature. First, the parting line can said to be a "stepped" parting line because it is non-planar and at two different levels. Second, the draft actually steps out the drafted face at one level of the parting line. Step draft keeps the face intact and introduces an intentional mismatch ledge (step) at the parting line.
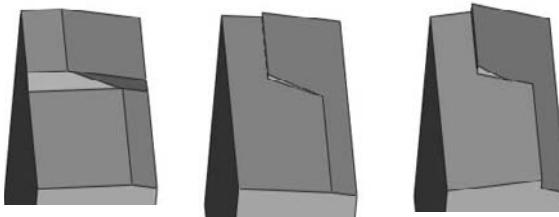
Figure 32.12 shows the difference between Parting Line draft and Step draft on a simplified part. The image on the left is the Parting Line draft. The middle image is Step draft where a ledge is only created on one side of the parting line. The image to the right is essentially double Step draft, where the total step size is minimized by distributing it across both sides of the stepped parting line.

FIGURE 32.12

Comparing Parting Line draft to Step draft

The draft in these images is slightly exaggerated to make it easier to see.

The workflow for Step draft is as follows:

1. **Start with a part that has a stepped split line, where the angled line makes angles of more than 90 degrees rather than less than 90 degrees.**

2. **Create a plane that defines the direction of pull and is located at the level of one of the split lines (for single step draft) or at the midpoint of the angled line (for double step draft).** The location of the plane will determine the "pivot" point for the drafted faces. Essentially the line of intersection between this plane and the drafted face will remain stationary and the rest of face will pivot around it.

3. **Initiate the Draft feature, and set the option to Step draft.** Consult your tooling people about whether to use tapered or perpendicular steps. Perpendicular are probably easiest to tool.

4. **Select the edges of the parting line.** Make sure that the yellow arrows indicate which faces you want to apply draft to.

The draft for the other face can be accomplished in a number of ways, but it does not require another Step draft feature. You can use a Neutral Plane feature, using the plane created in Step 2 as the Neutral Plane. It will maintain the steps created by the Step draft feature.

Some draft limitations

SolidWorks draft is powerful, but it is not omnipotent. There are things it cannot do. The main limitation of draft in SolidWorks that irritates me is that it cannot draft faces in both directions in the same feature. For example, if you have a parting line on a part, you must first draft the faces on one side of the parting line, then the faces on the other side of the parting line. This results in two separate features rather than a single feature. Is this a serious limitation? No, it is simply an inconvenience.

Further, you can only draft faces from a single body at a time. Again, this is an understandable limitation, but there are times when it is annoying and inconvenient.

However, its biggest limitation is that you can't draft a face if it has a fillet on one of its edges that runs perpendicular to the direction of pull. To get around this, you usually have to tinker with the feature order. On imported parts you might have to use FeatureWorks to remove the fillet or Delete Face to re-introduce the sharp corner.

What to do when draft fails

Draft is certainly one of those functions that require you to understand a little bit about the actual capabilities of CAD. Part of the key to success with the Draft feature is that you have your expectations aligned with the actual capabilities of the software. If you recognize a situation where the draft can not work, you may be able to correct the situation by changing feature order, combining draft features into a single feature, breaking the draft into multiple features, or changing the geometry to be more "draft friendly."

Sometimes the Allow Reduced Angle option can be used for Parting Line draft. If you use this, follow it up with a draft analysis to make sure that you have sufficient draft in all areas of the model. This option enables the software to cheat somewhat in order to make the draft feature work. The SolidWorks Help documentation actually has a more detailed explanation of when to use this option. I tend to just select it if a draft fails, particularly if the parting line used becomes parallel or nearly parallel to the direction of pull.

Draft can fail for a number of reasons, including tangent faces, small sliver faces, complex adjacent faces that cannot be extended, or faces with geometry errors. When modeling, it is best to minimize the number of breaks between faces. This is especially true if the faces will be drafted later. Generally, the faces you apply draft to are either flat faces or faces with single direction curvature. You can't just expect SolidWorks to draft any old junk you throw at it; you have to at least give it a fighting chance by making good clean geometry.

When draft does fail for a reason that doesn't seem obvious to you, you should use the Check utility under the Tools menu and also try a forced rebuild (Ctrl+Q) with Verification on Rebuild turned on.

DraftXpert

DraftXpert is a tool used to create multiple Neutral Plane draft features quickly. You can also use it to edit multiple drafted faces without regard for which features go to which faces.

Using Plastic Evaluation Tools

The plastic evaluation tools in SolidWorks enable you to automatically check the model for manufacturability issues such as draft, undercuts, thickness, and curvature. The tools used to do this are the Draft Analysis, Thickness Analysis, Undercut Checker, and Curvature tools.

Draft Analysis

The SolidWorks Draft Analysis tool is a must when you are working with plastic parts. The part shown in Figure 32.13 has many of the situations that you are going to encounter in analyzing plastic parts. The Draft Analysis tool has four major modes of display:

- Basic
- Gradual Transition
- Face Classification
- Find Steep Faces
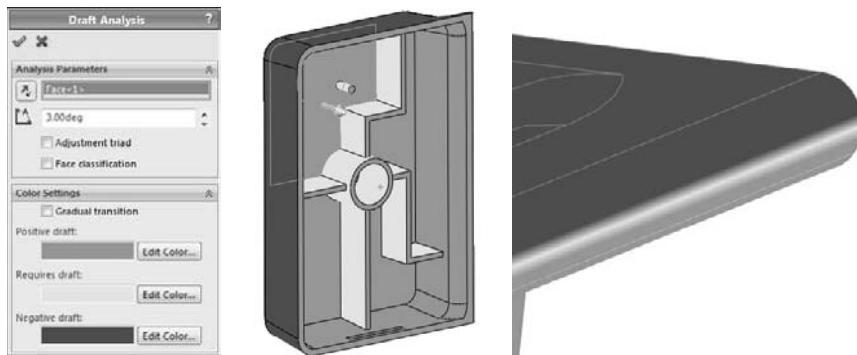
Draft Analysis is found in the View➪Display menu (instead of in the Tools menu), and is either selected or unselected, like the Section View tool. This is a benefit because it updates face colors dynamically as you model. It also has some drawbacks. The display method for the tool leaves the colors looking very flat, without highlights on curved faces, which makes parts — especially curved parts — very difficult to visualize.

Basic

The Basic draft analysis (with no options selected) simply colors faces red, green, or yellow. Colors may display transitioning if the draft shifts between two classifications. This transition type is shown in Figure 32.13 in the image to the right. For a clearer view of this method, look at the `Chapter 32 Draft Analysis.sldprt` part on the CD-ROM.

FIGURE 32.13

Basic draft analysis results

You can perform all types of draft analysis in SolidWorks by selecting a reference flat face or plane, and setting a minimum allowable angle. In Figure 32.13, all walls have at least a one-degree draft, except for the rounded edge shown in the image to the right and the dome. Both of these shapes transition from an angle less than one degree to an angle greater than one degree.
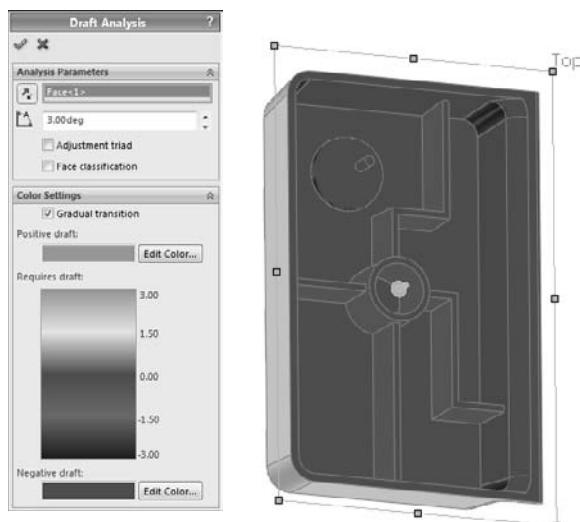
This basic analysis is good for visualizing changes in draft angle, but it also has some less desirable properties, which will become apparent as you study the other types of draft.

Gradual Transition

Although the Basic draft analysis is able to show a transitioning draft, the Gradual Transition draft analysis takes it a step further. With the Gradual Transition, you can specify the colors. It is also useful because it can distinguish drafts of different amounts by color. It may be difficult to tell in the grayscale image in Figure 32.14, but the ribs, which were created at one degree, have a slightly different color than the floor of the part, and the walls also have a different color. Notice that cavity and core directions have different colors, as well. You may want to open this part in SolidWorks, re-create the settings, and run the analysis so that you can see the actual colors.

FIGURE 32.14

The Gradual Transition draft analysis

Some problems arise when you use this display mode, the first being the flat, non-OpenGL face shading that is used to achieve the transitioning colors. This often makes it difficult to distinguish curved faces, and faces that face different directions. The second problem is that you cannot tell that the boss on top of the dome has absolutely no draft. In fact, there is no way to distinguish between faces that lean slightly toward the cavity and faces that lean slightly toward the core. The third problem is the strange effect that appears on the filleted corners. The corners were filleted after you applied the draft and before the shell, and so the filleted corners should have exactly the same draft as the sides; however, from the color plot, it looks to be a few degrees more.

Caution

Software can sometimes interpret things differently from the way that a person does. As a result, any computer analysis must be interpreted with common sense.

Due to this and some of the other problems that I mentioned earlier, I recommend using the Gradual Transition draft analysis in conjunction with one of the other tests. Gradual Transition gives an interesting effect, but it is not a reliable tool for determining on its own whether or not a part can be manufactured. ■

Face Classification
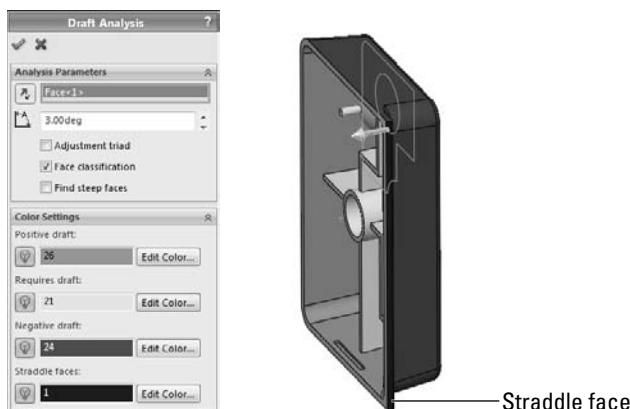
Face Classification draft analysis groups the faces into classifications using solid, non-transitioning colors. You will notice a big difference between the coloration of the Face Classification draft analysis faces and of the Basic or Gradual Transition faces. Face Classification uses OpenGL face shading, which is the same as that used by SolidWorks by default. This allows for better shading and differentiation between faces that face different directions. The Basic Analysis coloration looks like all the faces are painted the same flat hue, regardless of which direction they are facing, which makes shapes more difficult to identify. The non-Open GL alternate shading method makes it possible to display a transition in color. SolidWorks OpenGL shading cannot do this.

Another advantage of using the OpenGL shading is that the face colors can remain on the part after you have closed the Draft Analysis PropertyManager.

Face Classification draft analysis also adds a classification that is not used by the Basic draft analysis. *Straddle faces* refer to faces that straddle the parting line, or faces that, due to their curvature, pull from both halves of the mold. These are faces that need to be split. On this part, a straddle face is shown in Figure 32.15.

FIGURE 32.15

Face Classification draft analysis and a straddle face

Straddle face

The light bulb icons to the left of the color swatches enable you to hide faces by classification. This is useful when you are trying to isolate certain faces, or visualize a group of faces in a certain way. This can be an extremely useful feature, especially when you have a very complex part with a large number of faces, some of which may be small and easily lost in the mix with other larger faces.

The face counts that appear in the color swatches are a very helpful feature that is absent from the Basic draft analysis.

Best Practice
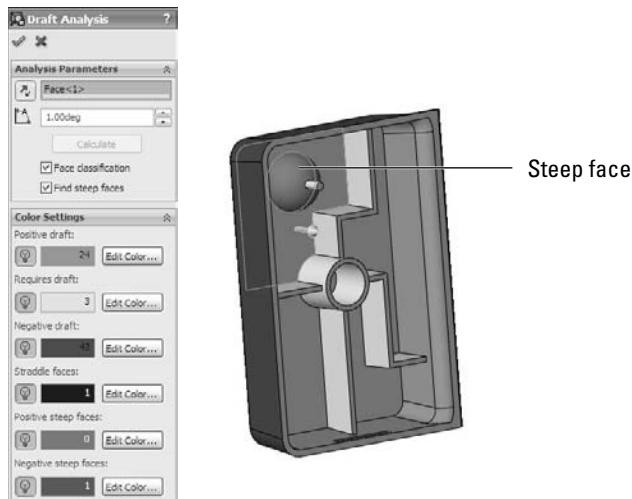
I prefer Face Classification draft analysis because it is the clearest. If I need additional detail regarding other types of faces, then I may run a Steep Face draft analysis as a supplement. The best practice here is not that you follow my favorite type of draft analysis, but that you understand what you need to know and then use the appropriate tools to find this information. This may include running multiple analyses to collect all the necessary information. ■

Find steep faces

A *steep* face is defined as a face that transitions from less than the minimum angle to more than the minimum angle. Steep faces are different from straddle faces in that straddle faces are actually positive *and* negative, while steep faces are either entirely positive or entirely negative. On this part, the dome inside the part is classified as a steep face, as shown in Figure 32.16.

FIGURE 32.16

A steep face

Steep face

Thickness Analysis

Thickness Analysis is part of the SolidWorks Utilities, which are part of SolidWorks Office or higher. After you have activated the Utilities add-in (Tools ➪ Add-ins ➪ SolidWorks Utilities), Thickness Analysis appears under the Tools menu.

Note

The tools from the Utilities add-in are always listed in the Tools menu, whether or not the add-in is loaded. When any of the tools is selected, the add-in is automatically loaded. SolidWorks has also done away with the Utilities menu, so if you are accustomed to using this functionality from previous versions, some new functionality awaits you.

You can run Thickness Analysis in two modes: Show Thin Regions and Show Thick Regions. Of these, Show Thick Regions is the most versatile.
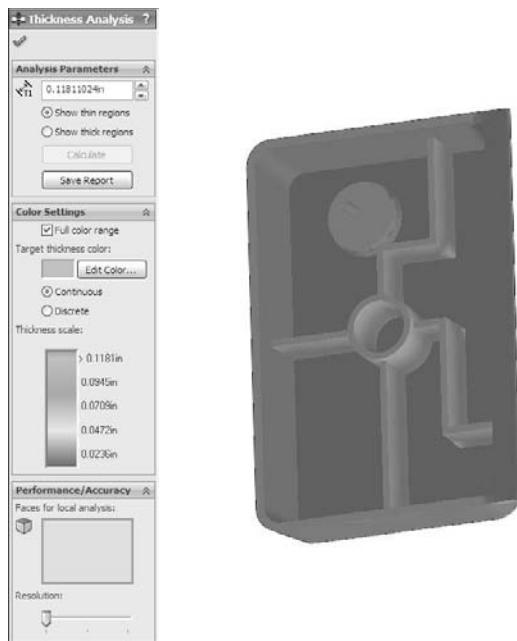
Show Thin Regions

The Show Thin Regions option, or the "Thinness" Analysis, requires you to input a minimum acceptable thickness. Every face with a thickness above this value is turned a neutral gray, and every face with a thickness below this value is displayed on a graduated scale.

Figure 32.17 shows the PropertyManager for this analysis and its result on the same part used for the draft work in the previous sections.

FIGURE 32.17

Results of the Thinness Analysis

One of the things to watch out for here is that some anomalies occur when you apply this analysis to filleted faces. The faces shown as colored were created by the Shell feature and should be exactly .100 inches thick. However, it does correctly represent the undercut on the end of the part and the thickness of the ribs. A nice addition to this tool would be the identification of minimum thickness faces. Perhaps you can submit an enhancement request.

Show Thick Regions

The Show Thick Regions option works a little differently from Show Thin Regions. You need to specify an upper thickness limit value, beyond which everything is identified as too thick. In these examples, the nominal wall thickness of the part is shown as .100 inches, and the thick region limit is set to .120 inches. For this type of analysis, the color gradient represents the thicknesses between .100 inches and .120 inches, while in the Thinness Analysis, the color gradient represents the values between .100 inches and 0 inches.

The analysis can produce some anomalous results, especially at the corners, and also in the middle. Again, this is a useful tool, if not completely accurate. You can use it to find problem areas that you may not have considered, but you should certainly examine the results critically.

The Treat Corners As Zero Thickness option should always be selected. I have never seen a situation where selecting it improved the results; in fact, I have found that deselecting it has always made corners and fillets behave worse.

This feature can generate a report, which to some extent answers questions about how or why it classifies faces in the way it does. To get a complete picture of the situation, it may be useful to look at the report when you are using the results to make design or manufacturing decisions. A sample of the report is shown in Figure 32.18.

FIGURE 32.18

A sample of a Thickness Analysis report

Summary

Total surface area analyzed	66.739in^2
Critical surface area(% of analyzed area)	1.533in^2 (2.30%)
Maximum deviation from target thickness	0.064in
Average weighted thickness on critical area	0.145in
Average weighted thickness on analyzed area	0.098in
Number of critical faces	6 Face(s)
Number of critical features	5
Minimum thickness on analyzed area	0.001in
Maximum thickness on analyzed area	0.182in

Analysis Details

Thickness range	Number of faces	Surface area	% of analyzed area
0.118in to 0.134in	4	0.57in^2	0.85%
0.134in to 0.15in	0	0.329in^2	0.49%
0.15in to 0.166in	0	0.359in^2	0.54%
0.166in to 0.182in	2	0.275in^2	0.41%

Undercut Detection

The Undercut Detection tool is in the View menu (relocated from the Tools menu) or on the Evaluate tab of the CommandManager. It is also an on or off display tool, which changes dynamically as you change the model. Undercut Detection is conceptually flawed in that it gives incorrect results every time. However, if you think of the labels as being changed slightly, the results become partially usable.

Even if you and your mold builder know that a part has absolutely *no* undercuts, the Undercut Detection tool will nonetheless always identify *all* the faces to be undercut. In fact, the only faces that this tool will identify as *not* undercut are faces that have no draft on them. The only time it correctly identifies an undercut is when it classifies the undercut as Occluded Undercut. Faces that have no draft and are occluded undercut are improperly identified as simply No Undercut.
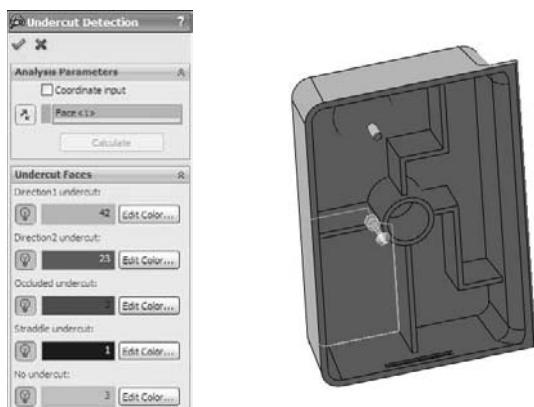
You may want to avoid this tool because too much interpretation of incorrect results is necessary; however, if you still want to use it, here is a translation guide that may help:

- **Direction 1 Undercut.** Should read Pull from Direction 2
- **Direction 2 Undercut.** Should read Pull from Direction 1
- **Straddle Undercut.** Should read Straddle faces
- **No Undercut.** Should read No draft in the primary draft direction, but may be occluded undercut faces
- **Occluded Undercut.** Should read Occluded Undercut faces that have draft in the completely irrelevant primary draft directions; does not include occluded undercut faces that have no draft in the primary direction

Figure 32.19 shows the PropertyManager for this function and the results. If you would like to test it for yourself, the part is on the CD-ROM with the filename `Chapter 32 Draft Analysis. sldprt`.

FIGURE 32.19

The results of the Undercut Detection tool

Working with the Mold Tools Process

The SolidWorks Mold Tools are intended to help you create cavity and core blocks for injection molds. They do not provide libraries or functionality for building the entire mold or mold components. Mold Tools entail a semiautomatic process to follow, with the tools in order on the toolbar. Mold Tools rely heavily on surfacing, and require a fair amount of manual intervention for certain types of parts. The next section deals with the manual intervention techniques. This section deals with the idealized semiautomatic process.

In order to fully understand the formalized Mold Tools process, it might be helpful to understand SolidWorks' capabilities with mold geometry in general. First, understand that to create cavity and core geometry in SolidWorks, you are not required to use the Mold Tools. You can manually model surfaces or solid features to accomplish the same tasks. Surface features are widely used for mold modeling because they allow you far more control than solid features.

You can also make mold geometry using an assembly of in-context parts or multi-body techniques. The formal Mold Tools uses the multi-body approach. This has benefits and drawbacks.

With the formal SolidWorks process, you start in part file with just the final plastic part in it, and then build both the cavity and core blocks around the plastic part. You also build any side actions or core pins within the part file.

Figure 32.20 shows the part of the Mold Tools toolbar that identifies the process. From the left to the right, the icons are:

- Split Line
- Draft
- Move Face
- Scale
- Insert Mold Folders
- Parting Lines
- Shut-off surfaces
- Parting Surfaces
- Tooling Split
- Core

FIGURE 32.20

The Mold Tools

Mold Tools are really meant for tooling engineers, but part designers often use the first part of the process to apply draft to parts. Tooling engineers often need to add or correct draft to plastic parts they receive from part designers without draft or that are not designed with any process in mind whatsoever.

Cross-Reference

The Split Line feature was covered in Chapter 7, and is not covered again here. Draft was covered earlier in this chapter. ■

The general workflow for using Mold Tools to create cavity and core blocks for an injection mold is as follows:

1. **Create split lines to add draft where needed.**

2. **Create draft as needed (Move Face can be used to angle faces much like the Draft feature).**

3. **Scale the part up to compensate for shrinkage during molding.**

4. **Identify the parting lines that separate cavity faces from core faces.**

5. **Create Shut-off faces, which are surfaces that close any through holes (windows or pass-throughs) in the part and represent places where the steel from the cavity side of the mold directly touches steel from the core side of the mold.** These openings in the part are capped by surface features.

6. **Create Parting surfaces.** These are the faces outside the part where the steel from opposite sides of the mold touch.

7. **Create the Tooling Split**. Tooling Split uses the faces of the Shut-offs and Parting Surfaces, and the faces of either the Cavity or the Core side to split a block into two sides.

8. **Create any Core features.** Core is an unfortunately named feature in SolidWorks. Even in mold lingo, the word has several meanings, and it doesn't become any clearer when translated into SolidWorks terminology. In this case, the word "core" refers to the material used to make core pins, side action, slide, lifter, or pull in a mold.

If you were to create a mold with manual modeling functions, you might go through roughly the same steps in the same order. The SolidWorks process often breaks down in the automated surface modeling areas, such as shut-offs and parting surfaces. You may need to manually intervene in the process for these steps. Fortunately, the SolidWorks process is flexible enough to allow for manual modeling as needed.

Each one of these process steps may have several steps of their own. Cavity and core creation is far from a push-button operation, but when you understand the overall process, the detailed steps become clearer.

Using the Scale feature
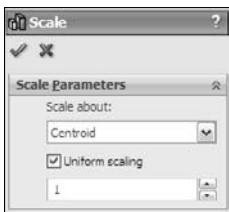
The Scale feature is used to make the plastic part slightly larger to compensate for plastic shrinkage during molding. Scale is driven by a multiplier value, so a part that is twice as big gets a scale factor of 2, and one half as big gets a scale factor of .5. Plastic materials have a shrink rate that is usually measured in thousandths of an inch per inch of part. Five thousands inch per inch is equal to a 0.5 percent rate. If the part is four inches long, the mold cavity to produce it must be 4.020 inches with that material. The 0.5 percent rate is equal to a scale factor of 1.005.

Some materials have *anisotropic* shrink rates, meaning they shrink different amounts in different directions. SolidWorks has a means to compensate for this, although it may not always be practical. Usually the shrink directions are identified as "in the direction of flow" and "across the direction of flow," and the direction of flow of molten plastic inside a mold cavity is not always a straight line. Any anisotropic shrink applied to a part in SolidWorks is an approximation at best. If you deselect the Uniform scaling option in the Scale feature, SolidWorks enables you to set different scale factors for X, Y, and Z directions. The Scale PropertyManager is shown in Figure 32.21.

FIGURE 32.21

The Scale PropertyManager

Insert Mold Folders

Mold Folders are folders that the Mold Tools add underneath the Surface body folders. You can add these folders manually using the Insert Mold Folders button on the Mold Tools toolbar. They are used to organize the different groups of faces used in separating the cavity and core solid bodies. The folders that are added are

- Cavity surface folder
- Core surface folder
- Parting surface folder
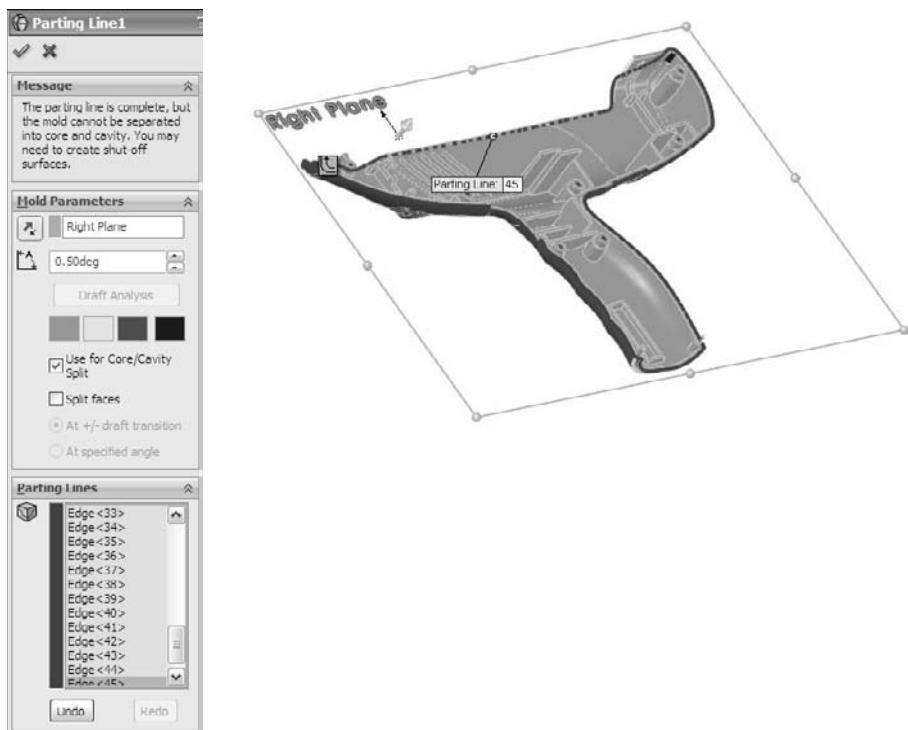
Parting Lines

The Parting Lines feature identifies (automatically or manually) the edges that separate the cavity faces from the core faces. Figure 32.22 shows the PropertyManager as well as the preview for this feature. The edge selections for this feature were mostly manual. SolidWorks intends for you to use the red arrow shown after you select an edge to propagate the selection around the part by pressing Y for yes if the red arrow indicates the correct next edge of the Parting Line or N for no if it does not.

The Parting Lines interface

In the example shown here SolidWorks gives me a message that says that the parting line is a complete loop around the part, but the part has some through holes, so it requires shut-off surfaces to close the holes.

The Parting Lines feature can also split faces if need be. You might need to split a face that straddles the parting line. For example, a filleted face might bridge across the parting line and need to be split.

Shut-off Surfaces
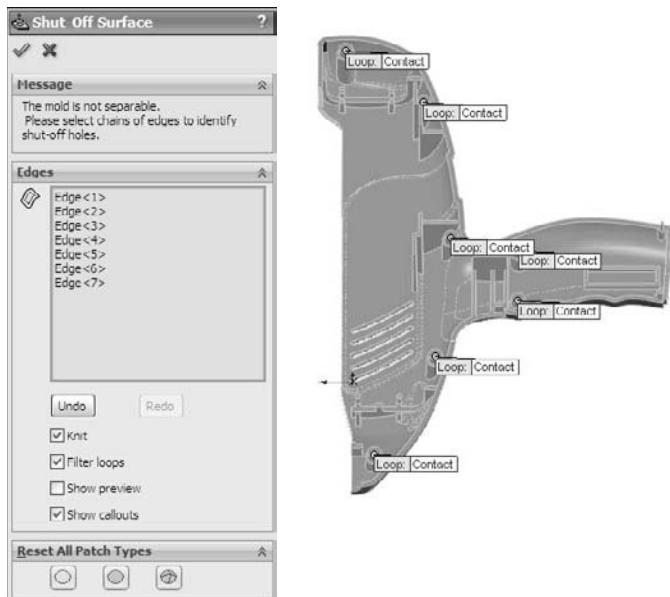
The screw holes that go through this housing require shut-off faces in order to create the mold cavity and core. You can't just seal off one end of the holes; you have to pay attention to which end of the hole is where the draft in opposite directions meet. In this case, the counterbored holes from the outside have to be drafted from the outside, so they must be sealed or shut off from the inside.

When you initiate the Shut-off Surfaces feature, SolidWorks identifies some of the necessary shut-offs for you. Figure 32.23 shows this.

FIGURE 32.23

Creating Shut-offs

When all appropriate edges around all the holes and slots are selected, the Shut-off Surfaces PropertyManager message window turns green and says "The mold is separable into core and cavity."

The tags on the loops in the graphics window will say either "No Fill," "Contact," or "Tangent." No Fill means that you do not want SolidWorks to create the shut-off surfaces. You will do these manually. Sometimes shut-off surfaces require complex or multi-feature shut-offs, which you have to do manually. The Contact condition means that the shut-off surface just needs to touch the edges, usually at a right angle. Tangent should be obvious.

Sometimes you need a combination of conditions in a single shut-off, in which case you will need to finish the feature manually. When the parting line and shut-off surfaces are complete, SolidWorks will automatically knit together all the surfaces in each Cavity and Core folder into a single surface body.

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Parting Surface

The Parting Surface in SolidWorks works best on planar parting lines that are convex all the way around. That is to say that it will work okay on a part with a parting line that looks like an "O" from the direction of pull, but may not work optimally on a part that looks like a "C." In fact, it might be safe to say that the Parting Surface feature is in many cases unusable for any but the simplest parts. The part that I have been using as an example for this section is too much for the Parting Surface feature for two reasons: it is non-planar and the parting line has two concave areas (corners where handle intersects the housing).
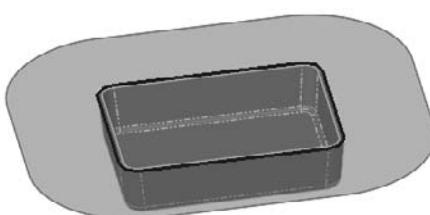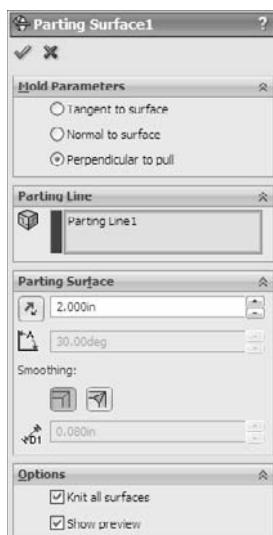
There are not enough options with this feature to make it work in situations in which it doesn't work by default. What this boils down to is for 70 percent or more of your Parting Surfaces, you will need to create your own manually, which I show you how to do in the next section.

Just to show an example that does work, I have created a very simple part and brought it to this point using the Mold Tools process. When the process works as it should, and even when you have to create surfaces manually, you will wind up with one complete surface body in each of the Mold Tools Folders — Cavity, Core, and Parting surfaces. From this you can see that the Parting Surface and Cavity Surface define the top side of the Cavity block. Likewise, the Parting Surface and the Core Surface define the top side of the Core block.

In Figure 32.24, the Parting Surface is transparent so you can see both the Cavity and Core surface bodies. The grayscale image may not show this distinctly, but if you open the part from the CD-ROM, it will become obvious.

FIGURE 32.24

A completed Parting Surface

Tooling Split
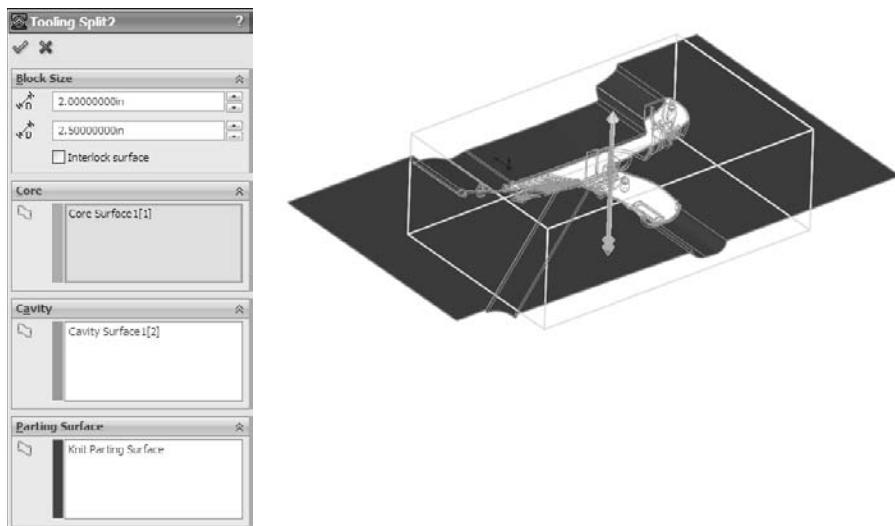
Assuming you have completed the Parting Surface either manually or through the SolidWorks Mold Tools, the next step is the Tooling Split. If you complete the Parting surface manually, make sure it is knit together as a single surface body, and then in the Surface Bodies folder, drag the knit surface into the Parting Surface folder. Tooling Split will not work unless all the surface bodies are in their correct folders.

Figure 32.25 shows the PropertyManager for the Tooling Split feature, along with a preview of the feature. The feature will produce two solid bodies, representing the cavity and core blocks of the mold. This model is included on the CD-ROM, under the name `Chapter 32 – frame mold tools.sldprt`.

The Tooling Split PropertyManager and finished product

A tooling engineer would probably change a few things about the layout of this split, but for the purposes of learning how the tools work, this is sufficient. The Parting Line of the front part of the device should probably face forward instead of up to prevent as much vertical steel in the mold as possible.

To send the cavity and core blocks to a shop for mold building, you will probably want to separate the multi-body part into individual part files. Use the techniques from Chapter 26 for this (Save Bodies, Insert Into Part, Insert Into New Part).

Note

To check the cavity and core blocks to make sure that they make the shape desired, make a new block that is larger than the original part, making sure to deselect the Merge Result option. Then use the Combine tool to subtract the mold parts from the new block. Then use the inverse scale to shrink it back down to finished part size (1original scale factor).

Also note the Interlock surface option in Figure 32.25. Most if not all of the examples of molds that you see created with SolidWorks mold tools are going to employ parting line interlocks. This is not because most molds are built that way, but because it is the main way that SolidWorks gets around the limitations in the Parting Line functionality. ■

Core
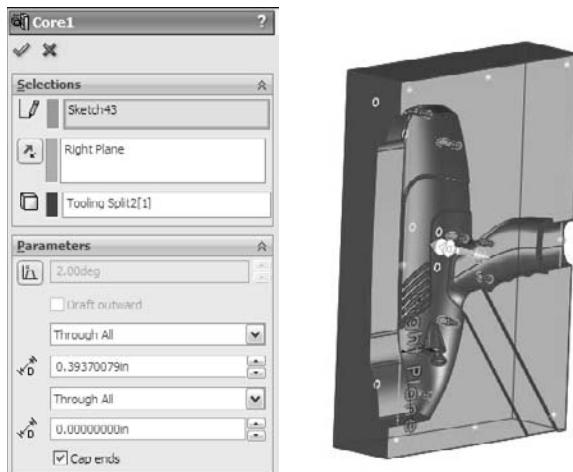
I will use the Core feature to create a set of core pins. All the standing steel that creates the counterbores for the screw bosses is made from separate replaceable pins. You can use many techniques to locate pins rotationally. This is not a lesson in mold design, but a lesson only in mold modeling techniques.

You can either pre-create a sketch or just make a sketch when the Core feature asks you for it. The Core feature is looking for a sketch that will cut out the block of mold material that you want to make a core of. Again, you can use this for side cores or core pins. In this case, I want to make several core pins.

To start, activate the Core feature; then sketch circles centered on each of the screw boss cores in the Cavity body. When I exit the sketch using the Confirmation Corner, SolidWorks prompts me for an extrusion depth for the sketch to create the feature. The Core PropertyManager and the feature preview are shown in Figure 32.26.

FIGURE 32.26

The Core feature

Again, you can save out these core pins as individual part files. You can use similar techniques to create side cores or lifters or other types of side actions.

Intervening Manually with Mold Tools

You have already seen that any sort of mold modeling resembling even a moderately complex part requires some level of manual intervention to get the Mold Tools to deliver usable results. You can do the entire mold modeling process manually, without using any of the semiautomated tools from Mold Tools. You may even come across situations where you do not need to use surface modeling at all. These situations will tend to be parts with a planar parting line, with no shut offs or cores.

I know several experienced mold designers, and they all tend to use different techniques, from cutting away chunks with solids, to using all manual surfacing methods, to using about 80 percent Mold Tools techniques and the rest manual surfacing. To me, it makes most sense to use the Mold Tools for the things they are good at, because they do speed up some tasks, such as planar shut-offs and separating out the cavity and core faces.

I want to run through two examples of manually intervening in the Mold Tools process. In the first, I will show you how to create a passing shut-off (shut-off with a stepped parting line), and in the second, I will show you how I created the Parting Surface shown in Figure 32.24.

Passing shut-offs

Snap features are often achieved in molds by using passing shut-offs rather than some sort of a lifter or horn pin slide. Eliminating actions from a mold can be economical, as long as the passing shut-off does not introduce wear or alignment problems. When creating parts that require this sort of feature in the mold, it is a good idea to consult your mold builder.

Passing shut-offs can be difficult to visualize, even for seasoned professionals. It might be a good idea to open up the part on its own and see the geometry for yourself. The filename on the CD-ROM is `Chapter 32 – passing shut off start.sldprt`. This is a clip that holds a CD in place in a plastic case. The draft analysis colors have been left on it to help you see which faces belong to which side of the mold. There are no undercuts on this part, as shown in Figure 32.27.

In this part I have actually modeled two pair of passing shut-offs.

Using the rollback bar is probably the best way to see what is going on with this part. The surfacing involved here may be confusing to you if you are not well versed with surfacing, but looking at the part and understanding the steps will help you learn. The basic steps to create the surface body called Shut-off 1 are as follows:

1. **Create Ruled surface for the planar edges.**
2. **Loft surfaces between the parting line edges and the Ruled surface.**
3. **Extrude a flat shut-off face at the parting line of the snap feature.**
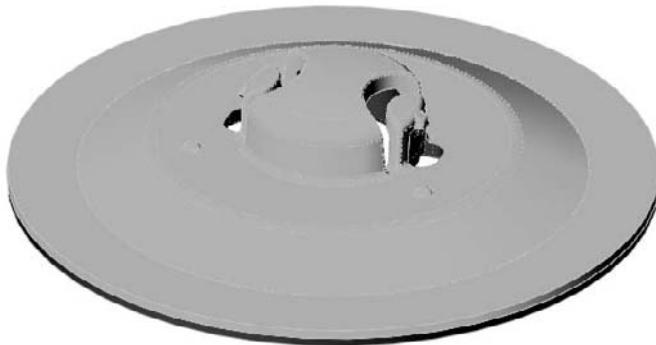
4. Use the Cavity or Core knitted body to trim the extruded surface.

5. Use the extruded surface to trim the ruled and lofted surfaces.

6. Knit the surface bodies together.

A part that requires passing shut-offs.

The hardest part of creating this passing shut-off is visualizing what the interface between the steel from opposite sides is going to look like. It is best to keep it as simple as possible. Tool builders request a wide range of angles for the passing shut-off (mold steel touching at steeply angled faces). I have heard them say that the minimum draft they can possibly stand is anywhere from 5 to 15 degrees of draft. I try to give at least 8 degrees, and more if I can. The tool builder will also look for a minimum land on the top of the shut-off boss, generally not less than 1 mm, or approx 0.050 inches, to work with round numbers.

Don't be discouraged if you don't completely understand this the first time around. The concept itself is difficult, and visualizing the geometry is extremely difficult.

Non-planar Parting Surfaces

Frankly, the method SolidWorks uses to create the Parting Surface is insufficient for most tasks. It will work well if you are molding a range of Frisbees or dinner plates, but it will not work well for handheld medical devices. Figure 32.28 shows the part on the CD-ROM named `Chapter 32 – frame parting surface.sldprt`. The result is entirely unacceptable for several obvious reasons.

From this you can learn that the SolidWorks Mold Tools are not reliable for concave parting lines or non-planar parting lines. Flat parting line disks and boxes work well. Beyond that, expect to need to do some manual surface modeling.

Note

If you want software that will do automatic parting surfaces for you, consider MoldWorks and SplitWorks from R&B software. This software also includes highly automated mold libraries and aids to help you model and document every aspect of mold hardware. ■

An automatically created Parting Surface for the handheld medical device

To manually create the parting surfaces for this part, I tackled the hard part first, which turns out to be easy once you know a couple of tricks. The first thing I did was to create a sketch and use it to lay out directions that I could pull off the non-planar sections of the parting line. Figure 32.29 shows three lines that identify the non-planar top, base of grip, and trigger areas. The sketch lines lead in directions that those edges could be projected without running into other geometry.
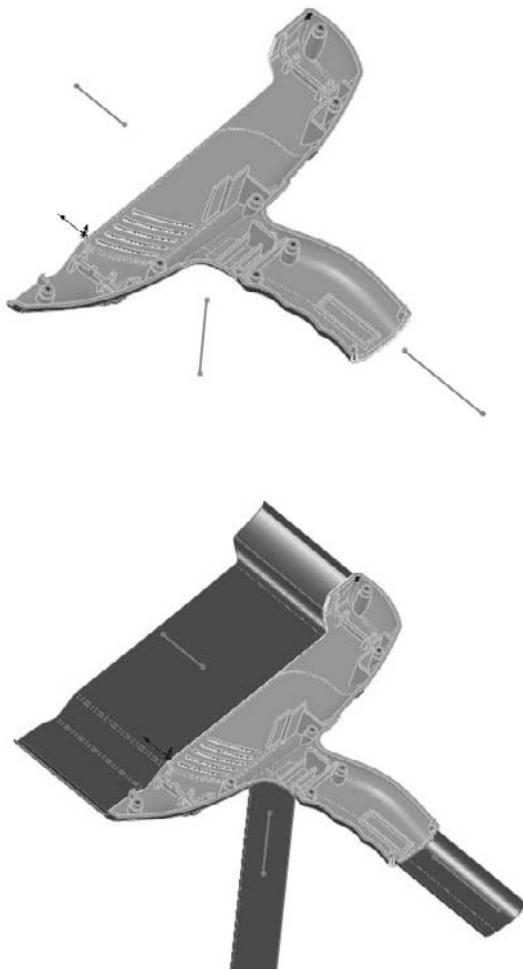
Then the edges of each non-planar portion of the parting line were converted into sketch entities in a 3D sketch, and extruded as a surface along each of these three directions. From there, it was simple to create planar surfaces between the non-planar sections. This technique may not work for all non-planar parting lines, but it does work for this one.

FIGURE 32.29

Projecting non-planar sections

Tutorial: Working with Plastic Features
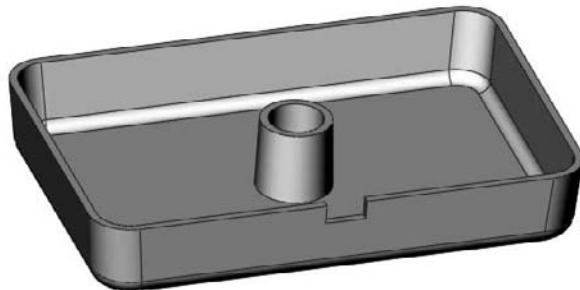
This tutorial walks you through adding several plastics features to a simple part, running some plastics evaluations on it, and then making the cavity and core blocks for the mold using a couple of different techniques. The goal of the tutorial is to make you familiar with the workflow of the tools rather than to teach every available option.

1. To create a simple plastic part, start by opening a new SolidWorks part file.

2. Draw a Centerpoint Rectangle on the Top XZ plane centered on the Origin, 4 inches (vertical) by 6 inches (horizontal).

3. Extrude 1 inch with 2 degrees of draft, using the Draft Outward option.

4. Apply fillets to the vertical edges with 0.5 inch radius.

5. Apply a fillet to the face nearest the Origin with a 0.25 inch radius.

6. Draw a circle on the Top plane centered on the Origin with a 0.75 inch diameter, and extrude it through the part as a cut, using 2 degrees draft, without the Draft Outward option.

7. Shell the part with a 0.10 inch thickness, removing the top face (large end of the extrusion).

8. Draw a centered rectangle on the Front plane 0.25 inch deep by 0.5 inch wide where the top of the rectangle is coincident with the top edge of the part. Cut through one side of the shelled block. To do this without cutting the boss in the center of the part you will have to use the From panel, extruding from an offset of 0.5 inch. Your model should look like Figure 32.30.

FIGURE 32.30

The tutorial model as of Step 8
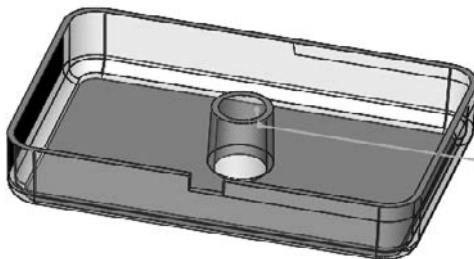
9. To create a Split Line, on the Front plane, draw a line from the bottom-right corner of the rectangular notch cut in Step 8 horizontally off the right side of the part. Make sure it goes past the part. Draw another short line from the bottom-right corner of the rectangular notch so that it makes a 100-degree angle with the horizontal line.

10. Use a Split Line to split all the faces that the lines project onto (should be a front, a back, two fillet faces, and a side for five total faces). Figure 32.31 shows the Split Line and the sketch.

FIGURE 32.31
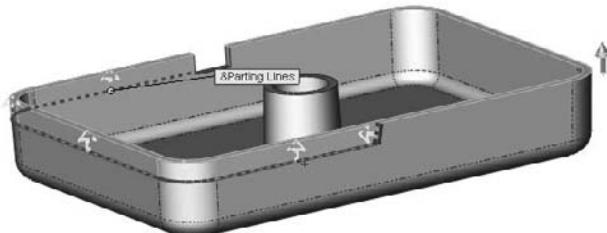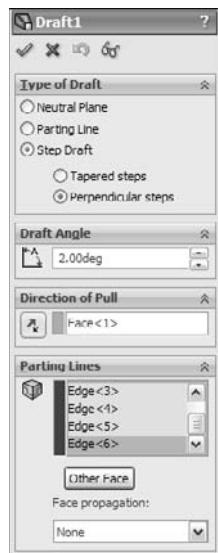
Setting up a split

11. **To create a Step Draft, initiate a Draft feature.** Use the Step Draft option. Select Perpendicular Steps. The draft angle should be 2 degrees. The direction of pull is the top thickness face of the box. Parting lines are the six edges of the split. Make sure all the yellow arrows are pointing to the same side of the split edges. Figure 32.32 shows the Step Draft in action.

FIGURE 32.32

The Step Draft feature in action

Note

You need to pay attention to the direction of the arrows for the draft (pointing up) and the selected faces (also pointing up). These arrows can have a mind of their own, and when you change one, it often changes the others without asking if that's what you want to do. You may have to individually select edges from the Parting Lines box and click Other Face to get all the arrows pointing in the right directions. ■

12. **Click OK to accept the feature.** Notice the drafted face steps out from the main part faces. Take a moment to examine the result of the Step Draft.

13. **To create Rib features, on the Front plane, create a sketch like the one shown in Figure 32.33.** Initiate a Rib feature, and make it 0.075 inch wide at the base (by selecting the At Wall Interface), with 1 degree of draft. Make sure you are using the Parallel to Sketch option (skyline).

14. **Open a sketch on the horizontal face of the rib that is 0.3 inch above the Origin, as shown on the lower image of Figure 32.33, and create the sketch shown, with a complete circle crossed by three lines.**

15. **Create another rib.** This time use the Perpendicular to Sketch (plan view) option. The thickness is again 0.075 inch at the base with 1 degree of draft. The part at this point should look like Figure 32.34.

16. **Add the Mold Tools to your CommandManager (right-click a tab and select Mold Tools), or if you are not using the CommandManager, turn on the Mold Tools toolbar (by right-clicking on a toolbar and selecting Mold Tools).**

17. **To scale the part, add a Scale feature with a factor of 1.008.** Scale about the Origin of the part.

18. **To initiate the Parting Line tool, use the Top plane as the Pull Direction, and set the draft angle to 1 degree. Click the Draft Analysis button.**

19. **Notice a purple parting line goes all the way around the part, but a warning message appears at the top of the Parting Line PropertyManager.** The warning says that the parting line is complete, but you need to also create a shut-off surface. Figure 32.35 shows the Parting Line PropertyManager and the model at this point.

   You might also notice that the two side faces of the notches don't have draft. For now, go ahead with creating the mold with the faces like this, and as an exercise, come back later and add the draft and watch it propagate through the surface features into the mold blocks.
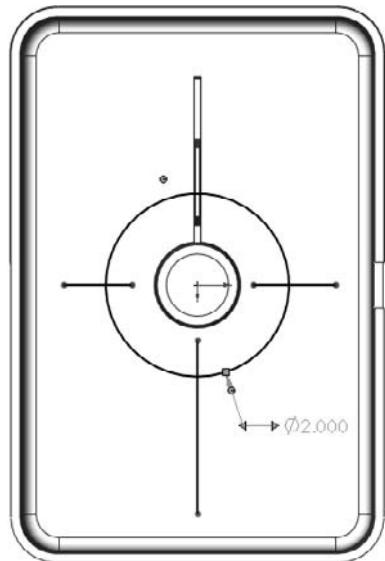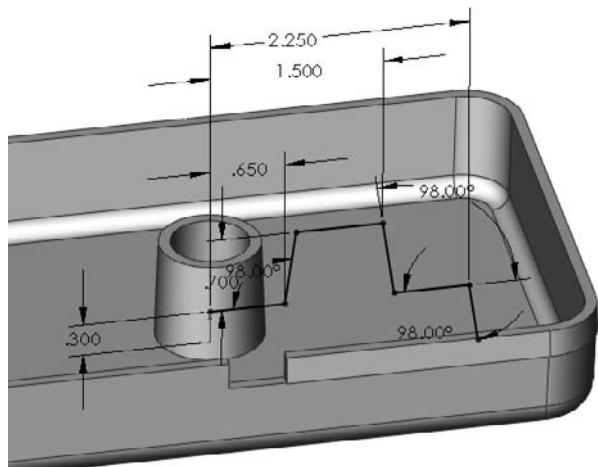
Note

You may need to deselect some edges around the rectangular notch. The edges selected for the Parting Line should be a clean single loop of edges that always separate the red faces from the green or yellow faces. Be careful that the Parting Line goes around the Step Draft faces correctly. ■
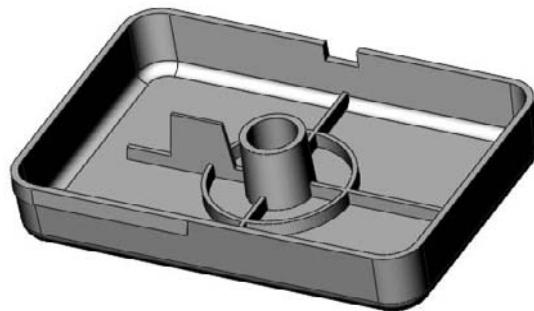
FIGURE 32.33

Creating a skyline rib
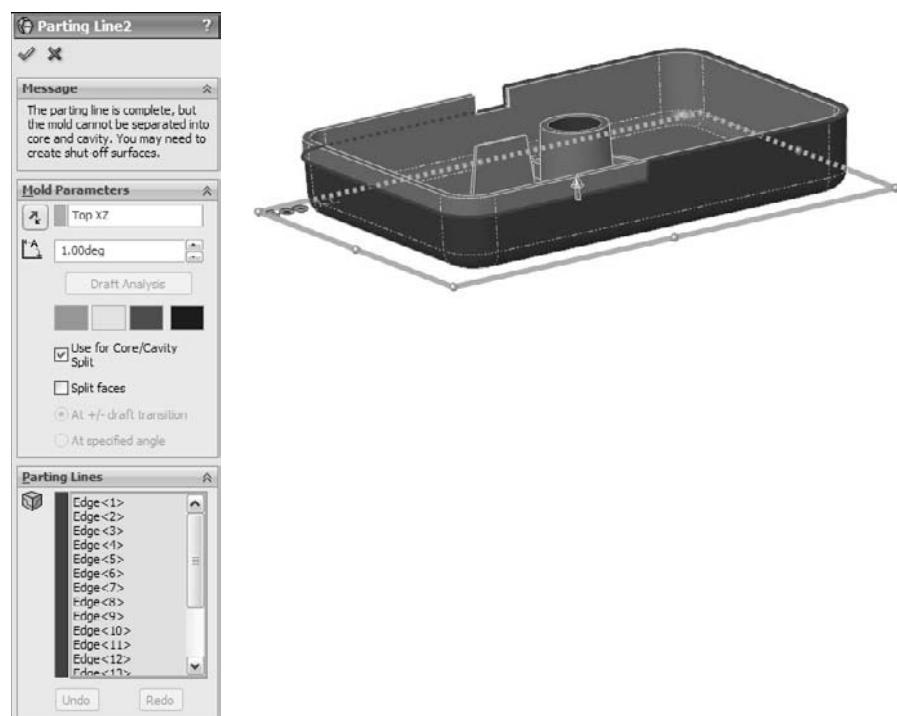
FIGURE 32.34

The part as of Step 15

FIGURE 32.35

The Parting Line PropertyManager and the model up to Step 19
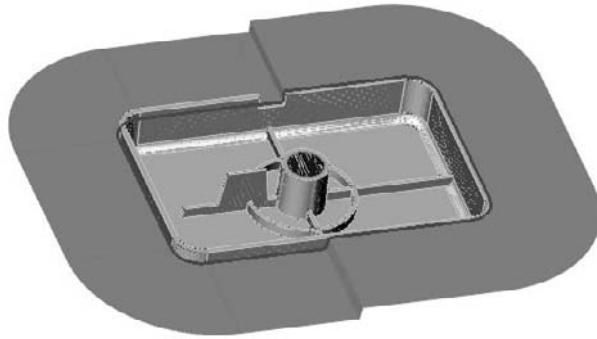
20. **Click the green check mark button to accept the Parting Line feature.**

21. **To create Shut-off faces, click the Shut-off Surface tool on the toolbar.** It should automatically find the hole in the middle of the part and understand where the shut-off needs to go. Click the All Contact icon at the bottom of the PropertyManager and then click the green check mark button to accept the feature.

22. **In the FeatureManager, expand the Surface Bodies folder.** Notice that it has subfolders for Cavity and Core Surface Bodies.

23. **To create the Parting Surfaces, click the Parting Surface icon in the CommandManager.** Assign a distance of 2 inches, and make sure that it automatically picked up the Parting Line feature. The Mold Parameters option should be set to Perpendicular to Pull.

24. **Examine the preview of the Parting Surface.** Notice that it looks good, but not perfect. In this case, you will call it "good enough," although the angled lines around the rectangular cut out are not really "good enough." Ideally, you might remodel that face later, given it should all be theoretically planar anyway. Notice how the surface handles the stepped parting line created by the split line feature and the Step draft.

25. **Click the green check mark button to accept the feature.** The part at this point is shown in Figure 32.36.

FIGURE 32.36

The model as of Step 25

26. **Click the Tooling Split icon in the CommandManager.**

27. **Select the Top plane, and draw a centerpoint rectangle centered on the Origin so that it is 6 inches by 8 inches, and the sketch fits within the bounds of the parting surface.** Make sure that you are looking at the part from the Top view.

28. **Exit the sketch by using the pencil icon in the ConfirmationCorner.** This enables the Tooling Split feature to continue.

29. **View the part from the side, and change the depth numbers at the top of the Tooling Split PropertyManager so that the block goes down .5 inch and up 1.5 inch.** Accept the feature when you are done.

30. **Right-click on the blue parting line in the graphics window and select Hide.** Also hide solid and surface bodies so you can see the inside of the block.

Look at the finished part provided on the CD-ROM if you need a reference. Use the controls on the Display Pane to hide and show bodies. Also use the Isolate option on right mouse button menus for the bodies folder and the body in the graphics window.

Summary

SolidWorks provides a vast amount of plastics functionality. In this chapter, I've given you an introduction but there is far more for you to learn as you go. These features will become second nature after you have used them a couple of times. The power and flexibility is amazing when you think of the incredible range of parts that you can make with these features.

SolidWorks Mold Tools, by contrast appear immature, but you can complete the mold process multiple ways, including using manual surfacing techniques. Rarely does a problem come up such that there is simply no way to do what you need to get done, but you do need to be inventive with work-arounds sometimes.

Animating with MotionManager

SolidWorks renamed the Animator product MotionManager, and it is now available in SolidWorks Standard. The MotionManager enables you to create movies of parts and assemblies. These movies can show something as simple as a part rotating or as complicated as complex machinery in motion, including motion constrained by assembly mates or motion driven by motors, springs, gravity, and contact.

This chapter does not cover Motion Analysis, formerly COSMOSMotion, because it is beyond the scope of SolidWorks Standard.

Overview

SolidWorks uses two different types of motion studies: Animation and Basic Motion. Animation uses key frames to drive the motion, and Basic Motion uses motors, springs, gravity (Physical Simulation), and collision (Physical Dynamics).

Understanding the terminology

The terminology used in this new product can be a little confusing. Here's an overview:

- **MotionManager.** Animator is now MotionManager. Animator as a separate product no longer exists. Its functionality has been absorbed into SolidWorks Standard.

- **Motion Analysis.** COSMOSMotion is now Motion Analysis. Motion Analysis is beyond the scope of this book.

- **Basic Motion.** Physical Simulation is now Basic Motion. Basic Motion uses motors, springs, gravity, and so on; it does not use key frames. It includes Physical Dynamics, which is the calculation of motion due to collisions.

- **Animation.** Assembly Motion is now Animation. Animation uses the *key frame* method, where the software interpolates between positions established by mates, free-hand drag, or positioning via Triad or XYZ values. Animation is not to be confused with *Dynamic Assembly Motion*, which is simply dragging parts in an assembly with the cursor to create motion.

Another method that you can use to capture screen motion to a movie file is to choose View⇨Screen Capture⇨Record Video. The Record Video tool also appears as a toolbar button on the Screen Capture toolbar. You can use Record Video to record whatever happens in the graphics window, including anything from using the Rollback Bar to running Basic Motion studies.

Formatting output

The MotionManager enables you to make animations within SolidWorks and output movie files as `*.avi` or a series of `*.bmp` or `*.tga` still images. You can use it with the default (OpenGL) SolidWorks display, RealView display, or in conjunction with PhotoWorks to create more realistic rendered animations.

Note

PhotoWorks will be replaced by PhotoView 360 in SolidWorks 2011. As of this writing, SolidWorks 2010 SP0 is the current version, and PhotoView 360 does not have any capability to be coupled with animation tools in SolidWorks. ∎

You can control the pixel size and frame rate of the recorded animation to help control finished file size, movie quality, and the amount of time it takes to record the animation. You can rotate or fly through single parts or assemblies. You can make assembly mechanisms move through animating mates, driving them with motors or manually positioning the parts in space.
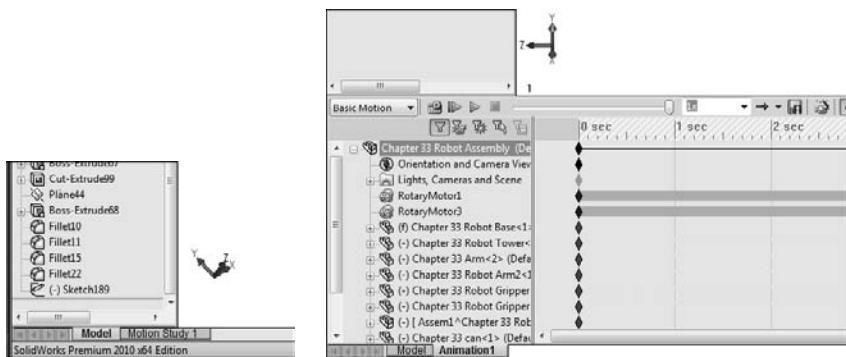
One of the beautiful things about SolidWorks animations is that you can save them to an eDrawings file. You can send eDrawings to non-SolidWorks users for review, and the file format is small so animations are especially size efficient.

Using the MotionManager interface

You can access the MotionManager interface in the lower left of the graphics window. The Model and Motion Study 1 tabs enable you to toggle the interface on and off. The Model tab shows the normal SolidWorks interface. You can add tabs to create multiple motion studies. Figure 33.1 shows the lower-left corner of the SolidWorks window with each of the buttons activated. If you cannot see this interface, you may need to turn on the MotionManager. To do this, right-click on a toolbar and select MotionManager from the list of toolbars.

FIGURE 33.1

Accessing the MotionManager interface

What can you animate?

You can animate the following:

- Distance mates
- Angle mates
- Part appearance (including display modes — shaded, wireframe, and so on)
- Part transparency
- Part visibility
- Part position
- View/zoom state
- Camera position and properties

When you animate colors and appearances, simple colors can fade from one color to another, but any appearance with a texture does not fade; it simply snaps to the next texture at the appropriate time. That is to say that you can fade red to blue, but you cannot fade marble to fabric.

You cannot animate the following:
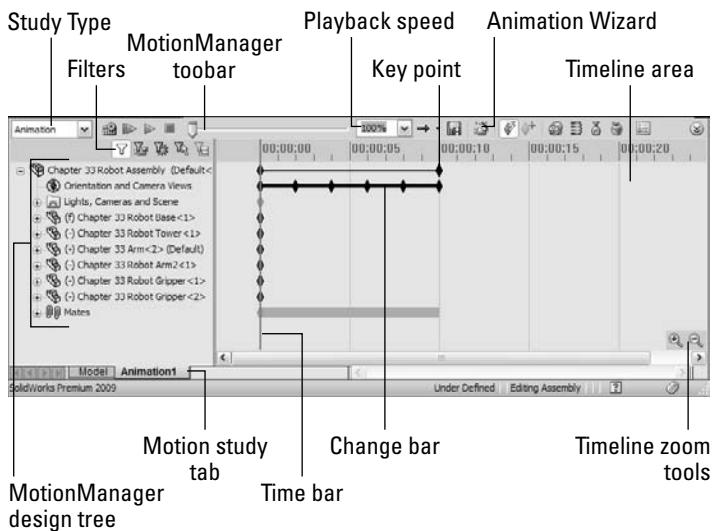
- Changing part dimensions
- Changing PhotoWorks materials
- Configurations

Identifying elements of the MotionManager

The parts of the interface you will use the most are the key points, the design tree, and the time bar. The filters help you select or view limited sets of items, and the tabs at the bottom enable you to set up alternative studies. Playback speed enables you to change the rate of playback to either take in a long animation more quickly, or to see motion in one area in more detail. The timeline zoom in and out tools enable you to rescale the time interval on the timeline. Figure 33.2 identifies the major elements of the MotionManager.

The major elements of the MotionManager

Using display options

When recording an animation to a movie file or a series of still images, you have several options for the type of display output to use. The first and easiest is the default SolidWorks display, without RealView. This is most appropriate for fast, technical presentations. You might want to use this to demonstrate the function of a particular mechanism or to simply rotate around a model to demonstrate the model in 3D rather than as a flat image or an eDrawing.

You can also turn on RealView and record the animation. If you do this, you should have appropriate appearances in use for individual parts. RealView appearances enable you to use reflective or textured materials on your parts.

The highest-quality images come through the PhotoWorks renderer. Using PhotoWorks takes much more time than the other options because each individual frame must be rendered just like a normal PhotoWorks rendering. PhotoWorks itself is beyond the scope of this book.

Planning an animation

It is often useful to plan any animation that is more involved than just a couple of moves on the screen. You can do this a couple of different ways. The easiest way is to write out a list of moves or positions you want display, with the approximate time of each position or action.

You might also use the storyboard technique professional video houses use. You create a series of images to represent the state of the animation at specific points in time. You can use static screen captures or hand sketches to do this, depending on the complexity of the geometry and animation.

Using the Animation Wizard

The easiest animations are those you can create with the Animation Wizard. The Animation Wizard accommodates two types. The first is where a part or assembly is simply rotated on the screen, and the second uses an existing exploded view from an assembly. You can combine, reorder, reverse, copy, or move both types of animation sequences within a larger animation.

Creating a rotating animation

To create a rotating animation, first click the Animation1 tab at the bottom-left corner of the graphics window. This opens the MotionManager. Remember that you can select or deselect the MotionManager in the list of toolbars. (Choose Tools ➪ Customize or View ➪ Toolbars or right-click on any toolbar.)

Click the Animation Wizard icon on the toolbar on top of the MotionManager. Figure 33.3 shows the dialog box that appears and gives you the options to rotate model, collapse, explode, import from Basic Motion, or import from Motion Analysis. All options are grayed out in this case, except rotate model. The model loaded does not have an exploded view, or Basic Motion or Motion Analysis data.

After you select the appropriate type of animation and click Next, you select an axis of rotation, the number of rotations, and the direction. An important thing to note here is that the X, Y, and Z axes do not refer to axes of the part; they refer to axes on the screen. Rotating about the X axis is like holding down the right-arrow key on the keyboard. The sample animation that appears in the corner of the Animation Wizard, shown in Figure 33.4, shows what you can expect. It will change direction if you change the option.
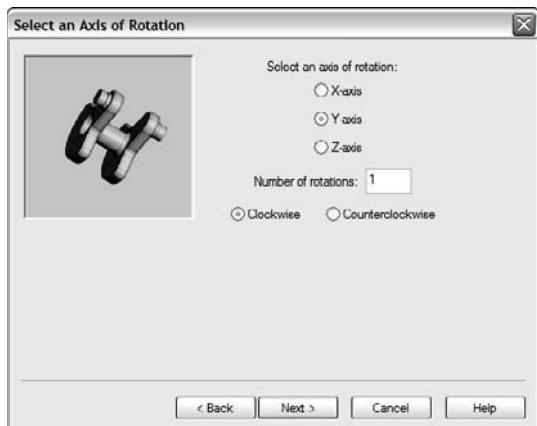
The first page of the Animation Wizard: Select an Animation Type

The second page of the Animation Wizard: Select an Axis of Rotation

The final step in creating the rotating animation is to determine how long the animation will last, and at what point in the overall animation it should start. Figure 33.5 shows the Animation Wizard page to set these options.

Note

Looping is only controlled during playback. The animation itself has a beginning and an end. If you want to play the finished movie with smooth looping, you should make sure that the start point and the end point of the animation are exactly the same. ∎

The third page of the Animation Wizard: Animation Control Options

After you click Finish, MotionManager populates the timeline with key points along the timeline for the Orientation and Camera Views. Instead of rotating the part, the software rotates the view. It seems like a semantic difference, but when you start working with moving parts in assemblies while changing the view, the difference becomes important. Notice the heavy black line with diamonds on the row for the Orientation and Camera Views in Figure 33.6. Each diamond (key point) represents a view angle, and the line between them represents that MotionManager will interpolate the view between the key points, making the view transition smoothly. You will learn how to create key points later in this chapter.

Change bars in the MotionManager
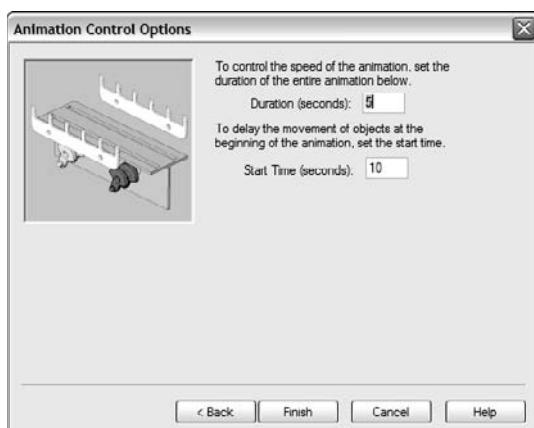
To play the animation, click the Play From Start or the Play button in the MotionManager tool-bar, shown respectively in the image in the margin.

Creating an exploded view animation

The sample assembly on the CD-ROM is named `Chapter 33 Robot Assembly`, and is saved with an exploded view. You can create your own or use the one I have provided. If you use this file, create a new animation to practice with. To use the Animation Wizard to create an animated explode and collapse, first start with an assembly that has an exploded view and activate the Animation Wizard. Figure 33.3 shows the first page of the Animation Wizard where you select the animation type. Select Explode and click Next. Figure 33.7 shows the second page. Explode animations skip the second step, which is used by Rotate animations.

FIGURE 33.7

The second page of the Explode Animation Wizard

If you added the explode at the end of the rotate, your animation does both: rotate then explode, each in sequence. Later you will learn how to copy and reverse the key points for the explode to make it collapse, and you will also learn how to adjust key points to make parts move faster, slower, or simultaneously.

Animating the View

You are not limited to the Rotate Animation Wizard to changing the view. You can manually create key points or drive a camera along a path to create the view or transition between views that help you visualize your geometry.

Animating view changes

Animating view changes is a simple task in the MotionManager, and once you learn it, you will be able to apply what you learn to making parts and mechanisms in an assembly move in much the same way.

Again, start with the robot assembly. First clear the timeline of any key points. One way to do this is to simply choose File ⇨ Reload to discard all changes, or you can right-click in the timeline area, choose Select All, and press Delete. Then set it to a Front view. An easy way to do this is to press the spacebar on the keyboard and double-click the Front view.

Orientation and Camera Views

The Orientation and Camera Views item in the MotionManager design tree is locked by default. You cannot manually change the view for the key point when this item is locked. To unlock it, right-click the Orientation and Camera Views entry, and deselect the Disable View Creation option. The icon changes from a black diamond with a red circle and line to a blue telescope.

The purpose of disabling the creation of new views is so you don't accidentally rotate the view and thus change the animation. I can tell you from experience that this is one of the most common mistakes I make when creating an animation.

Best Practice

The best way to handle the Orientation and Camera Views option is to select it (allow view changes) only when you want to establish the view key points, then deselect it when you are done. ■

To start this animation of the view, you need to deselect the Disable View Creation option, so that the blue telescope appears to the left of the entity in the design tree.

Introducing the time bar

The time bar is the vertical gray line in the timeline area that denotes the current time that you are editing in the animation. When you make a change to any element that can be animated, that change is applied at the time denoted by the time bar. To make a key-point driven animation, the workflow usually involves moving the time bar, making a set of changes, moving the time bar, making another set of changes, and so on. I do the same thing here to demonstrate how it works.

I start by making sure the time bar is set to zero (all the way to the left), and then positioning the view I want to start the animation with. In this case, bring up the View Orientation dialog box (spacebar) and double-click the view named 1.

I want the view to remain static for a couple of seconds when the animation starts. It might be too confusing to start the animation immediately with the view changing. To create this hesitation, I copy the first key point from the 0-second mark to the 2-second mark. It is as easy as it sounds. Click the key point in the same row as the Orientation and Camera Views, and then Ctrl+drag it to the right to the 2-second mark. This causes the first 2 seconds of the view to be static.

Creating key points
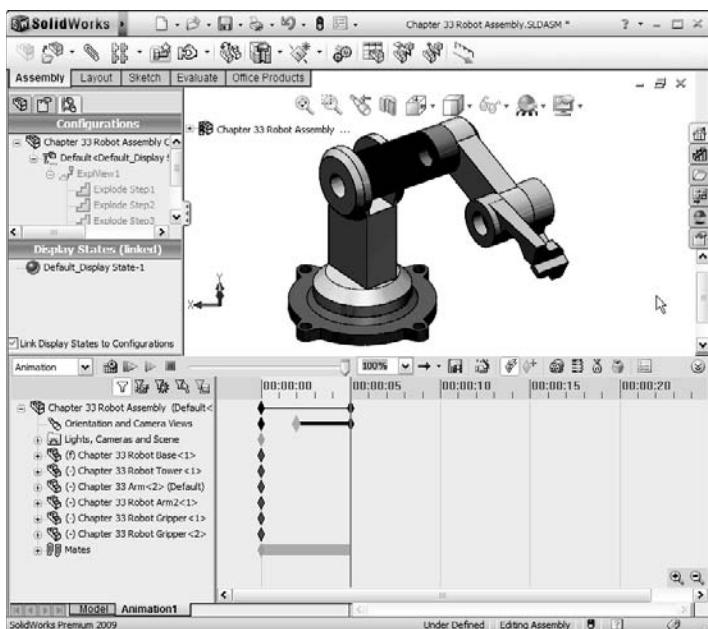
Next, move the time bar to the 5-second mark and bring up the View Orientation box again and activate view 2. This causes the view to swing around, and adds an additional key point to the timeline. The black bar between the 2-second key point and the 5-second key point indicates that MotionManager will interpolate the view orientation between the two defined points. The MotionManager now looks like Figure 33.8.

Note

There is a bit of odd functionality here. If you have the time bar selected and press the spacebar, the time bar advances 1 second. If you need to access View Orientation instead of advancing a second, first click in the graphics window or the timeline area to clear the selection before pressing the spacebar. ∎

FIGURE 33.8

The timeline at the 5-second mark

Zooming and free view manipulation

The next step is to zoom in to the grippers and simultaneously turn the view slightly to give a better view. Before changing the view, though, it would be nice to have another hesitation to give the viewer the chance to see what is there. To create the hesitation, click on the last key point in the Orientation and Camera Views row, and then Ctrl+drag it to the 7-second mark. Then move the time bar to the 10-second mark. Remember that the highlight color is now blue, not the green that

longtime users are used to. Also remember that the workflow for copying a particular key is to select, then Ctrl+drag, not just Ctrl+drag. If you Ctrl+drag without the initial select, you may be copying other key points that were also selected at the time. The select operation serves two functions: first to deselect anything else, and second to select only the key point you are interested in.
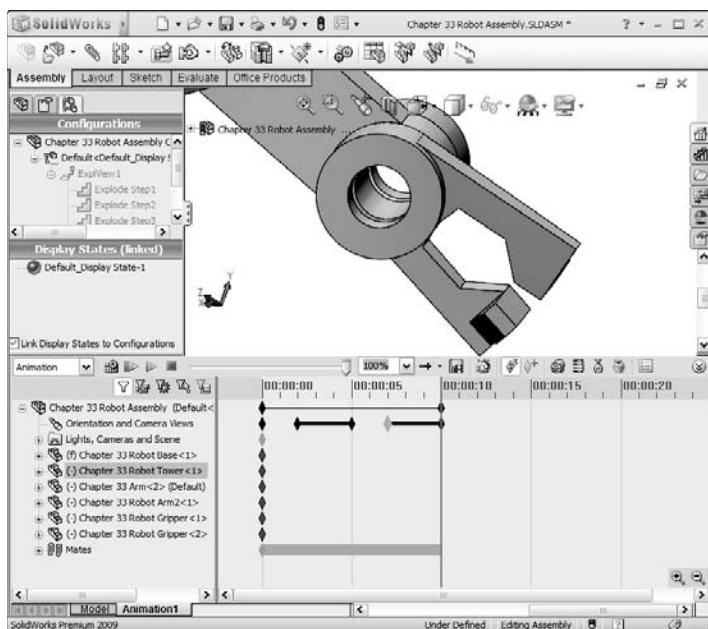
Note

When creating an animation, you have to be very careful about making changes to anything because those changes may be incorporated into the animation. If you just want to rotate the model to look at something, switch back to the Model tab near the lower-left corner of the SolidWorks window. The first tab always hides the MotionManager and you don't have to worry about changes to the views or positions of parts being recorded. ■

Once you have the time bar moved to the 10-second mark, zoom in on the grippers using whatever method you use to zoom: Shift+Z, middle mouse button (MMB) scroll, Zoom to Area, Zoom to Selection, or Zoom In/Out. When you are satisfied with the zoom, rotate the view slightly with the mouse, arrow keys, or any of the available toolbar tools. You may also want to pan the view slightly to get it positioned correctly. You need to be careful to make all the changes while the time bar is in a single location, or you may wind up with some very chopped-up view changes in the animation. The idea is to get a good partial side view of the grippers, such as that shown in Figure 33.9. Play the animation to see what you have created.

FIGURE 33.9

The timeline at the 10-second mark
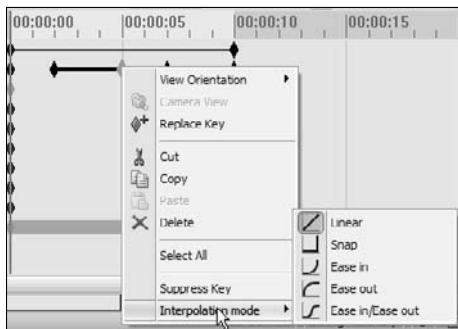
Using Interpolation modes

When you play the animation, it doesn't look very smooth. When the MotionManager interpolates between key points, either for changing views or part positions, the default interpolation mode is linear. That means that it changes between points at a constant speed. This creates the jerkiness because the motion starts and stops abruptly.

To remedy this, MotionManager offers several interpolation modes. Right-click one of the key points that you have created, and select Interpolation mode at the bottom of the list that appears. Another menu flies out, as shown in Figure 33.10.

FIGURE 33.10

Selecting Interpolation mode

The icons for the modes should be self-explanatory. My only complaint is that the Ease out icon seems upside down. In any case, curves make smoother motion than lines. Ease in/Ease out create the smoothest motion; Ease in works best at the beginning of a change, and Ease out works best at the end of a change. Snap and Linear should be self-explanatory.

The default is linear, so if you want to change all four of the key points, you have to go through this selection four times, right? No, there is an easier way. You can box+select all four key points, then right-click any of the selected key points, and change them all to the Ease in/Ease out mode. Now play the animation again. Notice how much smoother the view changes are.

Correcting mistakes

When you start to use the MotionManager, you will probably make mistakes. MotionManager does you the favor of recording them all for you in the form of adding key points to the change line for either the part position or view orientation. One way to troubleshoot these types of mistakes is to drag the timeline through the key points, identifying which key points need to be removed. To remove a key point, just click on it and press Delete.

If you are making a long animation that covers a long period of time, say more than 30 seconds, the key points may be close together and difficult to distinguish from one another. You can use the zoom tools in the lower-right corner of the timeline area to zoom the timeline in or out. Zooming in makes the key points appear further away from one another, enabling you to select one that might be right on top of another.

Other mistakes or animation problems will also come up, such as parts that don't move correctly. In most of these situations, the fastest way to deal with them is to delete the problem key points and re-create them. Troubleshooting some errors tends to be fruitless and takes longer than re-creation.

Using paths to control cameras

I introduced cameras in Chapter 5, so I will not go through the general details again. You might want to refer back to brush up on some of the controls. The main controls you need for animations are Target by selection, Position by selection, and Set roll by selection, in addition to the Field of View settings.

The main weakness of the Rotate Animation Wizard is that it rotates about the screen axis. When I first saw the part rotate that way, I wondered how I could change it. It isn't as easy as maybe it ought to be, but once you understand the process, you can make it as simple as you need to make it. I will use the example of making the camera revolve around the axis of a part regardless of the orientation of the part as the example of how to drive a camera along a path. You can make this process as simple or as complex as you need to. I will start simple and make it gradually more complex.

To state the problem explicitly, rotating the view around the axis of the screen the way the Rotate Animation Wizard does it makes the part look like it is wobbling in space, or spinning while dangling from a string. It doesn't look like it is sitting on a turntable and the table is rotating, which I would guess is the effect most people are looking for. In order to spin the view around the part axis, make a path on a plane perpendicular to the axis, and draw some sort of a path on it.

Starting with the robot assembly from the CD-ROM (`Chapter 33 Robot Assembly.sldasm`), move to a top view, and open a 3D sketch. When doing prep work like this, it is better if you can work using the Model tab, instead of the MotionManager. This prevents you from creating any unnecessary key points for animatable items.
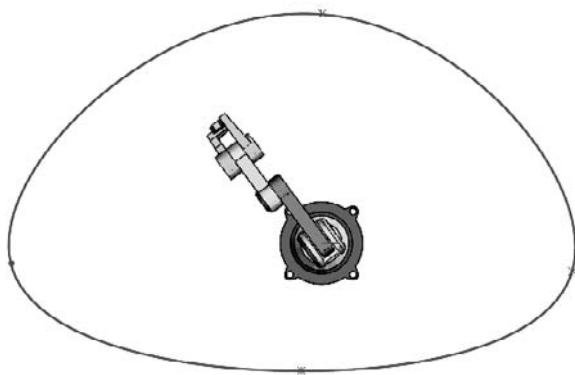
In the 3D sketch, from the Top view, draw a four-point closed loop spline, as shown in Figure 33.11. The reason I've created this in a 3D sketch is so that I can change the path to a non-planar path if I want to.

FIGURE 33.11

Creating a camera path

The path doesn't have to be perfectly circular; in fact, it might be better if it gets closer to the assembly on one side, making it rather kidney-shaped.

Note
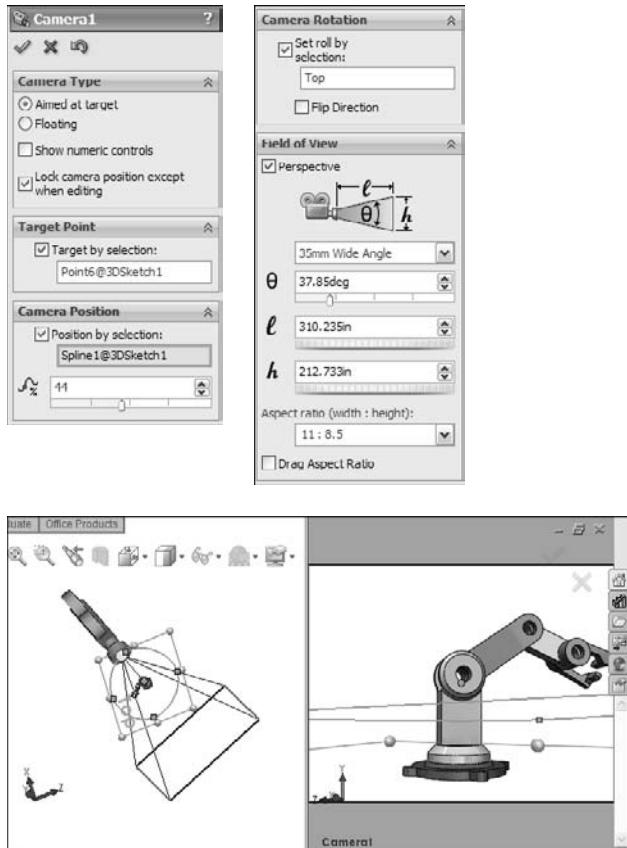
To greatly simplify this task, you can create an offset plane, and sketch an ellipse or circle on the plane rather than using the 3D spline. The 3D spline is intended to give you the most control and flexibility. ■

The camera will be attached to this spline. You might also want to have a target point for the camera to follow as it goes around the path. I placed a sketch point inside the joint between the Tower and Arm parts. If the assembly or even a part origin is in a convenient location, you can also use this as a place to point the camera.

Once the path exists, exit the sketch and insert a new camera. You can insert a camera by right-clicking the Lights, Cameras and Scene folder and selecting Add Camera. Remember that if the Lights, Cameras and Scene folder does not appear in the FeatureManager, you can make it appear by choosing Tools ➪ Options ➪ FeatureManager. Select the Show option. Figure 33.12 shows the PropertyManager for the Camera.

FIGURE 33.12

The Camera PropertyManager

Notice when you insert the camera, the SolidWorks graphics window splits into two viewports. The left viewport is your view of the camera, the model, and their surroundings. The right viewport is the view through the camera.

Using the fixed target method

To attach the camera to the spline and point it at the sketch point, make the following settings:

- **Aimed at target.** Select this radio button.

- **Target by selection.** Select this check box and select the 3D sketch point placed inside of the joint.

- **Position by selection.** Select this check box and select the spline.

- **Set roll by selection.** Select this check box and select Top plane.

You can set the camera angle now, but you should experiment a little first. In the left viewport, you can zoom the view up to a small window. Zoom to fit or otherwise zoom out until you can see the sketched spline. A yellow dot connects the lens of the camera to the spline. Drag the yellow dot around the spline slowly and observe the result in the right window. The image on the right in Figure 33.12 shows the arrangement of the viewports. As you drag the camera around the path, watch to make sure that the model stays in the field of view. Sections of the right viewport are grayed out to represent the visible field of view through the camera.

If portions of the model go out of the field of view, or you feel that the camera is too far away or too close to the model, you can move the camera or change the lens. To move the camera, exit the camera PropertyManager and edit the 3D sketch.

Note

Remember that when editing unconstrained 3D sketches, it is best to do it from orthogonal views. Any points you drag move in the plane of the screen. The best way to edit the size of the spline is to view it from the Top view, and drag out individual spline points. ∎

Once you have exited the camera PropertyManager, to get back to it is not exactly the same as most other features in SolidWorks. One way is to right-click it and choose Properties. The easiest way is to simply double-click the camera in the FeatureManager.

Using the sled method

If the first method was like filming an object on a turntable, this method is like walking around with a video camera on your shoulder. The first method attached the camera directly to the path, but in this method, you attach the camera to a dummy part, sometimes called a *sled*, and then move the sled around. The sled should be hidden when the animation is run and left visible for working purposes.

Caution

This functionality may or may not live up to your needs and expectations. You could politely call it "quirky," but it can be made to work within its limited range of capability, and works best when you do everything right the first time without needing edits. One of the cautions in the official SolidWorks documentation on the MotionManager states not to apply path mates to the sled. This seems odd, especially given this particular application looks perfect for the Path mate, but I can vouch that the Path mate does not work well in this application.

The SolidWorks documentation suggests that you use mates with the Use For Positioning Only option, so that the mate places the part, but is not added to the list of mates. I recommend you apply the mate but suppress it instead. If you need to put the part back to its original position, you can unsuppress and then resuppress the mate. The SolidWorks documentation recommends using the Move Component tool with the Along Entity setting, which is essentially the same as just dragging the part, but limiting the drag to a particular axis. ∎

Take a look at this simple animation using a sled Open the `Chapter 33 sled track assembly.sldasm`.

Start by creating a new motion study. Right-click the Motion Study 1 tab and select Create New Motion Study. Do your new work in this new motion study rather than trying to edit the existing motion study.

Positioning the sled

Notice that the assembly has two parts: the track and the sled. In this animation, the sled goes around the three sides of the sled and then stops. The first thing to do is to position the sled in the track. I would make a coincident mate between the bottom of the side and the bottom inside face of the track. Figure 33.13 shows this arrangement.

FIGURE 33.13

Positioning the sled in the track

Moving the sled

Next, create the motion of the sled around the track by using key points. Move the time bar to the next time you want to define the position of the sled, say 1 second, and then move the sled to the position it should be in at that time. You can use either simple dragging to get the part where it has to go, or you can use any of the options of the Move Component tool, shown in Figure 33.14.

When you move around the corners, remember that the sled is not going to follow an arc path; it interpolates linearly between key points. You can, however, make it rotate and translate at the same time. To do this from the end of a straight section, drag the sled with the left mouse button (LMB) to its new location, and then rotate the part by dragging with the RMB to get it headed down the next straight section. It is also recommended to use more key points when changes in curvature are greater. So around the corners, it may work better if you can place key points more densely than along the straight sections.

The assembly on the CD-ROM has an animation stored in it. You might want to check it out for reference.

Attaching the camera
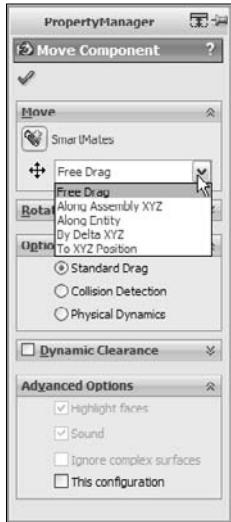
Create a new camera by right-clicking the Lights, Cameras and Scene folder. Use the steps I outlined earlier to attach the camera to the midpoint of the short side of the triangular sled, and target the pointy end of the sled. Be careful to not position or aim the camera such that it is half buried in the floor of the track or pointed at the floor. You want the camera to point in a direction parallel to the floor.

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FIGURE 33.14

Use the Move Component PropertyManager to move the sled.

You may want to mate the bottom of the sled at a distance from the bottom of the track so the camera is not too close the floor.

Hiding the sled

Hiding the sled turns out to be more of a task than you might have imagined. Because you have to be able to see the sled to position it, and you have to have a portion of the sled in the field of view if you want to use it to control the camera target, you will have created the animation with the sled always visible. If you hide it, the MotionManager actually make it transparent in the background, and fades from opaque at the other end of the timeline. You need to move the time bar to the other end of the timeline and hide the part there, too.

Turning the camera on

Once you have created the animation from the bird's-eye point of view, and you can see the camera actually travel around the track, you will want to see the animation through the camera. To do this, first deselect Disable View Key Creation by right-clicking the Orientation and Camera Views item; then right-click the camera in the MotionManager design tree and select Camera View. If you want to return to the bird's-eye view, just reverse the procedure.

In the end, using sleds has its own set of difficulties compared to simply moving the camera without attaching it to any thing. Neither is easy or convenient.

Animating with Key Points

In this chapter, I have already shown you a little about key points to introduce the idea, but here you will learn about how to use them in more detail. You can think of key points as snapshots at particular moments in time. If you said, "At the 4-second mark, the wheel needs to be three inches from the wall," this statement describes a key point. To create a key point, drag the time bar to a new time, and make a change. Any of the animatable items I listed earlier in the chapter can create a key point.
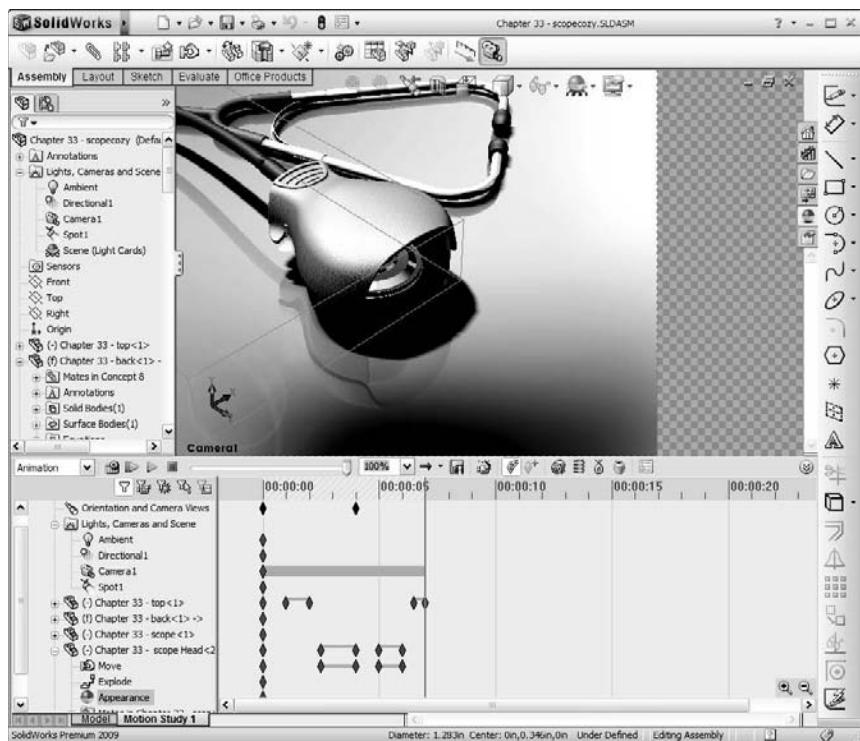
Getting started

Consider this easy and useful example: a customer wants you to make a little animation of a holder for a stethoscope that he will show to a potential client in PowerPoint. The holder opens, the stethoscope slides out, and then the animation is reversed.

The assembly with the animation saved in it is on the CD-ROM as `Chapter 33 – scopecozy. sldasm`. The assembly and the completed animation timeline is shown in Figure 33.15.

The stethoscope animation setup

This animation uses RealView display, which the customer has said is good enough for his purposes. This cuts down the time significantly compared to using PhotoWorks to render the animation.

Your first task is to set up the camera. You could do this without a camera, but cameras are a convenient way to store a particular view, along with settings such as lens angle, perspective, camera position, and target. Plus, if you do decide to use PhotoWorks later, cameras are the only way to get depth of field for additional realism. Another advantage of the camera is that you are able to control the area in view more closely. If you don't use a camera, the area of view is just whatever is available in the view port. With the camera, you can specify a size and aspect ratio, and the available area is cropped appropriately.

Because the stethoscope model is cut into pieces to enable different parts of it to be positioned, you need to position the parts and the camera such that the break between the head and ear pieces is not visible. Leave enough open area so that when the stethoscope comes out, it will not run out of the area of view.

Note

When adjusting the position of the camera, it is often easier to adjust the view itself than to manipulate the camera. In the Camera PropertyManager, deselecting the Lock Camera Position Except When Editing option enables you to manipulate the view directly. This setting is selected by default and will give you the camera with a red X icon if you try to rotate the view when the camera view is on. Switch to camera view and deselect Disable View Key Creation. ■

Using the time bar with key points
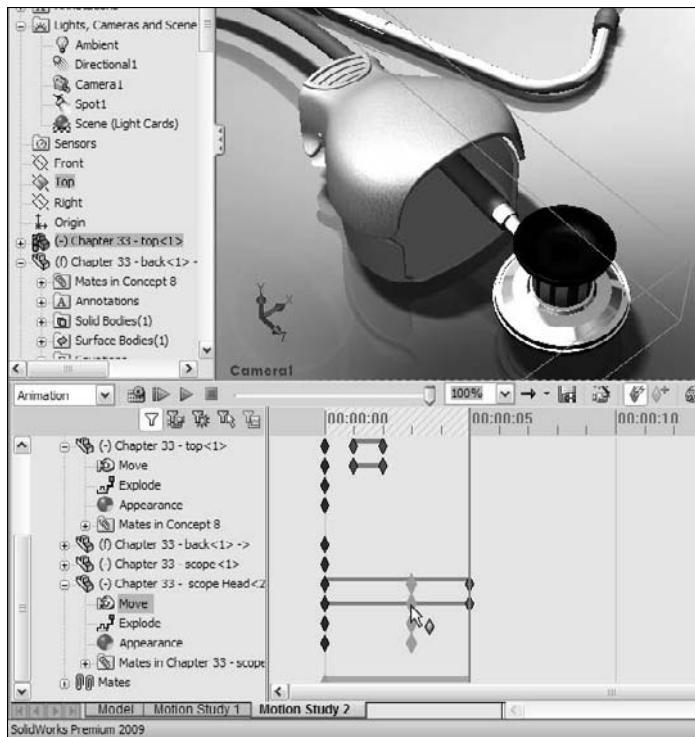
I like to start animations with some stillness. If you start an animation with motion, your viewer may not have the time to get settled. A second is usually enough time. Expand the `Chapter 33 – Top part`, click the key point for the Move row at the zero time mark, and Ctrl+drag it to the 1-second mark. This means that the top will not move between zero and 1 second. Now move the time bar to 2 seconds and open the top by just dragging it up slightly. It should only open about half an inch.

Now move the time bar to the 5-second mark. I want to create a mistake so you see one way to correct mistakes. At the 5-second mark, move the stethoscope out of the holder three or four inches. Try to make sure you do not go far enough that the rubber tube runs into the plastic parts.

Notice that this creates a change bar that shows the position of the scope head part moving continually from time 00.00.00 to time 00.00.05. The motion is supposed to start at the 3-second mark. I've shown you how to fix this in Figure 33.16.

Fixing a timeline problem

Click the key point for the motion of the scope head part. Then Ctrl+drag it from the 0-second mark to the 3-second mark.

If you run the animation at this point, you may see that the scope head does some unexpected movement, and a yellow line appears to the left of the change bar for the part. To fix this, click the Calculate button next to the Play button. I don't have a good answer for why this happens, but I have seen it on several animations.

Copying and mirroring motion
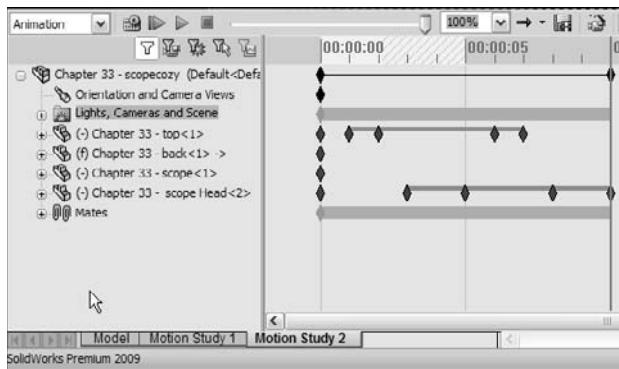
The animation is essentially done at this point, except that now the stethoscope needs to go back into the holder and it has to close. You don't have to manually create all the steps to close the device, although you could if you wanted to. It is more efficient to simply copy and reverse the paths that you have already made.

To copy both sets of motion, the top opening and the head sliding out, drag a window around the key points to select them all, and then Ctrl+drag them to the 6-second mark. Notice that this creates the situation shown in Figure 33.17. If you play the animation at this point, it is not at all what you want. It simply stacks the same motion on top of the original motion. You want it to be reversed.

Copying motion of parts

With the newly copied key points still selected, right-click one of them and select Reverse Path. Notice that this now shows symmetrical key points.

Adjusting the speed of actions

The animation is getting close to complete, but now you notice that it would be better if the second half of the animation went by faster than the first half. To do this, move the key points on the right side of a change bar toward the left. You might want to move both key points for the top part closing so that it starts closer to the time when the scope head is back inside the holder. You could even make some of the motion overlap, so the top starts closing before the scope head is fully inside.
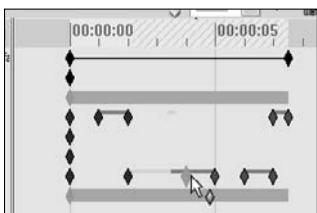
Again, if you see a strange effect like the scope head not going all the way back to where it belongs, trying clicking the Calculate button again. Calculate essentially rebuilds the animation after changes.

To make the motion a little smoother, right-click in empty space inside the timeline area, and choose Select All; then right-click one of the key points and select Interpolation Mode. Click the Ease in/Ease out option. Click Calculate again to watch the smoother animation.

If want variable speed, say for the scope head coming out of the holder (for example, it starts coming out slowly and then speeds up), you need to add at least one more key point. To do this, position the time bar to the left of the middle of the first scope head change bar, and click Place Key. This adds a key point in the existing change bar. This is shown in Figure 33.18. Then move the key point to the right. Make sure the new key point uses the Ease in/Ease out interpolation mode. Recalculate, and run the animation again.

FIGURE 33.18

Adding and moving a key point in an existing change bar

If you decide that the entire animation is too fast or too slow, you can also adjust this easily. Drag the right-most key point on the very top row with the Alt key depressed. This scales the entire animation up or down.

Getting output

Once the animation suits you, click the Save Animation toolbar button. This brings up the Save Animation to File dialog box, shown in Figure 33.19. The options for output formats are `*.avi`, or a `*.bmp` or `*.tga` series of still images. You could combine the still images to make an animated GIF to use on a Web site. Other types of output like Flash or QuickTime are not available directly from the SolidWorks software. Movie format converters are available on the Web for this purpose.

The options for the renderer are simply the SolidWorks screen or PhotoWorks. In this example, I used RealView and the SolidWorks screen renderer, which provides sufficient quality for my purposes. The main advantages of PhotoWorks over RealView are it offers a better choice of backgrounds, anti-aliasing, and more shadow control.

Other options
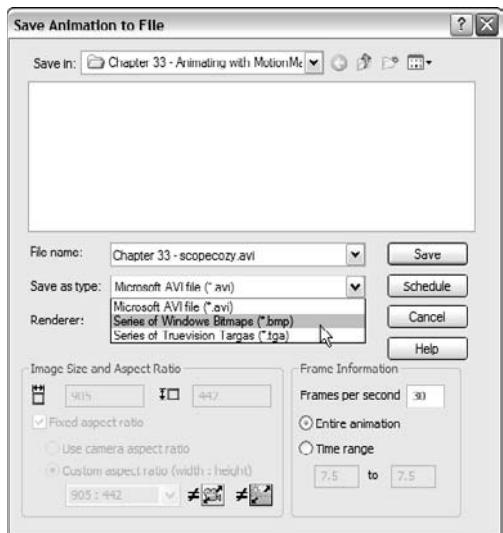
Image Size and Aspect Ratio options are available only when you do not use a camera. Without the camera, you are at the mercy of the size and shape of the SolidWorks graphics window until you save the animation to a file.

The Schedule button enables you to schedule the output for a more convenient time. You would normally use this option when using the PhotoWorks, because rendered animations can take many hours to complete depending on render settings, length of animation, and the frames per second setting.

FIGURE 33.19

Saving output data for your animation

Frame Information enables you to set the quality of the finished rendering. Low frame rates result in choppy motion. High frame rates will be much smoother, but the files may become unmanageably large. High-quality animations generally fall into the 25 to 30 frames per second (fps) range. The human eye cannot resolve faster rates, so it is usually pointless to go any higher than 30.

Test animations

Depending on the length of the animation and the other settings, test animations might run in the 10 fps range. You might also consider using a specific range of time to test just a portion of the animation.

Unfortunately, many of the decisions that you make regarding animation quality settings directly relate to the time you have to produce the final movie file. The biggest time saver is to avoid PhotoWorks. If RealView suits your needs, you are well ahead on time.

Selecting a compressor

When you go to save the animation, the software prompts you to select a video compressor (codec). Typical options are the Microsoft Video and the Cinepak compressors. Sometimes when you record or play back a movie with a particular compressor, you get a lot of video garbage in the movie. If this happens, try another compressor. I used Microsoft Video for the first movie I recorded of this animation, and there was a lot of video noise. I switched to Cinepak and it worked perfectly.

Using Animations to Flex Parts

You cannot change part dimensions when animating. You cannot use features like the Flex to bend parts for an animation. If you can't do any of this, how do you make parts flex during an animation? This calls for another workaround, but it is the only way to actually make parts change shape with the MotionManager. You have to find a way to model the part such that you can make the changes using distance and angle mates in an assembly. That means that you have to do in-context modeling.

A lot of people treat in-context modeling as if it is some terrible infectious disease, but as I pointed out in Chapter 16, it is nothing to be afraid of if you know how to handle it. Still, when you're flexing parts with in-context tricks, it is best if you can keep it simple.

Figure 33.20 shows an assembly that uses this technique to twist the strap as the fork rotates. The assembly used to create this animation is on the CD-ROM, and is called `Chapter 33 yoke link.sldasm`. The finished movie file is also on the CD-ROM, and is called `Chapter 33 yoke link.avi`.

FIGURE 33.20

Twisting a strap during an animation

The main trick here is that the strap is made from three separate features. One feature is a revolve that fits the bottom shape of the pin. This feature is part of a part that moves with the fork and pin as they rotate. Another feature is an extrude made from an in-context feature. The in-context relation goes to a dummy part that remains stationary with respect to the rotating fork. And the final feature (really a pair of features) is a loft that goes from the ends of the rotate feature to the ends of the extrude feature.

As the fork part rotates, the in-context relation causes one feature to stay still while the other rotates, and the constant rebuilds the MotionManager does make it look like the strap is flexing.

You can use the same sort of idea to flex living hinges in plastic parts, or just about anything. A cleaner way to do this is to create planes in the in-context part, and then everything else is sketches. If you can drive the flex illusion with planes, you can get this trick to work in a wide range of situations. You usually wind up needing a loft feature, because no other feature is quite as flexible as a loft. It is possible to sometimes use other types of features, but I like the loft the best.

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To create the actual motion in the MotionManager, you handle things just like in the other examples, but when you move the time bar, you double-click the angle mate and change the value. Driving animations with angle or distance mates is an effective way to get more exact motion.

In this case, the angle mate is the only thing driving the motion. There is some view orientation movement at the end of the movie, where I also copied the motion from the first half of the movie, and ran it again a little faster with different angle values closer up.

Animating with Basic Motion

Basic Motion is the functionality formerly known as Physical Simulation. It involves setting motors to turn parts, gravity to move parts and springs, and collisions to create animations that cannot be driven by mates or free motion. It uses a different solver than the rest of the animations in this chapter.

Basic Motion does not take into account effects such as momentum, bounce, resistance/friction, viscosity, reaction forces, and so on. To analyze for these effects, you will need to use Motion Analysis (formerly COSMOS Motion).
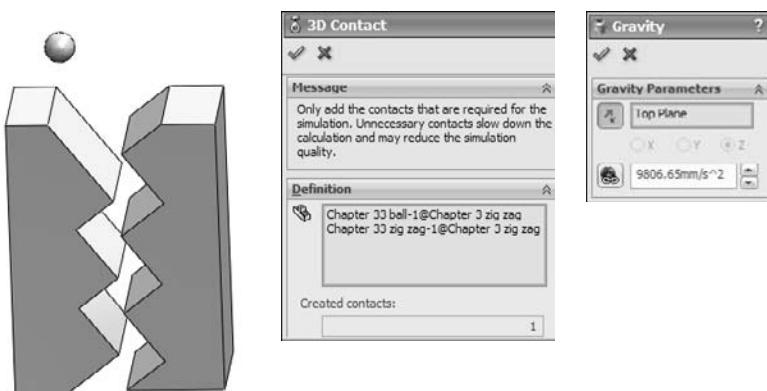
The study type selection box appears in the upper-left corner of the MotionManager. You will need to use Basic Motion, shown in Figure 33.21, for this example.

Using gravity and contact

Figure 33.21 shows an assembly that demonstrates the gravity and contact functions of Basic Motion. The problem is easy to set up. The part that is to move (ball) is underdefined, using only one mate to keep it in plane as it moves. The zigzag part uses a Fixed constraint. The assembly used in this example is on the CD-ROM and is called `Chapter 33 zig zag.sldasm`.

FIGURE 33.21

Setting up contact and gravity
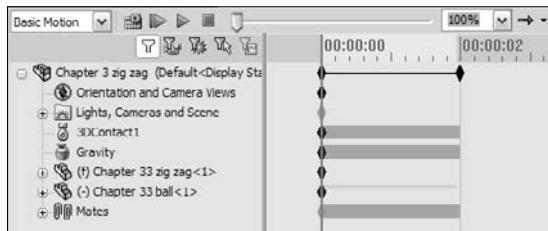
When you have added the physical simulation items, the MotionManager design tree looks like Figure 33.22. Editing items such as contact and gravity does not use the interface options that have been available in the rest of the SolidWorks software starting in the 2008 release. Left-click (select) does not bring up a context toolbar; you have to right-click and access the full RMB menu.

Because this example goes by so quickly, you may wish to use the Playback Speed drop-down menu to get a better look at it. You can also set playback looping options with the drop-down menu to the right of the Playback Speed.

FIGURE 33.22

The MotionManager design tree with added items
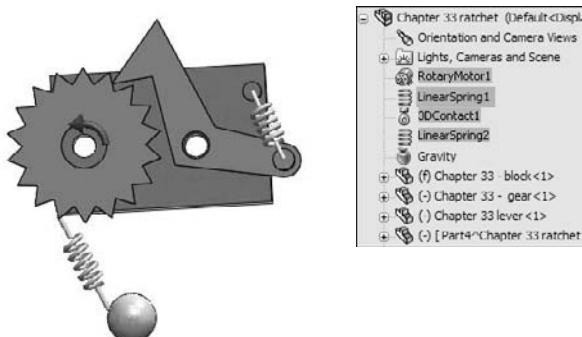
Using motors and springs

The use of motors does not necessarily require Basic Motion, but if you include springs, or contact or gravity problems, it does. Torsion springs require Motion Analysis, but linear springs only require Basic Motion. This example (on the CD-ROM as `Chapter 33 ratchet.SLDASM`) shows a motor driving a gear with a ratchet held to the gear teeth by a spring. I've added a swinging ball on a spring to show this isn't simple 2D functionality.

FIGURE 33.23

A motor, gear, and ratchet assembly driven by Basic Motion

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To set up this example, apply a counterclockwise motor to the inside circular edge of the gear, and select the block as the part to move relative to. In this case, I started the motor with a slow rpm (revolutions per minute), moved the time bar out a few seconds, and assigned a faster speed, so the motor speeds up over time.

The linear spring is easy to apply. Select the locations of both ends. In this case, I have circular bosses on the block and the ratchet to hold the spring.

Summary

If you keep your animation relatively simple, the MotionManager tools in SolidWorks Standard should be able to give adequate results. If you are using mates to drive motion, be sure to follow mates best practice recommendations. If you are manually positioning parts, remember to place key points closer together if the motion curvature changes.

The changes with PhotoWorks and PhotoView 360 will happen by SolidWorks 2011. Expect a lot of functionality to be added to PhotoView 360, including a link to motion studies.

Part VIII

Appendixes

The appendixes are a collection of information that doesn't fit well within the rest of the text. In particular, Appendix A lists several readily available sources for help with SolidWorks that you can use for free. Appendix B familiarizes you with the CD-ROM that comes with this book.

Finding Help

S olidWorks software has been around for more than 15 years now. In that time, the resources available to users seeking help have increased dramatically. These resources take many forms, from personal Web sites with information from individual experience to commercial online magazines or forums with advanced Flash interfaces. In this appendix, I have assembled some of the more worthy sources of quality information.

It is not the goal of this book to endorse any commercial sites or services, but some of the listed resources are commercial in nature, and may feature advertisements, logins, or both.

SolidWorks Help

The SolidWorks Help file contains most of the information you need to research how functions work. There are topics for which searchability is poor, or that do not appear under expected names, but this is the exception rather than the norm. It is more common to find that some features are poorly documented or not documented at all. For example, when you look up the significance of the Engage Belt option in the Belt/Chain sketch function described in Chapter 4, SolidWorks Help says "Engage belt. Clear to disengage the belt mechanism," which is incredibly obscure?

However, some functions, such as sheet metal bend allowances and the referenced documents search routine, are extraordinarily well documented. Isolated topics are surprisingly well documented.

SolidWorks Help is available in traditional Help files on the computer, as well as online. SolidWorks Web Help was created because it is easier for

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IN THIS APPENDIX

Getting SolidWorks Help

Accessing information on SolidWorks Web sites

Benefitting from user groups

Getting support from online forums

Using non-commercial Web sites

SolidWorks to keep it up to date. You can access the Web Help by turning on the Use SolidWorks Web Help option in the Help menu in SolidWorks or at `http://help.solidworks.com`.

SolidWorks Web Help

The SolidWorks Web Help is new in SolidWorks 2010. Web Help was introduced to help SolidWorks keep the Help files updated without having to send out help updates in the service packs. It also enables the use of other search tools and links to online data sources to make finding help that much easier.

You can access SolidWorks Web Help through the Help ⇨ SolidWorks Help menu selection as usual, but you need to make sure that the Use SolidWorks Web Help option also in the Help menu is activated. The following three sections only pertain to the traditional (non-Web) Help.

Contents

SolidWorks terminology has been a sticking point at times in the writing of this book because terms are either unclear or overlap. Still, it is difficult for two people to talk about the software if they are not using the same terminology. The Glossary, found at the bottom of the Contents list, is one of the most useful and yet most underused portions of the Help files. Often when a new user asks me a question, it can be impossible to discern what the user is talking about because he is not familiar with the SolidWorks terminology, is substituting AutoCAD or Inventor terminology, or is assuming all modeling terminology is universal. As dull as it may be, this Glossary should be required reading for all new users. Simply understanding the language being used by the training materials, Help files, and other users can give you a big hea start when it comes to learning the software. Look through it. I promise you'll learn something.

Index

Starting in SolidWorks 2010, SolidWorks has removed the Index from the regular Help, and of course, it does not exist in the Web help either. An index is difficult to create, and this difficulty was the reason SolidWorks gave for not including it in the new version.

Search

The Search function is for when you are not exactly sure of what you are looking for. For example, you know there is a feature that has a funny name that uses stripes to analyze curvature across edges, but you cannot remember its name. Begin your search with the words *stripes* and *curvature*. If you use curvature, the Search function returns about 60 possibilities. The term you are looking for is Zebra Stripes, but you may not find it by scanning such a long list. If you search using the word stripe, Zebra Stripes appears at the top of the list.

The biggest complaint you may have about the Search function in Help is that it will sometimes return too many options, and the connection between the word you searched on and the topic title shown in the list is not immediately clear. Still, too much information is better than not enough.

Many users overlook the three options at the bottom of the Search window: Search Previous Results, Match Similar Words, and Search Titles Only. All three are useful in narrowing your search. With the new Web help, Search is improved, and includes a section that allows you to further narrow the search results that works like guided search.

Additionally, most of the Help linked to from the search results has a link to search the Knowledge Base on a related phrase. This search of the Knowledge Base (KB) may or may not include any results. I have seen several links from the Help to the KB that were empty.

SolidWorks Web Site

Most of the valuable information on the SolidWorks Web site (`www.solidworks.com`) is behind the subscription login, but some free information is also available. It may be worthwhile to explore the SolidWorks site a bit, because it includes a large amount of information ranging from graphics cards evaluations to training files.

Graphics Cards

The link to this area of the SolidWorks Web site is `www.solidworks.com/sw/videocard testing.html`, and appears on the SolidWorks Web site main page behind the text System/ Graphics Card Requirements. SolidWorks has tested the range of most popular graphics cards and drivers for compatibility with various versions of SolidWorks, and has rated them at various levels based on the following criteria:

> Passed all tests
>
> Passed with limitations
>
> Card has significant stability or repaint problems
>
> Uses the graphics card display settings for SolidWorks
>
> Multi-head hardware accelerated
>
> Supports RealView
>
> Provides 64-bit native support
>
> Supports 3D-Stereo effects

Customer Portal

The SolidWorks Customer Portal is full of useful information. It requires a login, and you can find it at `https://customercenter.solidworks.com`.

SolidWorks Forums

The SolidWorks Forums have areas of wide interest for most users. These include about 40 different topic areas, each with a constant flow of information. SolidWorks employees sometimes

answer questions, and knowledgeable users often give good answers and invaluable perspectives on not just modeling and CAD admin topics, but also general mechanical engineering.

You can get an account to read and post messages even if you are not on maintenance. You can find the forums at `http://forum.solidworks.com`.

Knowledge Base

If you think you have used the Knowledge Base before and found it less than satisfying, you owe it to yourself to try it again. Searches actually turn up a lot of useful information. Results may include tech support responses to customer issues, SPRs (software performance reports — also known as bug reports), white papers, articles, and so on. In addition, you can look up SPR numbers you have received from tech support to check the reports' statuses. I have consulted the Knowledge Base several times while writing this book. It has been built from vast amounts of internal SolidWorks corporate support documentation, as well as the support database. I give it very high usability marks!

Software downloads

Manually downloading and installing software and upgrades for SolidWorks is becoming outdated, although you can still do it. The SolidWorks Installation Manager works much like Microsoft Automatic Update. It downloads and even installs updates for you automatically. You can also make it work with administrative image installations. I particularly like that it can download service packs before the links on the SolidWorks Web site are active. Of course, if you need or simply want to download them manually, this option is also available.

Release Notes

All the Release Notes for all the service packs of the current version are also available from the main Customer Portal window. This is essential information for CAD administrators. Technical Alerts, changes to the System Requirements (such as SolidWorks 2011 dropping support for Windows XP), and new installation details are listed here.

Even if you think you do not need to know any of this information, it still makes for interesting (and at times alarming) reading. The Technical Alerts typically warn of severe bugs or other problems and how to workaround or fix them.

What's New

What's New is a great document to refer to when you are learning a new version of SolidWorks. For the SolidWorks 2010 version, the What's New document comes in HTML and PDF formats.

Installation and administration guides

Installation and administration guides are available for SolidWorks, eDrawings, and Cosmos. They contain the basics about the topics, and are not as detailed as other sources of information. For a complete installation and administration guide, please refer to the *SolidWorks Administration Bible* (Wiley, 2010).

PDMWorks Workgroup Vault Debug Guide

When the vault gives you an error code, you don't have to call tech support to find out what is wrong. It is listed in the PDMWorks Workgroup Vault Debug Guide. All error codes appear with the name of the error. This is a must-have document for the PDMWorks administrator.

FLEXlm End Users Guide

The FLEXlm End Users Guide is required reading for a network license administrator. While the network license is easy to install, set up, and maintain, this guide explains many of the details and options that are available.

User Groups

The main SolidWorks site for user groups is `www.swugn.org`. There are too many individual user groups to keep track of or list here, but you can find a user group in your area on the SolidWorks User Group Network (SWUGN) site. You can also find user group information on the SolidWorks Forums, discussed in detail in the next section.

Online Forums

There are many types of online forums for SolidWorks. I have already discussed the forum on the SolidWorks Web site. Other forums are not directly sponsored by SolidWorks Corporation, and they may vary in quality.

Blogs

Blogs for SolidWorks and related topics cover everything from opinion-based essays to speculation about future products, tips, and tricks or CAD industry news. Most are written by SolidWorks users rather than journalists, so you'll get specifics about a tool that you actually use.

I write a blog, which you can find at `http://dezignstuff.com/blog`. It covers advanced SolidWorks subjects such as surfacing, as well as consumer advocate type of articles, including information on enhancement request drives and subscription options. I also provide links to many other SolidWorks-related blogs, forums, and Web sites.

My blog has updates to books, notices of new books, lists of errors found in books, and a lot of other content. I deal with philosophical and ethical questions related to product design and CAD in general. I post advanced tips, and sometimes excerpts from books. I also post questions for readers and polls where readers can express their opinions; and I'm always looking for opinions as long as you can back up what you have to say.

Some SolidWorks blogs to check out include:

Matt Lombard at `www.dezignstuff.com/blog`

Ricky Jordan at `www.rickyjordan.com`

Rob Rodriguez – Rendering in SolidWorks at `www.robrodriguezblog.com`

Devon Sowell at `http://designsmarter.typepad.com/devonsowell/`

Devon Sowell – SolidWorks PDM at `http://pdmsolutions.blogspot.com/`

Jeff Cope – API at `http://extensiblecad.com/words/`

Lenny Kikstra — API at `http://designsmarter.typepad.com/lennyworks/`

Josh Mings at `www.solidsmack.com`

Matt Lorono at `www.fcsuper.com/swblog/`

Many other blogs exist, and you should be able to find most of them from the links on my blog and the others listed here. These blogs do not all fit the same mold. Some are highly optimistic; others focus on tech gadgets, social networking, tech tips, or CAD news; some simply parrot news releases; and so on. For the best list of other CAD, design, 3D, rendering, and engineering blogs, view the blog roll in the right column of my blog.

Forums

Some forums are commercial, which means they are likely to contain advertising. These forums include:

- `https://forums.solidworks.com.` This is SolidWorks' site with official forums. The forums are very active in a wide range of topics and lightly moderated. This requires a login, but do not require current SolidWorks subscription.

- `www.mcadforums.com.` This site has a lot of traffic and content, but also uses Flash advertisements.

- `www.eng-tips.com.` This forum has plenty of traffic, but has an irritating sign-in popup and is highly censored. My posts in response to a question about how to become a consultant were removed.

- `www.3dcadtips.com.` This site has new owners, and in general has a lot of information on general engineering topics as well as CAD.

Non-Commercial Web Sites

The sites listed here are run by individuals, lack advertising, and contain information created by the owner of the site. They are my favorite types of places to find information. Even though most of these folks are not professional HTML coders, the information is useful and presented well.

- **Rob Rodriguez** (`www.robrodriguez.com`) features rendering topics. Rob used to host the PhotoWorks rendering contest, but that has moved to `http://rendercontest.com/`.

- **Paul Salvador** (`www.zxys.com`) has some nice models and images.

- **Stefan Berlitz** (`http://solidworks.cad.de`) is the unofficial German SolidWorks site. If you read German, this site is loaded with great information. Although it does contain some advertisements, I still consider it a non-commercial site.

- **Scott Baugh** (`www.scottjbaugh.com`) has several sample models and a section on tips.

- **Mike Wilson** (`www.mikejwilson.com`) has some amazing things posted to his site that he has done with SolidWorks. They are great models from which to learn. The site has not been updated for some time, but the models and techniques shown are fascinating.

What's on the CD-ROM

The only file you see on this book's companion CD-ROM is a single Zip file. Extract the contents of the file to your hard drive in a location that is easy to access. The CD contains example and tutorial parts, assemblies, and drawings, as well as templates, macros, and tables as appropriate for each chapter. The files are organized within folders for each chapter and are named for the chapter and the function they demonstrate. Some of the files are starting points for tutorials, and some are finished models meant to be examined.

If you make changes to files, I recommend that you use the Save As command (File ➪ Save As) to keep the original file intact. You also can retrieve originals from the CD again if needed.

Caution
I do not recommend that you open files directly from the CD or from the Zip file, because SolidWorks will respond with messages about read-only files. ■

System Requirements

Make sure that your computer meets the minimum system requirements listed in this section. If your computer doesn't match up to these requirements, you may have a problem using the contents of the CD.

Windows versions

For Windows XP Professional:

- Intel and AMD processors, single dual or quad cores.
- 1GB RAM minimum (2GB recommended).
- Virtual memory twice the amount of RAM (recommended).
- A certified OpenGL workstation graphics card and driver. (Check the SolidWorks Web site for details; `www.solidworks.com`.)
- A mouse or other pointing device.
- Microsoft Internet Explorer 6 minimum (IE 7 recommended).
- A CD drive minimum (DVD drive recommended).

For Vista/Windows 7:

- 1GB RAM minimum (2GB recommended).
- Other minimum requirements are the same as for Windows XP Professional.

For more details about the system requirements for SolidWorks 2010 and a list of certified graphics cards and drivers, visit `www.solidworks.com`.

Some older systems that run Windows XP Professional may not be compatible with Vista. Any hardware that runs Vista and most hardware that runs Windows XP should be able to work with Windows 7. For the latest information on system compatibility with Microsoft operating systems, visit `www.microsoft.com`. At this time, SolidWorks does not run on any non-Windows operating systems such as OS X, Linux, or Google Chrome OS.

Realistically, you will never be satisfied with minimum requirements. If you are using PhotoWorks, PhotoView 360, or any simulation (Finite Element Analysis, or FEA) software, multiple processors or multiple cores are advantageous. Multi-body modeling makes use of multiple cores but also takes advantage of higher processor clock speeds. Maximum clock speeds are usually higher for a lower number of cores, so higher speeds take precedence over number of cores for general solid modeling. You may get better performance per dollar with dual core processors than with quad core processors for functionality other than rendering and FEA.

You can only take advantage of more RAM up to the limit needed by your data sets. You can check your Windows Task Manager to see how much memory your largest or most complex models consume. For example, if your largest models use 3GB of RAM, you should have at least 4GB of RAM, but will probably not see a benefit from 16GB. You should use 64-bit operating systems if you intend to use more than 3GB of RAM.

All hardware produced in the last several years is 64-bit capable, but in order to take advantage of it, you have to have a 64-bit operating system installed. XP, Vista, and Windows 7 all have 64-bit versions, but you need to have installed the 64-bit version to take advantage of the huge memory advantages of 64 bit. 32-bit systems can handle a little over 3GB of RAM. To handle more than that you need a 64-bit operating system.

Some software has 32- and 64-bit versions, and some does not. SolidWorks has both versions, and you need to make sure you have disks or downloaded data for the correct installation type. Data files are interchangeable between 32 and 64 bit, so the files on the CD will work with either.

SolidWorks versions

As of SolidWorks 2010, SolidWorks files from the 2010 version are not compatible with older versions. So, if you have a version of SolidWorks older than 2010, you will have difficulty reading most of the files on the accompanying CD. You may find some files that came from older versions on the CD, but this only happens where the text has not been updated for new versions. If you have old version software, the only thing you can do to get files you can read is to contact the author or Wiley technical support to access files in SolidWorks 2007 format, which was the earliest version of the software for which this book was written. You cannot get (because they do not exist), and the author will not create files in earlier versions.

The author is not responsible for Dassault Systemmes SolidWorks Corporation's choices in this matter of policy, and there is nothing the author can do to circumvent it. Your best bet if you have difficulties with this policy of version non-compatibility (sometimes referred to as backward compatibility) is to contact SolidWorks directly and voice your concerns.

This book was written using the SolidWorks 2010 version software, and while some of it may be applicable to previous versions, some of it may not due to annual changes that happen in the course of software development. Earlier versions of the book do exist (2009 and 2007) and are still available, which may benefit you if you have SolidWorks software in one of those versions.

Troubleshooting

If you have difficulty installing or using any of the material on the companion CD, try the following solutions:

- **Turn off any anti-virus software that you may have running.** Installers sometimes mimic virus activity and can make your computer incorrectly believe that it is being infected by a virus. (Be sure to turn the anti-virus software back on later.)

- **Close all running programs.** The more programs you're running, the less memory is available to other programs. Installers also typically update files and programs; if you keep other programs running, installation may not work properly.

- **Contact the author.** For problems with the content of the CD, visit the author's Web site (`www.dezignstuff.com`) or blog (`www.dezignstuff.com/blog`), or send an e-mail (matt@dezignstuff.com).

- **See the ReadMe file.** Please refer to the ReadMe file located at the root of the CD for the latest product information at the time of publication.

Customer Care

If you still have trouble with the CD, please call the Wiley Product Technical Support telephone number (800) 762-2974. Outside the United States, call 1 (317) 572-3994. You can also contact Wiley Product Technical Support at `http://support.wiley.com`. John Wiley & Sons will provide technical support only for installation and other general quality control items. For technical support on the applications themselves, consult the program's vendor or author.

To place orders or to request information about other Wiley products, please call (800) 225-5945.

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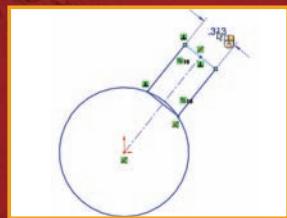
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The detail and depth you need to get the most out of SolidWorks

Whether you're new to SolidWorks or want to expand your current skills, this is the SolidWorks book you need. Offering in-depth content that exceeds most other publications on the subject, this book is a complete desk reference. Well-known SolidWorks consultant Matt Lombard covers all features, explains best practices, and provides pages of step-by-step tutorials and advanced techniques that will soon have you successfully using this powerful software for your own CAD projects.

- Understand feature- and history-based modeling, and sketching with parametrics
- Create sketches, parts, assemblies, and drawings
- Master the user interface, including menus, toolbars, macros, and more
- Work with mold tools, plastic and sheet metal parts, and weldments
- Learn advanced techniques for multibody modeling
- Build animations with SolidWorks MotionManager
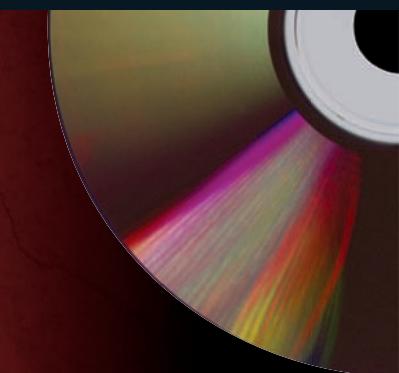
Use dimensions and sketch relations to modify sketches

Employ display techniques to grasp complex geometry

Master assembly tools to create better mechanisms

What's on the CD-ROM?

The CD includes many examples files of drawings, parts, and assemblies. You'll also find part templates and before-and-after files that follow the tutorials in the book.

System Requirements: See the CD Appendix in the book for details and complete system requirements.

Matt Lombard

is an independent mechanical design consultant. Because of the popularity of his SolidWorks blog dezignstuff and his sessions at SolidWorks World, Matt has built a reputation as the go-to resource for everything SolidWorks.

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