Rhinoceros®

Advanced Training Series

Modeling and Mold Making for Jewelry Designers

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Introduction

The purpose of this tutorial is to make a two-part mold that will let you produce a ring, or actually, half of the ring. Wax will be poured into the mold and the finished pieces will be used in lost wax technique production.

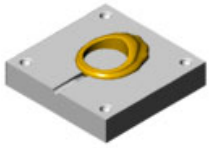
Rhinoceros and **VisualMill**, the latest advanced computer modeling and manufacturing programs, make it easy to:

- revise and change designs
- create a line of pieces based on one model
- reproduce fine detail
- create a library of reusable parts

This tutorial covers the basic steps for creating and milling a mold for prototyping or manufacturing.

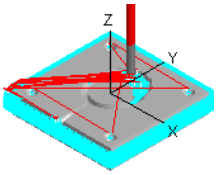
Model the part

- Create a model of a ring shape in Rhino.



Model the mold

- Use the model of the ring shape to create the mold.



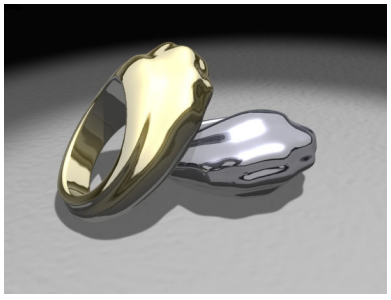
Create cutting instructions

- Import the model of the mold into VisualMill to create cutting instructions for a milling machine.
- Send the instructions to the machine to cut the part. This process varies according to the machine and is not covered in this tutorial.

To use the tutorial

1. If you do not have the **VisualMill** program installed, install the Evaluation version from the tutorial CD.
2. Work through the tutorial steps.

   Start by designing the curves that are needed to build the surfaces of the ring you are going to produce, and then you will work through the steps required for the construction of the two-part mold.



Product links

Rhinoceros
Start with sketches, drawings, physical models, or only an idea—Rhino provides the tools to accurately model your designs ready for rendering, animation, drafting, engineering, analysis, and manufacturing.
www.rhino3d.com

VisualMill
VisualMill is a 3-axis solid/surface milling package, which seamlessly integrates fast, gouge-free tool path generation with cut material simulation in an easy and fun to use package running on standard Windows hardware. **VisualMill** can drive a machine such as a CNC router or milling machine such as those manufactured by **Techno-isel** or **Roland Digital Group of America**. www.mecsoft.com

Roland Digital Group of America
Affordable desktop CNC scanning and milling machines such as the CAMM3, PNC 2500, and the new MDX 500 for use in the jewelry and educational markets.
www.rolanddga.com

Techno-isel
CNC routers for the sign making, woodworking, prototyping and educational markets. www.techno-isel.com

Start modeling with Rhino

You can start with a blank file and create the model yourself or you can open the model provided and follow along with the parts that are already there.

To start a new model

If you want to actually create this model yourself, you can start a blank Rhino file and draw all the parts as directed in this tutorial.

1. From the **File** menu, click **New** *(File menu: New)*.
2. In the **Template File** dialog box, select **Centimeters.3dm** and click **OK**.

To open the sample model

If you want to simply open the model provided, you can turn various layers on and off to see how the curves and surfaces were created.

1. From the **File** menu, click **Open** *(File menu: Open)*.
2. In the **Open** dialog box, select **Competed Mold.3dm**, and click **OK**.

   The initial curves are displayed on layers **01 Curves 1, 2, 3**, **02 Curves 4, 5**, and **03 Curve 6**.

   In addition, the Properties command for each curve lists the curve name, so you can follow along with the tutorial instructions.

To set the grid snap spacing

1. Open the **Document Properties** dialog box *(File menu: Properties)*.
2. On the **Grid** page, set the **Grid Snap Spacing** to **0.5** and click **OK**.
3. On the **Status bar**, click **Snap** to turn snap on.

Draw the first curves

While constructing the curves, it is very important to check their symmetry and appearance as you go along. Spend some time changing them or shaping them. The shape of the surfaces depends on the construction of the original curves.

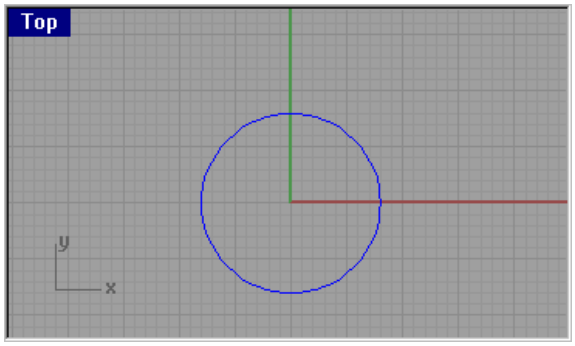
There is never one single strategy in building a surface. You can use your imagination to create the shapes you want, but keep the limitations of the manufacturing processes you are going to use in mind.

To draw curve 1

Curve 1 is the center circle of the ring.

1. Start the **Circle** command *(Curve menu: Circle > Center, Radius)*.
2. At the **Center of circle ...** prompt, type **0,0,0**.

   This sets the center of the circle at the origin point in space.
3. At the **Radius ...** prompt, type **8** or drag the radius 8 units.



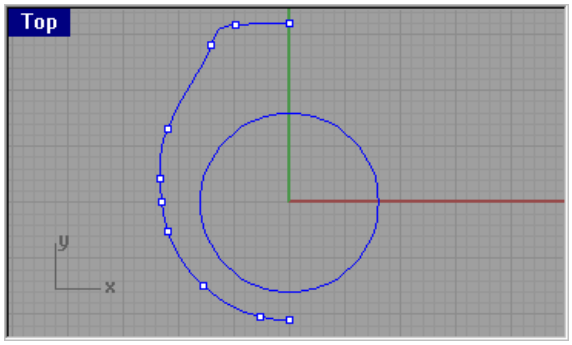
Curve 1 is a circle.

To draw curve 2

Draw the left half of the outline of the ring.

1. Start the **InterpCrv** command *(Curve menu: Free-form > Interpolate Points)*.
2. At the **Start of curve ...** prompt, start the curve on a snap point on the centerline (y-axis).
3. At the **Next point of curve ...** prompt, draw another point to define your curve

4 At the **Next point of curve ...** prompts, continue to draw points, ending on a grid snap point on the center line.



Draw curve 2.

Edit the second curve

You are going to make a mirrored copy of curve 2 to create the other half of the outside ring profile. This brings up an important point about the point where the two mirrored curves will meet. There should be no dip or point when the curves meet. To accomplish this requires just a little point editing on the curve.

To edit the control points for curve 2

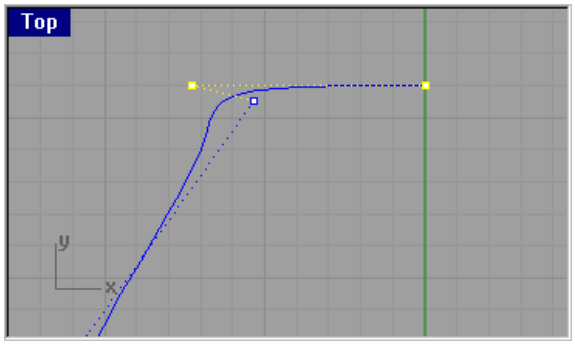
1 Turn on the **control points** for curve 2 *(Edit menu: Control Points > Control Points On (F10))*.

2 Look at each end of the curve.

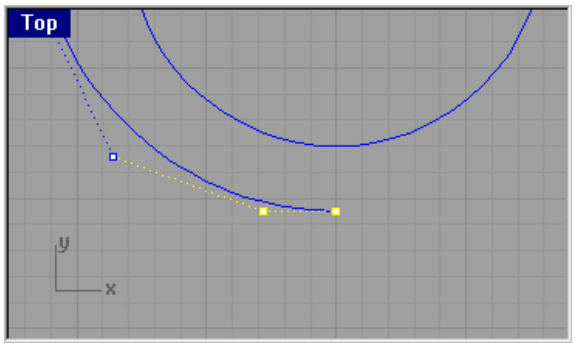
The important factor is that the control point at the end of the curve and the next control point are perfectly aligned horizontally.

The important factor is that the control point at the end of the curve and the next control point are perfectly aligned horizontally. If they are not, move the second control point using grid snap, a construction line, Ortho, or the **SetPt** command *(Transform menu: Set Points)*. This will ensure that after it is mirrored, the transition between the two halves of the mirrored curve will be tangent and horizontal.

For more information about this, see the Rhino Help file topic on *continuity*.



Line up the two control points at the top end of the curve.



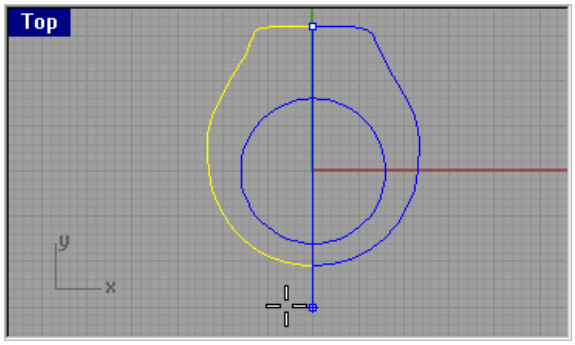
Line up the two control points at the lower end of the curve.

To mirror the curve

- **Mirror** the curve across the y-axis *(Transform menu: Mirror)*.

  Use **End** object snap and **Ortho** so the ends of the curve touch each other exactly.

  This creates curve 3.

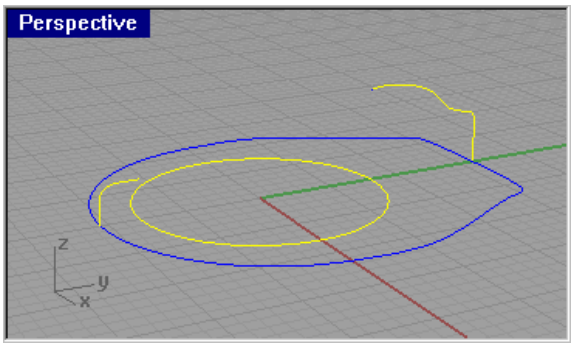


Mirror the curve in the Top viewport.

Define the side profile

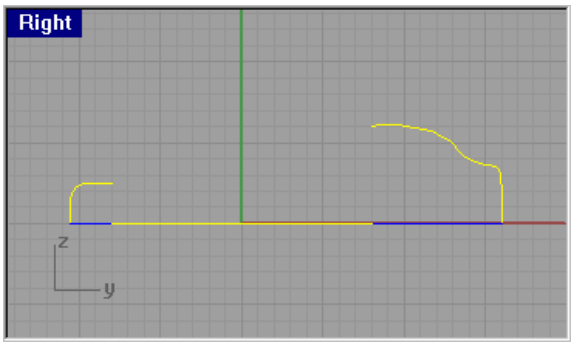
Design curves 4 and 5. These curves define the shape of the ring seen from the right side.

It is important to carefully place the start and end points of both curves with respect to the curves you have drawn so far. In the **Perspective** viewport you can see that the curves start at the endpoint of the mirrored curves.



Curves 4 and 5 in the Perspective viewport.

In the **Right** viewport, you can see that they end directly over the inner circle.



Curves 4 and 5 in the Right viewport, showing the relationship to the center circle.

To draw curves 4 and 5

1. Select the inner circle so you can see it highlighted and use it as a guide.
2. Start the **InterpCrv** command *(Curve menu: Free-form > Interpolate Points)*.

   Start drawing the curves in the **Perspective** or **Top** viewport.

   Use grid snap for the start of the curve to accurately place the beginning and end points.

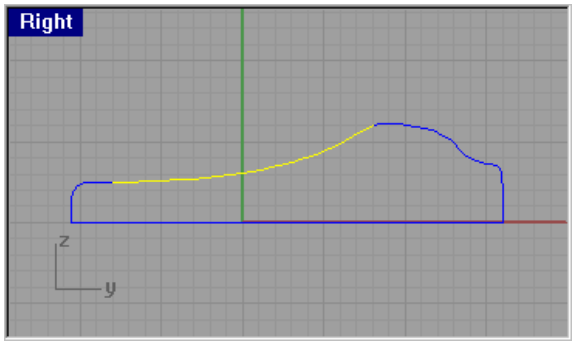
   To design your curve more freely, you can turn off grid snap for the intermediate points.
3. Continue drawing the curve in the **Right** viewport.
4. Use grid snap or point filters (.x, .y, .z) to line the end of the curve up with the circle.

Connect the curves

To complete the side profile of the ring, draw a curve to connect curves 5 and 6 then mirror the side profile curves to examine the whole shape.

To complete the side profile of the ring:

1. Use the **InterpCrv** command to draw curve 6 to connect curves 4 and 5 *(Curve menu: Free-form > Interpolate Points)*.

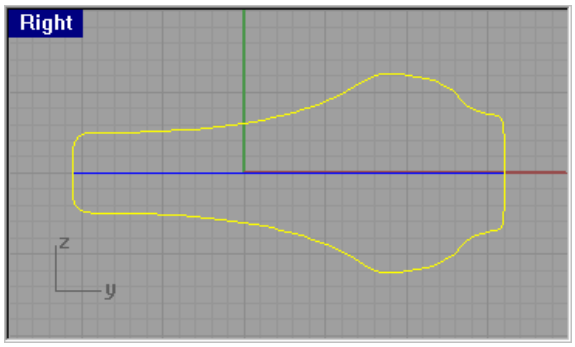
   Curve 6 connects curves 4 and 5.

You have defined the complete half of the outline of the ring viewed from the right side and from the top.

At this stage it is a good idea to check all the proportions and to see if some adjustment might be needed to give the ring a better shape.

2 For a better idea of how your design is developing, **Mirror** curves 4, 5, and 6 across the world y-axis in the right viewport so that you can see the complete ring profile as seen from the right side *(Transform menu: Mirror)*.

You will not be using this mirrored curve for constructing the model, so you can delete it when you are satisfied that the shape is what you want.



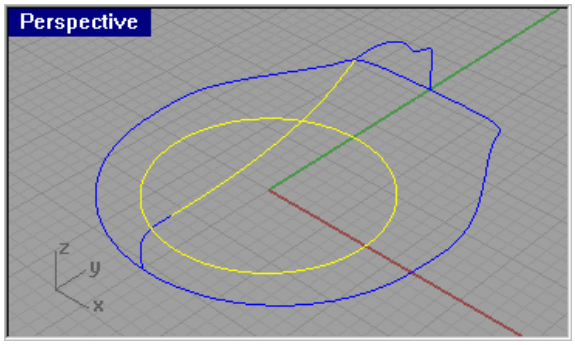
Mirror the curves to study the design of the entire profile.

Create the inner edge

You are now going to create a curve that defines the inner edge of the ring. This curve is shaped the way curve 6 is in the right view and the way curve 1 is in the top view. It is a sort of bent circle. Rhino has a command that creates this curve based on these curves.

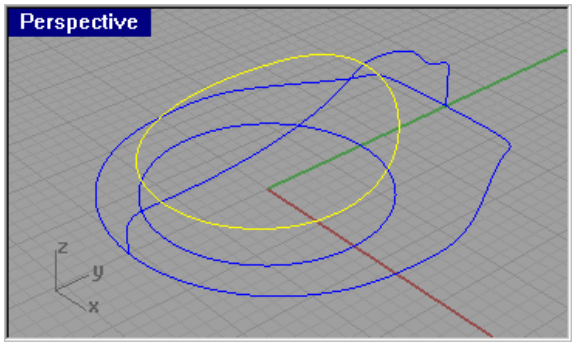
To create curve 7 from curves 1 and 6

1 Select curves 1 and 6.



Curves 1 and 6 are used to create curve 7.

2 Use the **Crv2View** command to create curve 7 *(Curve menu: From 2 Views)*.



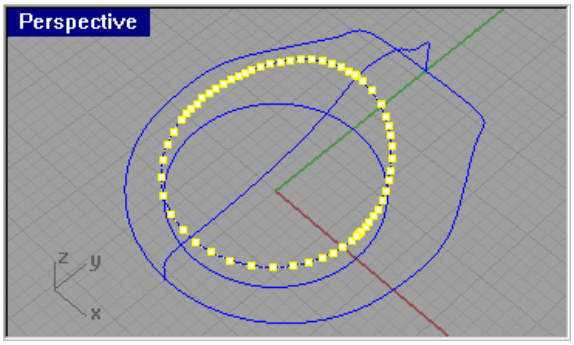
Curve 7, the top edge of the ring.

3 If curve 7 is not a single object, use the **Join** command to join the parts of the curve into one *(Edit menu: Join)*.

Refine the curve

There may be too many control points in this curve. Curves with very high number of control point create dense surfaces. It is better to use only as many control points as are necessary to maintain the shape you want.

The next step will be to reduce the number of control points in the curve.



Curve 7 has too many control points.

To rebuild the curve

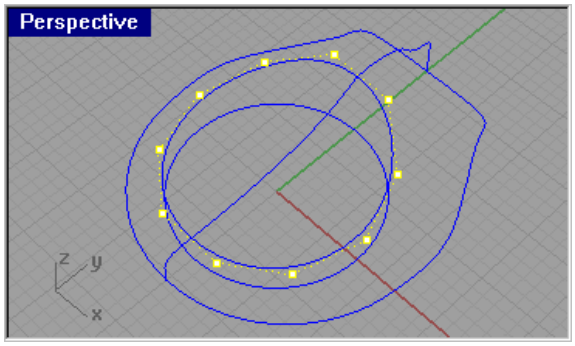
1 Turn on the **control points** on curve 7 *(Edit menu: Point Editing > Control Points On)*.

2 Press **Esc** to turn off the control points.

3 Use the **Rebuild** command to set the number of control points to 10 *(Edit menu: Rebuild)*.

Preview the curve to check whether the rebuilt curve is going to be too different from your original curve.

Use the **Delete input** option to remove the original curve.



Rebuild the curve with 10 control points.

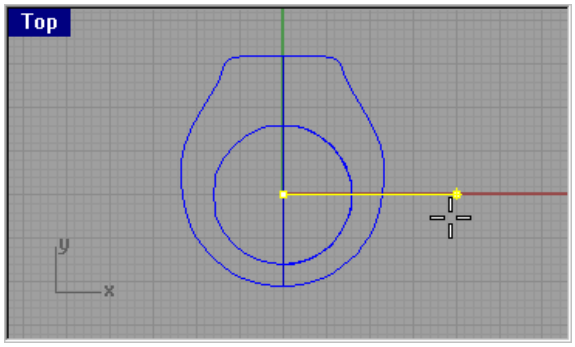
Draw the side profiles

Design the curves that define the profile of the ring when viewed in the **Front** viewport.

It will be useful to draw the line (curve 8) that will be used for finding the point where a line drawn from the center of the original circle intersects curve 2. This is the point where the side profile curve starts.

To draw the side profile curves

1. Use the **Line** command with **Snap** or **Ortho** to draw a horizontal line from the center of the original circle to out beyond curve 2 as shown *(Curve menu: Line > Single Line)*.

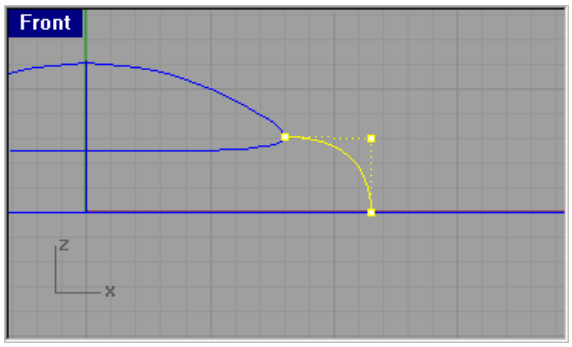


Construction line for drawing the side profile.

2. Use the **Curve** command and the **Intersection** object snap to draw a curve that starts at the intersection of curve 2 and your construction line *(Curve menu: Free-form > Control Points)*.

   Use the **Front** viewport for drawing the curve. It should end touching curve 7.

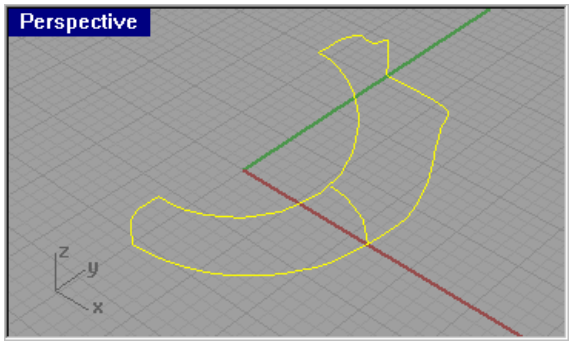
   Again, the control points should line up. In the example curve, there are only three control points, which form a right angle.



Draw the side profile curve.

3 **Trim** away the left half of the curve that forms the upper edge of the ring at the side profile curves *(Edit menu: Trim)*.

You now have all the curves you need for creating this model.



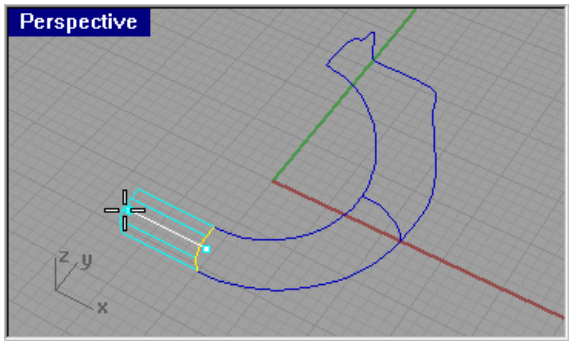
Use these curves to create the ring surface.

Create the first surface

You are about to build your first surface from the curves. You will be using only the curves on the right half of the model.

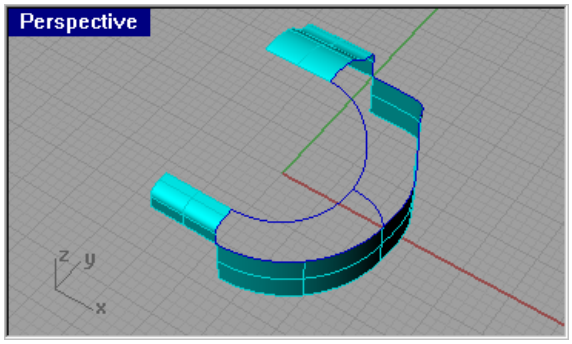
To create the first surface

1 **Extrude** with the **Straight** option the profile curve a short distance to the left (negative x-direction) as shown *(Surface menu: Extrude > Straight)*.



Extrude the profile curves at the bottom.

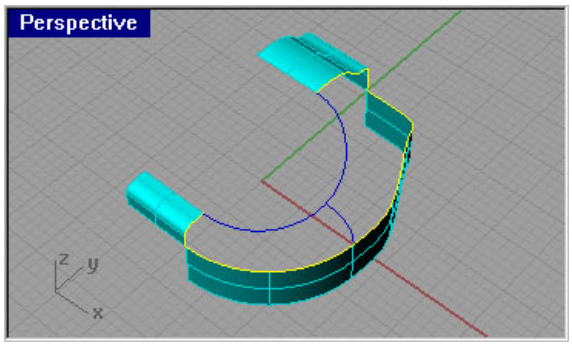
2 **Extrude** the other profile curve and the outline curve as shown *(Surface menu: Extrude > Straight)*.



Extrude the profile curve at the top and the outline curve.

This creates a series of construction surfaces. The **NetworkSrf** command can use these surfaces to establish a direction for the surface.

3 **Hide** the curves so they do not interfere with selecting the surface edges *(Edit menu: Visibility > Hide)*.

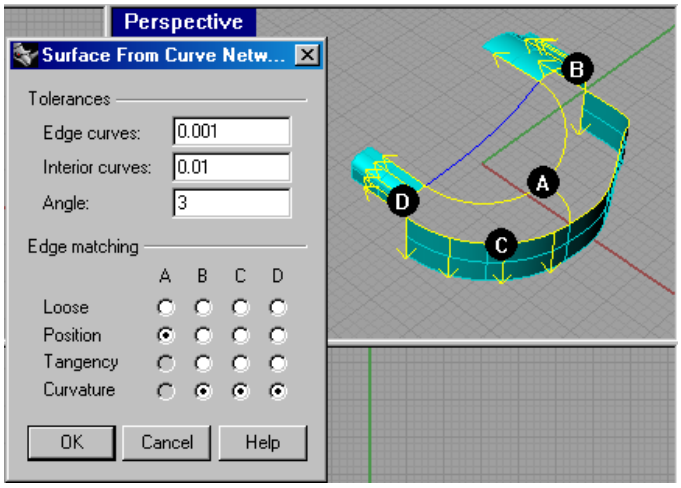


Hide the selected curves you used to extrude the construction surfaces.

4 Start the **NetworkSrf** command *(Surface menu: From Curve Network)*.

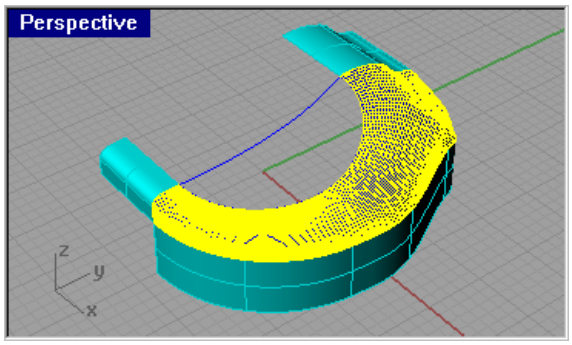
Select the edges of the dummy surfaces, the top curve, and the interior profile curve as shown.

Set the **Edge matching** to **Curvature** for the surface edges (B, C, and D in the illustration) and **Position** for the open curve at the top edge (A in the illustration).



Create a surface from curve network.

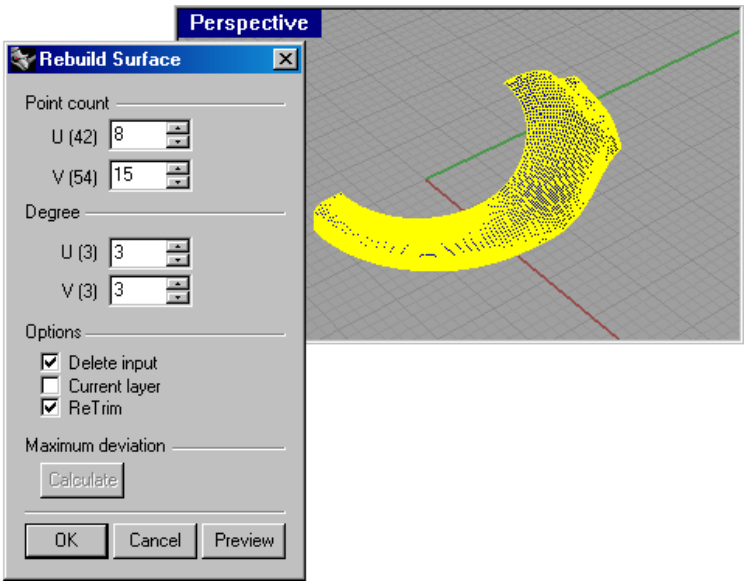
The surface may take a few minutes to generate.



The surface from curve network.

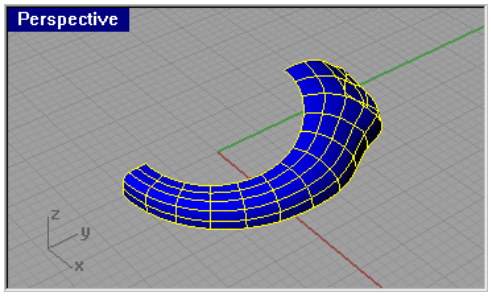
5 **Rebuild** the surface with 8 points in the u-direction and 15 in the v-direction. *(Edit menu: Rebuild)*

The u and v-directions may be different for your surface. The direction depends on the order you selected the curves for the **NetworkSrf** command.

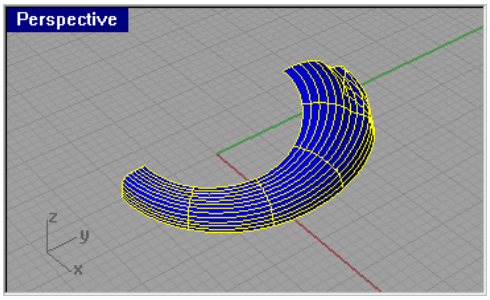


Rebuild the surface.

Be sure your picture looks like the image on the left, not the image on the right below.

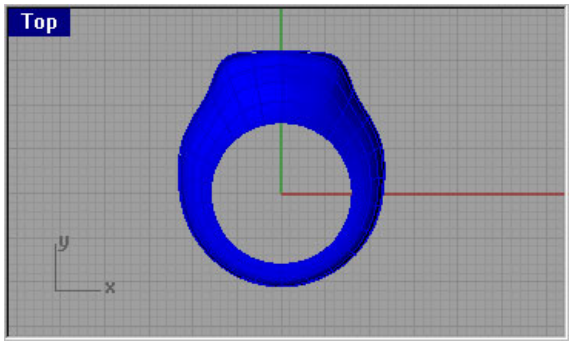


The surface should look like this.



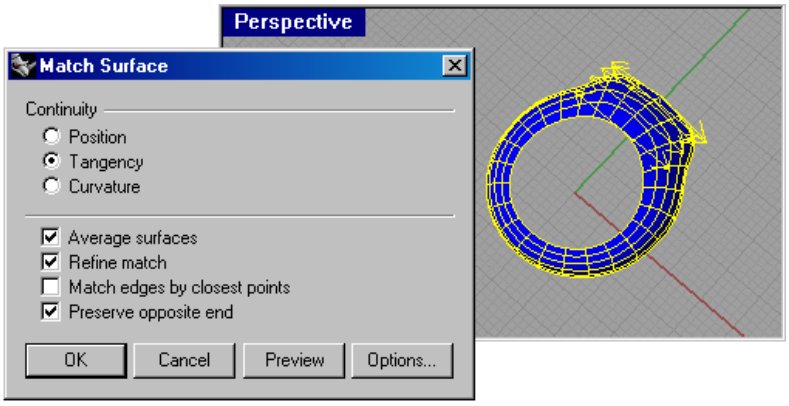
***Not** like this.*

6 **Mirror** the ring half *(Transform menu: Mirror)*.



Mirror the rebuilt surface across the y-axis.

7 Use the **MatchSrf** command to match the two surface edges for tangency with the settings shown *(Surface menu: Surface Edit Tools > Match)*.



Match the two surfaces with tangent matching.

8 Use the **MergeSrf** command to turn the two surfaces into one *(Surface menu: Surface Edit Tools > Merge)*.

When selecting the surfaces, pick near the top of the ring in the **Front** viewport.

Reshape the surface

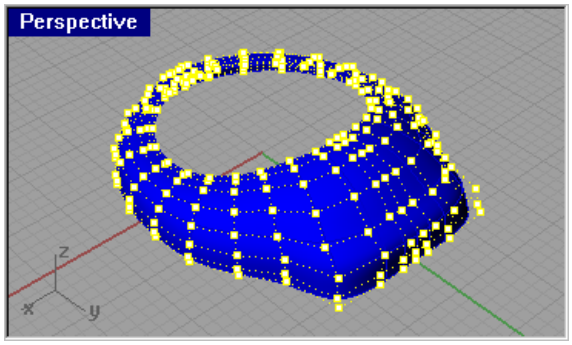
The surface is quite dense in spite of efforts at keeping the control point count low on the input curves.

To make the surface easier to work with, rebuild the surface so that it has fewer control points.

To reduce the control point count for the surface

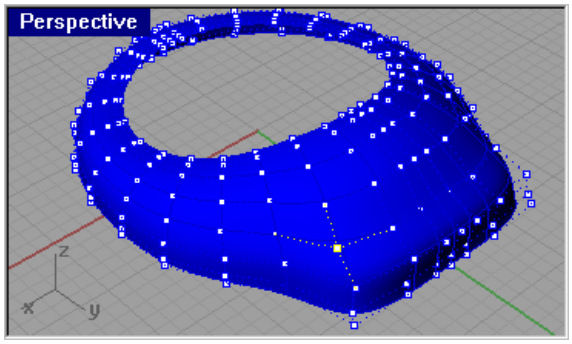
1 Turn on the control points of the surface *(Edit menu: Point Editing > Control Points On)*.

As you can see there are plenty to allow for modifications.



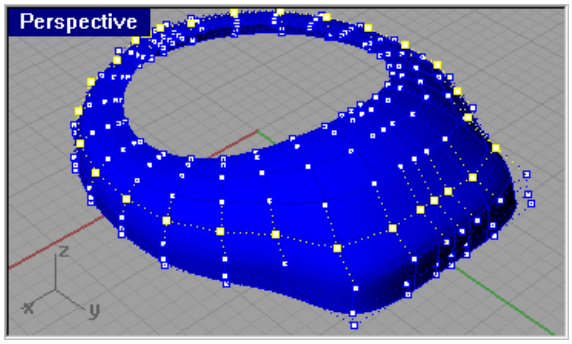
Control points on the surface.

2 Select one control point in the fourth row from the larger edge like the one shown below.



Select one point in a row of points.

3 Use the **SelV** command to select all the control points in the same v-direction row *(Edit menu: Control Points > Select Control Points > Select V)*.



Select the row of points with the SelV command.

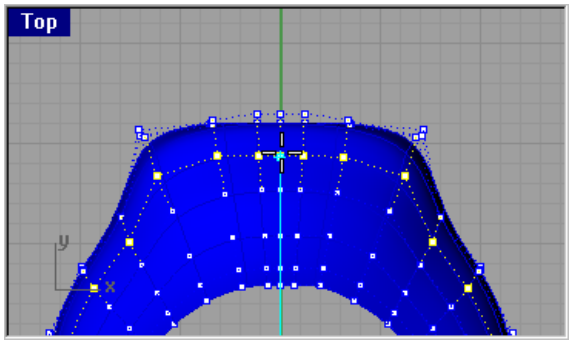
Note: The surface direction may be different from the example model. If this is the case, use the **SelU** command to select the row of points so it matches the image above.

4 Use the **Scale2D** command to move all the control points in this row toward the center of the ring *(Transform menu: Scale > Scale 2-D)*.

5 For the origin point, type **0,0,0**.

This is the center of the inner circle if you have not moved your model.

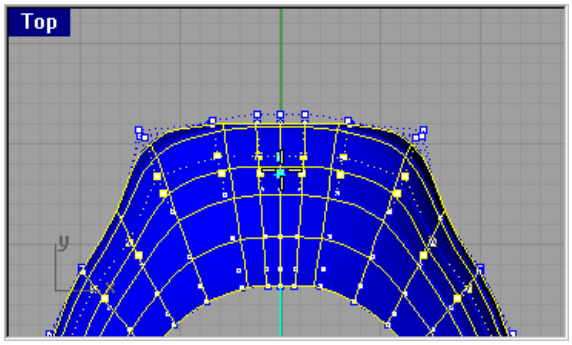
6 For the first reference point, in the **Top** viewport, drag the line out from the origin to near the row of points.



Scaling brings all the points toward the center.

7 For the second reference point, drag the line down toward the center of the circle until the control points in your selected row are near the next inner row.

Watch the isocurves. Make sure the isocurve associated with the row of points you are scaling does not cross over the next row toward the center. This will cause an undercut in the ring surface.



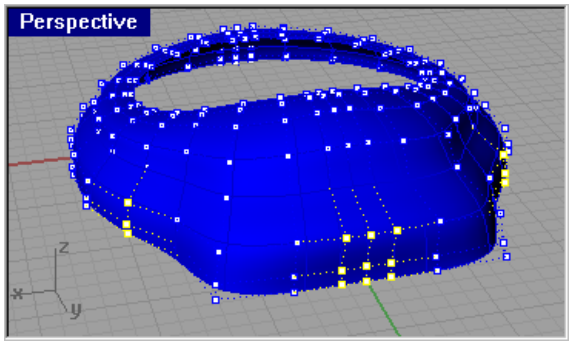
Use Scale2D to move the row of points in.

Add interest

The free-form modification of the surface is not over yet. You need to work on it a little bit more to make it more interesting. In the next step you are going to edit groups of control points to add some bulginess to the surface.

To create bulges in the surface

1 Select the three groups of control points shown below: the three rows at the top center and matching groups on each side.

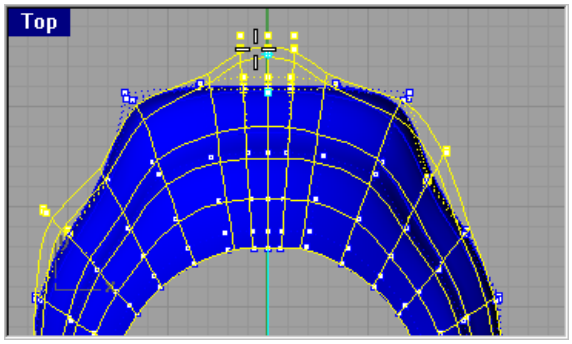


Select rows of control points.

2 Use the **Scale2D** command to move all the control points in this row away from the center of the ring *(Transform menu: Scale > Scale 2-D)*.

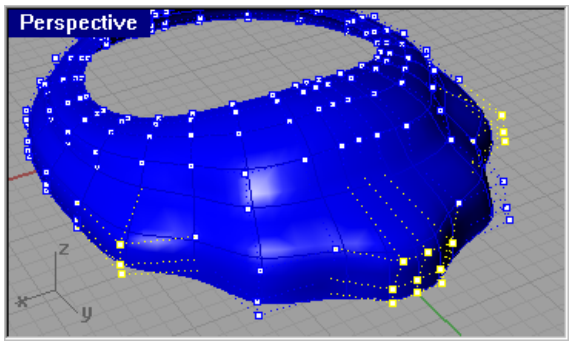
3 For the origin point, type **0,0,0**.

4 For the first reference point, in the **Top** viewport, drag the line out from the origin to the row of points.



Use grid snap to maintain control over the location of your reference points.

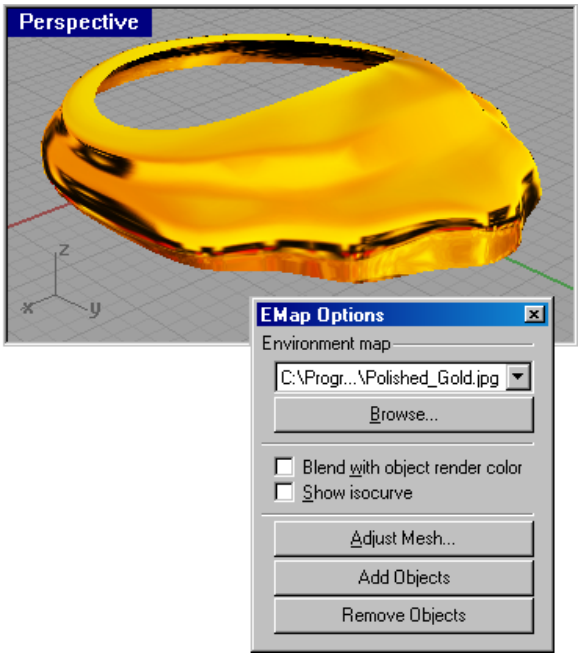
5 For the second reference point, drag the line up to the grid snap point two units away from the original point.



Scale the control points out from the center.

This creates three bulges in the ring surface.

6 Use the **EMap** command to examine the ring with a gold texture map *(Analyze menu: Surface > Environment Map)*.



The EMap command displays the ring with a gold texture.

You can see that the overall shape of the ring, especially from the top looking at the border, is changing considerably. Working with control points might be sometimes a bit time consuming, but it is a method for manipulating surfaces that is worth trying.

Feel free to try any solution you may find interesting or useful to modify the control points. This is just one of many possibilities.

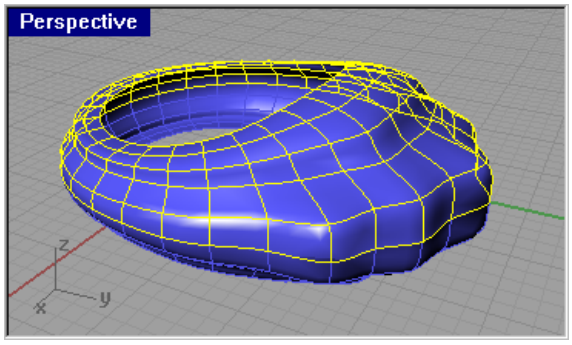
Mirror the surface

Mirror the ring half, match and merge the two ring halves into one surface.

To create the other half of the ring

1 Use the **Mirror** command to create the other ring half *(Transform menu: Mirror)*.

Use the **Front** or **Right** viewport.



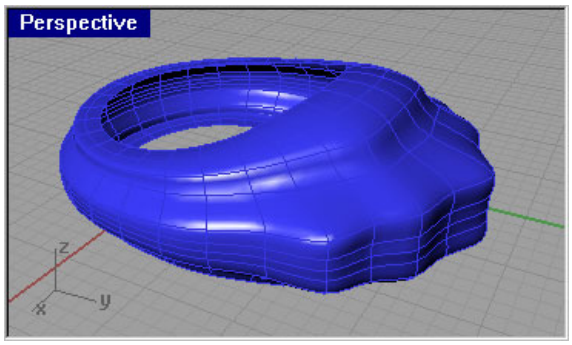
Mirror the ring half.

2. Use the **MatchSrf** command to match the tangency of the surface edges *(Surface menu: Edit Tools > Match)*.

3. Use the **MergeSrf** command to make the two surfaces into one *(Surface menu: Edit Tools > Merge)*.

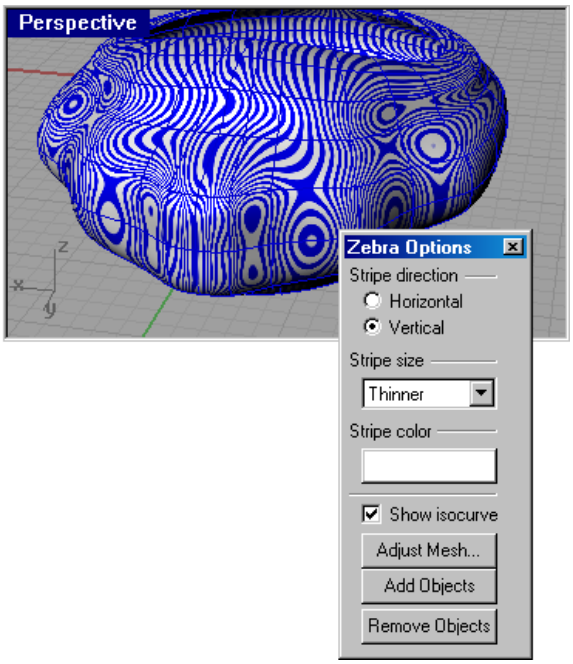
4. At the **Select untrimmed surface to merge ...** prompt, set the **Smooth** option to **No**.

   Since you already matched the surface edges so they are tangent, you do not need to smooth the surfaces together with the **MergeSrf** command.



The merged ring halves.

5. Check the ring with the **Zebra** command *(Analyze menu: Surface > Zebra)*.



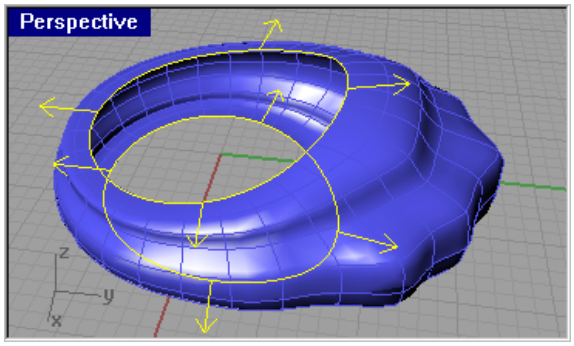
Check the surface with Zebra analysis.

Finish the interior

You now only have the interior surface of the ring to finish. Use the **Loft** command to complete this surface.

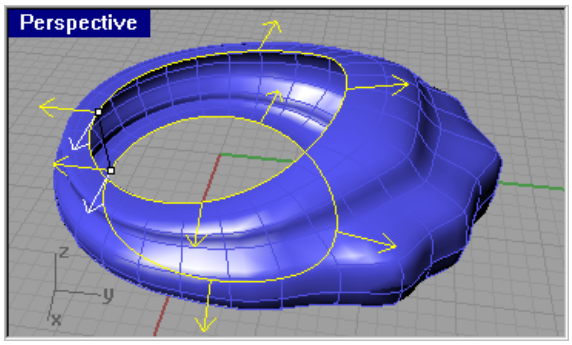
To create the interior surface

1. Start the **Loft** command and select the two edges of the surface as shown below *(Surface menu: Loft)*.



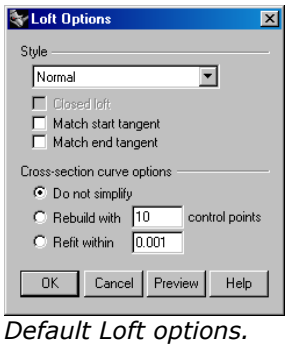
Loft the surface edge curves.

2. At the **Adjust curve seams ...** prompt, press **Enter** if the curve seams line up as they do in the image below.



Adjust curve seams.

3. Accept the default **Loft Options**.



Default Loft options.

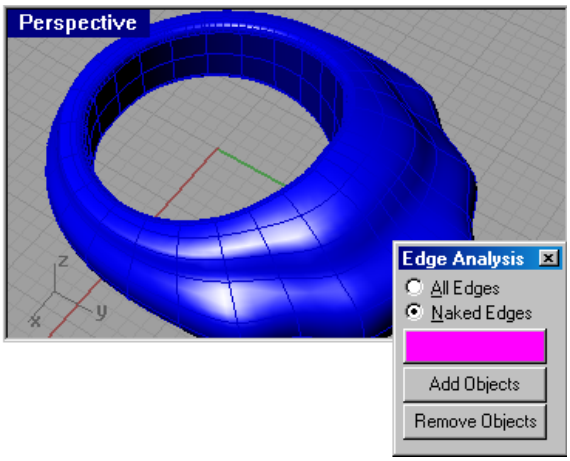
4. **Join** all the parts to create one object from the interior loft surface and the main ring surface *(Edit menu: Join)*.

5 As a clean-up of the finished product, use the **FilletEdge** command to create a fillet with a **0.5 mm** radius at both of the interior edges of the ring so that there is a smooth transition between the two surfaces *(Solid menu: Fillet Edge)*.

Check model for unjoined edges

- Use the **ShowEdges** command to check the joined ring model for unjoined (naked) edges *(Analyze menu: Edge Tools > Show Edges)*.

  Since the model will eventually be exported to a milling program, a closed model will be required. It is a good idea to check the model at intervals to catch modeling problems like this early.

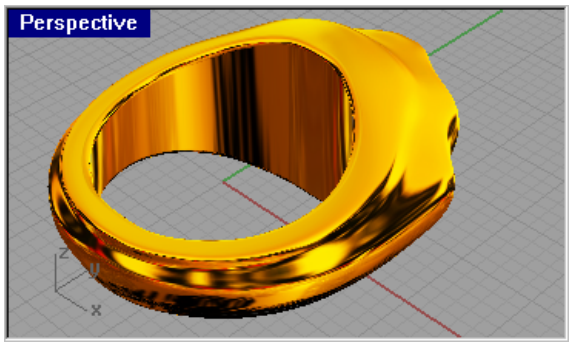


Check for unjoined edges.

Examine the finished ring with environment mapping

Look at the ring in its entirety using the **EMap** command.

1 With the **EMap** command, try **polished_gold.jpg** to represent the material to be simulated *(Analyze menu: Surface > Environment Map)*.



Examine the finished ring with environment mapping.

2 Use the **Turntable** command to continuously rotate the view of the ring.

Any modification to the finished piece should be done now, because the next steps concentrate on making the mold.

Save a copy

In the next sections you are going to cut up the ring to make mold parts, so make a copy of the completed ring surface and store it on a separate layer.

To prepare the ring model for making the mold model

1. Use the **Copy** command with the **InPlace** option to create a copy of the ring *(Edit menu: Copy)*.
2. Use the **ChangeLayer** command to place the ring copy on a new layer *(Edit menu: Layers > Change Object Layer)*.
3. Turn the new layer off.

Trim half the ring

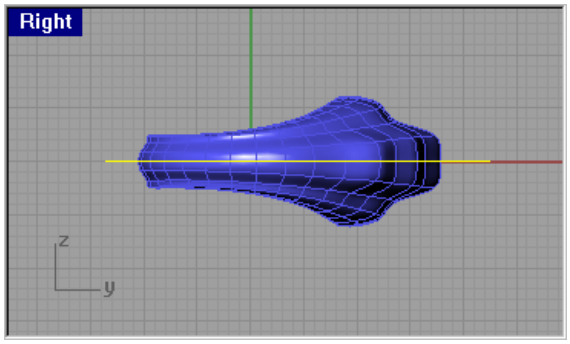
A surface is needed that will subtract an amount of volume from the thick part of the ring in order to make it hollow.

One of the strategies used to accomplish this task is to create section curves of the ring surface. Since the upper and the lower part of the ring are evidently symmetrical, only the lower half is needed.

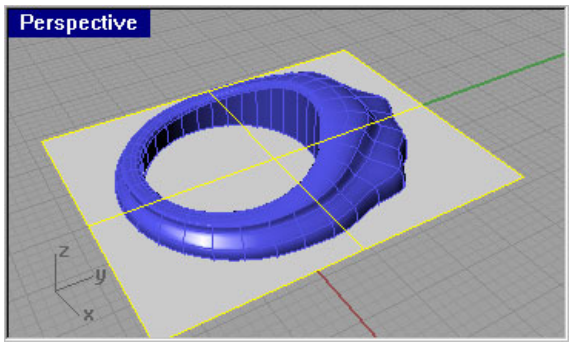
You created the whole ring so you could complete the design, but you only need half the ring to make a mold, so before you start, trim away the top half the ring.

To split the ring into two halves

1. In the **Right** viewport, use the **CutPlane** command to draw a cutting plane along the world y-axis that extends beyond the ring *(Surface menu: Plane > Cutting Plane)*.



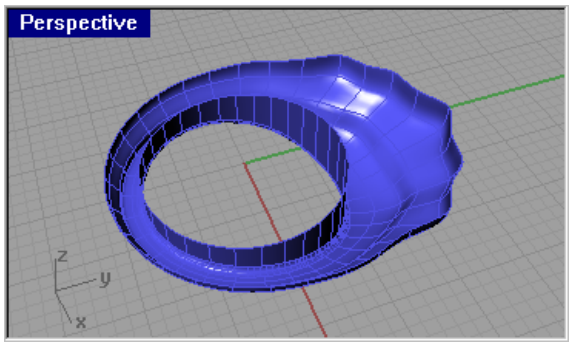
Create a cutting plane through the ring.



The cutting plane.

2 Use the **Trim** command to trim away the top half of the ring with the cutting plane *(Edit menu: Trim)*.

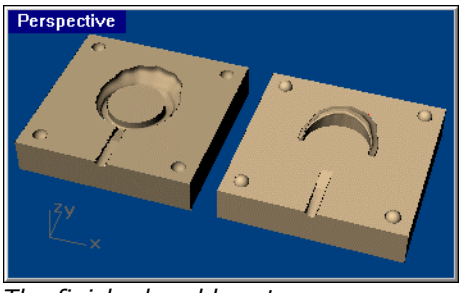
3 Delete the cutting plane surface.



The trimmed ring half.

Hollow the ring

Create a box from which to subtract a part, leaving a hollow space that will be exactly shaped like half the ring. If you were to use the entire ring as it is now, it would make a heavy, solid lump of metal that require a lot of time-consuming and expensive handwork to bring it to an acceptable weight.



The finished mold parts.

For the next part of this tutorial, you will create a hollow in the ring. You want to produce a mold that lets you control the thickness of the finished piece. To finish the piece, a goldsmith will weld the two parts and complete the finishing process.

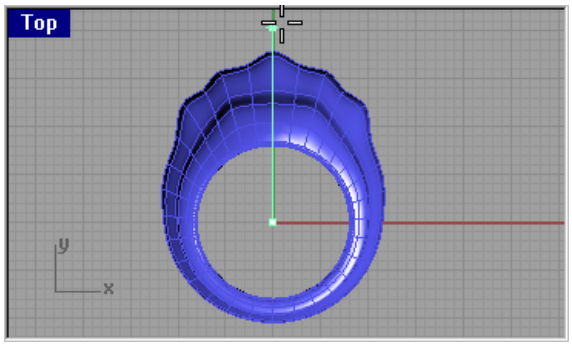
Create curves on the surface

First make curves that lie on the ring surface. Then offset these curves by 1 mm toward the inside. These inner curves will be used to create the hollowing surface.

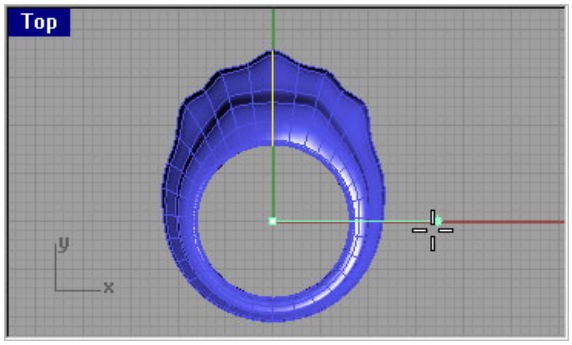
You might think to use the **OffsetSrf** command in this case, but it will create a surface that passes through itself. Unfortunately, the solution is not that easy. Offsetting surfaces that have complex curvatures may cause self-intersecting surfaces. This is unacceptable, so an alternative solution will have to be found an in order to build the surface. Offsetting curves does not create this problem, so you can offset curves based on the surface and use those curves to create the new interior surface.

To create section curves

1. Use the **Section** command to create curves that radiate out from the origin point *(Curve menu: From Objects > Section)*.
2. At the **Select objects for sections** prompt, select the ring half.
3. In the **Top** viewport at each **Start of section prompt**, type **0,0,0** and drag the section-defining line out beyond the ring as shown below.
4. Draw one section along the x-axis and one along the y-axis.

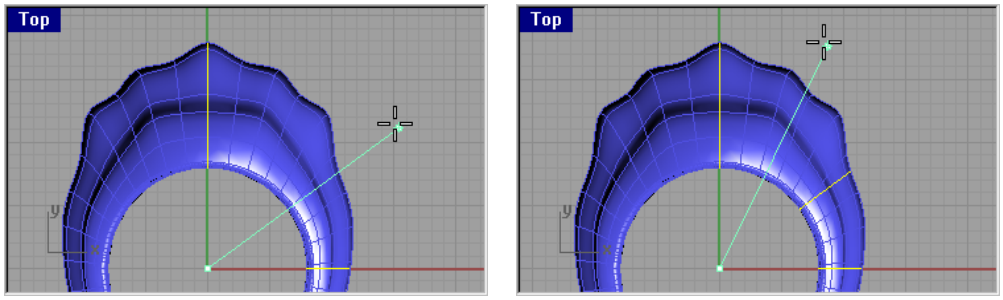


Draw a section line through the ring along the y-axis.



Draw a section line through the ring along the x-axis.

5 Place two more sections more or less evenly spaced between the x- and y-axes as shown highlighted below.



Draw two more section lines.

Offset the curves

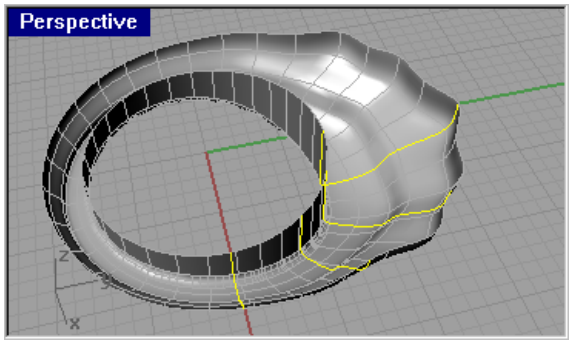
Offset these sections to obtain smaller curves and from these build a surface inside the ring.

The **Offset** command requires that the curve be parallel to the construction plane. Since the curves do not fit this requirement, set the construction plane to temporarily match a curve, offset the curve and then set the construction plane to match the next curve.

To offset the section curves

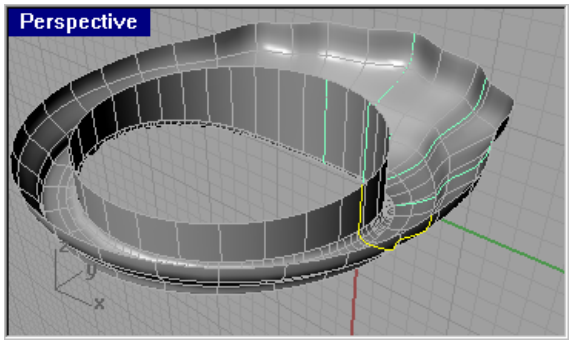
1 Before you start, use the **Lock** command on the ring surface so you can see it, but not select it *(Edit menu: Visibility > Lock)*.

This will help you keep track of where you are.



Lock the ring surface.

2 In the **Perspective** viewport, use the **CPlane** command with the **Object** option to align the construction plane with the first curve *(View menu: Set CPlane > To Object)*.

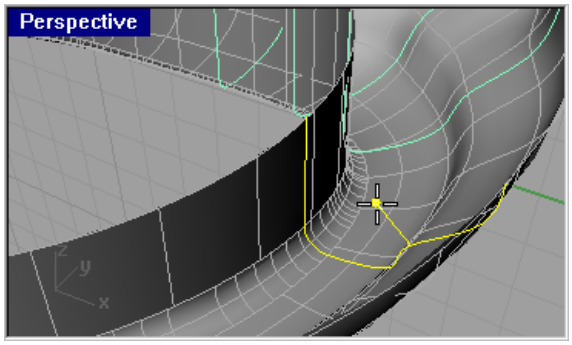


Set the construction plane to the section curve.

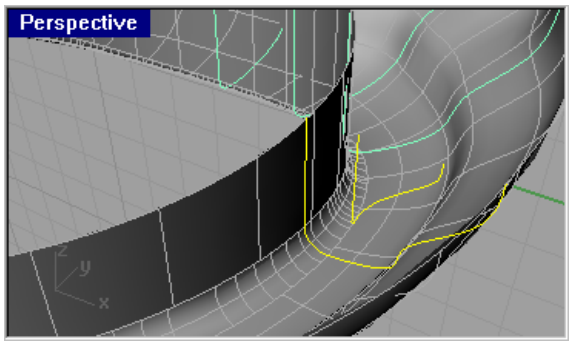
3 Use the **Offset** command to create the inside curve *(Curve menu: Offset Curve)*.

4 Set the distance to **1**.

5 At the **Side to offset ...** prompt, click to the inside of the ring.



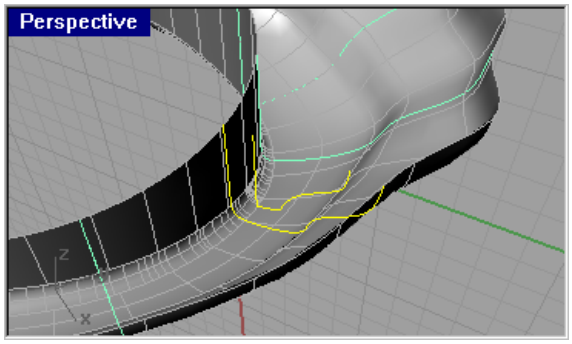
Offset the curve toward the interior of the ring.



The offset of the first section curve.

6 Use the **CPlane Object** command again to change the construction plane to the next section curve *(View menu: Set CPlane > To Object)*.

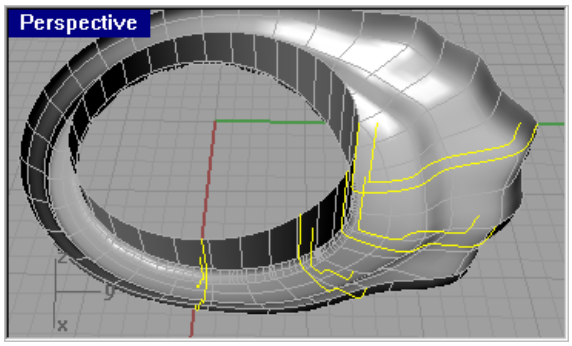
7 Use the **Offset** command to create the next inside curve *(Curve menu: Offset Curve)*.



Repeat the offset for each section curve.

8 Repeat steps 6 and 7 for the other two curves.

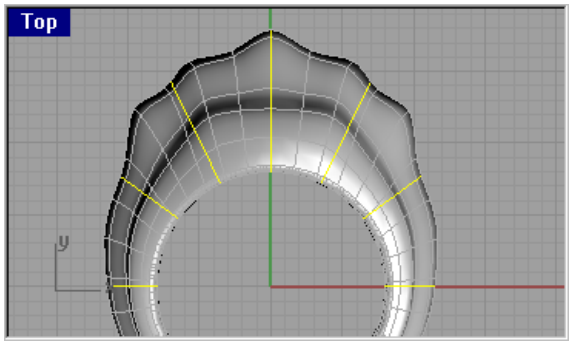
9 When you have offset all the curves, use the **CPlane** command **World Top** option to set the construction plane back to its normal position *(View menu: Set CPlane > World Top)*.



Reset the construction plane to World Top.

10 Use the **Mirror** command in the **Top** viewport to mirror the new offset curves to the other side of the ring *(Transform menu: Mirror)*.

Do not mirror the curve that lies on the y-axis.



Mirror the offset curves across the y-axis.

Create the hollow surface

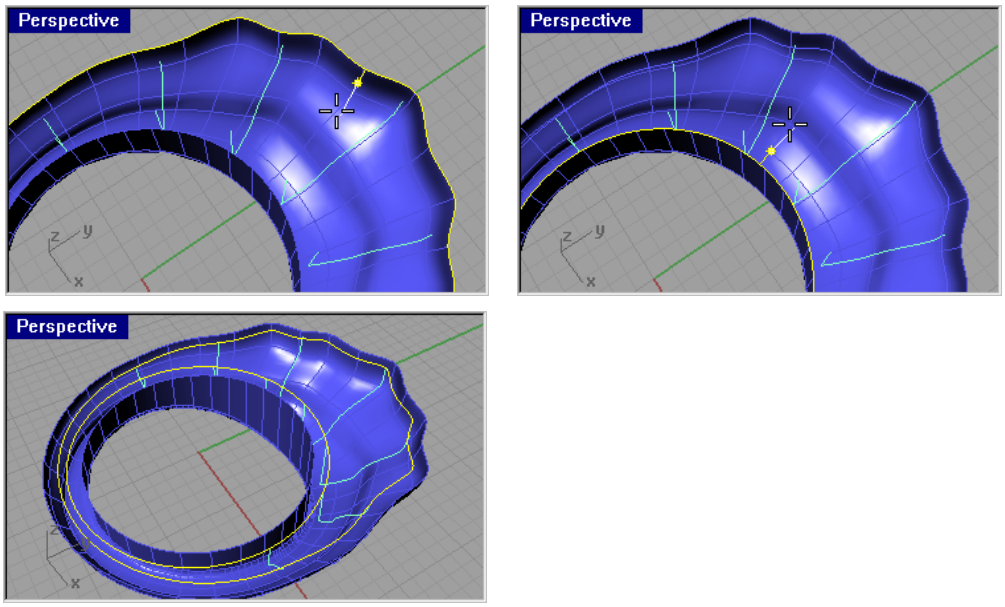
Once all the curves in the right position, offset the borders of the ring surface (internal and external borders shown in yellow), 1 mm to meet the end points of the offset curves. Use these curves to help define the surface.

To loft the inner surface

1. **Unlock** the ring polysurface *(Edit menu: Visibility > Unlock)*.
2. Use the **Offset** command to create the edge curves for the inner surface *(Curve menu: Offset Curve)*.

   Start the **Offset** command before selecting the surface edges.

   You cannot pre-select surface edges, but once you start the command, you will be able to do this.

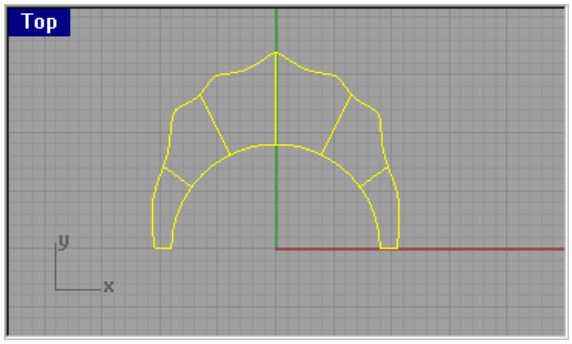


Offset the edge curves of the ring surface toward the interior.

3. **Hide** the ring surface *(Edit menu: Visibility > Hide)*.
4. Use the two smallest offset curves at the x-axis to **Trim** the offset border curves since you will use only part of them *(Edit menu: Trim)*.

   Set the **UseApparentIntersections** option to **On** in the **Top** viewport.

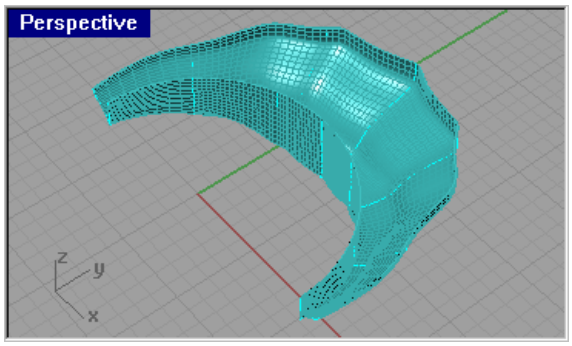
   You want the results to look like this:



The curves defining the interior surface.

5. Select all these curves and use the **NetworkSrf** command to make the inner surface *(Surface menu: From Curve Network)*.

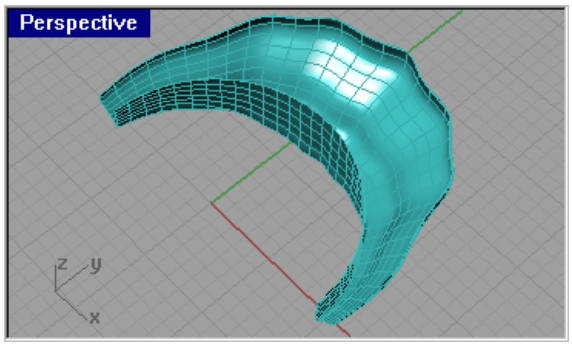
   Accept the default settings in the **Surface From Curve Network** dialog box.



The surface from curve network.

6 **Rebuild** the surface to reduce the complexity.

Use enough control points to keep the shape. In this case 33 points in the u-direction and 18 points in the v-direction were used.



The rebuilt surface.

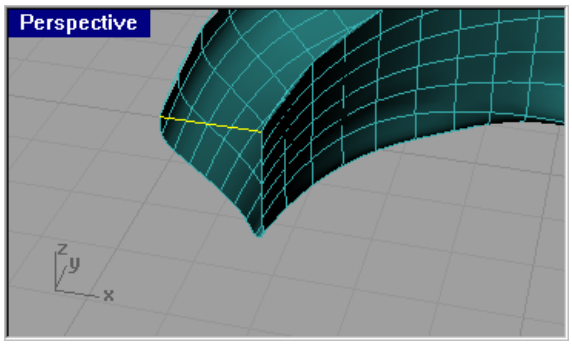
Cap the surface ends

A surface at the ends is needed.

To create the end surfaces

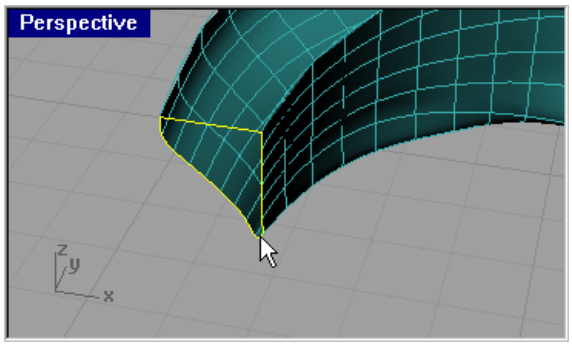
1 Draw a **Line** to connect the ending points of the surface *(Curve menu: Line > Single Line)*.

Use the **Endpoint** object snap.

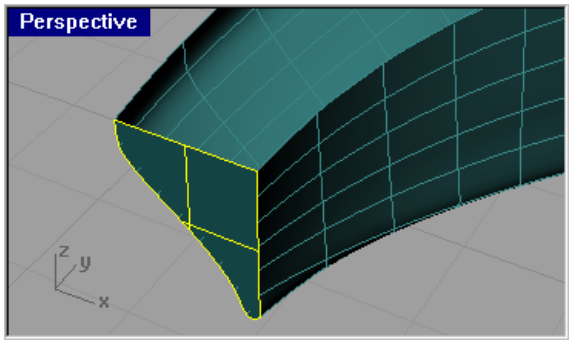


A line defines the top edge of the end surface.

2 Use the **PlanarSrf** command selecting the line and the surface edge to create the end *(Surface menu: Planar Curves)*.



Select the line and the edges of the surface to create the end surface.



The end surface.

3. Repeat this for the other side.
4. Use the **Join** command to make these three surfaces into one *(Edit menu: Join)*.
5. Use the **Cap** command to close the resulting polysurface.
6. Turn off the layer for the hollow.

Create the first mold part

The mold will be created in two parts: one half for the ring surface and one half for the hollow surface. You are now going to build the ring surface part of the mold.

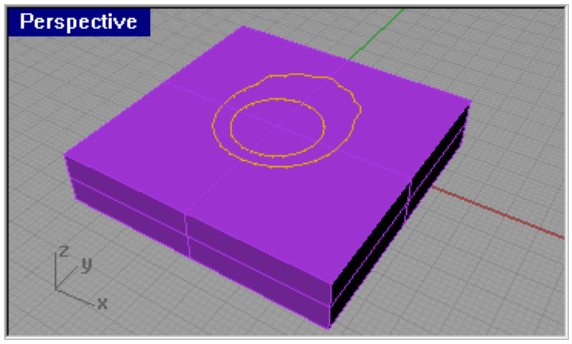
To create the mold part

1. Turn on the layer with the ring surface.
2. Use the **Box** command with the **Center** option to create a box with its top face lying on the construction plane *(Solid menu: Box > Corner to Corner, Height)*.

   The box has to be deep enough to enclose the ring surface.

   Start the center of the box at **0,0,0**.

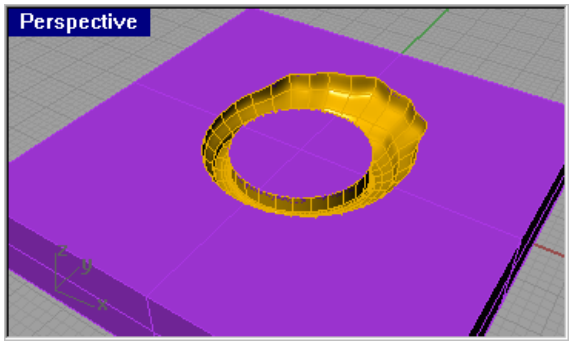
Make the **Length** and **Width 50 mm** and the **Height -10 mm**. This will center the box and make the top of the box line up with the construction plane. Remember that the lower half of the ring is below the construction plane.



Create the mold box.

3 Use the **BooleanDifference** command to create the ring shape in the box *(Solid menu: Difference)*.

Set the **DeleteInput** option to **No**. You may need the original ring surfaces later.

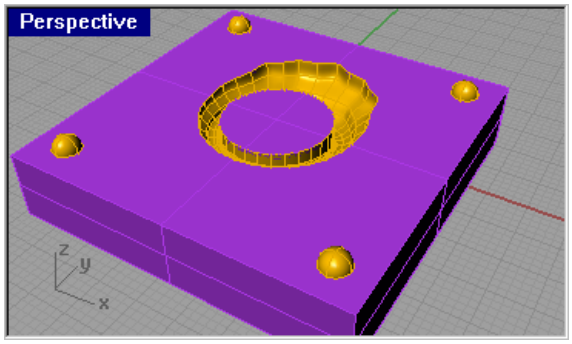


Cut the ring shape into the mold box with a Boolean.

4 Use the **Sphere** command to create four registration keys (notches) at the corners of the mold half *(Solid menu: Sphere > Center, Radius)*.

Use the **Snap** to draw the spheres with a diameter of **2 mm**.

You can draw one sphere and then use the **Mirror** command to create the other three.

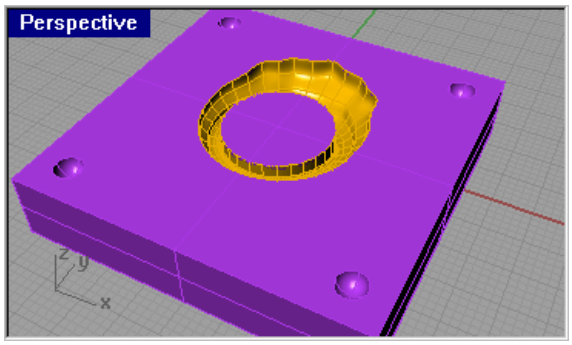


Create the spherical registration keys.

5 Use the **BooleanDifference** command to cut the registration keys into the mold *(Solid menu: Difference)*.

Set the **DeleteInput** option to **Yes**. The spheres were easy to draw and you will not be needing them again.

Registration keys help get a perfect alignment between this part of the mold and the other half. The second half will have four protruding half spheres that fit into to the four notches.



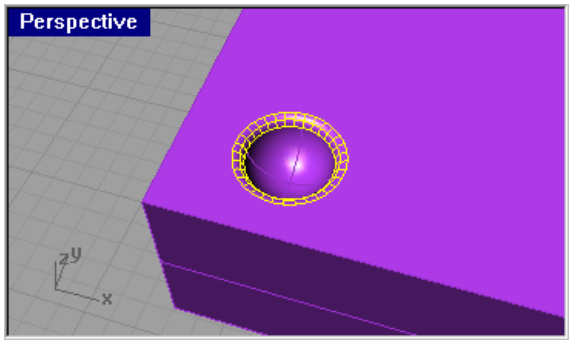
Cut the registration keys into the mold box with the spheres.

A very little touch-up is needed to finish the mold: filleting where the four little hemispheres join the upper face of the mold. This is to allow easier coupling between the two parts of the mold.

To round the corners

- Use the **FilletEdge** command to create the rounded edge of the notches *(Solid menu: Fillet Edge)*.

  The fillet radius is **0.5** mm.

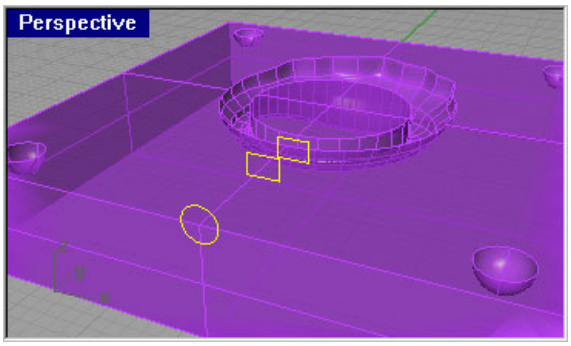


The rounded edges.

This part of the mold is complete.

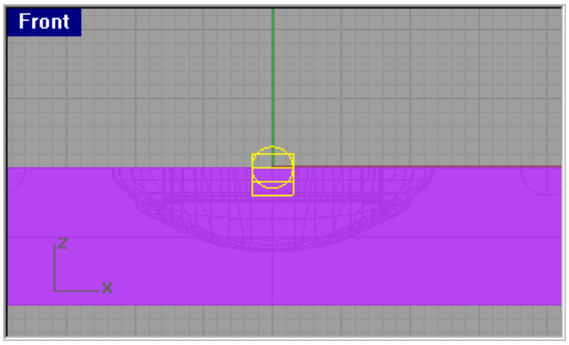
Create the gate

There is one missing part: the gate. The gate is the spout where the wax will be injected into the mold. To build the gate, use three closed curves as shown below.



The curves that define the gate.

Here is what the curves look like in the **Front** viewport.



The gate curves in the Front viewport.

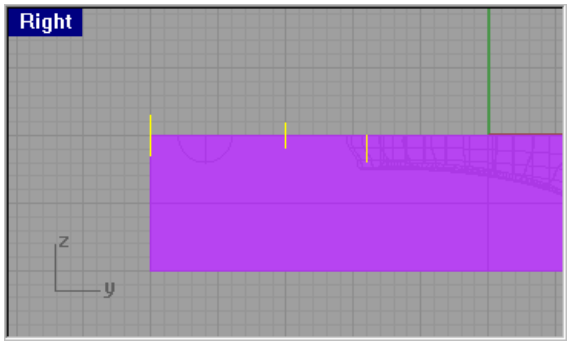
To create the gate:

1. Use the **Circle** command to draw a circle at the edge of the mold box *(Curve menu: Circle > Center, Radius)*.
2. Use the **Rectangle** command to draw two rectangles, one inside the edge of the ring surface and one about half way between the box edge and the ring edge *(Curve menu: Rectangle > Center, Corner)*.

   The rectangle closest to the ring is placed slightly below the other curves so that the wax will reach the part to be filled.

   This is visible in the **Right** viewport.

   The easiest way to do this is to create the circle and rectangles on the construction plane and then move them into place in the **Right** viewport.



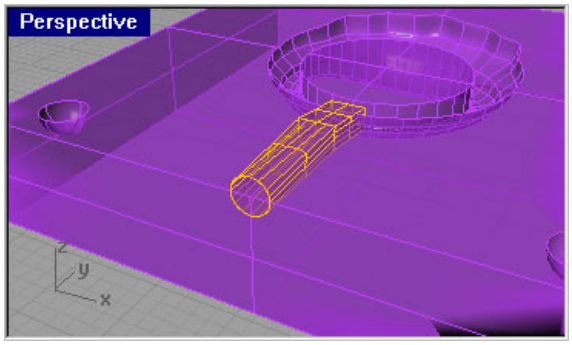
The gate curves in the Right viewport.

Considering the geometry of the piece, the rectangular shape is used to eliminate undercuts.

To create the gate

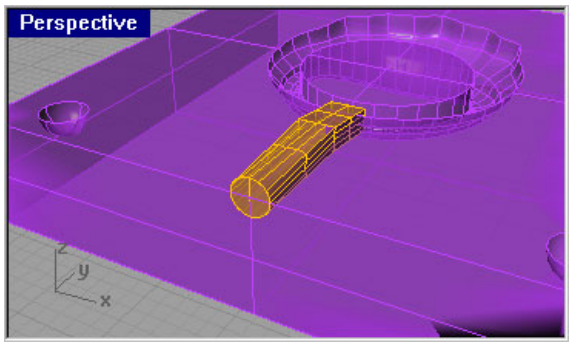
1 Use the **Loft** command to create a surface that connects the three closed curves *(Surface menu: Loft)*.

Use the **Straight** *sections* style option.



The gate surface.

2 Use the **Cap** command to close off the ends of the gate surface and make it into a solid *(Solid menu: Cap Planar Holes)*.



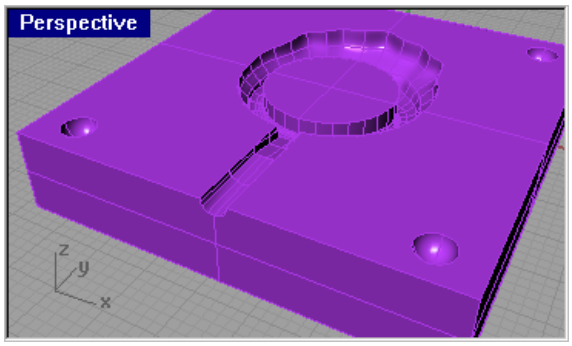
The capped gate surface.

This makes it easier to use the **BooleanDifference** command to cut the gate into the mold box. The Boolean commands work best when you use closed solids.

3 Use the **BooleanDifference** command to subtract the gate surface from the mold box *(Solid menu: Difference)*.

Set the **DeleteInput** option to **No**. You will use this part later to cut the gate into the upper part of the mold.

4 **Hide** the gate cutting solid *(Edit menu: Visibility > Hide)*.



The mold part.

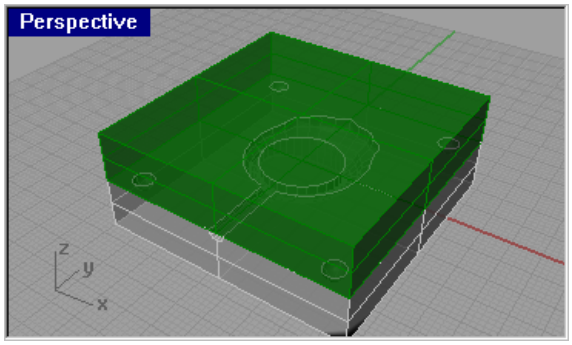
Create the second mold part

The mold is made of two halves. You will create the second half the same way as the first.

To create the second mold part

1 Use the **Lock** command on the mold part 1 you just created so you can work around it without accidentally selecting it *(Edit menu: Visibility > Lock)*.

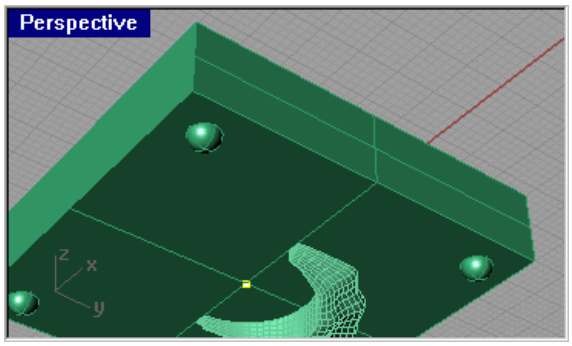
2 Use the **Box** command to create a new box right on top of the box used for mold part 1 *(Solid menu: Box > Corner to Corner, Height)*.



The second mold part.

3 Use the **Sphere** command to draw four small spheres with a radius of **1.95** that will form the other half of the registration keys *(Sphere menu: Center, Radius)*.

Use grid snap to center the spheres inside the notches you created in part 1. These new spheres will be slightly smaller than the ones used for the notches. This is to ensure some tolerance when the two pieces will be coupled. You can reasonably use a parting tolerance value of 0.05 mm; therefore, use a radius of **1.95**.



Draw the spheres for the registration keys.

4 **Unlock** mold part 1 and **Hide** it.

Create the second part

The second part of the mold is similar to the first; however, the model parts protrude from the box solid instead of being cut into it.

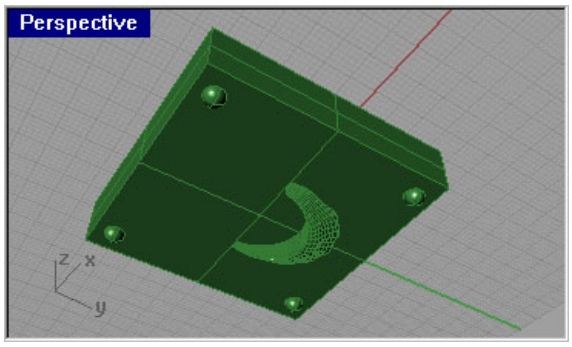
To cut and join the mold parts

1 Make the hollowing surface visible.

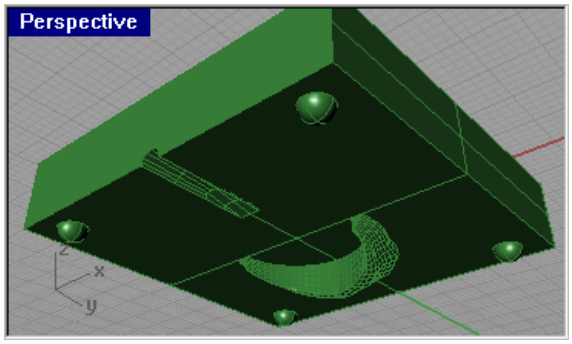
2 Use the **BooleanUnion** command to join the spheres to the box to create the registration keys for this half of the mold *(Solid menu: Union)*.

3 Use the **BooleanUnion** command to join the hollowing solid to the box.

If the Boolean fails, try moving the solid surface very slightly up toward the box.

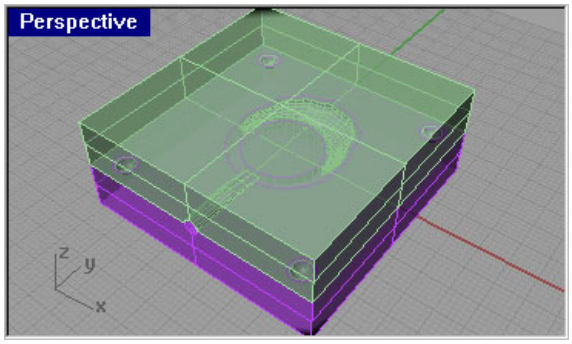


4 To create a gate in this part, use **BooleanDifference** command to subtract the gate solid you created earlier *(Solid menu: Difference)*.

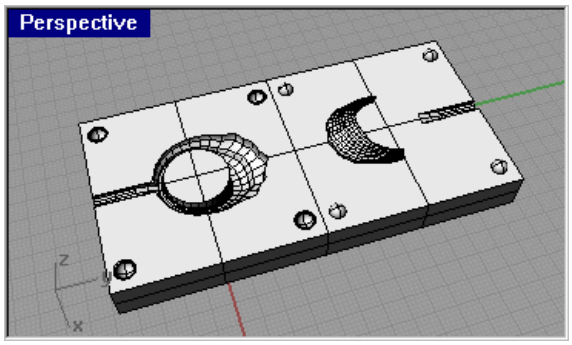


The model of the two-part mold looks complete at this point. Spend a few moments checking what you have just built.

The **GhostedViewport** mode has given an almost transparent appearance to the mold so you can see how things are looking where the wax will be poured.



For illustration purposes only, the two parts of the mold are now placed side by side and opened up.



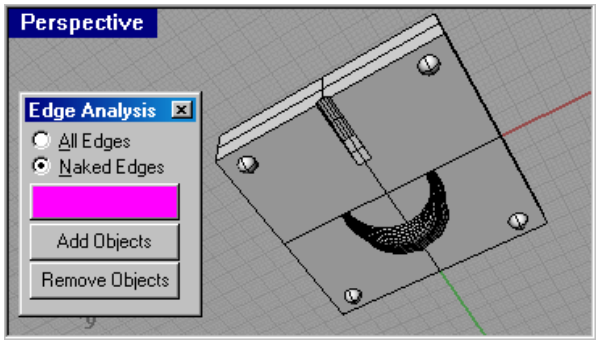
Analyze the model

Rhino lets you analyze a surface in several ways. You will first analyze the model to determine if there are unjoined edges. Most machining programs require that the model be a closed solid with no unjoined edges.

To check for unjoined edges

- Use the **ShowEdges** command to check for unjoined edges *(Analyze menu: Edge Tools > Show Edges)*.

  Make sure your part is a closed polysurface (no naked edges).



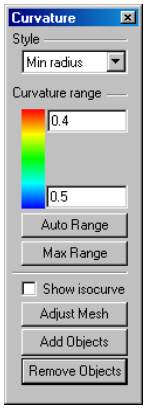
Mold top half with edge analysis.

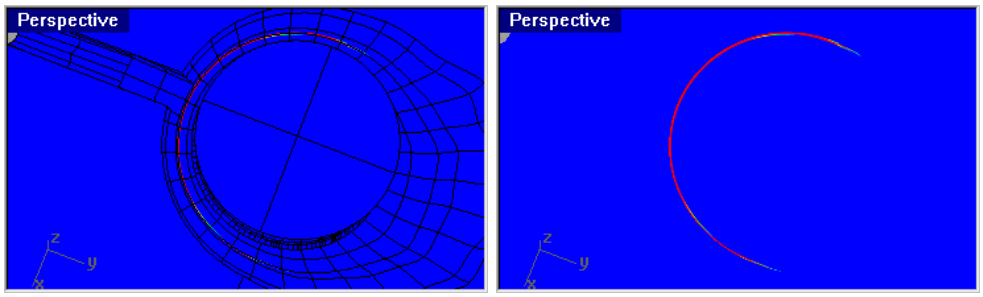
You should also examine the model for minimum radius of the curved surfaces. This will tell us what can be machined by tools of various radii.

To analyze the surface for minimum radius

1. Select mold part 1.
2. Use the **CurvatureAnalysis** command to check the surface *(Analyze menu: Surface > Curvature Analysis)*.

   Use the **Min radius** option.
3. Set the red upper limit to **.4**.
4. Set the blue lower limit to **.5**.



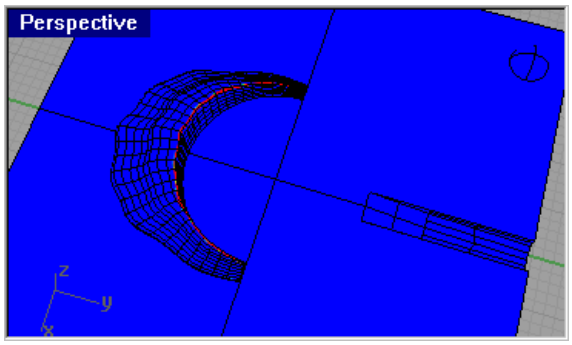


Curvature analysis with isocurves on (left) and off (right). The red areas are easier to see with the isocurves turned off.

The red areas of this piece have a radius smaller than the upper *Min radius* value. This means that a tool with a radius of 0.4 mm will leave a fraction of material uncut.

The curvature analysis is a representation of the tooling that will be used in the machining phase. In most extremely small radius situations, you will have to compromise between the uncut material and the reasonable size of the tool. As you will see in the VisualMill part of this tutorial, the smallest tool used is a 1-mm cutter.

5 Do the same analysis on mold part 2.



At this point you have completed the modeling and analysis.

Using VisualMill

The next step of the project will be to import the mold parts into VisualMill, and to develop a cutting strategy. Models are saved to use for this purpose, so you do not have to complete the model.

VisualMill will read both Rhino NURBS model files and Rhino mesh files. Normally, you can simply read in the Rhino model into VisualMill, but sometimes it works best to create the mesh in Rhino. In either case, export only the mold parts or a mesh created from the mold parts to a separate model.

The machining process

Numerically controlled (NC) machines, use a variety of cutting tools similar to drills or router bits to cut a piece of stock material to a shape. Milling programs such as VisualMill generate a list of instructions (G-Code) for the milling machine from a 3-D model. The G-Code instructions for the machine list which cutters (tools) to use, in what order, and where to cut (tool path). G-Code files are typically loaded by the NC controller and then "played" in order to move the cutter over the stock.

In the first stages of machining, called roughing, the object is to quickly remove the bulk of the waste material. This leaves stair-stepped shapes in the material.

In the final stages, called finishing, smaller, more refined tools are used to form the final profile, removing the stair-steps left behind in the roughing procedures.

After final machining stage the piece normally still needs hand polishing and finishing work before the article is complete.

Open the model

There are several ways to import geometry produced in Rhino into VisualMill 4.0. You can export your model as an STL, VRML, or RAW mesh file, or you can export it as an IGES or Parasolid file. VisualMill can also open Rhino files directly and use either the surface data or mesh data in the Rhino file.

The RhinoCAM plug-in for Rhino V3 is another option. If you have VisualMill 4 installed, with RhinoCAM you can be in a Rhino session and open the entire Rhino file or just selected objects in VisualMill. RhinoCAM eliminates the need to save your Rhino data to an intermediate file format before opening it in VisualMill.

In the first half of this tutorial, you will open the Rhino file that contains the top half of the ring mold you created in the first part. You will be guided through all of the steps required to generate the machine G-Code required to actually cut the part.

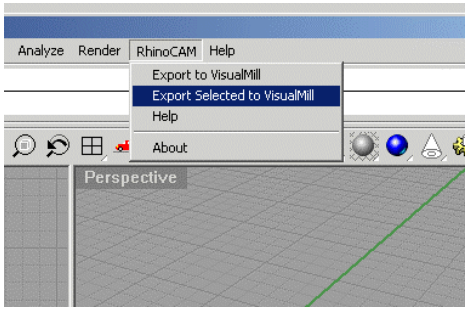
Each new operation will be shown in detail. If an operation is repeated in the tutorial, less detail will be provided, since you will have seen it before. For specific information on VisualMill options, see the VisualMill Help file.

The second half of the tutorial will cover the bottom half of the ring mold.

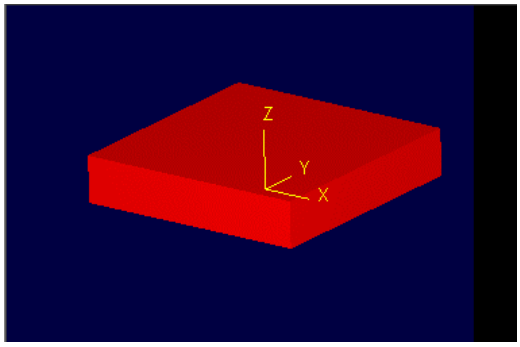
Warning: Operation of machine tools is a dangerous business. Cutters and material can break and fly around at high speed causing serious injury. Before actually milling a part with the G-Code generated in this tutorial, you should be well versed with all of the safety procedures required for safe operation of your equipment.

To export the part to VisualMill with the RhinoCAM plugin

1. Select the top half of the mold.
2. In Rhino, select the mold top half and from the **RhinoCAM** menu, select **Export Selected to VisualMill** *(RhinoCAM menu: Export Selected to VisualMill)*.



VisualMill automatically preserves the location, orientation, and units from the Rhino file.



Mold top half in VisualMill.

Orientation the part

Once the geometry has been successfully imported into VisualMill, you will orient the model for machining and set the *machine zero position*. The machine zero position is the reference point for all the machining operations (MOPs) and represents the initial physical position where the tip of the cutter will be positioned. For example, if the zero position will be the center-top based on the part's bounding box, you will move the cutter tip to the upper center point of the stock material when it is loaded in the milling machine.

The part is in the same orientation (upside-down) that it was in Rhino. It will need to be flipped so it can be cut.

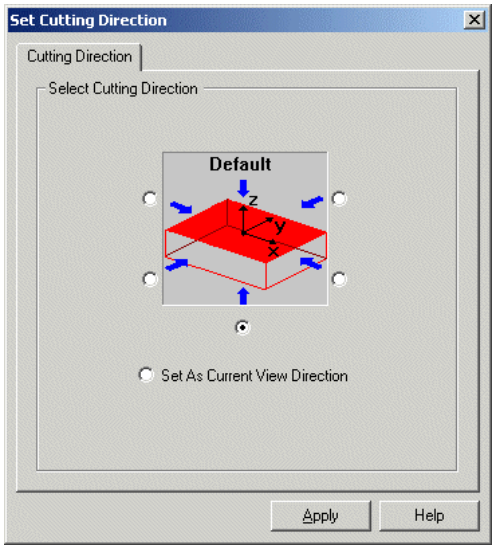
To set the part orientation

1. In the **Machining** toolbar, click the **Set Cutting Direction** button.

Button	Function
	Set Cutting Direction

2. In the **Set Cutting Direction** dialog box, change the direction to the bottom as shown in the image below.

This will flip the part so the detail is positioned under the cutting tool. The cutter on a 3-axis mill cuts from the top of the part.



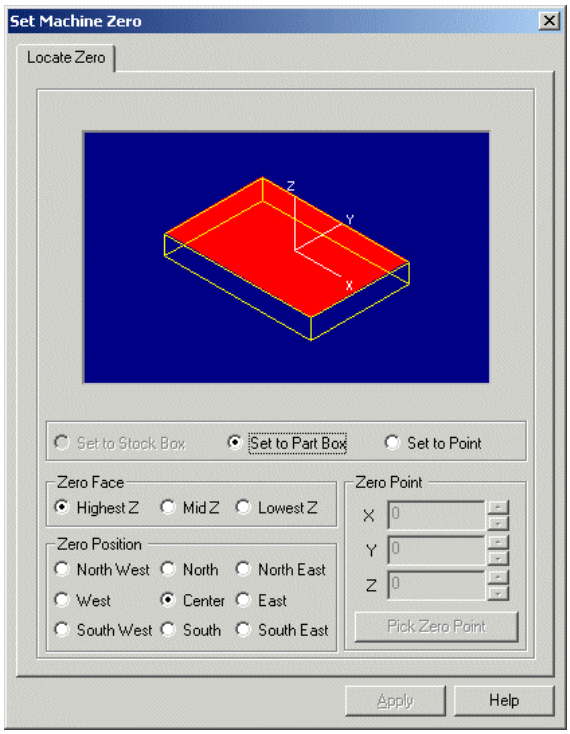
Set cutting direction.

To set the zero position

1. In the **Machining** toolbar, click the **Set Machine Zero** button.

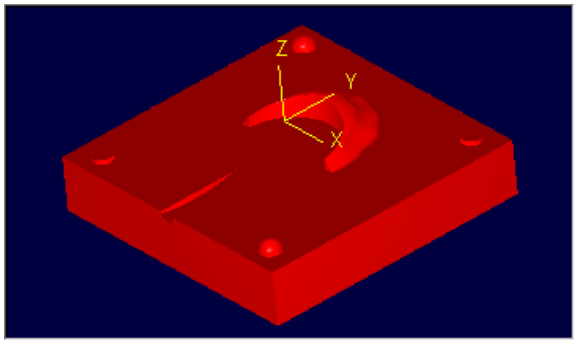
Button	Function
	Set Machine Zero

2. In the **Set Machine Zero** dialog box, set the machine zero position to the top center of the part box as shown in the image below.



Set machine zero.

The image below shows the part flipped and the machine zero position set.



Part setup complete.

Create a stock box

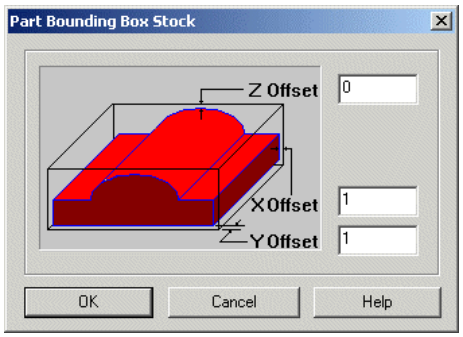
The next step is to create a stock box. The stock box represents the ideal raw stock that will be loaded into the mill. Assume the highest point (z) on the part will be even with the stock. Leave a little offset space around the sides in the x- and y-directions to allow for easier loading of the stock into the mill. After you create the stock, hide it.

To create a stock box

1 From the **Machining** bar, click **Create/Load Stock** and from the menu, select **Part Box Stock** *(Stock menu: Part Box Stock)*.

Button	Function
	Create/Load Stock

2 In the **Part Bounding Box Stock** dialog box, set the **X Offset** and **Y Offset** values to **1** as shown in the image below.



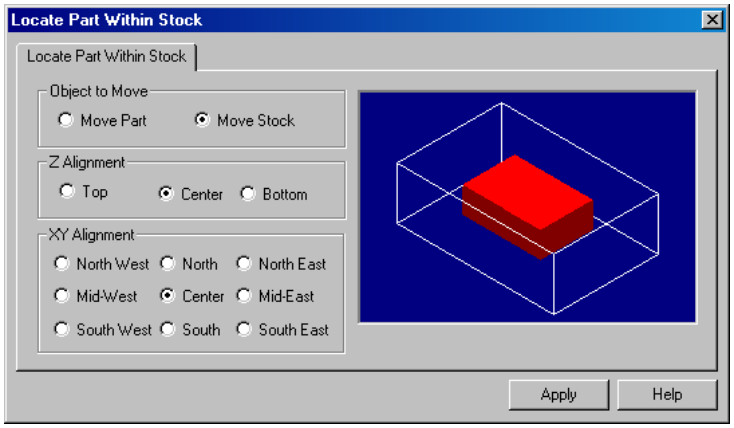
Part box stock.

To locate the part within the stock

1 From the **Machining** bar, click **Locate Part Within Stock**.

Button	Function
	Locate Part Within Stock

2 In the **Locate Part Within Stock** dialog box, set the **Object to Move** to **Move Stock** and the **Z Alignment** and **XY Alignment** to **Center** as shown in the image below.



Move the part to the center of the stock.

To hide the stock

- From the **View** toolbar, click the **Hide Stock** button *(View menu: Hide Stock)*.

Button	Function
	Hide Stock

Create cutting tools

You are now ready to create a set of virtual cutting tools that will be used to machine the mold half. Normally, you will have created a library of cutting tools that you maintain for your mill. No library has been created for this tutorial, so you will create the tools you need. The virtual cutting tools you create in your milling program must match the actual cutting tools that are used in your milling machine. There are many different mill manufacturers and cutting tool makers. Each cutting tool has its own dimensions (tool length, flute length, diameter). Each tool you create for your VisualMill library must match an actual tool you have for your milling machine.

To create a cutting tool

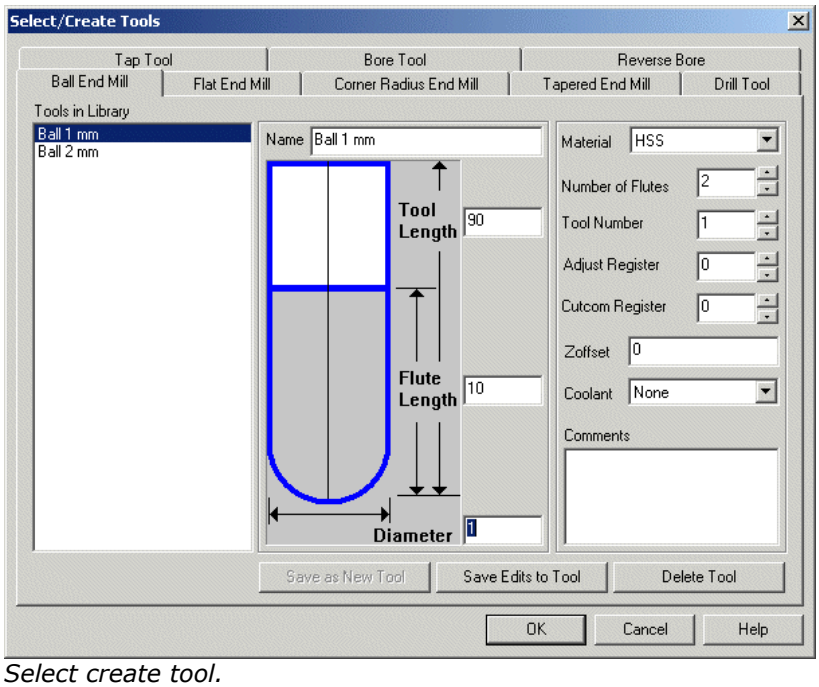
1. From the **Machining** toolbar, click the **Create/Select Tool** button *(Tool menu: Create/Select Tool)*.

Button	Function
	Create/Select Tool

2. Using the **Select/Create Tools** dialog box, **Ball End Mill** and **Flat End Mill** pages, create three cutting tools with parameters as shown in the table below.

   Label the tools descriptively so you can easily tell them apart.

Name	Tool Length	Flute Length	Diameter
Ball 1 mm	90	10	1
Ball 2 mm	90	10	2
Flat 4 mm	90	60	4



Select create tool.

Check speeds and feeds

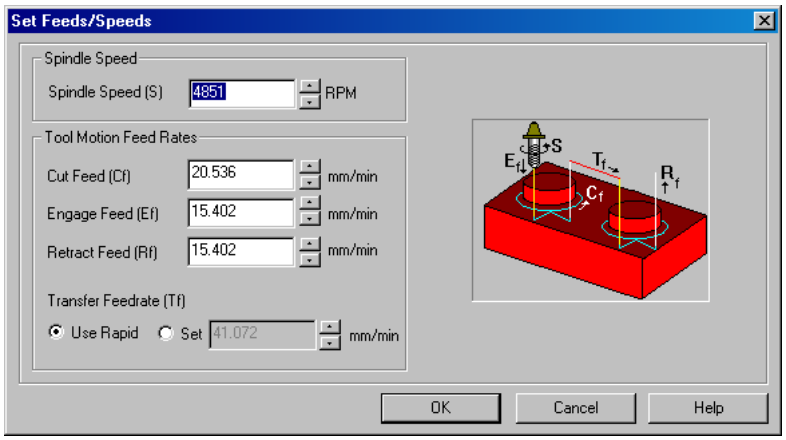
Typically, at this point, spindle speeds, and tool feed rates are established. These settings depend on the size of the cutting tool, the cutting tool composition, the stock being cut, the make, model, and condition of the milling machine, and the experience of the machinist.

To check the feeds and speeds

1 Click the **Set Feeds/Speeds** button *(Feeds/Speeds menu: Set Feeds/Speeds)*.

Button	Function
	Set Feeds/Speeds

2 Look at the current settings.



Set Feeds/Speeds dialog box.

If the cutting tool you are going to use were very small or the cuts were deep, it would be necessary to slow the cutting speed and rate of movement of the stock part through the machine. You do not need to change these settings for this particular operation, but it is a good idea to be aware of this issue at all times.

Note: A meaningful discussion of the topic of machine speed is beyond the scope of this tutorial. Consult the proper references for information on this subject. The information that comes with your milling machine will have some information on this subject. In addition, an industry standard work on the subject is *Machinery's Handbook*, by Erik Oberg, Franklin Day Jones, Henry H. Ryffel; Industrial Press; ISBN: 0831126256.

Define a machine operation

The first cutting strategy you will use is horizontal roughing. This operation uses a relatively large tool and is used for the majority of the stock removal. The horizontal roughing operation removes material in levels. The tool starts at the top of the stock and removes material moving only in the xy-plane. Once this level is completed, the tool moves to the next lower level and removes material in this xy-plane. With each successive operation you get closer to the finished part.

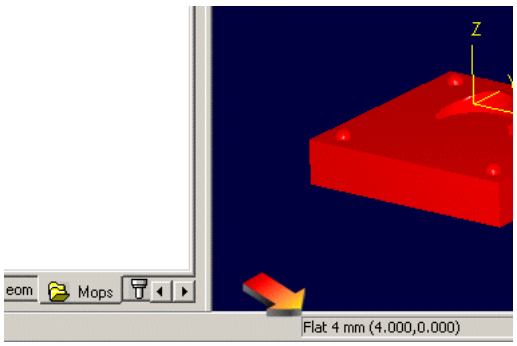
The ability to determine which cut strategy works best for each situation comes from experience and training as a machinist.

To define a horizontal roughing operation

1 To set the active tool, in the **Select/Create Tools** dialog box, select the **Flat 4 mm** tool *(Tool menu: Create/Select Tool)*.

Button	Function
	Create/Select Tool

Check the status bar to verify that this tool is active.

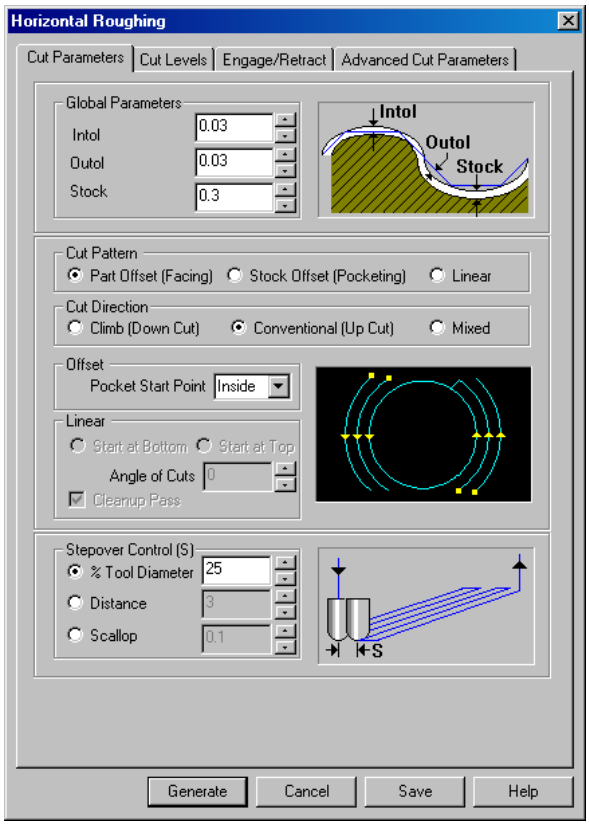


Flat 4-mm tool.

2 From the **Machining** toolbar, click the **3-Axis Machining** button, and then from the menu, select **Horizontal Roughing** *(3 Axis Milling menu: Horizontal Roughing)*.

Button	Function
	3-Axis Machining

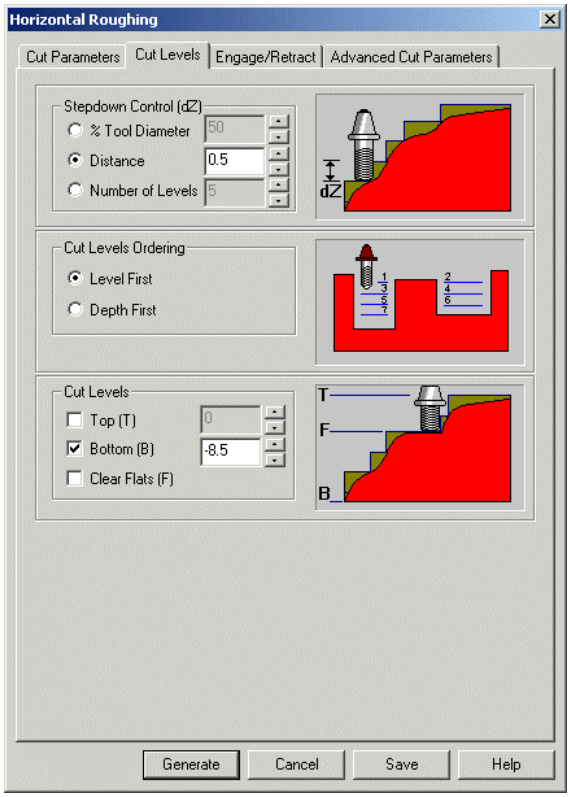
3 In the **Horizontal Roughing** dialog box, on the **Cut Parameters** page, set the values as shown below.



Set horizontal roughing parameters.

4 In the **Horizontal Roughing** dialog box, on the **Cut Levels** page, set the values set the parameters as follows:

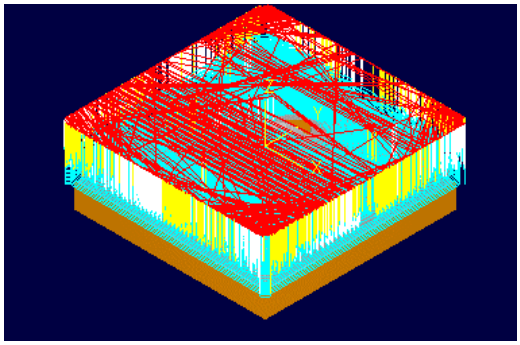
Stepdown Control Distance: **0.5**
Cut Levels Ordering: **Level First**
Cut Levels Bottom: **-8.5**



Horizontal roughing parameters.

To generate the milling operation tool paths

1 In the **Horizontal Roughing** dialog box, click the **Generate** button to create the milling operation (MOP) tool paths.



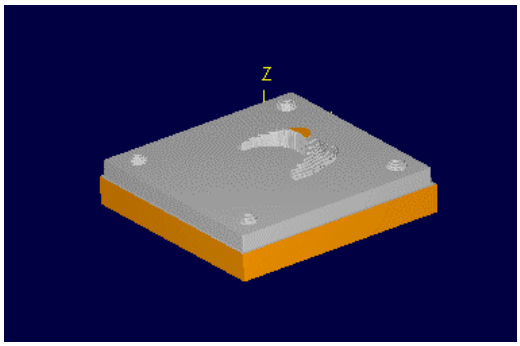
Horizontal roughing tool paths

2 Using the **View** toolbar, hide the part model, the grid, and the tool path, and show the stock.

3 On the **VCR** toolbar, click the **Simulate** button.

Button	Function
	Simulate

This will process the tool paths and display what your stock will look like after the initial roughing process.



Horizontal roughing simulation.

4 In the browser pane, click the **Mops** page.

5 Double-click an option that defines the roughing operation to change its parameters.

You can also rename the operation to make it more descriptive. If you make a change, a red star appears on the **Stock** option in the browser to indicate the display is not current with the MOP definition.

6 In the **VCR** toolbar, click the **Simulate** button update the display.

Remove material from flat areas

You will use plateau machining to remove the remaining stock from the flat areas. In plateau machining the cutter is restricted to machine areas in the part that are shallower than a specified angle from the horizontal plane. Plateau machining is used to finish areas that were not machined completely by a roughing operation.

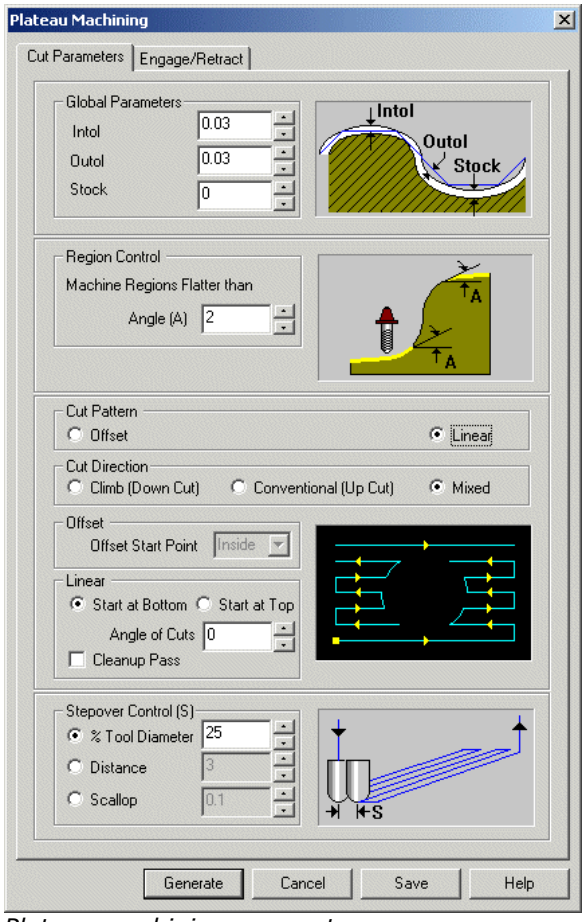
To define a plateau machining operation

1 Be sure the **Flat 4 mm** tool is active.

2 From the **3-Axis Machining** menu, select **Plateau Machining** *(3 Axis Milling menu: Plateau Machining)*.

Button	Function
	3-Axis Machining

3 Set the parameters as shown below.



Plateau machining parameters.

The **Region Control Angle** is defined in degrees from the horizontal. Any area within the flatness region will be machined. A value of 2 degrees will ensure that all of the flat parts will be cut down to a maximum of a 2-degree angle.

4 Click **Generate** to create the milling operation.

5 On the **VCR** toolbar, click **Simulate** to update the stock model display.

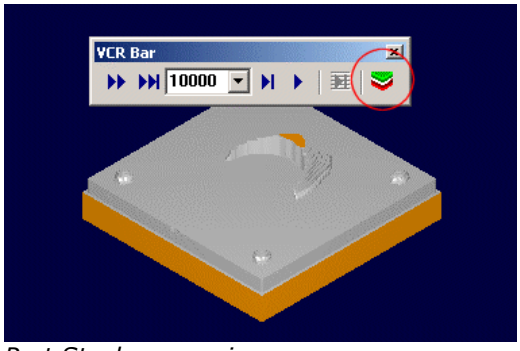
Button	Function
	Simulate

Use analysis tools

At this time, run the comparison of the part to the stock. This will show you the differences between the modeled part and the stock material remaining.

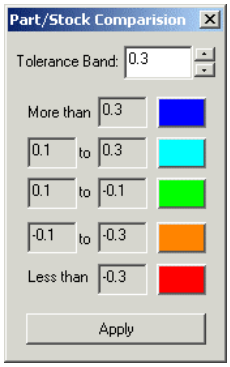
To compare the stock to the part

1 In the **VCR** toolbar, click the **Part/Stock** comparison button.



Part Stock comparison.

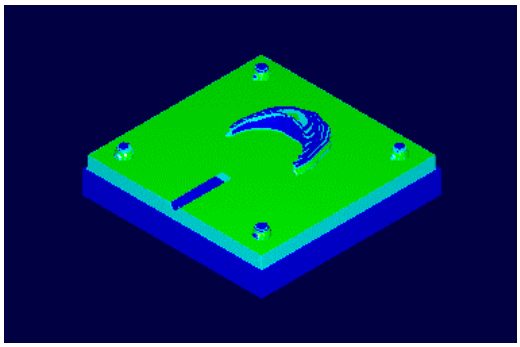
2 In the **Part/Stock Comparison** dialog box, set the **Tolerance Band** value to **0.3**, and click the **Apply** button.



Part/Stock comparison dialog.

In the roughing machine operation, a stock thickness of 0.3 mm was left. In the plateau machining operation, the additional stock thickness was eliminated so the flat areas of the mold were cut to final elevation. Setting the **Tolerance Band** to **0.3** will accurately show the areas of the stock model that still have excess material to be machined away. The green areas indicate good finished surfaces. The dark blue areas indicate where more material still needs to be removed. The remaining machining operations you create will remove the remaining excess material.

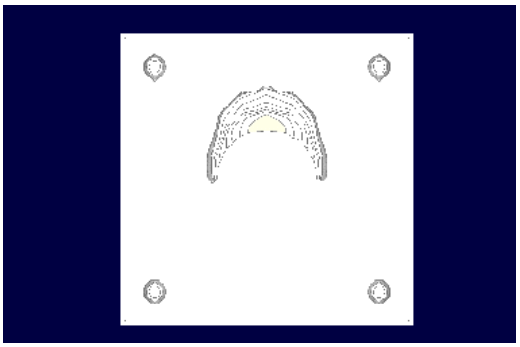
Note: VisualMill 4 has two different simulation methods for modeling the Part/Stock comparison display: 3 and 4-Axis simulation model. Depending on the geometry of your model, one method may create a more accurate display than the other. The trade-off is speed. The 3-axis method is faster while the 4-axis method is more accurate. For the images in this demonstration, the 4-axis method is used. The setting can be changed from the Preferences menu, Machining Preferences.



Part/Stock comparison.

Remove stepover notches

In the top view shown below, you can see excess stock around the four registration keys. This is left over from the relatively coarse step-over value used in the plateau machining operation. You will use a horizontal finishing strategy to remove this excess stock.



Top view showing extra material at registration keys.

To define a horizontal finishing operation

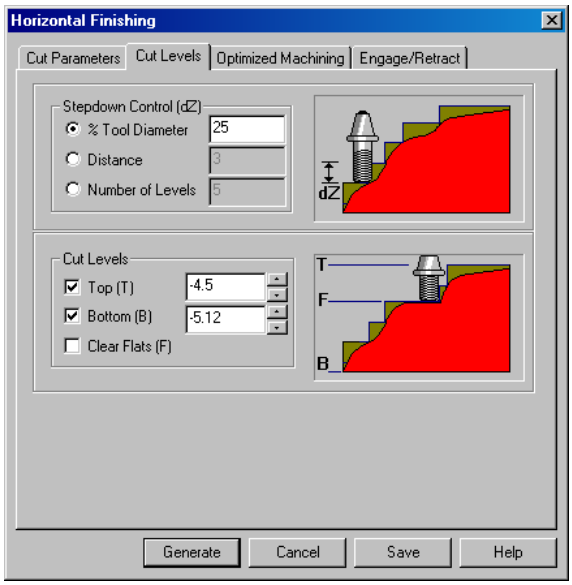
1. With the 4-mm flat-end mill active, from the **3-Axis Milling** menu, select **Horizontal Finishing** *(3 Axis Milling menu: Horizontal Finishing)*.

Button	Function
	3-Axis Machining

2. On the **Cut Levels** page, under **Cut Levels**, set the top value to **–4.5** and the bottom value to **-5.12**.

   This will limit the machining to a horizontal finishing pass in a single level and single path. The distance from the top of the part to the flat surface is 5.131. There is no need to use a cut level any higher than –4.5 since the tool will cut nothing at that level. Try setting a higher level and generate the tool paths. You will observe that additional tool paths are created, but there is no material left to cut at that level.

   The Stepdown Control moves the tool down 1 mm (25% of the tool diameter) with each pass. Experiment with the control to get as close as possible to -5.131.



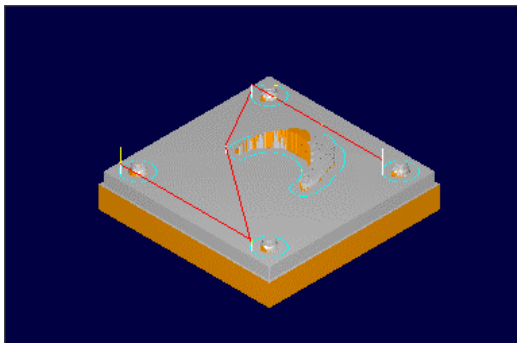
3 Click the **Generate** button the machine operation

4 On the **VCR** toolbar, click **Simulate** to update the display

Button	Function
	Simulate

5 Turn on the tool paths to verify the route the tool will take.

If there is more than one tool path around the areas to be cut, the **Top** level is too high. If there is only one tool path around the whole piece, the **Bottom** level is too low.



Horizontal finishing tool path.

Define regions

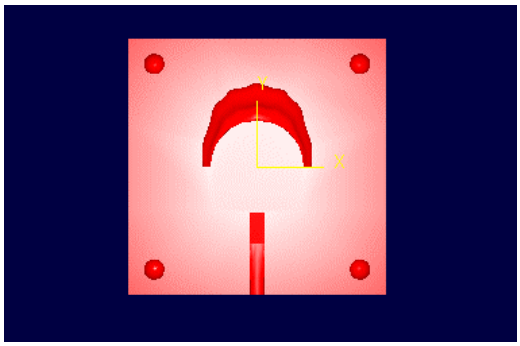
Now that the final machining for the flat areas is substantially completed, you are down to finish strategies for the four registration keys, ring-hollowing plug, and gate. It would be a waste of time to let the cutter go over the flat regions of the stock since they are complete. Restricting tool paths to regions is more efficient. First, create the six regions. Then make them active and generate tool paths within the active regions.

To set up the display

1. Set your display to the top view with the part model displayed.
2. **Hide** the stock and tool paths.
3. Set the display for the grid at your choice.
4. Make sure you do not have **Hide Machining Regions** turned on.

   You will want to see your regions as you create them.

   Your display should look something like this:



Top view.

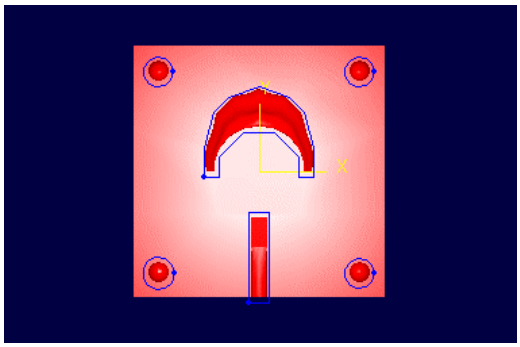
To create machining regions

1. From the **View** menu and select **Toolbars** *(View menu: Toolbars > Curves/Regions Bar)*.
2. Turn on the **Curves/Regions** toolbar.
3. Use the rectangle, circle, and polygon/polylines tools to create regions around the six regions that define the four registration keys, the gate, and the ring.

   The region creation has a few basic pointing tools to assist you in creating the regions including object snap and grid snap.

   It is not critical to precisely match the region boundaries to the part features. The regions should just completely surround the features with a minimum of extra space. These regions are only used to minimize unnecessary tool paths in areas that do not need to be machined.

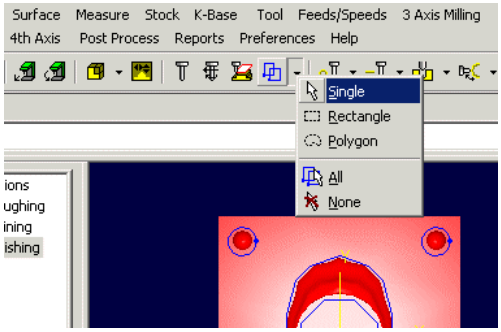
   After creating the six regions, your display should look something like this:



Machining regions.

To activate the regions

1 In the **Machining** toolbar, from the **Select Regions** button menu, select the **Single** tool.

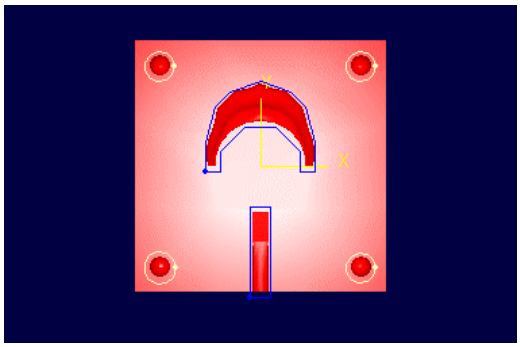


Select regions.

2 Select the four regions that surround the four spherical registration keys.

They will turn yellow to indicate they have been selected.

Pres and hold the **Ctrl** key to add all four regions.



Active regions highlight yellow.

Finish the registration keys

You will now use a ball-end tool and parallel finishing to finish the registration keys.

To finish the registration keys

1 From **Create/Select Tool** button menu, select the **2 mm ball end** mill *(Tool menu: Create/Select Tool)*.

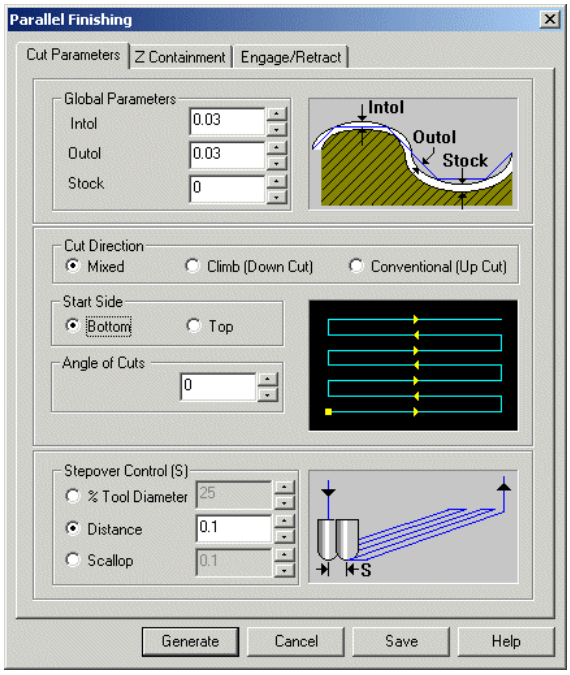
Button	Function
	Create/Select Tool

2 Check the status line to verify this is the active tool.

3 From the **3-Axis Milling** menu, select **Parallel Finishing** *(3 Axis Milling menu: Parallel Finishing)*.

Button	Function
	3-Axis Machining

4 Set the parameters as shown below:

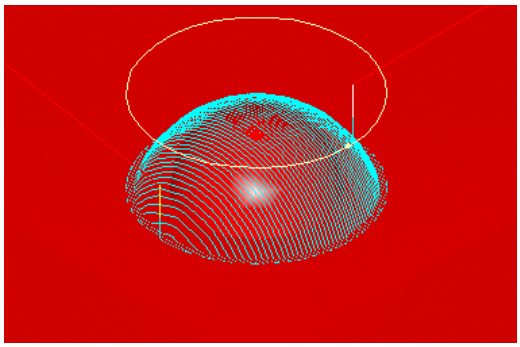


Parallel finishing settings.

5 Click **Generate** to create the tool paths.

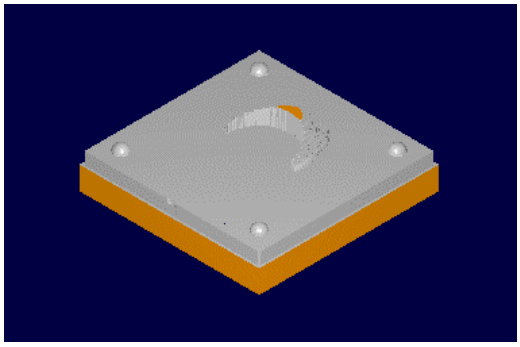
6 Show tool paths to see them.

They will only be created within the four active regions.



Detail of parallel finishing on a registration key.

7 Turn off the part model, turn on the stock, hide the tool paths, and run **Simulate** to see the results of the machining operation.



Result of parallel finishing for the registration keys.

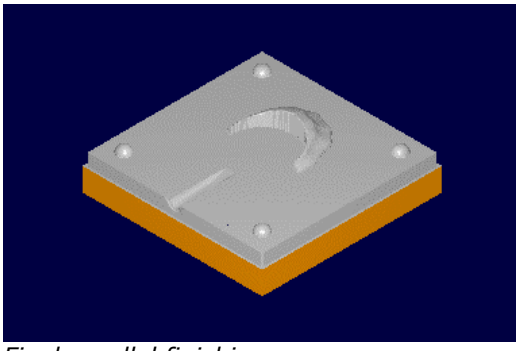
Final finishing

Use the same process and settings to create two more parallel finishing machine operations for the ring and gate regions using the **2 mm ball end** mill.

An additional option to consider when creating the machining operation for the gate is to rotate the angle of the tool paths. The default **Angle of Cuts** is zero (0). This orients the tool paths across the gate. Changing the **Angle of Cuts** to **90 degrees** will align them with the gate.

If you forget to activate the proper regions or tool before you generate the tool paths, you can select them later. Just right-mouse click on the appropriate parameter within the machining operation definition, make the correct selection, regenerate the tool paths, and run simulate to update the stock model.

After these two parallel finishing machining operations, your stock model should look like this:



Final parallel finishing.

Finish the gate

The only area where there is still an excess of stock is in the gate itself. This is because it transitions from a circular cross-section to a rectangular shape. The 2-mm ball end mill has left a some uncut stock. You will use a smaller tool and a new machining strategy to remove more of this excess material and complete the gate.

To finish the gate

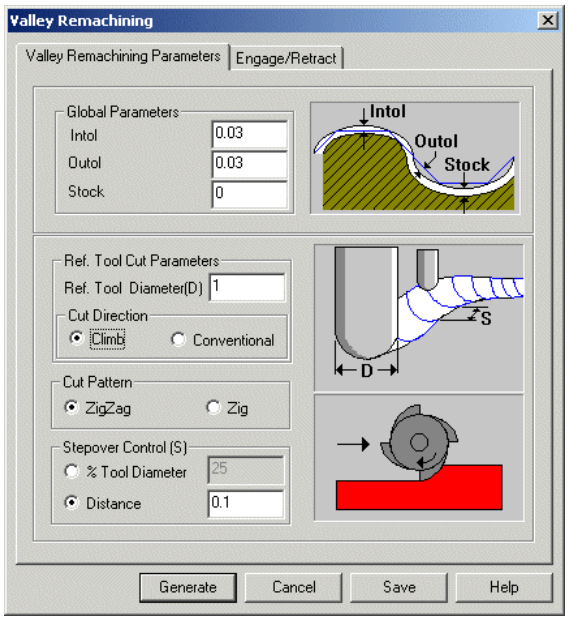
1 Make the **1 mm ball end** mill tool current and make the gate-machining region active *(Tool menu: Create/Select Tool)*.

Button	Function
	Create/Select Tool

2 From the **3-Axis Milling** menu, select the **Valley Re-Machining** strategy *(3 Axis Milling menu: Valley Re-Machining)*.

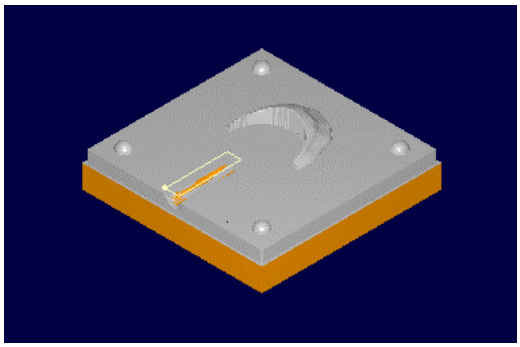
Button	Function
	3-Axis Machining

Set the parameters as follows:



Valley Re-machining settings.

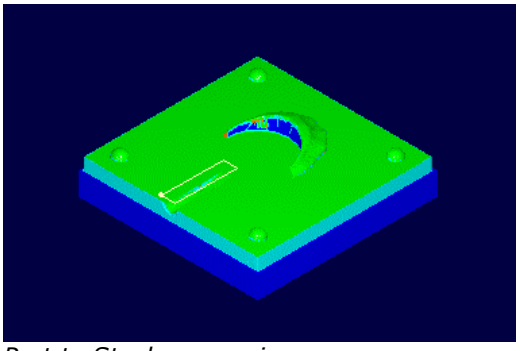
3 **Generate** the tool paths and run **Simulate** to update the stock model. This half of the ring mold is now completed.



Valley re-machining results.

Compare the part and stock

Use the **Part/Stock Comparison** tool to see if the machine operations you created removed all of the excess material. The only excess material appears to be on the backside of the ring. This apparent excess is because the surface was modeled slightly undercut. Since a 3-axis machine cannot cut undercuts, this excess material can be ignored in this case. This situation is normal and does not adversely affect the purpose of this modeled feature. It is very rare when all of the excess material can be removed by machining due to the physical size limitations of the tools used and the modeling techniques employed in the original idealized model.



Part to Stock comparison.

Generate tool paths

The last step is to post process your tool paths to produce G-Code for your particular NC controller. This post-processes the tool paths, tool changes, speeds and feeds, into the particular "flavor" of G-Code the NC controller on your mill requires.

To produce machine G-Code

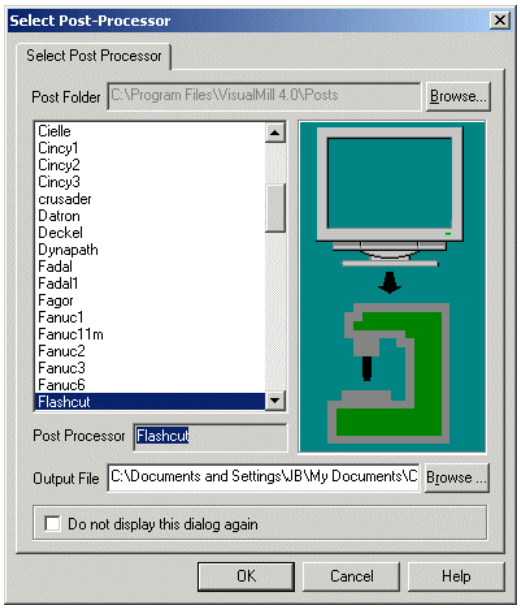
1. From the **Post Process** menu, select **Post Process** *(Post Process menu: Post Process)*.

Button	Function
	Post Process

2. Select the appropriate post processor for your controller, specify a file name and location for your file, and click **OK**.

   Note: This feature is disabled in the demo version of VisualMill.

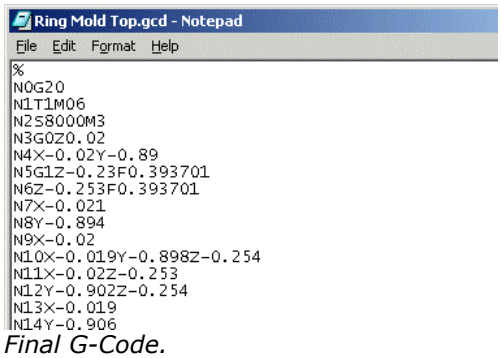
   If your controller is not listed, contact technical support at MecSoft (support@mecsoft.com) for help is selecting or creating a post processor.



Select the appropriate post-processor.

VisualMill will process the information and create the specified file. When this is complete, the output file will be displayed in the Windows Notepad editor where it can be reviewed and modified (if needed) by an experienced CNC code writer. It is common to add descriptions and tool requirements to the file for archiving purposes.

It is beyond the scope of this tutorial to enter into a discussion of G-Code editing.



Final G-Code.

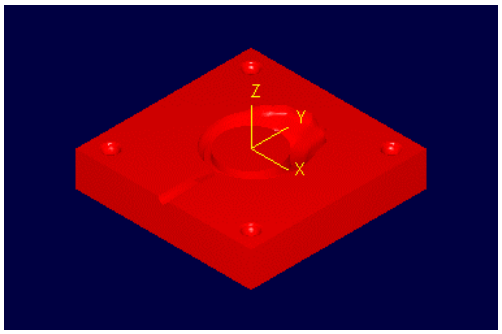
Set up the second part

The second part of the tutorial uses many of the same machining strategies as the first part. A few new strategies will be introduced and a new tool will be needed to efficiently machine this part.

To export the second part to VisualMill

- Select the lower half of the mold model in Rhino, and from the **RhinoCAM** menu, select **Export Selected to VisualMill**.

Note: If VisualMill is not currently running, it will be launched and the geometry will be exported to VisualMill. If an instance of VisualMill is already running, the geometry in the current VisualMill session is overwritten with the geometry sent from Rhino. The machining operations are not overwritten. They can be used with the newly exported geometry if they are appropriate. In this case they are not.



The second mold part in VisualMill.

To set machine zero position

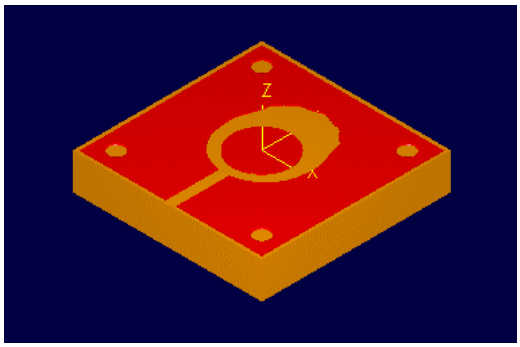
- Use the **Set Machine Zero** tool to set the following:

Locate Zero	Set to Part Box
Zero Face	Highest Z
Zero Position	Center

To create stock box definition

- Use the **Create/Load Stock** tool to define a **Part Box Stock** model with the following settings *(Stock menu: Part Box Stock)*:

Offset	Value
Z Offset	0
X Offset	1
Y Offset	1



Stock box and part.

To locate the part within the stock

- Use the **Locate Part Within Stock** tool to set the location of the stock as follows:

Action	Setting
Object to Move	Move Stock
Z Alignment	Center
XY Alignment	Center

Create cutting tools

You will need three ball end cutting tools to machine this part. Create them with the following settings *(Tool menu: Create/Select Tool)*:

Name	Tool Length	Flute Length	Diameter
Ball 1 mm	90	10	1
Ball 2 mm	90	10	2
Ball 3 mm	90	10	3

Create a machine operation

The first milling operation for the second part is a horizontal roughing operation. This MOP will remove most of the excess material.

To define the horizontal roughing operation

1. Make the **3 mm ball end** mill the current tool *(Tool menu: Create/Select Tool)*.

   Verify the current tool in the status bar.

2. From the **3-Axis Milling** menu, select **Horizontal Roughing** *(3 Axis Milling menu: Horizontal Roughing)*.

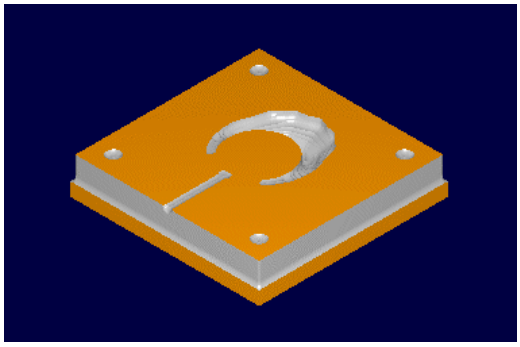
3 Set the cut parameters as follows:

Name	Value
Global Parameters	Intol: 0.03
	Outol: 0.03
	Stock: 0.3
Cut Pattern	Stock offset
Cut Direction	Climb
Offset: Pocket Start Point	Inside
Stepover Control	25% of tool diameter

4 Set the cut levels as follows:

Name	Value
Stepdown Control	Distance: 0.5
Cut Levels	Ordering: Level first
Cut Levels	Bottom: –6.5

5 **Generate** the operation, hide the part, hide the tool paths, and **Simulate** the operation to update the stock model.

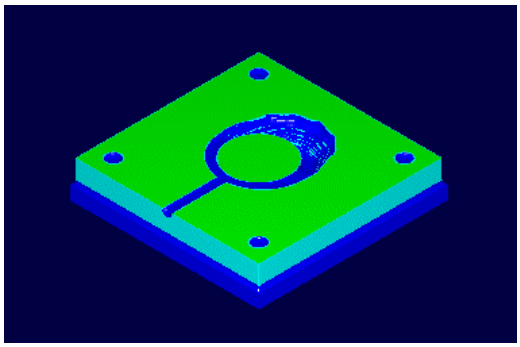


Horizontal roughing.

To check for excess stock

- Use the **Compare Part/Stock** tool to see where the excess stock material is.

  Use a **Tolerance Band** setting of **0.3**, the amount of excess stock left over after the roughing MOP. The green areas are finished while the dark blue areas indicate excess material thicker than 0.3 mm when compared to the ideal stock model.



Part to stock comparison.

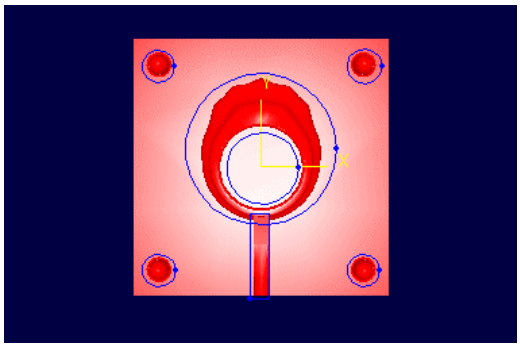
Define regions

Define machining regions to isolate the pockets of excess material for efficient removal in subsequent machining operations.

To define and set machining regions

1. Using the **Curves/Regions** toolbar, create the regions around the registration keys, ring area and gate area as shown below *(View menu: Toolbars > Curves/Regions Bar)*.

   Use the top view using the part model as a reference.



Create regions.

Define parallel finishing

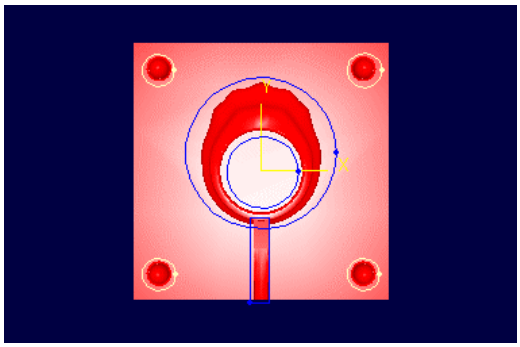
The next step is to set the active machining regions and current cutting tool, and then define a parallel finishing operation for the four spherical registration keys.

To define parallel finishing operations

1. Use the **Select Regions** tool to activate the regions around the registration keys as shown below.

Button	Function
	Select Regions

   The first parallel finishing operation will use these regions.



Select regions.

2 Use **Create/Select Tool** to make the **2-mm ball end** tool current *(Tool menu: Create/Select Tool)*.

Button	Function
	Create/Select Tool

Verify the selection in the status bar.

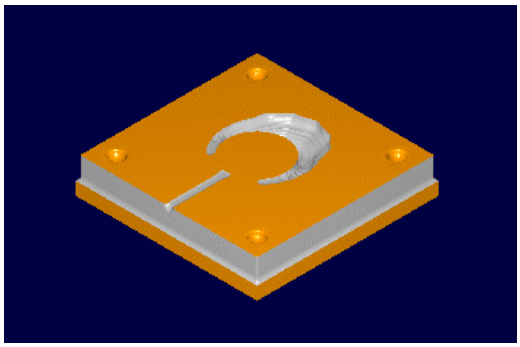
3 From the **3 Axis Machining** menu, select **Parallel Finishing** *(3 Axis Milling menu: Parallel Finishing)*.

Button	Function
	3-Axis Machining

Use the following parameters.

Name	Value
Global Parameters	Intol: 0.03
	Outol: 0.03
	Stock: 0
Cut Direction	Mixed
Start Side	Bottom
Angle of Cuts	0
Stepover Control	Distance: 0.1

4 **Generate** and **Simulate** the MOP to update the stock model.

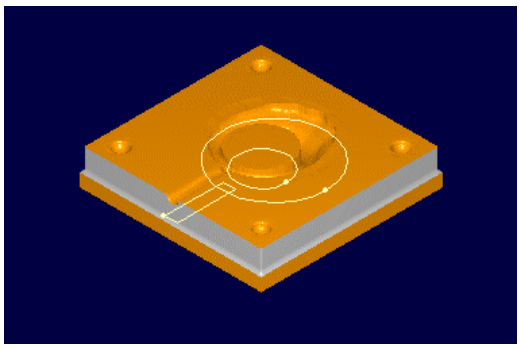


Parallel finishing for the registration key cavities.

To machine the ring and gate cavities

- Create another parallel finishing milling operation that uses the regions that surround the ring and gate cavities.

  Use the **2-mm ball end** tool and the same parameters and as the previous parallel finishing MOP.



Parallel finishing of the ring and gate cavities.

Define pencil tracing

A new milling strategy called *pencil trace* machining can be used either as a pre-finishing or a re-finishing operation. Pencil trace operations typically use a ball end cutter restricted to follow a path where two or more contact points are maintained with the part. This path typically follows the valleys and the corners of the part. Pencil trace can clean up the scallops left after a parallel finishing operation, or it can be used to remove excess material from valleys and corners.

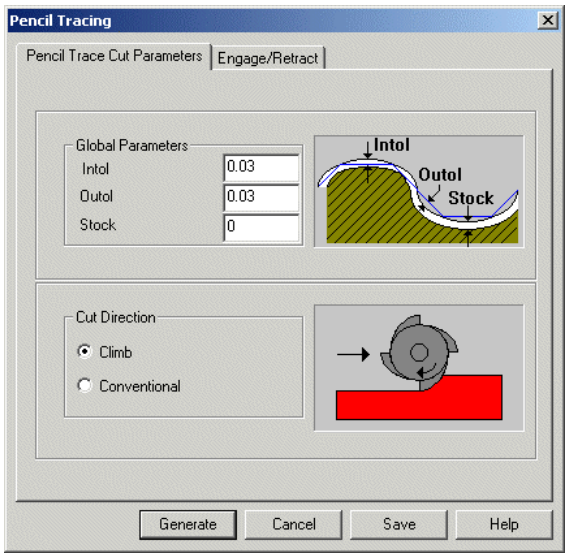
Warning: For any cutting strategy you use, consider the shape and material you are working on. If the tool is forced to take deep cuts and work with a very large amount of material on its blades, it creates a very dangerous situation. Tools and stock can shatter, sending sharp metal projectiles in all directions. Never overload a tool. It is much safer to take several shallow cuts than a few overly deep ones.

A pencil trace operation with a stock value of zero will generate a tool path that drives the cutter along the valleys and corners of the part model, thereby eliminating all the excess stock where the ball end mill has a double tangency condition. Double tangency is where the side and the point of the tool are cutting at the same time.

To define a pencil trace operation

1. Use the same **2-mm ball end** tool for the pencil trace operation *(Tool menu: Create/Select Tool)*.
2. From the **3-Axis Machining** tool, select **Pencil Tracing** *(3 Axis Milling menu: Pencil Tracing)*.

   You will use it as a pre-finishing operation. Set the parameters as shown below, then generate the MOP.



Pencil trace settings.

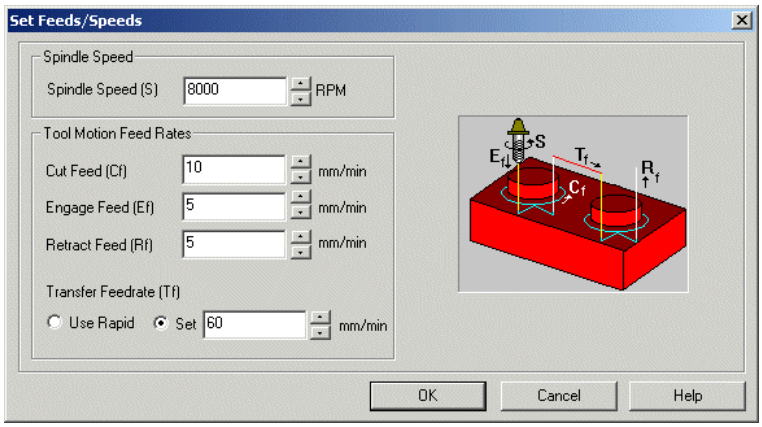
Set speeds and feeds

Before you update the stock model by simulating the MOP, change the speeds and feeds. This is not intended as a discussion on specific settings. In light of the hazards associated with the increased tool load from the dual contact point strategy used in pencil tracing, it is probably a good idea to reduce the feed rate.

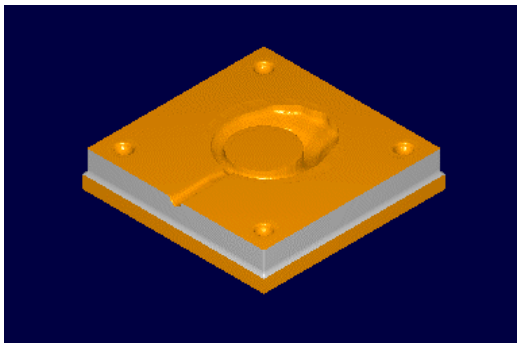
To set the feed rate

- In the **Set Feeds/Speeds** dialog box, reduce the feed rates to the settings shown below, **Generate** the tool paths, and **Simulate** to update the stock model *(Feeds/Speeds menu: Set Feeds/Speeds)*.

Button	Function
	Set Feeds/Speeds



Speeds and feeds settings.



Results of the pencil trace operation.

Final finishing

The final parallel finishing operations use the smallest cutter with a very small stepover. These operations will be confined to the ring cavity and gate regions. The only difference will be the **Angle of Cuts** setting. The two MOPS will be generated with the tool paths set 90 degrees to each other.

To define the final parallel finishing operations

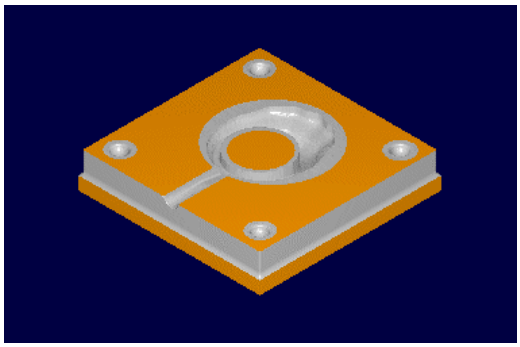
1. Set **Ball 1 mm** as the active tool *(Tool menu: Create/Select Tool)*.
2. From the **3-Axis Milling** menu, select **Parallel Finishing** *(3 Axis Milling menu: Parallel Finishing)*.

Use the following settings to generate the two MOPs.

Name	Value
Global Parameters	Intol: 0.03
	Outol: 0.03
	Stock: 0
Cut Direction	Mixed
Start Side	Bottom
Angle of Cuts	0
Stepover Control	Distance: 0.1

Name	Value
Global Parameters	Intol: 0.03
	Outol: 0.03
	Stock: 0
Cut Direction	Mixed
Start Side	Bottom
Angle of Cuts	90
Stepover Control	Distance: 0.1

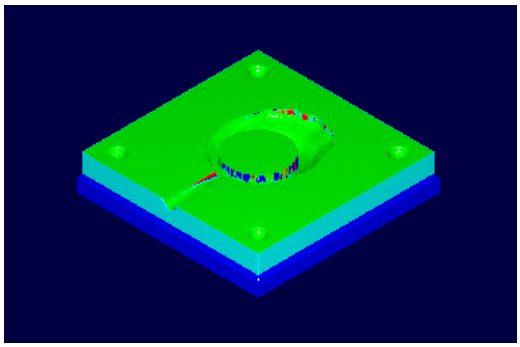
3 **Generate** the tool paths and **Simulate** the stock removal to update the stock model.



Final finishing results.

4 Check the part with the **Part/Stock Comparison** to see how well they match.

This looks like a good job, and will require very little, if any, hand finishing.



Part stock final comparison.

Generate the tool paths

Use the **Post Process** tool to generate the G-Code instructions for your mill and save the file *(Post Process menu: Post Process)*. (Note: This feature is disabled in the demo version of VisualMill.)

Button	Function
	Post Process

Congratulations!

You now have completed everything required to machine both halves of the ring mold. Hook up your machine and go!