Advanced Training Series

Modeling and Mold Making for Designers

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Introduction

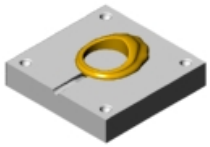
Rhinoceros and **VisualMill**, the latest advanced computer modeling and manufacturing programs, make it easy to:

- 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.

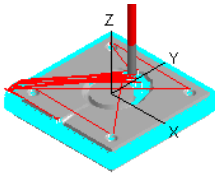
- **First, model the part.**

  Create a model of a ring shape in Rhino.



- **Second, model the mold.**

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



- **Third, create cutting instructions.**

  Import the model of the mold into VisualMill to create cutting instructions for a milling machine.

  The last step is to send the instructions to the machine to cut the part. This process varies according to the machine and is not covered in this tutorial.

On the CD

- The online *Modeling and Mold Making for Designers* tutorial file.
- A 64-page printable version of *Modeling and Mold Making for Designers*.
- The completed models used in the tutorial.
- **Rhinoceros** Evaluation version with exercises.
- **VisualMill** Evaluation version with additional tutorial and *Getting Started Guide*.

To use the tutorial

1. Put the CD into your CD drive.

   The opening menu lets you choose your next step.

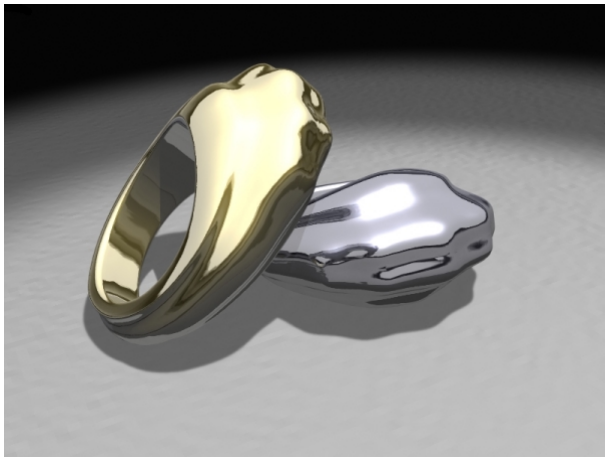
2. If you do not have the Rhino and VisualMill programs installed already, install the Evaluation versions of these programs that are included on this CD.

   This tutorial is advanced Rhino training. If you have not used Rhino before, go through the exercises included with the Rhino Evaluation version on this CD.

3. Start the online *Modeling and Mold Making for Designers* tutorial and work through the steps.

   If you would rather work from a printed version, print the PDF version of *Modeling and Mold Making for Designers* and use this to work through the steps.

We will start by designing the curves that are needed to build the surfaces of the ring we are going to produce. Then we will go all the way 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*

Mecsoft/VisualMill
VisualMill is a 3-axis solid/surface milling package, which seamlessly integrates fast, gouge-free toolpath generation with cut material simulation in an easy and fun to use package running on standard Windows hardware. *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. *http://www.rolanddga.com*

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

Part 1: Model the Ring in Rhino

Starting a command

Rhino has a very flexible and customizable interface. You can start commands three different ways depending on how you like to work.

You can:

- Use the pull-down menu.
- Click a toolbar button.
- Type a command at the command prompt.

At the beginning of each section, a chart will show you the button, the menu location, and command name in a table like the one below. Soon you will find a method that suits your style. Most people settle into one method that they use most of the time.

When you see the directions "Start the command," use the method you like best to activate the command.

Button	Command	Menu
	New	File > New
	Open	File > Open
	GridOptions	Tools > Options

Get started

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.

Button	Command	Menu
	New	File > New
	Open	File > Open
	GridOptions	Tools > Options

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. Start a new Rhino model.

2 In the **Template File** dialog box, select **4 viewports shaded.3dm** and click **OK**.

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**.

2 In the **Open** dialog box, select **Ring Model.3dm**, and click **OK**.

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

Set the grid snap spacing

1 From the **Tools** menu, click **Options**.

2 On the Grid tab, set the **Snap Spacing** to **0.5** and click **OK**.

3 On the **Status bar**, click **Snap** to turn snap on.

Draw the first curves

Button	Command	Menu
	Circle	Curve > Circle > Center, Radius
	InterpCrv	Curve > Free-form > Interpolate Points

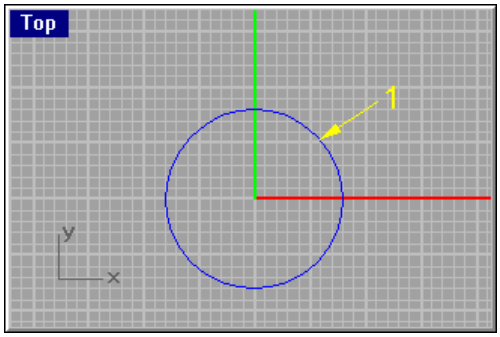
Draw curve 1

1 Start the **Circle** command.

2 At the **Center of circle (Vertical AroundCurve)** prompt, type **0,0,0**.

3 This sets the center of the circle at the origin point in space.

4 At the **Radius <1> (Diameter)** prompt, type **8** or drag the radius 8 units.



Draw curve 2

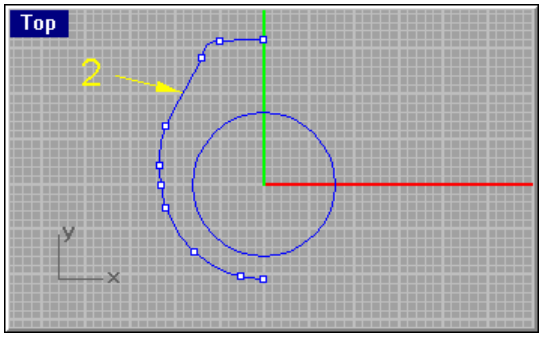
Draw the left half of the outline of the ring.

1 Start the **InterpCrv** command.

2 At the **Start of curve (Tangent Knots=SqrtChord Degree=3)** prompt, start the curve on a snap point on the centerline (y-axis).

3 At the **Next point of curve (Tangent Knots=SqrtChord Degree=3 Undo)** prompt, draw another point to define your curve

4 At the **Next point of curve. Press Enter when done (Close Tangent Knots=SqrtChord Degree=3 Undo)** prompts, continue to draw points, ending on a grid snap point on the center line.



Edit and mirror curve 2

Button	Command	Menu
	PtOn	Edit > Point Editing > Control Points On (F10)
	InterpCrv	Curve > Free-form > Interpolate Points
	Mirror	Transform > Mirror

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.

Edit the control points for curve 2

1 Turn on the control points for curve 2.

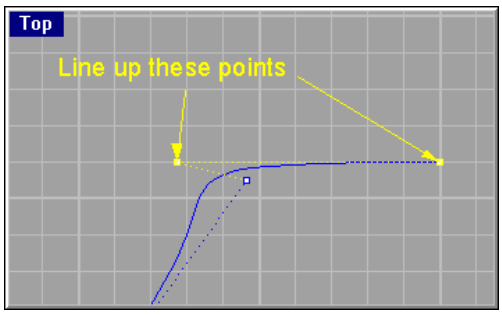
2 Look at each end of the curve.

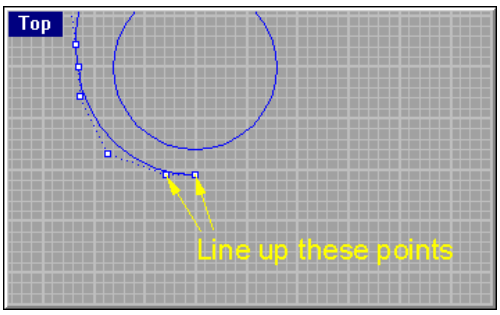
The important factor is that the control point at the end of the curve and the next control point are perfectly aligned horizontally.

If they are not, move the second control point using grid snap, a construction line, ortho or any other tools that you need to get the two points lined up.

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*.

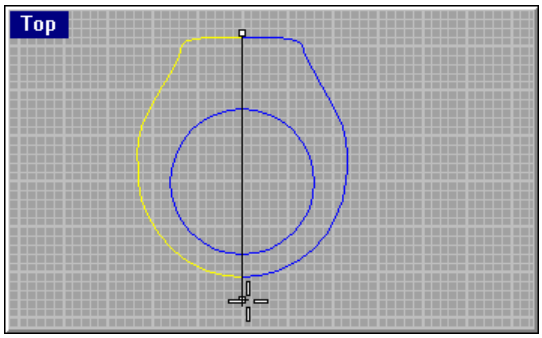




Mirror the curve

- **Mirror** the curve across the y-axis.

  Be sure to use grid snap so the ends of the curve touch each other exactly.

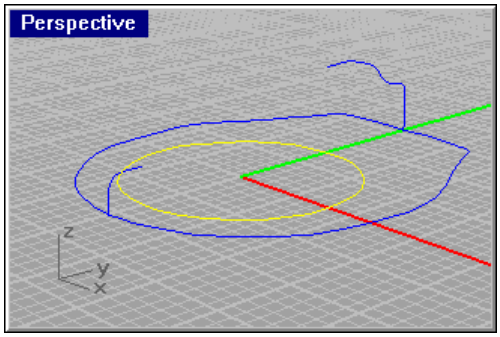


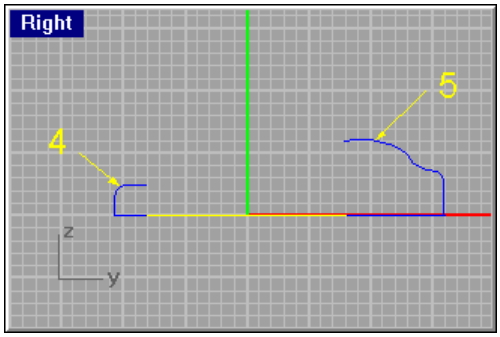
Draw curves to define the side profile

Button	Command	Menu
	InterpCrv	Curve > Free-form > Interpolate Points

Now design two more curves. Curves 4 and 5 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 we have drawn so far. In the **Perspective** viewport you can see that the curves start at the endpoint of the mirrored curves. In the **Top** viewport, you can see that they end directly over the inner circle.





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.

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

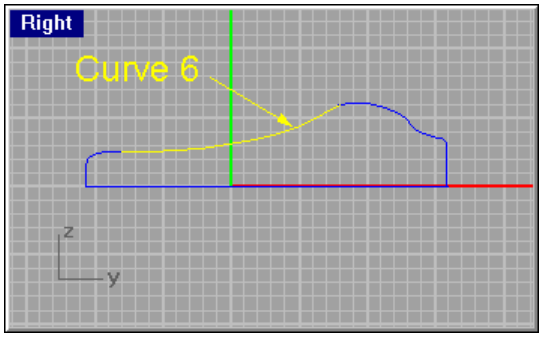
   Use grid snap for the start of the curve to accurately place the beginning and end points.

   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 to line the end of the curve up with the circle.

Connect curves 4 and 5

Button	Command	Menu
	InterpCrv	Curve > Free-form > Interpolate Points
	Mirror	Transform > Mirror

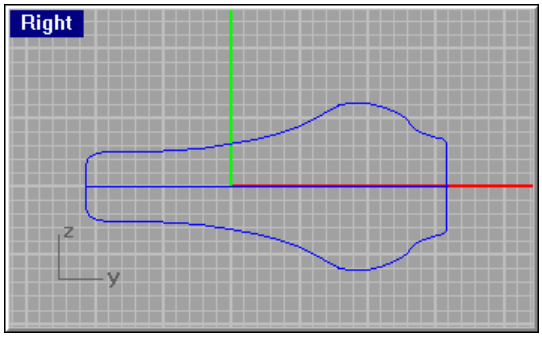
- To complete the side profile of the ring, design curve 6 to connect curves 4 and 5.



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

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

For a better idea of how your design is developing, it is advisable to mirror curves 4, 5, and 6 across the world y-axes in the right viewport so that you can see the complete ring profile as seen from the right side. 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.



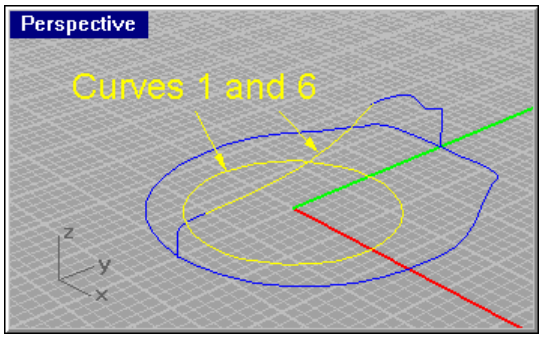
Create edge curve for ring

Button	Command	Menu
	Crv2View	Curve > From 2 Views
	Join	Edit > Join

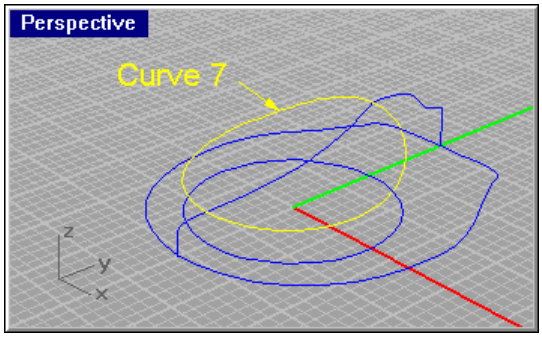
You are now going to create a curve that defines the inner edge of the ring. This curve is shaped the way curve 6 is in the right view and the way curve 1 is in the top view. It's a sort of bent circle. Rhino has a command that creates this curve based on these curves.

Create a curve from curves 1 and 6

1 Select curves 1 and 6.



2 Use the **Crv2View** command to create curve 7.



Curve 7 comes in three pieces and needs to be joined.

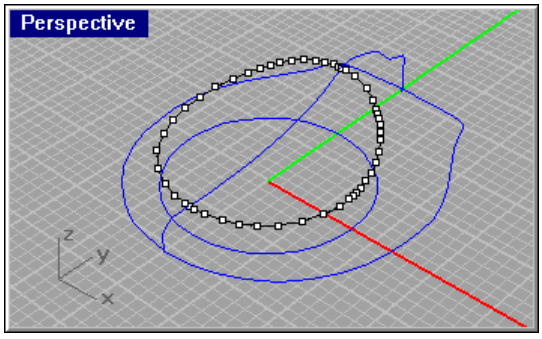
3 Use the **Join** command to join the three parts of the curve.

4 You want this curve to be a joined loop. If Rhino prompts to accept a small distance between the curve ends, click Yes.

Rebuild curve 7 with fewer control points

Button	Command	Menu
	PtOn	Edit > Point Editing > Control Points On
	Rebuild	Curve > Edit Tools > Rebuild

Curve 7 has a very high point count. Turn on the control points to see them.

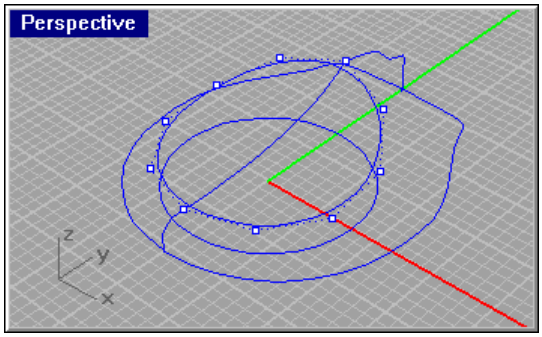


There are probably too many points in this curve, so the next step is to reduce the number of points.

Rebuild the curve

1. Press Esc to turn off the control points.
2. Use the **Rebuild** command to set the number of control points to 10.

   You can 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.

Draw the side profile curves

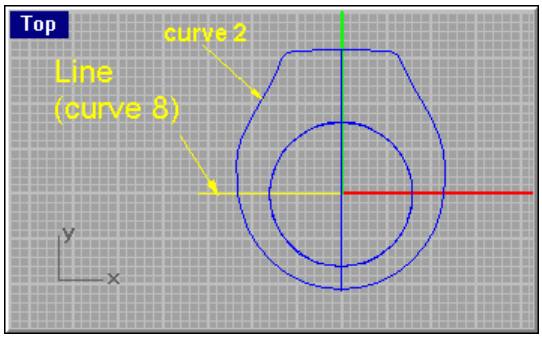
Button	Command	Menu
	Line	Curve > Line > Single Line
	InterpCrv	Curve > Free-form > Control Points
	Mirror	Transform > Mirror

You are now going to 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 we will start the side profile curve.

Draw a construction line

- Use the **Line** command, with grid snap or ortho to draw a horizontal line from the center of the original circle to out beyond curve 2 as shown.

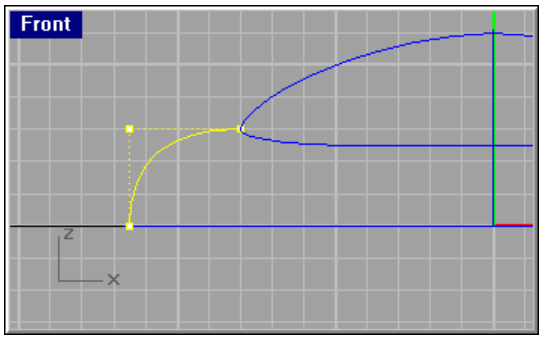


Draw the side profile curves

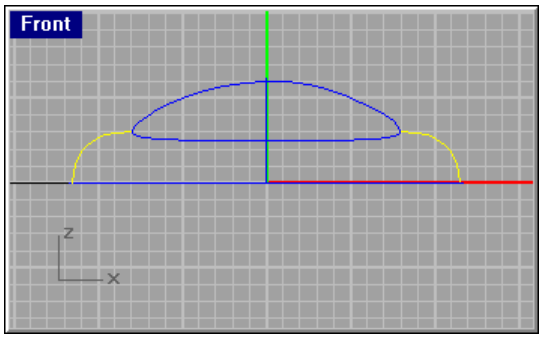
1 Use the **Curve** command and the Intersection object snap to draw a curve that starts at the intersection of curve 2 and your construction line.

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

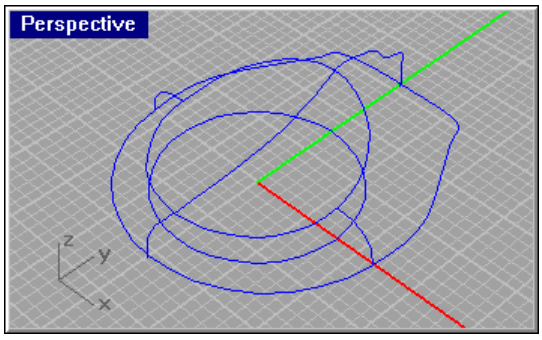
Again, the control points should line up. In the example curve, there are only three control points, which form a right angle.



2 **Mirror** the curve in the Front or Top viewport using the center (0,0) as the mirror point.



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



Curve construction

It is very important to construct the curves, to check their symmetry and how they look. Spend some time changing them or shaping them. What we are about to build depends largely on the construction of the original curves.

Also, it is important to know, there is not only one strategy in building a surface. There are many different ones! You can use your imagination to create the shapes you want. However, it is good to keep in mind the limitations of the processes you are going to use to manufacture your model in the real world.

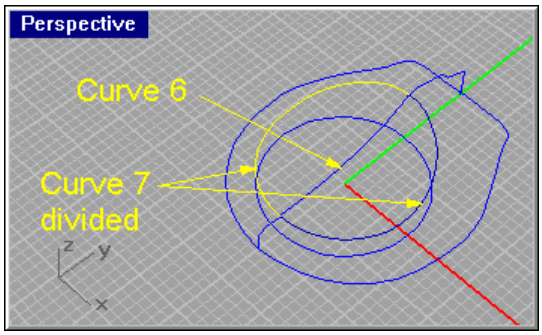
Create the first surfaces

Button	Command	Menu
	Split	Edit > Split
	NetworkSrf	Surface > From Curve Network
	Mirror	Transform > Mirror

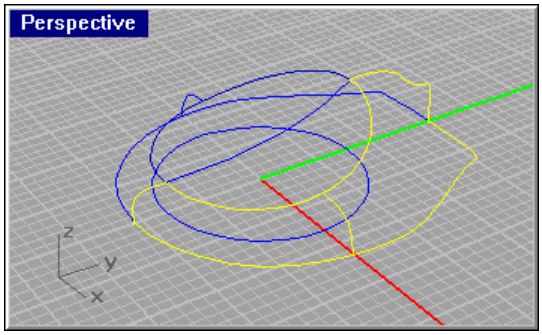
We are about to build our first surface, which will show a quarter of our ring.

1 Use the **Split** command to divide curve 7 into two halves.

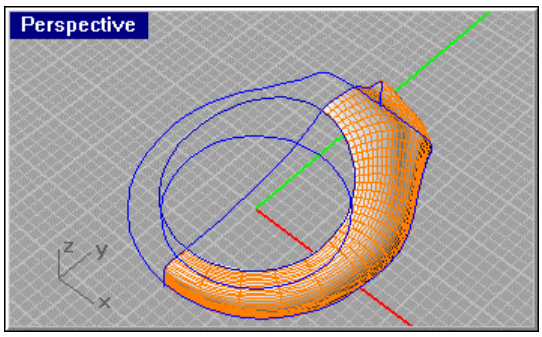
Use curve 6 as the splitting object.



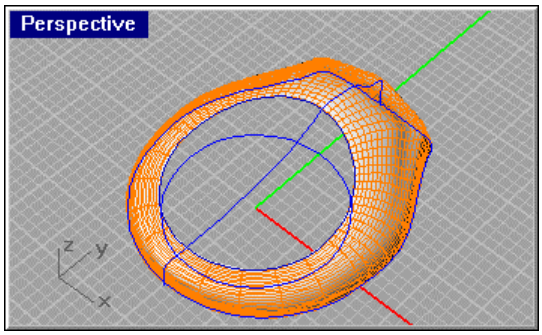
2 Select the curves shown in yellow below.



3 Use the **NetworkSrf** command to create a surface from the selected curves.



4 Use the **Mirror** command to create the other half of the outside ring surface.



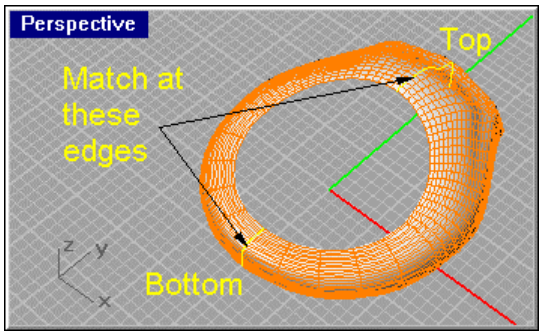
Match the surface edges

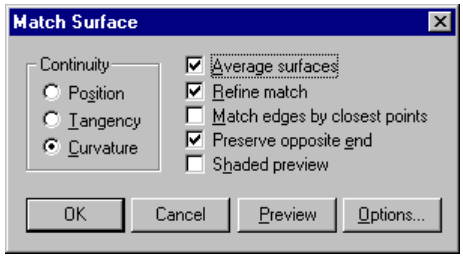
Button	Command	Menu
	MatchSrf	Surface > Edit Tools > Match

1 To ensure a smooth continuous surface across the two mirrored halves, use the **MatchSrf** command to match the edges where the two mirrored surfaces touch each other.

This happens in two places.

For the edges at the top part of the ring shown below, use these options in the **Match Surface** dialog box: under **Continuity**, choose **Curvature**; check **Average surfaces**, **Refine match**, and **Preserve opposite end**.

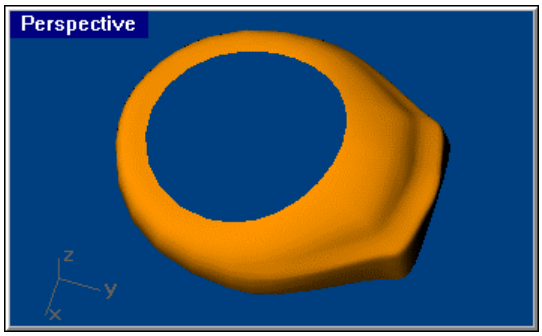




You will probably not actually see the results of this command as any changes are likely to be subtle.

2 Use the **MatchSrf** command again for the bottom edge. This time, under **Continuity**, choose **Tangency**; then check **Average surfaces**, **Refine match**, and **Preserve opposite ends**.

With a shaded preview, we can have an idea of what the result will look like.



You now have two surfaces, which are not joined, but they have the kind of *continuity* we were looking for.

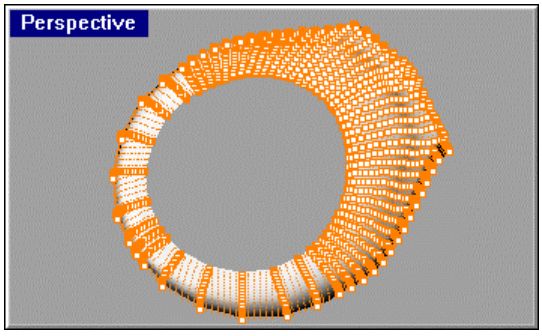
Merge the two ring half surfaces into one

Button	Command	Menu
	MergeSrf	Surface > Edit Tools > Merge
	PtOn	Edit > Point Editing > Control Points On (F10)
	RebuildSrf	Surface > Edit Tools > Rebuild

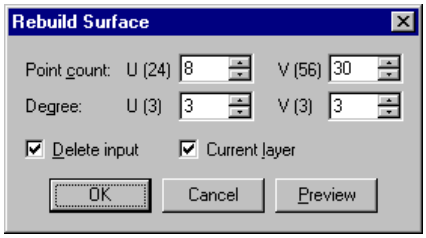
1. Use the **MergeSrf** command to make the two surfaces into one.
2. At the **Select untrimmed surface to merge (Tolerance Smooth=Yes)** prompt, type **S** to change the **Smooth** option to **No**.

   Since the continuity of the two surfaces have been already defined you don't need the MergeSrf command to automatically smooth the surface. We established the continuity manually with MatchSrf so we could set the continuity of the two edges separately.

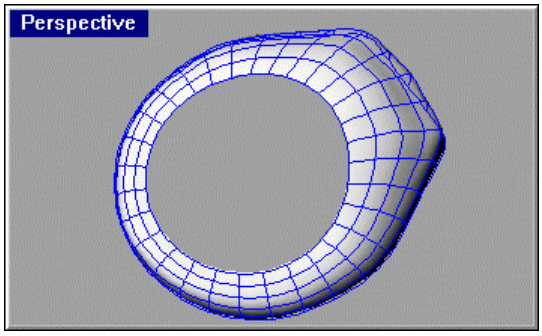
   The new surface has a great number of isoparms (the curves that build the wireframe). This indicates a large number of control points on the surface.
3. Select the surface and press F10 to turn on control points to see the density. For our purposes, we don't need so many control points.



4. To reduce the number of control points, use the **RebuildSrf** command.
5. Set the number of points to **8** for the **u-direction** and **30** for the **v-direction**.



You can now see that the new surface has a much less dense isoparm spacing.



The reason for the rebuilding of the surface is so we can make some free-form modifications to it. We want to use enough control points to give good control on the free-form modification, but not so high that it will be almost impossible to handle them. This surface is a good compromise.

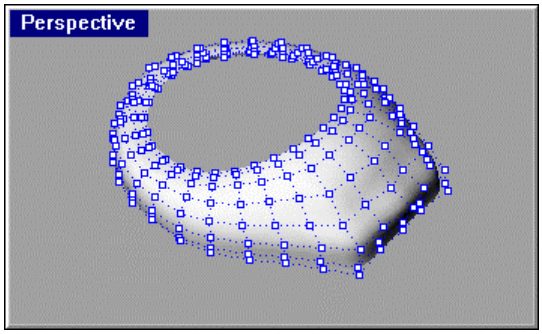
6 **Mirror** the surface at any stage to see what the other half of the ring will look like once it is finished.

Edit control points to shape the surface

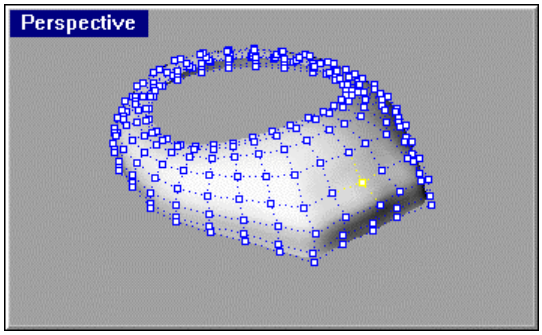
Button	Command	Menu
	PtOn	Edit > Point Editing > Control Points On (F10)
	SelV	Edit > Select Control Points > Select V
	Scale2D	Transform > Scale > Scale 2-D

1 Turn on the control points of the surface.

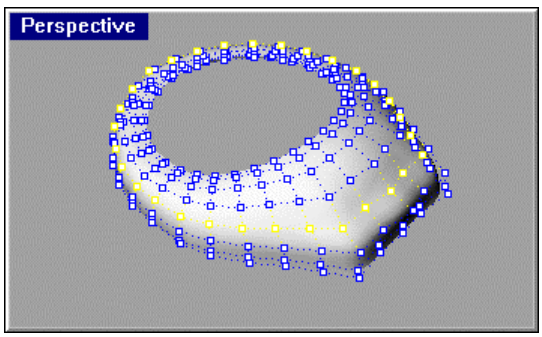
As you can see there are plenty to allow for modifications.



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



3 Use the **SelV** command to select all the control points in the same v-direction row.



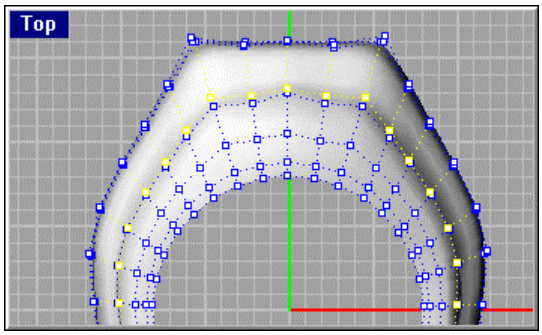
4 Use the **Scale2D** command to move all the control points in this row toward the center of the ring.

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

This is the center of the inner circle.

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

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.



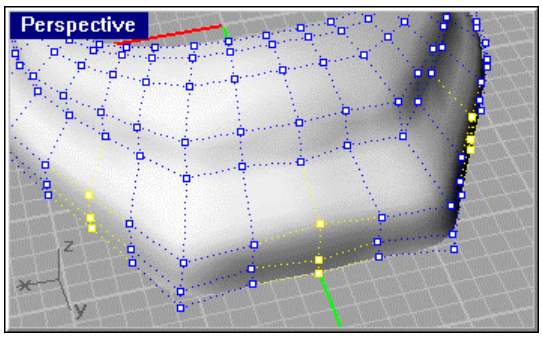
Add further interest to the surface

Button	Command	Menu
	PtOn	Edit > Point Editing > Control Points On (F10)
	SelV	Edit > Select Control Points > Select V
	Scale2D	Transform > Scale > Scale 2-D
	ShadeOptions	Tools > Options > Shade tab > Use OpenGL

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.

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

Tip: Hold the **Shift** key while selecting the points using a window. Rotate the view to see the other side of the ring.

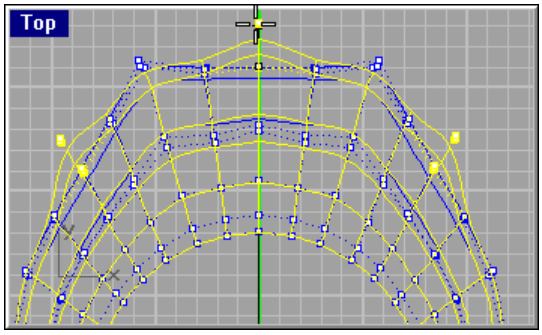


2 Use the **Scale2D** command to move all the control points in this row away from the center of the ring.

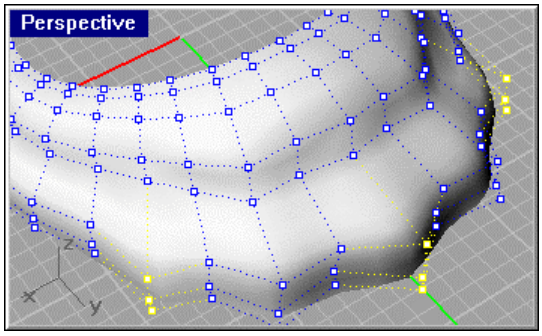
3 Again, for the **Origin point**, type **0,0,0**.

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

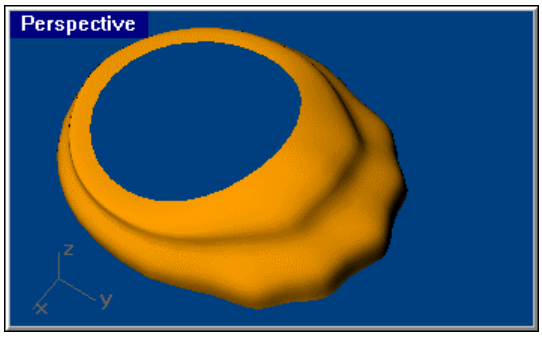


This creates three bulges in the ring surface.



6 **Shade** the viewport.

Note: To achieve the gold effect with shading in this image, we have changed the layer color to gold and used **OpenGL shading** to use the layer color.



We can see that the overall shape of the ring, especially if seen from the top and 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 giving a try.

Feel free to try any solution you may find interesting or useful to modify the control points.

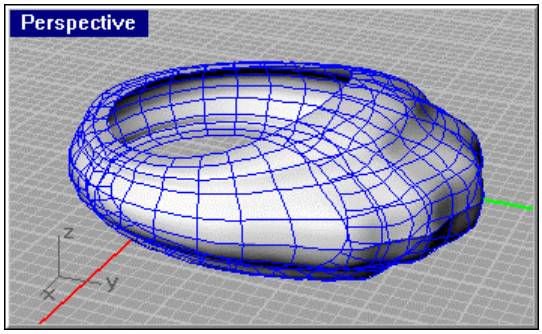
This is just one of many possibilities.

Create the other half of the ring

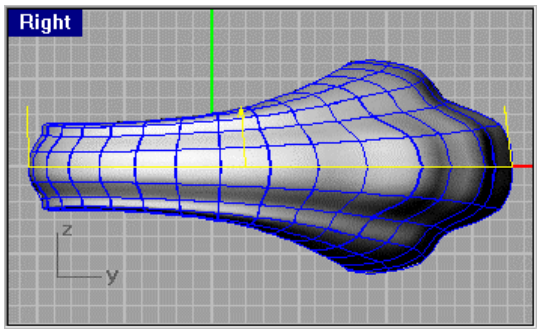
Button	Command	Menu
	Mirror	Transform > Mirror
	MatchSrf	Surface > Edit Tools > Match
	MergeSrf	Surface > Edit Tools > Merge

1 Use the **Mirror** command to create the other ring half.

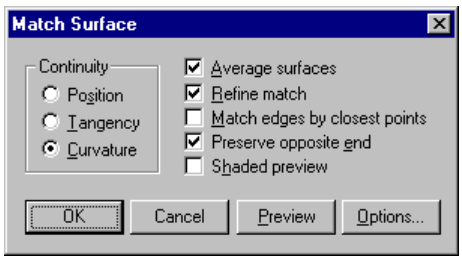
Use the Front or Right viewport.



2 Use the **MatchSrf** command to control how the edges will remain smooth across the transition.



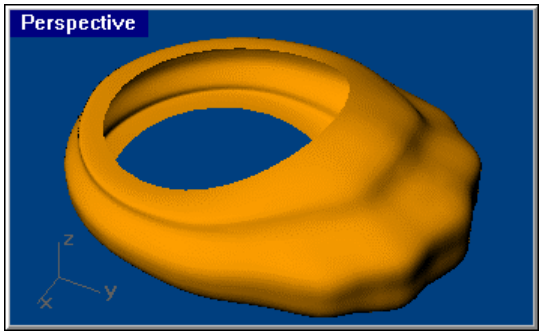
For the edges at the top part of the ring shown below, use these options in the **Match Surface** dialog box: under **Continuity**, choose **Curvature**; check **Average surfaces**, **Refine match**, and **Preserve opposite end**.



3 Use the **MergeSrf** command to make the two surfaces into one.

4 At the **Select untrimmed surface to merge (Tolerance Smooth=Yes)** prompt, type **S** to change the **Smooth** option to **No**.

Since the continuity of the two surfaces have been already defined you don't need the MergeSrf command to automatically smooth the surface. We established the continuity manually with MatchSrf so we could set the continuity of the two edges separately.

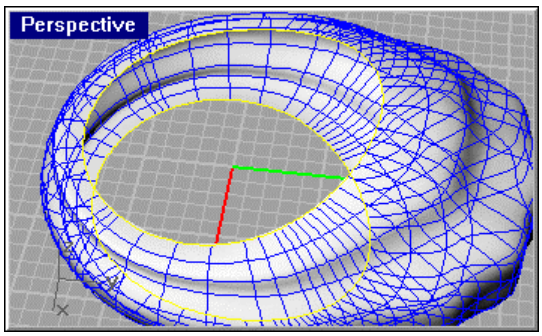


Finish the interior surface

Button	Command	Menu
	Loft	Surface > Loft
	FilletSrf	Surface > Fillet
	Join	Edit > Join
	Emap	Analyze > Surface > Environment Map
	Turntable	

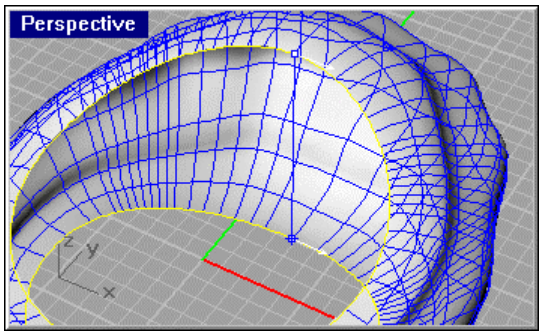
You now only have the interior surface of the ring to finish. Use the **Loft** command to complete this surface.

1 Start the **Loft** command and select the two edges of the surface as shown below.

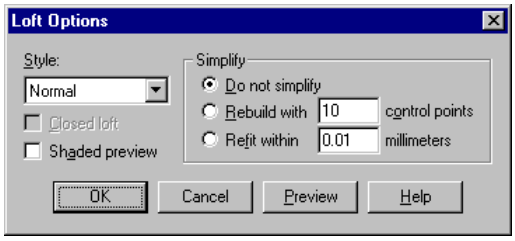


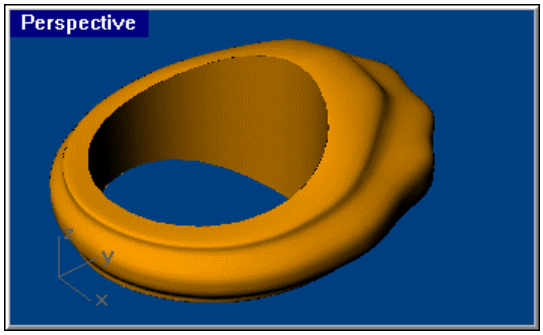
2 At the **Select seam point to adjust. Press Enter when done (FlipDirection Automatic Natural)** prompt, type **A** for automatic adjustment.

You will see the two arrows, one for each curve, going in the same direction, at corresponding points.

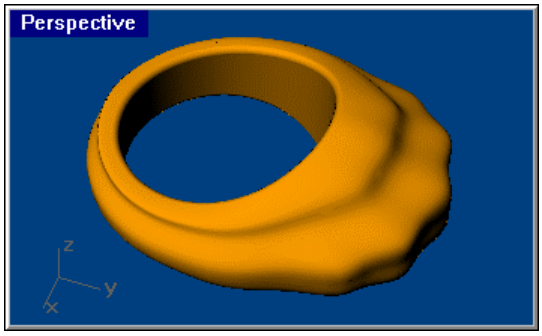


3 Accept the default **Loft Options**.





4 As a clean up of the finished product use the **FilletSrf** 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.



5 **Join** all the parts to create one object from the interior loft surface, the fillet surfaces, and the main ring surface.

Examine the finished ring

We can now look at the ring in it's entirety using the **Emap** command.

1 Try **polished_gold.jpg** to represent the material to be simulated.

2 Use the **Turntable** command to view the ring.

Any modification to our finished piece should be done now, because we are next going concentrate on making the mold. It would be too bad to end up with a beautiful mold of an ugly piece.

Make a copy of the ring to save

Button	Command	Menu
	Copy	Edit > Copy
	ChangeLayer	Edit > Layers > Change Object Layer

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

1. Use the **Copy** command with the **InPlace** option to create a copy of the ring.
2. Use the **ChangeLayer** command to place the ring copy on a new layer.
3. Turn the new layer off.

Trim away half the ring

Button	Command	Menu
	CutPlane	Surface > Rectangle > Cutting Plane
	Trim	Edit > Trim

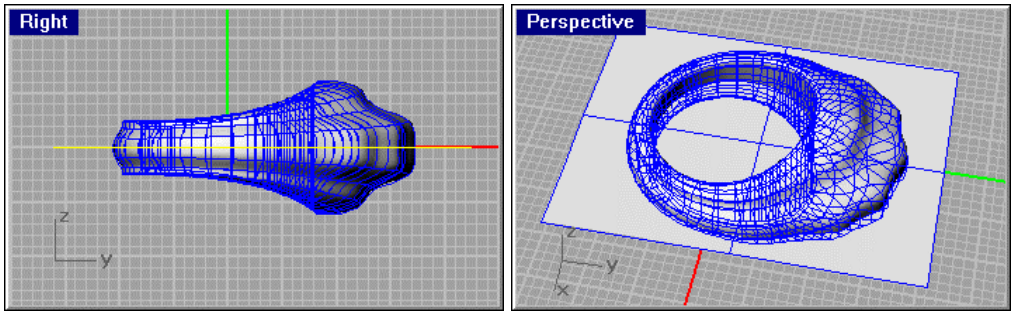
We need to create a surface that will subtract an amount of volume from the thick part of the ring in order to make it hollow.

To accomplish this task, one of the strategies we will use is to create section curves of the ring surface. Since the upper and the lower part of the ring are evidently symmetrical, we will do this just on the lower half.

Before we start, we will trim off half the ring. We created the whole ring so we could complete the design, but we only need half the ring to make a mold.

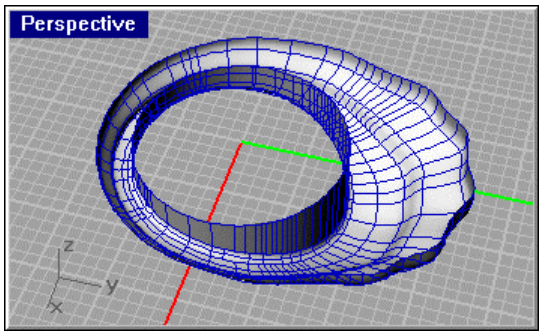
Split the ring

1. In the **Right** viewport, use the **CutPlane** command to draw a cutting plane along the world y-axis that extends beyond the ring.



2. Use the **Trim** command to trim away the top half of the ring with the cutting plane.

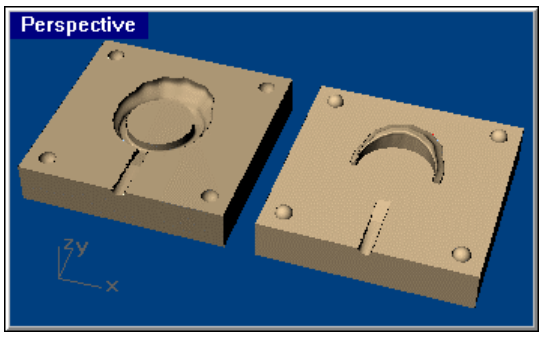
3 Delete the cutting plane surface.



Hollow out the ring

The purpose of this tutorial is to make a two-part mold that will let us 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.

To better describe our goal, we are going to create a box from which we are going to "subtract" a part, leaving a hollow space that will be exactly shaped like half the ring. We don't want to use the entire ring as it is now, this would make a solid lump of metal that would be so heavy that a lot of handwork would be needed to bring it to an acceptable weight. This would be time consuming and definitely not cost effective.



For the next part of this tutorial, we are going to be working on creating a "hollow" in the ring. We want to produce a mold that will allow us to have control over the thickness of the finished piece. A goldsmith will then have to weld the two parts and complete the finishing process.

It should be enough to offset the surface that defines our half ring a given amount towards the inside. Unfortunately, the solution is not that easy. For several geometrical reasons, offsetting surfaces that have complex curvatures may cause self-intersecting surfaces.

This is unacceptable, so we will have to find an alternative solution in order to build our surface.

Create curves on the ring surface

Button	Command	Menu
	Section	Curve > From Objects > Section

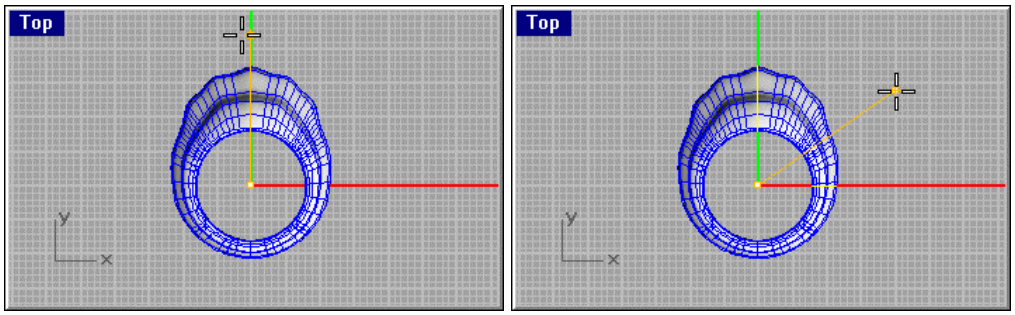
First we need to make curves that lie on our ring surface. Then we will offset these curves by 1 mm toward the inside. These inner curves will be used to create the "hollowing" surface.

1 Use the **Section** command to create curves that radiate out from the origin point.

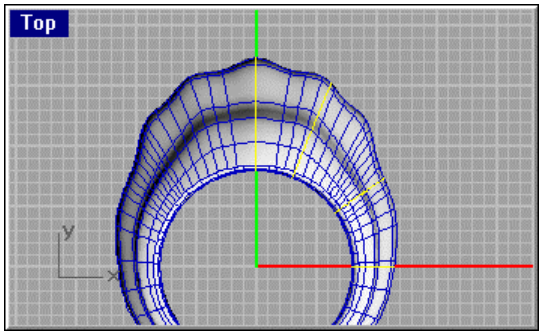
2 At the **Select objects for sections** prompt, select the ring half.

3 In the **Top** viewport at each **Start of section prompt**, type **0** 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.



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



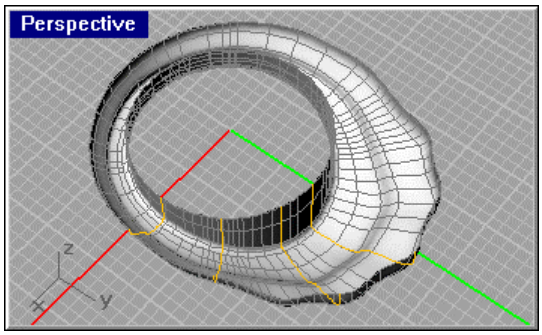
Offset the curves

Button	Command	Menu
Lock	Lock	Edit > Visibility > Lock
	CPlaneToObject	View > Set CPlane > To Object
	Offset	Curve > Offset
	CPlaneTop	View > Set CPlane > World Top
	Mirror	Transform > Mirror

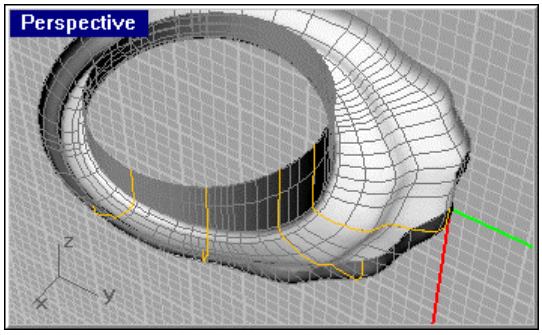
Now we are going to offset these sections to obtain smaller curves and from these build a surface inside the ring.

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

1 Before you start, use the **Lock** command on the ring surface so you can see it, but not select it.



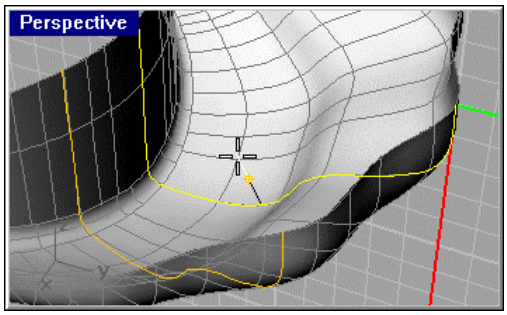
2 In the **Perspective** viewport, use the **CPlaneToObject** command to align the construction plane with the first curve.

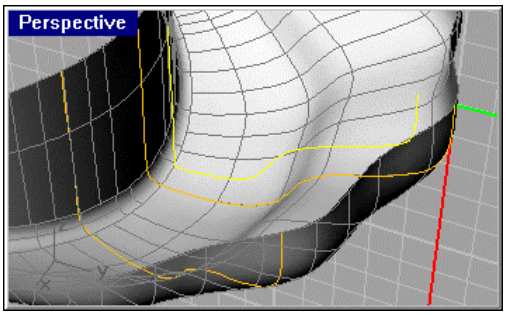


3 Use the **Offset** command to create the inside curve.

4 At the **Select curve to offset (Distance=1 Corner=Sharp ThroughPoint)** prompt, type **D** to set the distance to **1** if it is not already set.

5 At the **Side to offset (Distance=1 Corner=Sharp ThroughPoint)** prompt, be sure the marker is pointing to the inside of the ring.

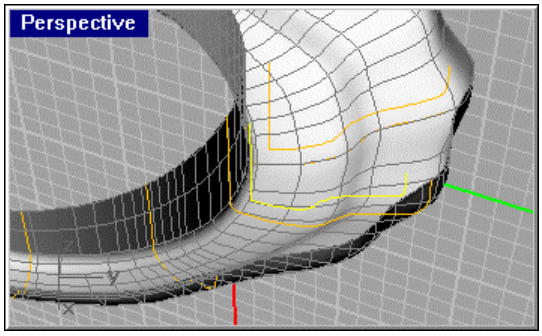




The yellow curve shown above is an offset of the section curve. Note that there is no self-intersection on offset curves.

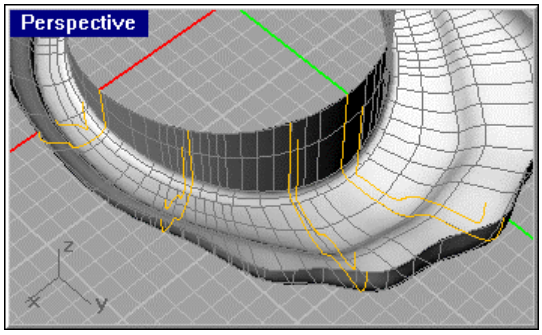
6 Use the **CPlaneToObject** command again to change the construction plane to the next section curve.

7 Use the **Offset** command to create the next inside curve.



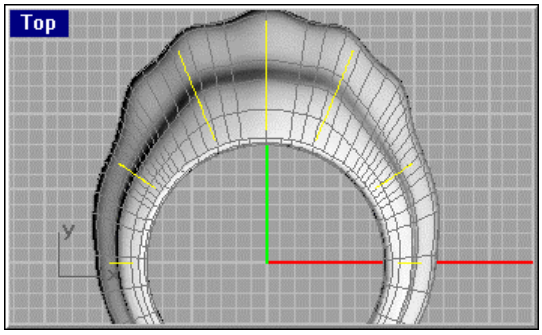
8 Repeat steps 6 and 7 for the other two curves.

9 When you have offset all the curves, use the **CPlaneTop** command to set the construction plane back to it's "normal" position.



10 Use the **Mirror** command in the **Top** viewport to mirror the new offset curves to the other side of the ring.

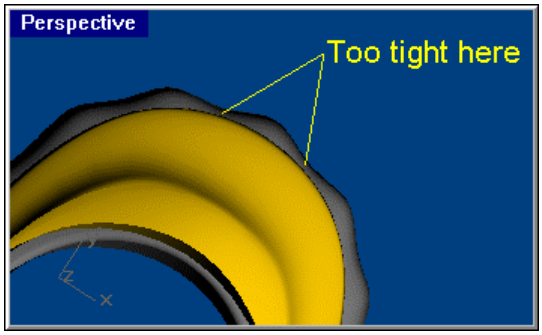
Do not mirror the curve that lies on the y-axis.



Create the inner surface

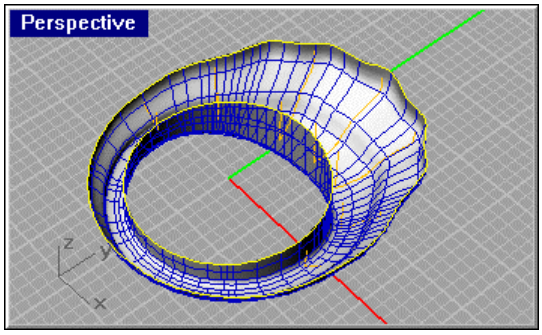
Button	Command	Menu
Lock	Unlock Right-click button	Edit > Visibility > Unlock
	Offset	Curve > Offset
Hide	Hide	Edit > Visibility > Hide
Trim	Trim	Edit > Trim
	NetworkSrf	Surface > From Curve Network

Having all the curves in the right position, one of the strategies we could use to build this surface would be to "loft" all the offset curves. This might be a good idea, but in this particular case we could have a problem. The distance between the outside edge of the half ring's surface and the edge of the lofted surface is not constant and at certain points the distance between the inner and outer surfaces is too small. The metal in this are would be thin.



Two solutions might be suitable at this point: one is to make more sections of the half ring's surface and work on them as we did to obtain more offset curves exactly where the distance between the borders gets to it's minimum value. This would work, but it would take a long time.

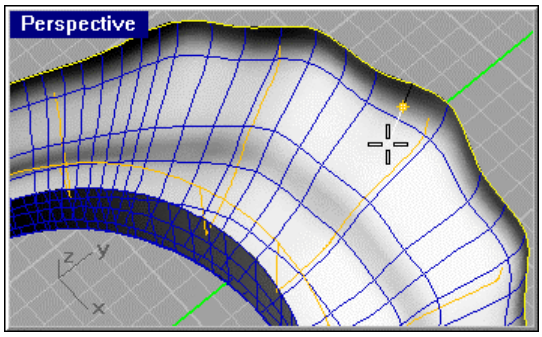
Our choice is to offset the borders of the ring surface (internal and external borders shown in yellow), 1 mm to meet the end points of the offset curves.



We will then use the curves to help define the surface.

1. **Unlock** the polysurface.
2. Use the **Offset** command to create the edge curves for the inner surface.

   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.

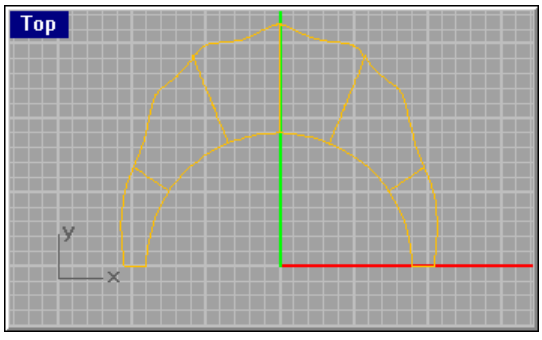


3. **Hide** the ring surface.
4. Use the two smallest offset curves at the x-axis to **Trim** the offset border curves since we will use only part of them.

   If the **Trim** command does not work, it is because the end points of the offset curves are not close enough to the curves that have to be trimmed.

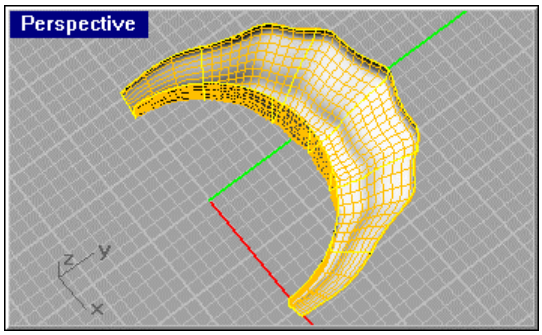
   There are several possible solutions to this, but the easiest is to draw a line in the Top viewport along the x-axis that overlaps the curves and use that for trimming.

   You want the results to look like this.



5 Select all these curves and use the **NetworkSrf** command to make the inner surface.

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



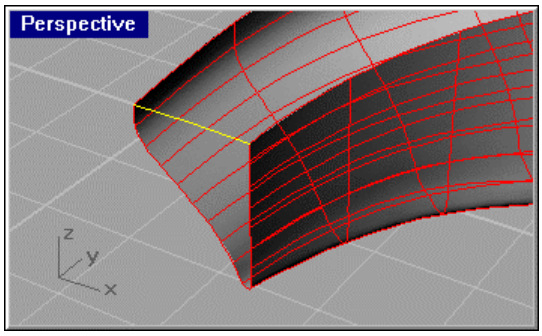
Create ends and join the inner surface

Button	Command	Menu
	Line	Curve > Line > Single Line
	EdgeSrf	Surface > Edge Curves
	Join	Edit > Join

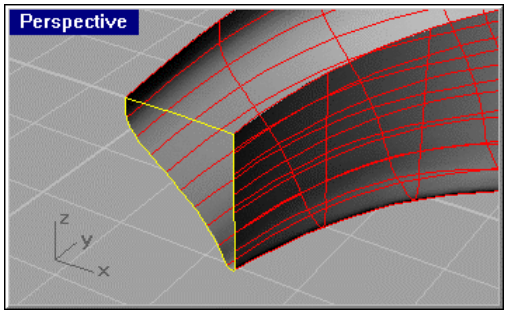
We need to fill the surface at the ends.

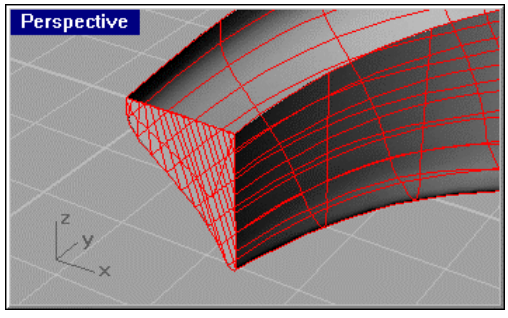
1 Draw a **Line** to connect the ending points of the surface.

Use the Endpoint object snap.



2 Use the **EdgeSrf** command selecting the line and the surface edge to create the end.





3 Repeat this for the other side.

4 Use the **Join** command to make these three surfaces into one.

5 Turn off the layer for the hollow.

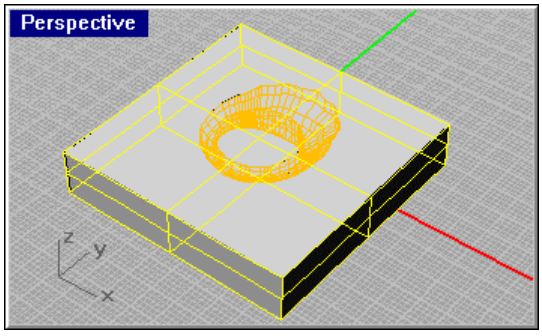
Create the first mold part

Button	Command	Menu
	Box	Solid > Box > Corner to Corner, Height
	ExtractSrf	Solid > Extract Surface
	Hide	Edit > Visibility > Hide
	Trim	Edit > Trim
	Show (Right-click button)	Edit > Visibility > Show
	Join	Edit > Join
	SphereD	Solid > Sphere > Diameter
	BooleanDifference	Solid > Difference

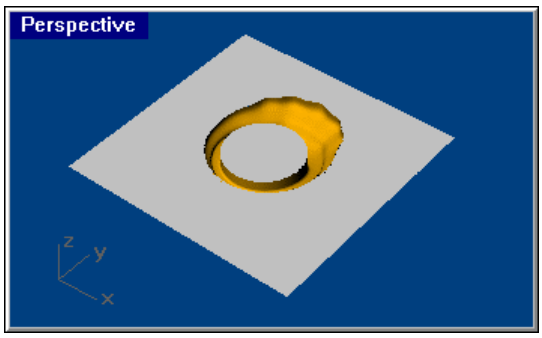
The mold will be created in two parts: one half for the ring surface and one half for the "hollow" surface. We are now going to build the ring surface part of the mold.

1. Turn on the layer with the ring surface.
2. Use the **Box** command to create a box with it's top face lying on the construction plane.

   The box has to be deep enough to enclose the ring surface. Check the size in the front viewport.

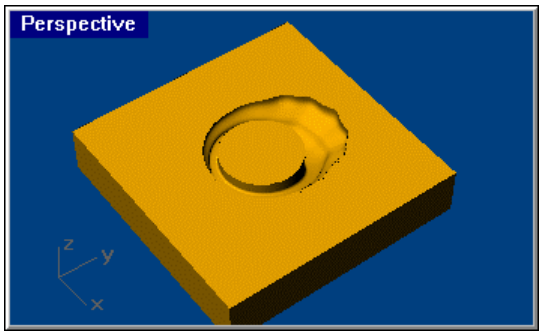


3. Use the **ExtractSrf** command to separate the top face of the box.
4. **Hide** the lower part of the box to get it out of your way temporarily.
5. Use the **Trim** command to trim out the area where the ring surface is.



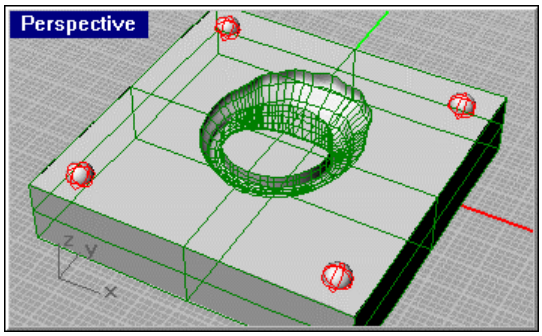
6. **Show** the lower part of the box.

7 Use the **Join** command to join all the parts together again.



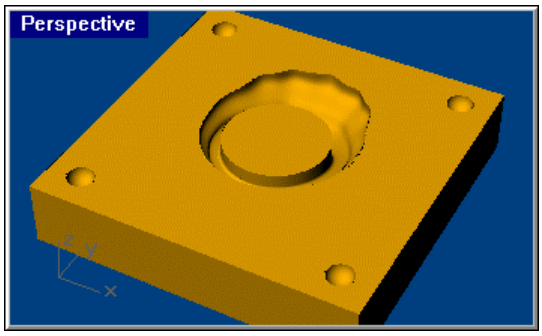
8 Use the **SphereD** command to create four registration keys (notches) at the corners of the mold half.

Use the grid snap to draw the spheres with a diameter of 4 mm.



9 Use the **BooleanDifference** command to cut the registration keys into the mold.

This helps us get a perfect alignment between this part of the mold and the other half that we will make next. The second half will have four protruding half spheres that correspond to the four notches.

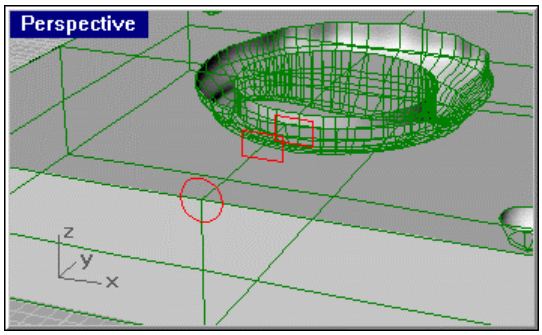


Create the gate

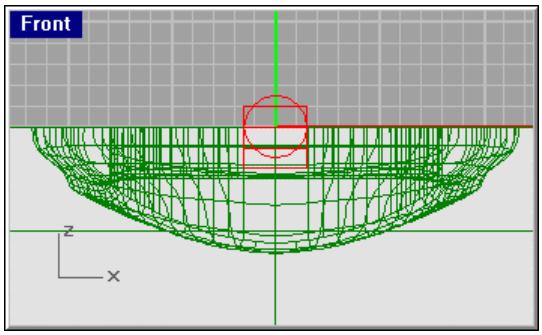
Button	Command	Menu
	Circle	Curve > Circle > Center, Radius
	RectangleCen	Curve > Rectangle > Center, Corner
	Loft	Surface > Loft
	Cap	Solid > Cap Planar Holes
	BooleanDifference	Solid > Difference
	FilletEdge	Solid > Fillet Edge

There is one missing part, which is the gate. This is the spout where the wax will be injected into the mold.

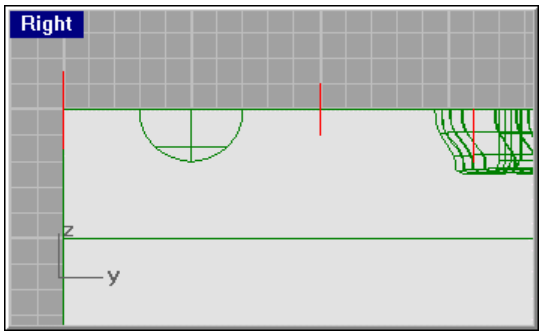
To build this we are going to use the three closed curves as shown below.



Here's what the curves look like in the Front viewport. We are using a circle and two rectangles.



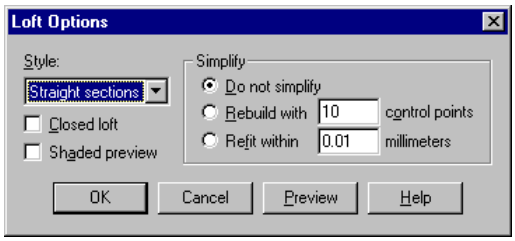
The rectangle closest to the ring is placed slightly below the other curves so that the wax will reach the part to be filled. This is visible in the Right viewport.

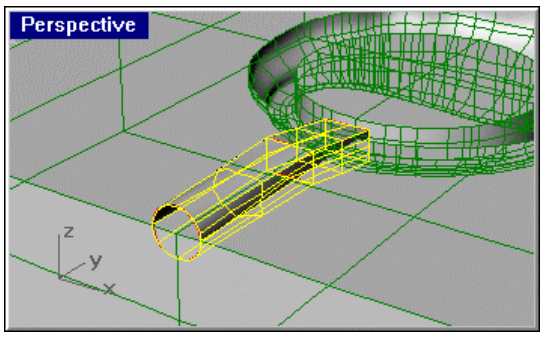


Considering the geometry of the piece, the rectangular shape is used to eliminate undercuts.

1. Select the three closed curves.
2. Use the **Loft** command to create a surface that connects them.

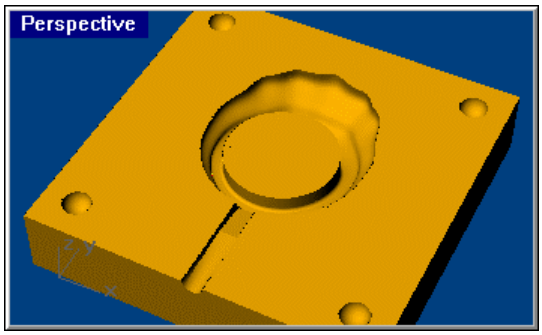
   Use the *Straight sections* style option.





3. Now use the **Cap** command to close off the ends of the gate surface and make it into a solid.

   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 two closed solids.
4. Make a copy of the gate solid and place it on another layer and turn that layer off. You will use this later to create the gate in the second mold part.
5. Use the **BooleanDifference** command to subtract the gate surface from the mold box.

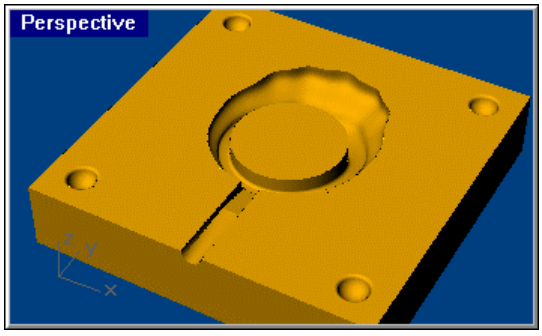


Round the corners of the registration notches

There is only a very little touch-up we can add to our half mold and that is filleting where the four little hemispheres join the upper face of the mold. The reason for this is to allow easier coupling between the two parts of the mold.

- Use the **FilletEdge** command to create the rounded edge of the notches.

  The fillet radius is **0.5** mm.

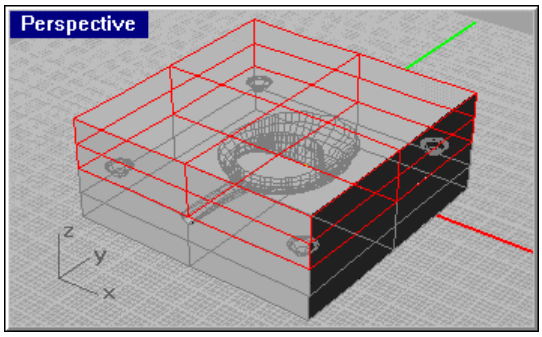


This part of the mold is complete.

Create the second mold part

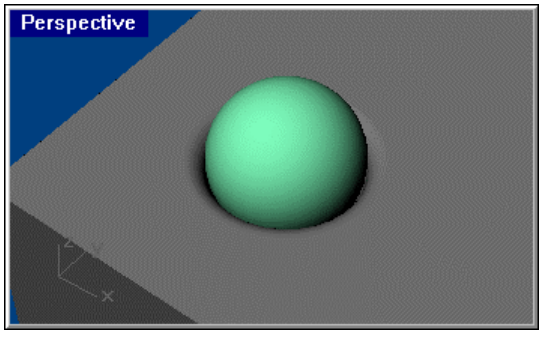
Button	Command	Menu
	Lock	Edit > Visibility > Lock
	Box	Solid > Box > Corner to Corner, Height
	Sphere	Sphere > Center, Radius

1. Use the **Lock** command on the mold part 1 you just created so you can work around it without accidentally selecting it.
2. Use the **Box** command to create a new box right on top of the box used for mold part 1.



3. Use the **Sphere** command to draw four small spheres that will form the other half of the registration keys.

   Use the grid snap to center the spheres inside the notches we 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. We can reasonably use a parting tolerance value of 0.05 mm. Therefore, use a radius of **1.95**.

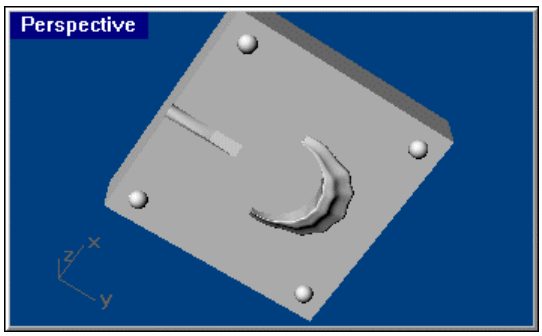


4. **Unlock** the part 1 solid and **Hide** it.

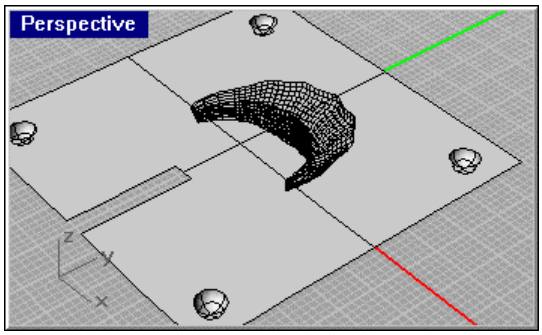
Cut and join the parts for the second mold part

Button	Command	Menu
	BooleanUnion	Solid > Union
	BooleanDifference	Solid > Difference
	ExtractSrf	Solid > Extract Surface
	Trim	Edit > Trim
	Join	Edit > Join

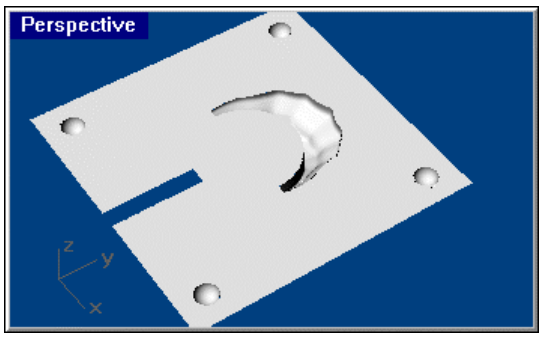
1. Make the "hollowing" surface visible.
2. Use the **BooleanUnion** command to join the spheres to the box to create the registration keys for this half of the mold.
3. To create a gate in this part, use **BooleanDifference** command to subtract the gate solid you copied earlier from the box.



4. Use the **ExtractSrf** command to separate the lower surface of the mold box.
5. **Hide** the larger part of the box, leaving the extracted surface showing.



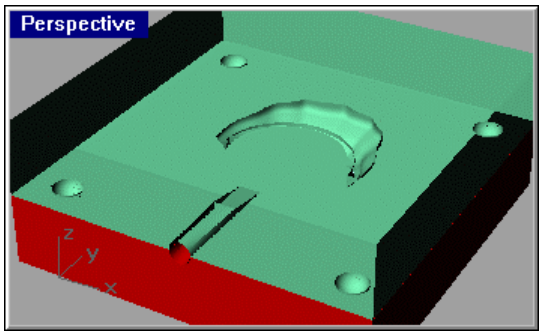
6. Use the **Trim** command to cut a hole in the box surface with the "hollowing" ring surface.



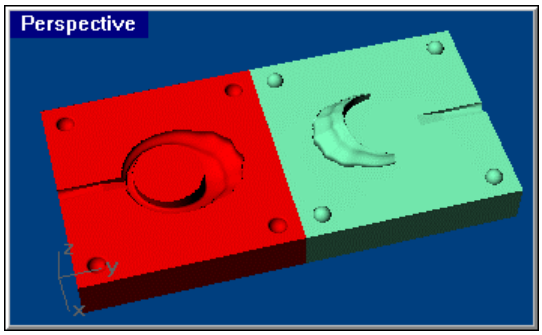
7 Use the **Join** command to join the inner ring surface, the extract surface of the box and the rest of the box back together.

The model of the two-part mold looks complete at this point. Let's spend a few moments checking what we have just built.

With the help of the Rhino rendering capabilities, we have given an almost transparent appearance to the upper part of the mold to 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. To create our files for VisualMill, we exported part 1 and part two as separate files.



Determine the smallest tool required

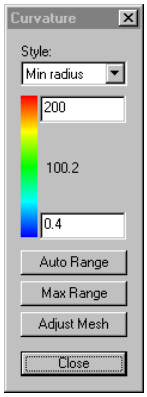
Button	Command	Menu
	CurvatureAnalysis	Analyze > Surface > Curvature Analysis

Rhino lets you analyze a surface in several ways. The method we want to use is to display the minimum radius. This will tell us what can be machined by tools of various radii.

1 Select mold part 1.

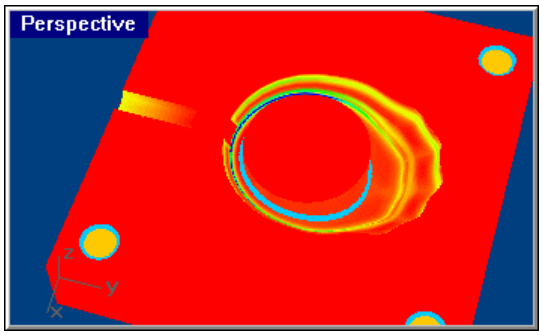
2 Use the **CurvatureAnalysis** command to check the surface.

Use the *Min radius* option.



3 Set the red upper limit to **200**.

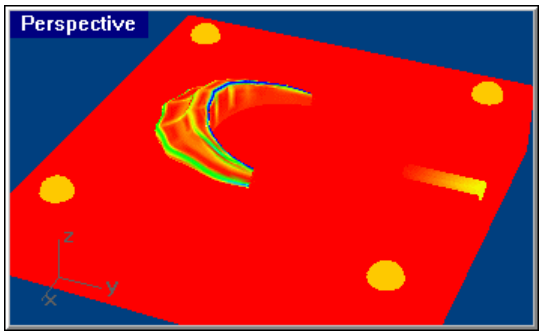
4 Set the blue lower limit to **.4**.



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

It is important to remember that these are actually representations of the tooling that will be used in the machining phase. In most extremely small radius situations, we will have to deal with a 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.



We have at this point completed our modeling and analysis.

The next step of our project will be to import mold part 1 and mold part 2 into VisualMill, and to develop a cutting strategy.

Models are saved to use for this purpose, so you don't have to complete the model.

In the next part we will create a real mold from our virtual model.

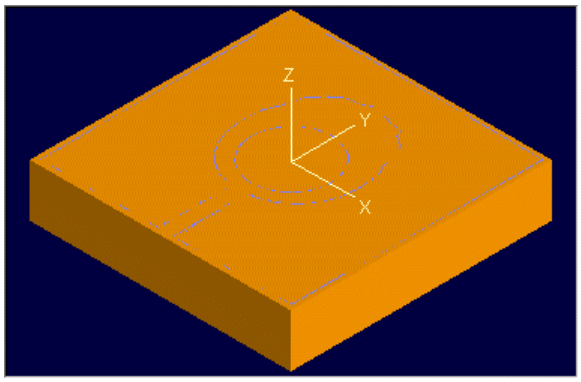
VisualMill will read both Rhino NURBS model files and Rhino mesh files. Normally, you can simply read in the Rhino model into VisualMill, but sometimes it works best to create the mesh in Rhino. In either case, export only the mold parts or a mesh created from the mold parts to a separate model.

Part 2:
Create tool paths with VisualMill

Get started with VisualMill

Button	Menu
	File > Open
	Enable material removal simulation
	Run to end with update
	View > Shade
	Part > Part Transform

There are two ways to import geometry produced with Rhino in VisualMill. One is to create a mesh in Rhino from the NURBS geometry and then open the Rhino mesh file in VisualMill. The second is to open the Rhino NURBS geometry file in VisualMill and let VisualMill do the meshing as the part is imported.



Parts of the VisualMill Screen

The VisualMill screen contains a menu bar at the top of the screen, toolbars docked at the top and sides, a status bar, a viewport, and a tabbed browser pane containing the machining operations, tools, and region definitions.

Start VisualMill and display the milling simulation

1. Start VisualMill.

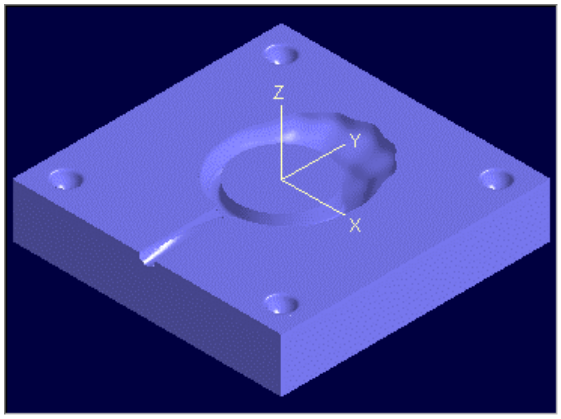
   Before you get started creating your own VisualMill part, open the part that comes with the tutorial and examine it with simulation turned on.
2. Open the model **Mold_part1.vmp**.
3. On the **VCR** bar, click **Enable material removal simulation**.
4. Set the increment to **100**.
5. Click **Run to end with update**.

Open the Rhino model

In this case, we have created a Rhino mesh, so we can import it.

1. Using the **Rhino Mesh Files (*.3dm)** option, open the file **Mold_Part1.3dm**.
2. **Shade** the part.

Once the geometry has been successfully imported into VisualMill, you will set the zero position. This is the reference point for all the cutting processes and the physical position where which we will place the tip of the cutter. For example, if the zero position will be center-top as referred to the part's bounding box, then we will have to move the cutter's tip to the upper center point of the stock material that has been loaded in the milling machine.



Set the zero position

1. Use **Part Transform** tool to locate zero at top center.
2. In the **Part Transform** dialog box, on the **Locate Zero** tab, set **Zero Face** to **Highest Z** and **Zero Position** to **Center**.
3. Experiment with the other settings to see how to change the zero location.

In the VisualMill viewport the zero position is shown by the white axes icon.

A note about the object colors

In VisualMill you can set the colors for the objects by selecting the color blocks in the status bar. For the graphics in this tutorial the colors are set as follows:

Object	Color
Part	
Stock	
Cut Stock	
Tool	
Background	
Regions	

Set up the cutting tools

Button	Menu
	Tool > Create/Select Tool

The next step will be to define a set of tools that we will use in our work.

For this project we define three ball-end mill tools with 1-, 2-, and 3-mm diameters.

Create a cutting tool

1. In the **Select/Create Tool** dialog box, on the **Ball End Mill** tab, set the **Tool Length** to **90**, the **Flute Length** to **10**, and **Diameter** to **1**
2. In the **Name** box, enter a name or accept the default name.
3. Click **Create New**.
4. Create two more ball end mill tools so you have a total of three tools with the following specifications:

Name	Tool Length	Flute Length	Diameter
Ball End 1 mm	90	10	1
Ball End 2 mm	90	10	2
Ball End 3 mm	90	15	3

5. Select the 3-mm tool.
6. Click **OK** to close the dialog box

   The highlighted tool will be the active tool. The name of the active tool appears in the status bar.

Set the specifications for the stock material

Button	Menu
	Stock > Part box stock
	Enable material removal simulation

We now have to work on the specifications for our stock material.

Set the stock specifications

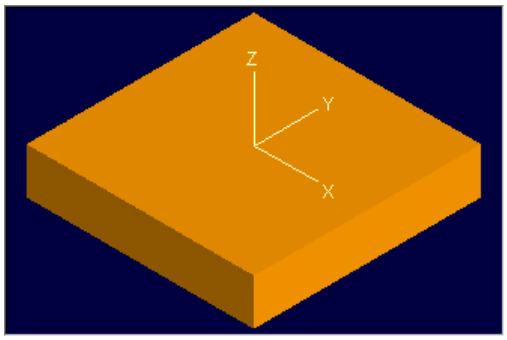
For our purpose, leave the z-offset at zero. This means that the top face of our stock will have the same z-value as the highest z-value of our part. The stock's top face is a flat surface. The x- and y-sides have a 1-millimeter offset.

Set the stock specifications

1. In the Part Bounding Box Stock dialog box, set the **X Offset** and **Y Offset** to **1**.

   Once you have defined the specifications of the stock, the top face of the part and the top face of the stock that are on the same plane.

2 To see the stock, on the **VCR** bar, click **Enable material removal simulation**.



Set the active tool

Before you start defining a machine operation, you have to set the active tool.

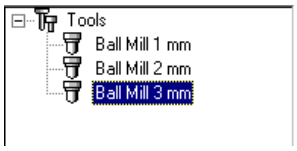
1 Check the status bar for the active tool.

It should look like this:

If it does not, you must select the active tool.

2 To set the active tool, in the browser pane click the **Tools** tab.

3 In the **Tools** list, double-click **Ball End Mill 3 mm**.



4 In the **Mill Tool** dialog box, click **OK**.

This sets the 3-mm ball-end mill tool to be the active tool. The active tool name appears in the status bar.

Define the horizontal roughing operation

Button	Menu
	3-Axis Milling > Horizontal Roughing

The first cutting strategy we are going to use is **Horizontal Roughing**. This will let us remove larger amounts of stock.

The horizontal roughing operation removes material in levels. The tool starts at the top of the stock model 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. Typical tools used for this kind of operation are flat end mills or corner radius end mills.

Set up the cut parameters

- In the **Horizontal Roughing** dialog box, on the **Cut Parameters** tab, set the values as follows:

Name	Value
Intol	.03
Outol	.03
Stock	.3
Cut Type	Offset
Cut Control	Climb
Pocket Start Point	Inside
Stepover Control	% Tool Diameter 25

Go to the next step before exiting this dialog box.

Set the cut levels

Button	Menu
	3-Axis Milling > Horizontal Roughing

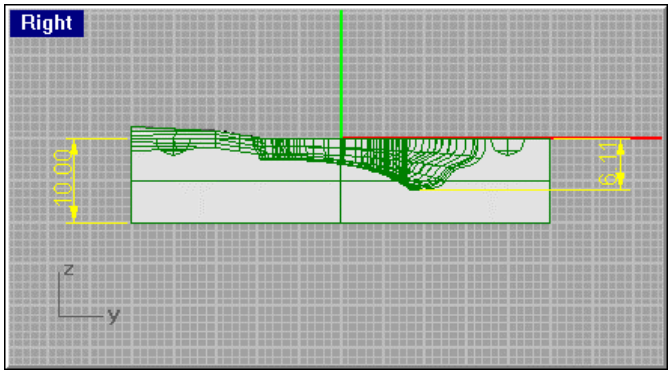
For cut levels it is important to have an adequate stepdown control. A stepdown of 0.5 millimeter is about how much we are going to cut for each stepdown level in this example. This depends on the material and the milling machine.

Set up the cut parameters

1. In the **Horizontal Roughing** dialog box, on the Cut Levels tab, under **Stepdown Control**, select **Distance** and then set the **Distance** to **.5**.
2. Under **Cut Levels Ordering**, select **Level First**.
3. Under **Cut Levels**, select **Bottom**, and set the level to **-6.5**.

   These options give some limits to the depth of the toolpath. A bottom value of -6.5 will guarantee us that all the cuttings we are interested in will be performed.
4. Click the **Generate** button.

   If you look at the dimensions on the Rhino model, you can see why the bottom cut level of -6.5 is satisfactory for our purposes. The maximum depth of the area you are going to cut is only 6.11.

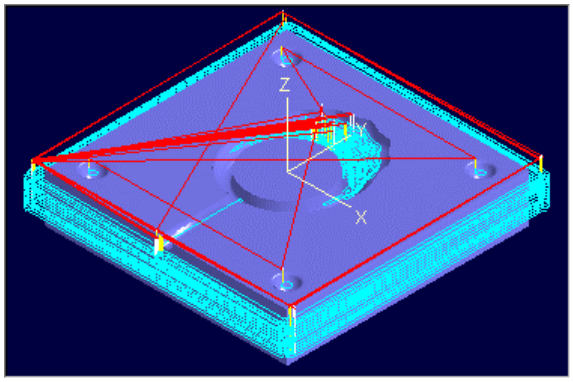


Show the toolpath

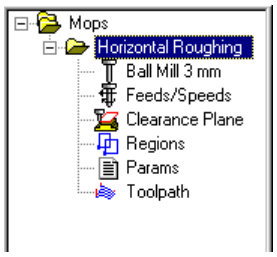
Button	Menu
	View > Hide toolpath

- **Show** the toolpath for the part.

  You can see the toolpath displayed on the viewport. The red lines show the transfer of the tool from one point to another.



A new Horizontal Roughing folder has been added in the browser pane on the Mops tab, which contains all the information that belongs to this toolpath.



Show the cutting simulation

Button	Menu
	View > Hide Toolpath
	Run to end with update

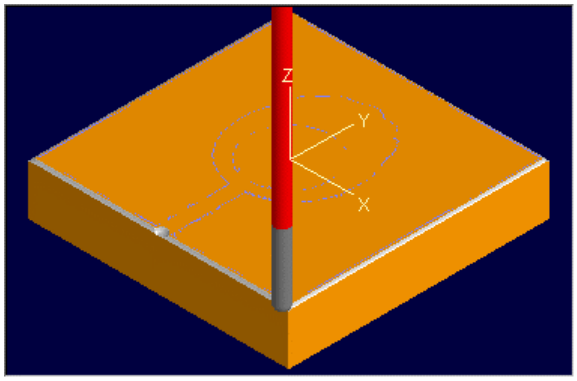
Cutting simulations show you what the results of cutting the stock will be.

Run the cutting simulation

1. Hide the toolpath.
2. On the **VCR** toolbar, set the increment to 100, and click **Run to end with update**.

   The orange part is the uncut material, the light gray parts show the cut stock.

This image shows that there are some uncut parts. This is because the size of the tool will not fit the cavities of the part.



Define cutting regions

Button	Menu
	Regions > Create Regions

With VisualMill you can define and use *regions*. They are 2-D shapes that limit where the cutting is going to be performed.

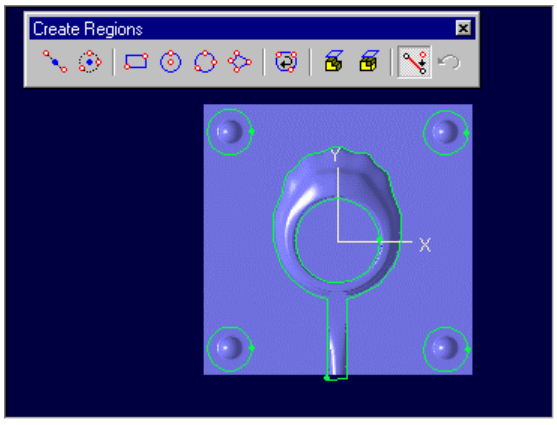
Since we are using flat stock that has been prepared, we do not need to cut on all the surfaces of the stock. The flat, prepared areas can be left alone. To do this you will define regions where cutting will take place.

They will be four circular regions where the registration notches are, a circular one going from the center of the part to the internal side of the ring, and another one that encompasses the gate and the outer limit of the ring.

Create regions

1. Change to a top view of the part.
2. From the menu, click **Regions**, and then click **Create Regions**.

   This opens the Create Regions toolbar.
3. Select a region tool.



As you create regions, they are added to the region browser pane and automatically numbered.

Apply regions to the machine operation

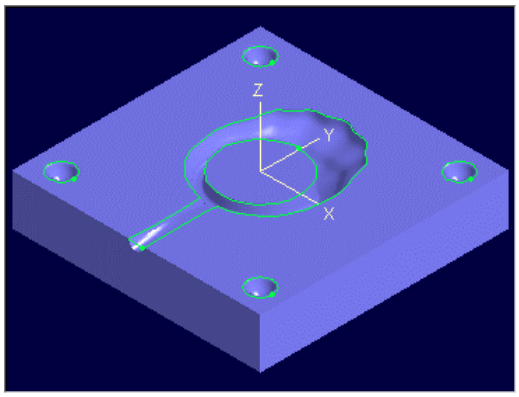
Button	Menu
	View > Hide Regions
	Single select region
	Regions > Select Active Regions

You want this operation to apply only to the regions around the notches.

Set active regions for machine operation

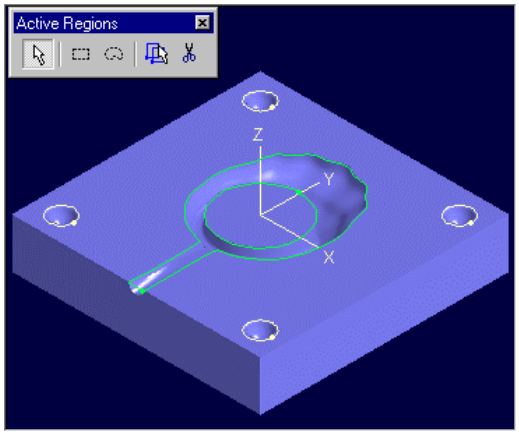
Regions define areas where the cutting operation will occur. Regions can be irregular polygons, rectangles and circles. These areas can also be nested within each other.

1. Show the region display.



2. In the browser pane, under the **Parallel Finishing** operation, double-click **Regions**.
3. Select the regions you want to be active for this machine operation.

   In this case select the four circular regions around the notches.



4. Right click in the browser pane and from the menu, click **Generate**.

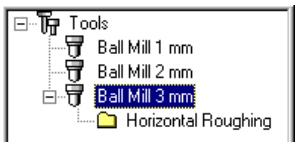
Define the parallel finishing operation

Button	Menu
	3 Axis Milling > Parallel Finishing

Parallel finishing can be used either as a pre-finishing or a finishing operation. The cutter (typically a ball end mill tool) follows a part's contours in the z-direction while being locked to parallel vertical planes. The orientation of these planes about the xy-plane is constant and can be defined by an angle from the x-axis.

- Set the **2-mm ball end tool** active.

  While you are looking at the **Tool** tab, notice that there is new information under **Ball End Mill 3 mm**. This tool is used for horizontal roughing, so that information is added to the tool folder.



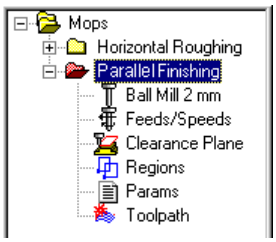
Set up the cut parameters

1. In the **Parallel Finishing** dialog box, on the Cut Parameters tab, under **Global Parameters**, set the **Stock** amount to **0**.
2. Under **Stepdown Control**, select **Distance** and then set the **Distance** to **.1**.

   Lower stepover values result in longer cutting times but a smoother surface.
3. Click the **Save** button when you have set all the parameters.

   The new machine operation appears in the browser. The red folder indicates that this toolpath has not yet been generated.

   Click the + to display the parameters for the parallel finishing.



   Before generating the toolpath, you will apply the regions you created in the last step.

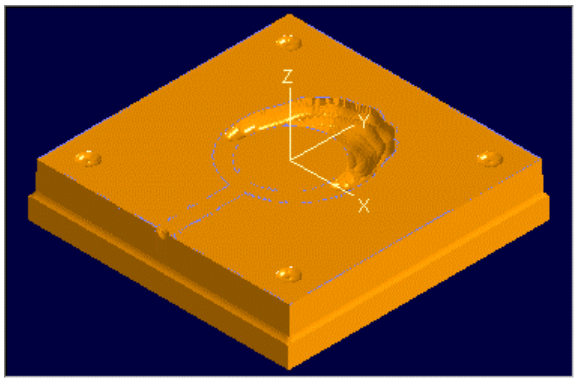
Show the simulation for the parallel finishing

Button	Menu
	Enable material removal simulation
	Run to end with update
	3 Axis Milling > Parallel Finishing

The cutting simulation will show what the cut will look like once it is performed.

Run the cutting simulation

1 On the **VCR** toolbar, set the increment to 100, and click **Run to end with update**.



As you can see, the four notches have a very smooth finish. The amount of stock left over from the cutting process is very limited.

Create the toolpath for the ring

Define a new parallel finishing operation with the central ring area regions active. Use the same technique as you used to create the notches.

Define a pencil trace operation

Button	Menu
	3 Axis Milling > Pencil Trace Roughing

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 sides of the cutter are in simultaneous contact with the part. This path typically follows the valleys and the corners of the part. Pencil trace can clean up scallops left after a parallel finishing operation or it can be used as a pre-finishing operation to remove material from the valleys and corners.

For any cutting strategy you may want to use, you need to consider the particular shape you are working on. If the tool will be forced to work on a very large amount of its blade, this can create a dangerous situation.

A pencil tracing operation with a Stock value of 0 will generate a toolpath that drives the cutter along the valleys and corners of the part thereby eliminating all the areas where the ball-end mill tool will have a double tangency condition. Double tangency is where the side and the point of the tool are cutting at the same time.

Use this strategy to prepare the area so that the stock removal can be as gentle as possible during the following stages.

Define the pencil tracing operation

1 Set the **1-mm ball end mill tool** active.

2 Activate the **Pencil Tracing** tool.

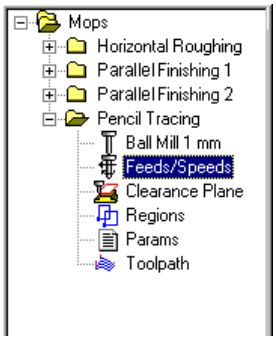
3 In the **Pencil Tracing** dialog box, on the **Pencil Trace Cut Parameters** tab, fill in the settings.

4 Click the **Save** button.

Set the feeds for the pencil trace tool

For this operation, you are going to set the speed and feed rate for the tool. By lowering the feed rates, we will reduce the risk of breaking the tool.

1. In the browser pane on the **Mops** tab, from the **Pencil Tracing** folder, double-click **Feeds/Speeds**.

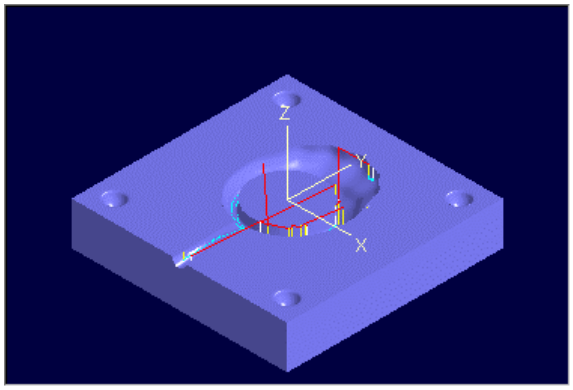


   The Set Feeds/Speeds dialog box appears.

2. Set the values as shown below:

Name	Value
Spindle Speed	8000 RPM
Cut Feed	10 mm/min
Engage Feed	5 mm/min
Retract Feed	5 mm/min
Transfer Feedrate	60 mm/min

3. Right-click in the browser pane and click **Generate**.
4. Select the regions that apply to the ring and gate areas.



Define a new parallel finishing operation

Button	Menu
	3 Axis Milling > Parallel Finishing

Now that the part has been prepared, we can perform another parallel finishing operation.

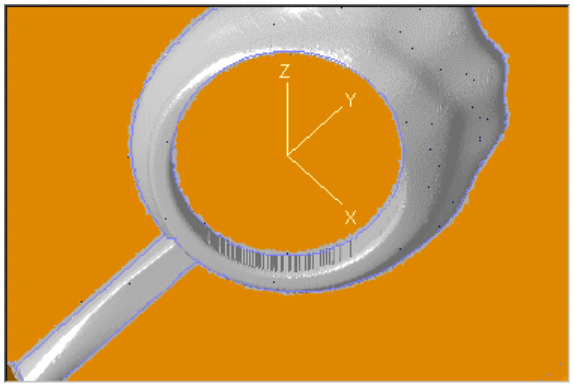
1. Check that the **1-mm ball end mill tool** is still active.
2. Activate the **Parallel Finishing** tool.

3 Set up the values as shown below.

Name	Value
Global Parameters Stock	0
Angle of Cuts	0
Stepover Control Distance	0.1

4 Set the regions involved in this cutting process to the same interior ring regions that you used last time.

5 In the browser pane on the **Mops** tab, right-click the **Parallel Finishing** operation and click **Generate**.



You can now see the result of the parallel finishing. Look at the area where the gate and the ring meet to see two channels that have been produced by the tool.

Define a final parallel finishing operation

Button	Menu
	3 Axis Milling > Parallel Finishing

To get an even smoother surface, it is a good idea to perform the same parallel finishing operation with the cuts oriented at 90 degrees to the previous one.

1 Check that the **1-mm ball end mill tool** is still active.

2 Activate the **Parallel Finishing** tool.

3 Set up the values as shown below. The difference from the other tools is the **Angle of Cuts** value is set to **90**.

Name	Value
Global Parameters Stock	0
Angle of Cuts	90
Stepover Control Distance	0.1

Set the regions involved in this cutting process to the same interior ring regions that you used last time.

In the browser pane on the **Mops** tab, right-click the **Parallel Finishing** operation and click **Generate**.

Review the model

Button	Menu
	
	View > Hide Toolpath
	View > Shade
	View > Hide Stock
	View > Reset Stock Color

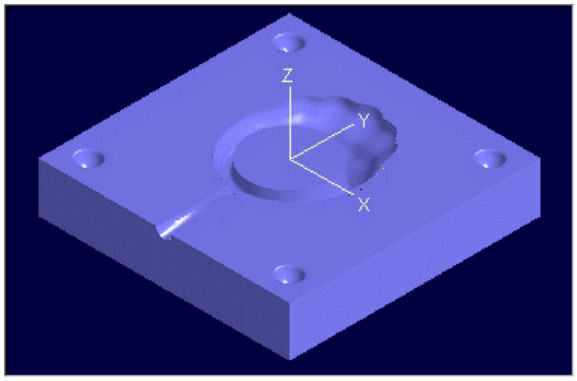
Now is the time to review your piece.

Set up for final review

1. On the **VCR** toolbar, click the **Run to end** (fast forward) button.
2. **Hide** the toolpath.
3. **Reset** the stock color.

   This sets the color of the freshly cut areas to the normal stock color.

   What we are actually seeing is what we will have in our hands when the milling is carried out.
4. As a comparison, shade the part.

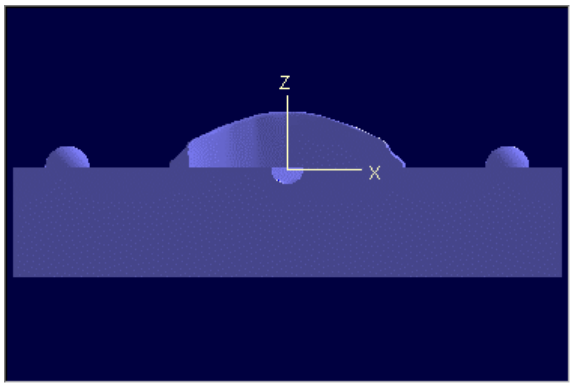


Create the second part

Button	Menu
	File > Open
	View > Front View
	Part > Transform

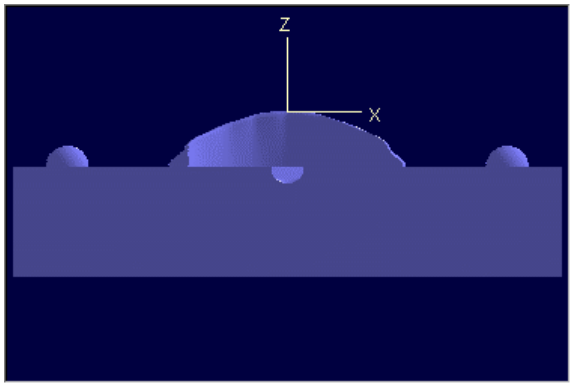
Mold Part 2 is the second (top) part of the mold. We will open the Rhino model and set the zero position. The zero position should be center top.

1. Using the **Rhino Files (*.3dm)** option, open the file **Mold_Part2.3dm**.
2. Set the view to the **Front** view.



As you can see from the image, the zero position is on the flat area of the part.

3. In the **Part Transform** dialog box, on the **Locate Zero** tab, set **Zero Face** to **Highest Z** and **Zero Position** to **Center**.

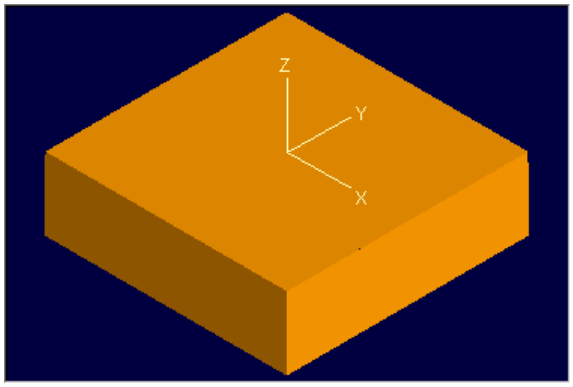


Set the specifications for the stock material

Button	Menu
	Stock > Part stock box
	Enable material removal simulation

1 In the **Part Bounding Box Stock** dialog box, set the **X Offset** and **Y Offset** to **1**.

2 Enable the material removal simulation.



Create the tools

Button	Menu
	Tool > Create Tool

1 Create the following tools for mold part 2.

Name	Tool Length	Flute Length	Diameter
Flat End 4 mm	90	60	4
Ball End 2 mm	90	15	2
Ball End 1 mm	90	10	1

2 Select the **4-mm flat end** tool.

This is the tool we are going to use for our first operation.

Define and generate the horizontal roughing

Button	Menu
	3 Axis Milling > Horizontal Roughing
	

1 Begin with horizontal roughing.

In the **Horizontal Roughing** dialog box, on the **Cut Parameters** tab, set the following values:

Name	Value
Intol	.03
Outol	.03
Stock	.3
Cut Type	Offset
Cut Control	Climb
Pocket Start Point	Inside
Stepover Control	% Tool Diameter 25

2 On the **Cut Levels** tab set the following parameters:

Name	Value
Stepdown control	Distance = 0.5
Cut Levels Ordering	Level First
Cut Levels	Bottom = -8.5

Remove material from the flat areas

Button	Menu
	3-Axis Milling > Plateau Machining

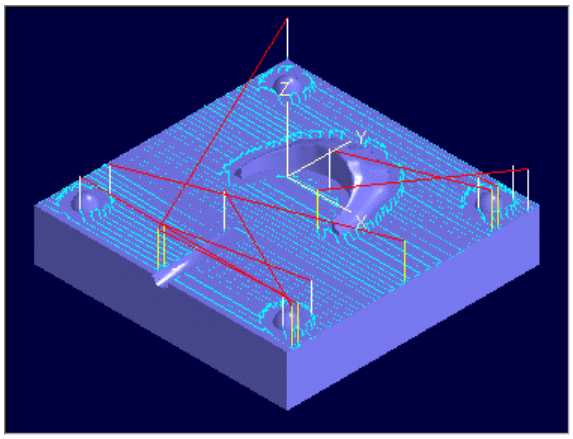
Use plateau machining to remove the stock from the flat areas.

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 horizontal finishing operation.

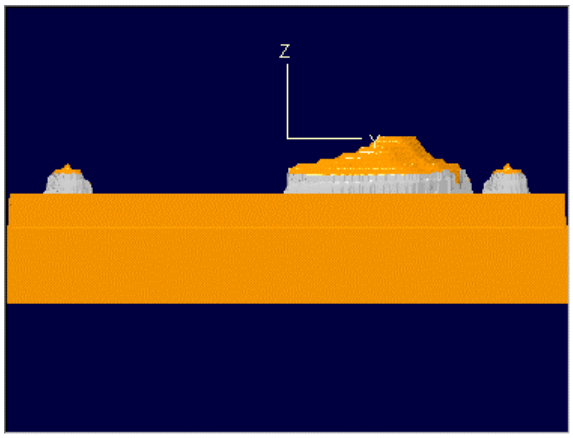
1 In the **Plateau Machining** dialog box, set the following values:

Name	Value
Intol	.03
Outol	.03
Flatness Angle	2
Cut Pattern	Zig Zag
Cut Start Point	SE
Angle of Cuts	0
Stepover Control	% Tool Diameter 25

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



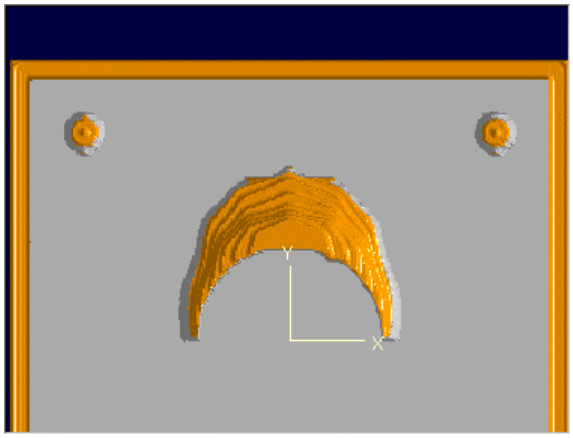
A side view of our work. It looks like we are getting there!



Cut down the stock around the notches

Button	Menu
	3 Axis Milling > Horizontal Finishing

In the top view shown below, you can see that because of the stepover value we have used in the plateau remachining, the finishing of the four notches and the central surface is still rough. Some stock still needs to be removed.



There is a quick solution to this situation.

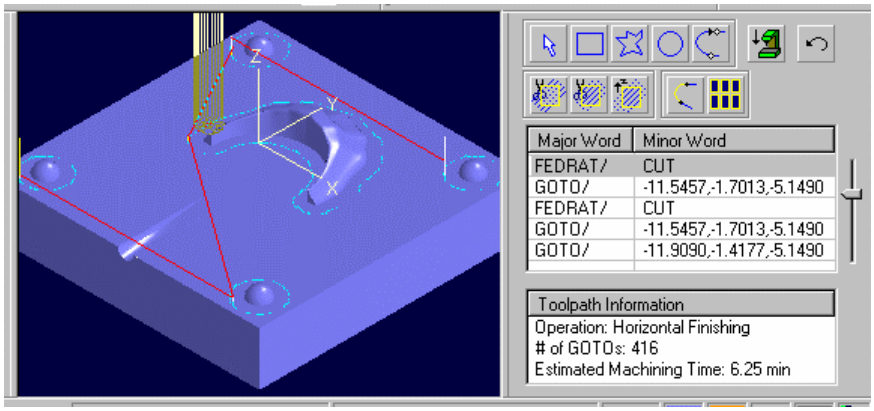
1. Create a **Horizontal Finishing** operation.
2. In **Horizontal Finishing** dialog box, on the Cut Levels tab, define a stepdown value that is limited to one level using a **Top** value of **0** and **Bottom** value of -5.149.

   Using this value forces Visual Mill to perform a horizontal finish at only one level and on one path.
3. **Generate** the toolpath.

Check the toolpath interactively

1. In the browser pane, on the **Mops** tab, under **Horizontal Finishing**, double-click **Toolpath**.

   The toolpath commands display to the right.



2. Move the slider down to see a visual display of the tool moving along the path.
3. To close the toolpath commands, in the browser pan, double-click **Toolpath** again.

Create regions for parallel finishing

Button	Menu
	Regions > Create Regions
	3 Axis Milling > Parallel Finishing

Now you are going to refine the finish of the mold with a parallel finishing operation.

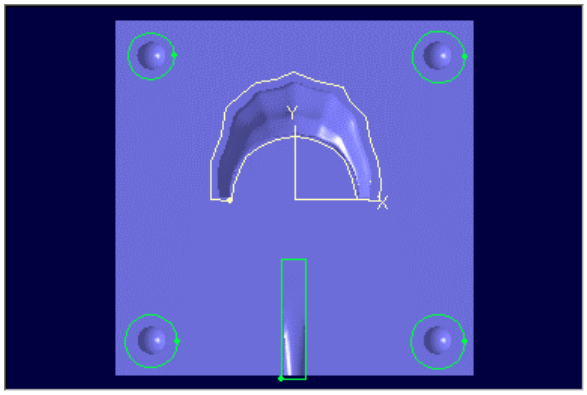
1. With the **Parallel Finishing** operation selected, from the menu, click **Regions**, and then click **Create Regions**.

   This opens the Create Regions toolbar.

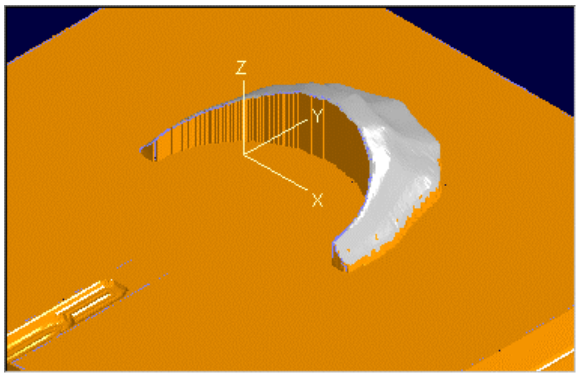
2. Select a region tool.

   The view automatically changes to a top view of the part.

   Create four circular ones where the spheres are, a polyline region where the large protruding surface is, and a rectangular one on the bottom of the part where the gate cut will be.



3. Create a **Parallel Finishing** operation using the **2-mm Ball End Mill** tool with the polyline around the main ring part as the active region.
4. Set the **Stepover Control Distance** to **0.1** mm to produce a very smooth piece.

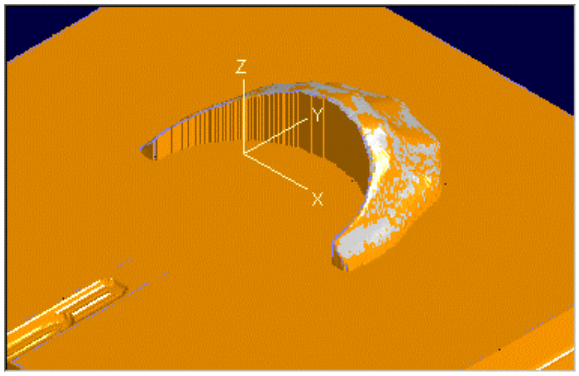


Create the second parallel finishing operation

Button	Menu
	3 Axis Milling > Parallel Finishing

1 Create another **Parallel Finishing** operation, but this time the cutting pattern will be at right angle with the previous one.

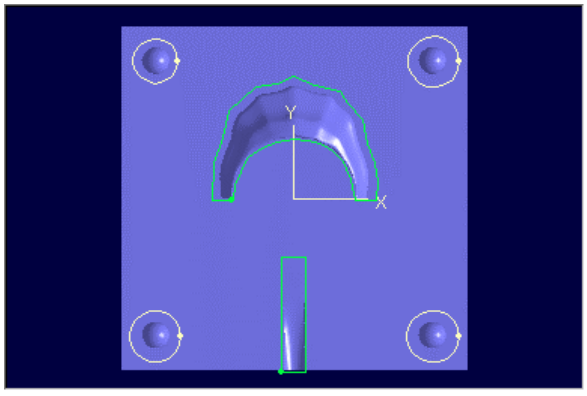
2 In the **Parallel Finishing** dialog box, on the **Cut Parameters** tab, set **Angle of Cuts** to **90**.



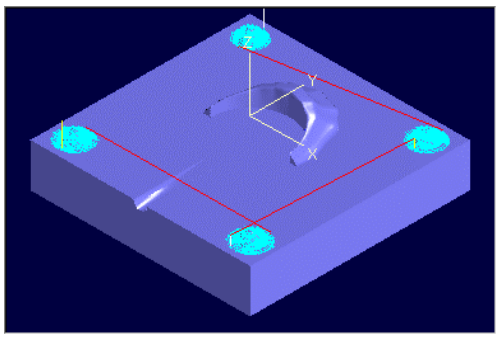
Finish the registration keys

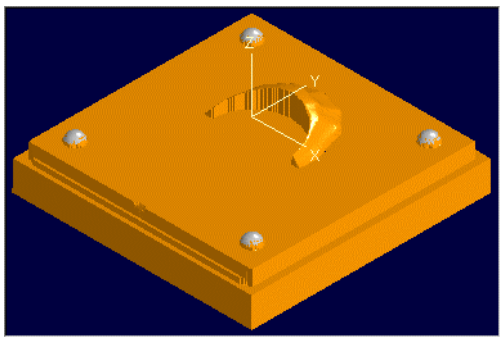
Button	Menu
	3 Axis Milling > Parallel Finishing

1 Using the **2-mm Ball End Mill** tool and the four circular regions, create a new **Parallel Finishing** operation.



2 Set the **Stepover Control Distance** to **0.1 mm**.



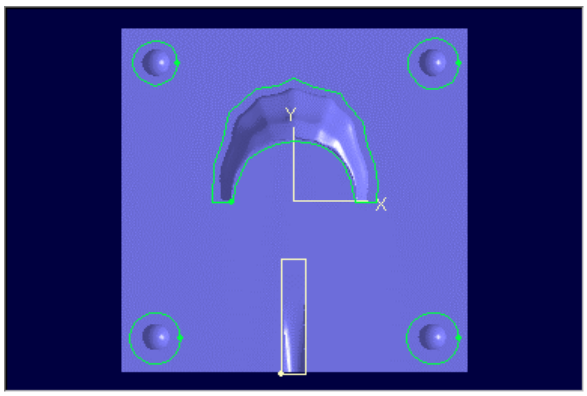


Finish the gate

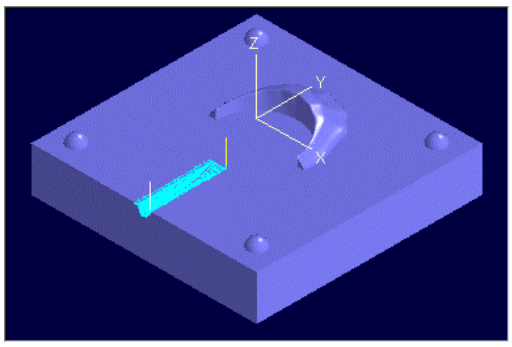
Button	Menu
	Finishing > Parallel Finishing
	3 Axis Milling > Valley Re-Machining

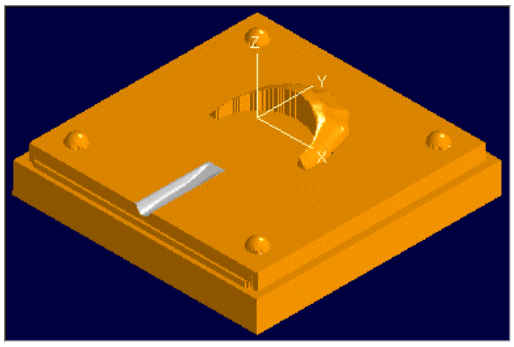
Use parallel finishing for the first step

1 Select the region around the gate.



2 Create a **Parallel Finishing** operation to finish the gate using the **2-mm ball end mill** tool.





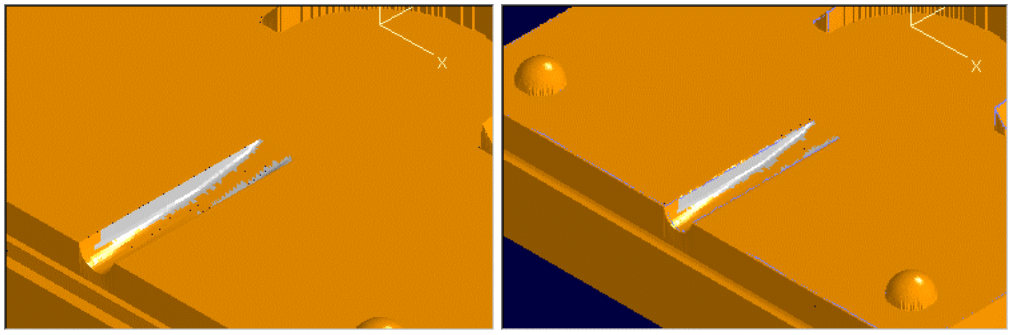
The gate area has a portion of its geometry that is circular, with a transition to a rectangular portion. In this area a 2-mm ball end mill tool will leave a fair amount of uncut stock. We will then work with a smaller radius tool that will ensure a better cut of this particular area of the region.

Use valley re-machining for the final finish

1 For the final finish, create a **Valley Remachining** operation.

Valley remachining is a toolpath method that can be used as a re-finishing operation to clean up regions in a part that could not be reached by a larger tool.

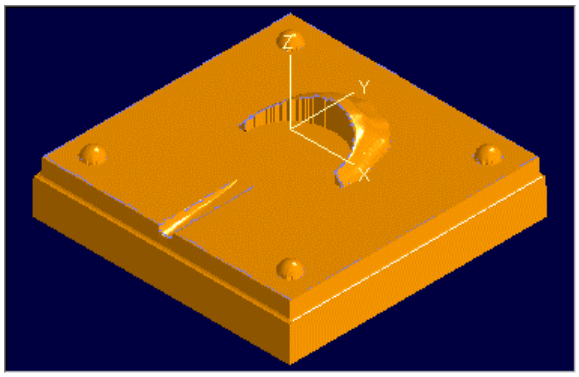
2 In the **Valley Remachining** dialog box, use the **1-mm ball end mill** tool for the valley remachining.



Final finishing and checking

Button	Menu
	View > Hide Stock
	View > Shade

Now compare the cut stock and the imported Rhino model. You should not notice a great difference.



Now hook up your milling machine and go!

After milling the part, you will have to couple the two mold parts and do some minor hand work to match the gate's inlet to your wax-injector's nozzle shape.

By the time you started cutting the parts the wax may not have warmed yet!

The browser pane has three tabs that display the machine operations, tool definitions, and region definitions.

To learn more

To learn more about Rhinoceros, tutorials are included with the evaluation version on the CD. Visit *www.rhino3d.com* for links to training classes, tutorials, the Rhino newsgroup, and other resources.

To learn more about VisualMill, check out the VisualMill tutorial and the *Getting Started with VisualMill* booklet in either US (8.5" x 11") or A4 (21 cm x 29.7 cm) paper size also on the CD. Visit *www.mecsoft.com* for further links to information about VisualMill.

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