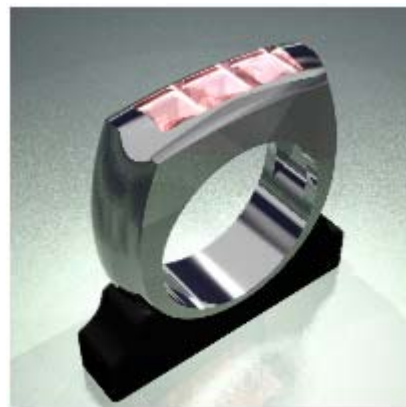
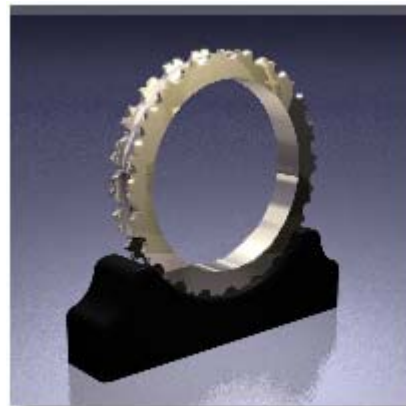
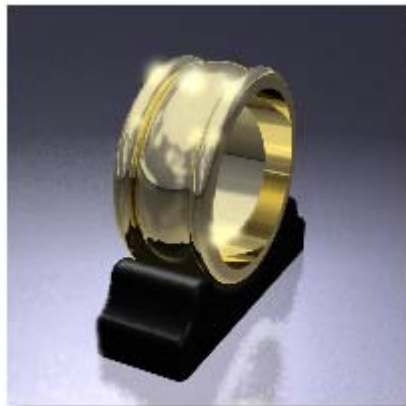


# Basic Jewelry Modeling in Rhinoceros®





Copyright © 2005 Robert McNeel & Associates. All rights reserved.

**RhinoCeros** is a registered trademark of Robert McNeel & Associates.

Revised 22 March 2005.

Tutorials provided by:

Alex Antich, [TechJewel](#)

Giuseppe Massoni, [Massoni Design](#)



## Table of Contents

<b>1</b>	<b>Create a Ring with a Concave Top Surface .....</b>	<b>5</b>
	Starting a Model with a Template .....	5
	Modeling the Shank Outline .....	6
	Creating the Inside Shank Curve .....	10
	Extruding Curves to Create the Solid Ring.....	13
	Cutting Out the Shank .....	14
<b>2</b>	<b>Create a Ring with Decorative Top Elements .....</b>	<b>16</b>
	Modeling the Shank.....	16
	Creating the Inside Curve.....	24
	Extruding Curves to Create the Solid Ring.....	26
	Cutting Out the Shank .....	28
<b>3</b>	<b>Create a Simple Band Ring.....</b>	<b>29</b>
	Modeling the Shank Inner Curve .....	29
	Creating the Shank Cross-Section Profile .....	31
<b>4</b>	<b>Create an Eternity Ring with Claw Settings.....</b>	<b>35</b>
	Modeling the Shank.....	35
	Create the Claw to Array .....	36
	Create a construction circle for the outside of the shank .....	36
	Create construction lines .....	37
	Arraying the Claw Profile Around the Ring .....	41
	Extruding the Curves to Create the Solid Ring .....	41
	Creating the Channel.....	42
	Cut the Channel.....	44
<b>5</b>	<b>Create a Ring with Gemstones .....</b>	<b>46</b>
	Draw the Initial Shank Outline Shape.....	46
	Adding Gemstones .....	48
	Importing a gemstone model .....	48
	Place another gemstone .....	49
	Place the third gemstone .....	50
	Making the Finished Shank Curves.....	52
	Making the Interior Opening .....	53
	Edit the interior curve's shape .....	54
	Making the Curves that Define the Exterior Shape.....	57
	Making the Exterior Surfaces .....	61
	Making the Interior Surface .....	62
	Making the Front and Back Surfaces .....	63



Making a Solid to Cut the Opening in the Top Surface.....	64
Cut the Opening in the Top Surface .....	65



## 1 Create a Ring with a Concave Top Surface

In this tutorial, you will learn how to make a ring with a concave top surface.

You will learn how to:

- Use **grid snap** and **object snaps**
- Use **layers**
- Make **circles** and **arcs** to define the shape of the ring
- **Extend** arcs
- **Join** the arc segments into a single polycurve
- **Offset** a polycurve to make a second polycurve
- Turn on and move **control points**
- **Extrude** the 2-D curves to make solids
- Use **zoom** commands to change the view
- Use the **BooleanDifference** command to hollow the ring's interior



### Starting a Model with a Template

When starting a model in Rhino you can select a template to begin the modeling process. Templates contain information that helps you get started such as units, layers, viewport layouts, and toolbars.

Selecting a default template sets the **units** for your model. For these examples, we will be using millimeters as our units.

The templates include six predefined **layers**. Layers are a way of organizing objects so you can manipulate them separately or keep track of them in some way. It may help you to think of layers as transparent sheets where you can place different parts of your model. The predefined layers have color properties that will let you distinguish various parts more easily.

**Modeling aids** can help you draw the 2-D geometry with more precision. In this tutorial you will use the **snap** and **object snap** (osnap) modeling aids. You can find the controls for modeling aids on the **status bar** at the bottom of the Rhino window.

When **Snap** is on, the marker "snaps" between grid snap points. In this template, the grid snap is set at one-millimeter intervals to match the grid line intersections.

**Object snaps** make it easy to snap precisely to various parts of the geometry. In this tutorial, we will use the **End** object snap to attach geometry at the end of another piece of geometry.

---

**Note** Object snaps are crucial to accurate modeling. Never try to place things visually on the screen; it is too difficult, and it is impossible to ensure accuracy.

---



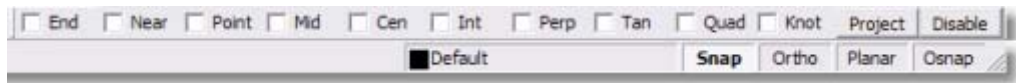
### To start the model

- ▶ On the **File** menu, click **New**, select **Millimeters.3dm**, and then click **Open**.

### To turn on modeling aids

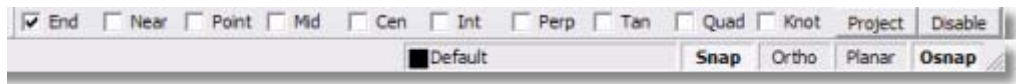
- ▶ On the **status bar**, click the **Snap** and **Osnap** panes.

These controls are toggles. Click a pane once to turn the aid on; click again to turn it off. When modeling aids are turned on, the text in the pane is bold.



Clicking the **Osnap** pane toggles the display of the **Osnap** toolbar.

The **Osnap** pane text is bold only when one or more of the object snap check boxes are selected.



You will be using the **Osnap** toolbar in this tutorial, so make sure it is displayed.

## Modeling the Shank Outline

We will start by making an outline of the shank. First, we will maximize the Front viewport to make it easier to see.

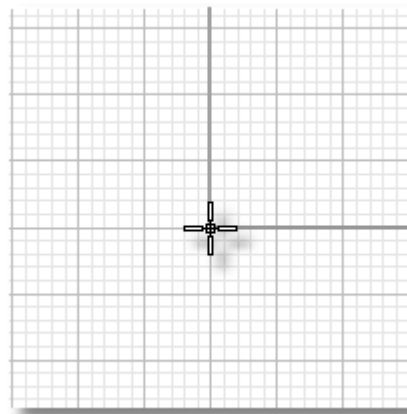
### To maximize the Front viewport

- ▶ In the **Front** viewport, double-click the viewport title.

### To make the inside of the shank

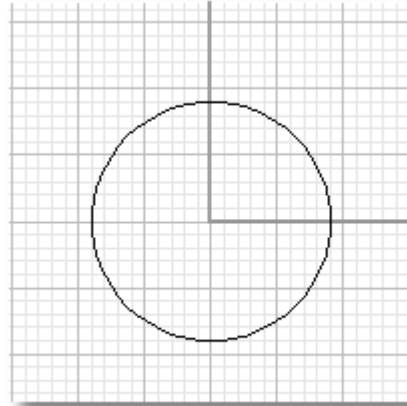
- 1 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.
- 2 At the **Center of circle ...** prompt, move your mouse pointer to the origin of the viewport construction plane (where the red and green axis lines meet) and click.

Since grid snap is on, the cursor will snap exactly to the origin point where the axes meet.



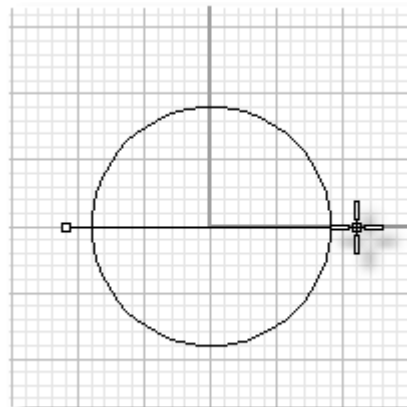


- 3 At the **Radius ...** prompt, type **9**, and press ENTER.

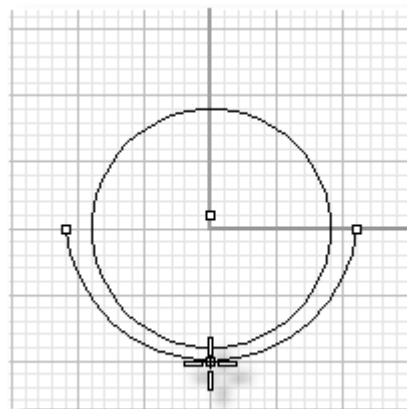


#### To make the outside of the shank

- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Point**.
- 2 At the **Start of arc** prompt, click a point **2 mm** (two grid lines) to the left of the left quadrant of the circle.
- 3 At the **End of arc** prompt, click a point **2 mm** to the right of the right quadrant of the circle.



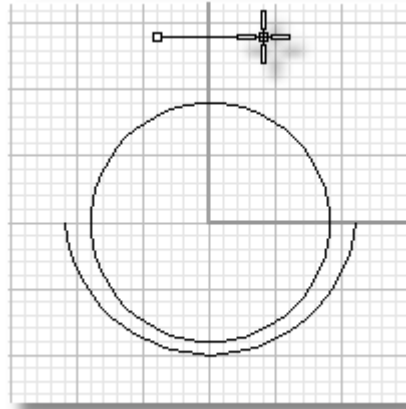
- 4 At the **Point on arc** prompt, click a point **1 mm** from the bottom of the circle.



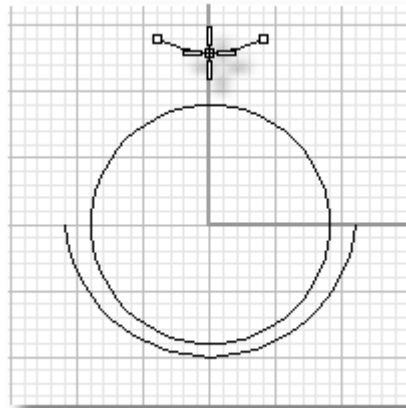


### To make the top of the shank

- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Point**.
- 2 At the **Start of arc** prompt, click a point approximately **5 mm** up from the top and **4 mm** to the left of the top of the circle.
- 3 At the **End of arc** prompt, click a point **8 mm** to the right of the start point.



- 4 At the **Point on arc** prompt, click a point **4 mm** from the top of the circle.



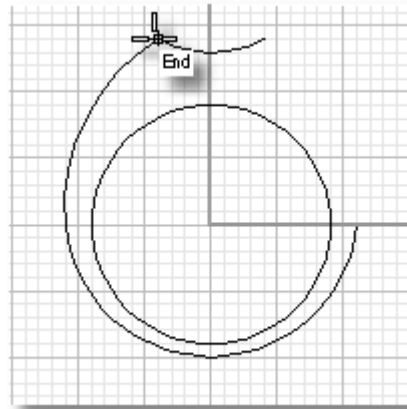
### To finish the curves for the shank

- 1 On the **Osnap** toolbar, select the **End** check box.
- 2 On the **Curve** menu, click **Extend**, and then click **By Arc to a Point**.
- 3 At the **Select curve to extend ...** prompt, click the upper left end of the arc that forms the bottom of the shank.

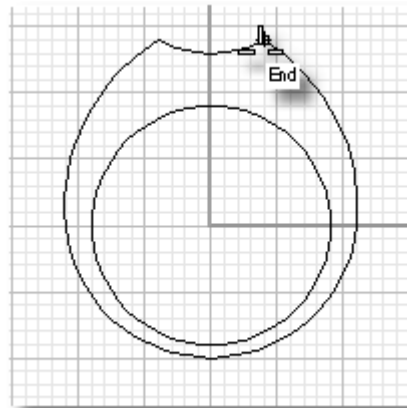




- 4 At the **End of extension ...** prompt, move your mouse pointer until the **End** object snap causes it to snap to the left end of the arc at the top and click.



- 5 **Repeat** steps 2 through 4 to extend the arc on the right side of the shank.





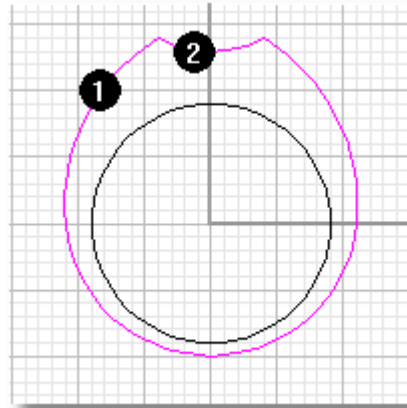
## Creating the Inside Shank Curve

Next, we will offset the outside curve to make another curve that defines the inside of the shank.

### To join the outside shank curves

- 1 On the **Edit** menu, click **Join**.
- 2 At the **Select object to join** prompt, click curve **1**.
- 3 At the **Select curve to join** prompt, click curve **2**.

This will join the two curves into one polycurve.



### To set the current layer

The Layer pane displays the name of the current layer. In this case, the Layer pane should display the layer name "Default."

- ▶ On the **status bar**, click the **Layer** pane, and then select the **Layer 01** check box.

This sets the current layer to Layer 01 so that the new offset curve will display in red, the color of Layer 01.

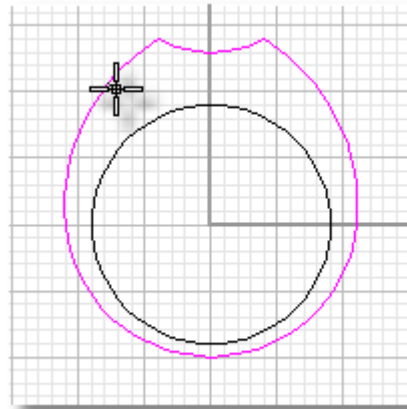
### To offset the joined curve

- 1 **Select** the joined curve, if it is not already selected.
- 2 On the **Curve** menu, click **Offset Curve**.
- 3 At the **ThroughPoint ...** prompt, type **.8**, and press ENTER.

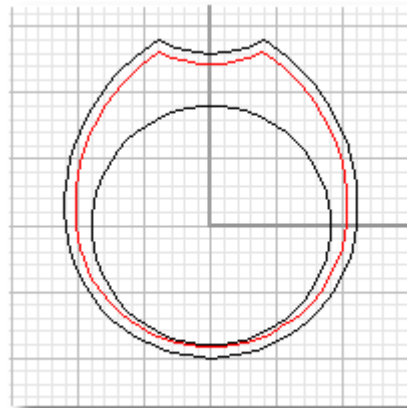
This sets the offset distance to 0.8 mm.



- 4 At the **ThroughPoint...** prompt, click inside the selected curve.



A new curve is created 0.8 mm inside the original curve.

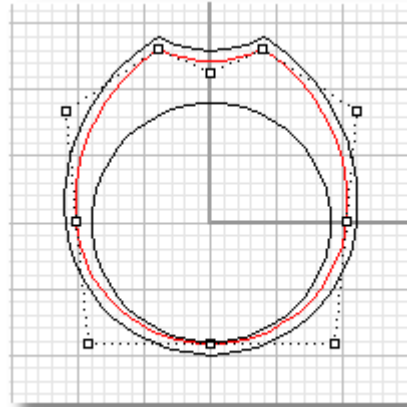




### To edit the interior curve

- 1 **Select** the red offset curve.
- 2 On the **Edit** menu, click **Control Points**, and then click **Control Points On**.

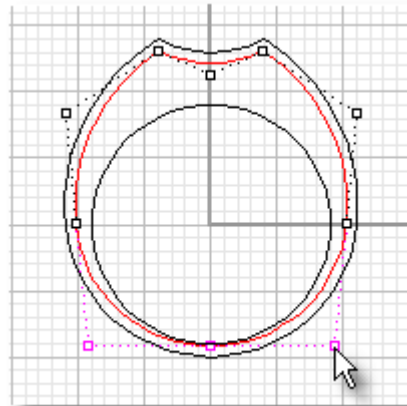
Control points appear surrounding the curve. These display as small squares. Control determine the shape of the curve.



- 3 **Select** the three control points at the bottom of the curve.

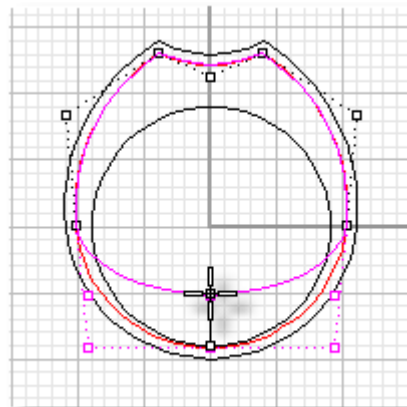
To select the points individually, hold the SHIFT key and click.

Another way to select the points is to click and drag a window from left to right that encloses the three points.



- 4 On the **Transform** menu, click **Move**.
- 5 At the **Point to move from** prompt, click a point in the center of the ring.
- 6 At the **Point to move to** prompt, click a point approximately **4 mm** above the first point.
- 7 On the **Edit** menu, click **Control Points**, and then click **Control Points Off**.

This is so the solid created from this curve that will be used to hollow the ring will not cut through the lower part of the band.





## Extruding Curves to Create the Solid Ring

The next step is to make two extruded solids; one for the ring and another that will be used to cut out the inside of the shank. This operation will be easier to visualize if you can see all four viewports.

### To restore the four-viewport layout

- 1 In the **Front** viewport, double-click the viewport title.
- 2 On the **View** menu, click **Zoom**, and then click **Extents All**.

The view zooms in each viewport to display all of the geometry.

You can also rotate the mouse wheel to zoom in and out in the active viewport.

### To set the current layer

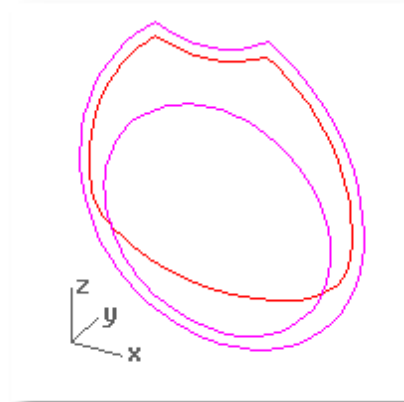
- ▶ On the **status bar**, click the **Layer** pane, and then select the **Default** check box.

This sets the current layer to Default.

### To extrude the shank curves

- 1 **Select** the outer curve and the original circle.
- 2 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 At the **Extrusion distance** ... prompt, move your mouse pointer to the command line and click the options so they are set as follows:

**BothSides= Yes**  
**Cap= Yes**  
**Mode= *Straight***  
**DeleteInput= No**



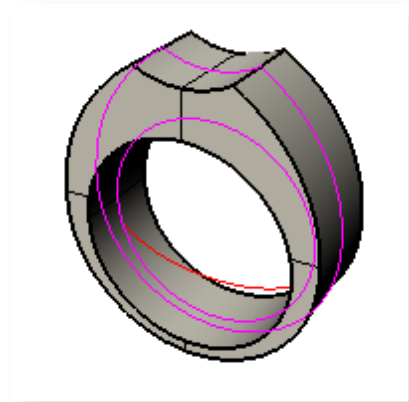
- 4 At the **Extrusion distance** ... prompt, type **4**, and press ENTER.

The curves are extruded 4 mm on either side of the original curves making ring surfaces 8 mm wide, and adding flat surfaces on the two sides to make a complete solid.



### To shade the viewport

- ▶ In the **Perspective** viewport, right-click the viewport title, and then click **Shaded Display**.



### To set the current layer

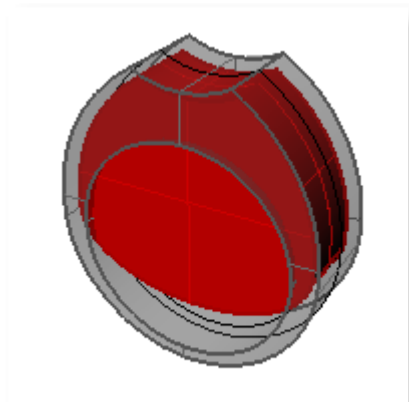
- ▶ On the **status bar**, click the **Layer** pane, and then select the **Layer 01** check box.

### To extrude the interior curve

- 1 **Select** the interior curve.
- 2 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 At the **Extrusion distance ...** prompt, type **3**, and press ENTER.

The options should be set as before:

**BothSides= Yes**  
**Cap= Yes**  
**Mode= Straight**  
**DeleteInput= No**



## Cutting Out the Shank

We will finish the ring by removing the extruded interior solid from the ring solid using a Boolean operation.

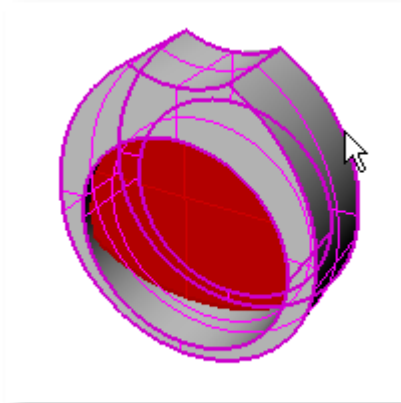
The three Boolean operations in Rhino are **Union**, **Difference**, and **Intersection**.

The **BooleanUnion** command merges two or more intersecting solids into one piece. The **BooleanDifference** command removes one or more solids from another. The **BooleanIntersection** command leaves only the intersecting part of two or more solids.

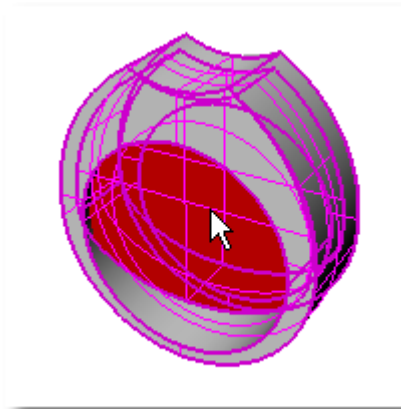


**To remove the center from the ring**

- 1 **Select** the ring solid.
- 2 On the **Solid** menu, click **Difference**.



- 3 At the **Select second set of surfaces or polysurfaces ...** prompt, click the interior solid, and press ENTER



The smaller cutout solid is removed from the shank of the ring.

- 4 In the **Perspective** viewport, right-click and drag to rotate the view.





## 2 Create a Ring with Decorative Top Elements

In this tutorial you will learn how to make another style of ring.

In addition to practicing the techniques from the previous tutorial, you will learn how to:

- Use **Ortho** to draw at 90-degree increments.
- Use the **Mirror** command to model symmetrical objects.
- Use **Fillet** to make rounds between two curves.



---

**Note:** You should complete the tutorial in the previous chapter before starting this tutorial.

---

### To start the model

- ▶ On the **File** menu, click **New**, select **Millimeters.3dm**, and then click **Open**.

### To turn on modeling aids

- 1 On the **status bar**, click the **Snap** and **Osnap** panes.
- 2 On the **status bar**, click the **Ortho** pane.

When Ortho is on, cursor movement is restricted to points at multiples of a specified angle from the last point created. The default angle is 90 degrees.

## Modeling the Shank

We will start by making the outline of the shank.

### To maximize the Front viewport

- ▶ In the **Front** viewport, double-click the viewport title.

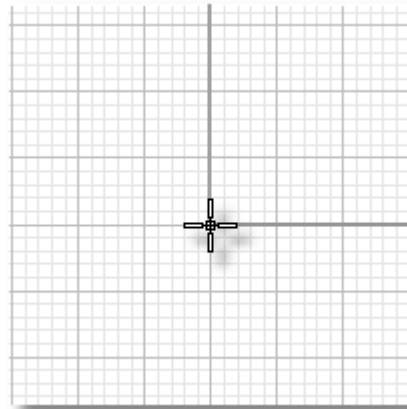
### To make the inside of the shank

- 1 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.

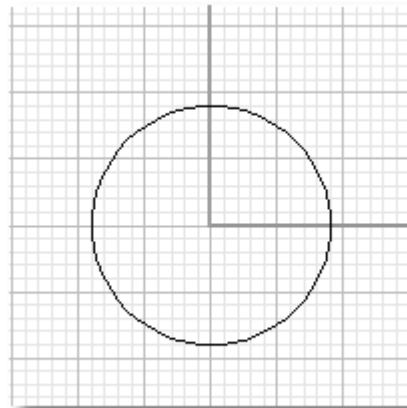




- 2 At the **Center of circle ...** prompt, move your mouse pointer to the origin of the viewport construction plane and click.

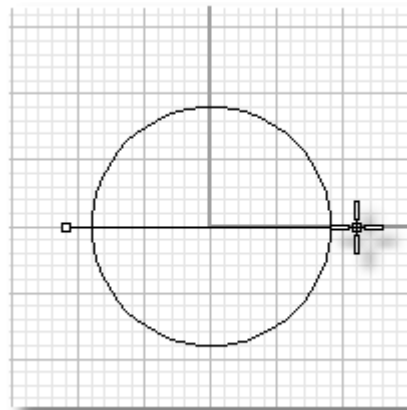


- 3 At the **Radius ...** prompt, type **9**, and press ENTER.



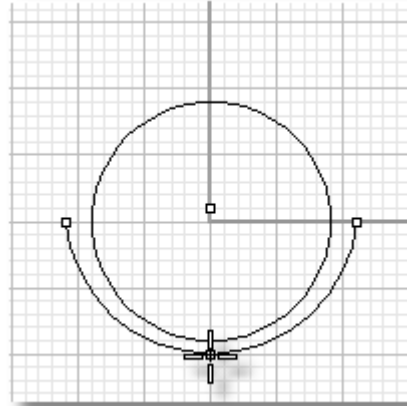
### To make the outside of the shank

- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Point**.
- 2 At the **Start of arc** prompt, click a point **2 mm** (two grid lines) to the left of the left quadrant of the circle.
- 3 At the **End of arc** prompt, click a point **2 mm** to the right of the right quadrant of the circle.



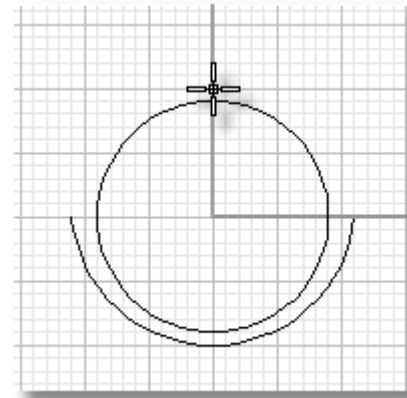


- 4 At the **Point on arc** prompt, click a point **1 mm** from the bottom of the circle.



#### To finish the outside of the shank

- 1 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 2 At the **Start of line ...** prompt, set the **BothSides** option to **Yes**.  
This lets you make a line from a middle point with symmetry.
- 3 At the **Middle of line...** prompt, click a point **1 mm** up from the top quadrant of the circle.



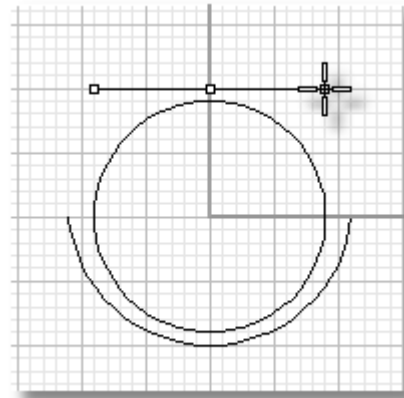
- 4 At the **End of line** prompt, type **9**, and press ENTER.

Typing a value and pressing ENTER sets a constraint for the length. The result is no matter where you click, the line will be 9 units from the middle point on both sides.



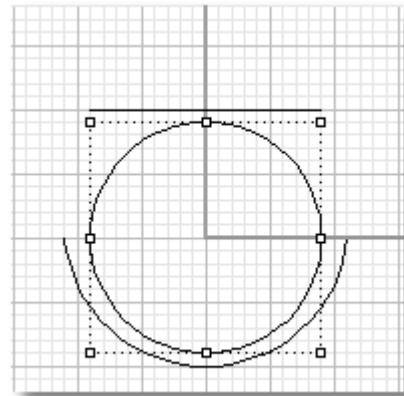
- 5 At the **End of line** prompt, move your mouse to the right and click.

The cursor movement is constrained to 90-degree angles by the Ortho setting and to 9 mm by the distance constraint.

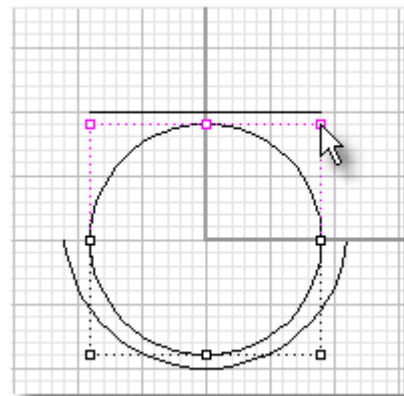


### To make the top inside part of the shank oval

- 1 **Select** the circle.
- 2 On the **Edit** menu, click **Control Points**, and then click **Control Points On**.

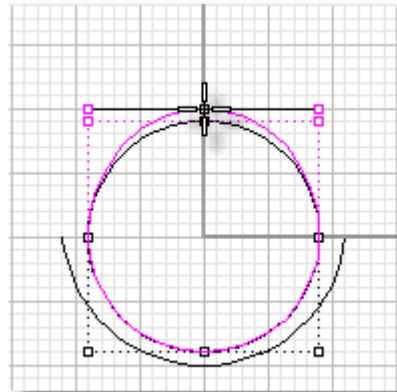


- 3 **Select** the three control points at the top of the circle.





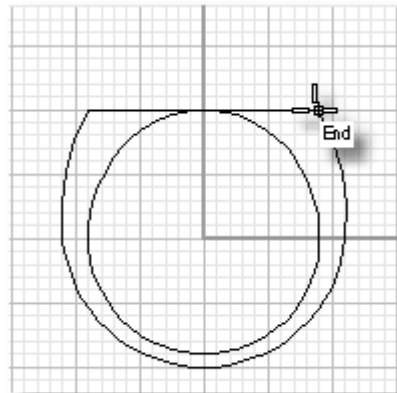
- 4 On the **Transform** menu, click **Move**.
- 5 At the **Point to move from** prompt, click the point at the top of the circle.
- 6 At the **Point to move to** prompt, click the middle of the line.



- 7 On the **Edit** menu, click **Control Points**, and then click **Control Points Off**.

#### To finish the curves for the shank

- 1 On the **Osnap** toolbar, select the **End** check box.
- 2 On the **Curve** menu, click **Extend**, and then click **By Arc to a Point**.
- 3 At the **Select curve to extend ...** prompt, use the **End** object snap to select the upper left end of the arc that forms the bottom of the shank.
- 4 At the **End of extension** prompt, use the **End** object snap to select the the left end of the line at the top and click.



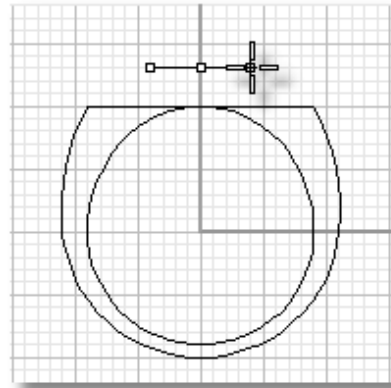
- 5 **Repeat** these steps to extend the arc on the right side of the shank.

#### To make the center arc for the top of the ring

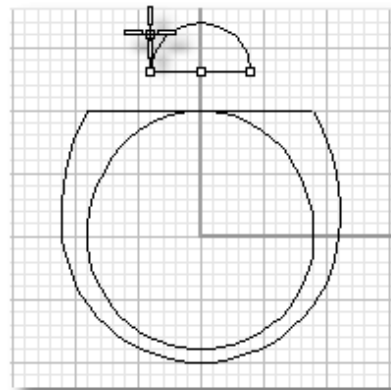
- 1 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 2 At the **Start of line ...** prompt, set the **BothSides** option to **Yes**.
- 3 At the **Middle of line...** prompt, click a point **3 mm** above the top quadrant of the oval.
- 4 At the **End of line** prompt, type **4**, and press ENTER.



- 5 At the **End of line** prompt, move your mouse pointer to the right and click.

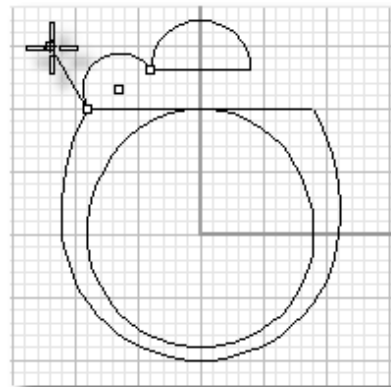


- 6 On the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.
- 7 At the **Start of arc** prompt, snap to one end of the line and click.
- 8 At the **End of arc** prompt, snap to other end of the line and click.
- 9 At the **Direction at start** prompt, click a point directly above the first point.



#### To make the side arcs

- 1 On the **status bar**, click the **Ortho** pane to toggle Ortho off.
- 2 On the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.
- 3 At the **Start of arc** prompt, use the **End** object snap to start the arc at the left corner of the shank.
- 4 At the **End of arc** prompt, use the **End** object to end the arc at the left end of the upper arc.
- 5 At the **Direction at start** prompt, drag and click to make the arc as illustrated.

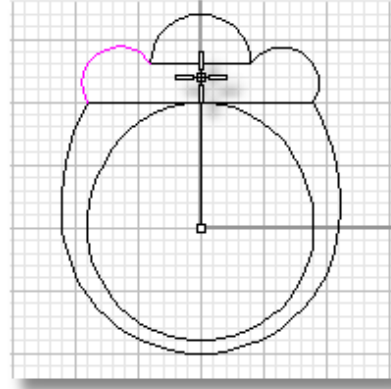


#### To mirror the arc to the other side

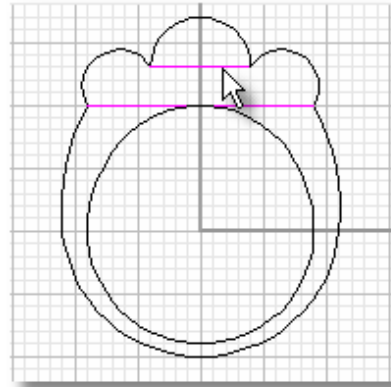
- 1 Turn **Ortho** on.



- 2 **Select** the arc you just created.
- 3 On the **Transform** menu, click **Mirror**.
- 4 At the **Start of mirror plane ...** prompt, click in the center of the ring.
- 5 At the **End of mirror plane ...** prompt, use Ortho to click directly above the first point.



- 6 **Select** the two construction lines, and press DELETE.



#### To make a radius between the arcs

- 1 On the **Curve** menu, click **Fillet Curves**.
- 2 At the **Select first curve to fillet ...** prompt, type **.2**, and press ENTER.  
This sets the fillet radius to 0.2.
- 3 At the **Select first curve to fillet ...** prompt, click the **Join** option.

This joins the arcs at the same time it makes the round.

Check the options to see that they are set as follows:

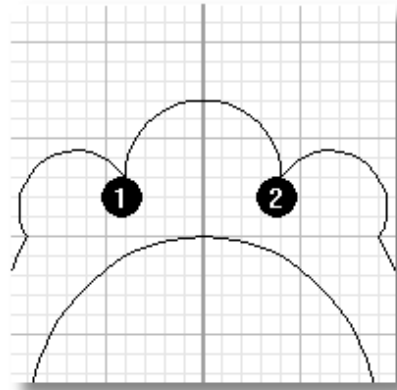
**Radius=0.2**  
**Join=Yes**  
**Trim=Yes**  
**DeleteInput=No**



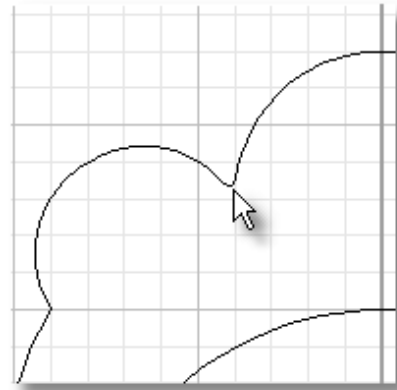
4 At the **Select first curve to fillet ...** prompt, click near the end of one of the arcs **(1)**.

5 At the **Select second curve to fillet ...** prompt, click near the end of the adjacent arc.

Another arc is drawn between the arcs, the original arcs are trimmed back to the ends of the fillet, and the three parts are joined.



6 **Repeat** this step for the other side **(2)**.





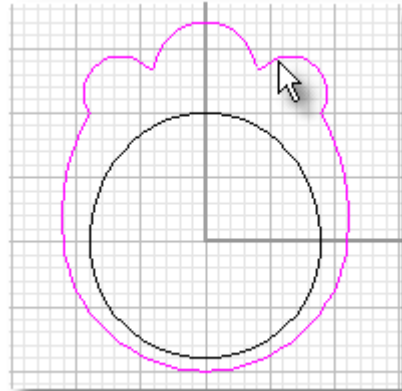
## Creating the Inside Curve

Next we will make a copy of the outside of the ring to use to cut out the inside of the shank.

### To make the curve that defines the inside of the shank

- 1 **Select** all the curves that form the outer part of the ring.
- 2 On the **Edit** menu, click **Join**.

This will join the curves into one polycurve.

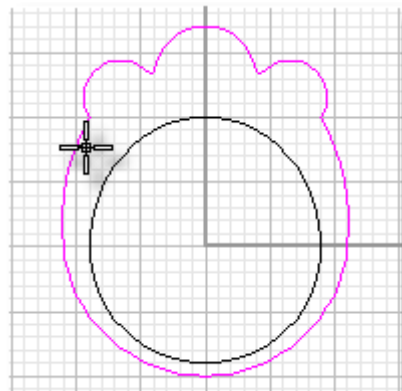


### To set the current layer

- ▶ On the **status bar**, click the **Layer** pane, and then select the **Layer 01** check box.

### To offset the curve

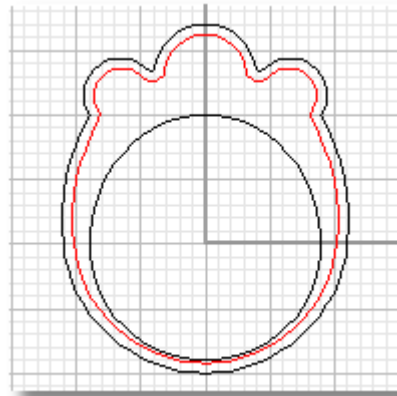
- 1 **Select** the joined curve.
- 2 On the **Curve** menu, click **Offset Curve**.
- 3 At the **ThroughPoint ...** prompt, type **.8**, and press ENTER.
- 4 At the **ThroughPoint ...** prompt, click inside the selected curve.





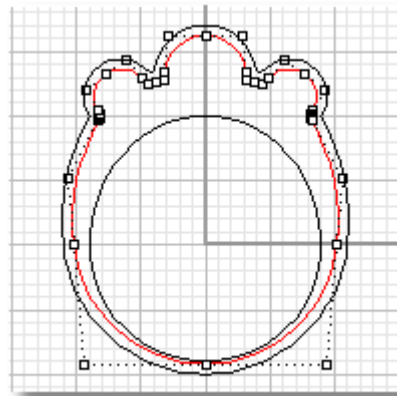


A new curve is created 0.8 mm inside the original curve.

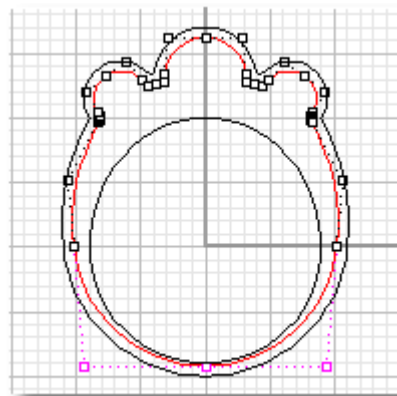


**To adjust the curve to move the lower part out of the way of the band**

- 1 **Select** the offset curve you just created.
- 2 On the **Edit** menu, click **Control Points**, and then click **Control Points On**.

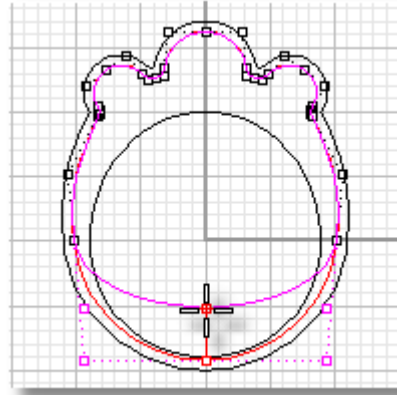


- 3 **Select** the three control points at the bottom of the curve.





- 4 On the **Transform** menu, click **Move**.
- 5 At the **Point to move from** prompt, click a point at the bottom of the curve.
- 6 At the **Point to move to** prompt, click a point approximately **4 mm** above the first point.



- 7 On the **Edit** menu, click **Control Points**, and then click **Control Points Off**.

## Extruding Curves to Create the Solid Ring

The next step is to make two solid extrusions; one for the ring and another for the inside of the shank. This section will be easier if you can see all four viewpoints.

### To restore the four-viewport layout

- 1 In the **Front** viewport, double-click the viewport title.
- 2 On the **View** menu, click **Zoom**, and then click **Extents All**.

### To set the current layer

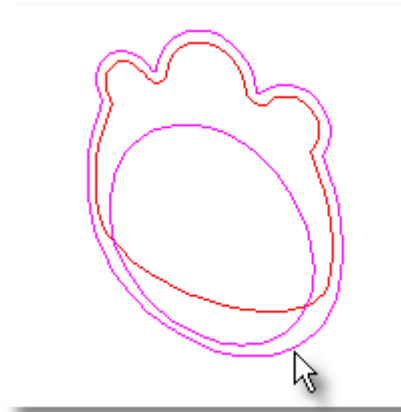
- ▶ On the **status bar**, click the **Layer** pane, and then select the **Default** check box.

This sets the current layer to Default.

### To extrude the shank curves

- 1 **Select** the outer curve and the circle.
- 2 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 At the **Extrusion distance ...** prompt, move your mouse pointer to the command line and click the options so they are set as follows:

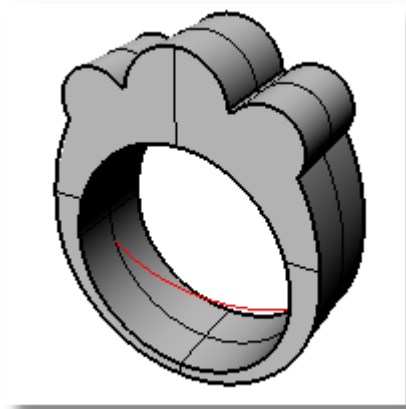
**BothSides= Yes**  
**Cap= Yes**  
**Mode= Straight**  
**DeleteInput= No**





- 4 At the **Extrusion distance ...** prompt, type **4**, and press ENTER.

The curves are extruded 4 mm on either side of the original curve giving the ring a thickness of 8 mm.



#### To set the current layer

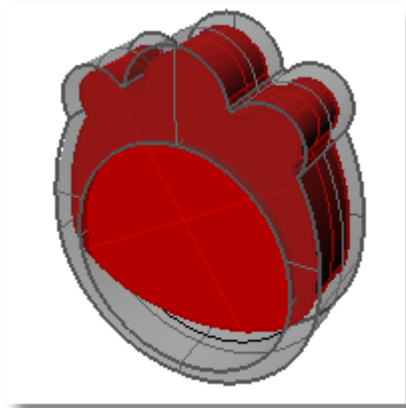
- ▶ On the **status bar**, click the **Layer** pane, and then select the **Layer 01** check box.

#### To extrude the interior curve

- 1 **Select** the interior curve.
- 2 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 3 At the **Extrusion distance ...** prompt, type **3**, and press ENTER.

The options should be set as before:

**BothSides= Yes**  
**Cap= Yes**  
**Mode= *Straight***  
**DeleteInput= *No***



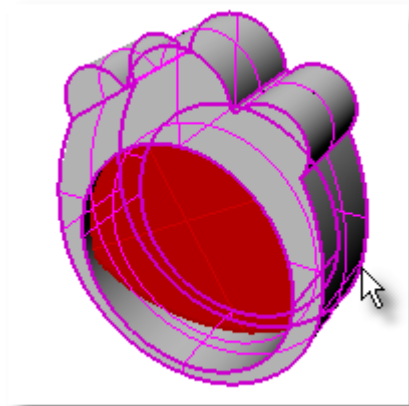


## Cutting Out the Shank

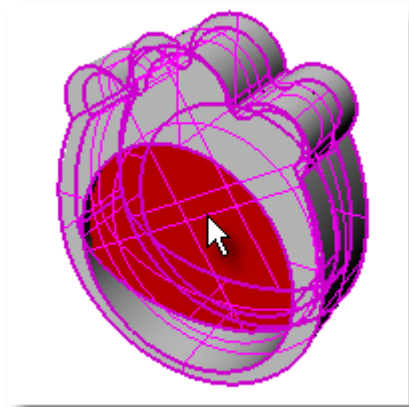
We will finish the ring by removing the extruded interior solid from the ring solid using a Boolean operation.

### To remove the center from the ring

- 1 **Select** the ring extrusion.
- 2 On the **Solid** menu, click **Difference**.

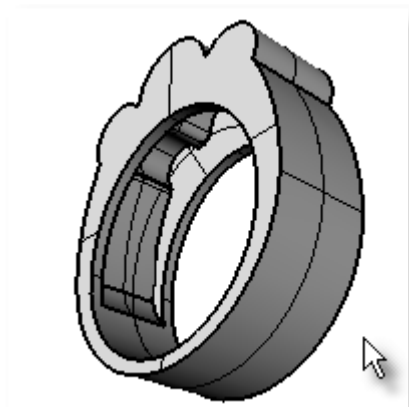


- 3 At the **Select second set of surfaces or polysurfaces ...** prompt, select the interior solid.



- 4 At the **Select second set of surfaces ...** prompt, press ENTER.

The smaller extrusion is removed from the shank of the ring.



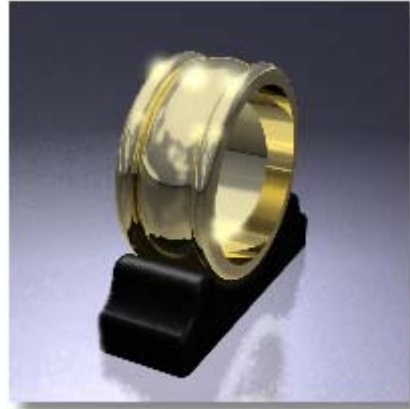


### 3 Create a Simple Band Ring

In this tutorial you will learn how to make a simple band ring with scallop detailing.

In addition to practicing the techniques you learned in the previous chapters, you will learn how to:

- Make a surface with a **1 Rail Sweep**



---

**Note:** You should complete the tutorials in the previous chapters before starting this tutorial.

---

#### To start the model

- ▶ On the **File** menu, click **New**, select **Millimeters.3dm**, and then click **Open**.

#### To turn on modeling aids

- ▶ On the **status bar**, click the **Snap**, **Ortho**, and **Osnap** panes.

### Modeling the Shank Inner Curve

We will start by making an outline of the shank.

#### To maximize the Front viewport

- ▶ In the **Front** viewport, double-click the viewport title.

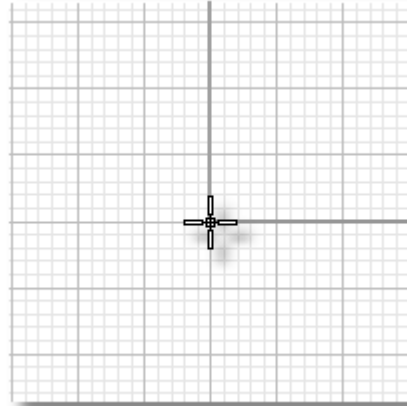
#### To make the inside of the shank

- 1 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.

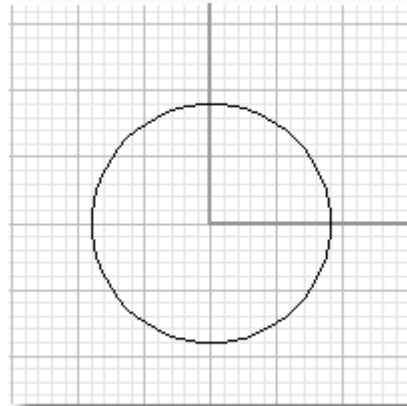


- 2 At the **Center of circle ...** prompt, move your mouse pointer to the origin of the viewport construction plane (where the red and green axes lines meet) and click.

Since grid snap is on, the cursor will snap exactly to the origin point where the axes meet.



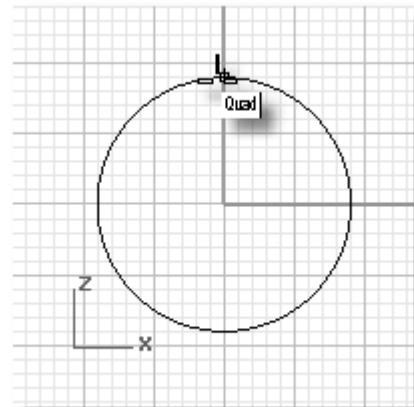
- 3 At the **Radius ...** prompt, type **9**, and press ENTER.



#### To make a reference point on the circle

- 1 On the **Osnap** toolbar select the **End**, **Point**, and **Quad** check boxes.
- 2 On the **Curve** menu, click **Point Object**, and then click **Single Point**.
- 3 At the **Location of point object** prompt, use the **Quad** object snap to place the point at the top quadrant of the circle.

This will give us a reference point for creating the cross-section of the band.





## Creating the Shank Cross-Section Profile

In this section you will create the cross-section profile curve that will be swept around the shank curve.

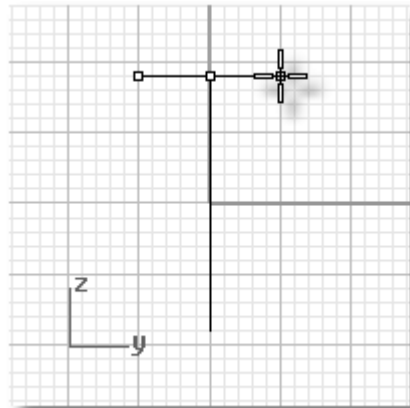
### To maximize the Right viewport

- 1 In the **Front** viewport, double-click the viewport title to restore the four-viewport layout.
- 2 In the **Right** viewport, double-click the viewport title.

### To make the construction lines for the cross-section curve

- 1 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 2 At the **Start of line ...** prompt, click **BothSides**.
- 3 At the **Middle of line ...** prompt, click the reference point you created in the previous section.
- 4 At the **End of line** prompt, type **5**, and press ENTER.
- 5 At the **End of line** prompt, move your mouse to the right and click.

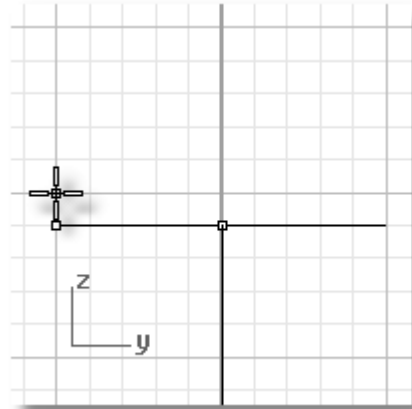
The cursor movement is constrained to 90-degree angles by the Ortho setting and to 5 mm by the distance constraint.



- 6 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 7 At the **Start of line...** prompt, click the left end of the line you just created.
- 8 At the **End of line** prompt, type **1**, and press ENTER.

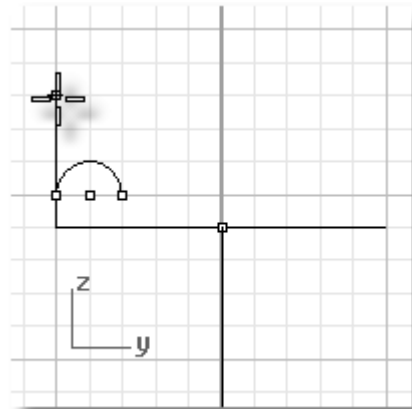


- 9 At the **End of line** prompt, click directly above the first point.



#### To create the first arc

- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.
- 2 At the **Start of arc** prompt, use the **End** object snap to start the arc at the upper end of the line you just created.
- 3 At the **End of arc** prompt, move your mouse pointer two grid points to the right and click.
- 4 At the **Direction at start** prompt, click a point directly above the first point.



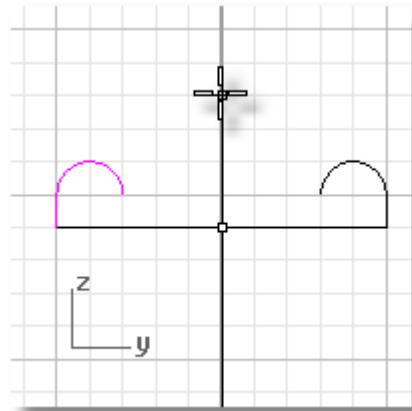
#### To mirror the arc and the line segment

- 1 **Select** the arc and the vertical line segment you just created.
- 2 On the **Transform** menu, click **Mirror**.
- 3 At the **Start of mirror...** prompt, snap to the reference point.



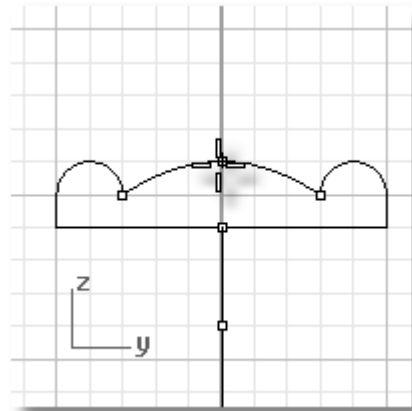


- 4 At the **End of mirror plane...** prompt, click directly above the first point.



**To finish the cross-section curve**

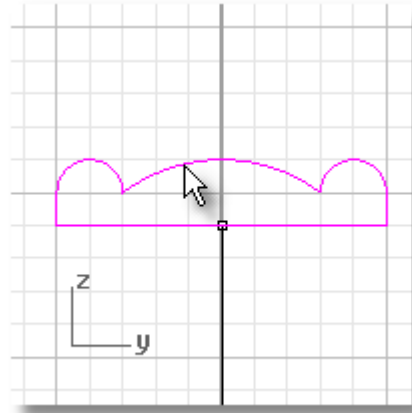
- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Point**.
- 2 At the **Start of arc** prompt, use the **End** object snap to start the arc at the end of one of the side arcs.
- 3 At the **End of arc** prompt, use the **End** object snap to end the arc at the end of the other arc.
- 4 At the **Point on arc** prompt, click a point **2 mm** up from the reference point.





### To join the cross-section curve

- 1 **Select** the three line segments and the three arcs.
- 2 On the **Edit** menu, click **Join**.

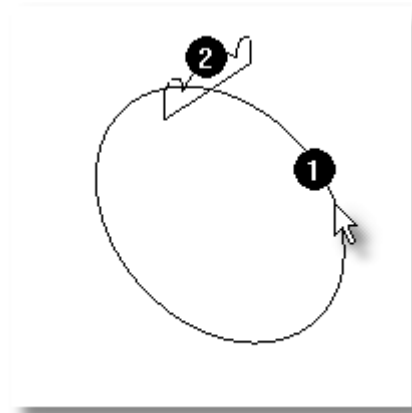


### To restore the four viewport layout

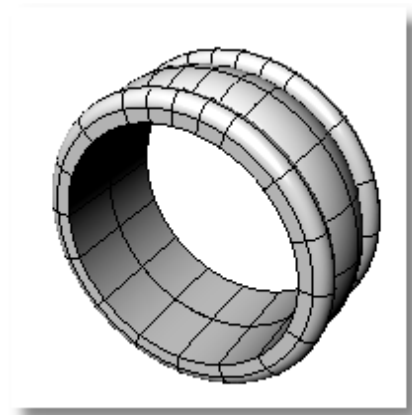
- ▶ In the **Right** viewport, double-click the viewport title.

### To finish the band ring

- 1 On the **Surface** menu, click **Sweep 1 Rail**.
- 2 At the **Select rail curve** prompt, click the circle (1).
- 3 At the **Select cross section curves ...** prompt, click the cross-section you just created (2), and press ENTER.



- 4 In the **Sweep 1 Rail Options** dialog box, click **OK**.



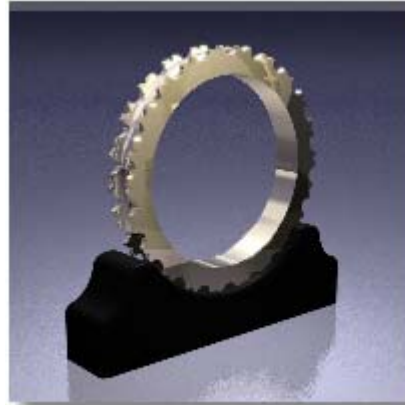


## 4 Create an Eternity Ring with Claw Settings

In this tutorial, you will learn how to make a ring with a specified number of elements evenly spaced around the shank of the ring. The ring we will make will have twenty-two elements.

In an eternity ring you usually have the problem of matching the size and a specific number of stones, or you may have a constraint given by the size of the stones. The polar array command will solve this problem.

A polar array is a repetition of items around a center where you can specify the number of elements and the angle to fill.



In addition to practicing techniques you learned in the previous chapters, you will learn how to:

- **Array** objects around a circle
- **Trim** and **chamfer** curves
- Make a **revolved** surface

---

**Note:** You should complete the tutorials in the previous chapters before starting this tutorial.

---

### To start the model

- ▶ On the **File** menu, click **New**, select **Millimeters.3dm**, and then click **Open**.

### To turn on modeling aids

- ▶ On the **status bar**, click the **Snap** and **Osnap** panes.

## Modeling the Shank

We will start by making an outline of the shank.

### To maximize the Front viewport

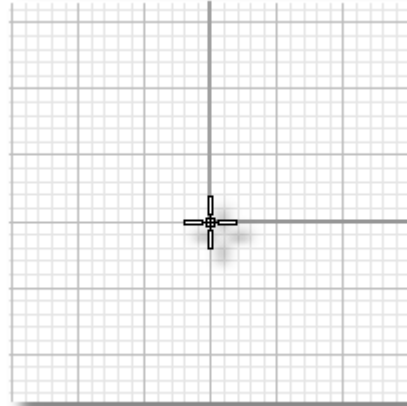
- ▶ In the **Front** viewport, double-click the viewport title.

### To make the inside of the shank

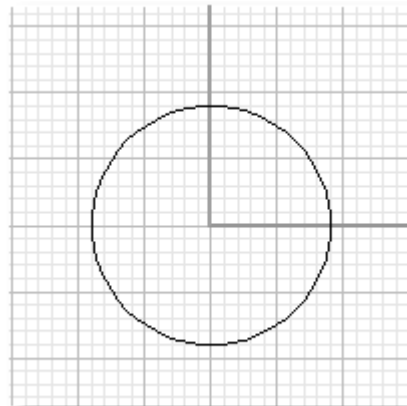
- 1 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.



- 2 At the **Center of circle...** prompt, move your mouse pointer to the origin of the viewport and click.



- 3 At the **Radius ...** prompt, type **9**, and press ENTER.



### Create the Claw to Array

Next we will make a copy of the inside of the ring to use for a construction curve for the outer part of the ring.

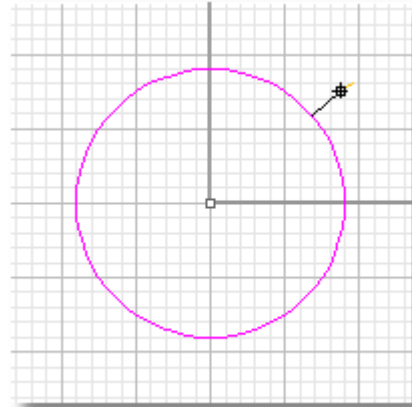
#### Create a construction circle for the outside of the shank

The construction circle provides the outer boundary of the arrayed elements



### To create the construction circle

- 1 **Select** the circle you just created.
- 2 On the **Curve** menu, click **Offset Curve**.
- 3 At the **Through point ...** prompt, type **.2**, and press ENTER.
- 4 At the **Through point ...** prompt, click outside the first circle.

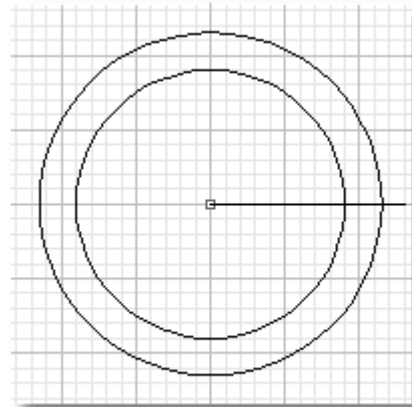


### Create construction lines

Next we will make the rest of the construction lines for the claw elements. We will start by making a single line and arraying around the circle 22 times. This will give us the proper spacing for the elements. The spacing is determined by the number of stones that will be placed in the ring.

### To make a construction line

- 1 On the **Osnap** toolbar, select the **Cen** check box.
- 2 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 3 At the **Start of line ...** prompt, move your mouse pointer over one of the circles, and click when the **Cen** osnap tooltip appears.
- 4 At the **End of line ...** prompt, click a point **2 mm** to the right of the right quadrant of the outer circle.

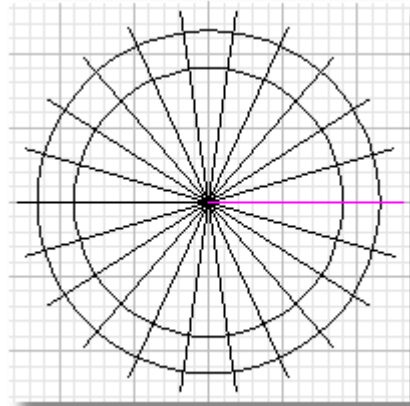


### To array the construction line in a circle

- 1 **Select** the line you just created.
- 2 On the **Transform** menu, click **Array**, and then click **Array Polar**.
- 3 At the **Center of polar array** prompt, snap to the center of the circles.
- 4 At the **Number of items ...** prompt, type **22**, and press ENTER.



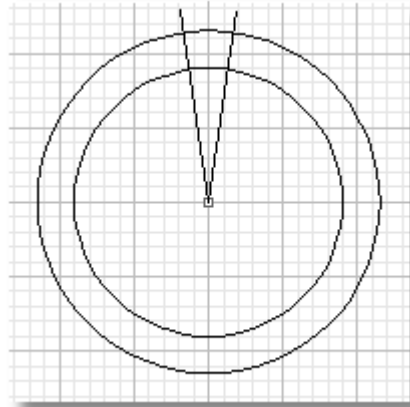
- 5 At the **Angle to fill <360>** prompt, press ENTER to accept the default value.



#### To delete the unnecessary lines

Now that the spacing for the elements has been established by the array, you can delete all but the two lines that you will use to construct the claw feature.

- ▶ **Select** all of the lines except the two that are located at either side of the top of the circle, and press DELETE.

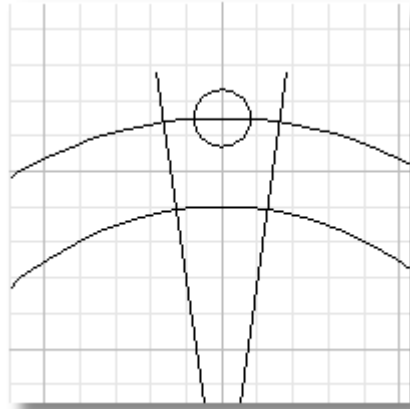


#### To make the construction curves for the starting element

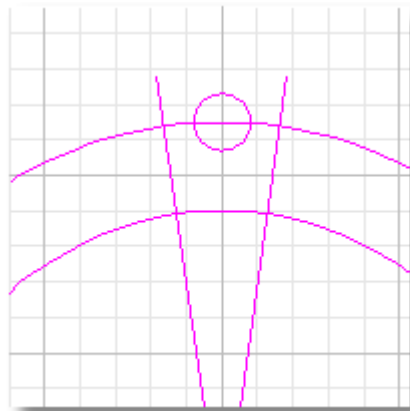
- 1 On the **Osnap** toolbar, select the **Quad** check box, and clear the **Cen** check box.
- 2 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.



- 3 At the **Center of circle ...** prompt, use the **Quad** osnap to place the center of the circle at the top quadrant of the outside construction circle.
- 4 At the **Radius ...** prompt, type **.8**, and press ENTER.



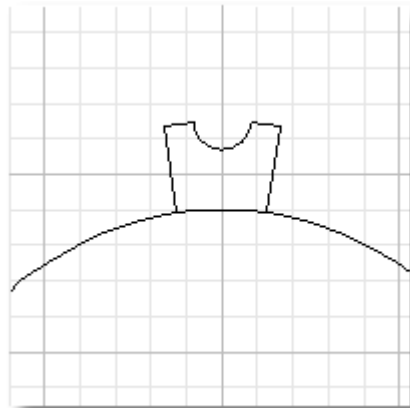
- 5 **Select** the three circles and two lines.



- 6 On the **Edit** menu, click **Trim**.
- 7 At the **Select object to trim** prompt, click the tops of the two lines, the left and middle of the outside circle, and the top of the small circle, and press ENTER.

The point on the curve you pick determines the part that will be removed.

The result should match the illustration.

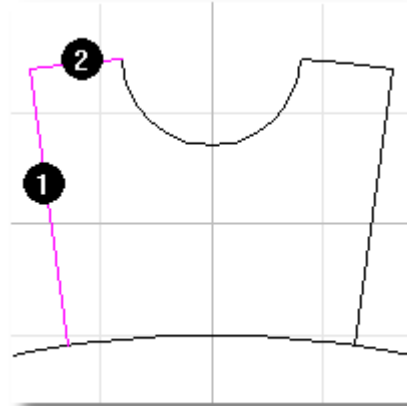


### To chamfer the edges of the starting element

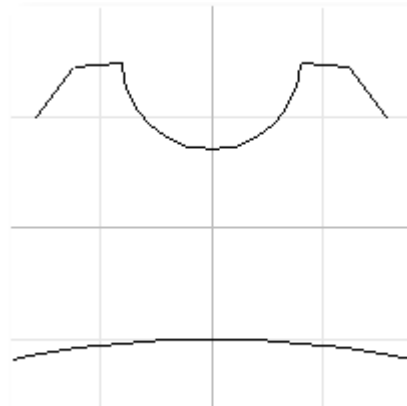
- 1 On the **Curve** menu, click **Chamfer Curves**.
- 2 At the **Select first curve to chamfer ...** prompt, click **Distances**.
- 3 At the **First chamfer distance ...** prompt, type **.4**, and press ENTER.



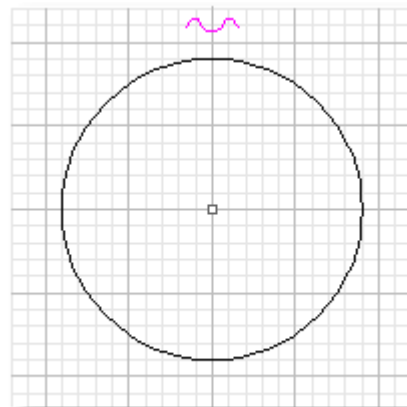
- 4 At the **Second chamfer distance ...** prompt, type **.4**, and press ENTER.
- 5 At the **Select first curve to chamfer ...** prompt, click on the line **(1)**.
- 6 At the **Select second curve to chamfer ...** prompt, click the arc segment **(2)**.



- 7 **Repeat** these steps to chamfer the line and the arc segment on the right side.
- 8 **Delete** the left and right construction lines.



- 9 **Select** the four remaining lines and arcs.
- 10 On the **Edit** menu, click **Join**.





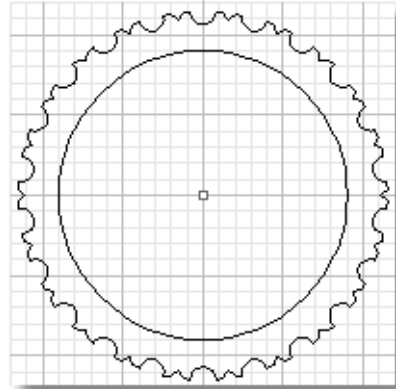


## Array the Claw Profile Around the Ring

Next we will array the single claw feature around the ring.

### To array the claw profile

- 1 On the **Osnap** toolbar, select the **Cen** check box, and clear the **Quad** check box.
- 2 **Select** the joined curves you just created.
- 3 On the **Transform** menu, click **Array**, and then click **Array Polar**.
- 4 At the **Center of polar array** prompt, snap to the center of the circle.
- 5 At the **Number of items <22>** prompt, press ENTER.
- 6 At the **Angle to fill <360>** prompt, press ENTER.



## Extruding the Curves to Create the Solid Ring

The next step is to extrude the inside and the outside curves to form the main part of the band. This section will be easier if you can see all four viewports.

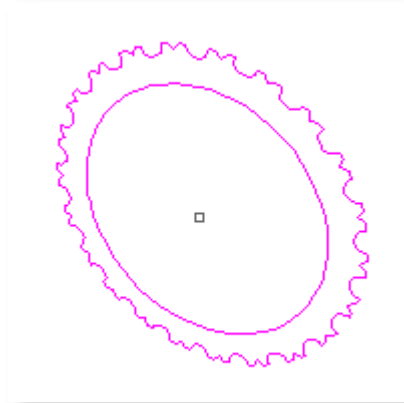
### To restore the four-viewport layout

- 1 In the **Front** viewport, double-click the viewport title.
- 2 On the **View** menu, click **Zoom**, and then click **Extents All**.

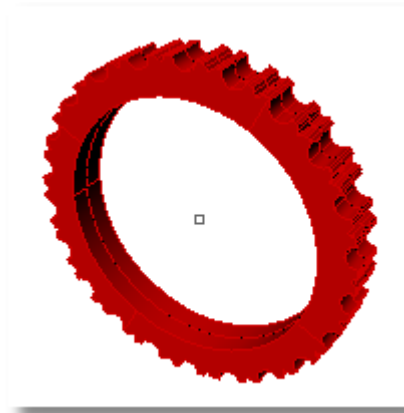


### To extrude the shank curves

- 1 Set the current layer to **Layer 01**.
- 2 **Select** the claw curves and the circle.
- 3 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 4 At the **Extrusion distance ...** prompt, click **BothSides**.



- 5 At the **Extrusion distance ...** prompt, type **1.5**, and press ENTER.  
  
The curves are extruded 1.5 mm on either side of the original curves giving the ring a thickness of 3 mm.



### Creating the Channel

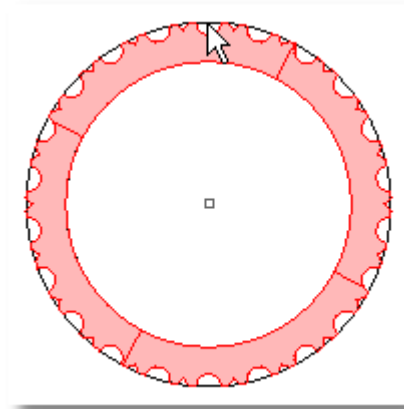
The next step is to make a revolved surface to cut out a channel in the outside of the ring.

#### To make the construction curves for the channel

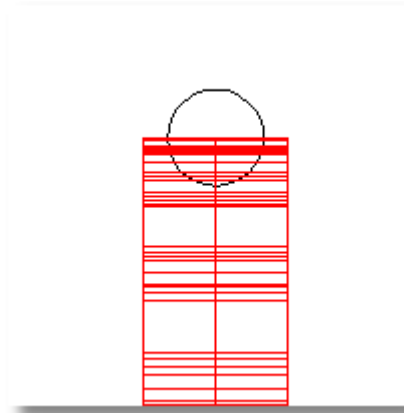
- 1 Set the current layer to **Default**.
- 2 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.



- 3 At the **Center of circle ...** prompt, use the **Cen** object snap to start the circle at the center of the shank circle.
- 4 At the **Radius ...** prompt, type **11.5**, and press ENTER.



- 5 On the **Osnap** toolbar, select the **Quad** check box, and clear the **Cen** check box.
- 6 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.
- 7 At the **Center of circle ...** prompt, in the **Right** viewport, use the **Quad** object snap to the circle quadrant at the top of the construction circle you just created.
- 8 At the **Radius ...** prompt, type **1**, and press ENTER.



#### To make the revolved surface

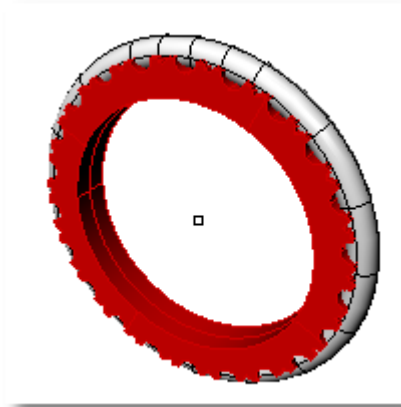
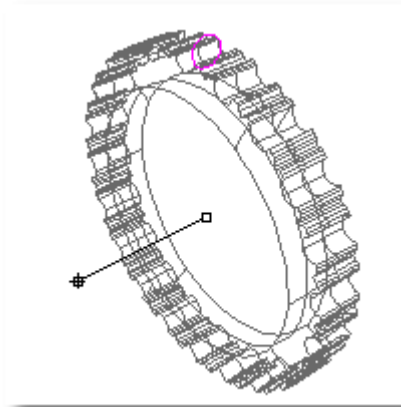
- 1 Turn **Ortho** on.
- 2 **Select** the circle you just created.
- 3 On the **Surface** menu, click **Revolve**.
- 4 At the **Start of revolve axis** prompt, type **0**, and press ENTER.

This sets start point at the construction plane origin.



- 5 At the **End of revolve axis** prompt, move your mouse pointer to the **Perspective** viewport, and then along the world y-axis and click.

A tubular surface is created around the circumference of the ring.

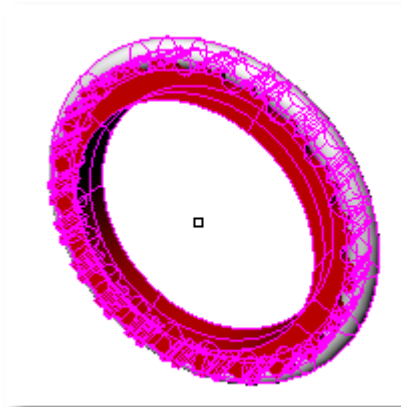


### Cut the Channel

We will finish the ring by using the revolved surface to cut a channel in the ring.

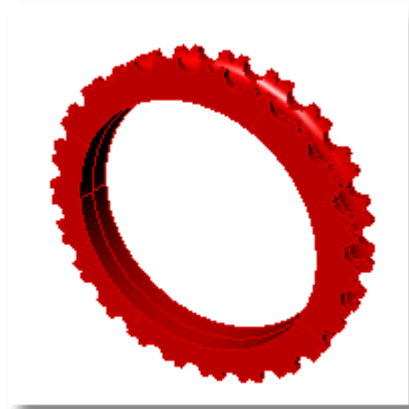
#### To cut the channel

- 1 **Select** the ring solid.
- 2 On the **Solid** menu, click **Difference**.

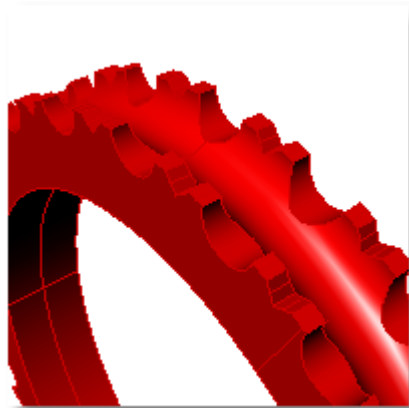




- 3 At the **Select second set of surfaces or polysurfaces ...** prompt, select the revolved surface, and press ENTER.



The revolved surface is removed from the outside of the ring.



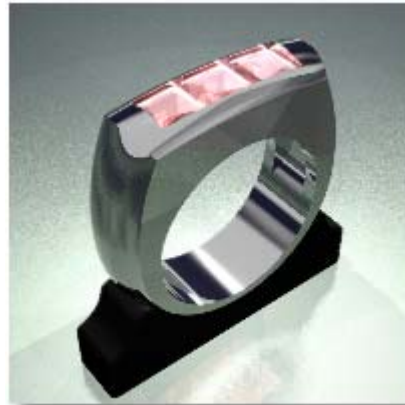


## 5 Create a Ring with Gemstones

In this tutorial you will learn how to make another style of extruded ring.

In addition to practicing the techniques you learned in the previous chapters, you will learn how to:

- **Import** a model to add pre-made features
- **Rotate** an object
- **Copy** curves
- Add control points to a curve by **inserting knots**
- **Hide** curves
- Make a surface **lofted** between curves
- Make surfaces from **planar curves**
- Use a **sweep a curve along two rail curves** to make surfaces
- **Join** surfaces into a closed polysurface



---

**Note:** You should complete the tutorials in the previous chapters before starting this tutorial.

---

### To start the model

- ▶ On the **File** menu, click **New**, select **Millimeters.3dm**, and then click **Open**.

### To turn on modeling aids

- ▶ On the **status bar**, click the **Snap** and **Osnap** panes.

## Draw the Initial Shank Outline Shape

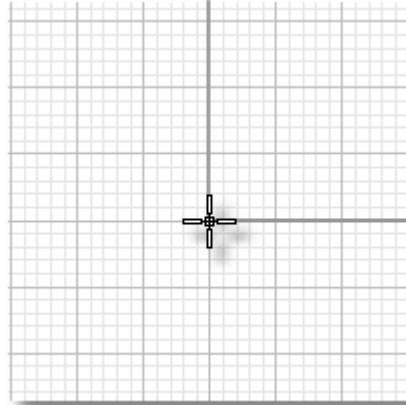
We will start by making an outline of the shank.

### To maximize the Front viewport

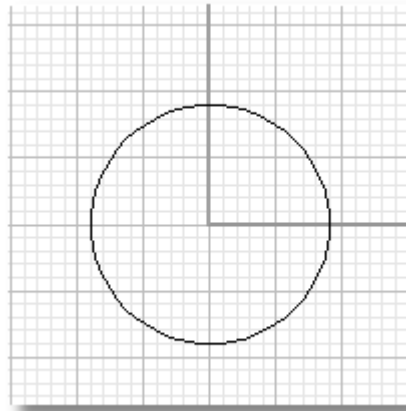
- ▶ In the **Front** viewport, double-click the viewport title.

**To make the inside of the shank**

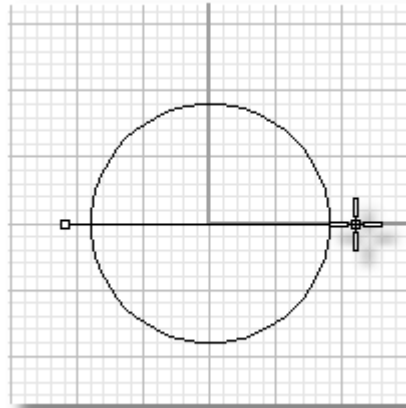
- 1 On the **Curve** menu, click **Circle**, and then click **Center, Radius**.
- 2 At the **Center of circle ...** prompt, move your mouse pointer to the origin of the viewport and click.



- 3 At the **Radius ...** prompt, type **9**, and press ENTER.

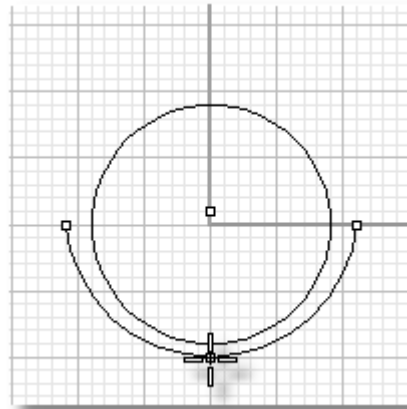
**To make the outside of the shank**

- 1 On the **Curve** menu, click **Arc**, and then click **Start, End, Point**.
- 2 At the **Start of arc** prompt, click a point **2 mm** (two grid lines) to the left of the left quadrant of the circle.
- 3 At the **End of arc** prompt, click a point **2 mm** to the right of the right quadrant of the circle.





- 4 At the **Point on arc** prompt, click a point **1 mm** from the bottom of the circle.



## Adding Gemstones

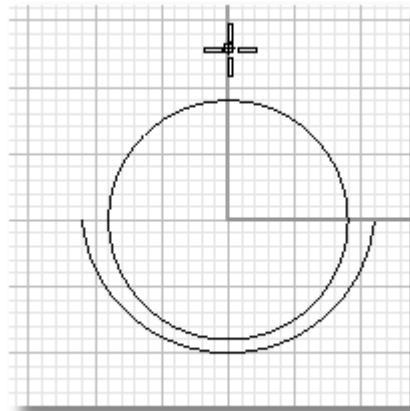
Because the size of the ring depends on the size and number of gemstones, the next step is to insert a model of a stone. We will then scale and position the stone and copy it. The rest of the model will be created around the size and position of the stones.

### Importing a gemstone model

Before importing a gemstone model, we will make a reference point to make placing the stone easier.

#### To make a reference point for inserting the stone

- 1 On the **Curve** menu, click **Point Object**, and then click **Single Point**.
- 2 At the **Location of point object** prompt, click a point **4 mm** above the top of the circle.



- 3 On the **Osnap** toolbar, select the **Point** check box, and clear all other check boxes.

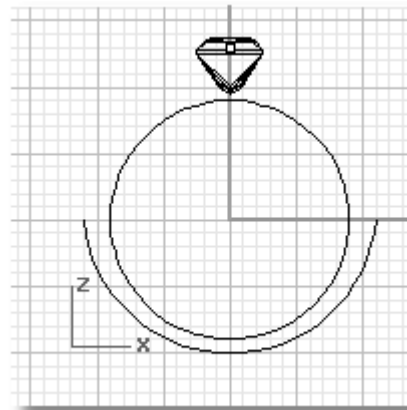
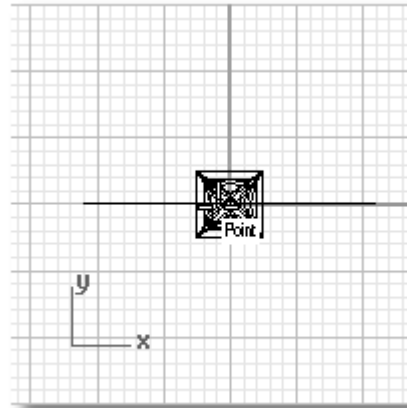
Since the gemstone model we are inserting is exactly one unit (millimeter) square, we can scale it when we insert it. We want to have a stone that is 5 mm square and 5 mm high.





### To insert the stone

- 1 On the **File** menu, click **Insert**.
- 2 On the **Insert** dialog box, click **File** and navigate to the folder where you saved these tutorial files, and select **Square\_05.3dm**.
- 3 Under **Insertion point**, select the **Prompt** check box.
- 4 Under **Scale**, select the **Uniform** check box, in the **X** box type **5.0**, and click **OK**.
- 5 At the **Insertion point** prompt, in the **Top** viewport, use the Point object snap to place the stone at the point object you just created.



### Place another gemstone

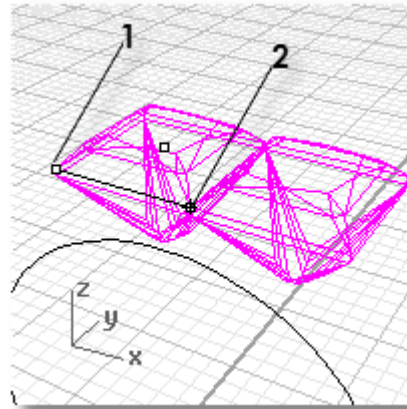
For the next gemstone, you will copy and rotate your original gemstone model.

### To copy the stone

- 1 Maximize the **Perspective** viewport.
- 2 **Select** the stone you just inserted.
- 3 On the **Transform** menu, click **Copy**.

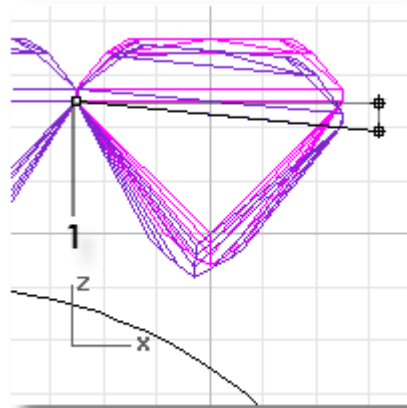


- 4 At the **Point to copy from** prompt, use the **End** object snap to start at the lower left end of the stone **(1)**.
- 5 At the **Point to copy to** prompt, use the **End** object snap to end at the lower right end of the stone **(2)**.



### To rotate the stone

- 1 **Maximize** the **Front** viewport.
- 2 **Select** the stone you just copied.
- 3 On the **Transform** menu, click **Rotate**.
- 4 At the **Center of rotation ...** prompt, use the **End** object snap to click the lower left end of the selected stone, as illustrated **(1)**.
- 5 At the **Angle or first reference ...** prompt, move your mouse pointer to the right and click.
- 6 Turn **Ortho** off.  
- Or -  
Hold SHIFT to suspend Ortho for the next mouse click.
- 7 At the **Second reference ...** prompt, move your mouse pointer down slightly and click.



### Place the third gemstone

You will now mirror the second gemstone around the center of the ring to create a third gemstone. In addition to the gemstone, we want to create a small construction line that will help define the edge of the mounting. You will also mirror this line to the other side to make the setting symmetrical.

### To make a construction line to help define the ring edge

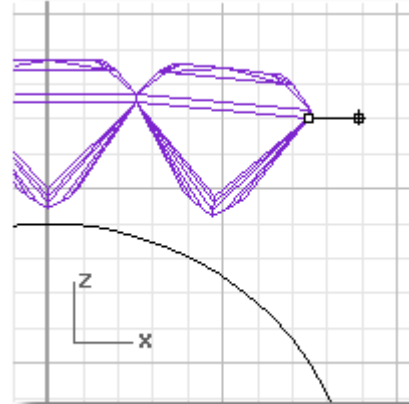
- 1 Turn **Ortho** on.
- 2 On the **Curve** menu, click **Line**, and then click **Single Line**.



- 3 On the **Osnap** toolbar, click **Project**.

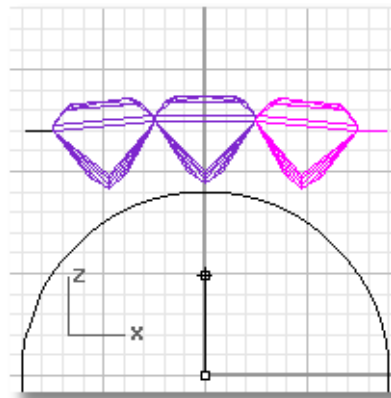
The Project object snap mode projects the object snap point to the construction plane. When you start the line using the End object snap, instead of snapping to the corner of the stone, the snap point is projected to the construction plane and the line will start at the center of the stone's edge where the construction plane and the stone intersect.

- 4 At the **Start of line ...** prompt, use the **End** object snap projected to the construction plane to start the line at the right edge of the stone as illustrated.
- 5 At the **End of line** prompt, type **1.4**, and press ENTER.
- 6 At the **End of line** prompt, move your mouse to the right and click.



#### To mirror the stone and the construction line

- 1 **Select** the copied stone and the line you just created.
- 2 On the **Transform** menu, click **Mirror**.
- 3 At the **Start of mirror plane ...** prompt, use the **Cen** object snap to start the mirror plane at the center of the ring.
- 4 At the **End of mirror plane ...** prompt, use **Ortho** to click directly above the first point.



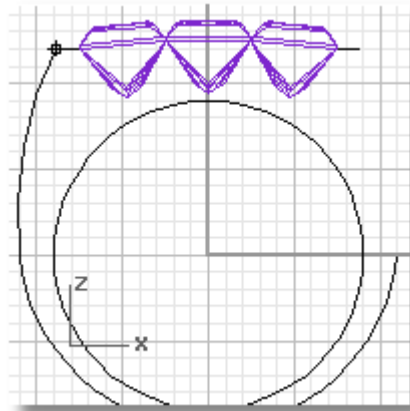


## Making the Finished Shank Curves

We can now add the curves that define the shape of the interior and exterior of the shank.

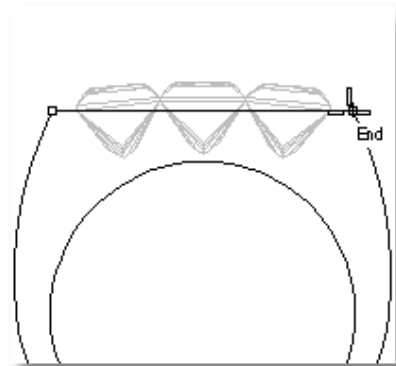
### To create the lower outside edge curve

- 1 On the **Curve** menu, click **Extend**, and then click **By Arc to a Point**.
- 2 At the **Select curve to extend ...** prompt, click the upper left end of the arc that will form the bottom of the shank.
- 3 At the **End of extension ...** prompt, use the **End** object snap to end the arc extension at the left end of the top arc.
- 4 **Repeat** this command to extend the arc on the right side of the shank.



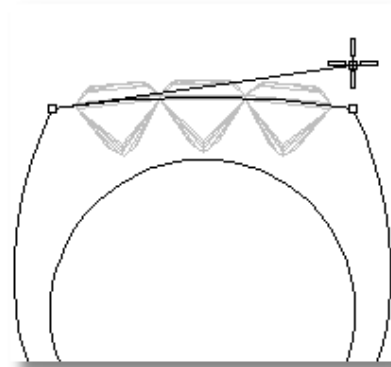
### To make an arc that defines the shape at the top

- 1 Turn **Ortho** off.
- 2 On the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.
- 3 At the **Start of arc ...** prompt, use the **End** object snap to start the arc at the top left end of the arc you just created.
- 4 At the **End of arc ...** prompt, use the **End** object snap to end the arc at the top right end of the arc you just created.





- 5 At the **Direction at start ...** prompt, move your mouse pointer up until the curve roughly follows the profile of the stones as shown and click.

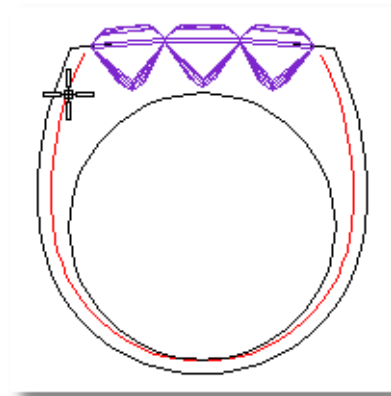


## Making the Interior Opening

Next we will make a copy of the outside of the ring to use to cut out the inside of the shank and the opening for the stones in the top of the ring.

### To make the curve that defines the inside of the shank

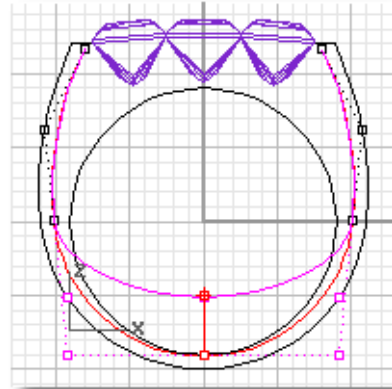
- 1 Set the current layer to **Layer 01**.
- 2 **Select** the curve that forms the outer part of the ring.
- 3 On the **Curve** menu, click **Offset Curve**.
- 4 At the **ThroughPoint ...** prompt, type **.8**, and press ENTER.
- 5 At the **ThroughPoint ...** prompt, click inside the selected curve.





### To edit the curve shape

- 1 **Select** the offset curve.
- 2 On the **Edit** menu, click **Control Points**, and then click **Control Points On**.
- 3 **Select** the three control points at the bottom of the curve.
- 4 On the **Transform** menu, click **Move**.
- 5 At the **Point to move from** prompt, click a point in the center of the ring.
- 6 At the **Point to move to** prompt, click a point approximately **4 mm** above the first point.



### Edit the interior curve's shape

To make adjustment to the upper part of the offset curve without changing the lower part of the curve, it will be necessary to add additional control points.

One way to add control points to the curve with control over their position, is to add knots to the curve. For each knot we add, a control point will also be added.

---

**Note:** A knot is property of the curve where the mathematical definition of the curve changes. Imagine a rope. If you hold it at the ends, the rope will sag according to the laws of nature (gravity, the stiffness of the rope, etc.) with a mathematical definition. If you tie it off somewhere along its length (by putting knots in it), there will be a different definition (sag) for each segment between the knots.

---

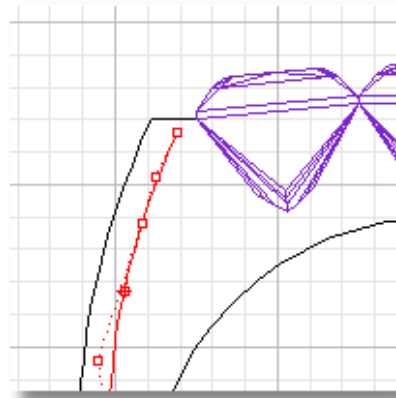
### To add control points

- 1 On the **Edit** menu, click **Control Points**, and then click **Insert Knot**.
- 2 At the **Point on curve to add knot ...** prompt, click approximately half way between the two uppermost control points on the offset curve.

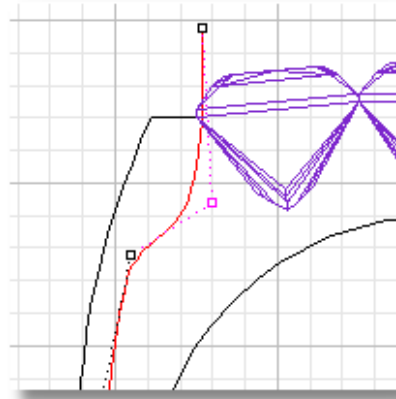


- 3 At the **Point on curve to add knot ...** prompt, click approximately half way between the inserted knot and the next control point down the offset curve.
- 4 At the **Point on curve to add knot ...** prompt, press ENTER to end the command.

The new control points will not be visible until the **InsertKnot** command has ended.



- 5 Drag the control points to match the illustration at the right.  
  
It is important that the vertical segment at the top intersects the edge of the stone.
- 6 On the **Edit** menu, click **Control Points**, and then click **Control Points Off**.



### To create the other side of the curve

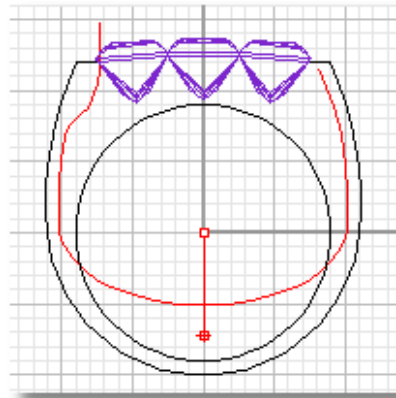
Since it would be difficult to repeat the previous steps exactly to make the other side of the curve symmetrical, the next two steps demonstrate how to make the curve symmetrical on each side of the ring.

- 1 Turn **Ortho** on.
- 2 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 3 At the **Start of line ...** prompt, type **0**, and press ENTER.

This starts the line at the construction plane origin.

- 4 At the **End of line** prompt, move your mouse down and click.

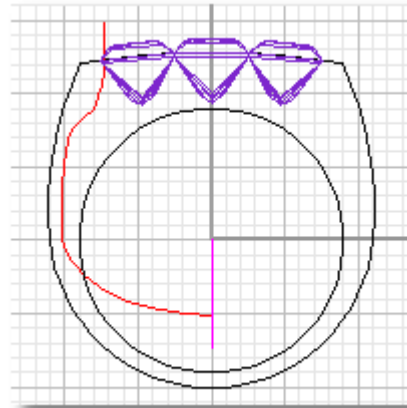
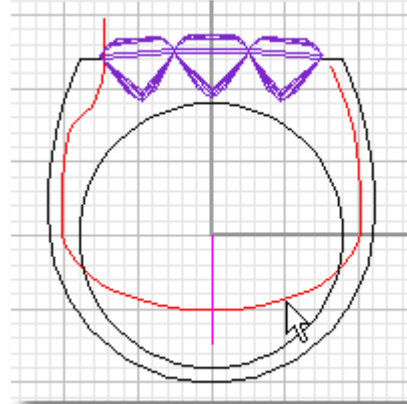
Make sure your line intersects the offset curve.





### To trim the offset curve

- 1 **Select** the line you just created.
- 2 On the **Edit** menu, click **Trim**.
- 3 At the **Select object to trim** prompt, click the right side of the offset curve.
- 4 At the **Select object to trim** prompt, press ENTER to end the command.



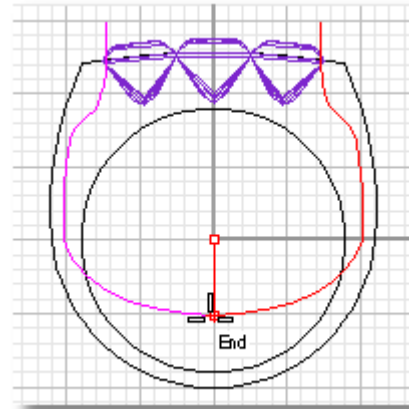
### To mirror the curve

- 1 **Select** the trimmed offset curve.
- 2 On the **Transform** menu, click **Mirror**.





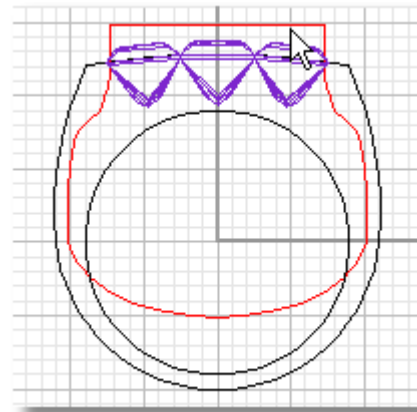
- 3 At the **Start of mirror plane ...** prompt, use the **Cen** object snap to start the mirror plane in the center of the ring.
- 4 At the **End of mirror plane ...** prompt, use the **End** object snap to end the mirror plane at the end of the offset curve.



To make the closed solid you will use to cut out the interior of the ring, connect the two arcs with a line and join these pieces.

#### To close the curves

- 1 On the **Curve** menu, click **Line**, and then click **Single Line**.
- 2 At the **Start of line ...** prompt, use the **End** object snap to start the line at the end of the offset curve.
- 3 At the **End of line** prompt, use the **End** object snap to end the line at the end of the mirrored curve.
- 4 **Select** the line you just created and the two offset curves.
- 5 On the **Edit** menu, click **Join**.



## Making the Curves that Define the Exterior Shape

The next step is to make copies of the curves you created. Then you will make construction curves to be used to make surfaces. Working in the Perspective viewport will make this step easier to complete.

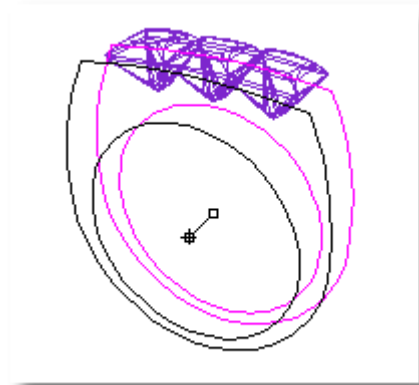
#### To set the layer

- 1 Set the current layer to **Default**.
- 2 Turn **Layer 01** off.

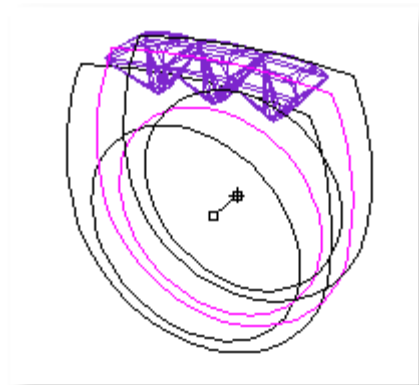


### To copy the shank curves

- 1 **Select** the outer curve, the circle, and the curve for the top.
- 2 On the **Transform** menu, click **Copy**.
- 3 At the **Point to copy from** prompt, type **0**, and press ENTER.
- 4 At the **Point to copy to** prompt, type **3.7**, and press ENTER.

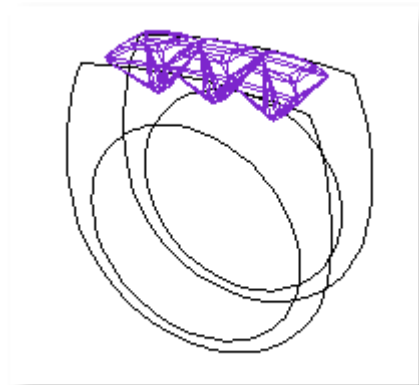


- 5 At the **Point to copy to** prompt, move your mouse pointer along the y-axis and click.
- 6 At the **Point to copy to** prompt, type **3.7**, and press ENTER.
- 7 At the **Point to copy to** prompt, move your mouse pointer the opposite direction along the y-axis and click.
- 8 At the **Point to copy to** prompt, press ENTER to end the command.



- 9 With the original curves still selected, from the **Edit** menu, click **Visibility**, and then click **Hide**.

This will remove the original curves from the view.



### To make a cross-section curve for the side and top surfaces of the ring

- 1 Turn **Ortho** off.
- 2 On the **status bar** click **Planar**.

Planar mode aids in creating planar objects with commands that allow free picking. Successive pick points will have the same construction plane elevation as the first point.



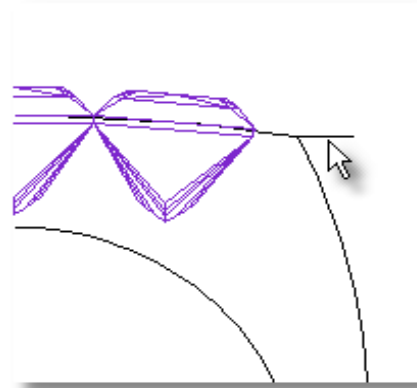
- 3 On the **Osnap** toolbar, click **Project**.

This will turn Osnap projection mode off, letting you to snap to the ends of the curves you copied, which are not on the construction plane.

- 4 On the **Curve** menu, click **Arc**, and then click **Start, End, Direction**.

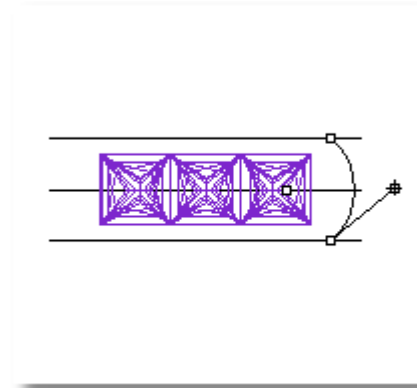
To make the curve, it will be necessary to move your mouse pointer between the **Perspective** viewport and the **Top** viewport while picking points.

- 5 At the **Start of arc ...** prompt, in the **Perspective** viewport use the **End** object snap to start the arc at the top right end of one of the curves you copied.



- 6 At the **End of arc ...** prompt, in the **Perspective** viewport, use the **End** object snap to end the arc click top right end of the other curve you copied.

- 7 At the **Direction at start ...** prompt, move your mouse pointer to the **Top** viewport, drag the arc shape, and click to approximate the direction illustrated.

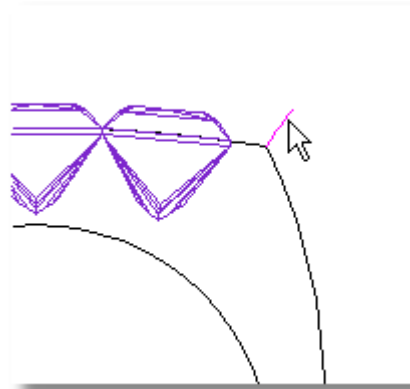
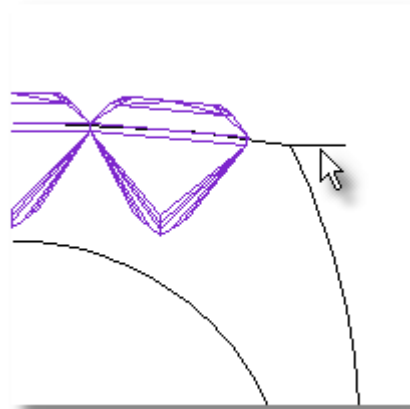


#### To rotate the cross-section curve

- 1 **Select** the curve you just created.
- 2 On the **Transform** menu, click **Rotate**.

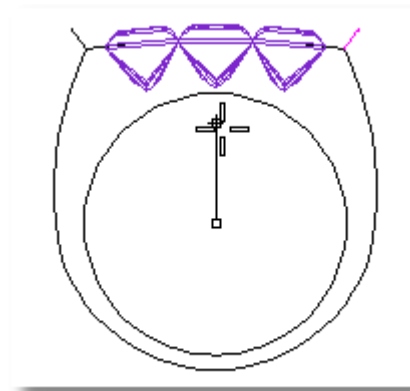


- 3 At the **Center of rotation ...** prompt, in the **Front** viewport , use the **End** object snap to place the center of rotation at the end of the arc.
- 4 At the **Angle or first reference point ...** prompt, move your mouse pointer to the right and click.
- 5 At the **Second reference point ...** prompt, move your mouse pointer up to approximate the direction illustrated.



#### To mirror the cross-section curve

- 1 Turn **Ortho** on.
- 2 **Select** the curve you just created.
- 3 On the **Transform** menu, click **Mirror**.
- 4 At the **Start of mirror plane ...** prompt, type **0**, and press ENTER.
- 5 At the **End of mirror plane ...** prompt, use **Ortho** to move your mouse pointer up and click.



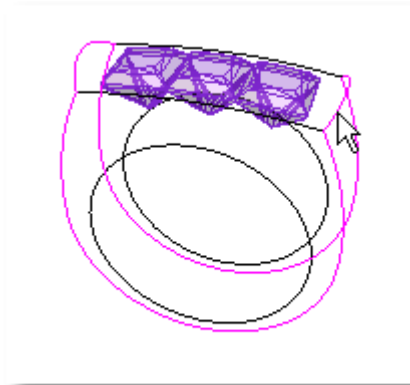


## Making the Exterior Surfaces

The next step is to make the surfaces for the outside of the shank and the top of the ring.

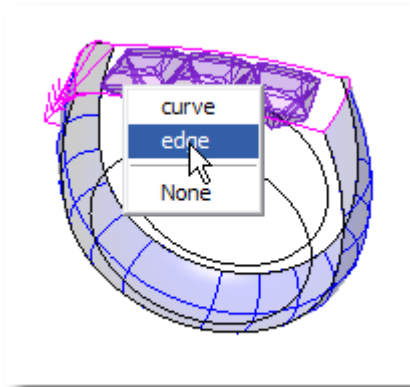
### To make the surface for the outside of the shank

- 1 Set the current layer to **Layer 02**.
- 2 **Select** the two curves that form the shank.
- 3 On the **Surface** menu, click **Sweep 2 Rails**.
- 4 At the **Select cross section curves ...** prompt, click one of the cross-section arcs you just created.
- 5 At the **Select cross section curves. ...** prompt, select the other arc you just mirrored, and press ENTER.
- 6 In the **Sweep 2 Rail Options** dialog box, click **OK**.



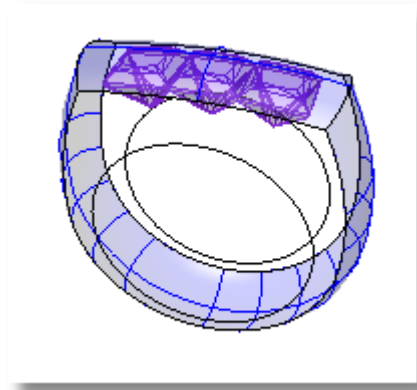
### To make the surface for the top of the ring

- 1 **Select** the two curves at the front and back of the top that run along the sides of the gems.
- 2 On the **Surface** menu, click **Sweep 2 Rails**.
- 3 At the **Select cross section curves ...** prompt, click the two top edges of the surface you just created.
- 4 At the **Select cross section curves ...** prompt, press ENTER.





- 5 In the **Sweep 2 Rail Options** dialog box, click **OK**.

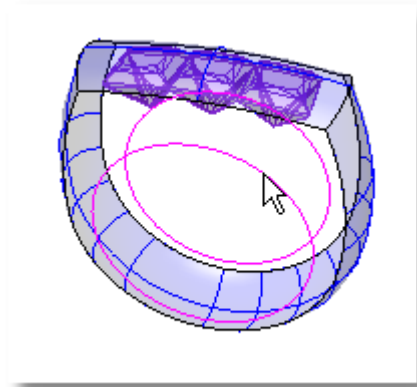


### Making the Interior Surface

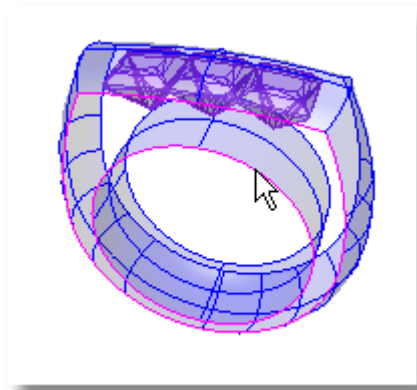
The next step is to make a surface for the inside of the shank.

#### To make the lofted surface

- 1 **Select** the two circles.
- 2 On the **Surface** menu, click **Loft**.
- 3 At the **Adjust curve seams ...** prompt, press ENTER.



- 4 In the **Loft Options** dialog box, click **OK**.



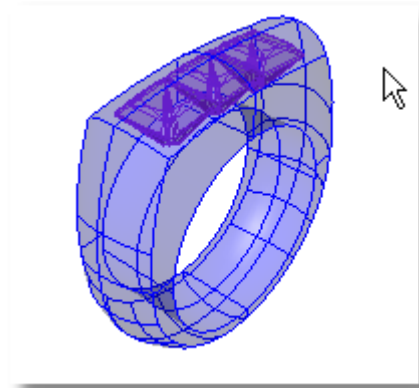


## Making the Front and Back Surfaces

The next step is to make a planar surface for the front and back of the ring.

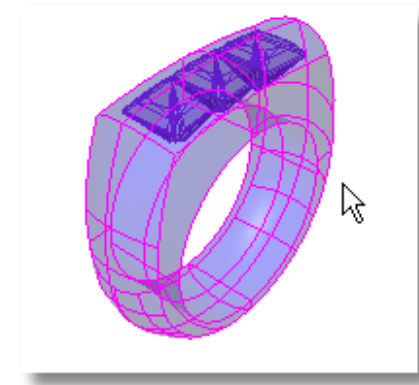
### To make the planar surface

- 1 On the **Surface** menu, click **Planar Curves**.
- 2 At the **Select planar curves to build surface** prompt, select the three surface edges at the front of the ring.
- 3 At the **Select planar curves to build surface ...** prompt, press ENTER.
- 4 **Repeat** these steps for the back of the ring.



### To join the surfaces into one closed polysurface

- 1 **Select** all five surfaces.
- 2 On the **Edit** menu, click **Join**.





## Making a Solid to Cut the Opening in the Top Surface

The next step is to make an extruded surface to cut out part of the outside of the ring.

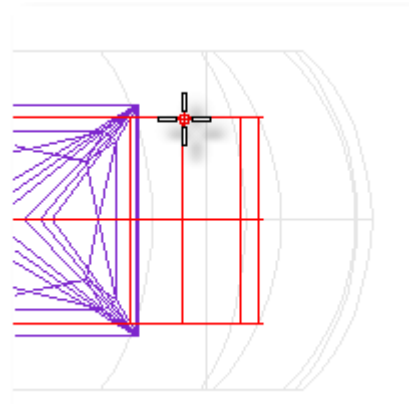
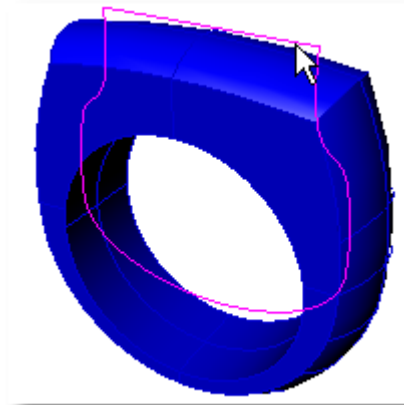
### To extrude the curve

- 1 Set the current layer to **Layer 01**, and turn the **Default** layer off.
- 2 **Select** the curve that defines the interior shape.
- 3 On the **Solid** menu, click **Extrude Planar Curve**, and then click **Straight**.
- 4 At the **Extrusion Distance ...** prompt, click the **BothSides** option.

Set the options as follows:

**BothSides= Yes**  
**Cap= Yes**  
**Mode= Straight**  
**DeleteInput= No**

- 5 At the **Extrusion Distance ...** prompt, in the **Top** viewport, adjust the thickness until it is slightly smaller than the height of the stones, and click.





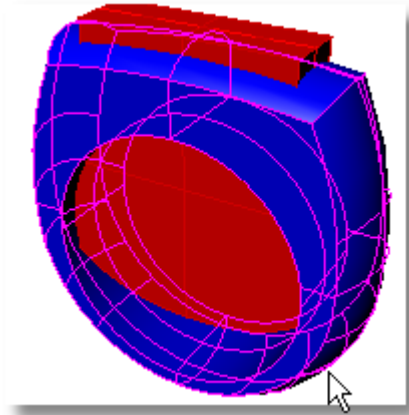


## Cut the Opening in the Top Surface

We will finish the ring by removing the extruded solid from the ring using a Boolean operation.

### To hollow out the ring

- 1 **Select** the ring extrusion.
- 2 On the **Solid** menu, click **Difference**.



- 3 At the **Select second set of surfaces or polysurfaces ...** prompt, select the extruded solid you just created.

