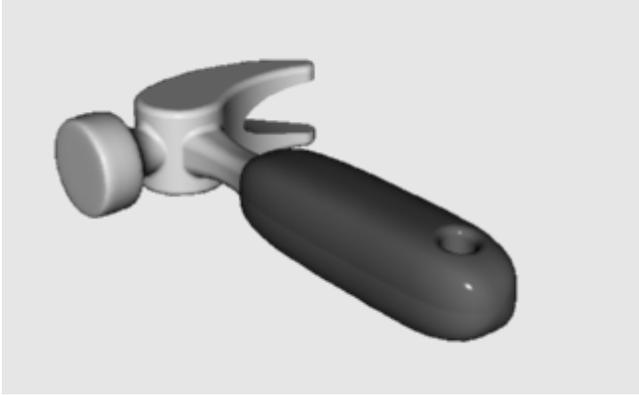


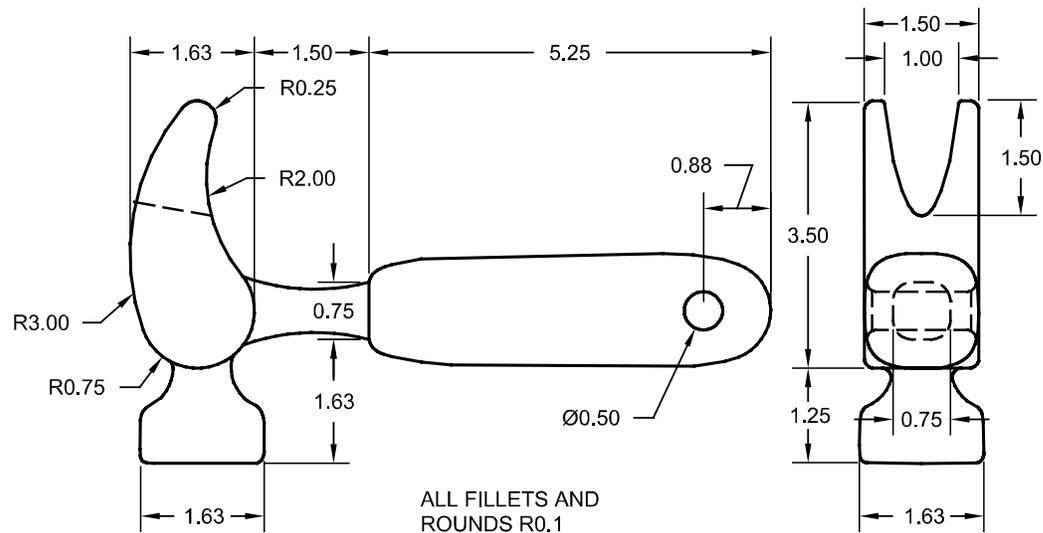
# Modeling a Toy Hammer

Some models require more attention to detail. This is an example of a model that requires precise modeling techniques. This exercise also requires a number of different surface creation techniques. The technical drawing is included to help you create a very precise model.



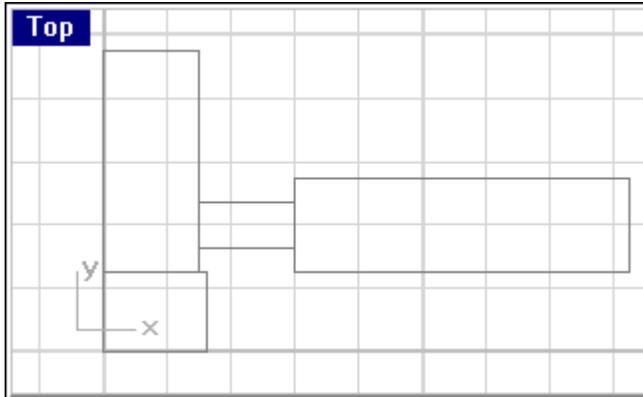
## Creating a toy hammer

- 1 Start a new model. Save as **Hammer**.
- 2 Create the following layers: **Curve, Handle, Tang, Head, Claw**.



3 Draw the outlines for the hammer.

Drawing outlines helps while drawing the curves. You can either draw lines, polylines, or rectangles to create the outlines. Use the dimensions on the technical drawing to get accurate outlines.



To create the claw:

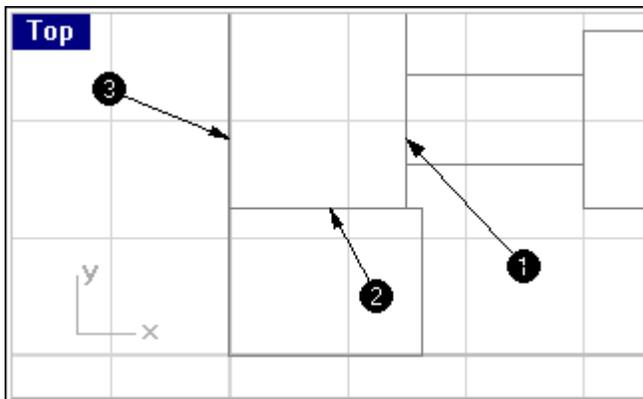
When modeling the shape of the claw you will use circles and arcs. You can trim the circles and arcs and then join them together to create the closed curve.

- 1 Change to the **Curve** layer
- 2 Draw a curve defining the shape of the claw in the top viewport.

You can use a freeform curve or use a combination of arcs and circles which are trimmed and joined to create the curve. Following is a step by step approach to creating the curve for the claw part of the hammer.

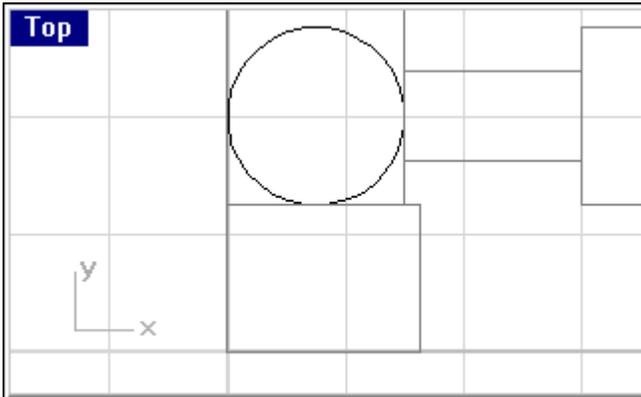
Start by drawing two circles.

- 3 From the **Curve** menu, click **Circle**, then click **Tangent to 3 Curves**.

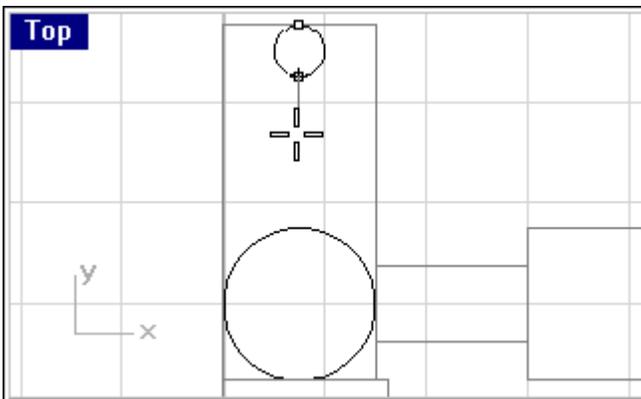


- 4 At the **Select first curve** prompt, select the curve at point one.
- 5 At the **Select second curve** prompt, select the curve at point two.

- 6 At the **Select third curve** prompt, select the curve at point three.  
The circle will be created tangent to all three curves you selected.



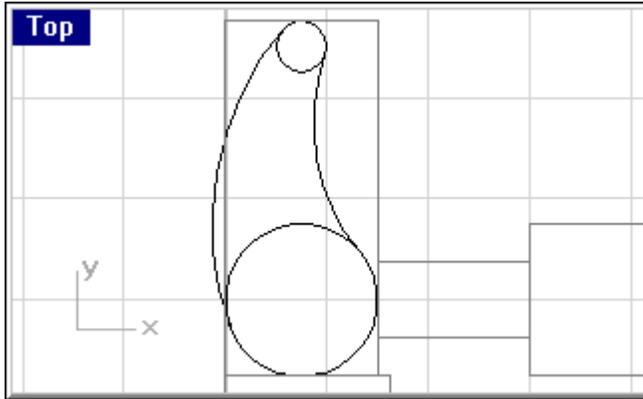
- 7 From the **Curve** menu, click **Circle**, then click **Diameter**.  
8 At the **Start of diameter (Vertical)** prompt, snap to the midpoint of the upper claw construction line.  
9 At the **End of diameter** prompt, type **.5** and press ENTER.  
10 At the **End of diameter** prompt, pick a point directly below the first point.



To create the tangent arcs:

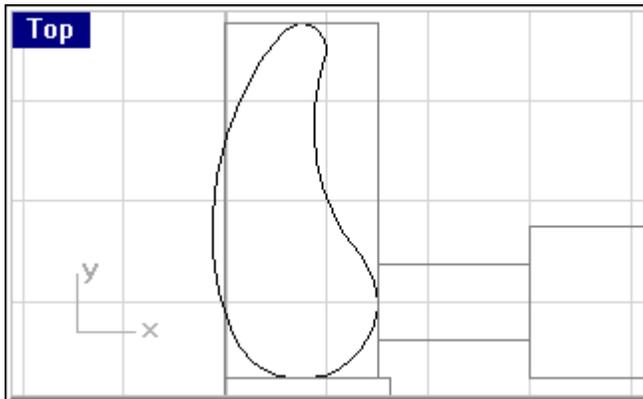
- 1 From the **Curve** menu, click **Arc**, then click **Tangent, Tangent, Radius**.
- 2 At the **Select first curve for ArcTTR** prompt, select a circle near the right edge.
- 3 At the **Select second curve for ArcTTR** prompt, select the other circle near the right edge.
- 4 At the **Radius <1>** prompt, type **2** and press ENTER.
- 5 Delete the parts of the arcs that you don't need.
- 6 From the **Curve** menu, click **Arc**, then click **Tangent, Tangent, Radius**.
- 7 At the **First curve** prompt, select a circle near the left edge.
- 8 At the **Second curve** prompt, select the other circle near the left edge.
- 9 At the **Radius <1>** prompt, type **3** and press ENTER.

- 10 Delete the parts of the arcs that you don't need.



To trim the circles:

- 1 From the **Edit** menu, click **Trim**.
- 2 At the **Select cutting edges (Options)** prompt, select one of the arcs.
- 3 At the **Select cutting edges (Options)** prompt, select the other arc.
- 4 At the **Select cutting edges (Options)** prompt, press ENTER.
- 5 At the **Select object to trim (Options)** prompt, select the insides of the circles.
- 6 At the **Select object to trim (Options)** prompt, press ENTER to end the command.



- 7 **Join** the curve segments.  
The command terminates when the curve closes.

To extrude the claw part of the hammer:

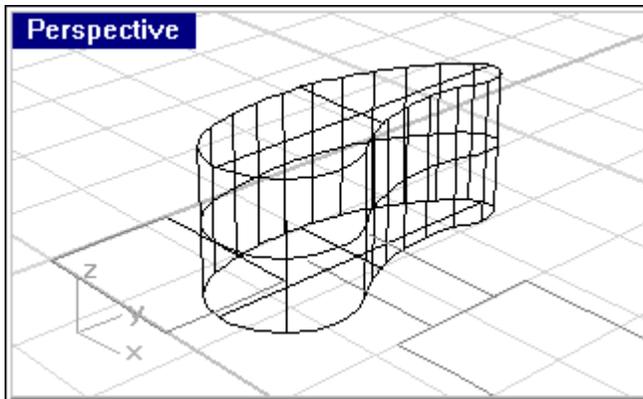
- 1 Change to the **Claw** layer.
- 2 Select the joined segments.
- 3 From the **Surface** menu, click **Extrude**, then click **Straight**.
- 4 At the **Extrusion distance (Direction Cap=no Bothsides Tapered)** prompt, type **C** and press ENTER.

This makes it a solid.

- 5 At the **Extrusion distance (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.

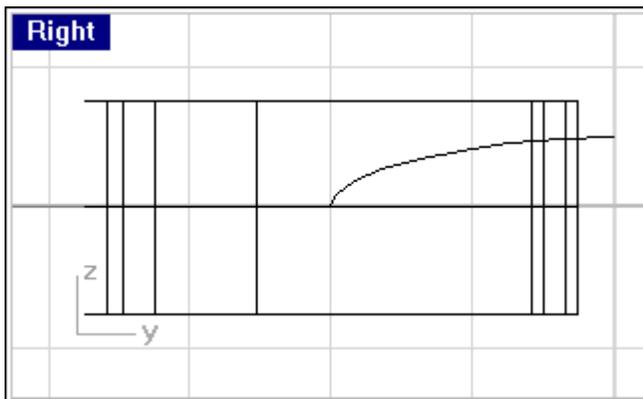
This extends the extrusion on both sides of the construction plane.

- 6 At the **Extrusion distance (Direction Cap=yes)** prompt, type **.75** and press ENTER.

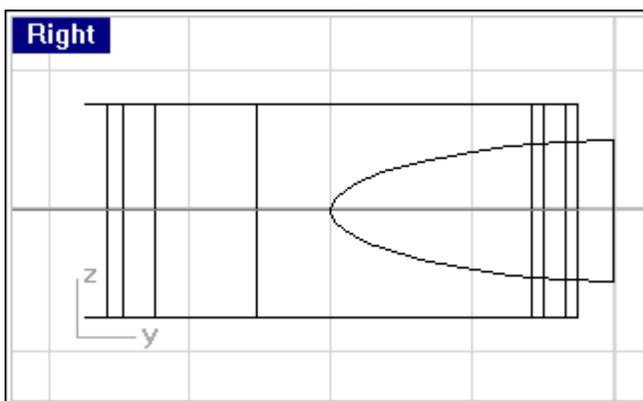


To create slot for the claw part of the hammer:

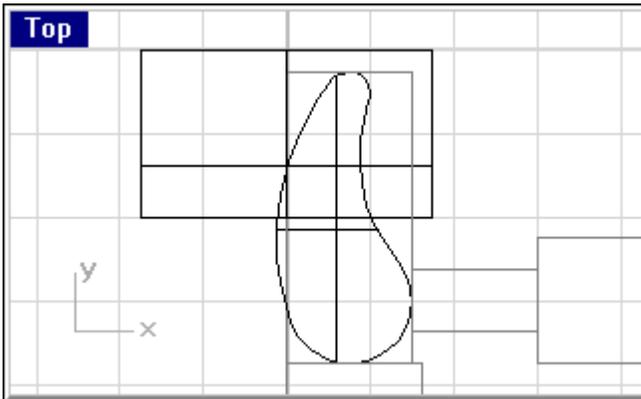
- 1 Change to the **Curve** layer.
- 2 Draw a **Curve** in the **Right** viewport, that starts in the middle of the extruded object, and extends to the right beyond the extrusion.



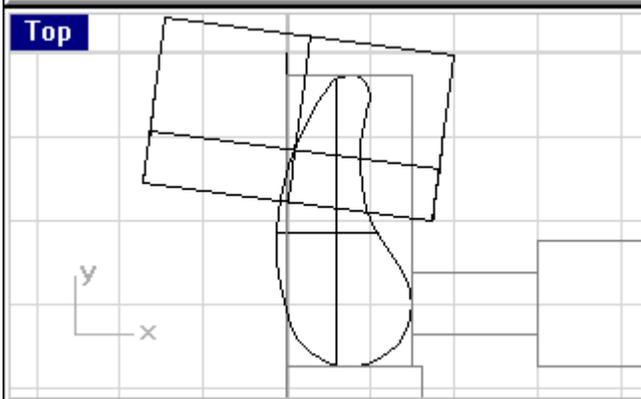
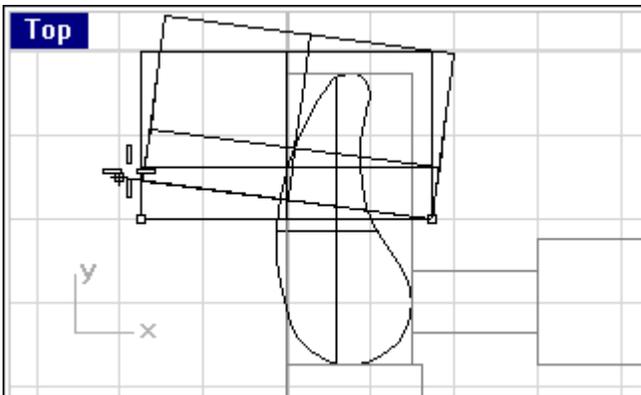
- 3 **Mirror** the curve.
- 4 Draw a **Line** between the end points, then **Join** the three curves.



- 5 Select the joined segments.
- 6 Change to the **Claw** layer.
- 7 From the **Surface** menu, click **Extrude**, then click **Straight**.
- 8 At the **Extrusion distance <.75> (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.
- 9 At the **Extrusion distance (Direction Cap=yes)** prompt, type **2** and press ENTER.



- 10 Rotate the slot in the **Top** viewport

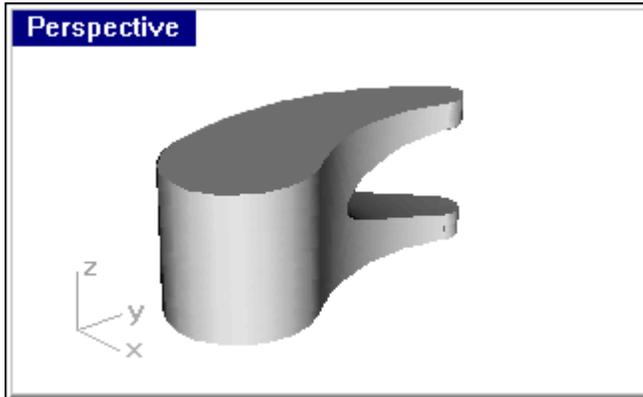


- 11 **Save** your model.

To remove the slot from the claw:

- 1 From the **Solid** menu, click **Difference**.

- 2 At the **Select surface or polysurface** prompt, select the claw extrusion.
- 3 At the **Select surface or polysurface to subtract** prompt, select the Slot extrusion.

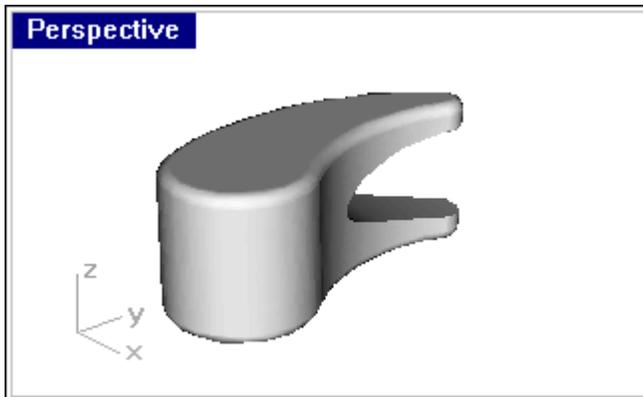


- 4 **Save** your model.

To make the fillets on the claw:

- 1 From the **Solid** menu, click **Fillet Edge**.
- 2 At the **Select edges to fillet ( Radius=1 )** prompt, select the top edge of the claw.
- 3 At the **Select edges to fillet ( Radius=1 )** prompt, select the bottom edge of the claw.
- 4 At the **Select edges to fillet ( Radius=1 )** prompt, type **.1** and press ENTER.
- 5 At the **Select edges to fillet ( Radius=0.1 )** prompt, press ENTER.

The edges have a round on them.

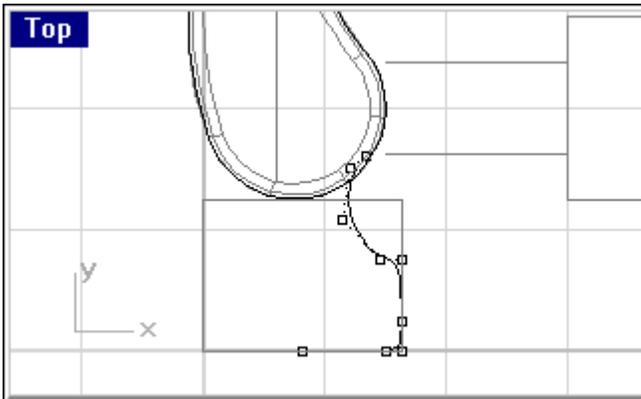


- 6 **Save** your model.

To create the Head:

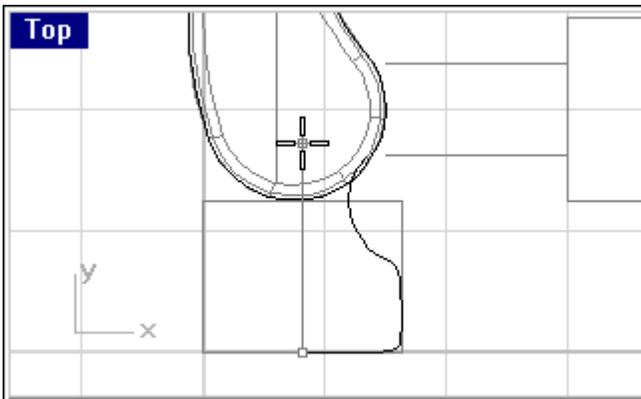
- 1 Change to the **Curve** layer.

- 2 Create the curve for the cross section of the head, using the **Curve** command.

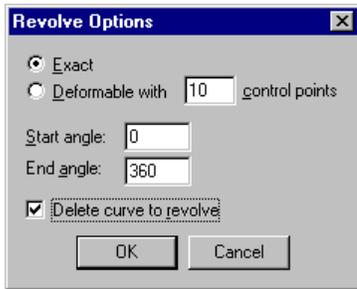


Make sure the curve intersects the claw part. This will make joining the two pieces easier.

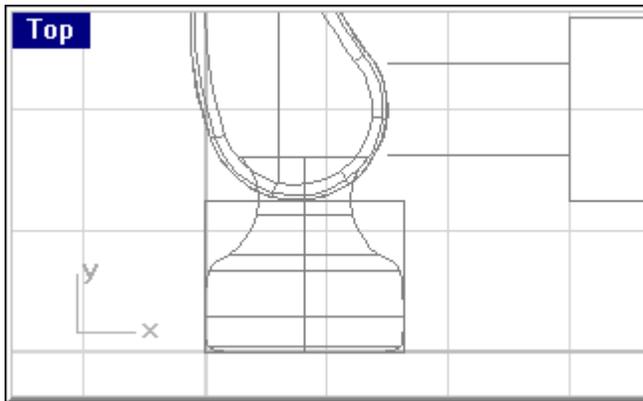
- 3 Change to the **Head** layer.
- 4 From the **Surface** menu, click **Revolve**.
- 5 At the **Select curve to revolve** prompt, choose the polyline for the cross-section of the head.
- 6 At the **Start of revolve axis** prompt, pick the end of the curve.
- 7 At the **End of revolve axis** prompt, toggle ortho on and pick a point above the first point.



- 8 In the **Revolve Options** dialog, click **OK**.



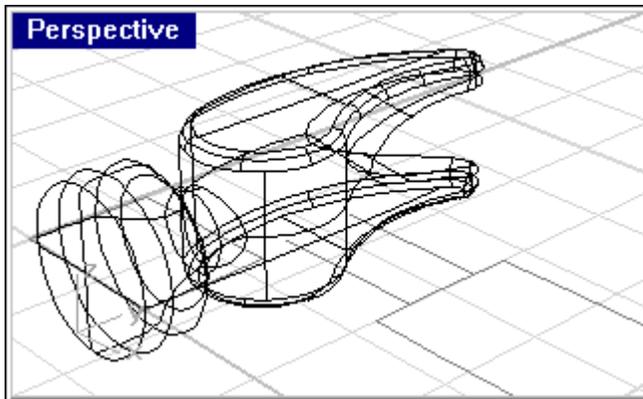
A surface is created.



- 9 **Save** your model.

To join the head and the claw:

- 1 From the **Solid** menu, click **Union**.
- 2 At the **Select surface or polysurface** prompt, select the **Head**.
- 3 At the **Select surface or polysurface to union** prompt, select the **Claw**.

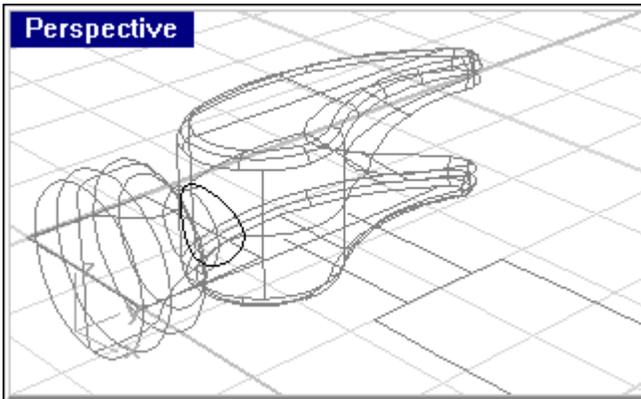


- 4 **Save** your model.

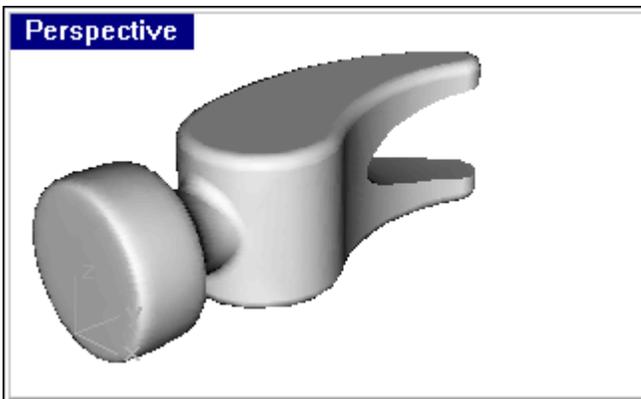
To make the fillet where the head and the claw meet:

- 1 From the **Solid** menu, click **Fillet Edge**.

- 2 At the **Select edges to fillet ( Radius=0.1 )** prompt, select the edge where the two parts join.



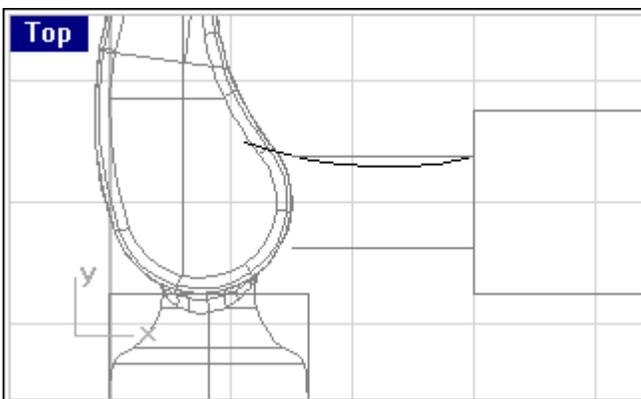
- 3 At the **Select edges to fillet ( Radius=0.1 )** prompt, press ENTER.  
The edge has a round on it.



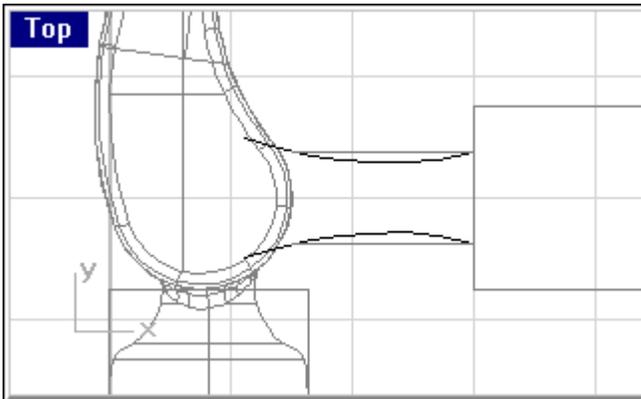
- 4 **Save** your model.

To create the Tang:

- 1 Change to the **Curve** layer.
- 2 Create one of the curves for the tang of the hammer.  
Make sure that it intersects the claw.



- 3 **Mirror** to create the other curve.

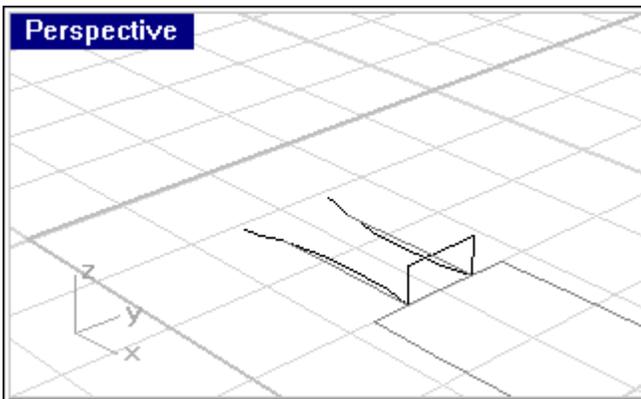


To create the shape curve for the tang and the handle:

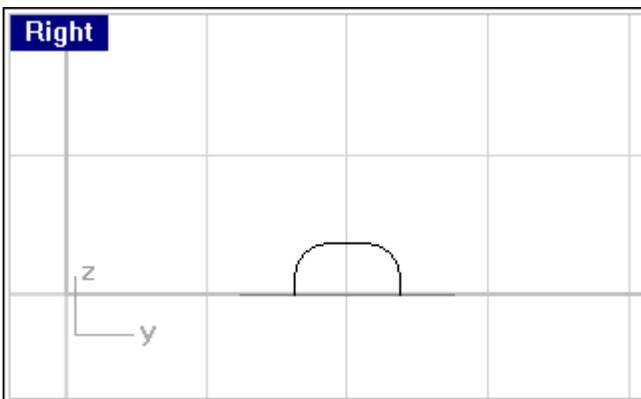
Create the shape curve for the tang in the right viewport. This curve will also be used for the handle.

- 1 Turn on ortho. Use the **Lines** command to create the curve for the tang cross-section curve. This curve will also be used for the handle

The height of the line is **.375** and the width is **.75**.



- 2 **Fillet** the corners with a radius of **.25**.

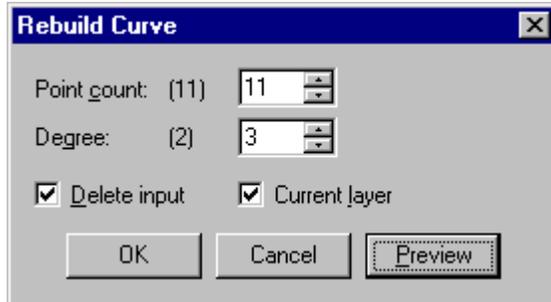


- 3 **Join** the curves
- 4 Select the joined curve.

- From the **Curve** menu, click **Edit Tools**, then click **Rebuild**.

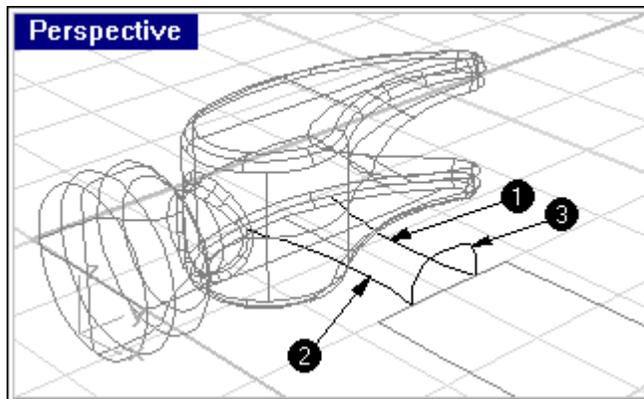
Rebuilding the curve eliminates any kinks, which will make a smooth surface with fewer potential problems later in the modeling process.

- In the **Rebuild Curve** dialog, click **Preview**. If the geometry matches the curve, then click **OK**. If the geometry doesn't match the curve, change the point count and click **Preview**.

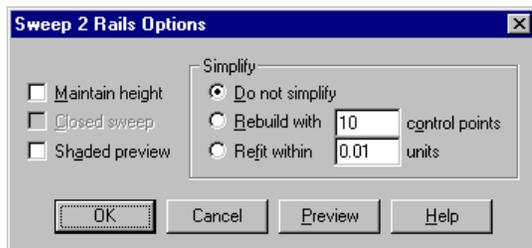


To create the surface for the tang:

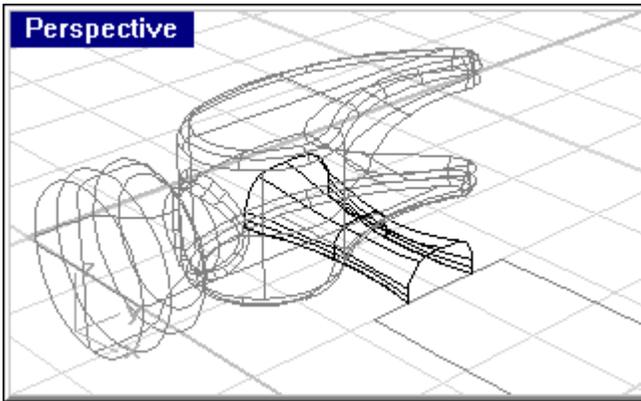
- Change to the **Tang** layer.
- From the **Surface** menu, click **Sweep 2 Rails**.



- At the **Select 2 rail curves** prompt, select one of the rail curves (1).
- At the **Select 2 rail curves** prompt, select the other rail curve (2).
- At the **Select cross-section curves (Point)** prompt, select the shape curve (3).
- At the **Select cross-section curves (Point)** prompt, press ENTER.
- In the **Sweep 2 Rails Options** dialog, click **OK**.

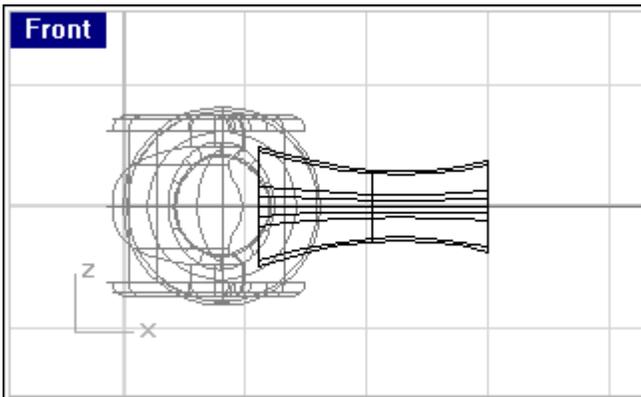


A surface is created.



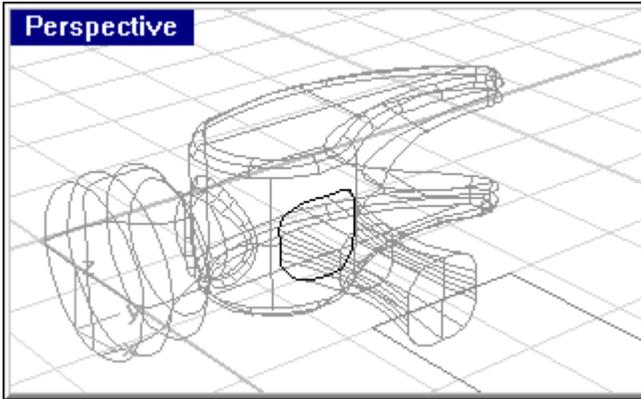
To finish the tang:

- 1 From the **Transform** menu, click **Mirror**.
- 2 At the **Select objects to mirror** prompt, choose the tang surface you created.
- 3 At the **Start of mirror plane** prompt, in the **Front** viewport snap to the endpoint at the lower left corner of the tang surface.
- 4 At the **End of mirror plane** prompt, toggle ortho on and pick a point to the right in the **Front** viewport.



- 5 **Join** the two surfaces together.
- 6 From the **Solid** menu, click **Cap Planar Holes**.
- 7 At the **Select surfaces or polysurfaces to cap** prompt, choose the tang.

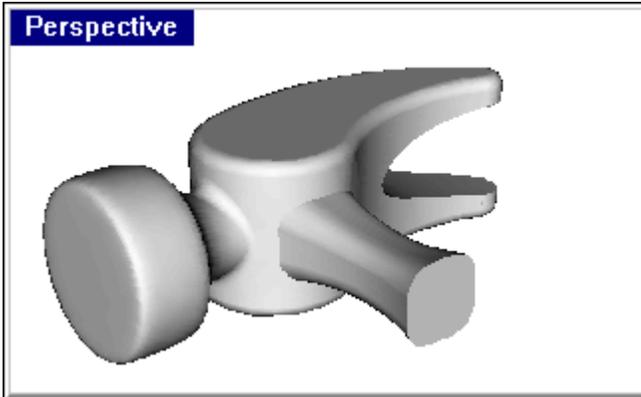
- At the **Select surfaces or polysurfaces to cap** prompt, press ENTER.



- Save your model.

To join the tang and the claw:

- From the **Solid** menu, click **Union**.
- At the **Select surface or polysurface** prompt, select the tang.
- At the **Select surface or polysurface to union** prompt, select the claw.

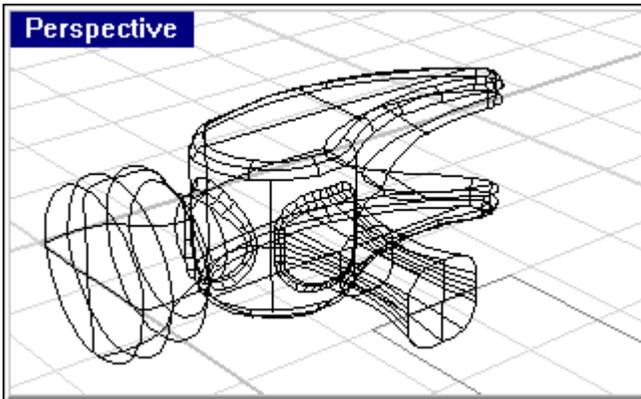


- Save your model.

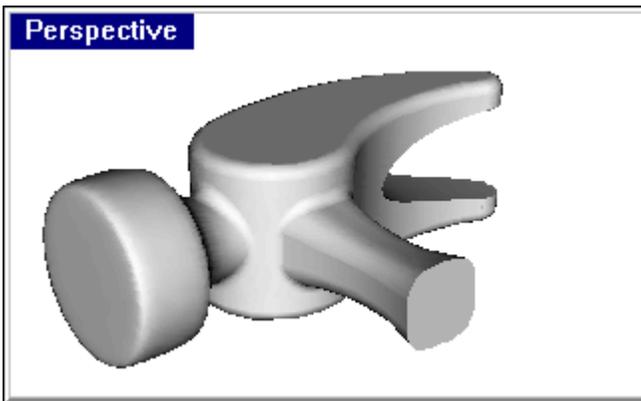
To make the fillet where the tang and the claw meet:

- From the **Solid** menu, click **Fillet Edge**.

- At the **Select edges to fillet ( Radius=0.1 )** prompt, select the edge where the two parts join.



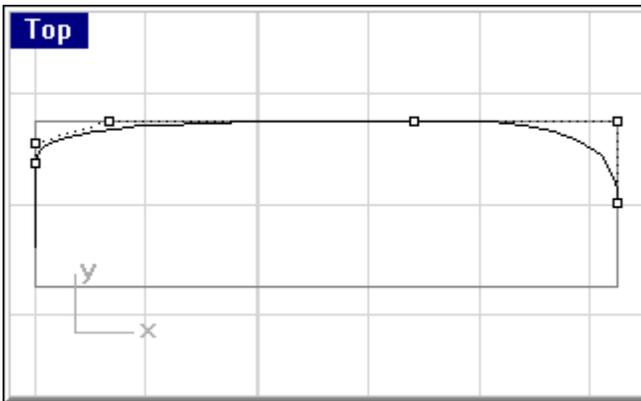
- At the **Select edges to fillet ( Radius=0.1 )** prompt, press ENTER.  
The edge has a round on it.



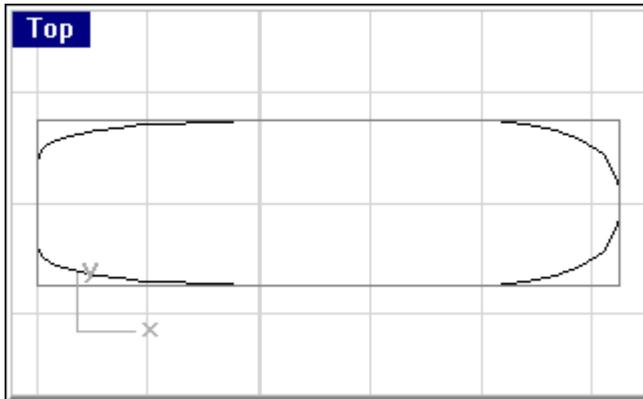
- Save** your model.

To create the Handle:

- Change to the **Curve** layer.
- Create one of the curves for the handle of the hammer. Make it start at the endpoint of the tang profile curve and end on the centerline.

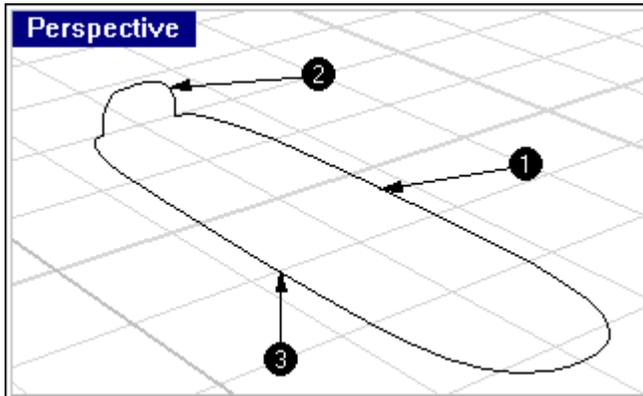


- 3 **Mirror** to create the other curve.



To create the surface for the handle:

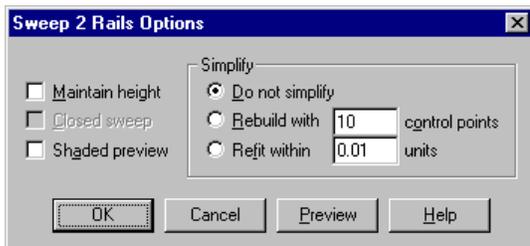
- 1 Change to the **Handle** layer.
- 2 From the **Surface** menu, click **Sweep 2 Rails**.



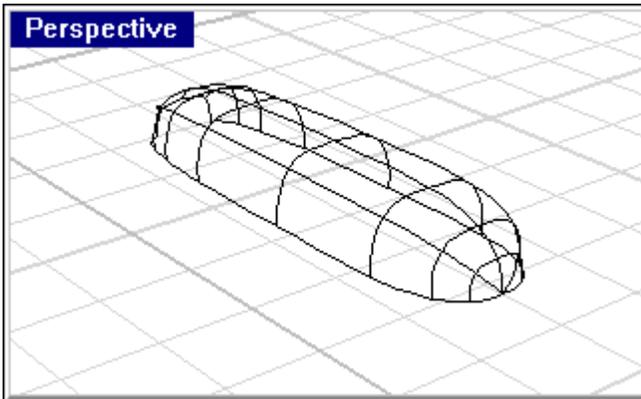
- 3 At the **Select 2 rail curves** prompt, select one of the rail curves.
- 4 At the **Select 2 rail curves** prompt, select the other rail curve.
- 5 At the **Select cross-section curves (Point)** prompt, select small the curve you drew for the tang.

This is the same curve you used to create the tang of the hammer.

- 6 At the **Select cross-section curves (Point)** prompt, press ENTER.
- 7 In the **Sweep 2 Rails Options** dialog, click **OK**.

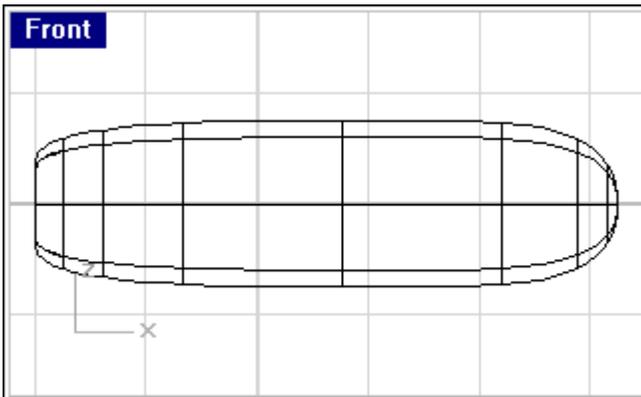


A surface is created.



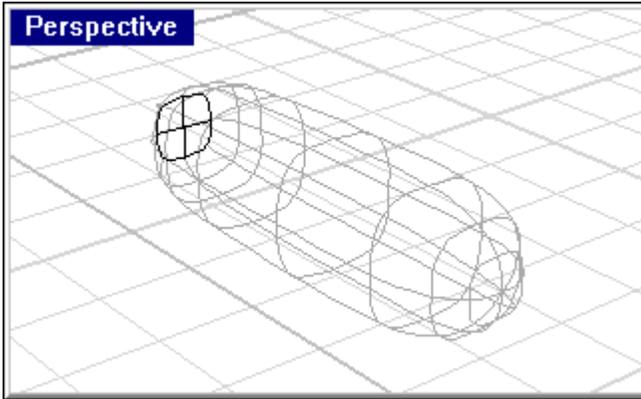
To finish the handle:

- 1 From the **Transform** menu, click **Mirror**.
- 2 At the **Select objects to mirror** prompt, choose the surface you created.
- 3 At the **Start of mirror plane** prompt, in the **Front** viewport snap to the one end of the handle edge.
- 4 At the **End of mirror plane** prompt, toggle ortho on and pick a point to the right in the **Front** viewport.



- 5 **Join** the two surfaces together.
- 6 From the **Solid** menu, click **Cap Planar Holes**.
- 7 At the **Select surfaces or polysurfaces to cap** prompt, choose the handle.

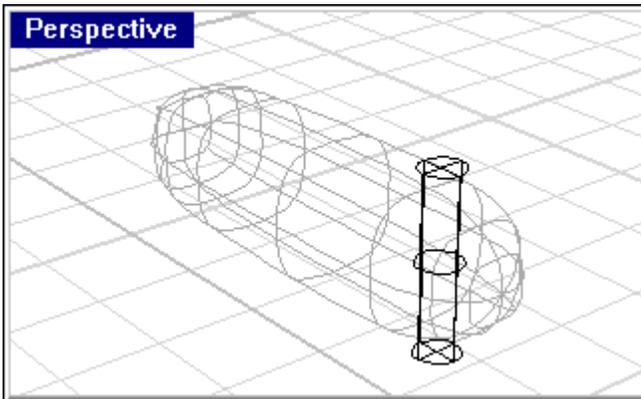
- 8 At the **Select surfaces or polysurfaces to cap** prompt, press ENTER.



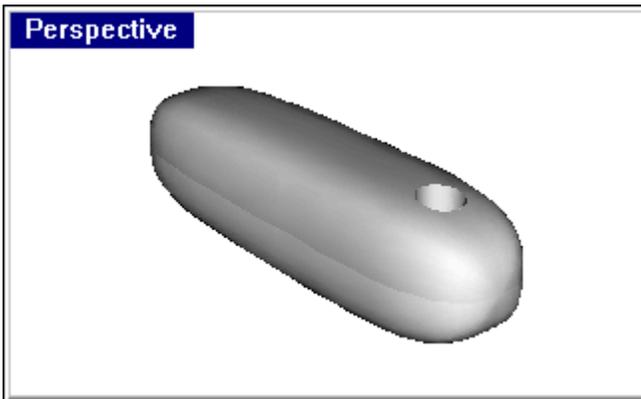
- 9 **Save** your model.

To create the hole for the handle:

- 1 Draw a **Circle** with the center .88 from the end of the handle. You may have to draw a construction line to help you position the circle.
- 2 From the **Surface** menu, click **Extrude**, then click **Straight**.
- 3 At the **Extrusion distance <2.00> (Direction Cap=yes Bothsides Tapered)** prompt, type **B** and press ENTER.
- 4 At the **Extrusion distance (Direction Cap=yes)** prompt, type **2** and press ENTER.



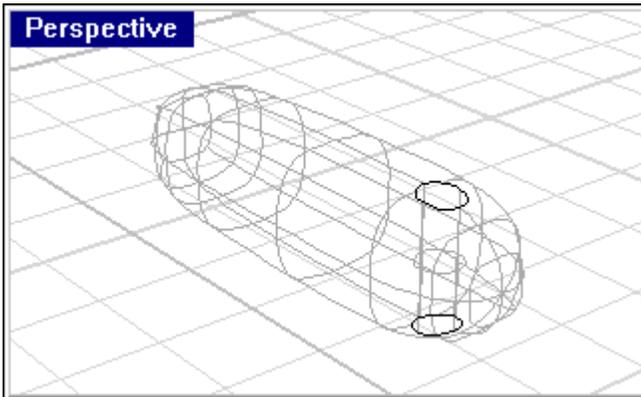
- 5 Use **Difference** to remove the cylinder from the handle.



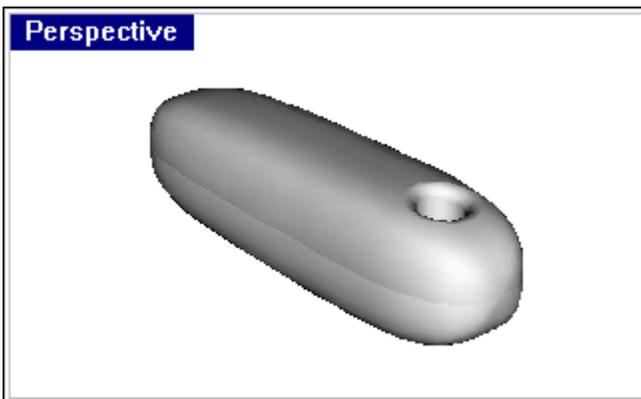
- 6 **Save** your model.

To create the fillets for the hole:

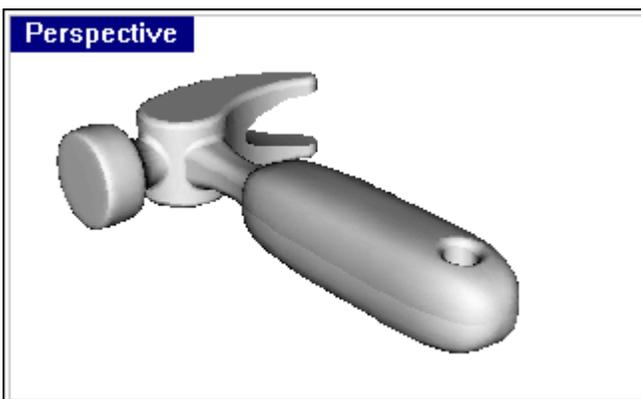
- 1 From the **Solid** menu, click **Fillet Edge**.
- 2 At the **Select edges to fillet ( Radius=0.1 )** prompt, type **.1** and press ENTER.
- 3 At the **Select edges to fillet ( Radius=0.1 )** prompt, select both edges of the hole.



- 4 At the **Select edges to fillet ( Radius=0.1 )** prompt, press ENTER.  
The edge has a round on them.



- 5 **Save** your model.
- 6 Click in the **Perspective** viewport.
- 7 From the **Render** menu, click **Shade**.



- 8 From the **Render** menu, click **Render**.

