Modeling Open Parts

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- create open parts
- set material side
- perform join operations
- perform cut operations
- create a solid from an open part

Before you begin...

Prerequisite tutorials:

1. Getting Started (I-DEAS™ Multimedia Training)
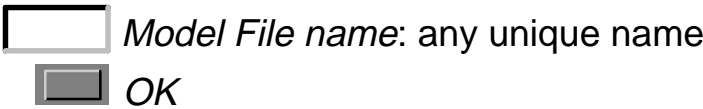
–or–

Quick Tips to Using I-DEAS
–and–
Creating Parts

2. Sketching and Constraining

3. Using Sections and Sketch Planes

4. Extruding and Revolving Features
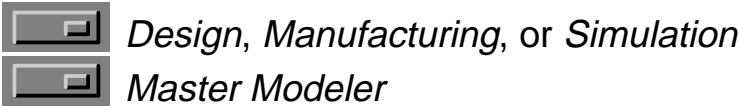
5. Adding Fillet, Shell, and Draft Features

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

File
  Open

Open Model File form

    Model File name: any unique name

    OK

Make sure you're in the following application and task:

    Design, *Manufacturing*, or *Simulation*

    Master Modeler

Since this tutorial creates free-form parts without dimensions, start with mm units and a workplane set to display as –100 x 100 mm square.

Options
  Units

    mm (milli newton)
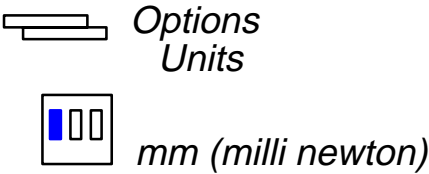
Workplane Attributes form

☑ *Display Border*

☐ *X Min. –100*

☐ *Y Min. –100*

☐ *X Max. 100*

☐ *Y Max. 100*

▭ *OK*

Save your model file.

⬒ *File*
   Save

Warning!

If you are prompted by I-DEAS to save your model file, respond:

▭ *No*

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

To reopen your model file to the previous save, press Control-z.

Open parts are parts that do not completely enclose a volume.

You can create open parts by extruding, revolving, lofting, or sweeping open sections, or by deleting faces from a solid part.

In this section you will use the *Revolve* icon to create an open part that is used throughout the remainder of the tutorial.
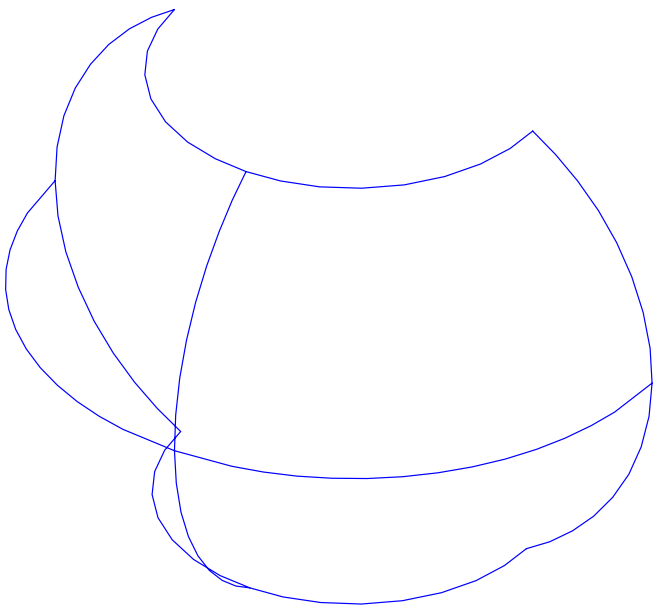
Sketch a line and an arc similar to those shown.
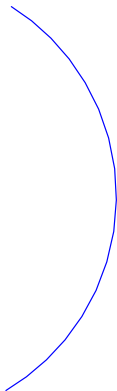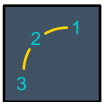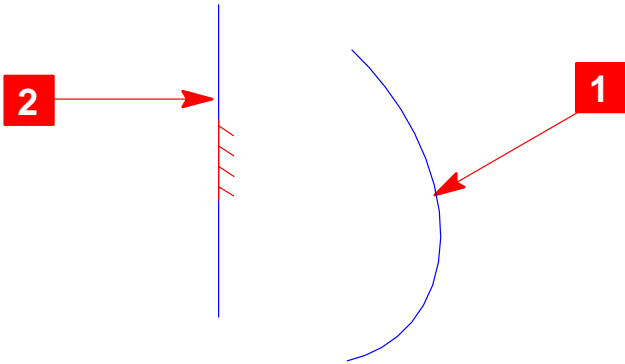

Change to isometric view.
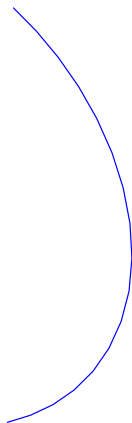
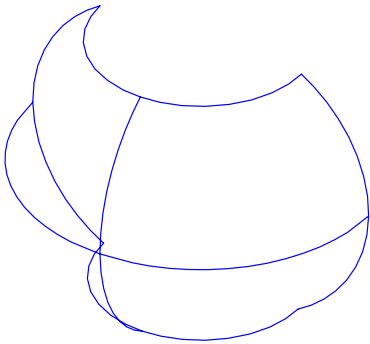Revolve the arc 180 degrees about the line.

1 pick anywhere on arc

 (Done)

2 pick line

Revolve Section form

Angle: 180

OK

Calculate the properties for this part.

 pick anywhere on part

Physical Properties form

 Calculate

Things to notice
Notice that the part has *Open Surface Area,* but zero *Solid Surface Area* and *Volume.*

 OK

Recovery Point
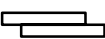
 File
         Save

To perform construction operations on open parts, it is important to understand the concept of "material side."

When you perform a cut or join on an open part with another part, the operation will behave just like the operation would between surfaces of any two solid parts.

For example, picture two parts (A and B) that overlap.
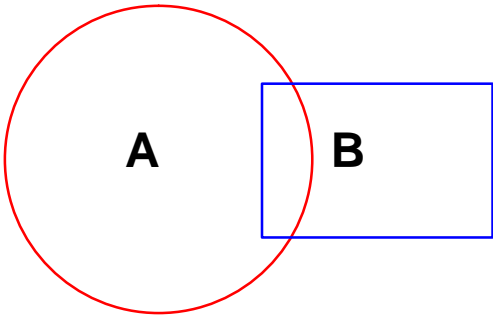
The material side of each of these parts is toward the center, since they are solid. If part B is cut from part A, the result would be:
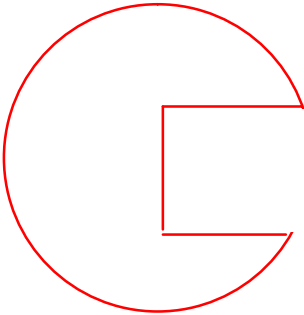
However, if part A is an open part, you would get different results depending on the direction of the material side of part A.

Picture parts A and B looking like this:
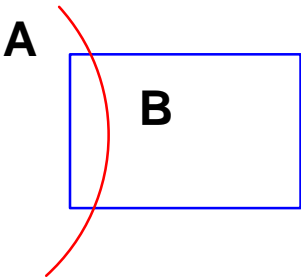
Material side can be on either side or none. In the parts shown above, the results of cutting B from A would be as follows, depending on the material side. The arrow shows the material side of surface A.
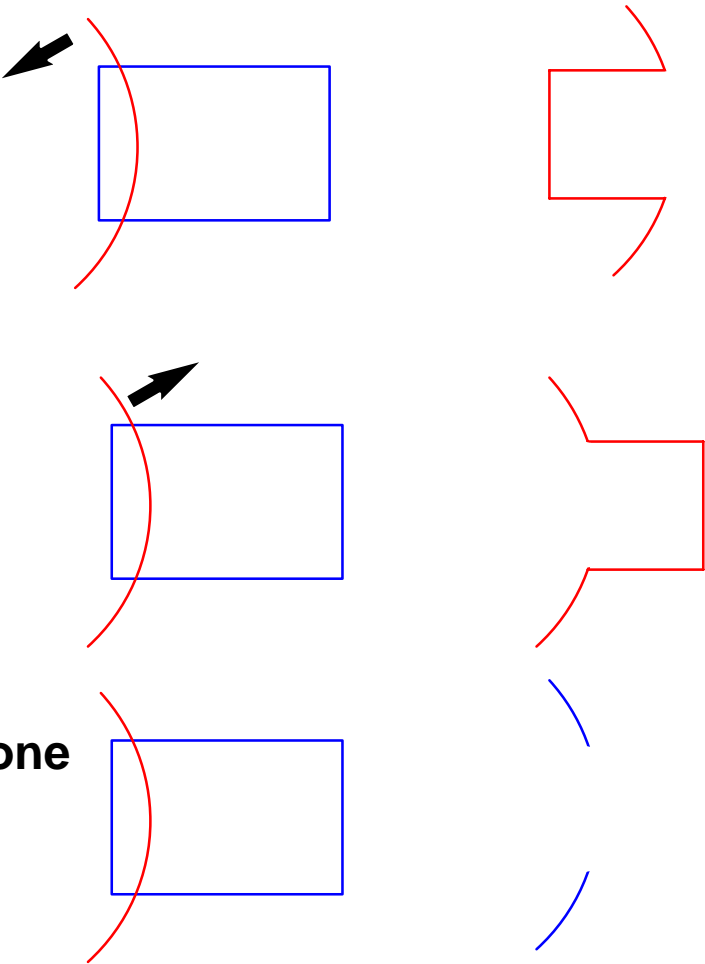
Set the material side of the part's surface to point toward the center.
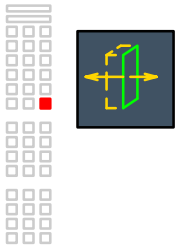
 pick surface of part

 (Accept)

 One

 No (this keeps arrow from flipping)

 (to deactivate icon)
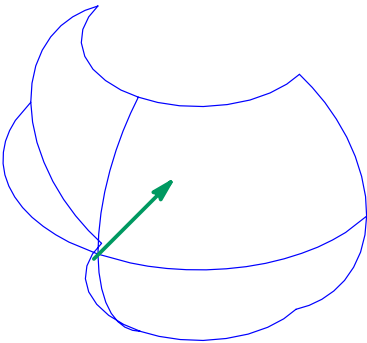
Switch to front view and sketch a circle on the same workplane.
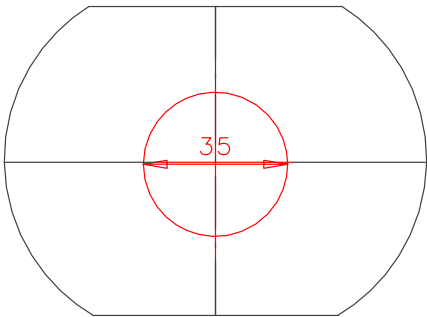
Switch back to isometric view and extrude the circle as a new part.

💡 Make sure the distance of the extrude is large enough so the surfaces cross. You may need to use dynamic viewing and preview the arrow before accepting the form.
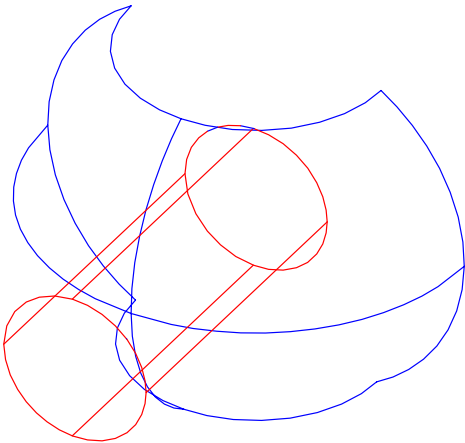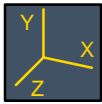
Check to make sure you have two parts on the workbench .

 Workbench

 Check *I-DEAS List*.

Recovery Point
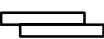
 File
    Save

 You'll be reopening your model file many times to this point to try construction operations with the material side set different ways. Be sure to save your model file only when the tutorial tells you to.
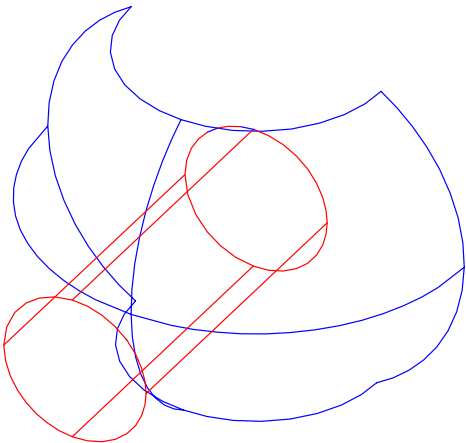
Join the cylinder to the open part, making sure that construct relations are turned off.

check the *Relations* switch in the menu

Select only if the menu says *Turn Relations OFF*. If the menu has *Turn Relations ON*, move the mouse pointer off the menu without selecting anything.
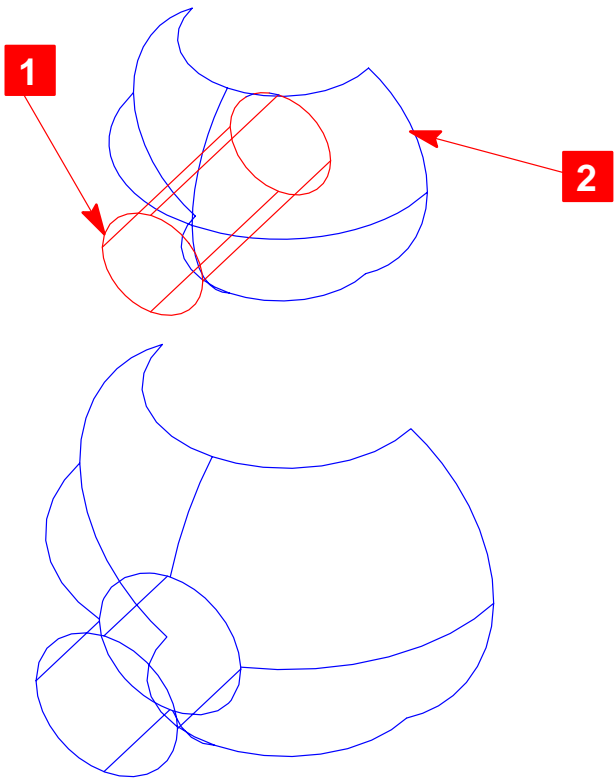
Try different displays and views to see the result. Rotate the part to see the inside.
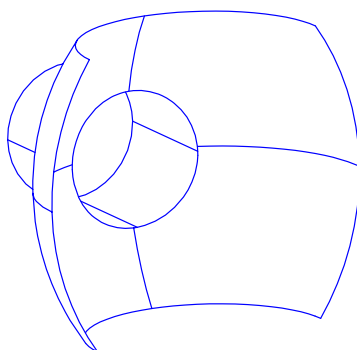
Hint

 Don't save your model file at this point.

Repeat the join operation, but this time with the material side the other way.

First, reopen your model file to the last save (Control-z).
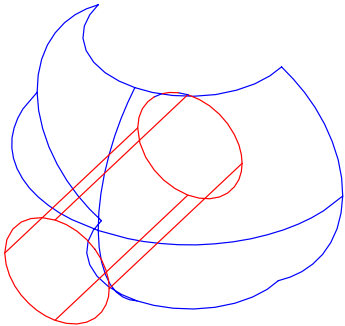
 pick surface of open part

 (Accept)

 One

 Yes (ok to flip material side)

 No (this keeps arrow from flipping)
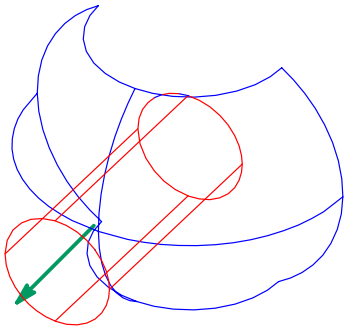
 (to deactivate icon)

Now, try the join again.
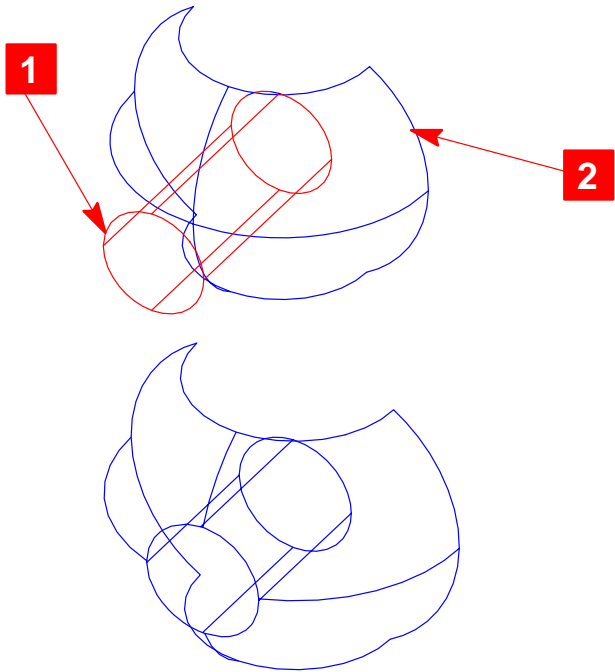
Try different views and displays to see the inside of the cylinder.
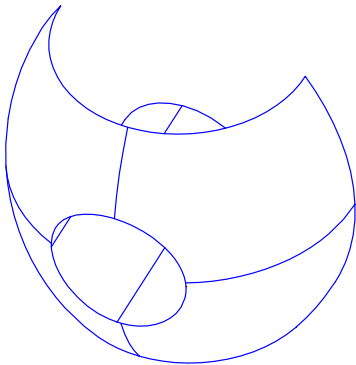
Don't save your model file at this point.

Try the join operation again, but this time with the material side set to none.

First, reopen your model file to the last save.
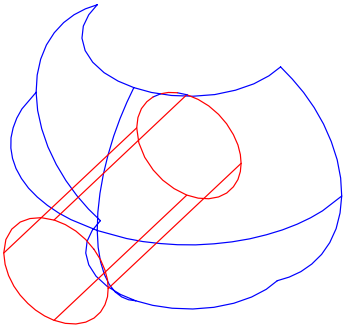
 pick surface of open part

 (Accept)

 None
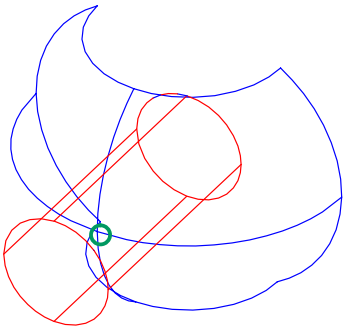
 (to deactivate icon)

Perform the join.
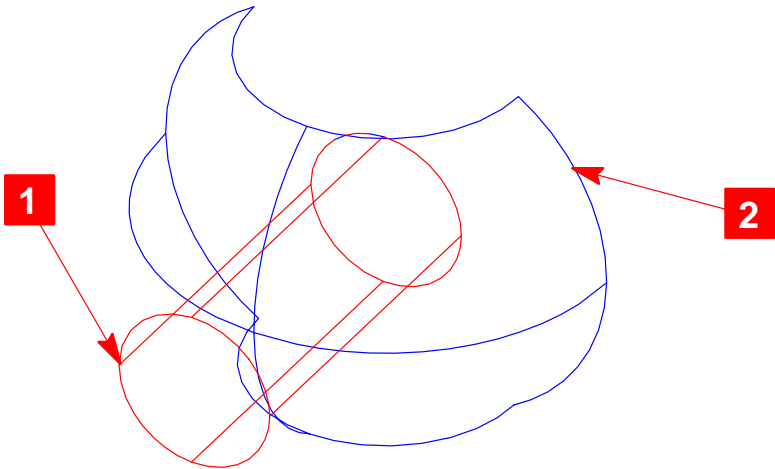
Try different views and displays of the result.
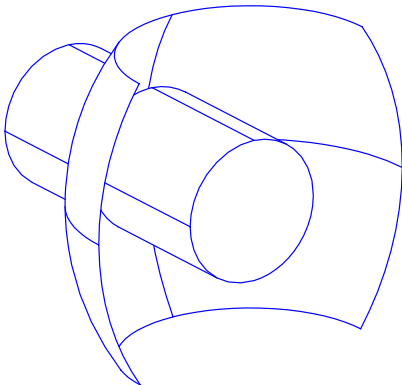
Don't save your model file at this point.

Using the parts on your workbench, perform a cut operation with material side both ways, and also set to none.

Use the cylinder as the cutter part in all cases.

First, reopen your model file to the last save.
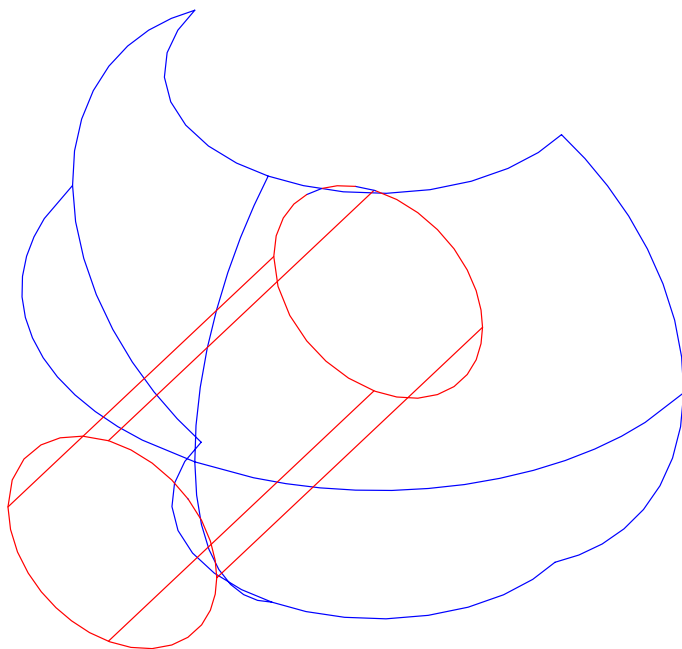
Set the material side of the part's surface to point toward the center.
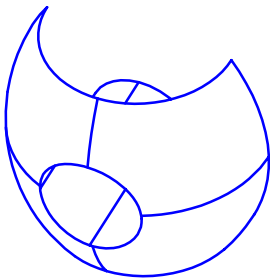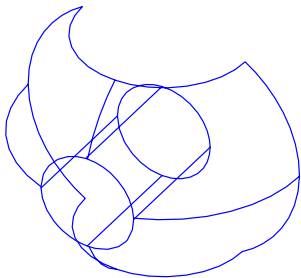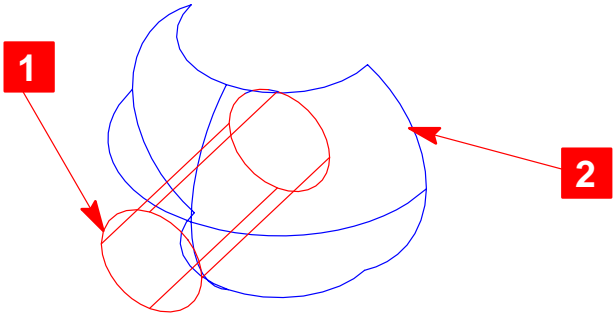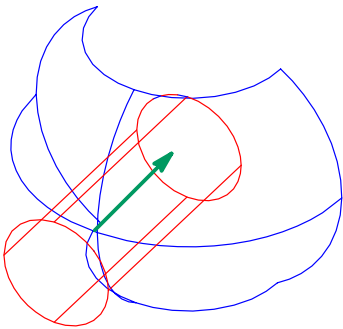
Don't save your model file at this point.

Reopen your model file to the last save.

Set the material side of the part's surface to "none."
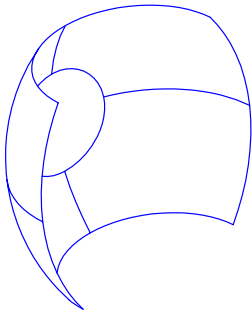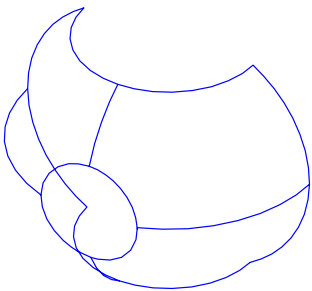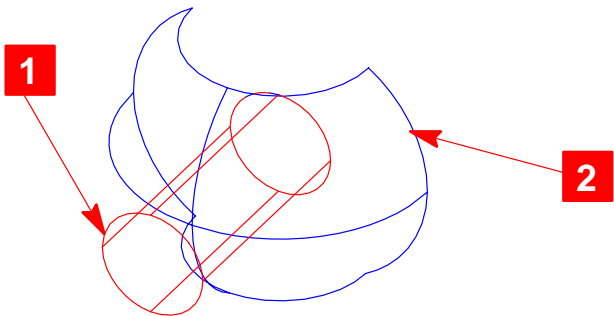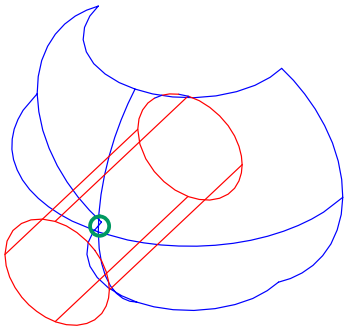
Don't save your model file at this point.

Reopen your model file to the last save.

Set the material side of the part's surface to point outward.
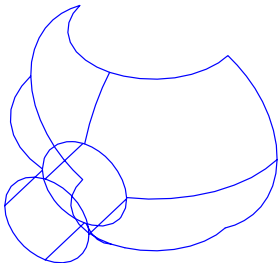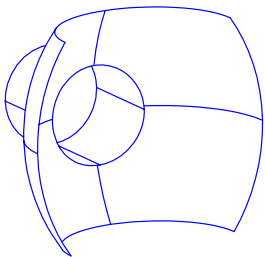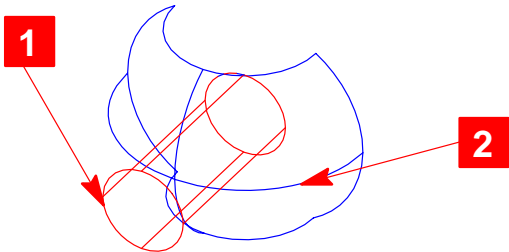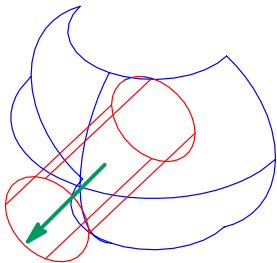
Recovery Point

File
   Save

After creating open parts and adding features, you will often want to make them into a solid. You can create a solid by stitching surfaces together (covered in the tutorial "Performing Surfacing Operations"), or by shelling to create a thin-walled solid (covered in the tutorial "Adding Fillet, Shell, and Draft Features").

In this section, you will shell the part to create a solid.
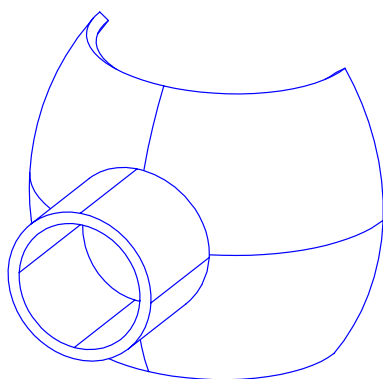
To create a solid, shell the part that you just cut .
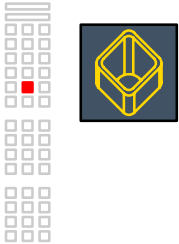
1 pick anywhere on part

[□■□] (Done)
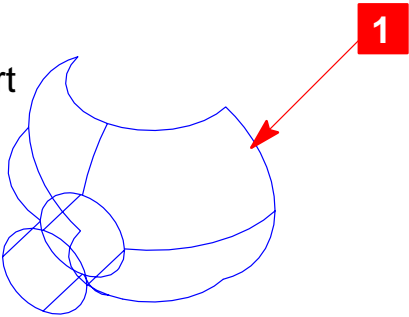
Shell form

Thickness: 2

pick _Delete Surface_ icon on form

2

[□■□]
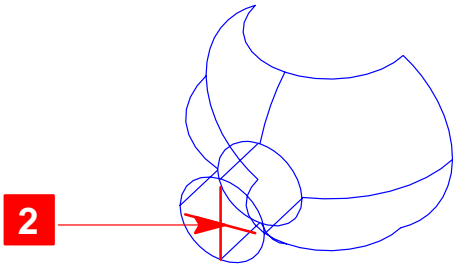
Shell form

[___] _OK_
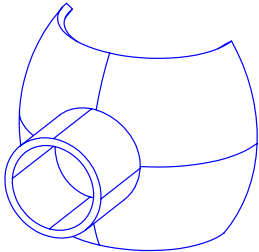
To practice the skills taught in this and preceding tutorials, try to create the stove burner drip pan shown below.

Top View
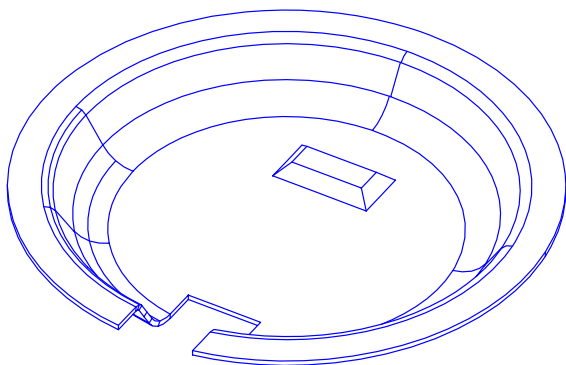
Bottom View
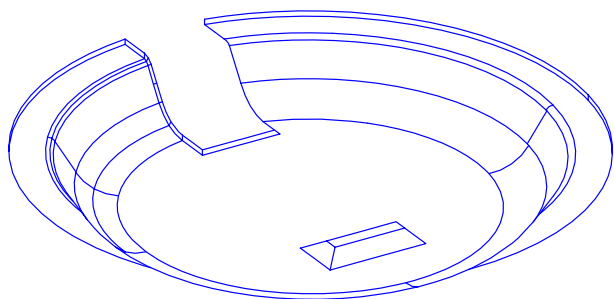
Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the part shown.

If you would rather quit this tutorial now and try the "On your own" later, skip to the last page for wrap-up instructions.

Step 1. Sketch the shape shown, revolve accepting all defaults, and switch to the isometric view.
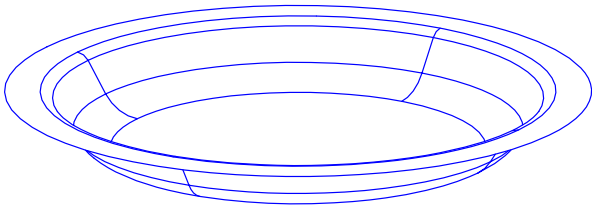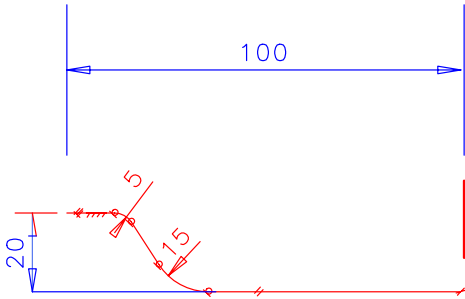
Step 2. Set the material side to none. Select sketch on face and extrude a rectangle (thicken) through the burner plate edge.
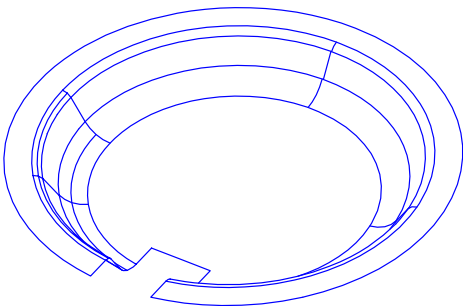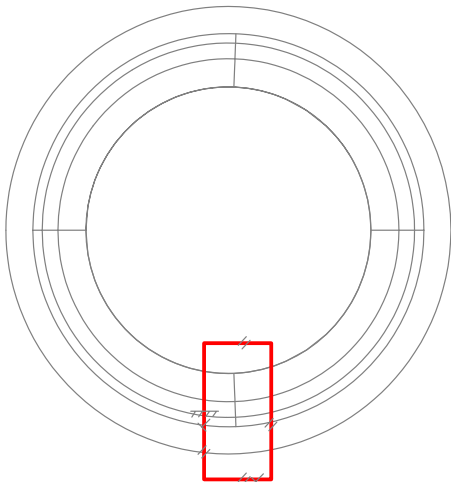
Step 3. Set material side to one (Down). Sketch a rectangle on the bottom of the burner plate. Extrude with a depth of 5 and a draft angle of –45.
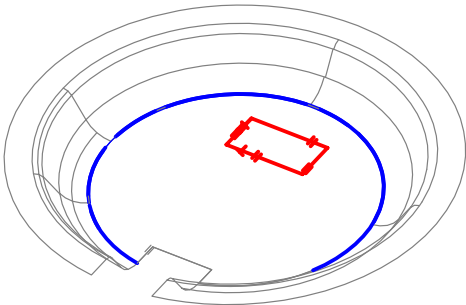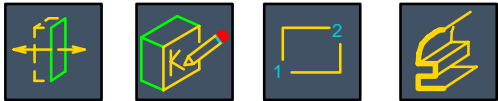
Top View
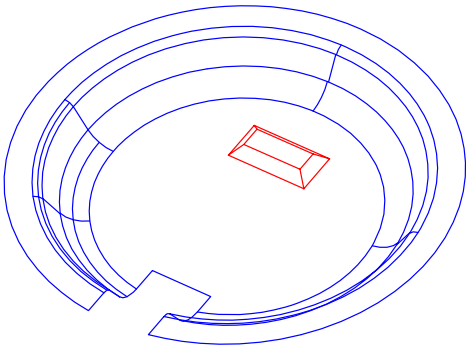
Bottom View
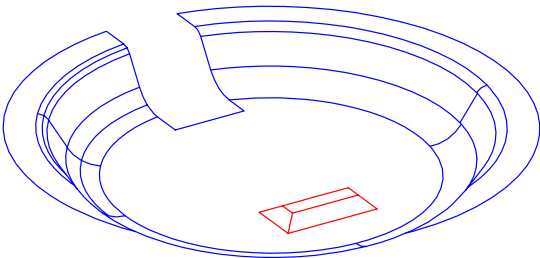
Step 4. Shell the burner plate to a thickness of 2.
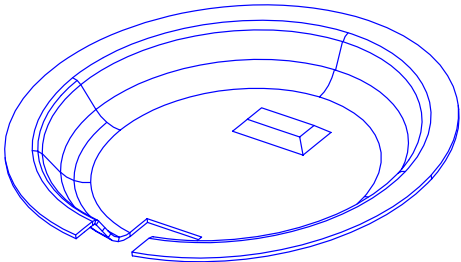
Tutorial Wrap-up

You have completed the Modeling Open Parts tutorial.

Delete or put away the parts. These parts are not used in any other tutorials.

See also...

For additional information on many of the concepts covered in this tutorial, see the following in the I-DEAS *Help* facility:

   ⬚ *Help, Manuals, Table of Contents*

Design User's Guide
   Design Concepts
      Modeling Parts
      Surface Creation

What's next?

After exiting, choose the Fundamental Skills tutorial that is next in the learning path you are following.

To exit this tutorial, select:

   ⬚ *File*
      Exit

Warning!

Do not use the menu in the *I-DEAS Icons* window to exit. Use the menu in the Acrobat Reader window.

I-DEAS Master Series™ Online Tutorials

Structural Dynamics Research Corporation
2000 Eastman Drive
Milford, Ohio 45150
(513) 576−2400