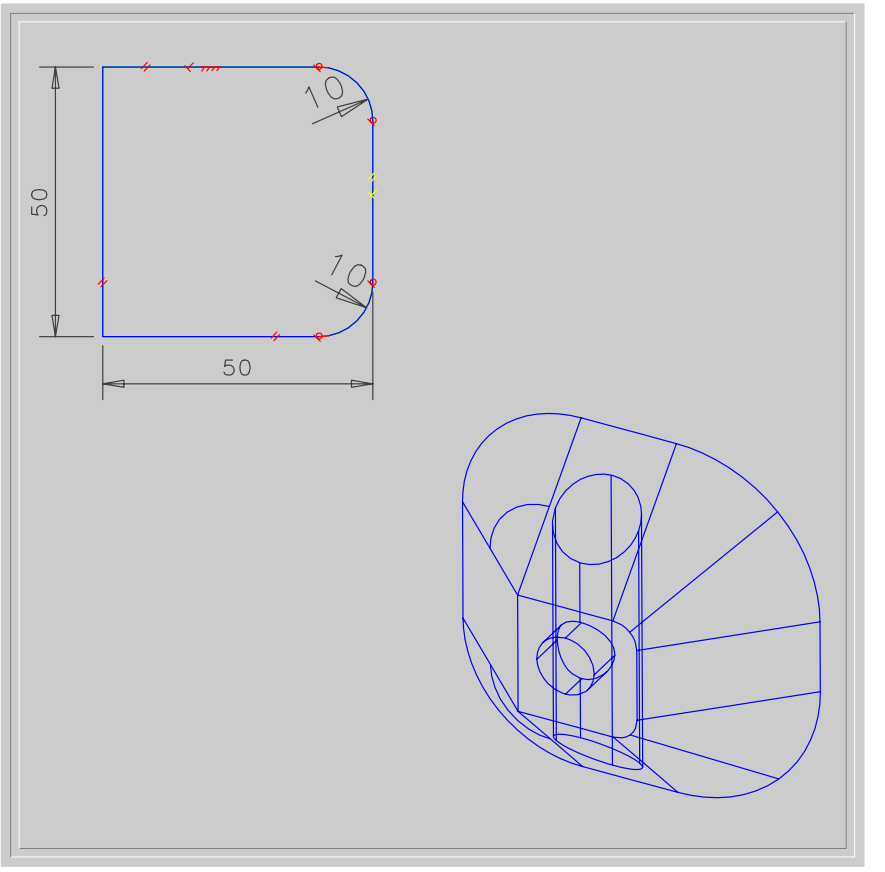


Extruding and Revolving Features

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- use extrude options
- use revolve options

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

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:

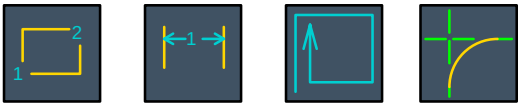
Design, *Manufacturing*, or *Simulation*
Master Modeler

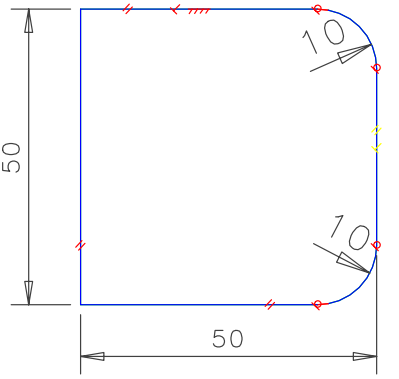
Set your units to mm.

Options
Units

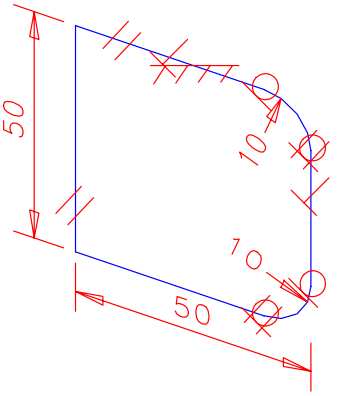
mm (*milli newton*)

Sketch the shape to the dimensions shown and fillet the two corners.





Switch to isometric view.



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.

You use *Extrude* to create solids that manifest your part's basic shape. Extruding closed wireframe produces a solid. *Extrude* includes the following capabilities:

Operations

- *Protrude*
- *Cutout*
- *Intersect*
- *New Part*

Depth

- *Distance* (enter value)
- *Thru All*
- *Until Next*

Draft Angle

Options

- *Vector*
  enter vector coefficients
  pick along vector
- *Corner radii*
  Constant radii
  Varying radii
  Varying radii and corner radius

Try some of the options that are available.

 pick anywhere on sketch

 (Done)

Extrude Section form

 *Distance*: 50

 *Draft Angle* (toggle on)

 *Draft Angle*: 45

 *New Part*

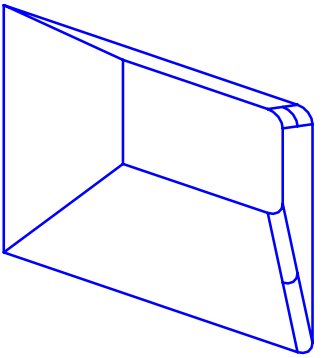
 *Options...*

Extrude Options form

 *Constant radii*

 *OK*

 *OK*



Things to notice

With a positive draft angle, the part gets larger in the direction of the extrusion. Using *Constant radii*, corner fillets are constant, but will not have the requested draft angle.

You can also extrude with varying radii. First, open your model file to the last save.

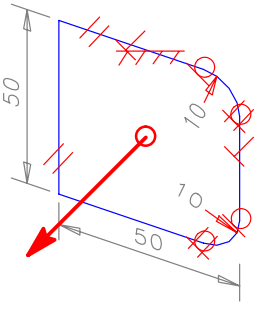
Hint

Deselect All

Hold down the Control key and press the letter z.

pick anywhere on sketch

(to indicate Done)



Extrude Section form

Distance: 50

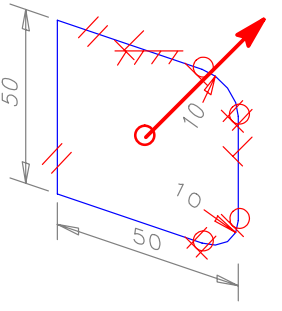
Draft Angle (toggle on)

Draft Angle: 45

flip direction

New Part

Options...

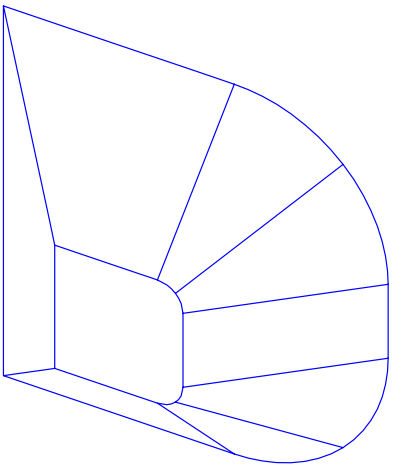


Extrude Options form

Varying radii (normally the default)

OK

OK

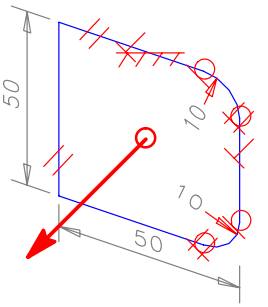


Repeat the process, but this time with varying radii and round corners. This is the only option that gives the exact requested draft angle on every surface.

Open your model file to the last save (Control–z).

 pick anywhere on sketch

 (Done)



Extrude Section form

 *Distance*: 50

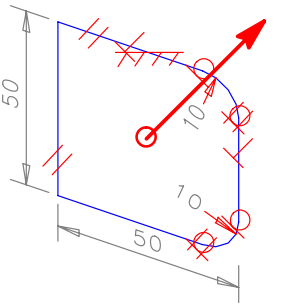
 *Draft Angle*

 *Draft Angle*: 45

 flip direction

 *New Part*

 *Options...*

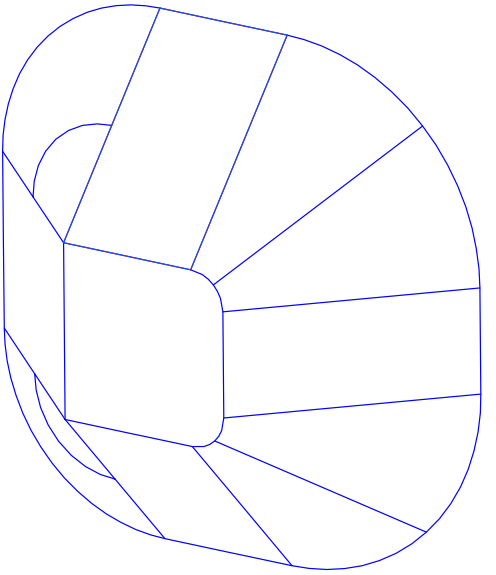


Extrude Options form

Varying radii, round corners

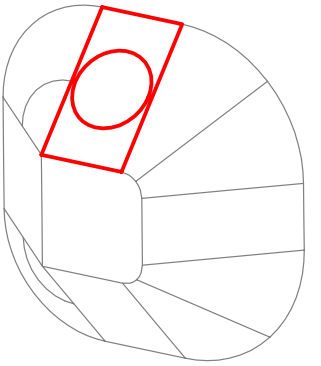
OK

OK



Another extrude option is *Extrude Along Vector*.

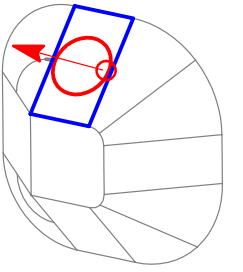
First, sketch a circle on the face shown.



Then, extrude the circle.

 pick anywhere on circle

 (Done)



Extrude Section form

Cutout

Depth:
Thru All

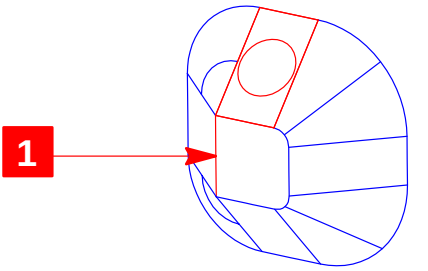
Options...

Extrude Options form

Along Vector

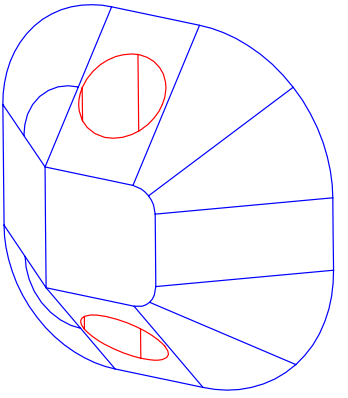
Pick Vector

1



OK

OK



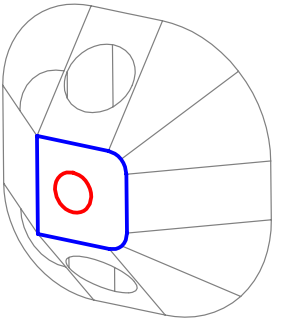
Recovery Point

File
Save

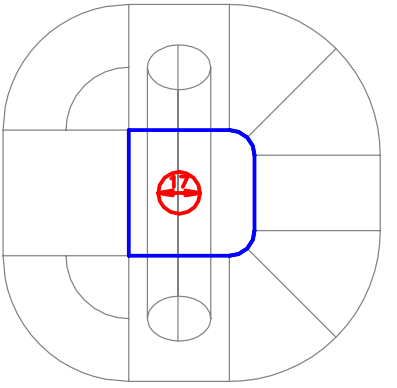
Another extrude option is *Extrude Until Next*

If a cutout is made to a specific distance and the part's dimensions are modified, the cutout will need to be modified also. However, *Extrude Until Next* will automatically change the cutout's distance.

To see how this works, sketch a circle on the face shown. Try to make the new circle smaller than the first and in the center of the face.



It might be easier if you switch to front view.



Next, cut out the circle to the first hole.

 pick anywhere on circle

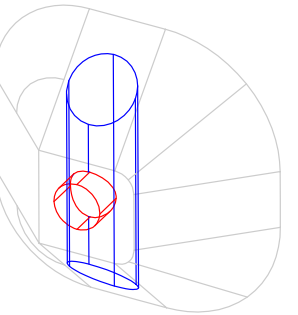
 (Done)

Extrude Section form

 *Cutout*

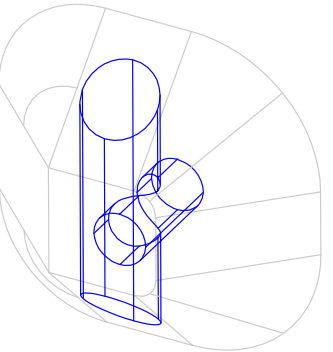
 *Depth:*
Until Next

 *OK*



Things to notice

If the cutout misses the first hole, the cutout will continue through until the bottom surface of the part, as shown below.



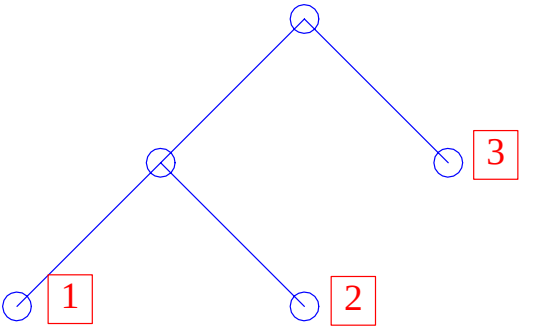
If this is the case, and you want to try again, open your model file to the last save. Resketch the circle and try the extrude again.

Display the history of this part. If modeled as directed, it should have the three leaves shown below.

 pick anywhere on part

 (Accept)

History Tree form



---

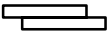
1. First extrusion with draft
2. Hole cut out through part
3. Hole cut out until next

Things to notice

Each wireframe sketch became a new leaf in the history tree.

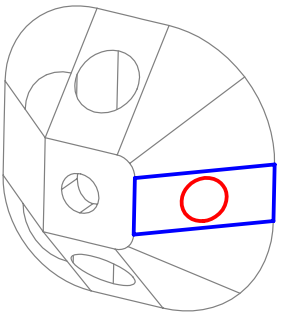
 *Dismiss*

Recovery Point

 *File*
Save

Another use of the *Extrude* command is the *Protrude* option.

First, sketch a circle on the face shown.



Protrude the circle a length of 25mm.

 pick anywhere on circle

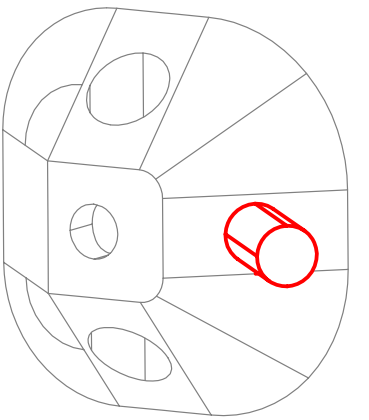
 (Done)

Extrude Section form

 *Distance*: 25

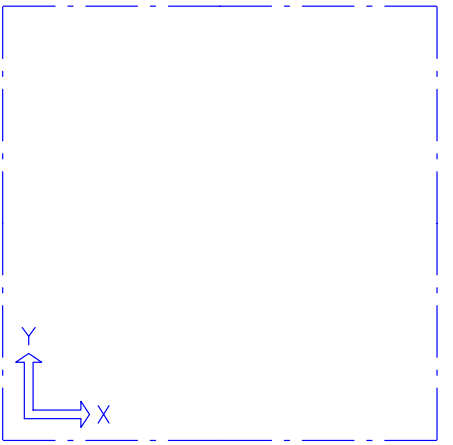
 *Protrude*

 *OK*

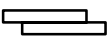


To get ready for the next section of the tutorial, delete or put away the part. The part is not used in any other tutorials.

Switch to front view.



Recovery Point



File
Save

Revolving closed wireframe also produces a solid.

The *Revolve* icon (similar to *Extrude*, except you need to pick a line to revolve about) includes the following capabilities:

Operations

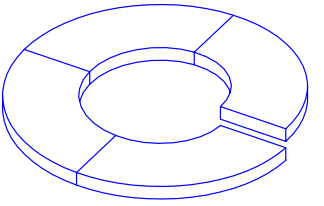
- *Protrude*
- *Cutout*
- *Intersect*
- *New Part*

Angle

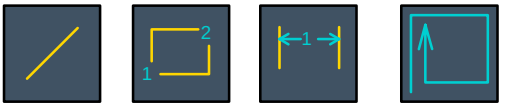
Options

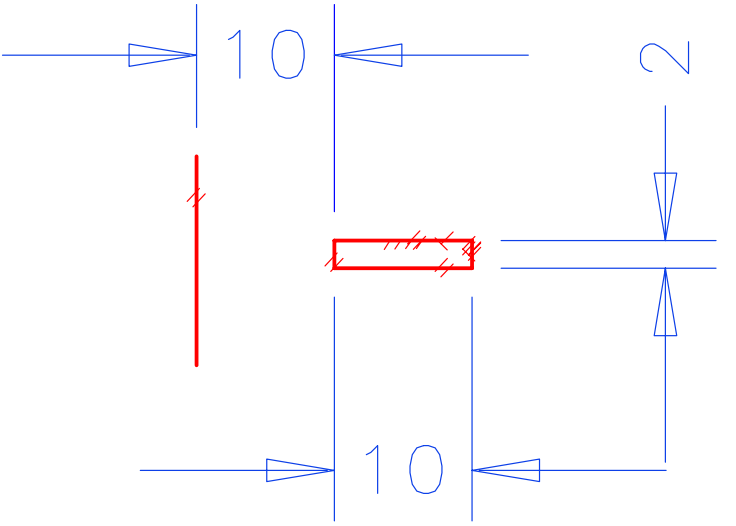
- *Translation along Axis*
- *Change in Radius*

In this section, you'll create a split washer, like the one shown below, by revolving a rectangle 360 degrees about a center line.



First, sketch a rectangle and a vertical line with the following dimensions.





Use dynamic viewing if the rectangle is too small on your screen (F1 and F2).

Recovery Point

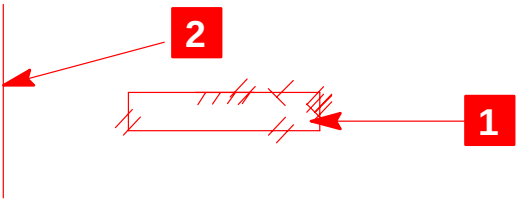
File
Save

Next, do the revolve.

1 pick anywhere on rectangle

 (Done)

2 pick axis



Revolve Section form

Angle: 360

Options...

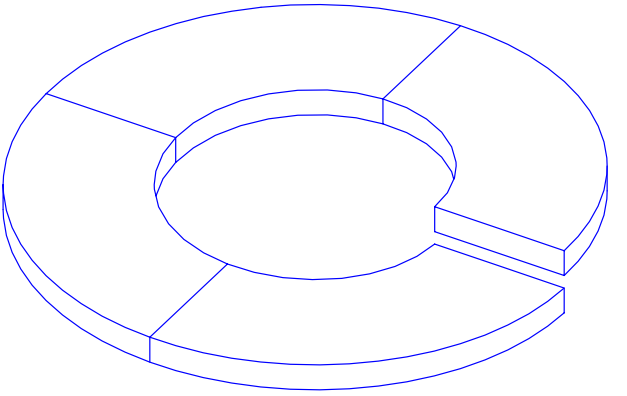
Revolve Options form

Translation along Axis

Translation along Axis: –3

OK

OK



Delete or put away the part. It is not used in any other tutorials.

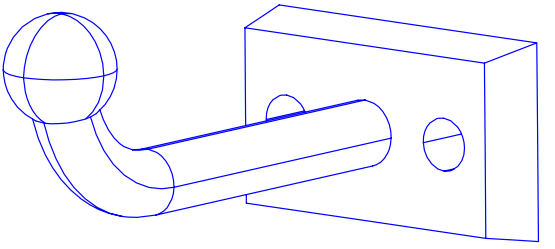
Before quitting the tutorial, try the two “On your own” exercises on the next few pages. If you would rather try the “On your own” at a later time, skip to the last page for wrap-up instructions.

There are two “On your own” projects:

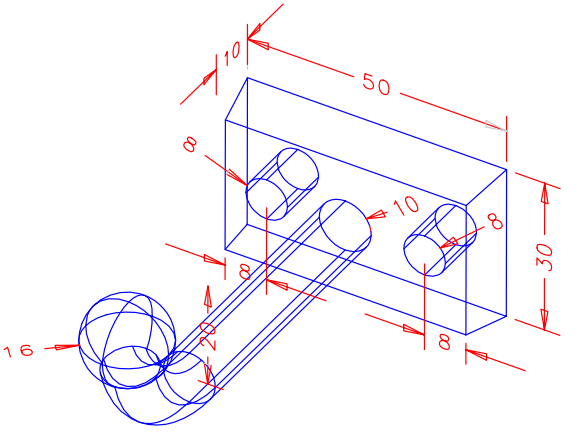
1. Create a coat hook
2. Create a spring

1. Create a coat hook

Create a coat hook using the *Extrude* and *Revolve* icons learned in this tutorial. Make sure each feature is protruded to or cut from the base feature.

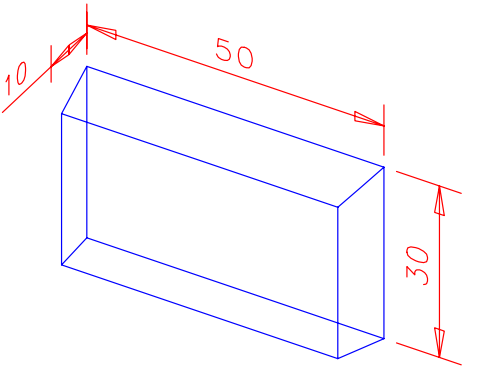


Use the following dimensions:

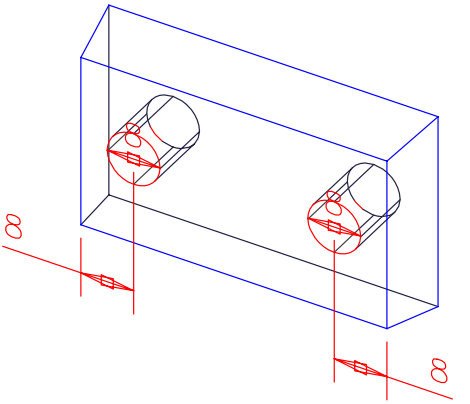


Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the part shown.

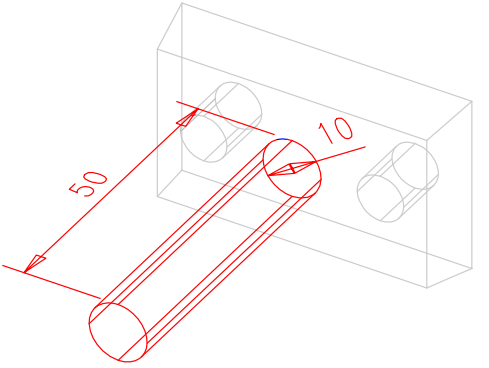
- Create the base with the following dimensions and a 10-degree draft angle:



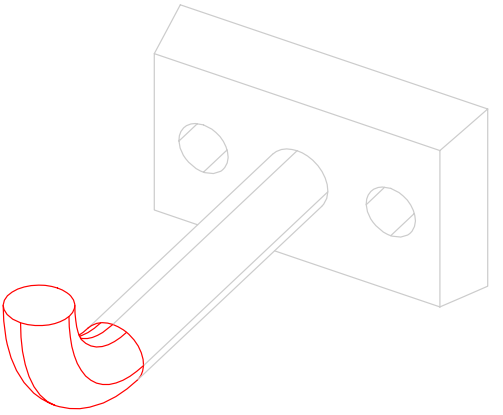
- Cut out the mounting holes.



- Extrude (protrude) the straight section of the hook.

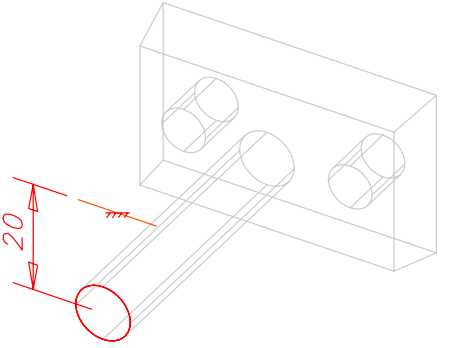


- Revolve the curved section of the hook –90 degrees.

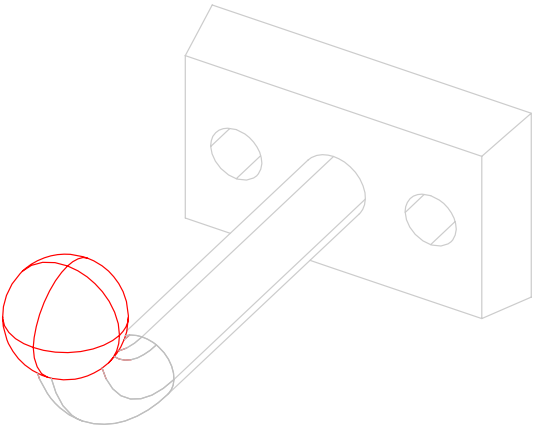


Hint

Use *Sketch in Place* and sketch a horizontal line. Do not sketch the circle. Revolve the existing face instead.

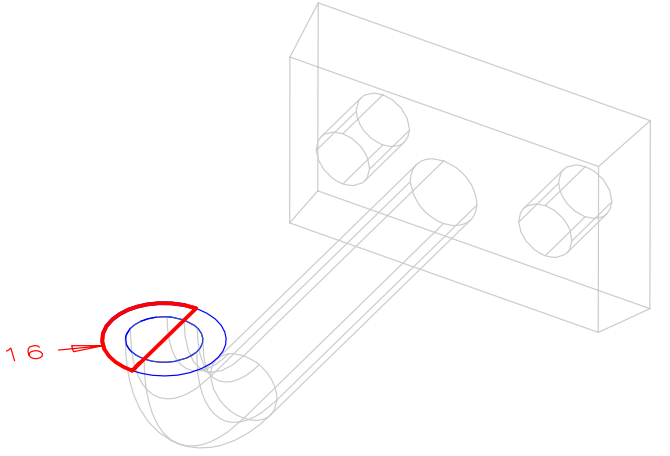


- Revolve a half circle to create a ball tip.

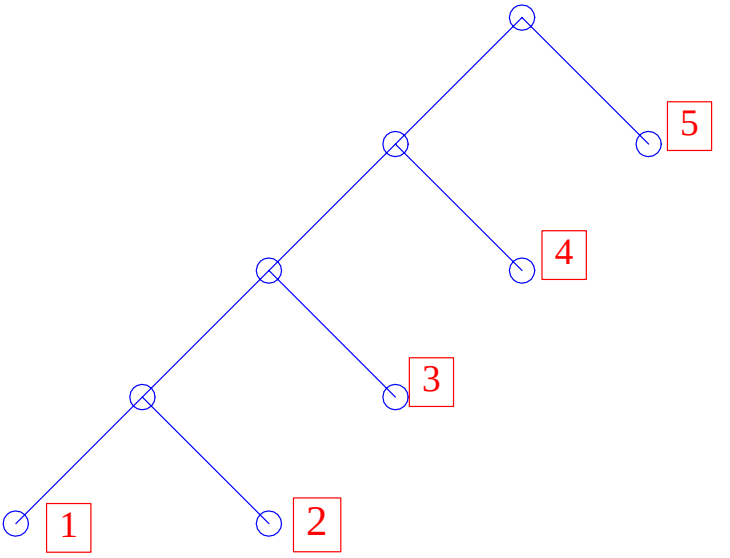


Hint

Sketch the complete circle, but use *Stop at Intersections* to pick a section on half the circle.



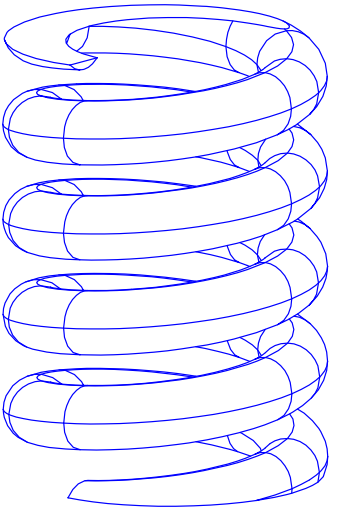
- The final part should have a history tree something like the following.



1. base
2. mounting holes
3. straight section
4. revolved section
5. ball tip

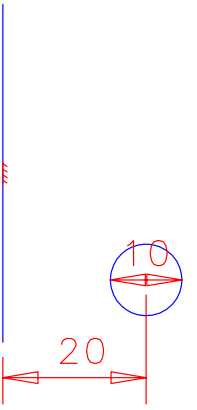
2. Create a spring

Create a 5-turn spring similar to the one shown.

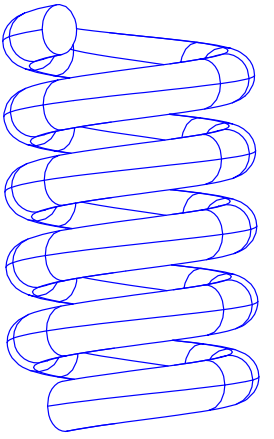


Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the part shown.

- Create a circle and a vertical line with the following dimensions:



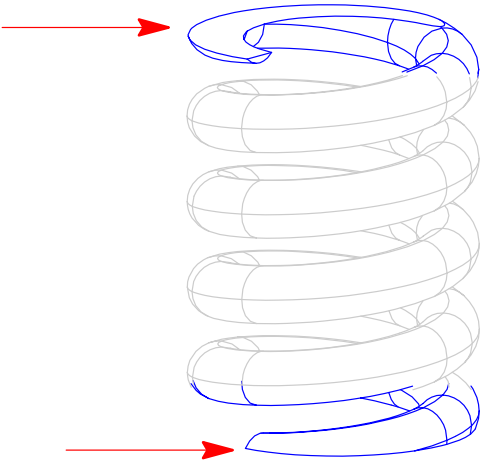
- Revolve the circle 5 times around the vertical line axis.



Hint

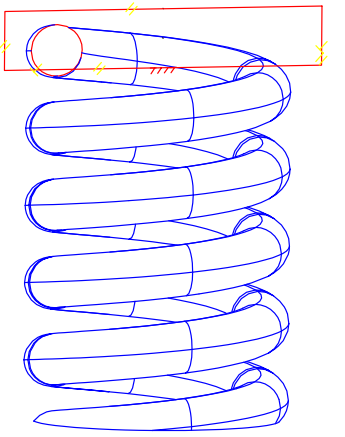
You can enter the angle as 360*5. Enter the total translation distance (distance/turn)*5. For example, to get a distance of 15 for each turn, enter 75.

- Trim the ends flat.



Hint

Sketch a rectangle on the spring end. Then use *Extrude*, *Cutout*, with the *Depth Thicken* option.



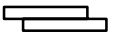
Tutorial wrap-up

You have completed the Extruding and Revolving Features tutorial.

Delete or put away any parts. They 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

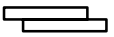
Design User's Guide
- Designer's Notebook
- Design Concepts
  - Modeling Parts
- Design Techniques and Examples

Design Reference Guide
- Master Modeler
  - Wireframe Extrusion and Revolution
    - Extruding
    - Revolving

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
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Warning!
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