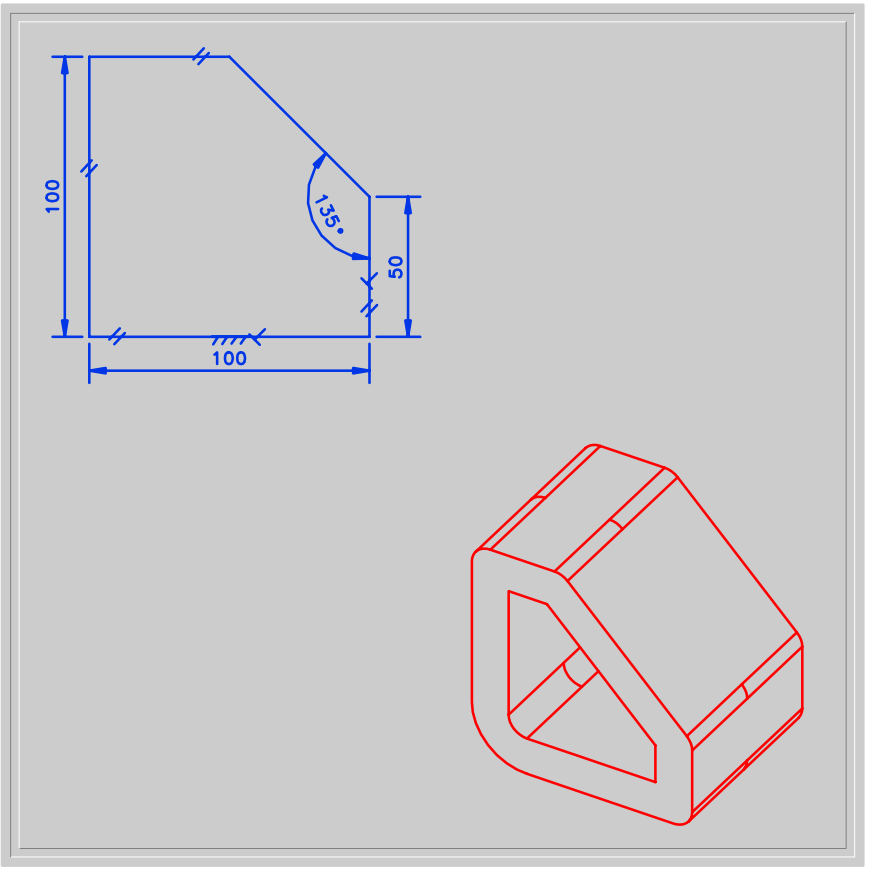


Using Sections and Sketch Planes

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- work with sections
- find free degrees of freedom
- select sketch planes

Before you begin...

Prerequisite tutorials:

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

   –or–

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

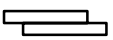
2. Sketching and Constraining

To begin this tutorial, make sure you're in the following application and task:

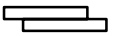

Design, *Manufacturing*, or *Simulation*


Master Modeler

If you saved the model file created in the tutorial "Sketching and Constraining," open it now and use the sketch that was created.

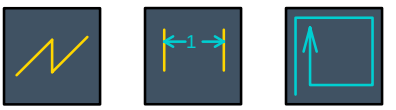

File
Open

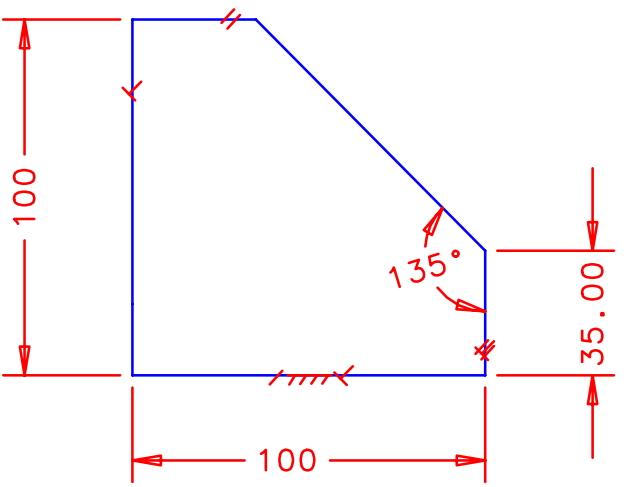
Set your units to mm.


Options
Units

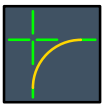

mm (*milli newton*)

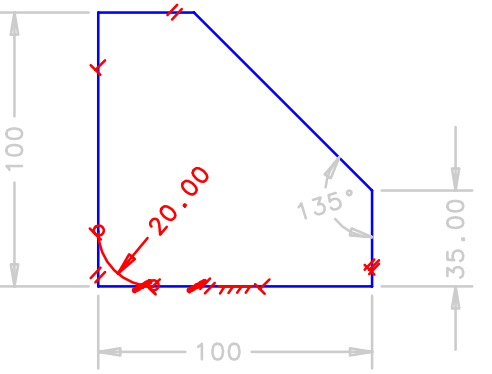
If, for some reason, you did not save the model file, or the sketch is no longer on the workbench, create the following sketch before continuing.





Add a fillet in the lower corner. Keep the untrimmed curves.



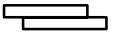


Hint

Radius: 20

Trim/Extend (toggle off)

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.

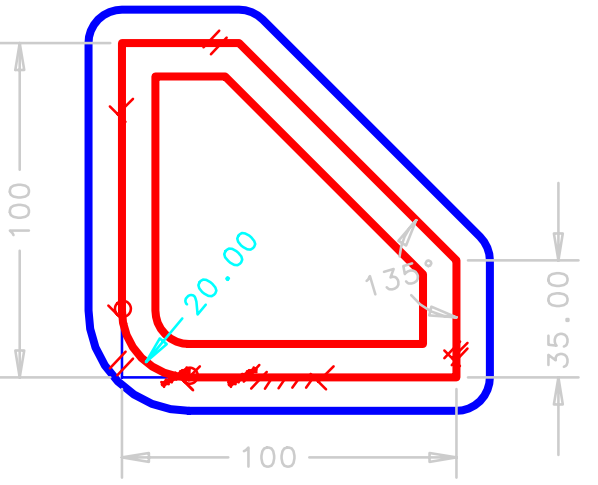
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

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

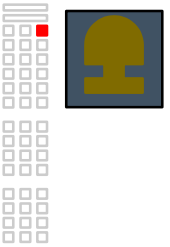
The *Extrude* and *Revolve* commands automatically create a section which chains around the geometry to be extruded or revolved. Sometimes it gives you more control to create the section first.

Sections can also be used as a way to group wireframe geometry for other operations, such as *Offset* or *Move*.



When creating sections, an important option on the Section Options form is *Stop at intersections*. This option is off by default. When you toggle this option on, the software prompts you to pick each segment of a section where there are intersecting lines, and the section could continue in more than one direction.

Build a section around the shape using the round corner.



Section Options...

Section Options form

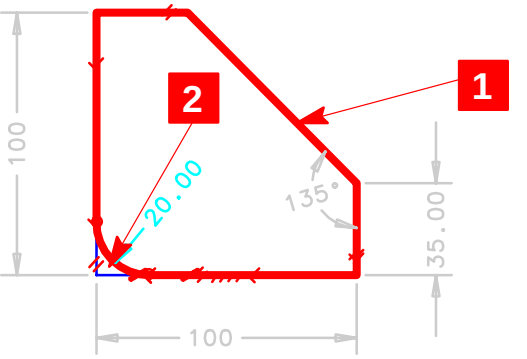
Stop at intersections

OK

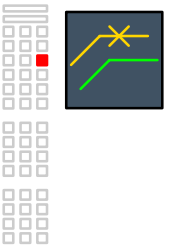
1

2

(Done)



Offset the section 10mm in both directions, with rounded corners.



1 pick anywhere on the section

(Accept)

Offset form

Distance: 10

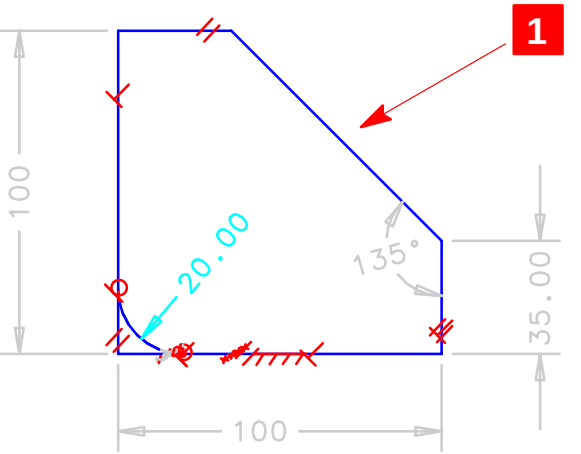
Round Corners

Associativity

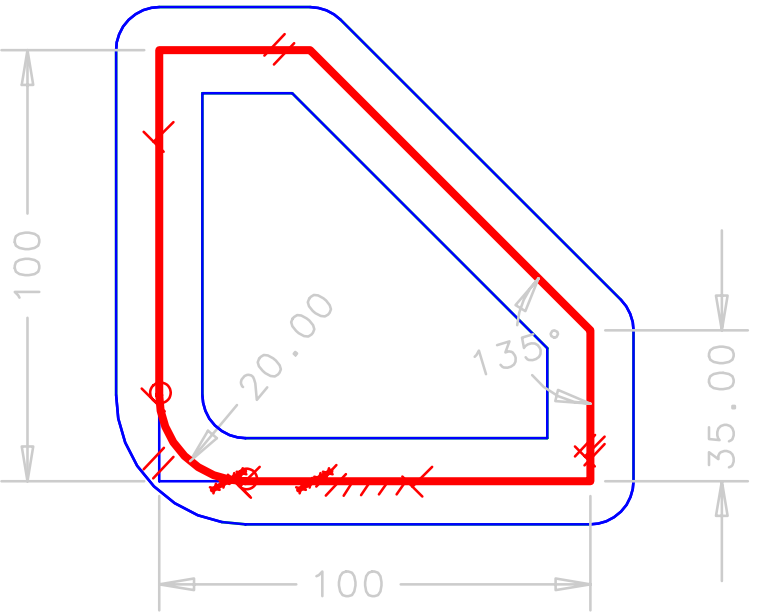
Wireframe

<–|–>

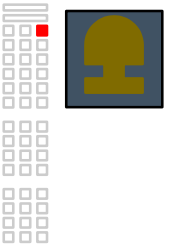
OK



Result



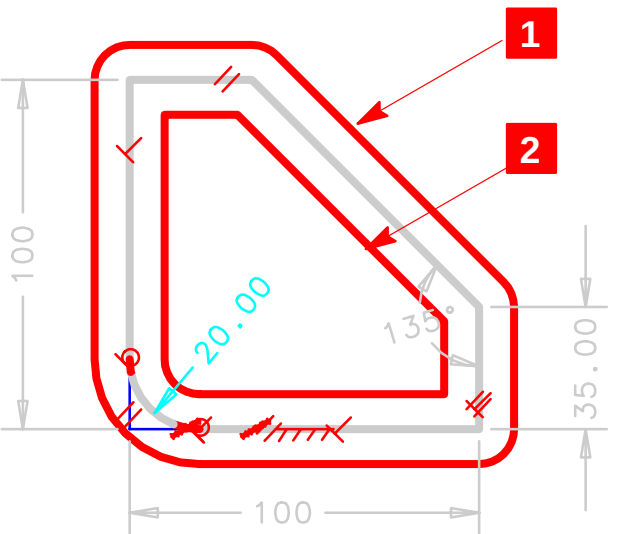
Create a new section from the inner and outer curves.



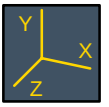
1

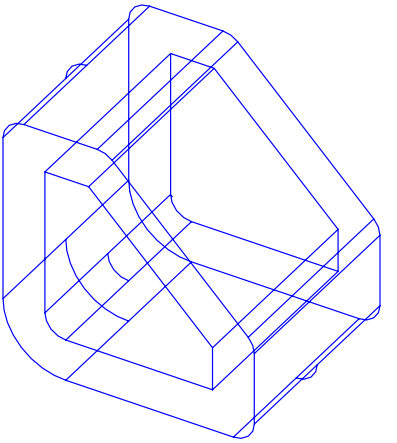
2

(Done)



Now extrude the new section 100mm to create the part shown below.

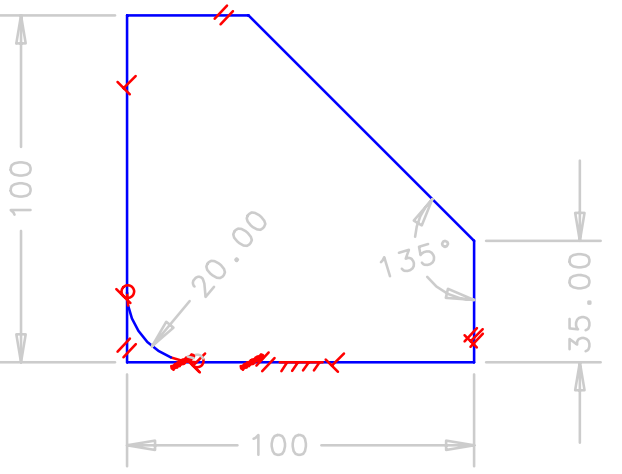




Open your model file to the last save, to get the original sketch on the workbench.

Hint

Control-Z



In any 2D drawing, there must be a certain number of dimensions to fully define a part so it can be manufactured. More dimensions than this number would be redundant, while fewer dimensions would leave ambiguities.

The same holds true with any 2D sketch. If the shape is not fully constrained, unpredictable results may occur when you make modifications.

This section shows you how to determine if you have any unconstrained geometry.

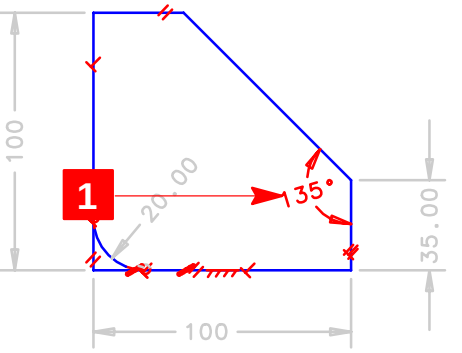
To demonstrate how to find free degrees of freedom, delete the angular dimension so that the section is under-constrained.

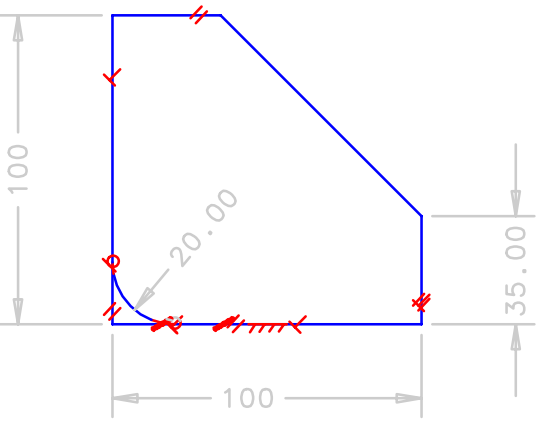
1

 (Done)

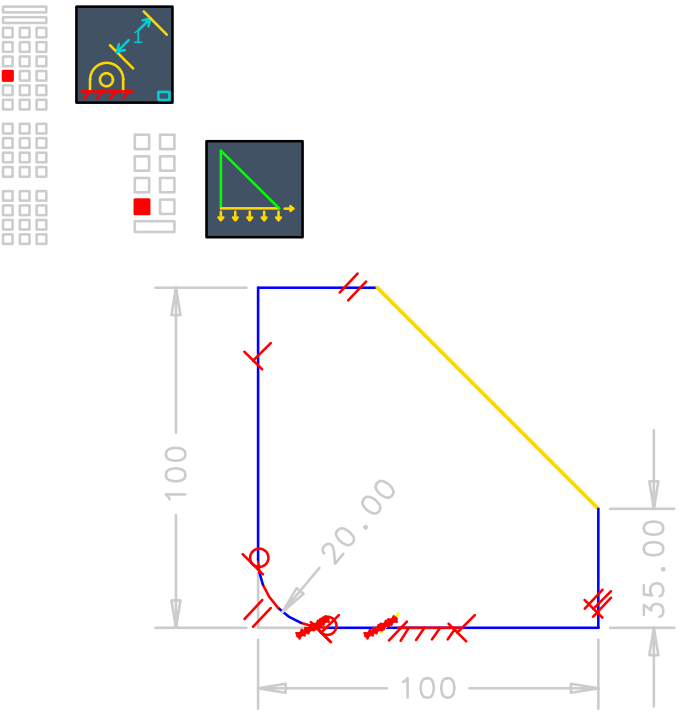
 *Yes*

 (to deactivate icon)





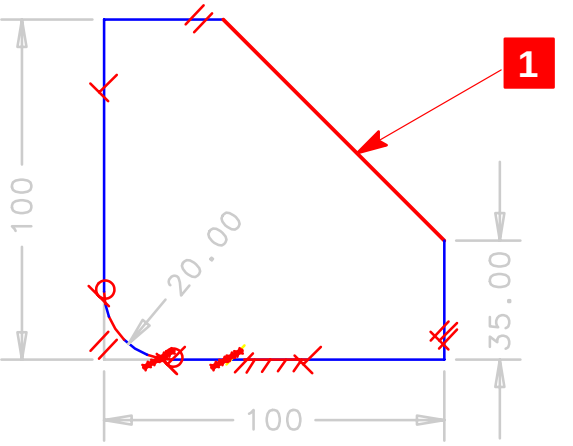
Now, use the *Show Free* icon to find out what is not constrained.



Things to notice

Notice how the diagonal line is shown in yellow, indicating that it is not fully constrained.

1 pick the diagonal line



 (Done)

Things to notice

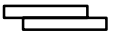
The arrows show you the geometry that is still free to move.

 (to deactivate icon)

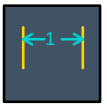
Close the *Constrain* panel.

 *Close*

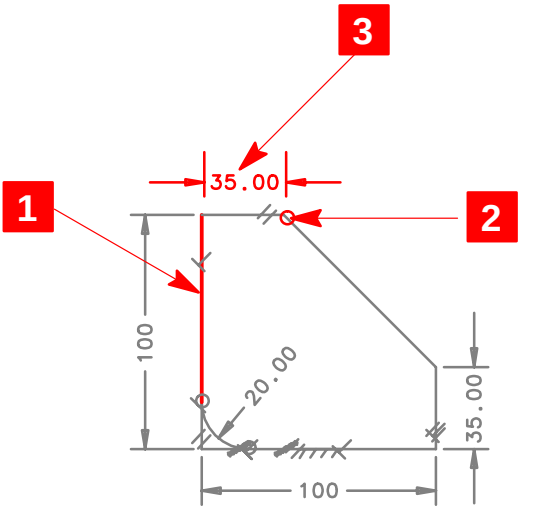
Recovery Point

 *File*
Save

Add a dimension that will fully constrain the section.

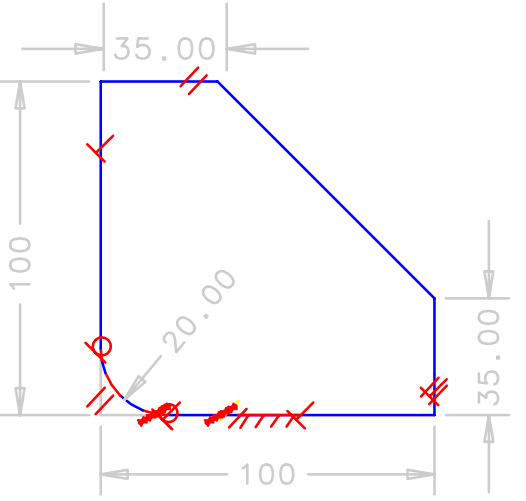


1. pick the line
2. pick the point
3. place the text

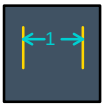


Things to notice

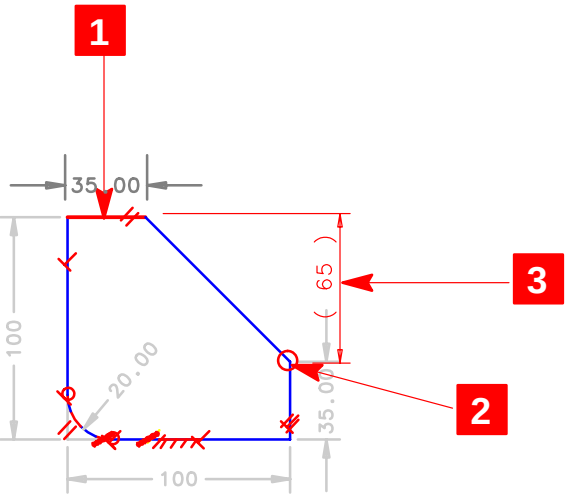
Notice how the line colors changed from yellow to blue as the dimension was added, since this dimension fully constrained the section.



Now, see what happens when you add a dimension to a fully constrained section.



1. pick the line
2. pick the point
3. place the text



Things to notice

Redundant dimensions are shown in parentheses and are called “reference” dimensions. They cannot be used to drive the geometry, but will change when other dimensions are modified.

Select sketch planes 1 of 7

It is important to understand that sketching is always done on a plane. There are two primary methods to orient the sketch plane:

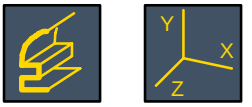
- *Sketch on Workplane*
  – the workplane (this is the default)

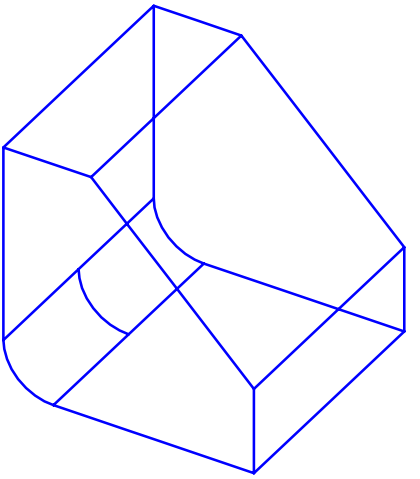
- *Sketch in Place*
  – face of the part
  – reference plane associated with the part
  – coordinate systems associated with the part

Sketch on Workplane is primarily used to create a new part. *Sketch in Place* is primarily used to add features to a part.

Select sketch planes 2 of 7

First, extrude the section on your workbench to create the part shown.



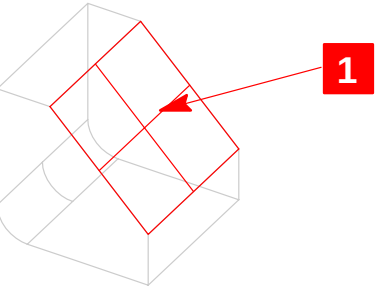


Recovery Point

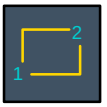
File
Save

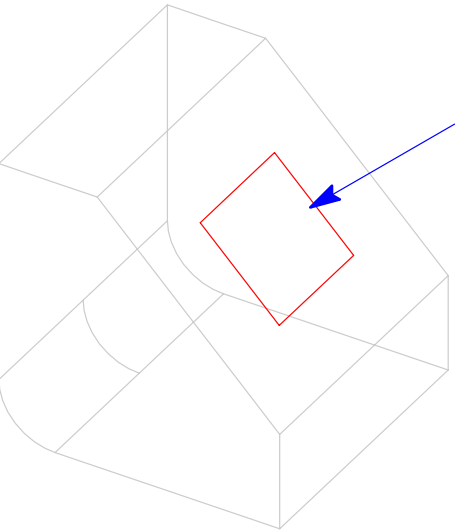
Sketch in place on the angled face.

1 pick face



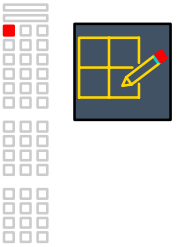
Now, sketch a rectangle on this face. Don’t worry about any dimensions that may appear.

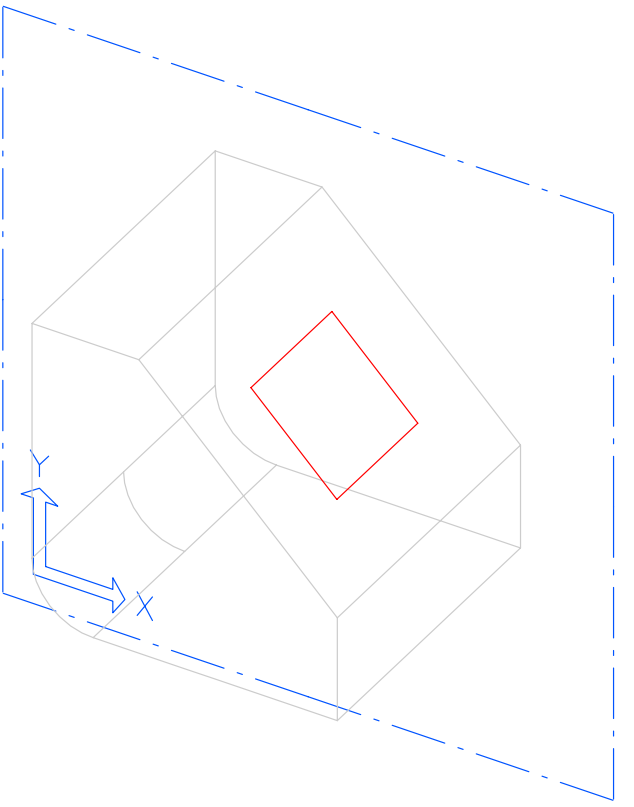




Select sketch planes 4 of 7

Revert to sketching on the workplane before continuing.





Sketch a rectangle on the workplane. Don’t worry about dimensions.

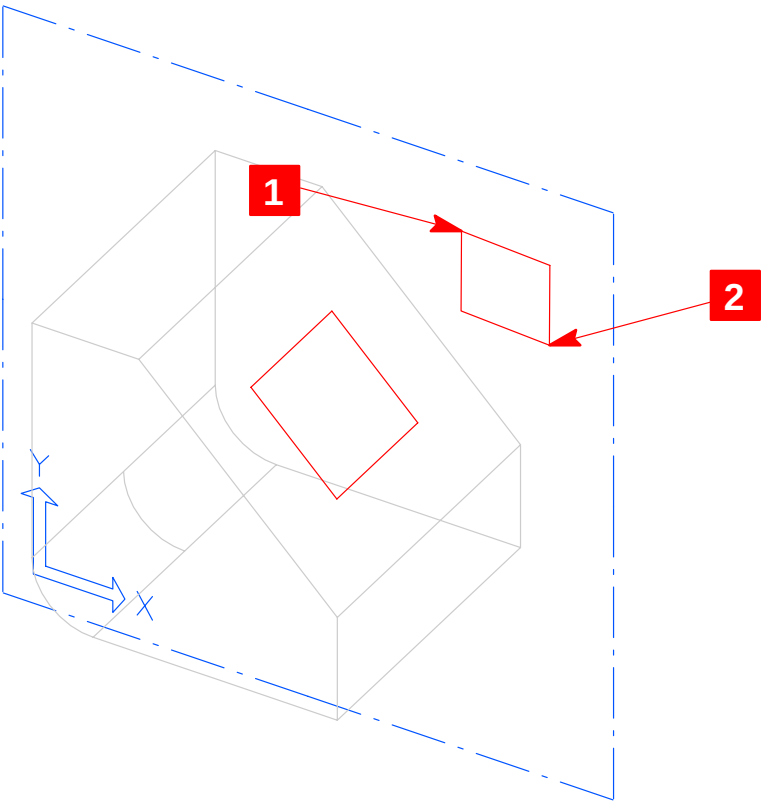
Before picking a point on the workplane, move the cursor back and forth above the other rectangle.

Notice that the Dynamic Navigator does not recognize the geometry on the other rectangle.

1

2

 (to deactivate icon)



Select sketch planes 6 of 7

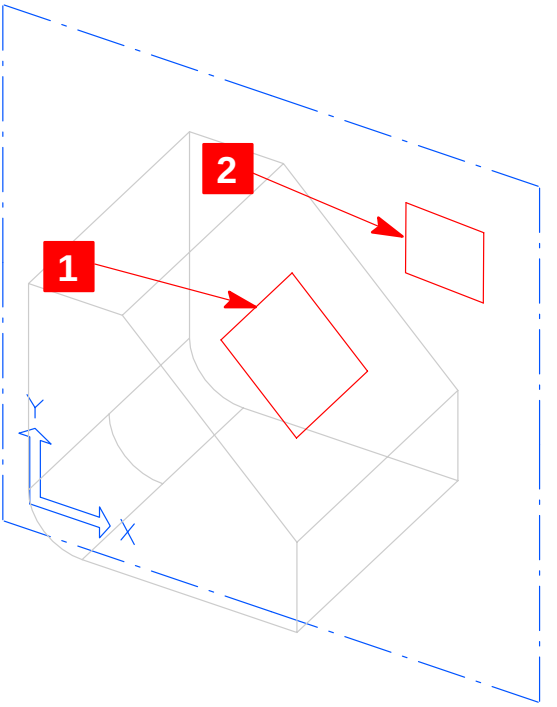
The two rectangles not only lie in different planes in space, they are not associated with the same sketch pad.

Use the *Info* icon and check each rectangle.

1

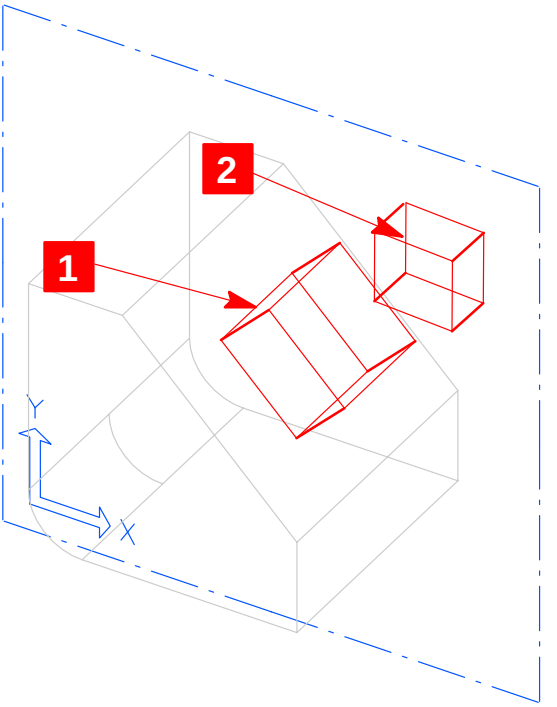
2 shift-pick

 (Done)



Check *I-DEAS List*.
Scroll up in the window to see that one rectangle is associated to the part, and the other with the workbench (notice the part name is “workbench_wireframe”).

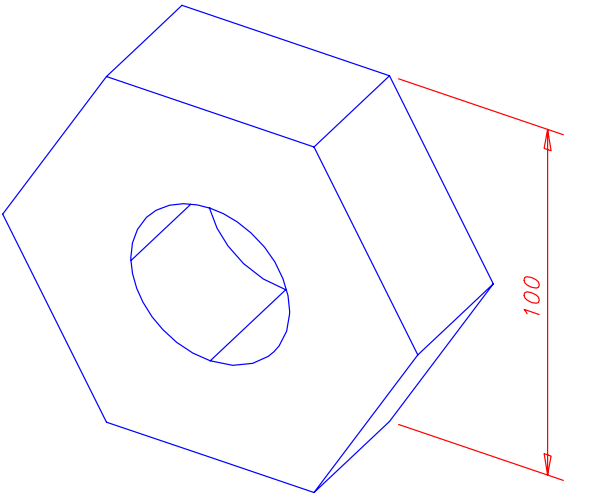
Extrude each of the rectangles. Notice the different default options when you extrude.



Things to notice

In one case, the default is to protrude from the part. In the other case, the default is to create a new part.

Create and constrain the wireframe geometry, and create a section that could be used to define a hexagon (like the one shown).



Match the dimensions so that changing one side's dimension will change them all.

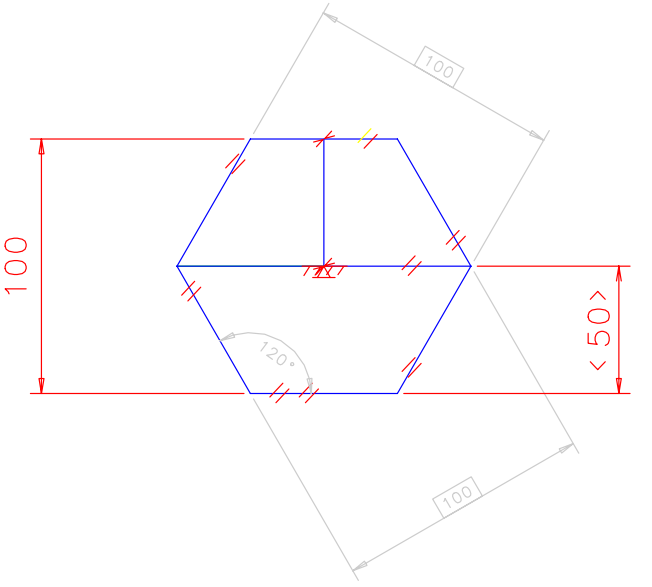
Try this on your own. If you need help, refer to the next few pages, which give you hints on how to create the section.

If you would rather try this at a later time, skip to the last page for the tutorial wrap-up.

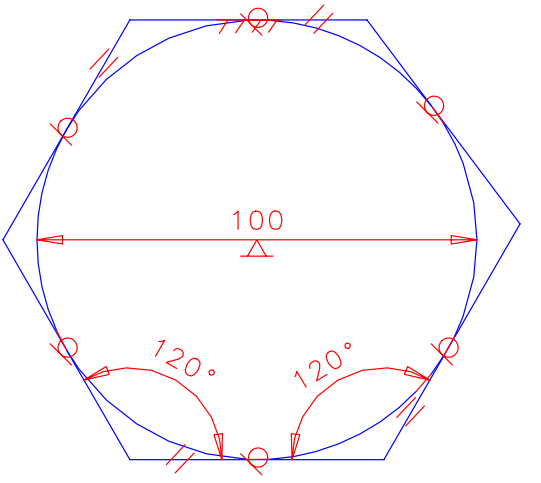
There are many different ways to constrain this shape. In both of the solutions shown, extra geometry was used in order to create constraints so that the center could be anchored.

To create the section, you should use *Stop at Intersections* in both cases.

Solution 1

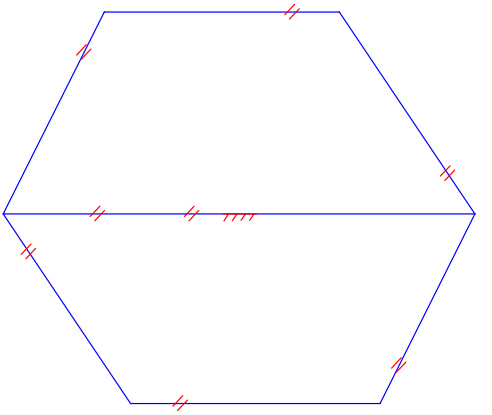


Solution 2



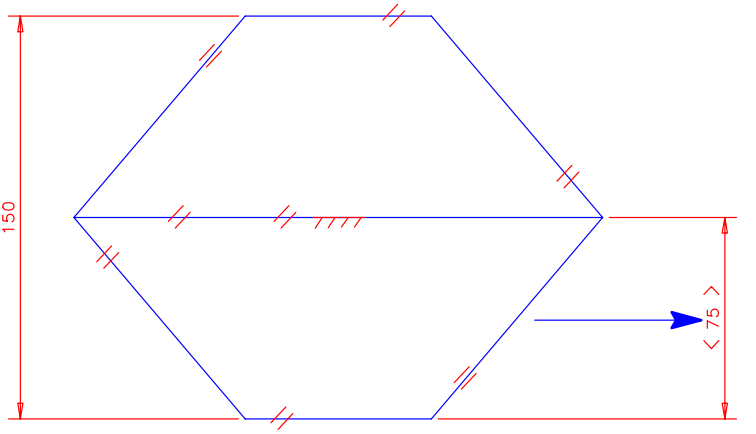
Solution 1 Step 1.

Step 2.



Make sure opposite sides are constrained parallel.

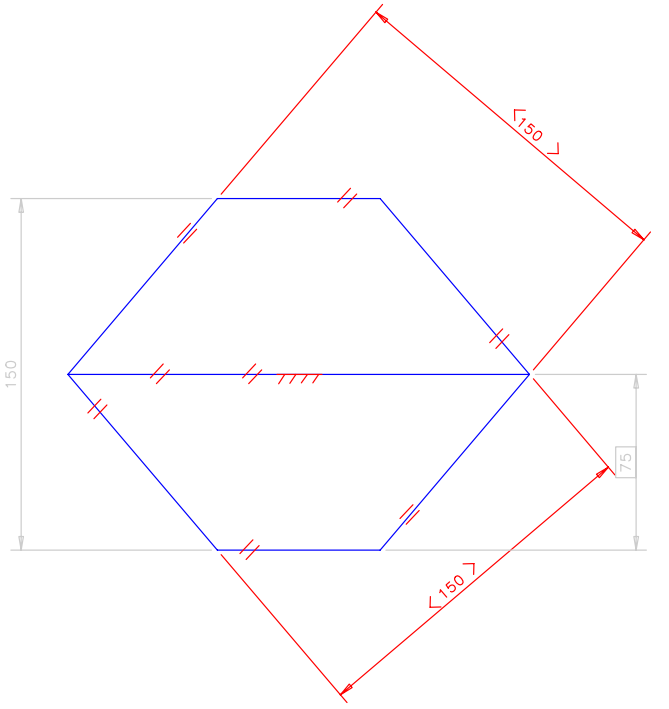
Solution 1 Step 3.



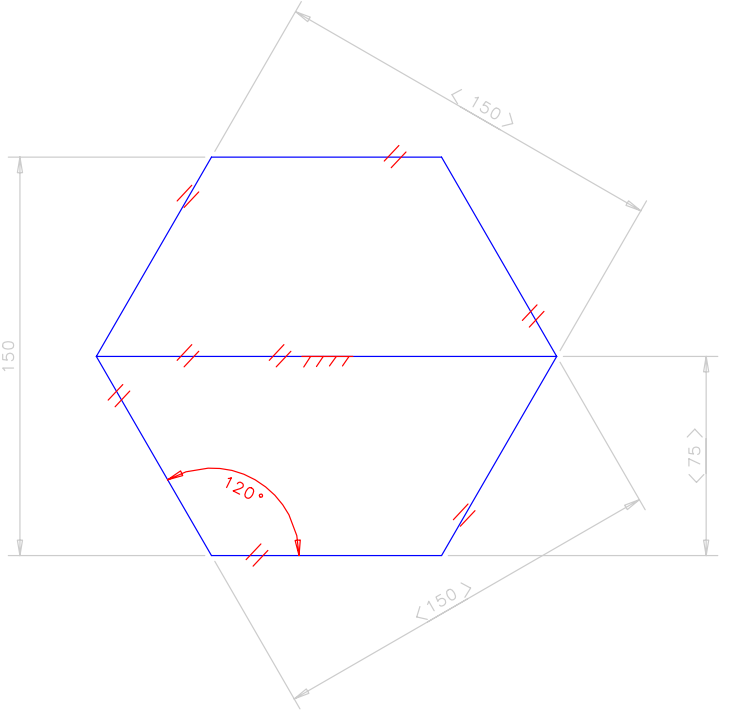
Hint

When you match dimensions, type in /2 to force the dimension indicated above to be one-half of the other. For example: D2 = D1/2.

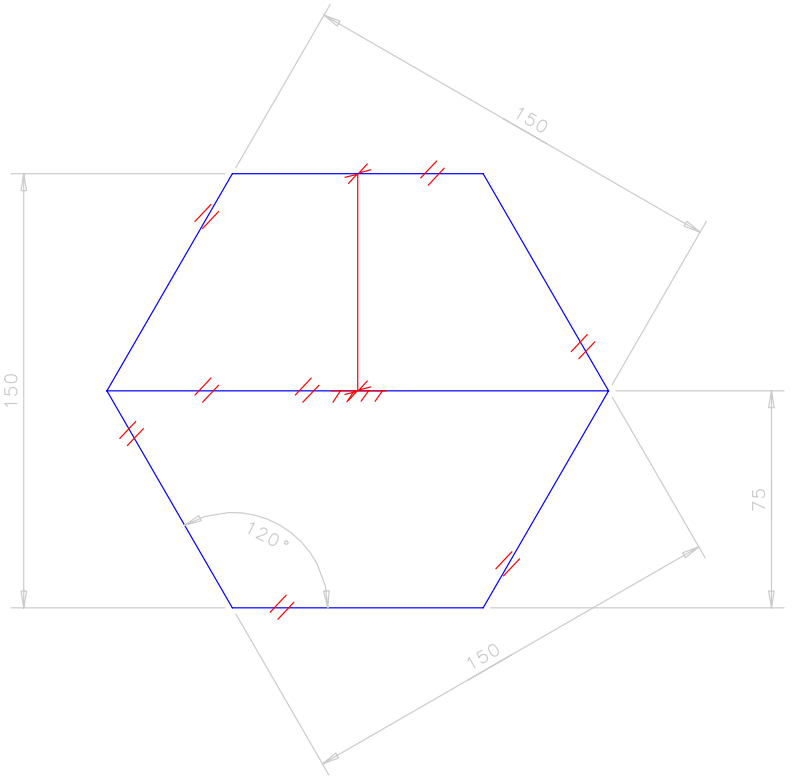
Solution 1 Step 4.



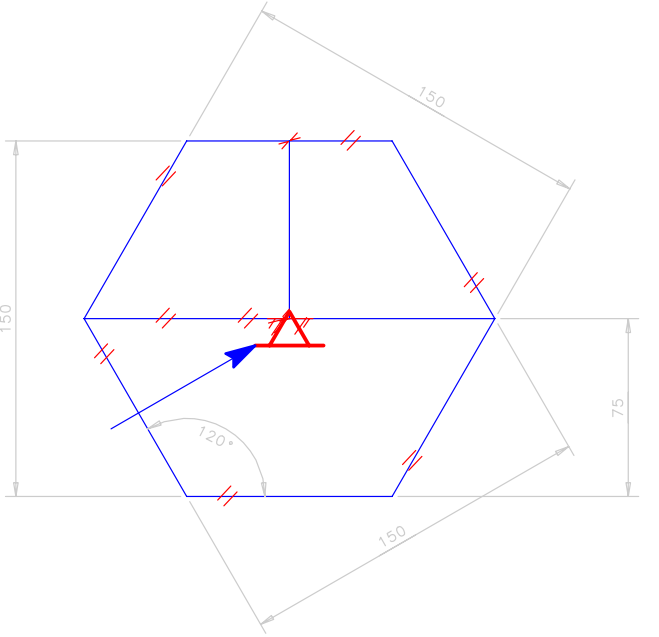
Solution 1 Step 5.



Solution 1 Step 6.

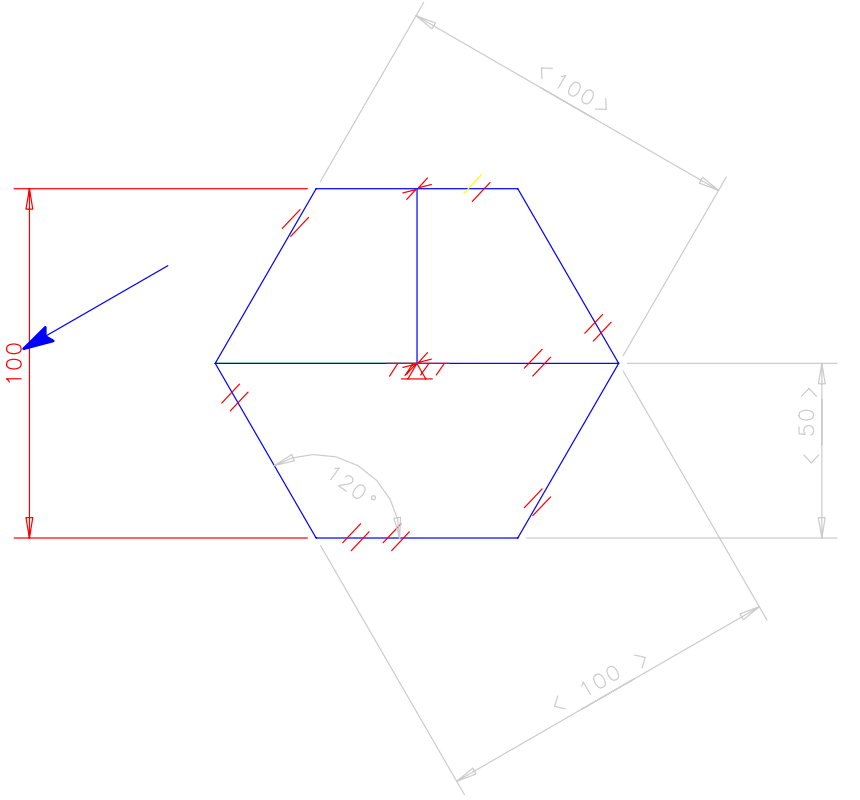


Solution 1 Step 7.

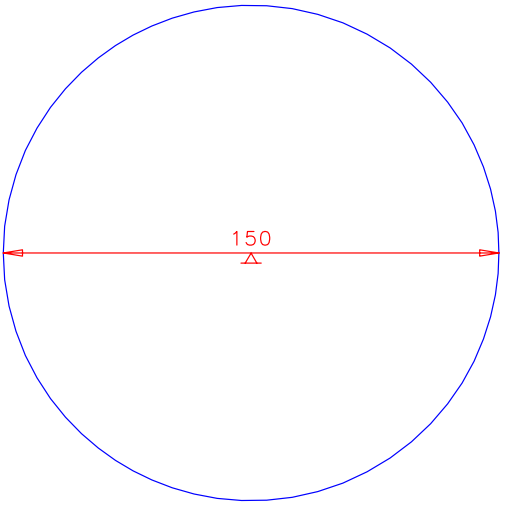


Solution 1 Final Test.

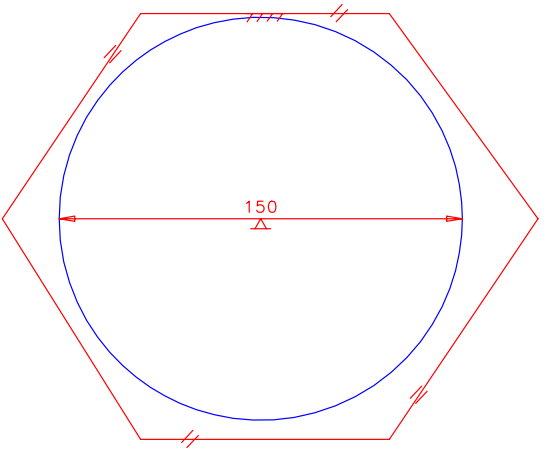
To get the final result and test your design, modify or drag the dimension shown to 100 to see if the other dimensions change accurately.



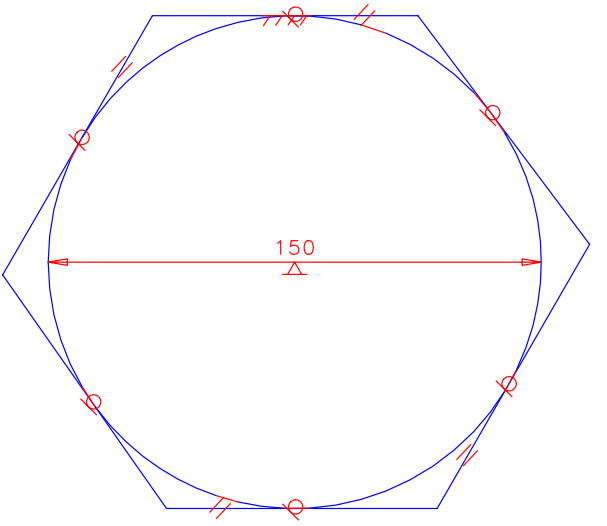
Solution 2 Step 1.



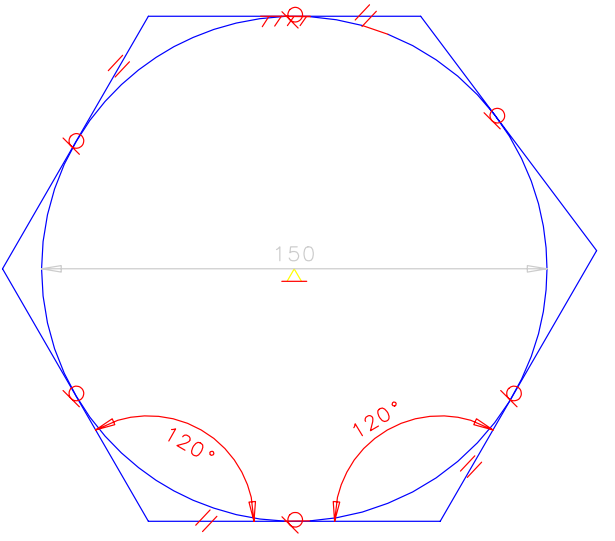
Solution 2 Step 2.



Solution 2 Step 3.

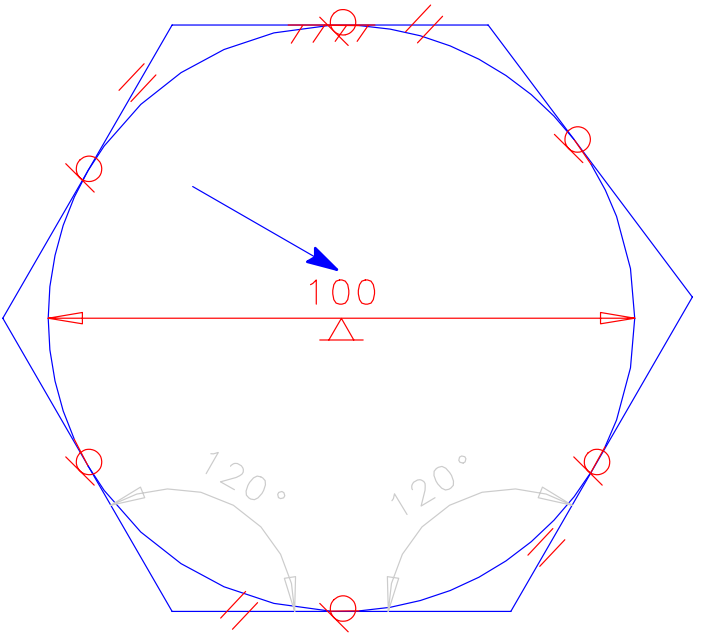


Solution 2 Step 4.



Solution 2 Final Test.

To get the final result and test your design, modify or drag the dimension shown to 100 to see if the other geometry changes accurately.



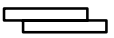
Tutorial wrap-up

You have completed the Using Sections and Sketch Planes 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*

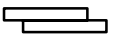
Design User's Guide
- Design Techniques and Examples
  - Modification Strategy
    - The Sketchpad: How Wireframe Geometry Is Stored
  - Feature Examples
    - Wireframes
    - Designing and Modifying Sections
    - Creating Features on Parts with Planar and Non-planar Surfaces

Design Reference Guide
- Master Modeler
  - Wireframe Geometry
    - Constraining Wireframe Geometry

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

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

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