Autodesk Inventor
Module 14
Revolving

Learning Outcomes

When you have completed this module, you will be able to:

1 Describe a centerline object and explain how it is inserted and used in a 2D Sketch.
2 Describe how a Base sketch is revolved with and without the use of a centerline to create a solid model.
3 Apply the REVOLVE command to create a solid model from a Base sketch.

Revolving

When drawing symmetrical objects it is much easier to create the model by *revolving* the Base sketch around an *axis* rather then extruding it. The axis, which can be one of the lines in the sketch or a centerline, must always be located in the center of the symmetrical model. The sketch can be revolved any angle between 0 and 360 degrees.

In this module, the basic features of the REVOLVE command are taught. The Inventor Advanced eCourse will cover the more advanced features.

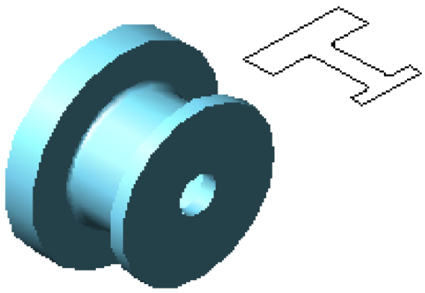


Figure 14-1
A 2D Sketch Revolved Around a Line in the Sketch

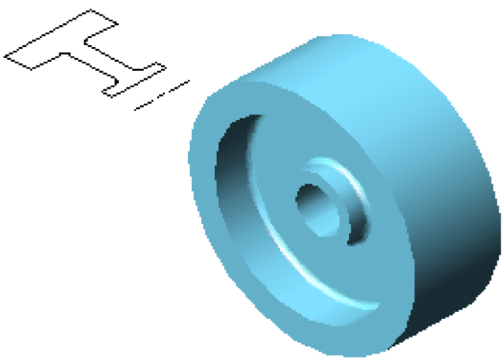


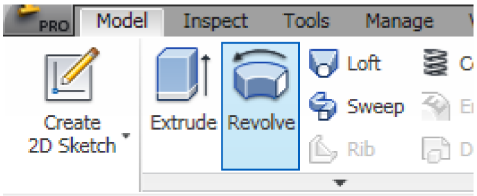
Figure 14-2
The Same 2D Sketch Revolved around a Centerline

The models in Figure 14-1 and 14-2 were created by revolving the same Base sketch around an axis. Take note how the two solid models that were created using the same sketch are quite different. In Figure 14-1, the line on left side of the sketch was used as the axis while in Figure 14-2, it was the centerline that was added to the sketch and used as an axis or revolution.

Inventor Command: REVOLVE

The REVOLVE command is used to create a solid model by revolving the Base sketch around an axis.

Shortcut: **R**



Centerline

A *centerline* is a line with its properties set to act as a centerline. In the REVOLVE command, a centerline is automatically recognized as the axis for the revolution. The two methods of drawing a centerline, which is similar to drawing construction a line, are as follows:

Method 1 Draw the line using the LINE command and then select it. While it is selected, click the Centerline icon.

Method 2 Enable the Centerline icon and then draw the line, using the LINE command.

The Centerline icon is shown in Figure 14-3. A centerline will display as the centerline linetype.

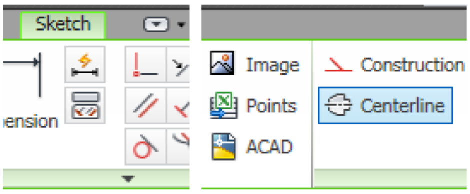


Figure 14-3
Centerline Icon

WORK ALONG

Revolving a Sketch Without using a Centerline

Step 1 Check the default project and if necessary, set it to Inventor Course.

Step 2 Using the NEW command, start a new part file using the template English-Modules Part (in).ipt.

Step 3 Save the file with the name Inventor Workalong 14-1. (Figure Step 3A and 3B)

Step 4 Start a new sketch on the Front or XZ Plane. Project the Center Point onto the sketch.

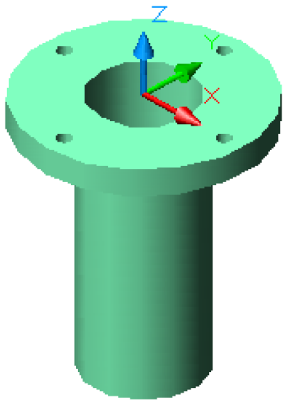


Figure Step 3A
3D Model

Author's Comments: In this and the next workalong, you will be constructing the same solid model. This one without the use of a centerline and the next one with the use of a centerline. Either method is an acceptable way of creating the solid model. These two workalongs will allow you to practice both methods.

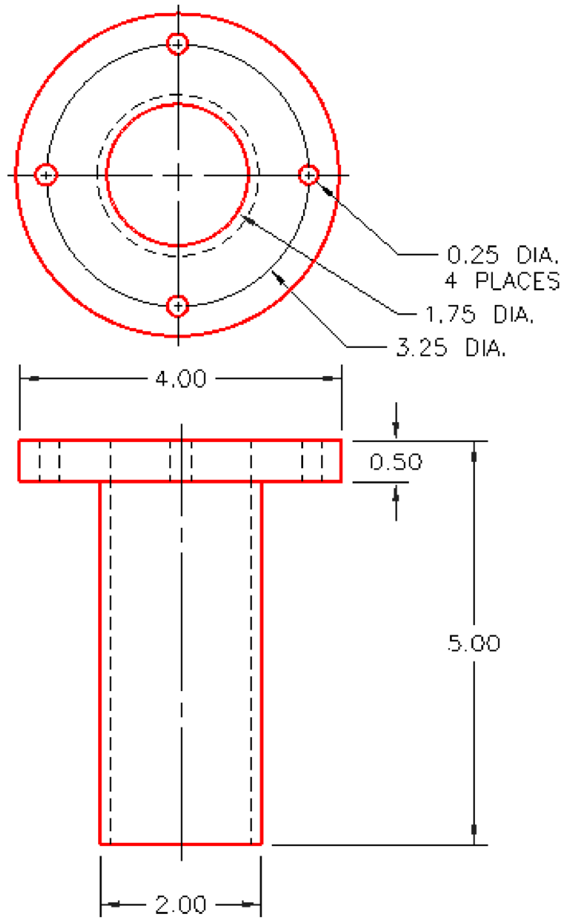


Figure Step 3B
Dimensioned Multiview Drawing

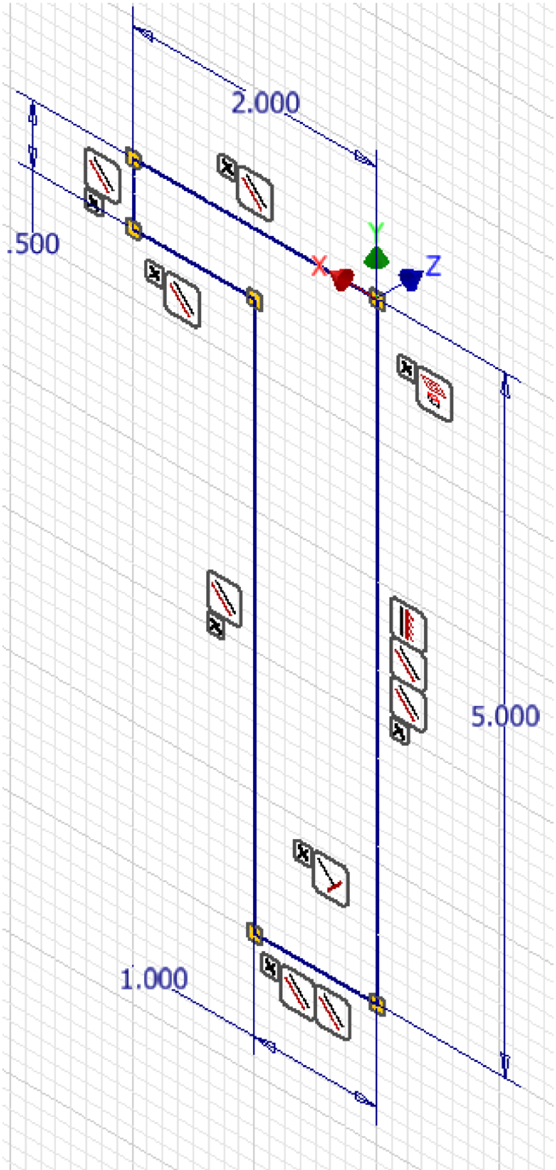


Figure Step 5

Step 5 Draw and dimension one-half of the Front view as shown in the figure. Ensure that the sketch is fully constrained (Figure Step 5)

Author's Comments: Note how the sketch is a profile of the object, that when revolved, will create the solid model. For now, ignore the center hole and the 4 small holes. The line located in the center of the model is used as the axis. The holes will be inserting after the sketch is revolved to complete the solid model.

Author's Comments: Your geometrical and dimensional constrains may not match the figure exactly. Ensure that the sketch is fully constrained.

Step 6 In Model mode, enter the REVOLVE command. It will highlight the sketch automatically as the area to revolve. (Figure Step 6)

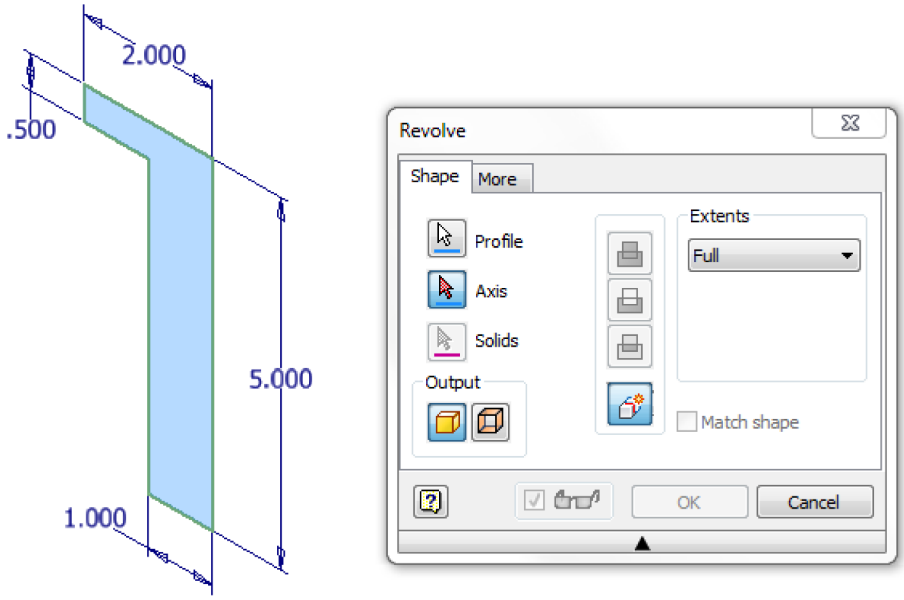


Figure Step 6

Step 7 In the Revolve dialogue box, set the Extents to Full and enable the Axis icon. Select the line on the right side of the sketch as the axis. The Full setting means that it will be revolved 360 degrees. (Figure Step 7)

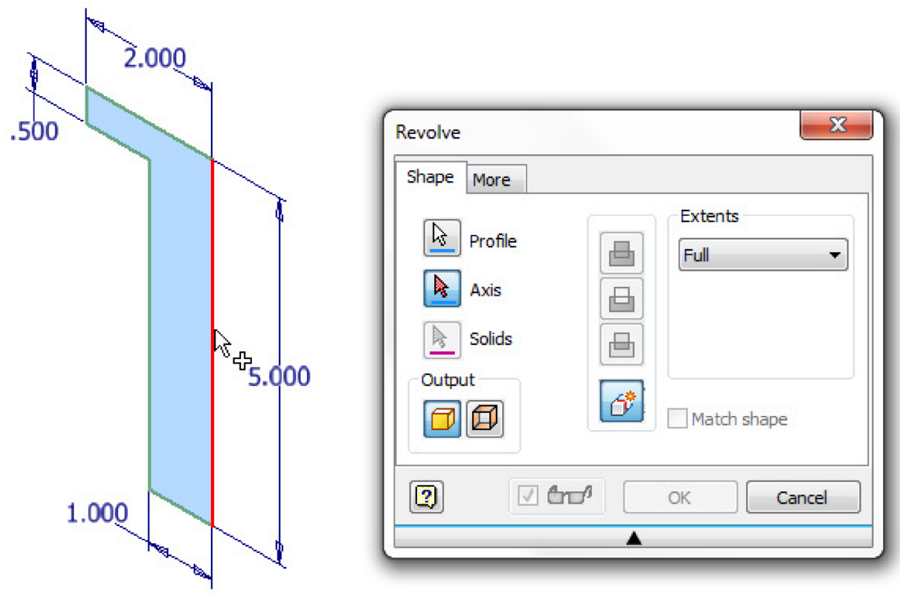


Figure Step 7

Step 8 After you select the axis, the REVOLVE command will display the Base Solid model as it is revolved. If this is the desired outcome, click OK. (Figure Step 8A and 8B)

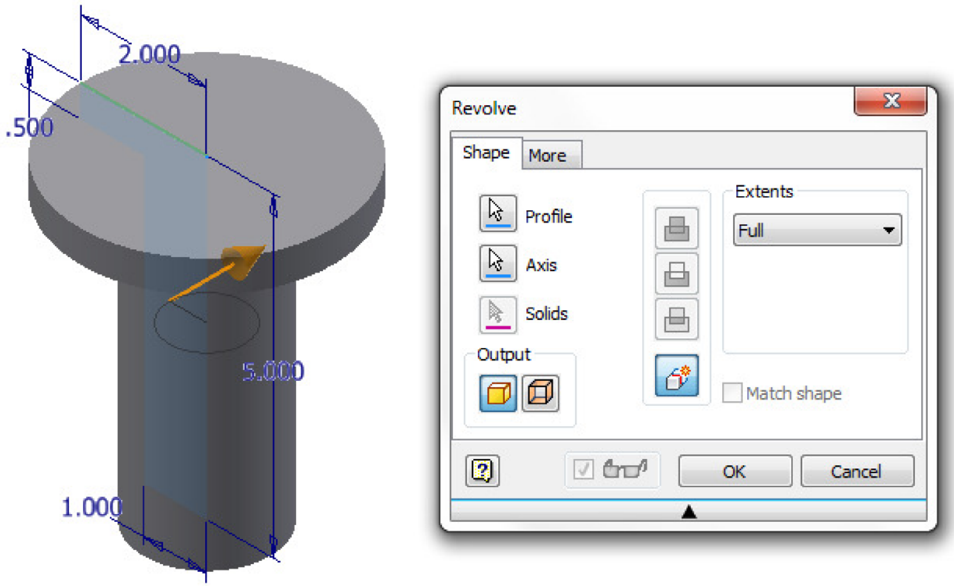


Figure Step 8A

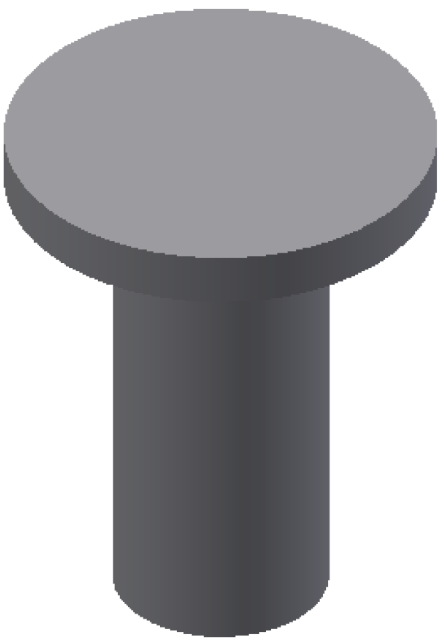


Figure Step 8B

Step 9 Start a new sketch on the top plane of the model. (Figure Step 9)

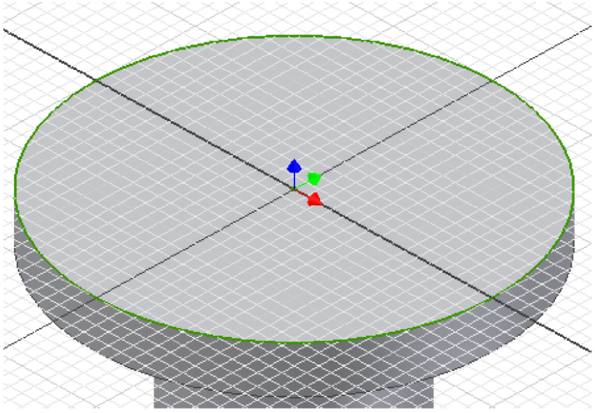


Figure Step 9

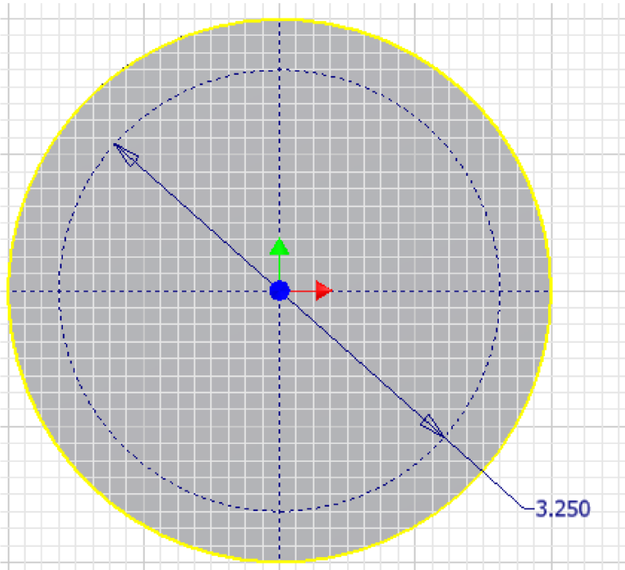


Figure Step 10A

Step 10 Using what you learned in Module 12, draw a construction circle and four construction lines. Insert a dimension for the diameter of the circle. Ensure that the sketch is fully constrained. (Figure Step 10A and 10B)

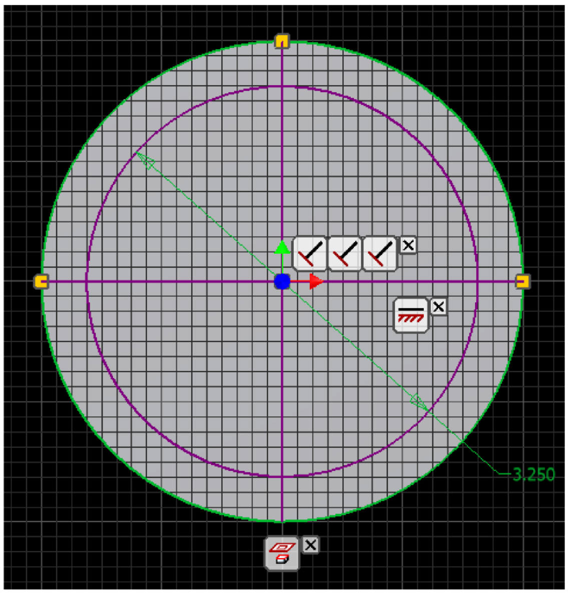


Figure Step 10B

Author's Comments: Ensure that you snap the lines correctly when you draw them. If you don't, you will have trouble fully constraining the sketch. If you have trouble doing this step, look at the workalong in Module12.

Step 11 Using the technique that you learned in Module 12, draw the 4 circles. Dimension one and constrain the additional 3 with an Equal constraint. (Figure Step 11A and 11B)

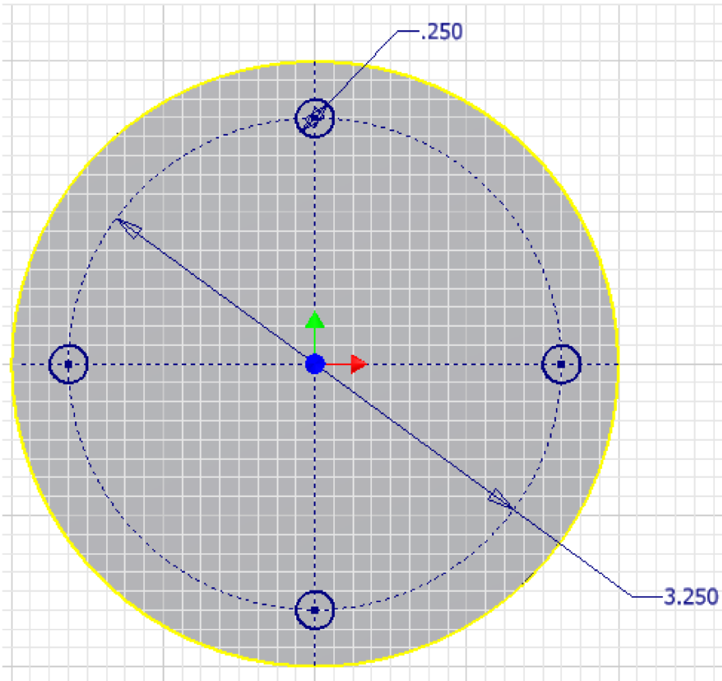


Figure Step 11A

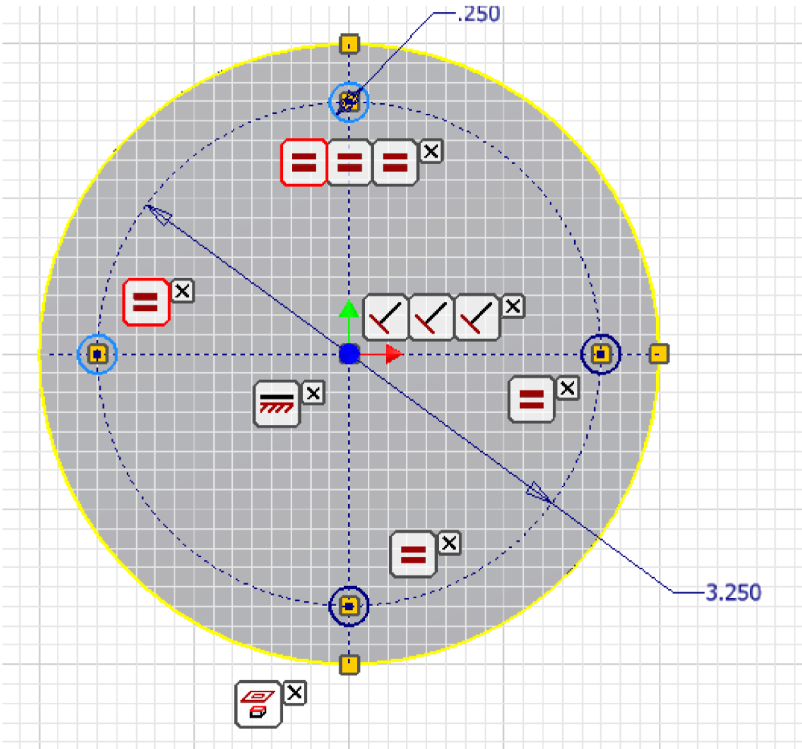


Figure Step 11B

Author's Comments: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

Step 12 Extrude the four circles to the To Next extents. (Figure Step 12)

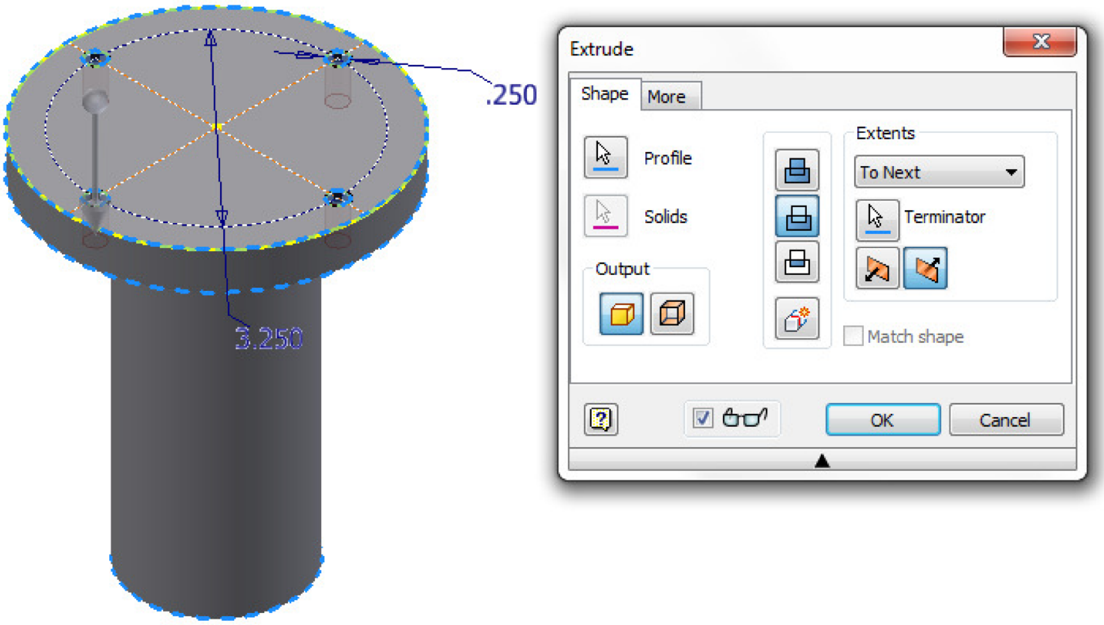


Figure Step 12

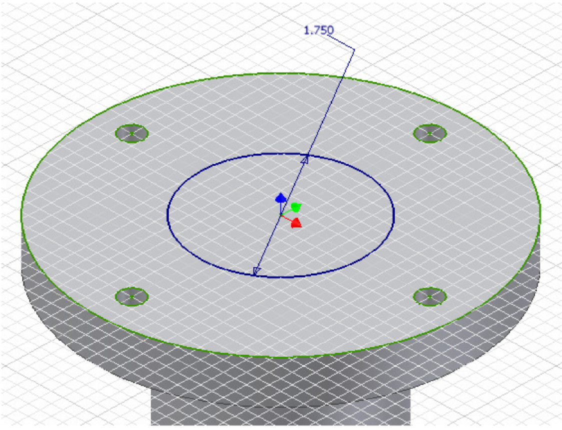


Figure Step 13

Step 13 Start another sketch on the top plane and draw a circle by offsetting the outside diameter. Dimension the circle and extrude it to complete the model. (Figure Step 13)

Author's Comments: Using an offset, will automatically constrain the circle.

Step 14 Change the color to Aluminum - Polished. (Figure Step 14)

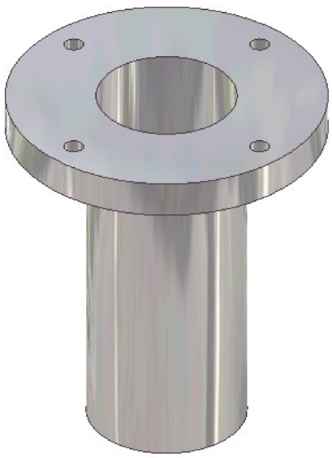


Figure Step 14

Step 15 Save and close the file.

WORK ALONG

Revolving a Sketch using a Centerline

Step 1 Check the default project and if necessary, set it to Inventor Course.

Step 2 Enter the NEW command to start a new part file using the template English-Modules Part (in).ipt.

Step 3 Save the file with the name Inventor Workalong 14-2. (Figure Step 3)

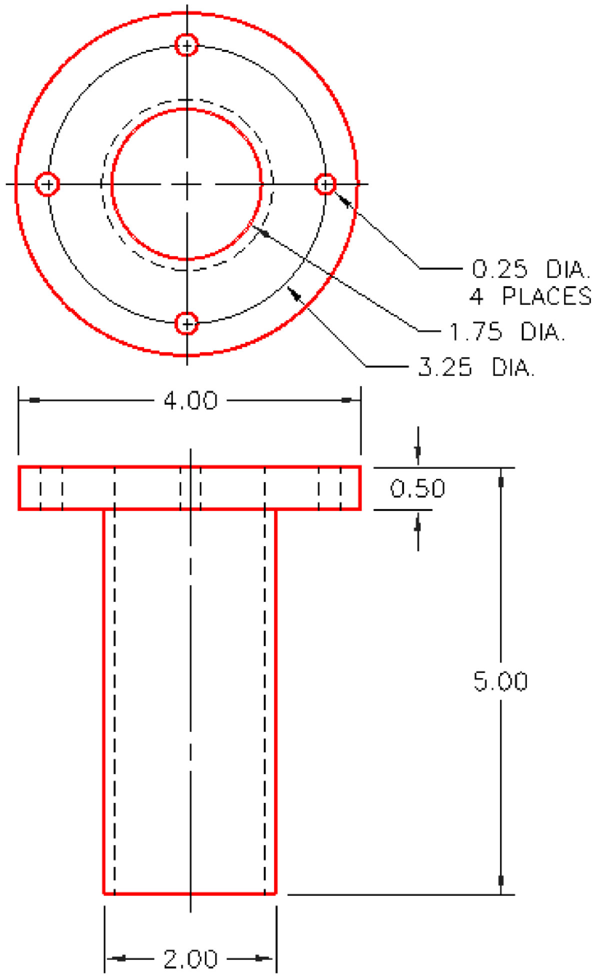


Figure Step 3
Dimensioned Multiview Drawing

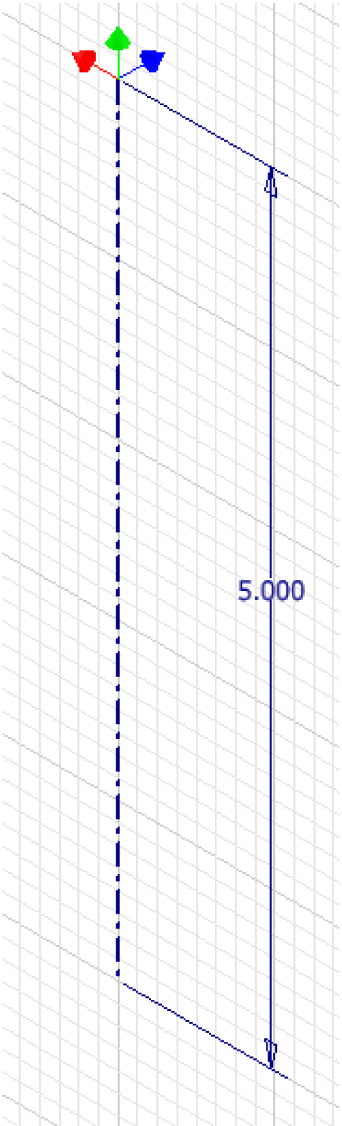


Figure Step 4A

Step 4 Start a new sketch on the Front or XZ Plane. Draw and dimension a line start it by snapping to the Center Point. Draw it 5 inches in the negative Y direction. This is the length of the model. This is centerline of the solid model. Change the line's properties to a centerline. (Figure Step 4A and 4B)

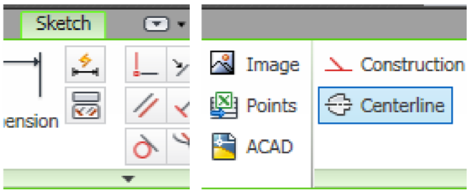


Figure Step 4B

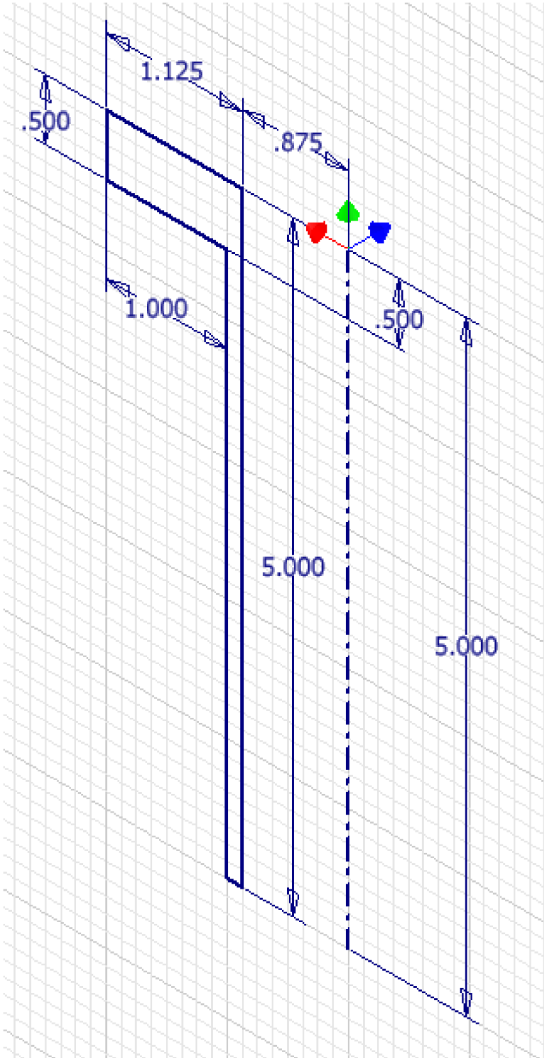


Figure Step 5A

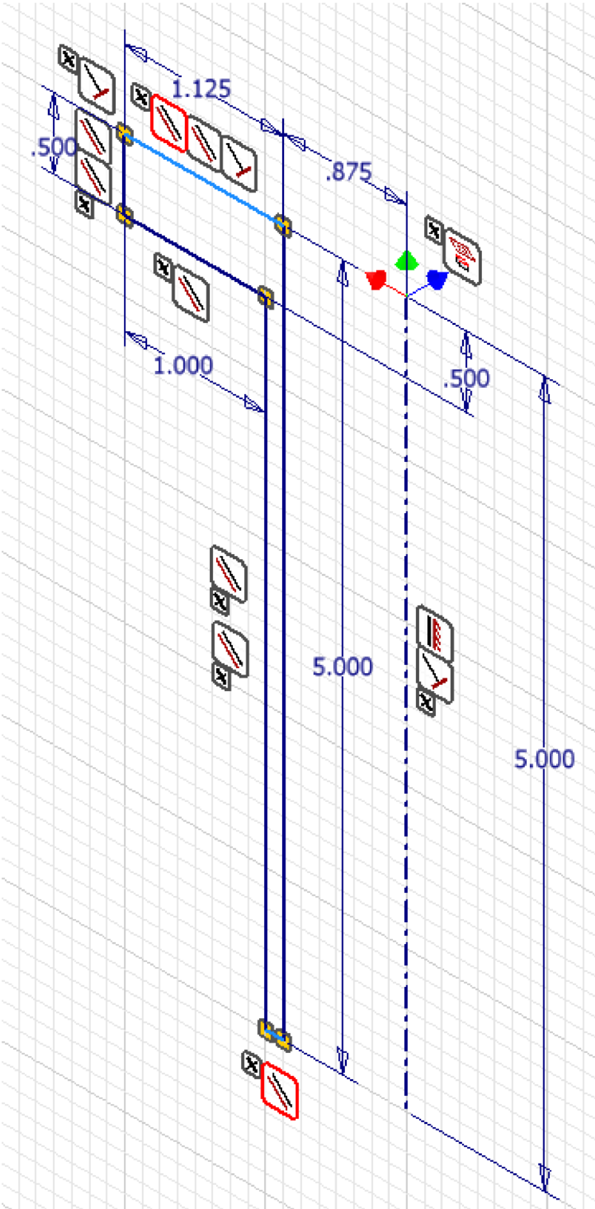
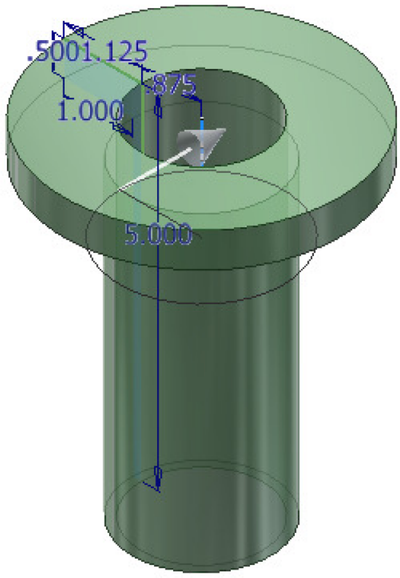


Figure Step 5B

Step 5 Draw and dimension one-half of the Front view as shown in the figure. (Figure Step 5A and 5B)

Author's Comments: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

Step 6 Return to Model mode and enter the REVOLVE command. Since a centerline is part of the sketch, the REVOLVE command will automatically use it as the axis to revolve the sketch around. It will display the outcome of the revolution. (Figure Step 6)



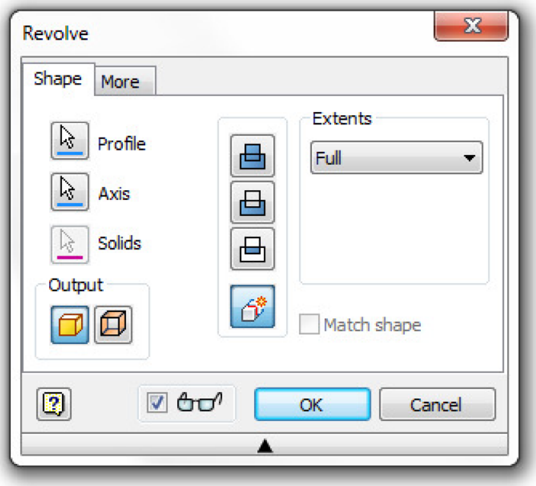


Figure Step 6

Step 7 In a new sketch, add the four smaller circles and extrude them to complete the model.

Step 8 Change the color to <u>Aluminum - Polished</u>.
(Figure Step 8)

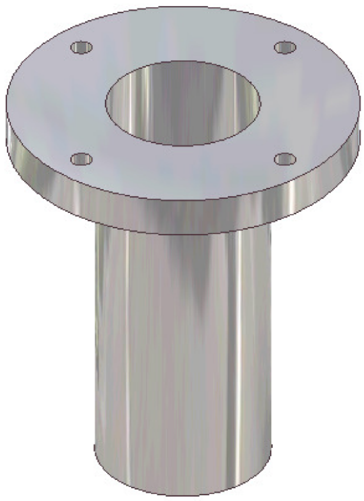
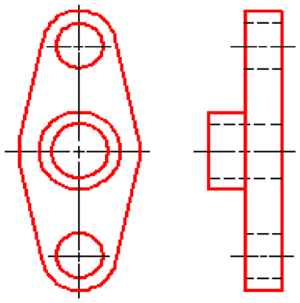


Figure Step 8

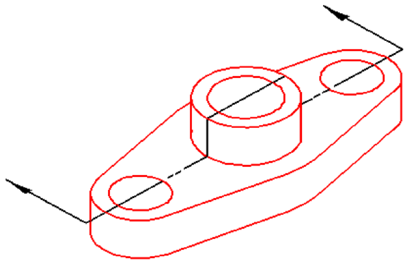
Step 9 Save and close the file.

Drafting Lesson
Cross Sections - Part 1

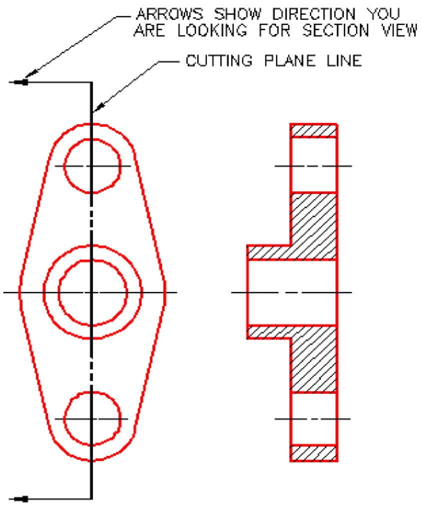


Normal Front and Right
Side View of an Object

When a view of an object requires a clearer description of the interior of the object or it is hard to dimension because of the hidden lines, a cross section view can be drawn in place of the normal multiview. Sometimes an additional view is drawn.

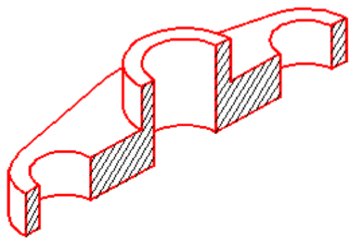


3D Model of the Object
Showing the Cutting Plane Line



A Front view and Right
Side Section View

A *cross section* view, also called a *section*, is a view of the object as if it were cut along a cutting plane and the two pieces pulled apart exposing the inside of the object. A *cutting plane* line is the line along the object where the cut would have been made. The arrows point in the direction that you are looking when you draw the view. The surfaces of the object that are solid when cut, are crosshatched.



3D Model of the Object
Showing it Cut on the
Cutting Plane Line

The Key Principles in Module 14

1 When drawing symmetrical objects, it is much easier to create the model by revolving the Base sketch around an axis rather then extruding it. The axis, which can be one of the lines in the sketch or a centerline, must always be located in the center of the symmetrical model.
2 A centerline is a line with its properties set to act as a centerline.

Lab Exercise 14-1		**Time Allowed: 60 Min.**
Part Name: Inventor Lab 14-1	**Project**: Inventor Course	**Units**: Millimeter
Template: Metric-Modules Part (mm).ipt	**Color**: Copper - Polished	**Material**: N/A

Step 1 Project the Center Point onto the Base plane.

Step 2 Note the location of X0Y0Z0. Draw the Base sketch and revolve it create the Base model of the object shown below. Revolve it by using a line in the sketch. Do not draw a centerline. (Figure Step 2A and 2B)

Step 3 Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.

Step 4 Apply the color shown above. (Figure Step 4A and 4B)

Step 5 Add the four small holes on a new sketch and extrude them.

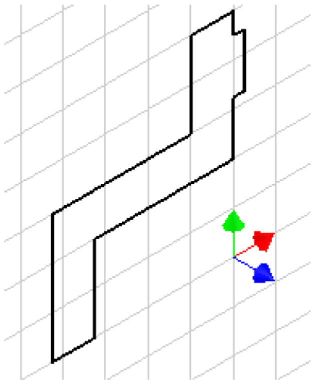


Figure Step 2A
Suggested Base Sketch - Right Side - YZ Plane

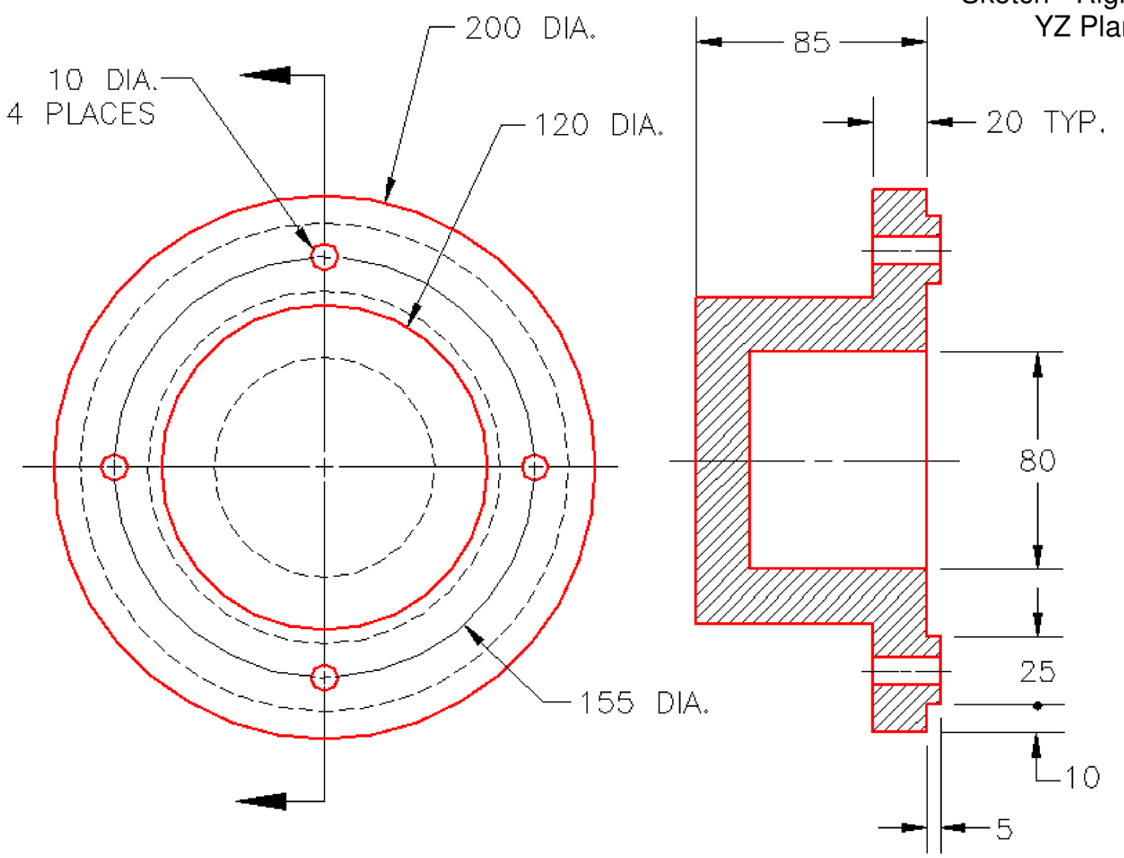


Figure Step 2B
Dimensioned Multiview Drawing

Author's Geometric Constrains: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints with the authors.

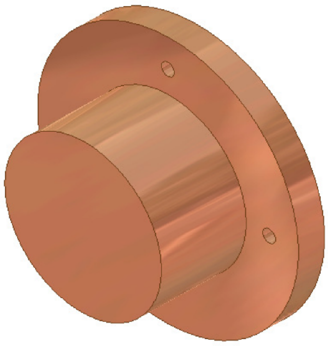


Figure Step 4A
Completed Solid Model
- Home View

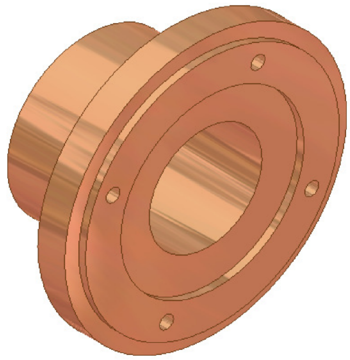
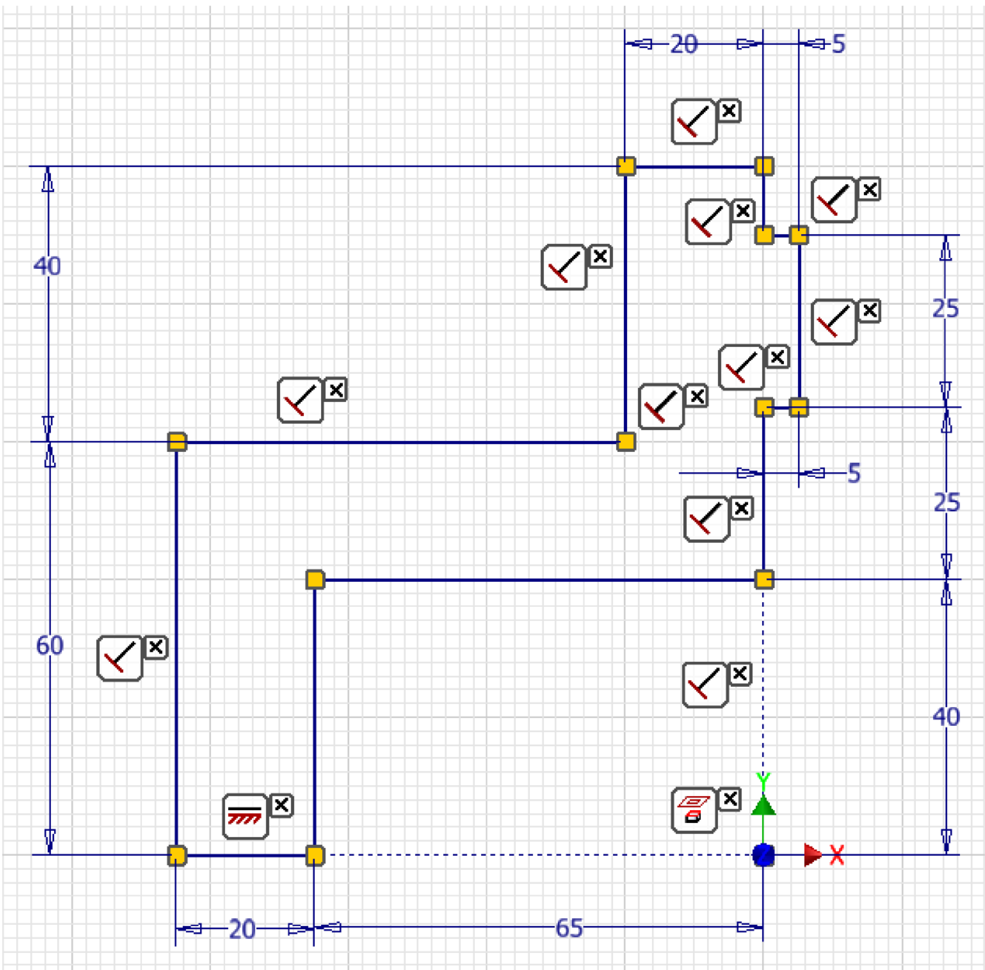
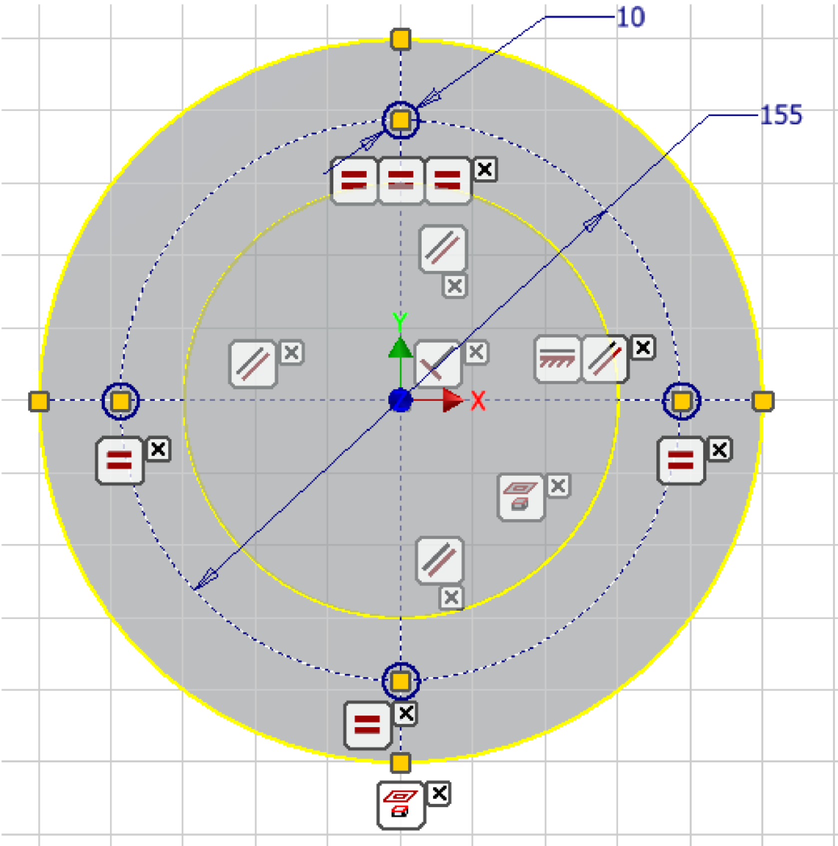


Figure Step 4B
Completed Model
- Rear View

Author's Comments: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

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155

Lab Exercise 14-2		**Time Allowed: 60 Min.**
Part Name: Inventor Lab 14-2	**Project**: Inventor Course	**Units**: Inches
Template: English-Modules Part (in).ipt	**Color**: Zinc	**Material**: N/A

Step 1 Project the Center Point onto the Base plane.

Step 2 Note the location of X0Y0Z0. Draw the Base sketch and revolve it create the Base model of the object shown below. Revolve it by using a centerline. (Figure Step 2A, 2B, 2C, 2D, and 2E)

Step 3 Apply all of the necessary geometrical and dimensional constraints to fully constrain all sketches.

Step 4 Apply the color shown above. (Figure Step 4A and 4B)

Step 5 Add the four small holes and the key on new sketches and extrude them.

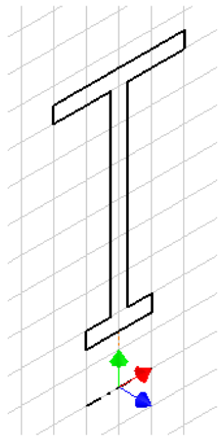


Figure Step 2A
Suggested Base Sketch - YZ Plane

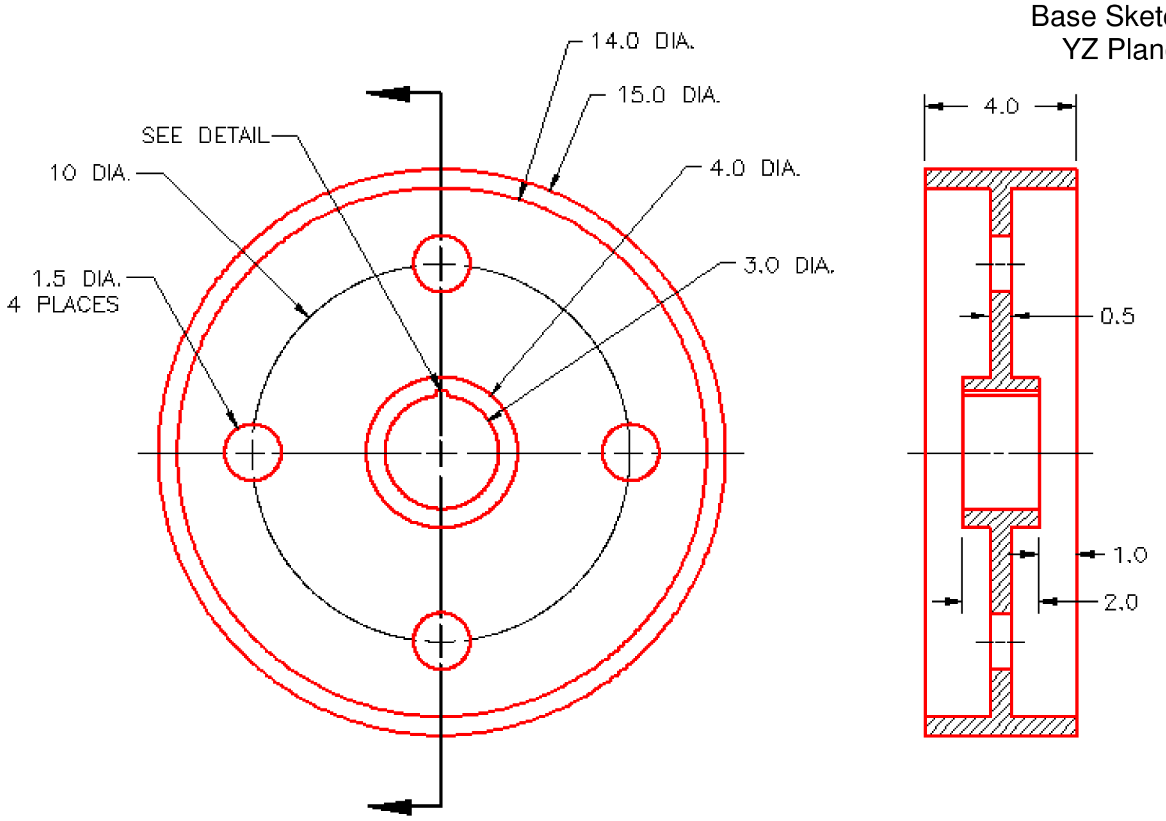


Figure Step 2B
Dimensioned Multiview Drawing

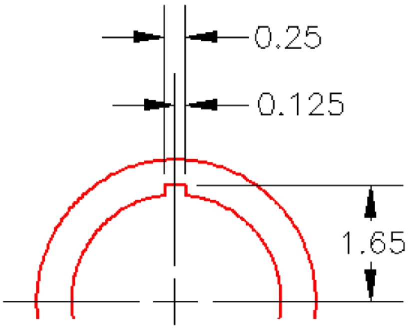


Figure Step 2C
Detail of Keyway

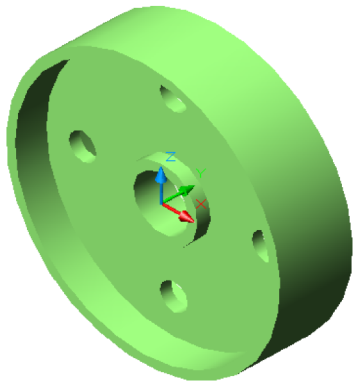


Figure Step 2E
Solid Model - Home View

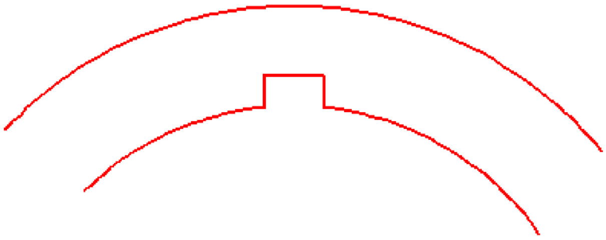


Figure Step 2D
Keyway is Flat Across the Top

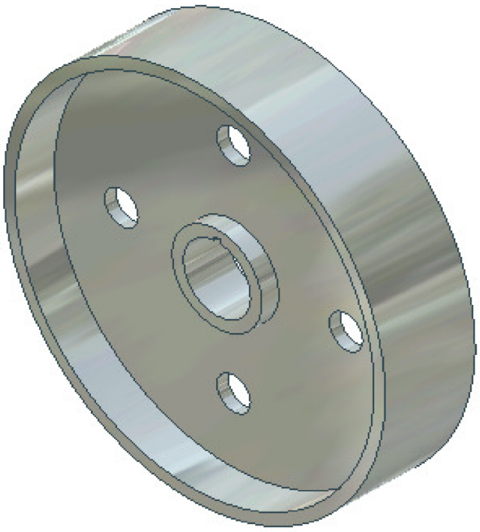


Figure Step 4A
Completed Model -
Home View

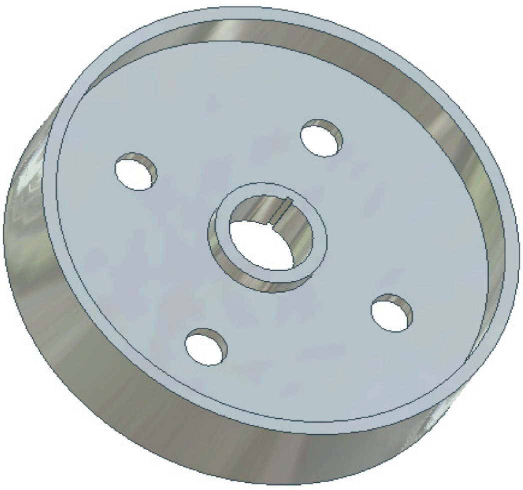
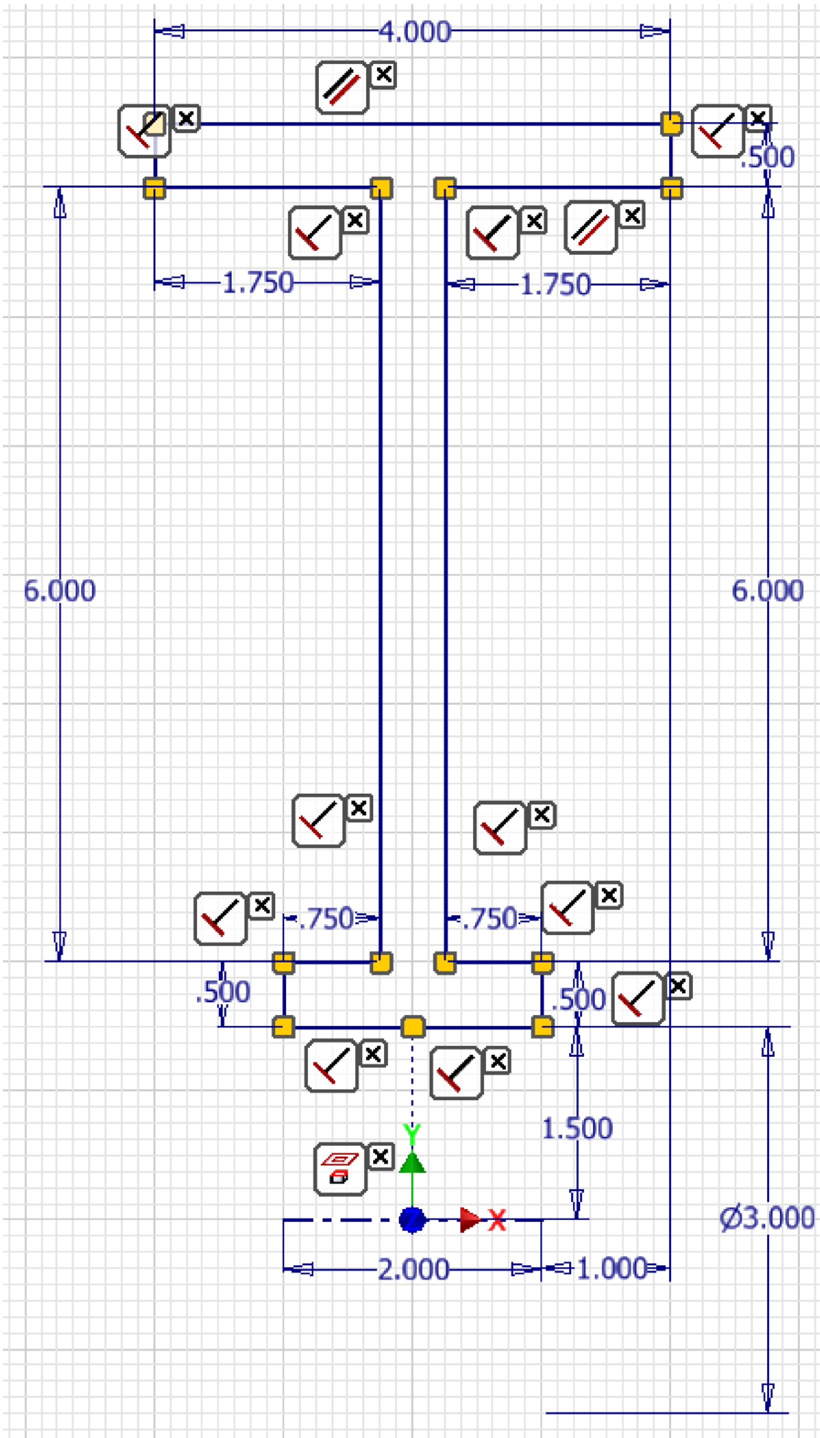
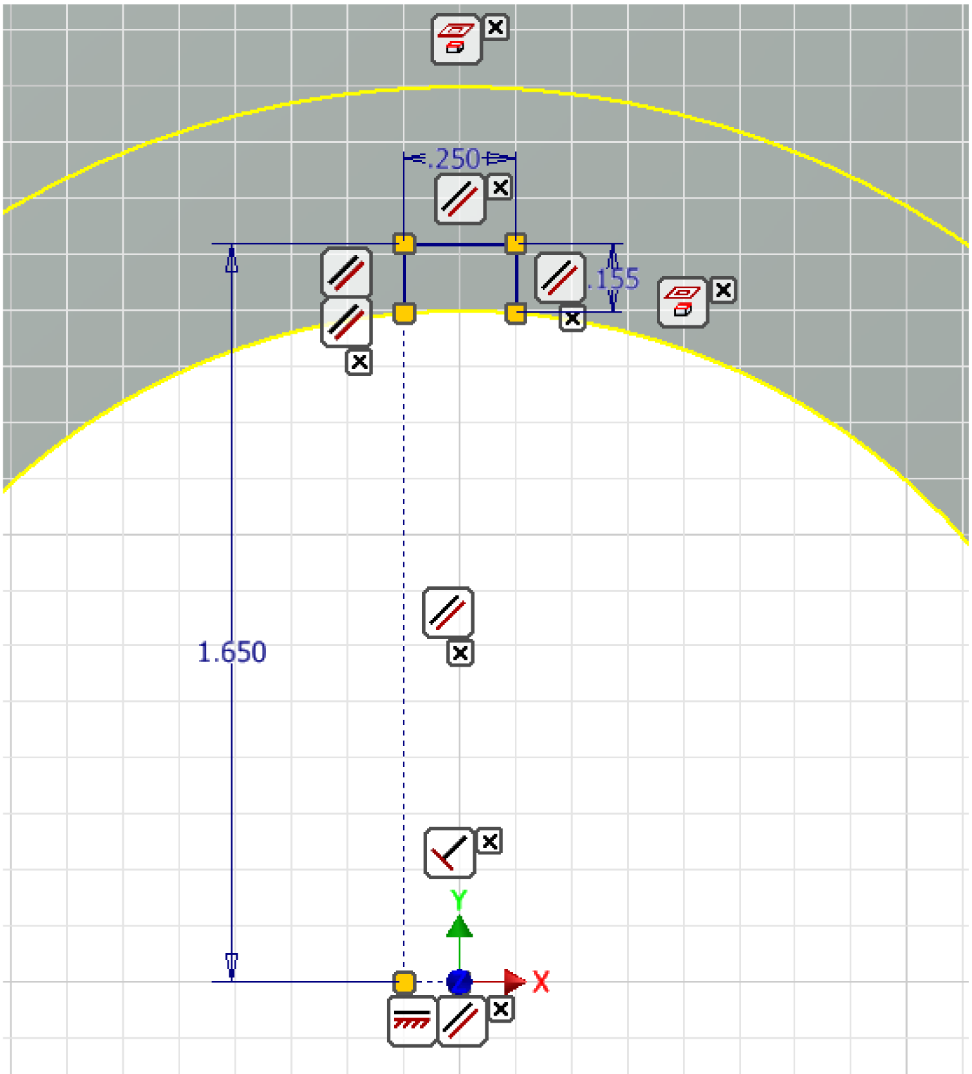


Figure Step 4B
Completed Model
- Rotated View

Author's Geometric Constrains: The following figures shows the sketch's construction method plus geometric and dimensional constraints suggested by the author to help you learn how to construct and constrain sketches. It is only the suggested method and if you can complete a fully constrained sketch using a different construction method and constraints, that is what is important. You may want to compare your construction method and constraints with the authors.

Author's Comments: Your geometrical and dimensional constrains may not match the figure exactly. Just ensure that your sketch is fully constrained.

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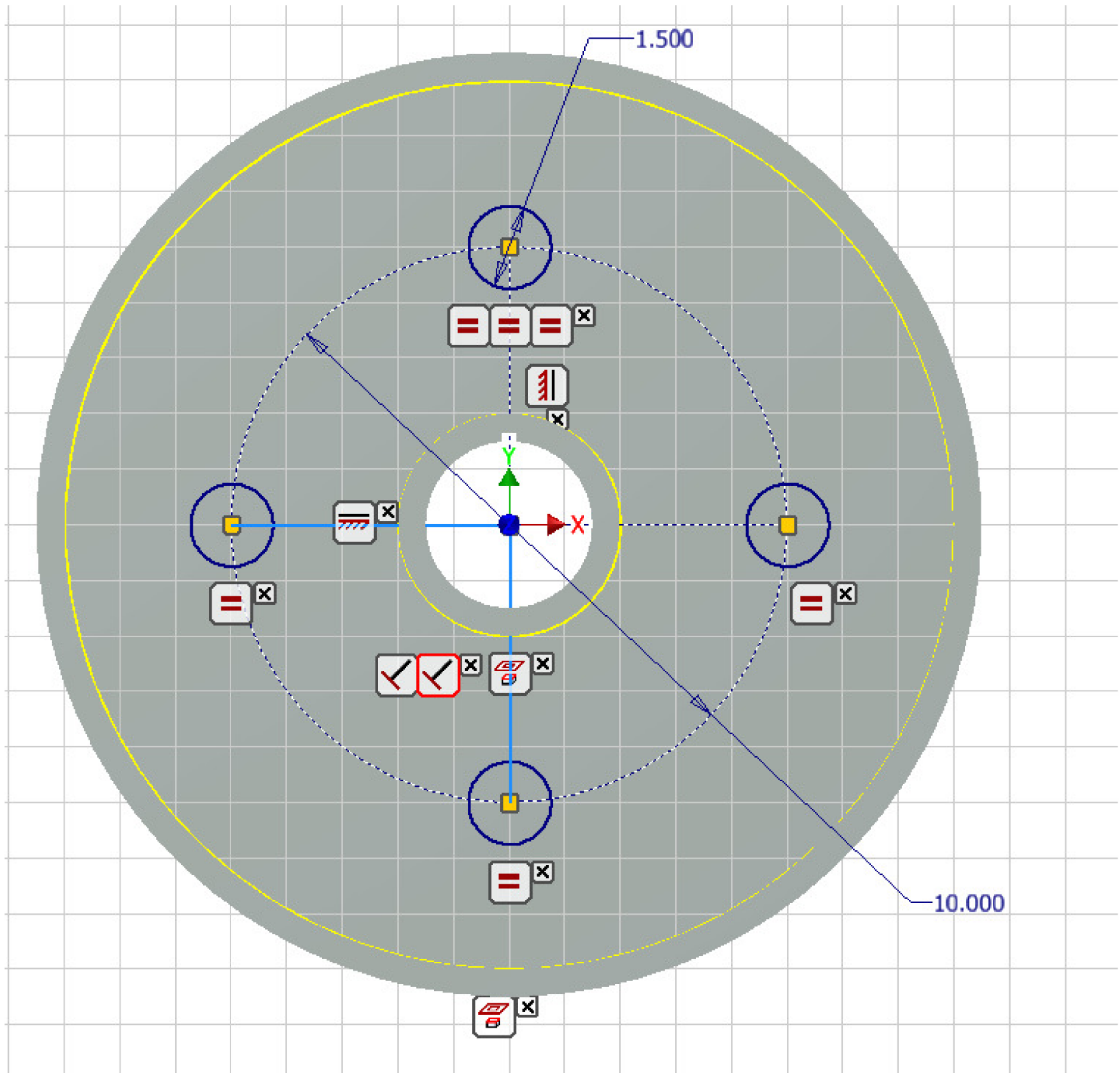
Y
X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10.000