Chapter 2: Part Modeling Basics

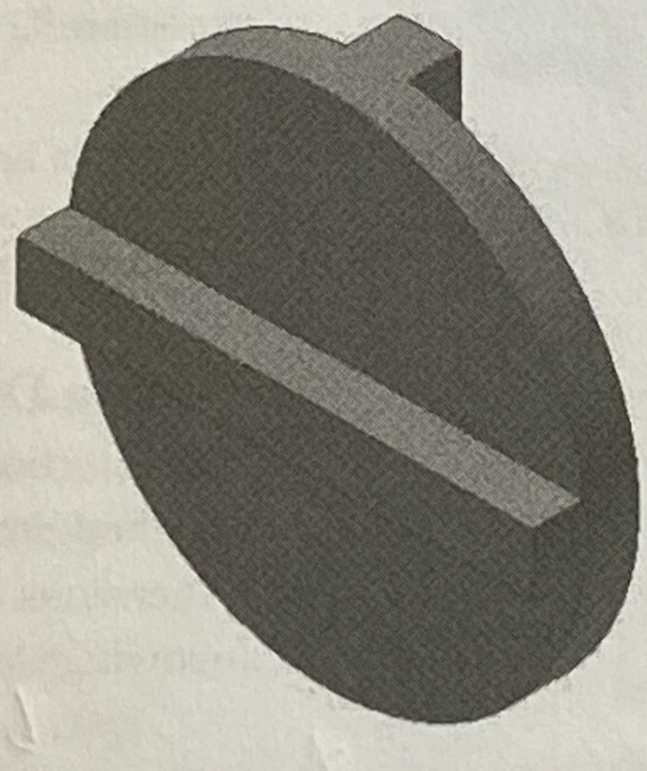
This chapter takes you through the creation of your first Inventor model. You create simple parts:

In this chapter, you will:

- Create Sketches
- Create a base feature
- Add another feature to it
- Create revolved features
- Create cylindrical features
- Create box features
- Apply draft

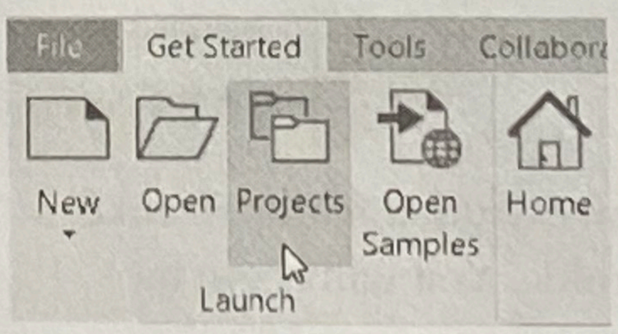
TUTORIAL 1

This tutorial takes you through the creation of your first Inventor model. You will create the Disc of an Oldham coupling:



Creating a New Project

1. Start **Autodesk Inventor 2020** by double-clicking the **Autodesk Inventor 2020** icon on your desktop.
2. To create a new project, click **Get Started > Launch > Projects** on the ribbon.



3. Click the **New** button on the **Projects** dialog.
4. On the **Inventor project wizard** dialog, select **New Single User Project** and click the **Next** button.
5. Enter **Oldham Coupling** in the **Name** field.
6. Enter **C:\Users\Username\Documents\Inventor\Oldham Coupling** in the **Project(Workspace) Folder** box and click **Next**.
7. Click **Finish**.
8. Click **OK** on the **Inventor Project Editor** dialog.

9. Click **Done**.

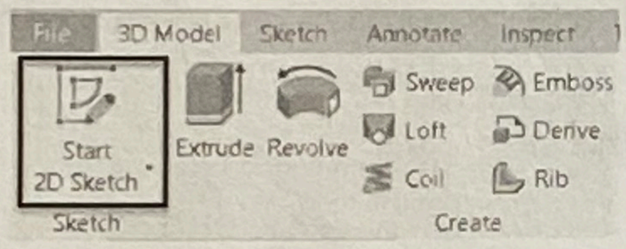
Starting a New Part File

1. To start a new part file, click **Get Started > Launch > New** on the ribbon.
2. On the **Create New File** dialog, click the **Templates** folder located the top right corner.
3. Click the **Standard.ipt** icon located under the **Part – Create 2D and 3D Objects** section.
4. Click the **Create** button on the **Create New File** dialog.

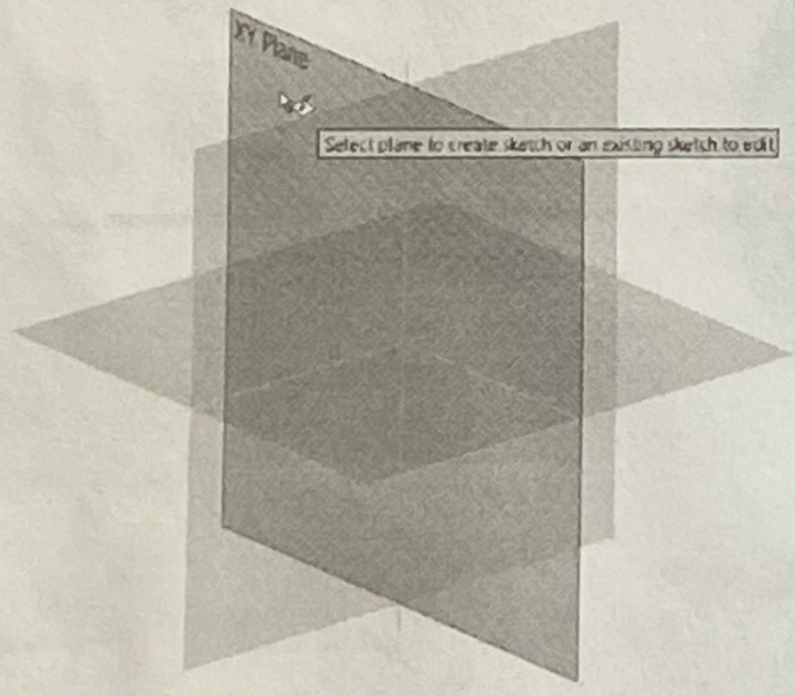
A new model window appears.

Starting a Sketch

1. To start a new sketch, click **3D Model > Sketch > Start 2D Sketch** on the ribbon.

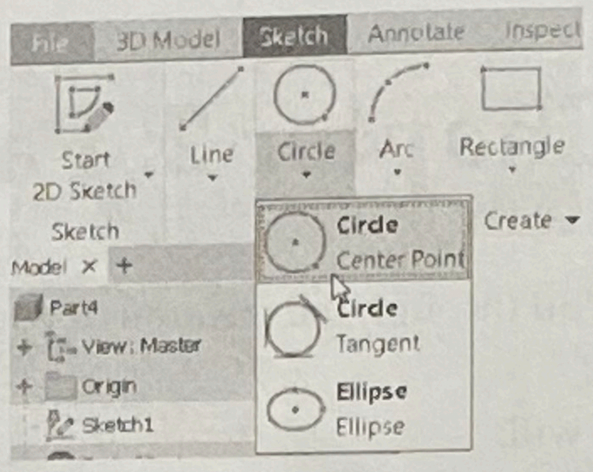


2. Click on the **XY Plane**. The sketch starts.

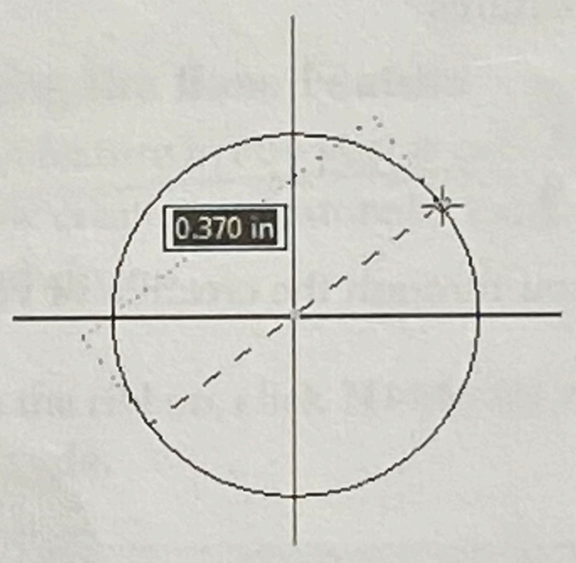


The first feature is an extruded feature from a sketched circular profile. You will begin by sketching the circle.

3. On the ribbon click **Sketch > Create > Circle > Circle Center Point**.



4. Move the cursor to the sketch origin located at the center of the graphics window, and then click on it.
5. Drag the cursor up to a random location, and then click to create a circle.



6. Press **ESC** to deactivate the tool.

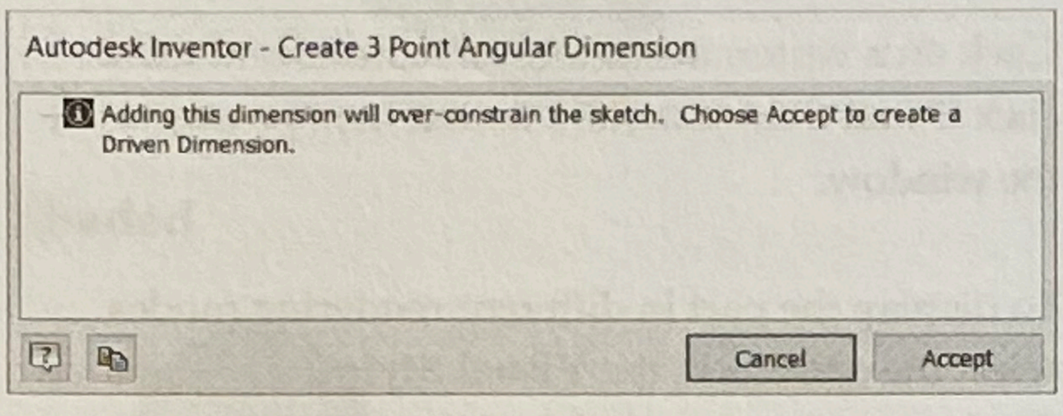
Adding Dimensions

In this section, you will specify the size of the sketched circle by adding dimensions. As you add dimensions, the sketch can attain any one of the following states:

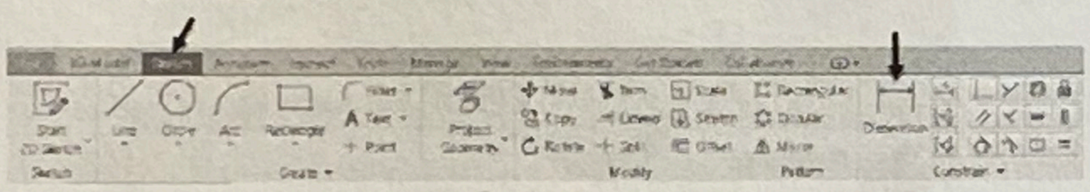
Fully Constrained sketch: In a fully constrained sketch, the positions of all the entities are fully described by dimensions, constraints, or both. In a fully constrained sketch, all the entities are a dark blue color.

Under Constrained sketch: Additional dimensions, constraints, or both are needed to specify the geometry completely. In this state, you can drag under constrained sketch entities to modify the sketch. An under constrained sketch entity is in black color.

If you add any more dimensions to a fully constrained sketch, a message box will appear showing that dimension over constraints the sketch. In addition, it prompts you to convert the dimension into a driven dimension. Click **Accept** to convert the unwanted dimension into a driven dimension.



1. On the ribbon, click **Sketch > Constrain > Dimension**.



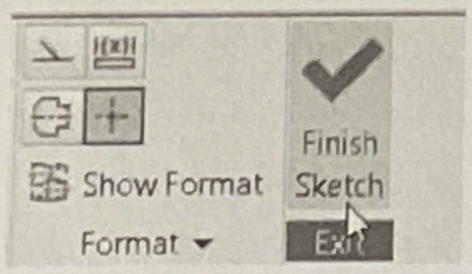
2. Select the circle and click; the **Edit Dimension** box appears.
3. Enter **4** in the **Edit Dimension** box and click the green check.
4. Press **Esc** to deactivate the **Dimension** tool.

You can also create dimensions while creating sketch objects. To do this, enter the dimension values in the boxes displayed while sketching.

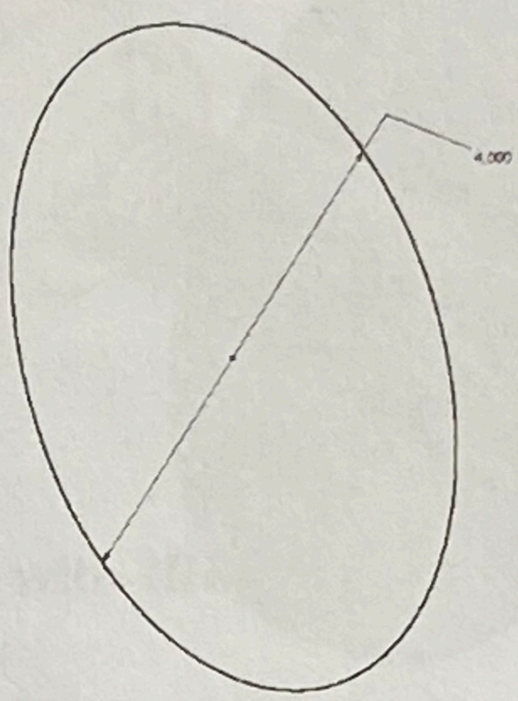
5. To display the entire circle at full size and to center it in the graphics area, use one of the following methods:

   - Click **Zoom All** on the **Navigate Bar**.
   - Click **View > Navigate > Zoom All** on the ribbon.

6. Click **Finish Sketch** on the **Exit** panel.



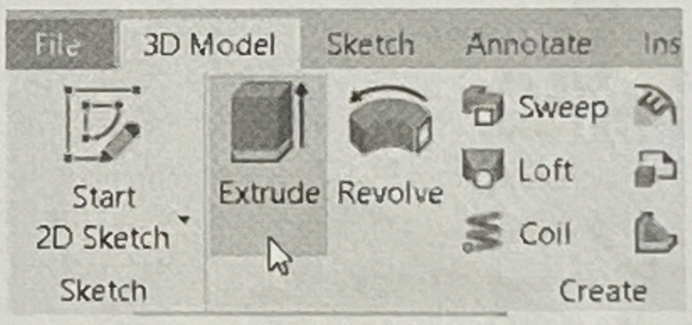
7. Again, click **Zoom All** on the **Navigate Bar.**



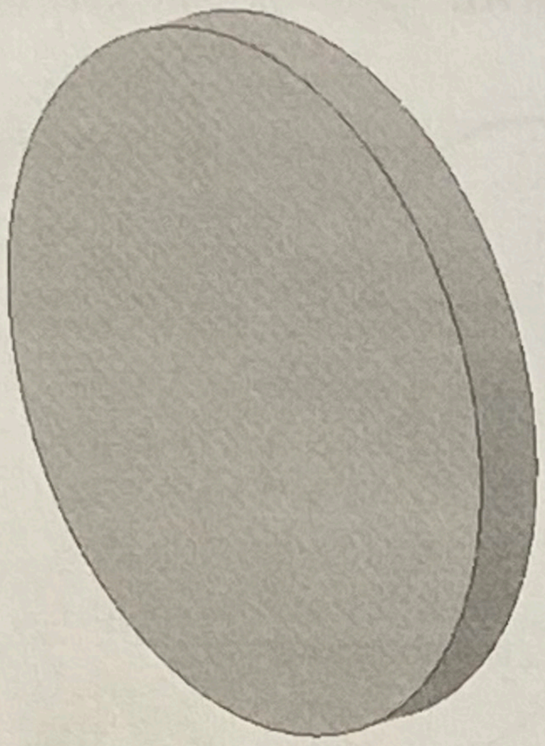
Creating the Base Feature

The first feature in any part is called a base feature. You now create this feature by extruding the sketched circle.

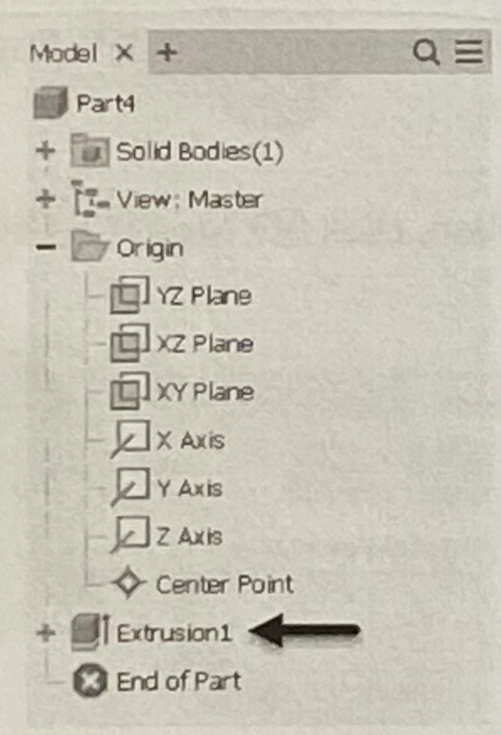
1. On the ribbon, click **3D Model > Create > Extrude.**



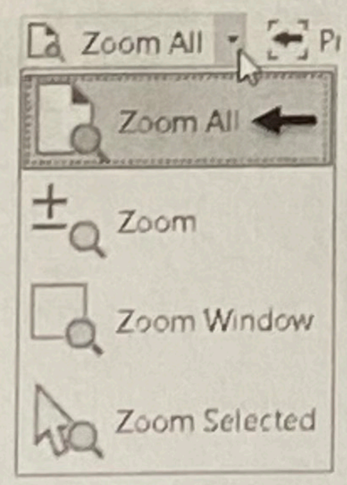
2. Type-in 0.4 in the **Distance A** box available on the **Extrude** Properties panel.
3. Click the **Direction > Default** icon under the **Behavior** section on the **Extrude** Properties panel.
4. Click **OK** on the **Extrude** Properties panel to create the extrusion.



Notice the new feature, **Extrusion 1**, in the **Browser window**.



To magnify a model in the graphics area, you can use the zoom tools available on the **Zoom** drop-down in the **Navigate** panel of the **View** tab.



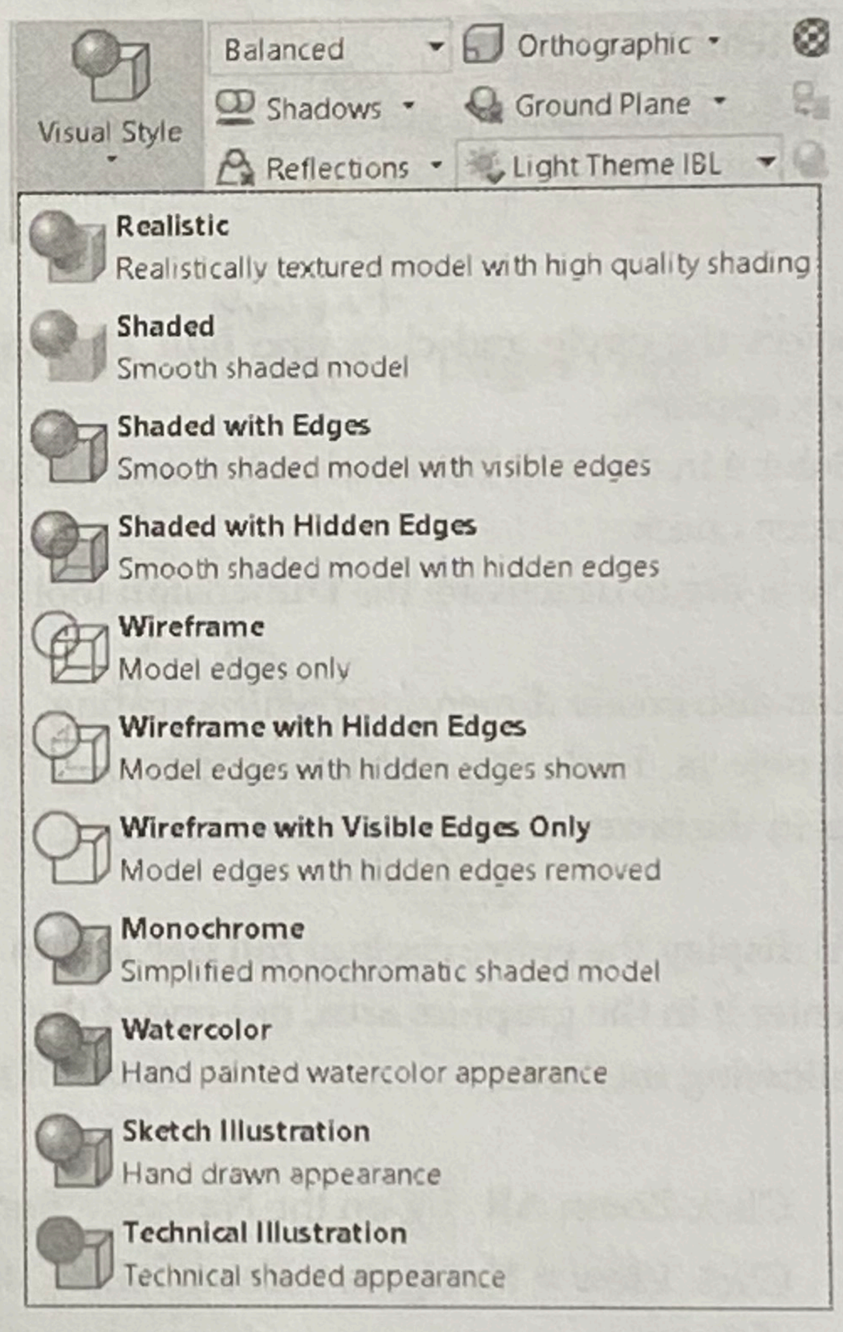
Click **Zoom All** to display the part full size in the current window.

Click **Zoom Window**, and then drag the pointer to create a rectangle; the area in the rectangle zooms to fill the window.

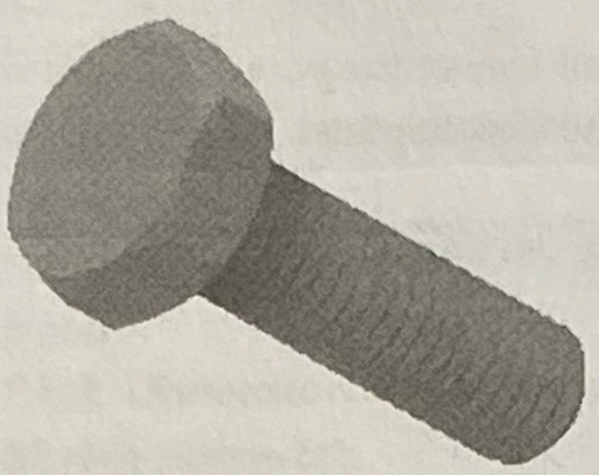
Click **Zoom**, and then drag the pointer. Dragging up zooms out; dragging down zooms in.

Click on a vertex, an edge, or a feature, and then click **Zoom Selected**; the selected item zooms to fill the window.

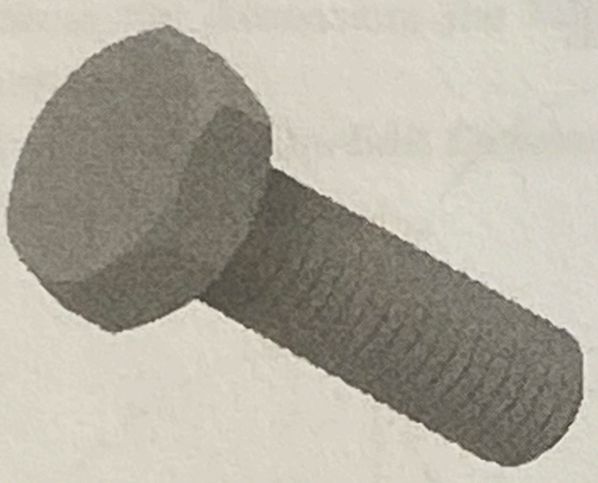
To display the part in different rendering modes, select the options in the **Visual Style** drop-down on the **Appearance** panel of the **View** tab.



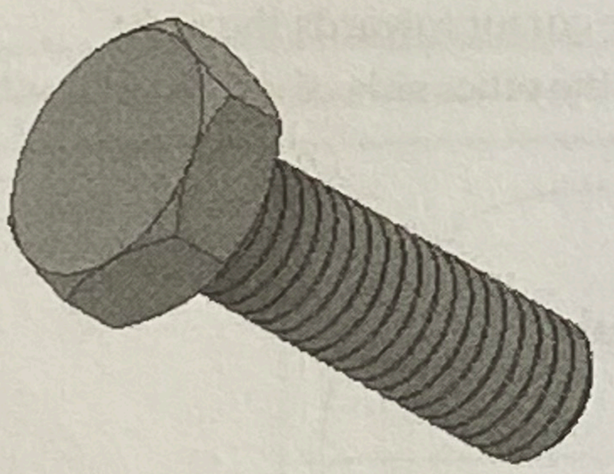
Realistic



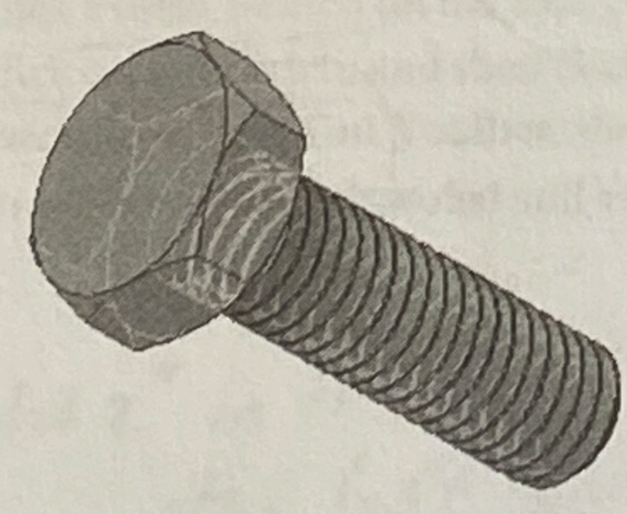
Shaded



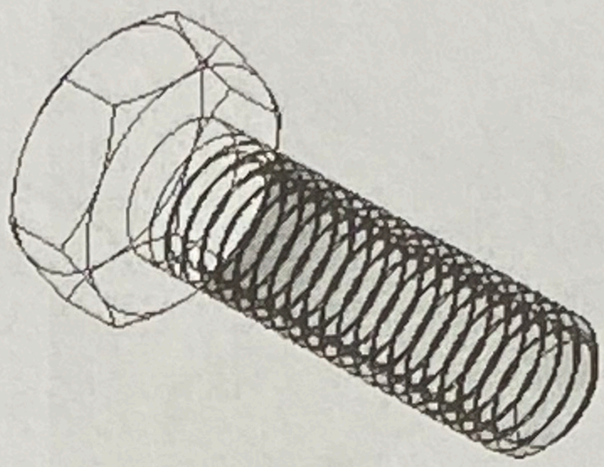
Shaded with Edges



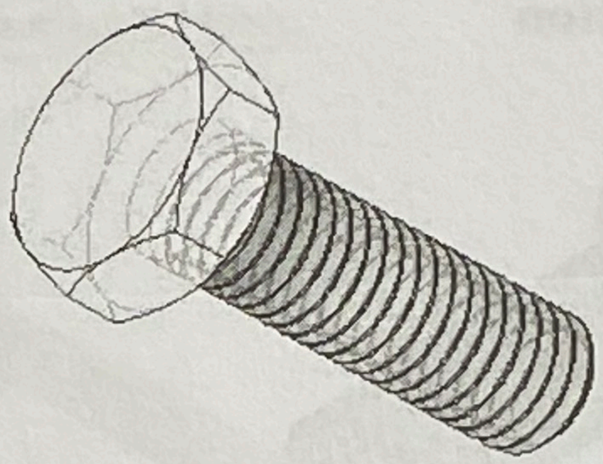
Shaded with Hidden Edges



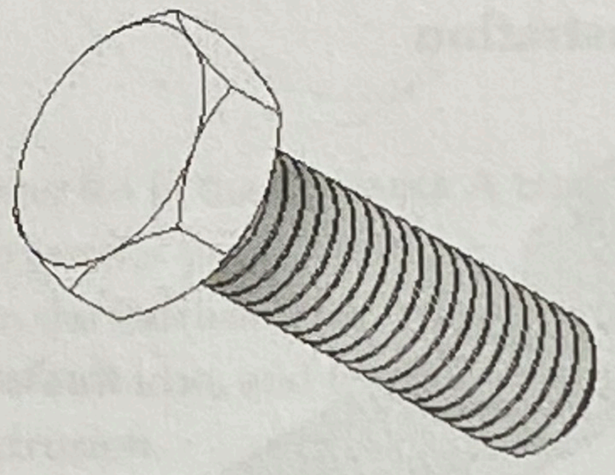
Wireframe



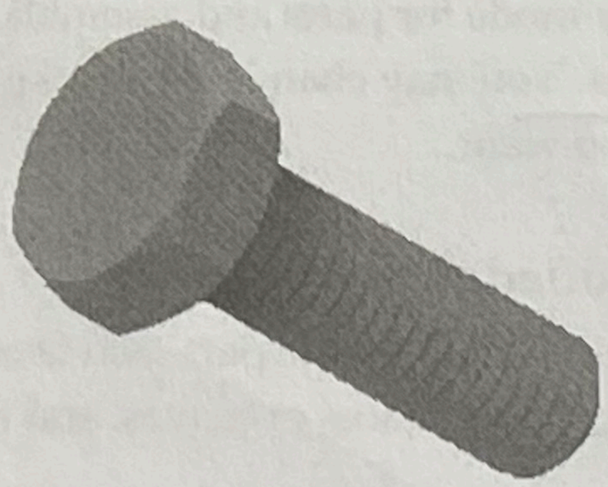
Wireframe with Hidden Edges



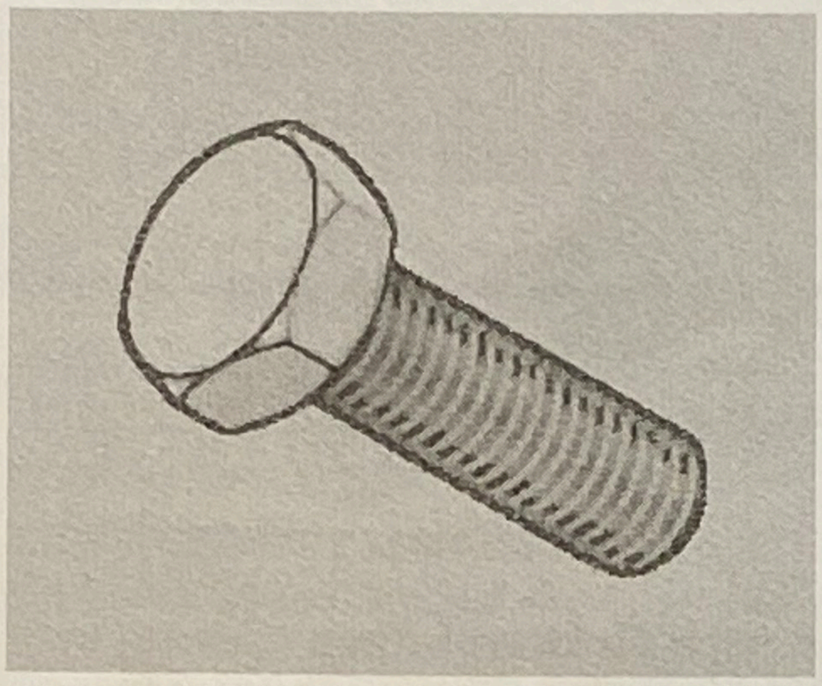
Wireframe with Visible Edges Only



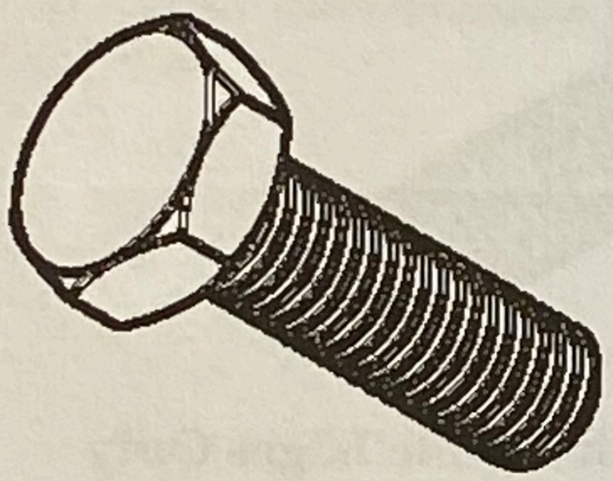
Monochrome



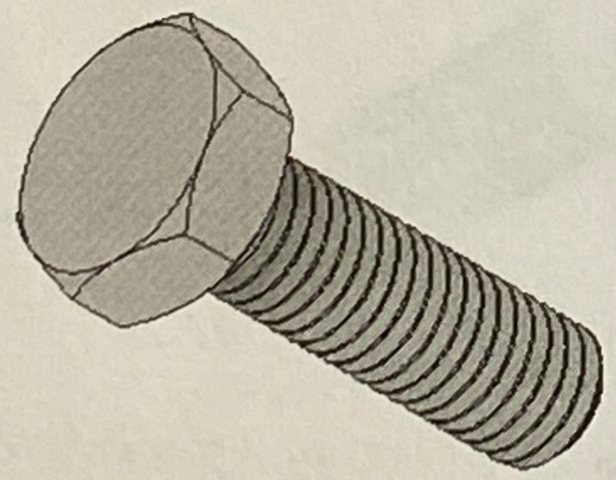
Watercolor



Sketch Illustration



Technical Illustration

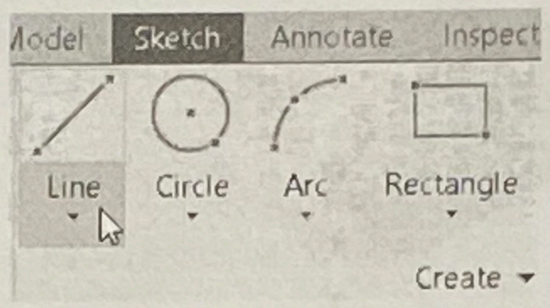


The default display mode for parts and assemblies is **Shaded with Edges**. You may change the display mode whenever you want.

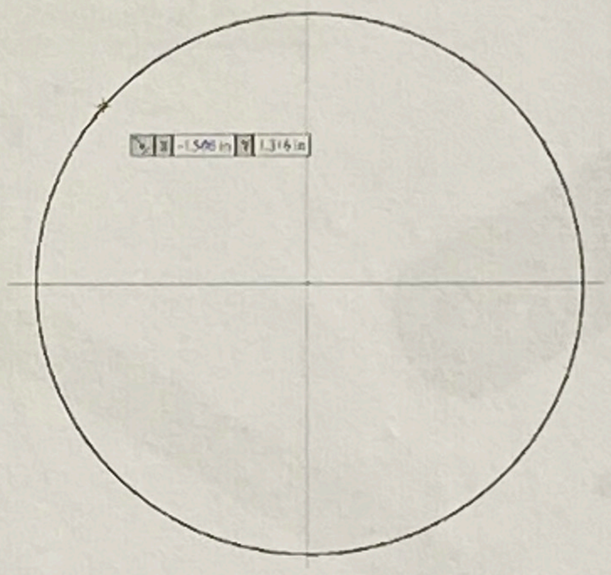
Adding an Extruded Feature

To create additional features on the part, you need to draw sketches on the model faces or planes, and then extrude them.

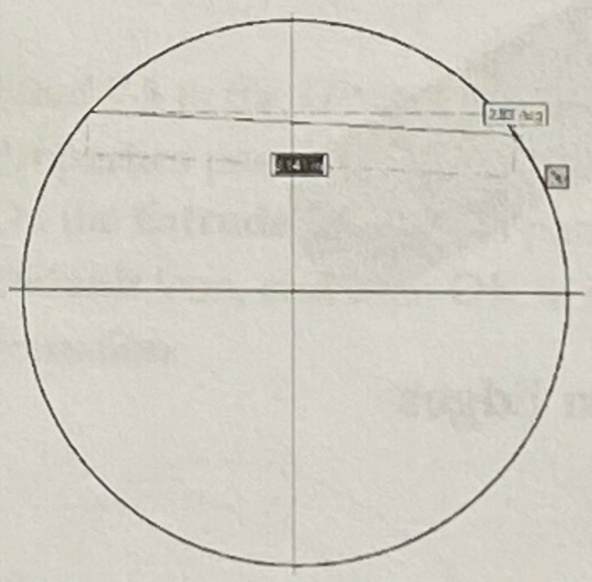
1. On the ribbon, click **View > Appearance > Visual Style > Wireframe**.
2. On the ribbon, click **3D Model > Sketch > Start 2D Sketch**.
3. Click on the front face of the part.
4. Click **Line** on the **Create** panel.



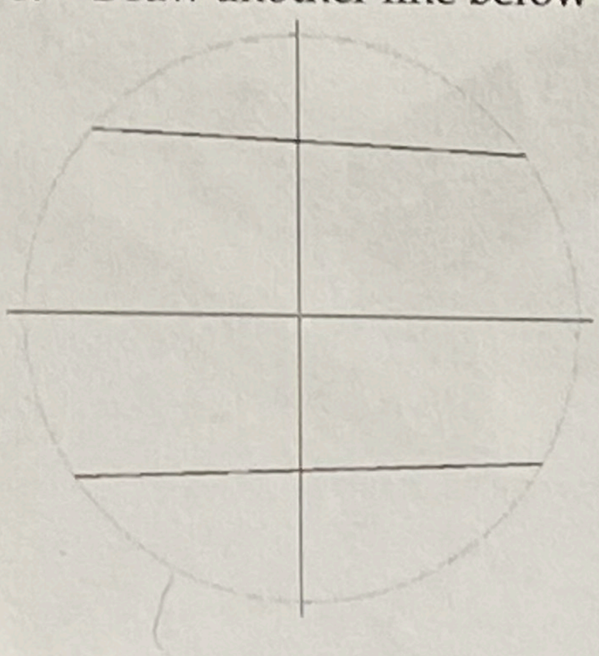
5. Click on the circular edge to specify the first point of the line.



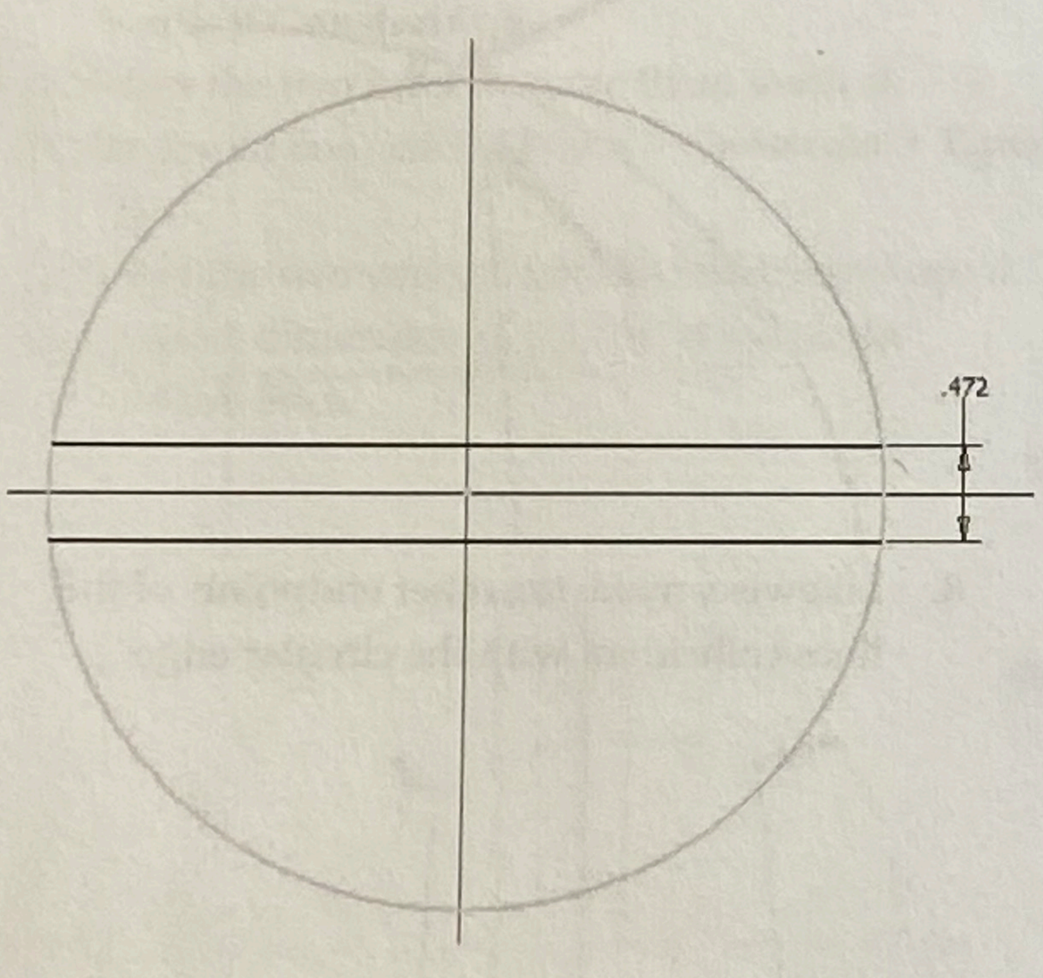
6. Move the cursor towards the right.
7. Click on the other side of the circular edge; a line is drawn.



8. Draw another line below the previous line.

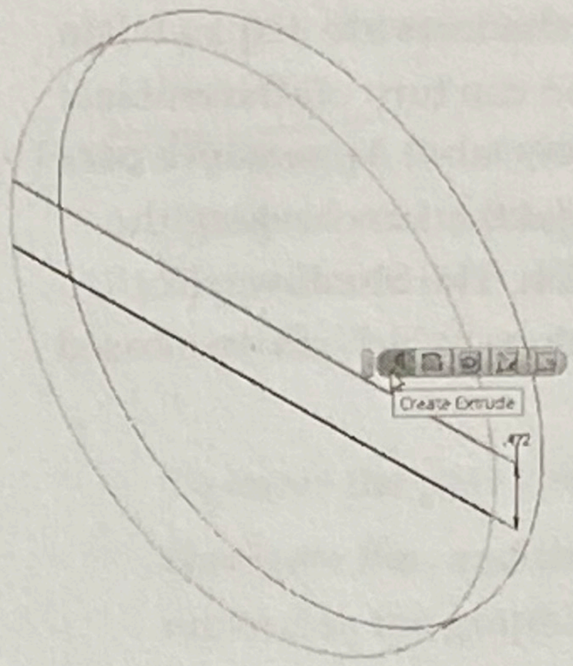


9. On the ribbon, click **Sketch > Constrain > Horizontal Constraint** .
10. Select the two lines to make them horizontal.
11. On the ribbon, click **Sketch > Constrain > Equal** .
12. Select the two horizontal lines to make them equal.
13. Click **Dimension** on the **Constrain** panel of the **Sketch** ribbon tab.
14. Select the two horizontal lines.
15. Move the cursor toward the right and click to locate the dimension; the **Edit Dimension** box appears.
16. Enter **0.472** in the **Edit Dimension** box and click the green check.

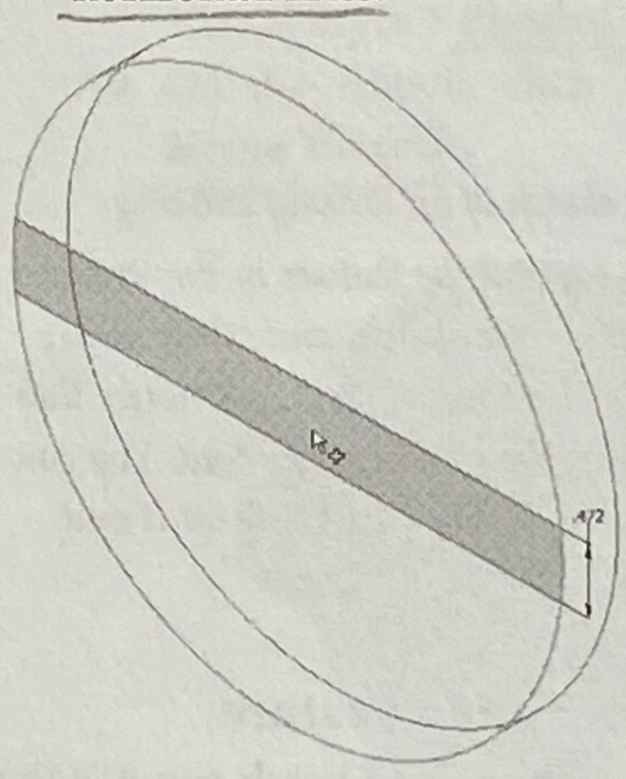


17. Click **Finish Sketch** on the **Exit** panel.
18. Click on the sketch, and then click **Create Extrude** on the **Mini Toolbar**; the **Extrude** Properties panel appears.

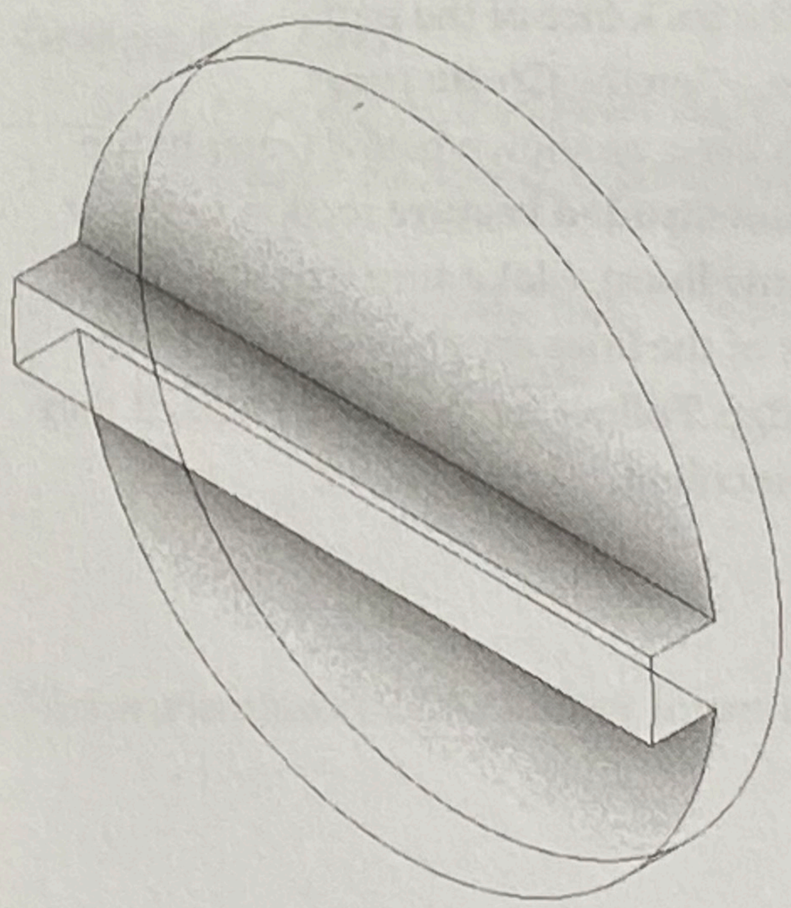
"Sketch 2" in the "model tree" along the left side of screen



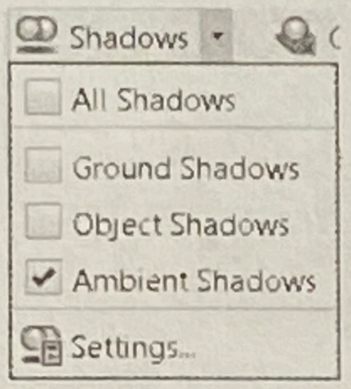
19. Click on the region bounded by the two horizontal lines.



20. Enter **0.4** in the **Distance A** box on the **Extrude** Properties panel.
21. On the **Extrude** Properties panel, click the **Default** icon, and then **OK** to create the extrusion.



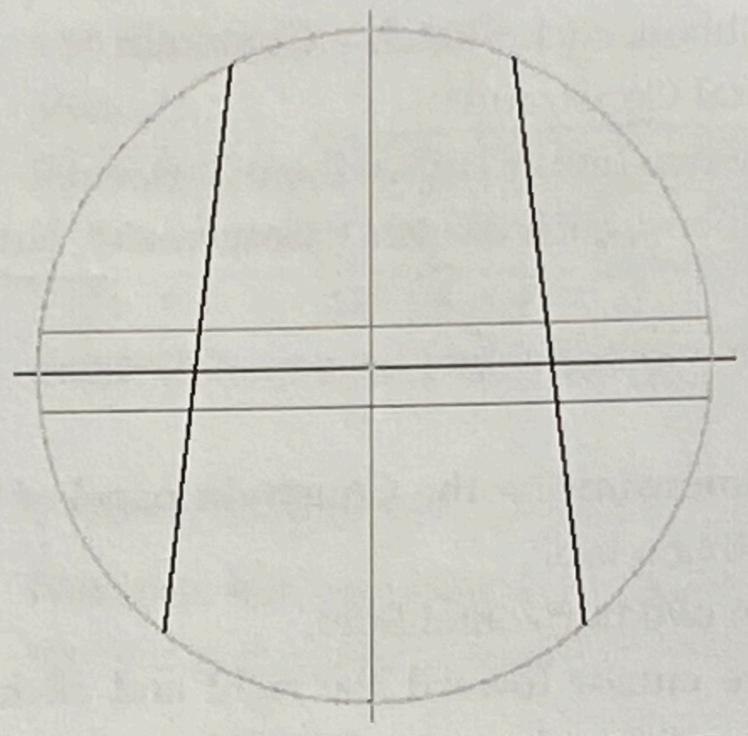
By default, the ambient shadows are displayed on the model. However, you can turn off the ambient shadows by clicking **View** tab > **Appearance** panel > **Shadows** drop-down, and then unchecking the **Ambient Shadows** option. The **Shadows** drop-down has two more options, which you use based on your requirement.



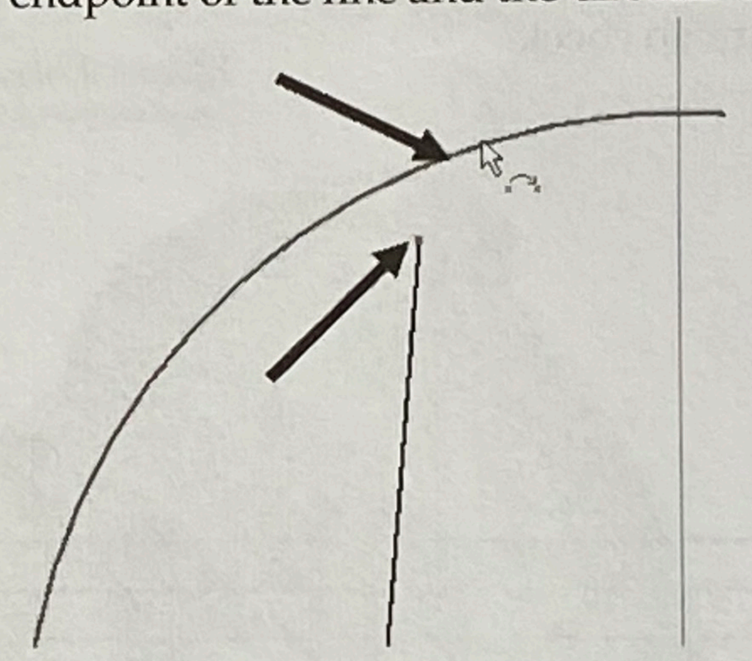
You can reuse the sketch of an already existing feature. To do this, expand the feature in the Browser Window, right click on the sketch, and select ***Share Sketch*** *from the shortcut menu. You will notice that the sketch is visible in the graphics window. You can also unshare the sketch by right-clicking on it and selecting* ***Unshare****.*

Adding another Extruded Feature

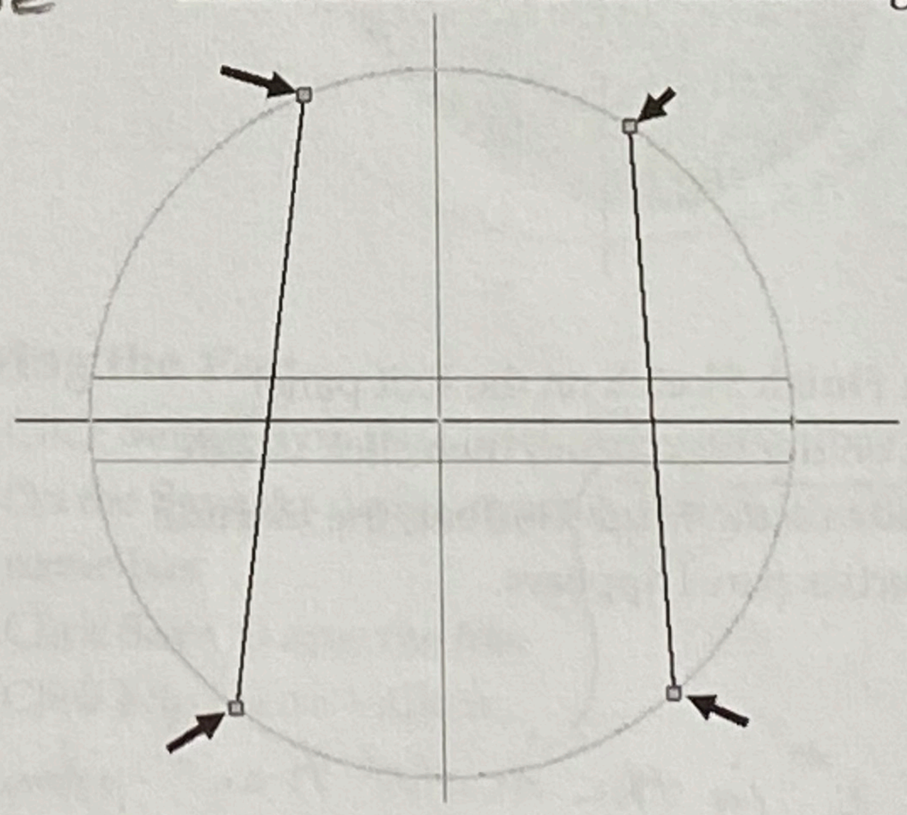
1. Click **Start 2D Sketch** on the **Sketch** panel of the **3D Model** ribbon tab. Just spin the view cube
2. Use the **Free Orbit** button from the **Navigate Bar** to rotate the model such that the back face of the part is visible.
3. Right click and select **OK**.
4. Click on the back face of the part.
5. Click **Line** on the **Create** panel.
6. Draw two lines, as shown below (refer to the **Adding an Extruded Feature** section to know how to draw lines). Make sure that the endpoints of the lines are coincident with the circular edge. Follow the next two steps, if they are not coincident.



7. On the ribbon, click **Sketch > Constrain > Coincident Constraint**. Next, select the endpoint of the line and the circular edge.



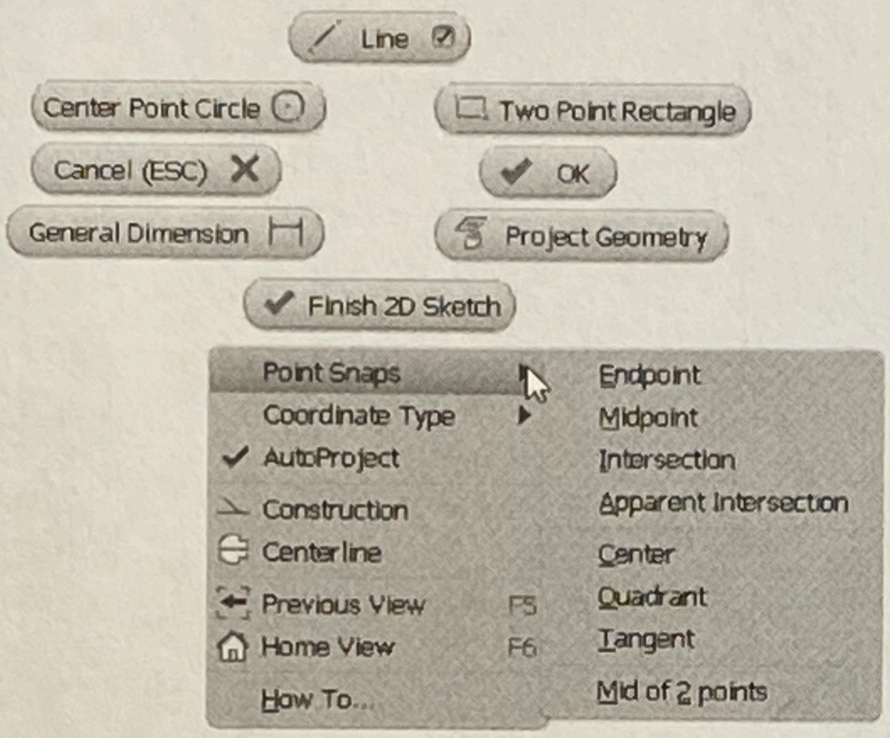
8. Likewise, make the other endpoints of the lines coincident with the circular edge.



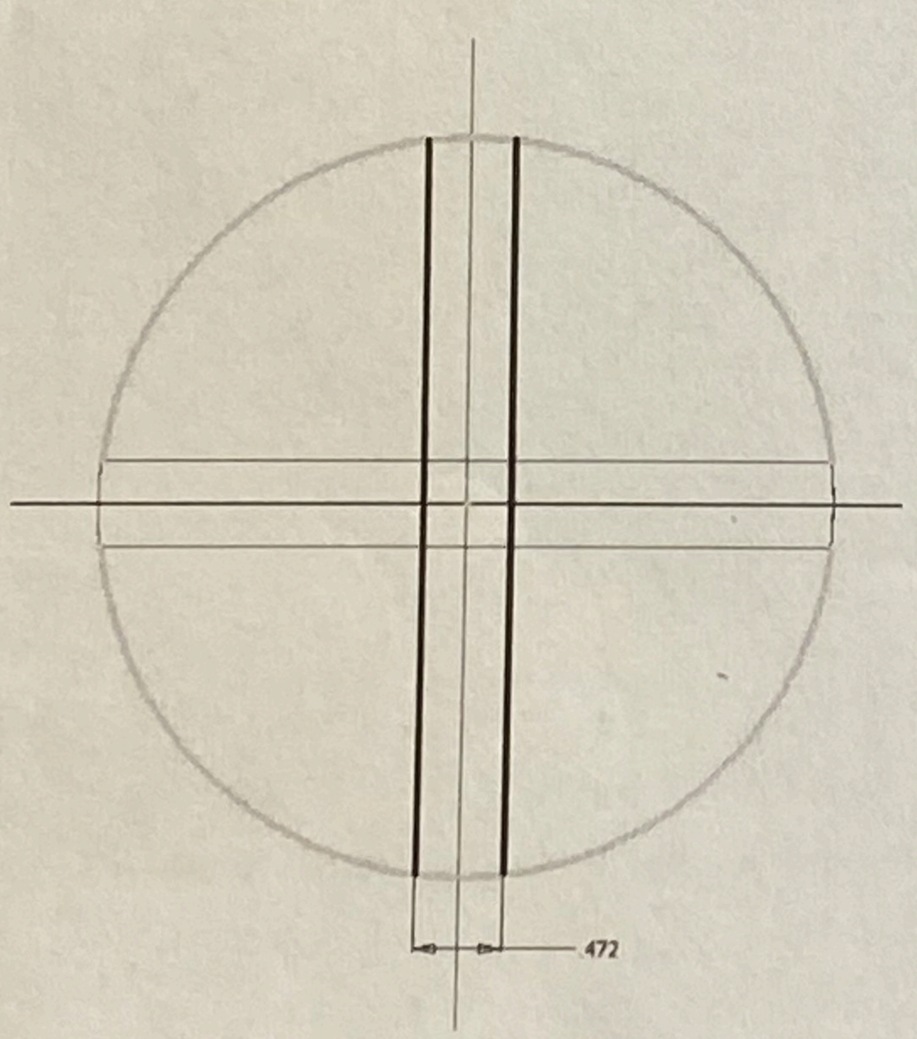
Skip the above two steps if the endpoints of the lines are coincident with the circular edge.

You can specify a point using various point snap options. To do this, activate a sketching tool, right click, and select ***Point Snaps****; a list of point snaps*

appears. Now, you can select only the specified point snap.



9. On the ribbon, click **Sketch > Constrain > Vertical Constraint**.
10. Select the two lines to make them vertical.
11. On the ribbon, click **Sketch > Constrain > Equal** =.
12. Select the two vertical lines to make them equal.
13. Create a dimension of 0.472in between the vertical lines.

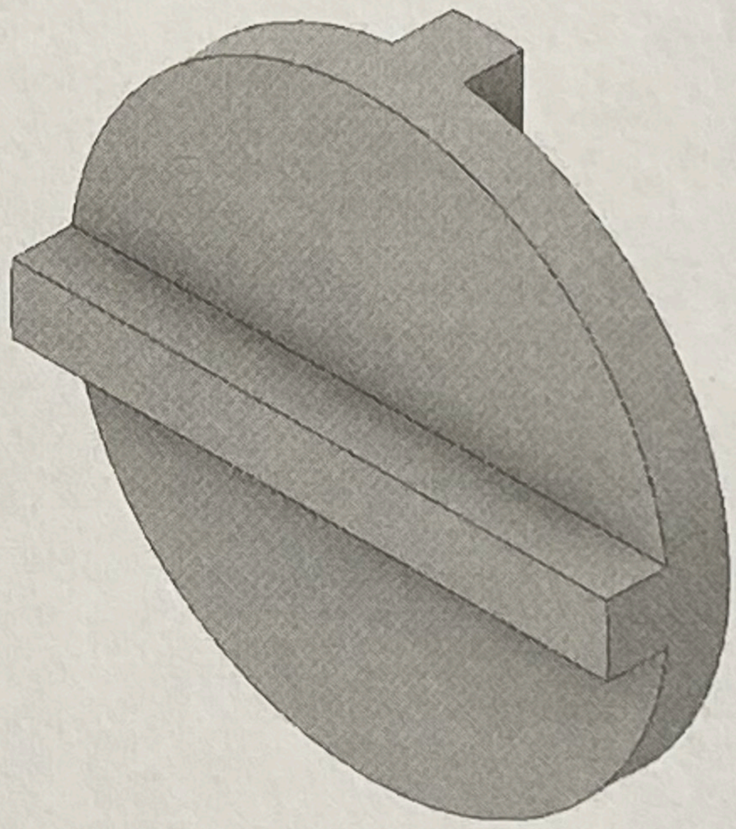


14. Click **Finish Sketch**.

15. On the ribbon, click **3D Model > Create > Extrude**.
16. Click inside the region enclosed by two lines, if they are not already selected.
17. Type 0.4 in the **Distance A** box on the **Extrude** Properties panel and click **OK**.

To move the part view, click **Pan** on **Navigate Bar**, and then drag the part to move it in the graphics area.

18. On the ribbon, click **View > Appearance > Visual Style > Shaded with Edges**.
19. On the ribbon, click **View > Navigate > Home View**.



Saving the Part

1. Click **Save** on the **Quick Access Toolbar**.
2. On the **Save As** dialog, type-in **Disc** in the **File name** box.
3. Click **Save** to save the file.
4. Click **File Menu > Close**.

Note:

*.ipt is the file extension for all the files that you create in the Part environment of Autodesk Inventor.