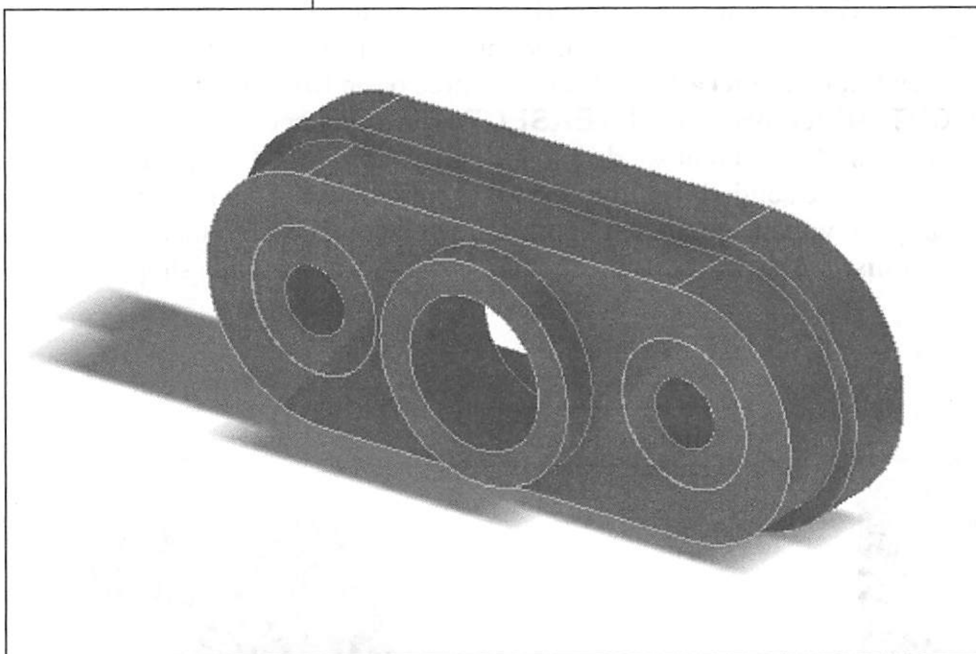
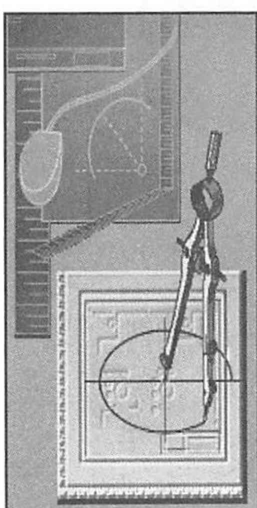
Chapter 3
# CSG Concepts and Model History Tree



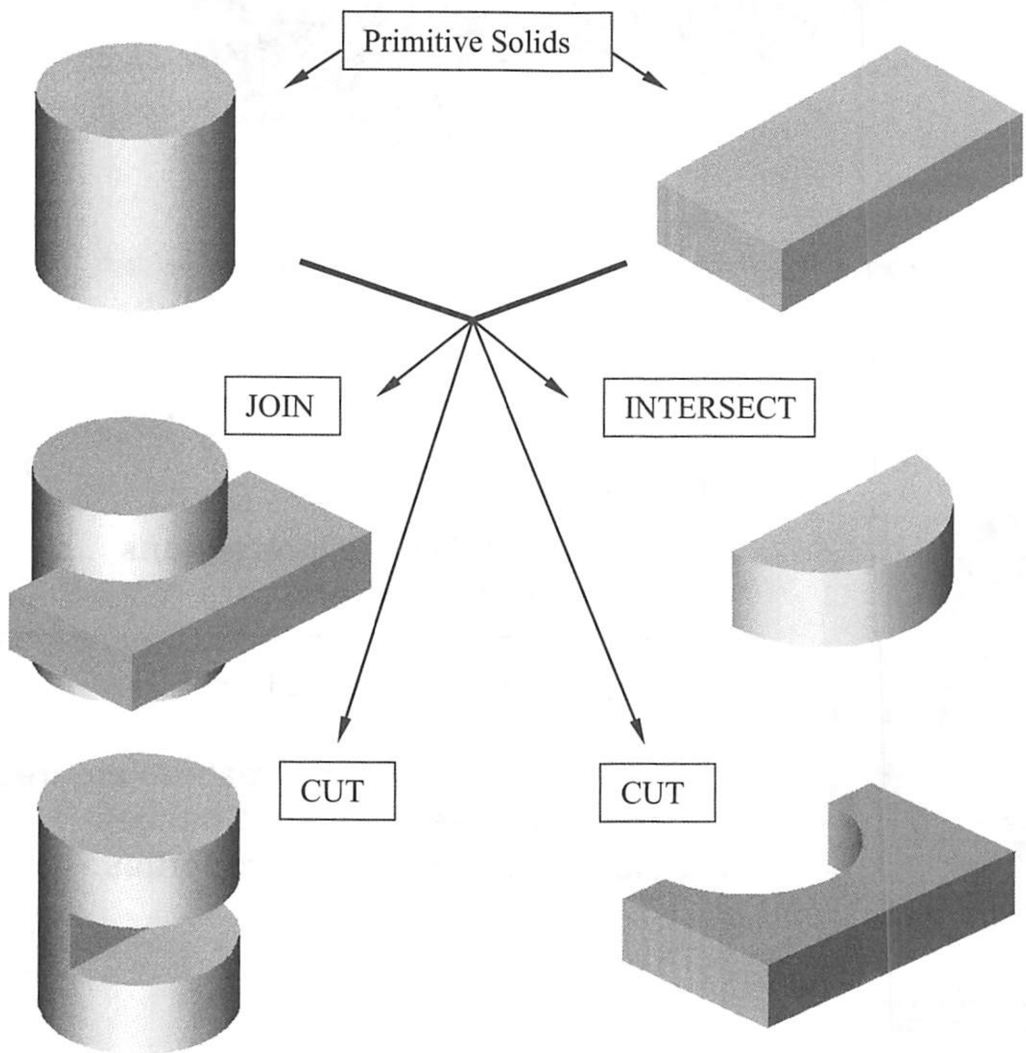


## Learning Objectives

- Understand Feature Interactions
- Use the Part Browser
- Modify and Update Feature Dimensions
- Perform History-Based Part Modifications
- Change the Names of Created Features
- Implement Basic Design Changes
- Calculate the Physical Properties

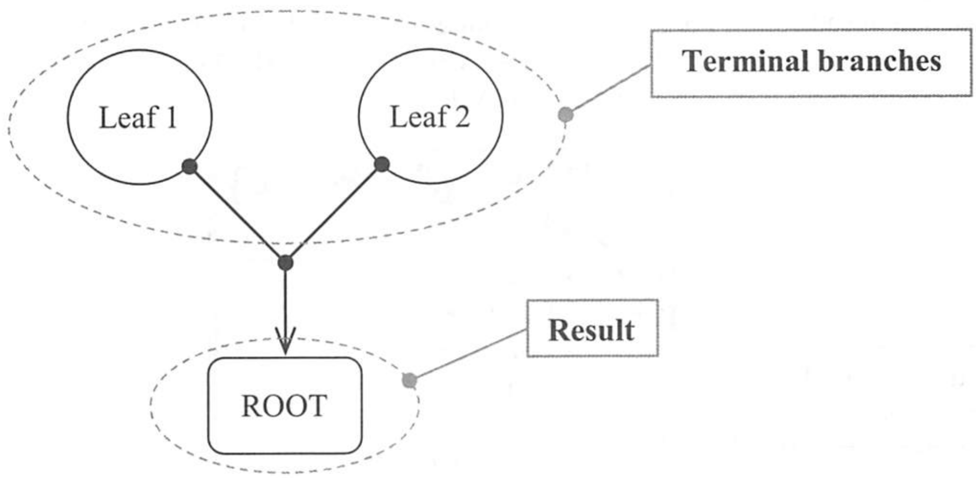
## Introduction

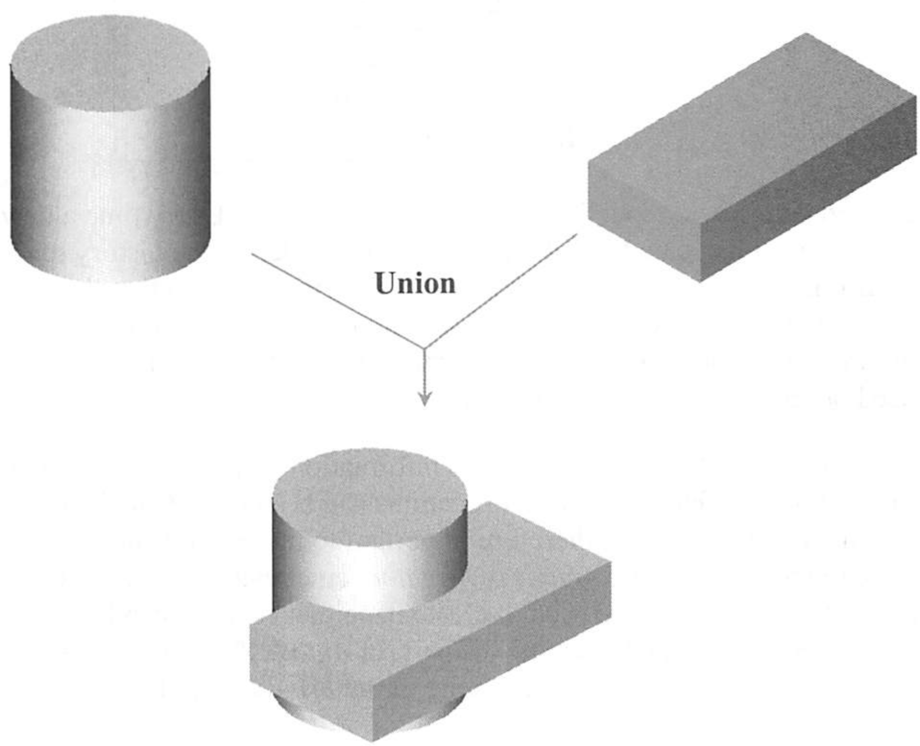
In the 1980s, one of the main advancements in **solid modeling** was the development of the **Constructive Solid Geometry** (CSG) method. CSG describes the solid model as combinations of basic three-dimensional shapes (**primitive solids**). The basic primitive solid set typically includes: Rectangular-prism (Block), Cylinder, Cone, Sphere, and Torus (Tube). Two solid objects can be combined into one object in various ways using operations known as **Boolean operations**. There are three basic Boolean operations: **JOIN (Union)**, **CUT (Difference)**, and **INTERSECT**. The *JOIN* operation combines the two volumes included in the different solids into a single solid. The *CUT* operation subtracts the volume of one solid object from the other solid object. The *INTERSECT* operation keeps only the volume common to both solid objects. The CSG method is also known as the **Machinist's Approach**, as the method is parallel to machine shop practices.



## Binary Tree

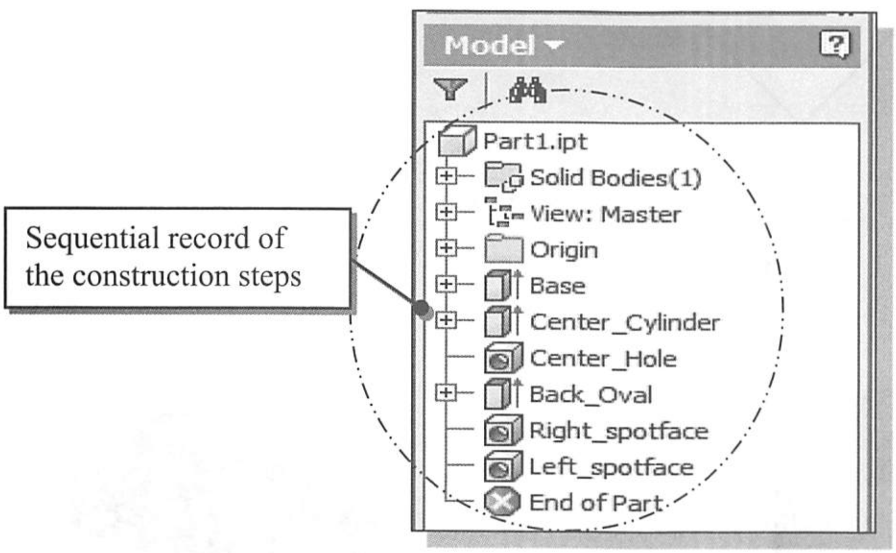
The CSG is also referred to as the method used to store a solid model in the database. The resulting solid can be easily represented by what is called a **binary tree**. In a binary tree, the terminal branches (leaves) are the various primitives that are linked together to make the final solid object (the root). The binary tree is an effective way to keep track of the *history* of the resulting solid. By keeping track of the history, the solid model can be re-built by re-linking through the binary tree. This provides a convenient way to modify the model. We can make modifications at the appropriate links in the binary tree and re-link the rest of the history tree without building a new model.



## Model History Tree

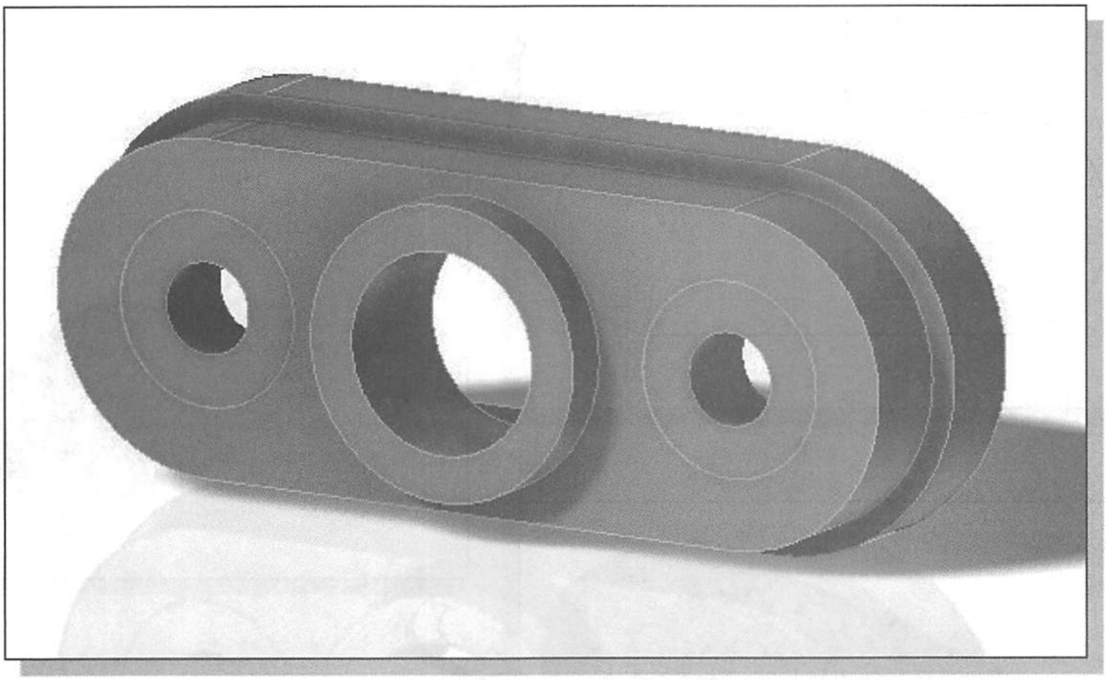
In *Autodesk Inventor*, the **design intents** are embedded into features in the **history tree**. The structure of the model history tree resembles that of a **CSG binary tree**. A CSG binary tree contains only *Boolean relations*, while the ***Autodesk Inventor* history tree** contains all features, including *Boolean relations*. A history tree is a sequential record of the features used to create the part. This history tree contains the construction steps, plus the rules defining the design intent of each construction operation. In a history tree, each time a new modeling event is created previously defined features can be used to define information such as size, location, and orientation. It is therefore important to think about your modeling strategy before you start creating anything. It is important, but also difficult, to plan ahead for all possible design changes that might occur. This approach in modeling is a major difference of **FEATURE-BASED CAD SOFTWARE**, such as *Autodesk Inventor*, from previous generation CAD systems.




Feature-based parametric modeling is a cumulative process. Every time a new feature is added, a new result is created and the feature is also added to the history tree. The database also includes parameters of features that were used to define them. All of this happens automatically as features are created and manipulated. At this point, it is important to understand that all of this information is retained, and modifications are done based on the same input information.

In *Autodesk Inventor*, the history tree gives information about modeling order and other information about the feature. Part modifications can be accomplished by accessing the features in the history tree. It is therefore important to understand and utilize the feature history tree to modify designs. *Autodesk Inventor* remembers the history of a part, including all the rules that were used to create it, so that changes can be made to any operation that was performed to create the part. In *Autodesk Inventor*, to modify a feature we access the feature by selecting the feature in the ***browser*** window.

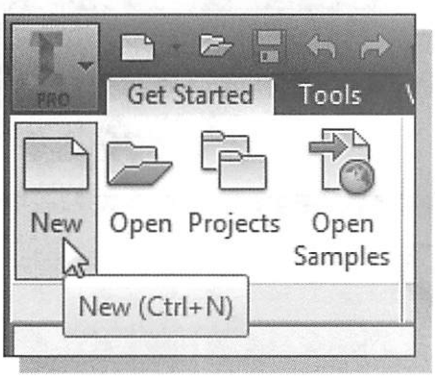
## The A6-Knee Part



❖ Based on your knowledge of *Autodesk Inventor* so far, how many features would you use to create the design? Which feature would you choose as the **BASE FEATURE**, the first solid feature, of the model? What is your choice in arranging the order of the features? Take a few minutes to consider these questions and do preliminary planning by sketching on a piece of paper. You are also encouraged to create the model on your own prior to following through the tutorial.

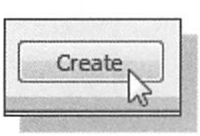
## Starting Autodesk Inventor

1. Select the **Autodesk Inventor** option on the *Start* menu or select the **Autodesk Inventor** icon on the desktop to start *Autodesk Inventor*. The *Autodesk Inventor* main window will appear on the screen.



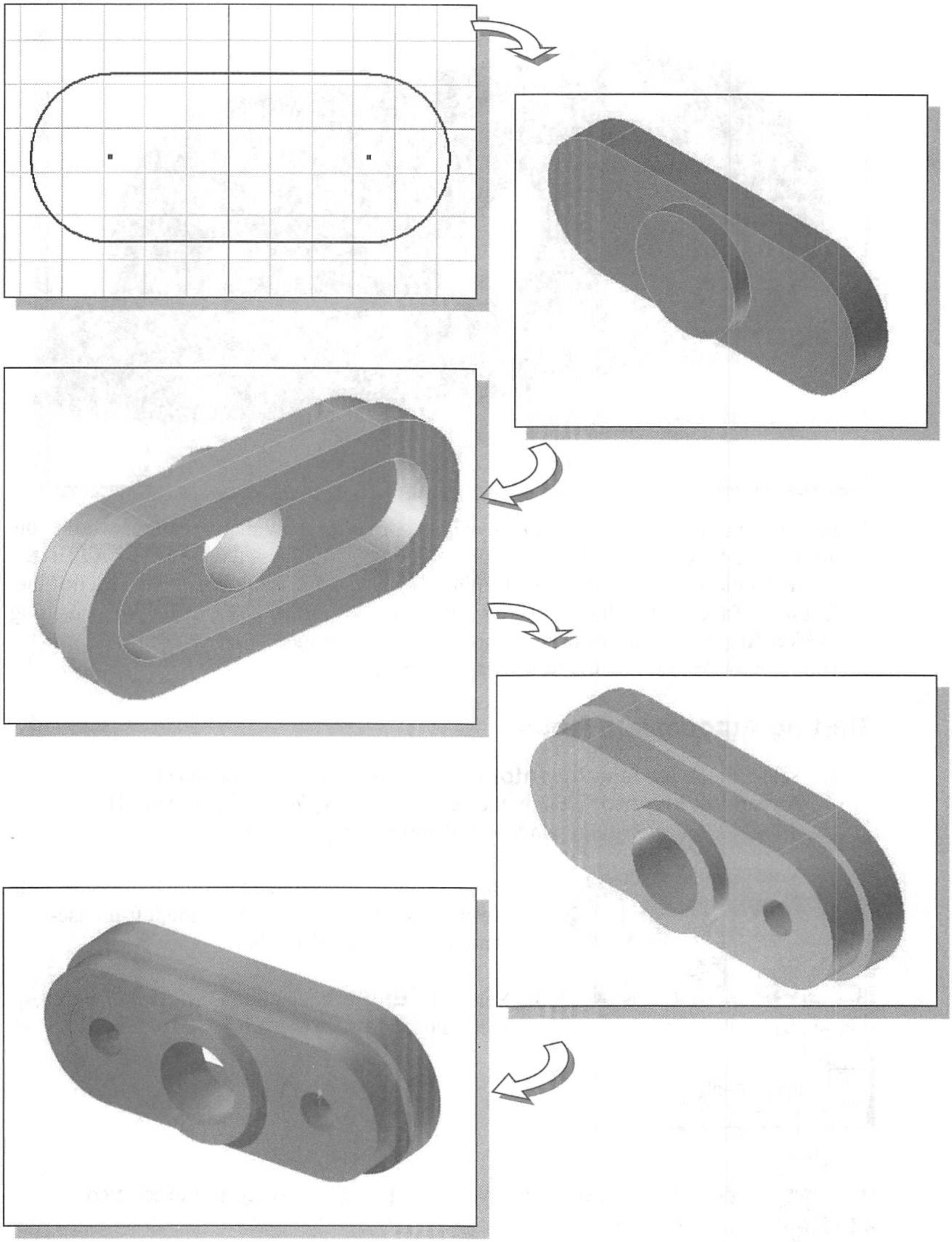

2. Once the program is loaded into memory, select the **New** icon with a single click of the left-mouse-button in the *Launch* toolbar.

3. Select the **English** tab and in the *Template* area, then select **Standard(in).ipt**.




4. Pick **Create** in the *New File* dialog box to accept the selected settings.

## Modeling Strategy

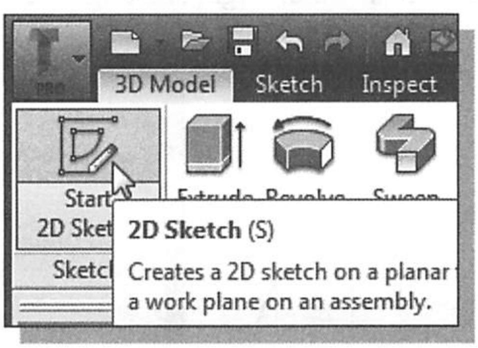


## The Autodesk Inventor Browser


- In the *Autodesk Inventor* screen layout, the ***browser*** is located to the left of the graphics window. *Autodesk Inventor* can be used for part modeling, assembly modeling, part drawings, and assembly presentation. The *browser* window provides a visual structure of the features, constraints, and attributes that are used to create the part, assembly, or scene. The *browser* also provides right-click menu access for tasks associated specifically with the part or feature, and it is the primary focus for executing many of the *Autodesk Inventor* commands.

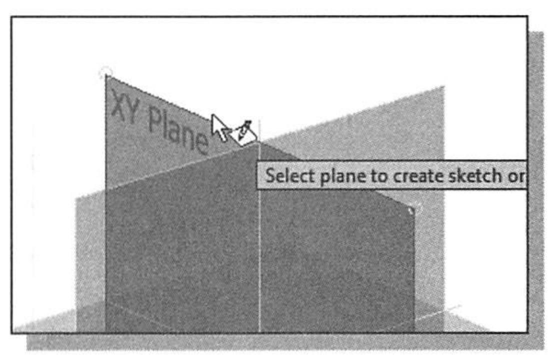

- The first item displayed in the *browser* is the name of the part, which is also the file name. By default, the name "Part1" is used when we first started *Autodesk Inventor*. The *browser* can also be used to modify parts and assemblies by moving, deleting, or renaming items within the hierarchy. Any changes made in the *browser* directly affect the part or assembly and the results of the modifications are displayed on the screen instantly. The *browser* also reports any problems and conflicts during the modification and updating procedure.

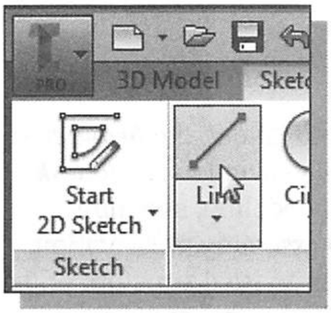
## Create the Base Feature




1. In the *Sketch* toolbar select the **Start 2D Sketch** command by left-clicking once on the icon to start the new 2D sketch.

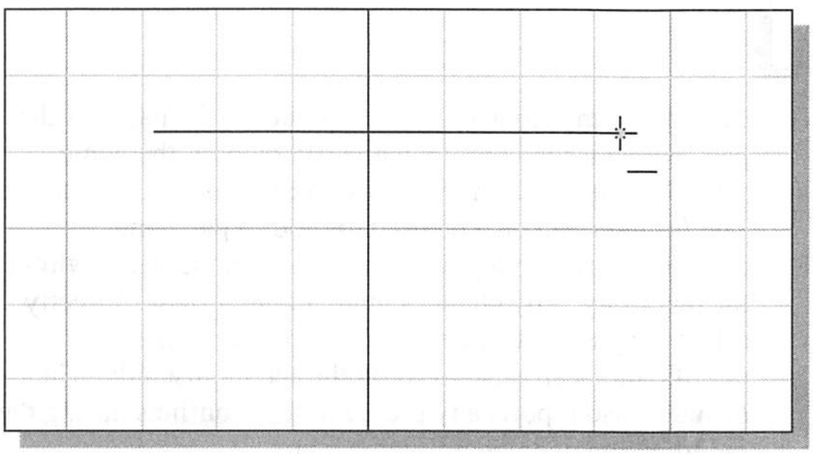
2. Move the cursor over the edge of the *XY Plane* in the graphics area. When the *XY Plane* is highlighted, click once with the **left-mouse-button** to select the *Plane* as the sketch plane for the new sketch.





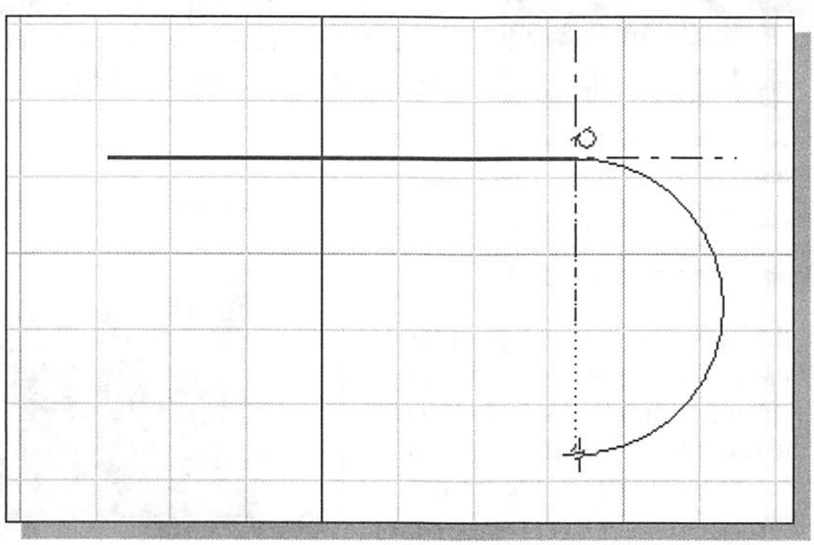

3. Move the graphics cursor to the **Line** icon in the *Draw* toolbar. A *Help tip* box appears next to the cursor and a brief description of the command is displayed at the bottom of the drawing screen "*Creates Straight line segments and tangent arcs.*" Click once with the left-mouse-button to select the command.

4. Select the icon by clicking once with the left-mouse-button; this will activate the Line command. In the *Status Bar* area, near the bottom of the *Autodesk Inventor* drawing screen, the message "*Specify start point, drag off endpoint for tangent arcs*" is displayed. *Autodesk Inventor* expects us to identify the starting location of a straight line.

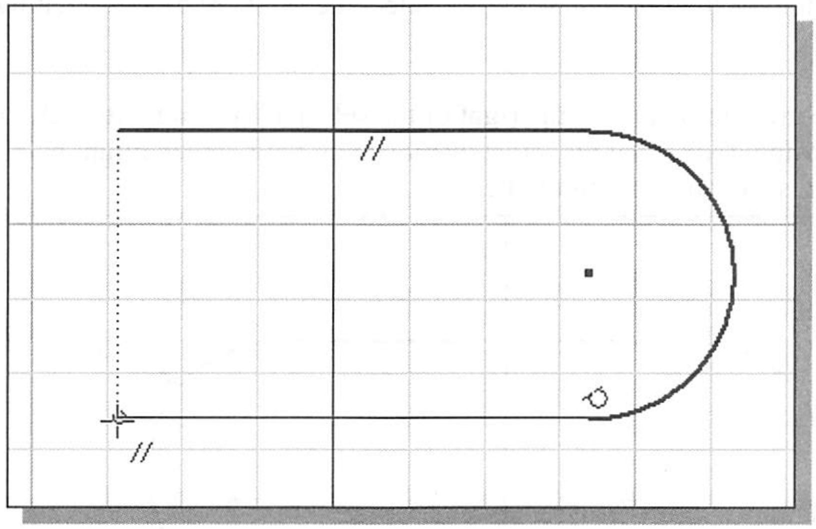


5. On your own, create a **horizontal line** from left to right as shown. Do not exit the Line command yet.

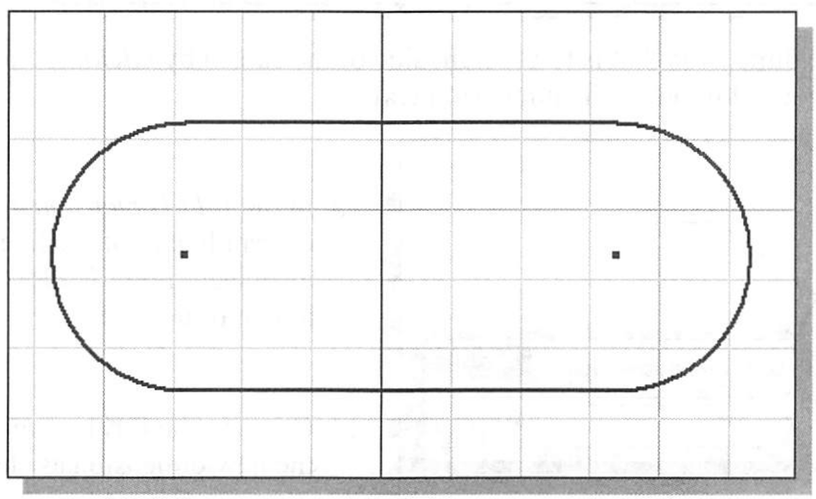
6. Move the cursor on top of the last point of the sketch and **drag** with the **left mouse button** to activate the **Arc** option. Create an arc on the right and align the other end of the arc as shown.



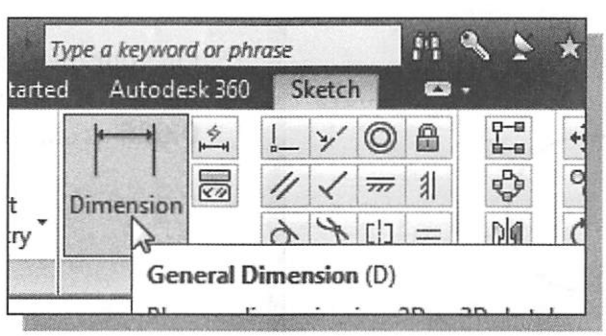
7. Create another horizontal line aligned to the arc as shown.



8. Move the cursor on top of the last point of the sketch and drag with the left-mouse-button to activate the **Arc** option. Create another arc on the left to form a closed region as shown.

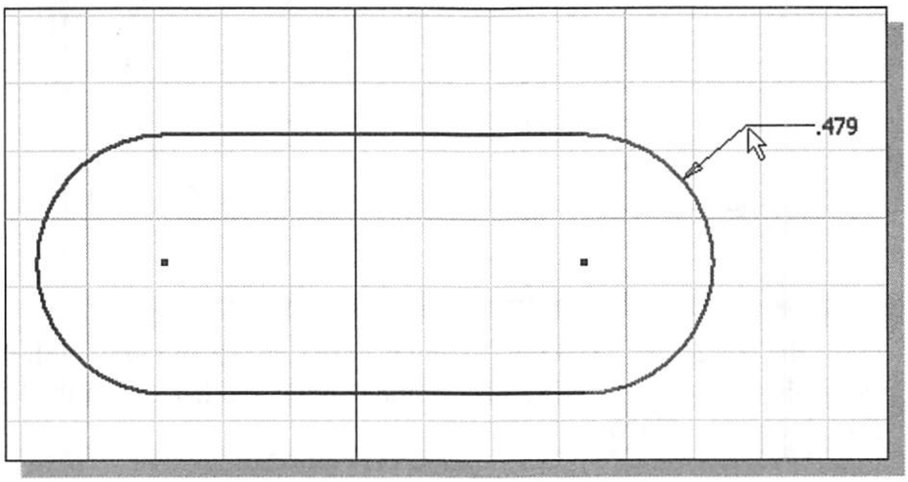


9. Activate the **General Dimension** command by clicking once with the left-mouse-button. The General Dimension command allows us to quickly create and modify dimensions.

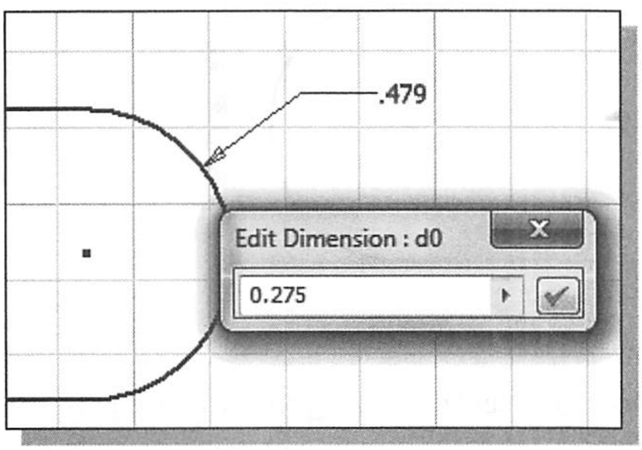


10. The message "*Select Geometry to Dimension*" is displayed in the *Status Bar* area at the bottom of the *Inventor* window. Select the right arc by left-clicking once on the arc.

11. Move the graphics cursor to the right of the selected line and left-click to place the dimension. (Note that the value displayed on your screen might be different than what is shown in the figure below.)

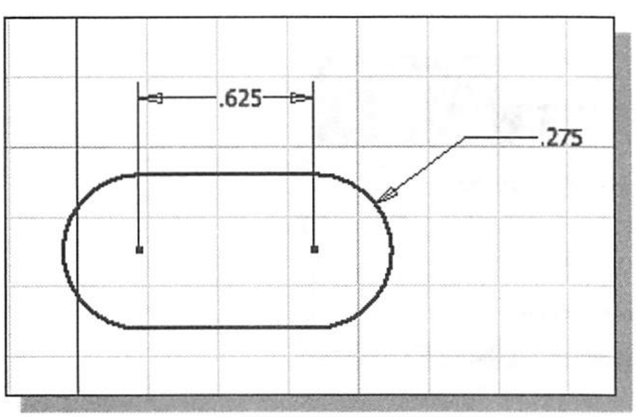



12. Select the dimension that is to the right side of the sketch by ***clicking*** once with the left-mouse-button on the dimension text.

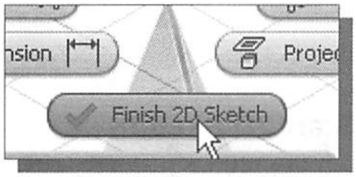



13. In the *Edit Dimension* box, the current length of the line is displayed. Enter **0.275** to set the length of the line.

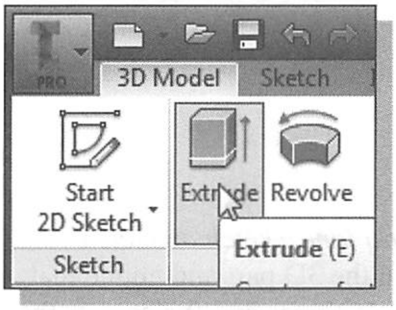
14. Click on the **OK** button to accept the new dimension as shown.




15. On your own, create and modify the center to center distance to **0.625** as shown.

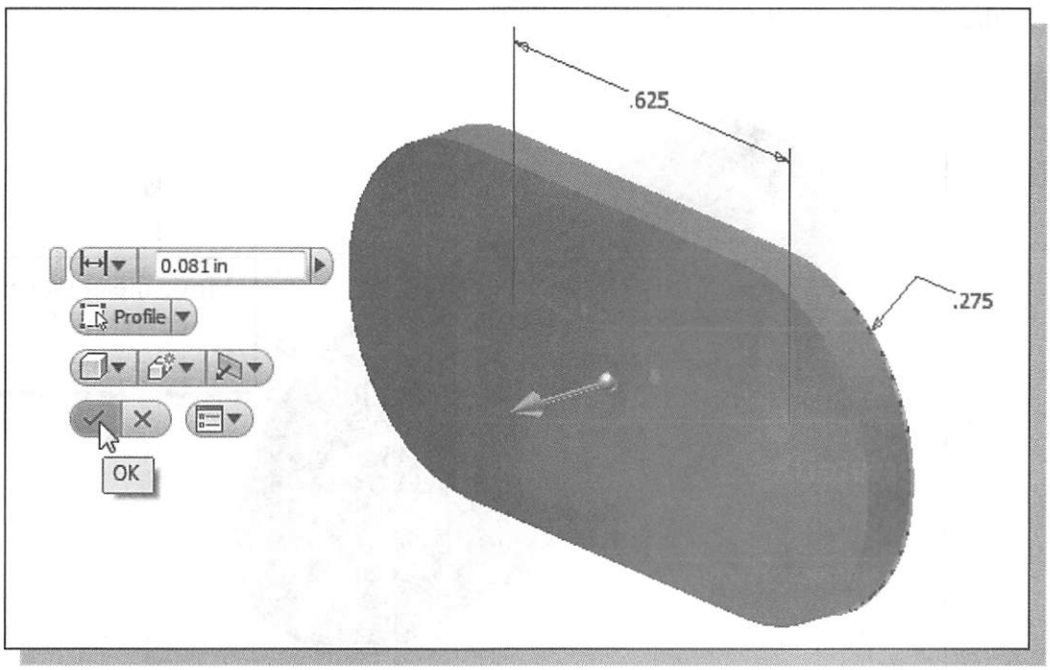



16. Inside the *graphics window*, click once with the right-mouse-button to display the option menu. Select **Finish 2D Sketch** in the popup menu to end the Sketch option.



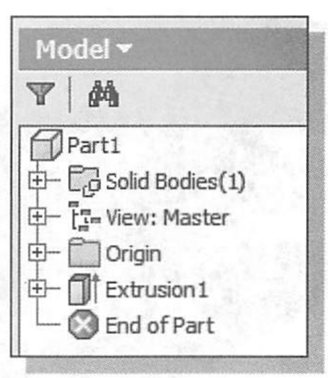

17. In the *Create* toolbar (the toolbar that is located to the right side of the *Sketch* toolbar in the *Ribbon*), select the **Extrude** command by releasing the left-mouse-button on the icon.

18. In the *Distance* option box, enter **.081** as the total extrusion distance.

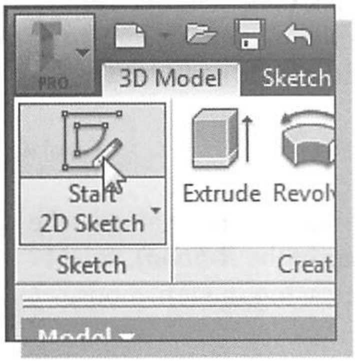



19. Click on the **OK** button to accept the settings and create the base feature.

➢ On your own, use the *Dynamic Viewing* functions to view the 3D model. Also note the extrusion feature is added to the *Model Tree* in the *browser* area.

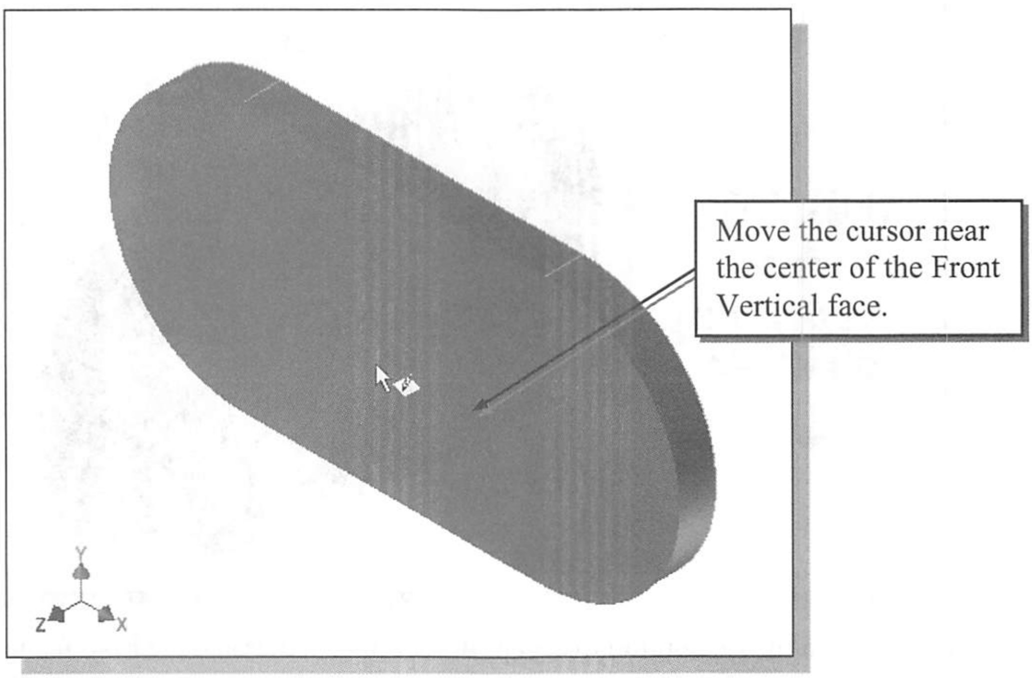


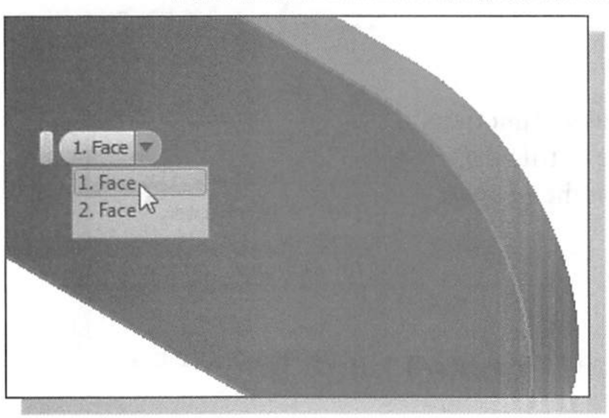
## Add the Second Solid Feature




1. In the *Sketch* toolbar select the **Start 2D Sketch** command by left-clicking once on the icon.

2. In the *Status Bar* area, the message "*Select face, workplane, sketch or sketch geometry.*" is displayed. Move the graphics cursor on the 3D part and notice that *Autodesk Inventor* will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Move the cursor inside the front face of the 3D object as shown below.

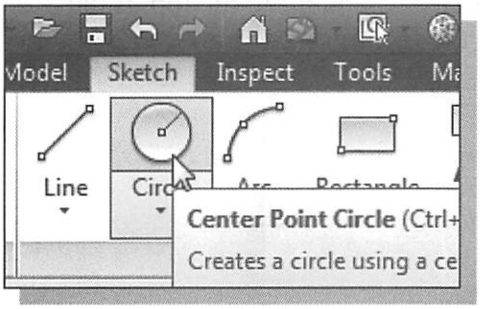



3. Pause the cursor on the surface until a selection list appears, click on the down arrow to examine all possible surface selections.

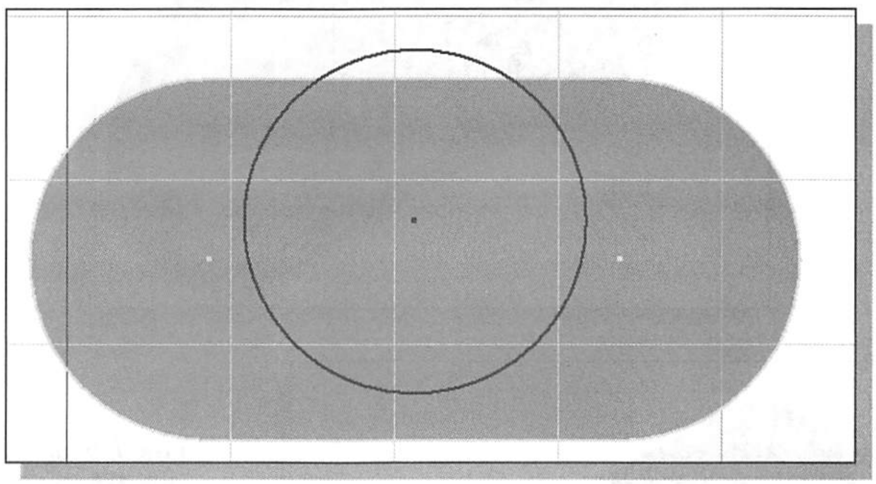
4. Select the **front face** of the solid model when it is highlighted as shown in the figure.

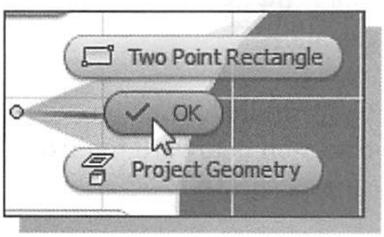
## Create a 2D Sketch



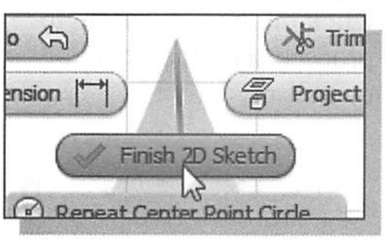

1. Select the **Center Point Circle** command by clicking once with the left-mouse-button on the icon in the *Draw* toolbar.

2. Create a circle that extends to the outside of the front face of the solid model, as shown. (Note that we will intentionally not add any dimensions to our sketch.)

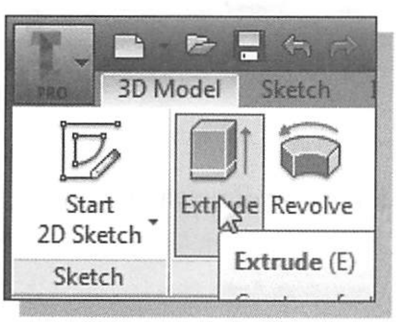





3. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **OK** in the popup menu to end the Circle command.

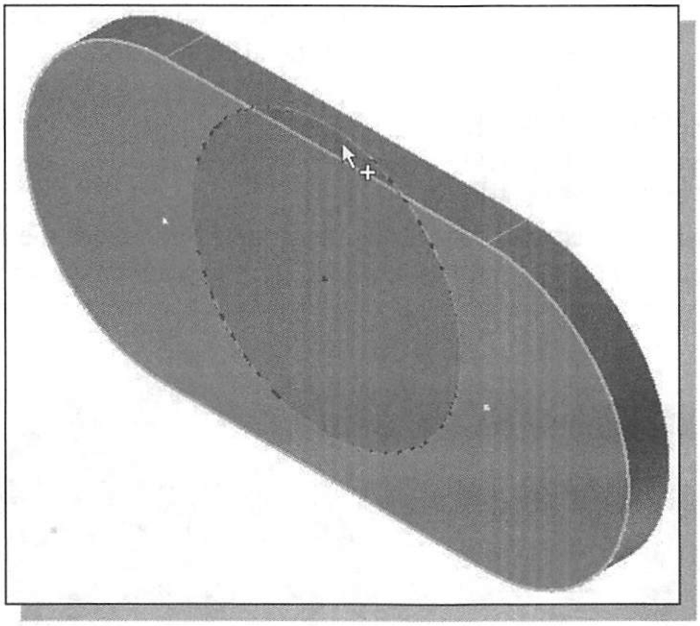


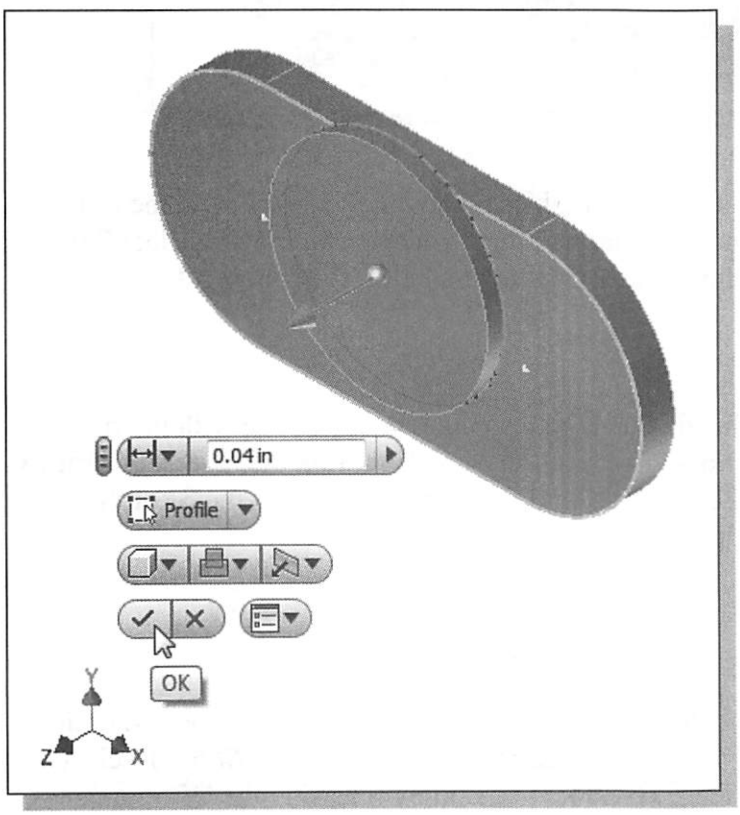

4. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Finish 2D Sketch** in the popup menu to end the Sketch option.




5. In the *Create* toolbar (the toolbar that is located to the right side of the *Sketch* toolbar in the *Ribbon*), select the **Extrude** command by releasing the left-mouse-button on the icon.

6. In the *Extrude* popup window, the **Profile** option is activated; *Autodesk Inventor* expects us to identify the profile to be extruded. Move the cursor to the top of the circle we just created and left-click once to select the region as the profile to be extruded.

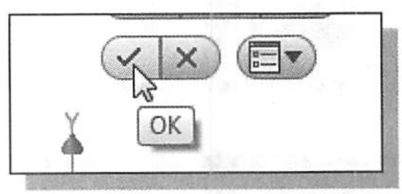





7. In the *Distance* option box, enter **.04** as the extrusion distance.

8. Confirm the extrusion direction is toward the front side as shown.

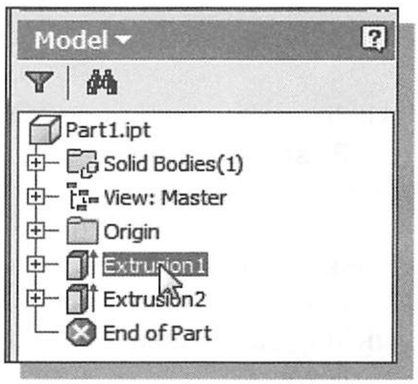
9. Click on the **OK** button to proceed with the *Join* operation.



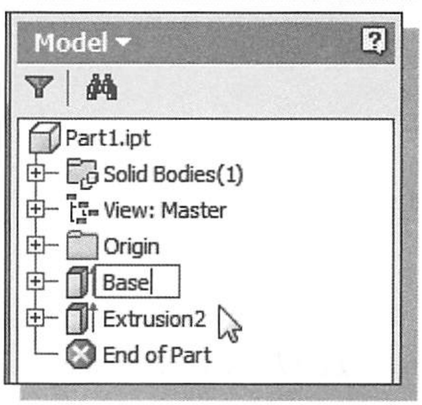

➢ Note that the cylindrical feature is created without adding any dimension to the 2D sketch.

## Rename the Part Features

♦ Currently, our model contains two extruded features. The feature is highlighted in the display area when we select the feature in the *browser* window. Each time a new feature is created, the feature is also displayed in the *Model Tree* window. By default, *Autodesk Inventor* will use generic names for part features. However, when we begin to deal with parts with a large number of features, it will be much easier to identify the features using more meaningful names. Two methods can be used to rename the features: (1) Clicking twice on the name of the feature, and (2) using the *Properties* option. In this example, the use of the first method is illustrated.




1. Select the first extruded feature in the *model browser* area by left-clicking once on the name of the feature, **Extrusion1**. Notice the selected feature is highlighted in the graphics window.



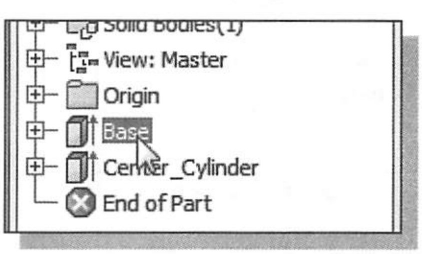

2. **Left-mouse-click** on the feature name again to enter the *Edit* mode as shown.

3. Enter **Base** as the new name for the first extruded feature.

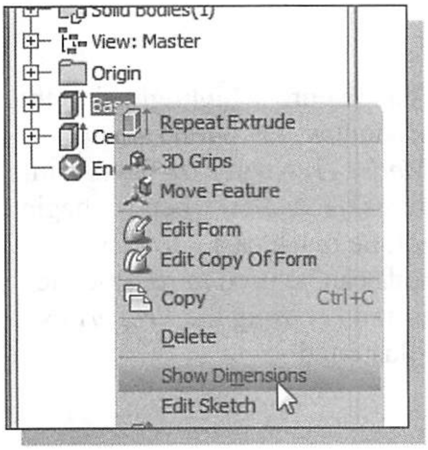
4. On your own, rename the second extruded feature to **Center_Cylinder**.

## Adjusting the Dimensions of the Base Feature

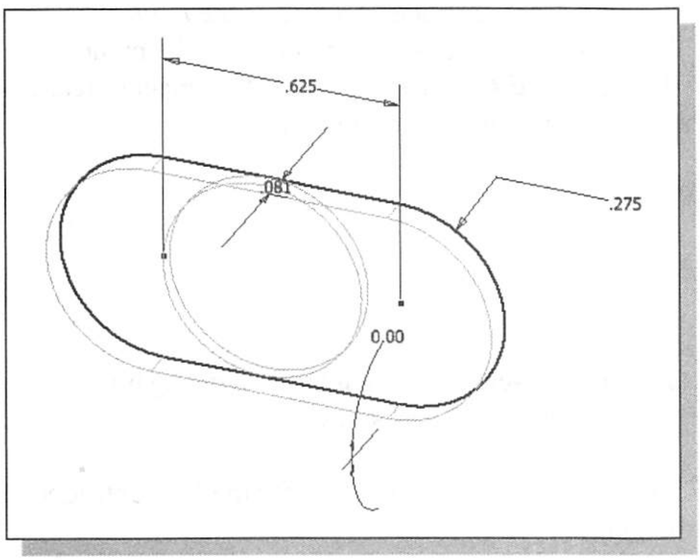
❖ One of the main advantages of parametric modeling is the ease of performing part modifications at any time in the design process. Part modifications can be done through accessing the features in the history tree. *Autodesk Inventor* remembers the history of a part, including all the rules that were used to create it, so that changes can be made to any operation that was performed to create the part.



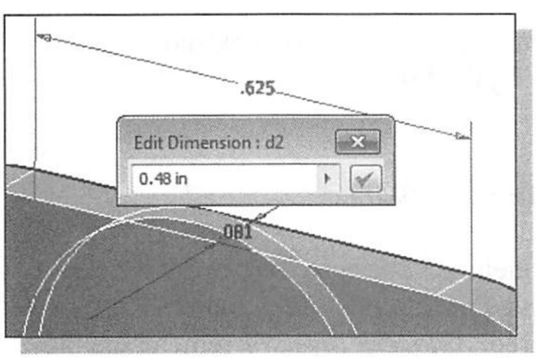

1. Select the first extruded feature, **Base**, in the *browser* area. Notice the selected feature is highlighted in the graphics window.




2. Inside the *browser* area, **right-mouse-click** on the first extruded feature to bring up the option menu and select the **Show Dimensions** option in the pop-up menu.



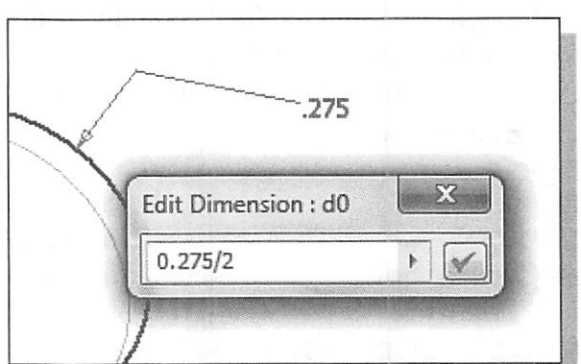

3. All dimensions used to create the **Base** feature are displayed on the screen. Select the overall width of the **Base** feature, the **0.625** dimension value, by **double-clicking** on the dimension text as shown.



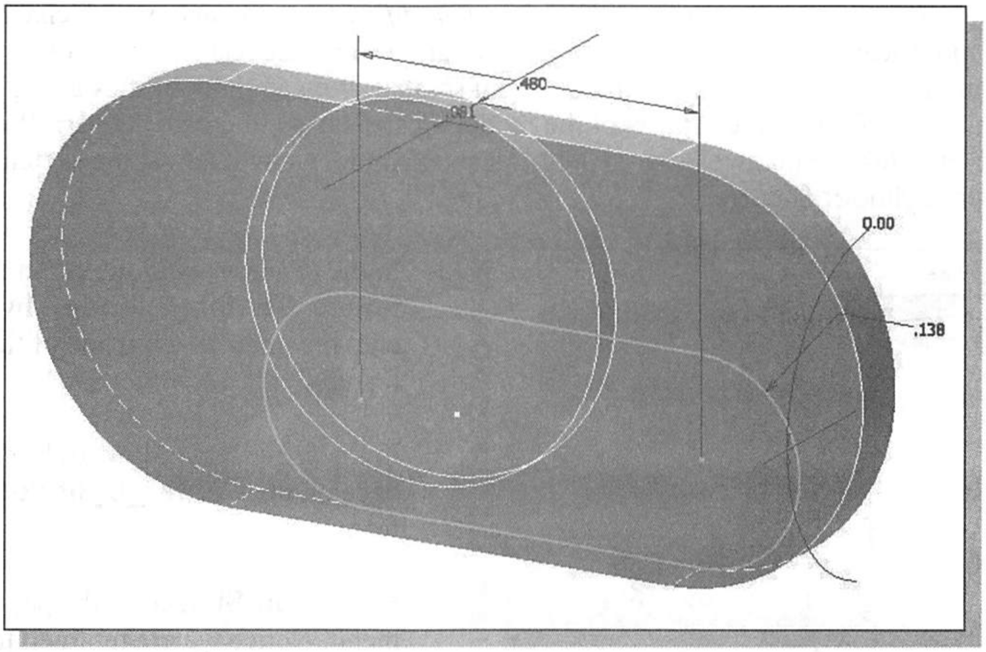

4. Enter **0.48** in the *Edit Dimension* window.

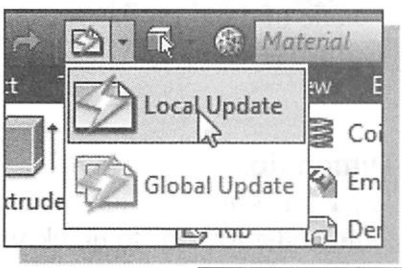
5. On your own, repeat the above steps and modify the radius to half its current value.

6. Enter **0.275/2** in the *Edit Dimension* window. (Note that *Inventor* allows the input of mathematical operations for dimensions.)

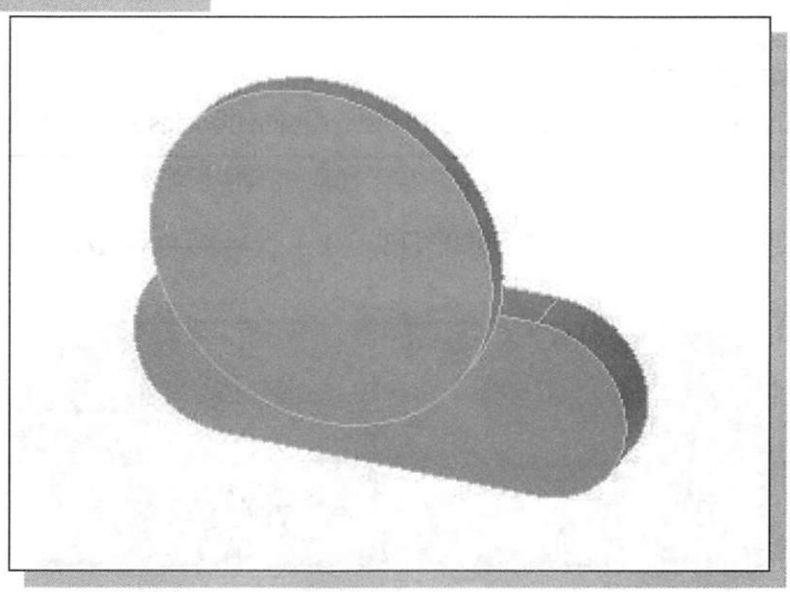


7. The 2D sketch has been updated to reflect the changes, but the solid feature still needs to be updated.



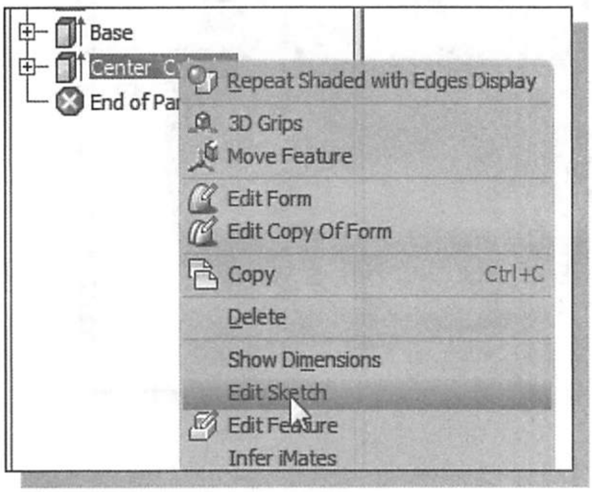

8. Click **Local Update** in the *Quick Access* toolbar.



➢ Note that *Autodesk Inventor* updates the model by re-linking all elements used to create the model. Any problems or conflicts that occur will also be displayed during the updating process.

## History-Based Part Modifications

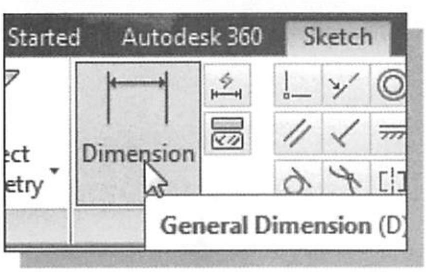
*Autodesk Inventor* uses the *history-based part modification* approach, which enables us to make modifications to the appropriate features and re-link the rest of the history tree without having to reconstruct the model from scratch. We can think of it as going back in time and modifying some aspects of the modeling steps used to create the part. We can modify any feature that we have created. As an example, we will adjust the sketch of the Center_Cylinder feature.




1. In the *browser* window, select the last feature, **Center_Cylinder**, by left-clicking once on the name of the feature.

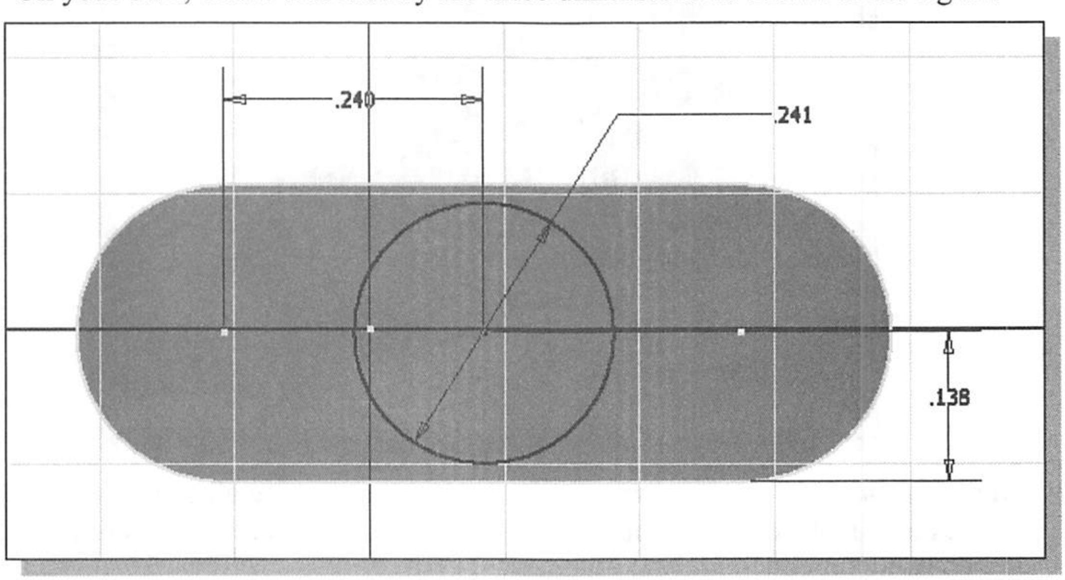
2. In the *browser* window, right-click once on the **Center_Cylinder** feature.

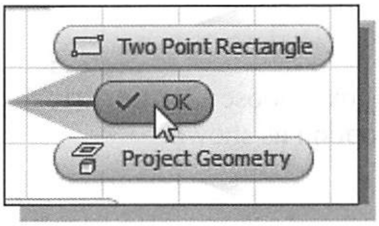
3. Select **Edit Sketch** in the pop-up menu. Notice we are returned to the *2D sketch* mode of the **Center_Cylinder** feature.



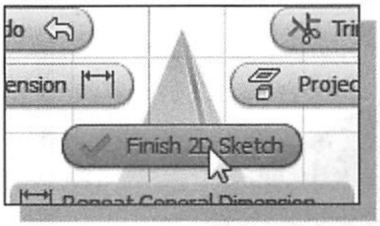

4. Activate the **General Dimension** command by clicking once with the left-mouse-button. The General Dimension command allows us to quickly create and modify dimensions

5. On your own, create and modify the three dimensions as shown in the figure.

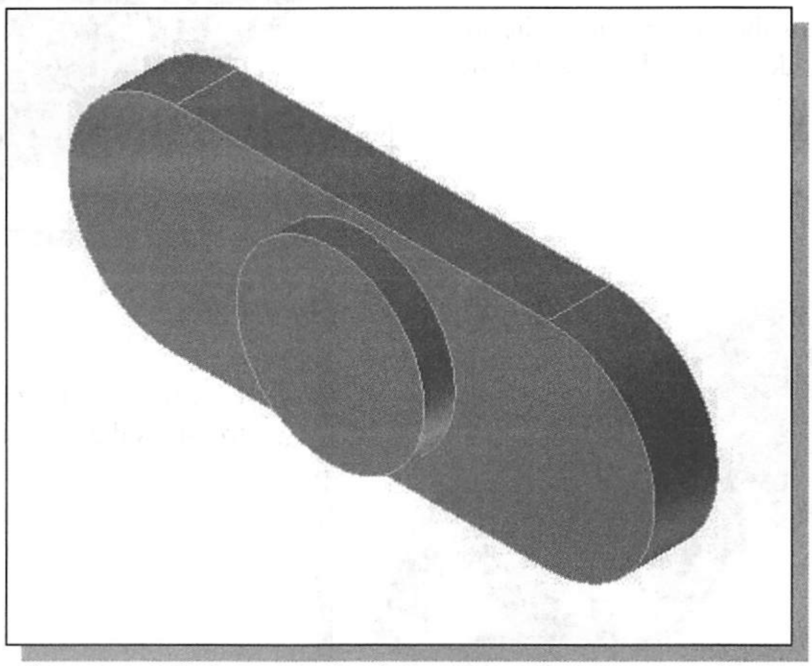





6. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **OK** in the popup menu to end the **Circle** command.

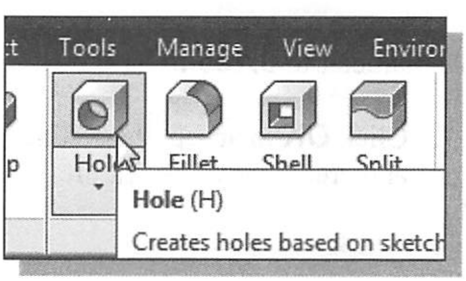



7. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Finish 2D Sketch** in the popup menu to end the **Sketch** option.

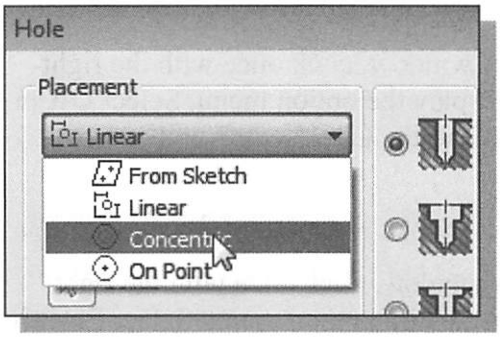


## Add a Placed Feature

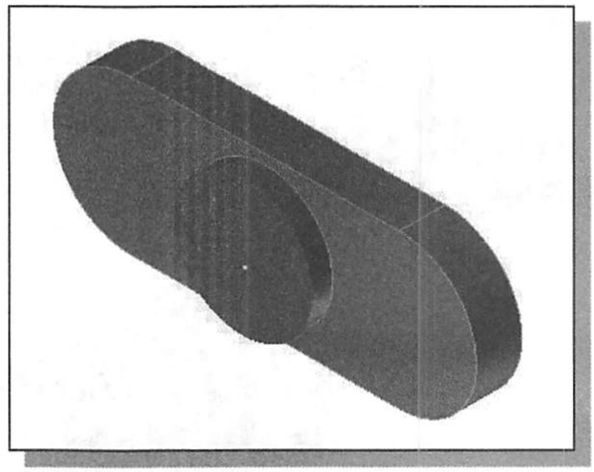
- In *Autodesk Inventor*, there are two types of geometric features: **placed features** and **sketched features**. The last feature we created is a *sketched feature*, where we created a rough sketch and performed an extrusion operation. We can also create a feature without creating a 2D sketch, which is known as a *placed feature*. A *placed feature* is a feature that does not need a sketch and can be created automatically. In parametric modeling, holes, fillets, chamfers, and shells are all placed features.



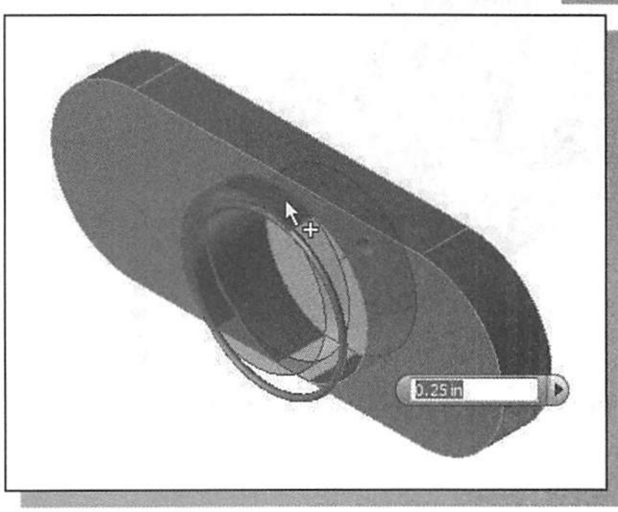

1. In the *Modify* toolbar, select the **Hole** command by clicking the left-mouse-button on the icon.



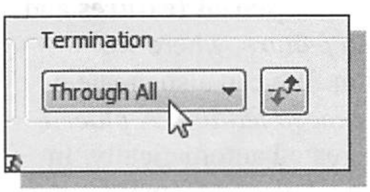

2. In the *Hole* dialog box, choose **Concentric** for the *Placement* option as shown.



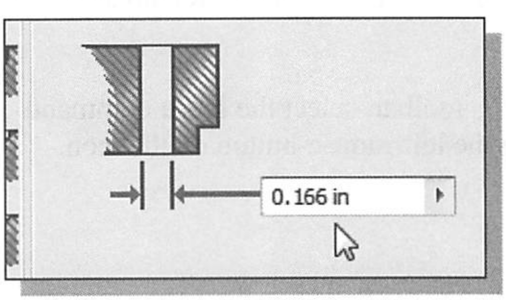
3. Pick the front plane of the solid model as the placement plane as shown.



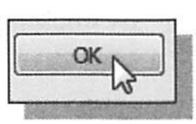

4. Pick the center cylindrical surface to use as the concentric reference.




5. Set the *Termination* option to **Through All** as shown.

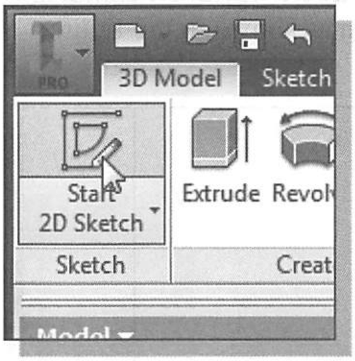



6. Set the hole diameter to **0.166 in** as shown.



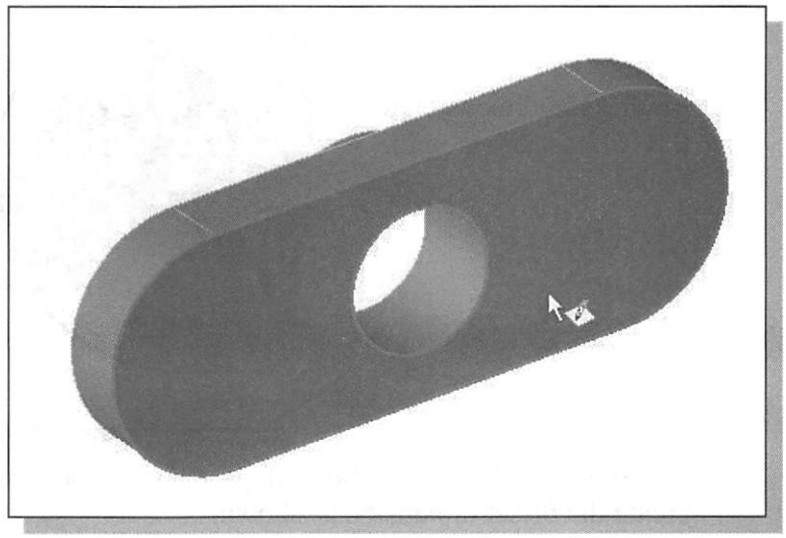

7. Click **OK** to accept the settings and create the *Hole* feature.

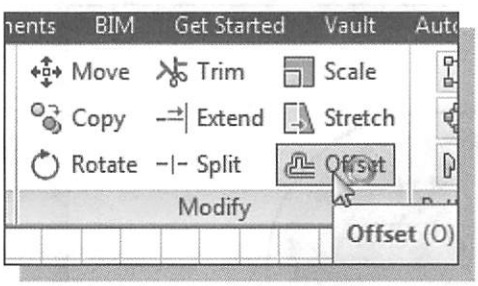
## Using the Offset Command to Create a Feature




1. In the *Sketch* toolbar select the **Start 2D Sketch** command by left-clicking once on the icon.

2. Pick the **back face** of the solid as shown.

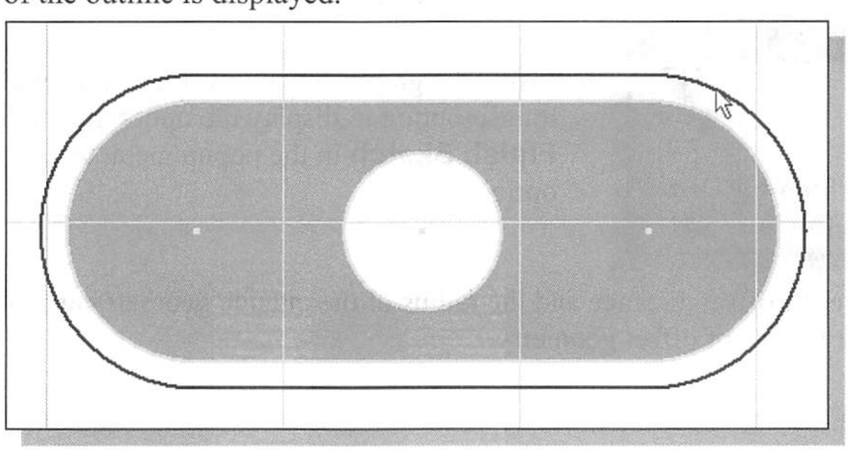


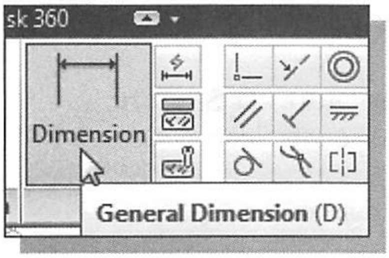



3. Click on the **Offset** icon in the *Modify* panel.

4. Select any edge of the **back face** of the 3D model. *Autodesk Inventor* will automatically select all of the connecting geometry to form a closed region.

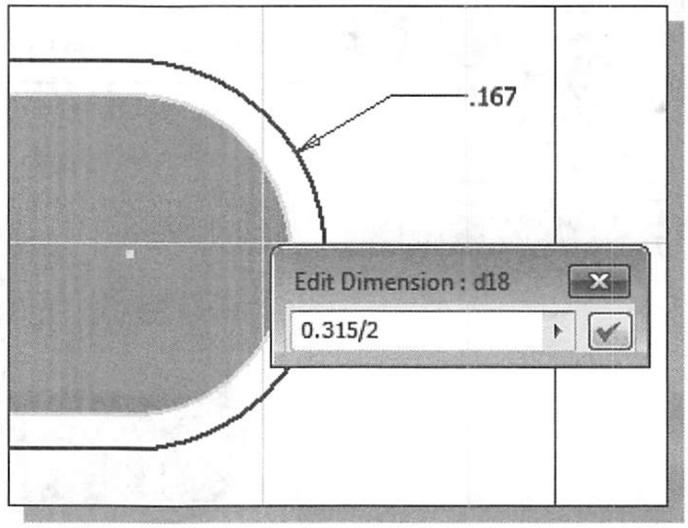
5. Move the cursor toward the outside of the selected region and notice an offset copy of the outline is displayed.



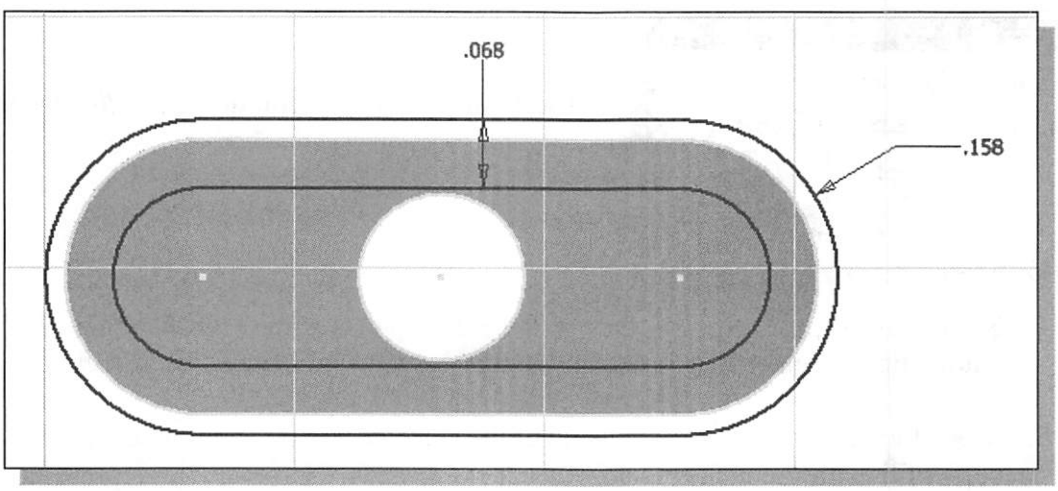


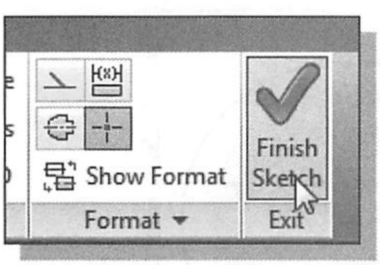

6. On your own, use the **General Dimension** command to create the radius dimension as shown in the figure below.

7. Modify the dimension to **0.315/2** as shown in the figure.



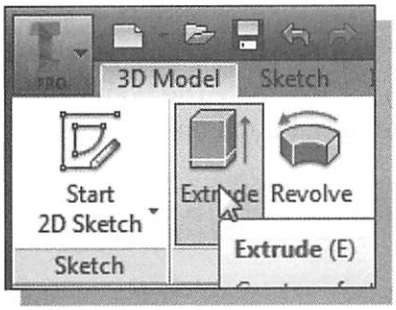

8. On your own, repeat the above steps and create another offset geometry; also create the offset dimension as shown.



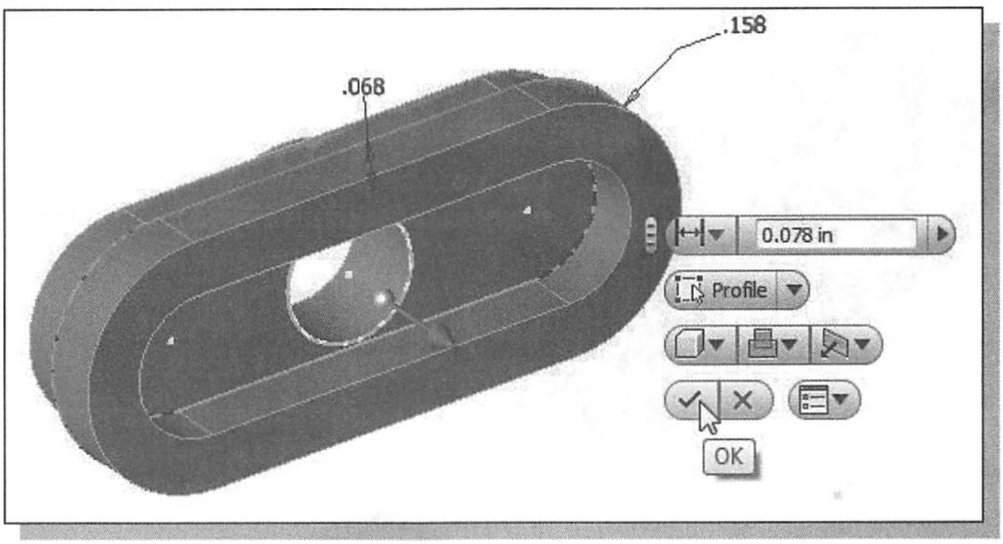

9. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Finish Sketch** in the popup menu to end the Sketch option.

➢ Note the offset distance and the radius of the outside geometry are used to control the two sets of offset geometry.

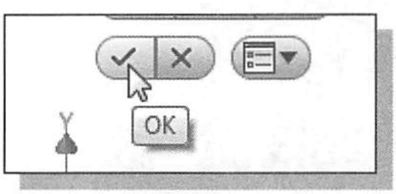



10. In the *Create* toolbar (the toolbar that is located to the right side of the *Sketch* toolbar in the *Ribbon*) select the **Extrude** command by releasing the left-mouse-button on the icon.

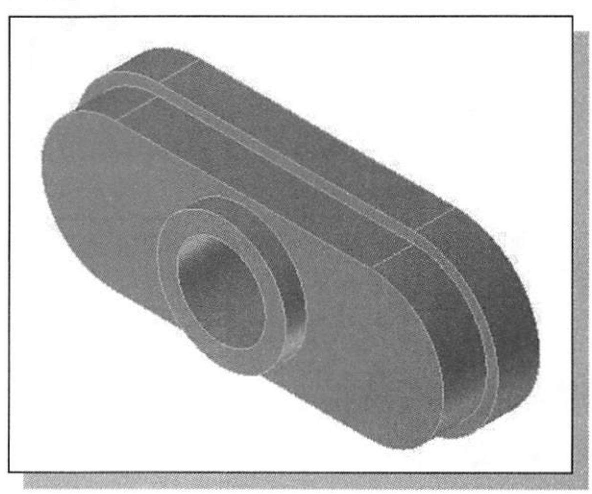
11. In the *Extrude* popup window, the **Profile** option is activated down; *Autodesk Inventor* expects us to identify the profile to be extruded. Select the **TWO regions**, located in between the two offset geometry we just created, as the profile to be extruded.



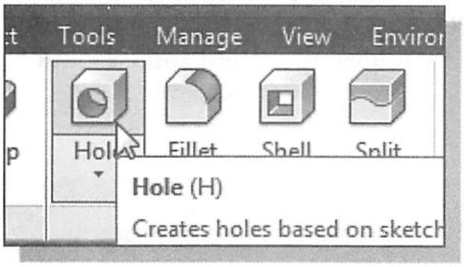

12. In the *Distance* option box, enter **.078** as the extrusion distance. Also confirm the extrusion direction is toward the back of the base feature as shown.



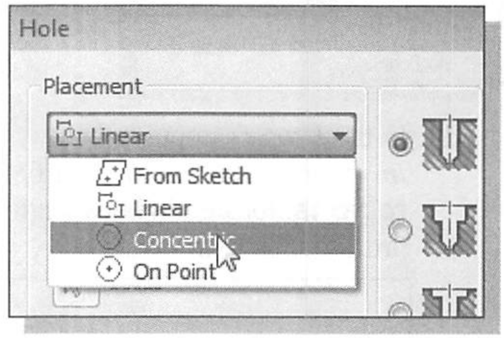

13. Click on the **OK** button to proceed with the *Join* operation.



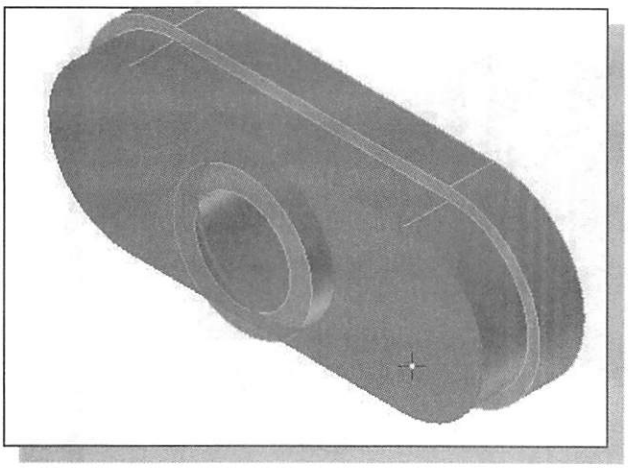
## Add another Hole Feature



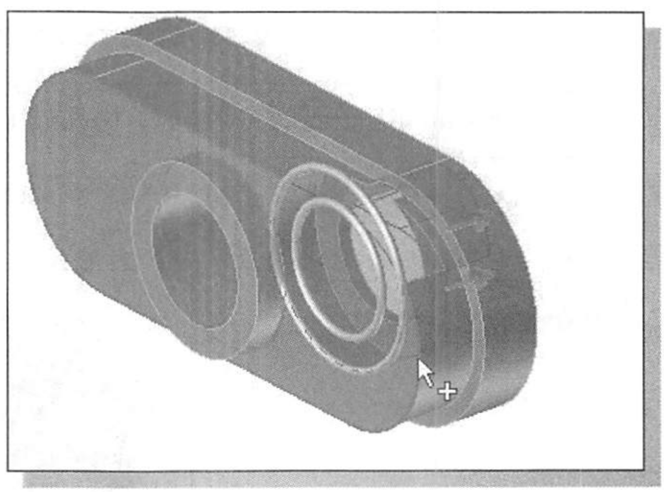

1. In the *Modify* toolbar, select the **Hole** command by clicking the left-mouse-button on the icon.



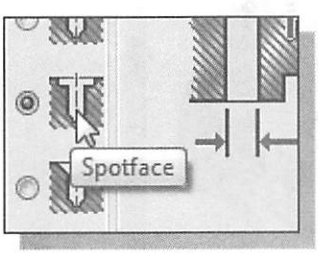

2. In the *Hole* dialog box, choose **Concentric** as the *Placement* option as shown.



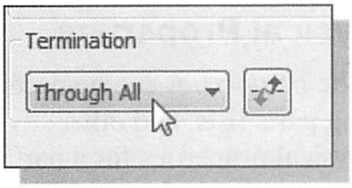
3. Pick the front plane of the **Base** feature as the placement plane as shown.



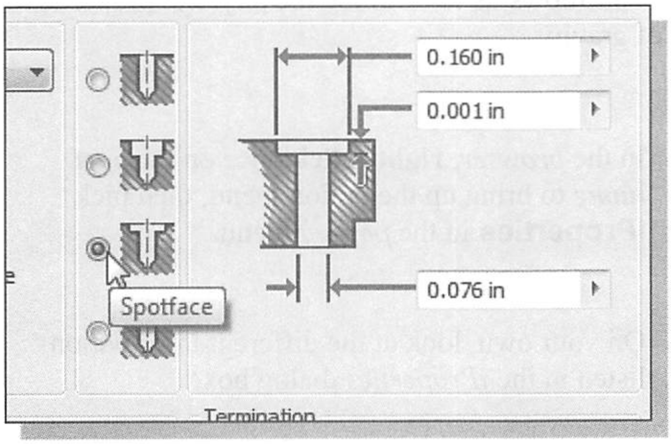
4. Pick one of the cylindrical surfaces or arcs on the right to use as the concentric reference.




5. Set the hole type to **Spotface** as shown.



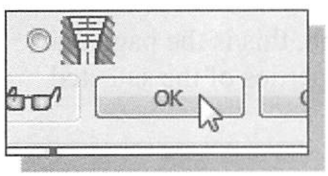

6. Set the *Termination* option to **Through All** as shown.




7. Set the hole diameter to **0.076 in** as shown.

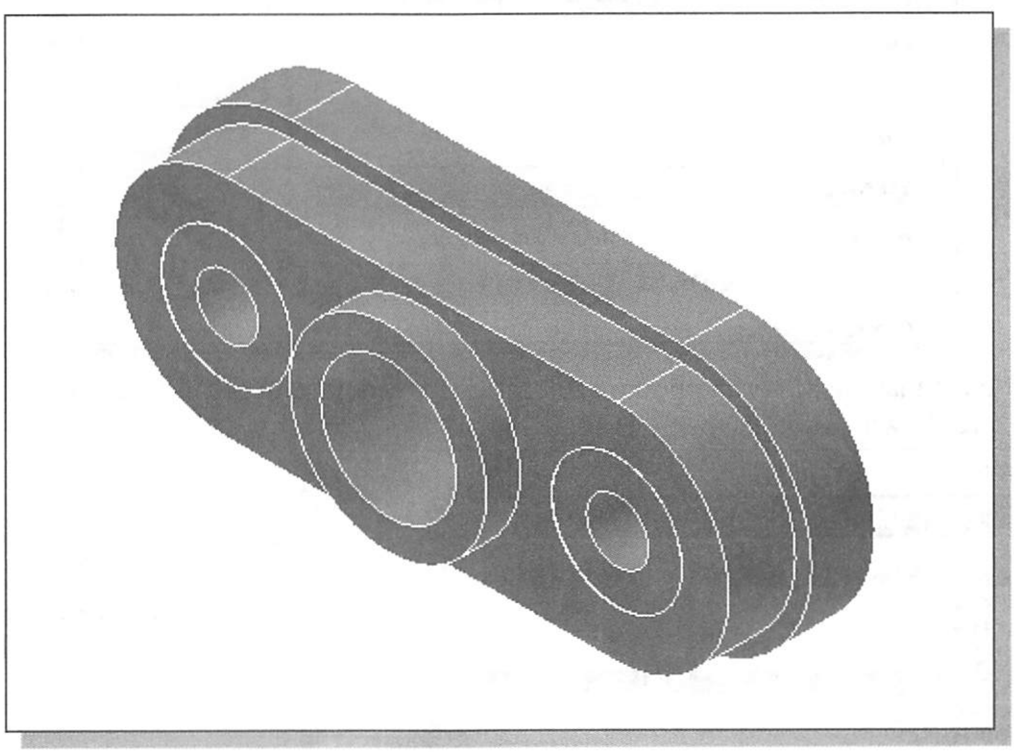
8. Set the spotface diameter to **0.160 in** as shown

9. Set the spotface depth to **0.001 in** as shown.



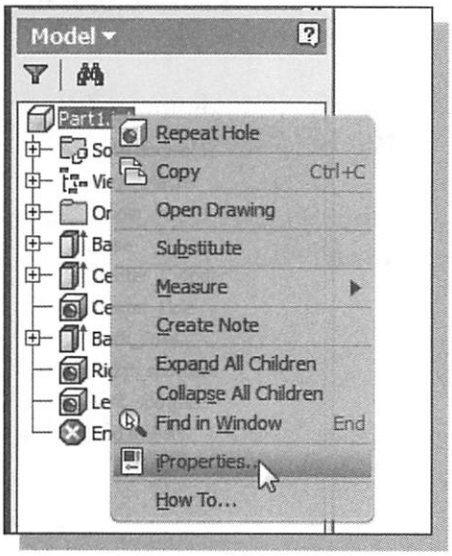

10. Click **OK** to accept the settings and create the *Hole* feature.

11. On your own, repeat the above steps and create another *spotface* feature on the other side as shown.



## Assigning and Calculating the Associated Physical Properties

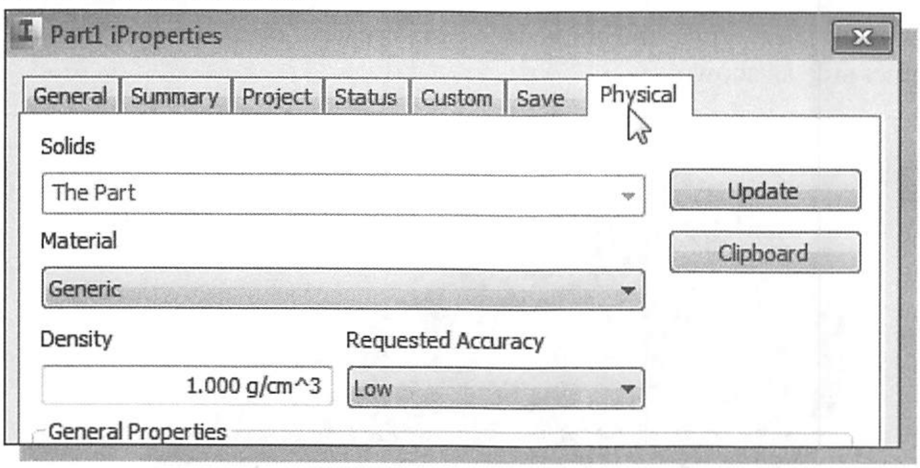
*Autodesk Inventor* models have properties called ***iProperties***. The *iProperties* can be used to create reports, and update assembly bills of materials, drawing parts lists, and other information. With *iProperties*, we can also set and calculate physical properties for a part or assembly using the material library. This allows us to examine the physical properties of the model, such as weight or center of gravity.



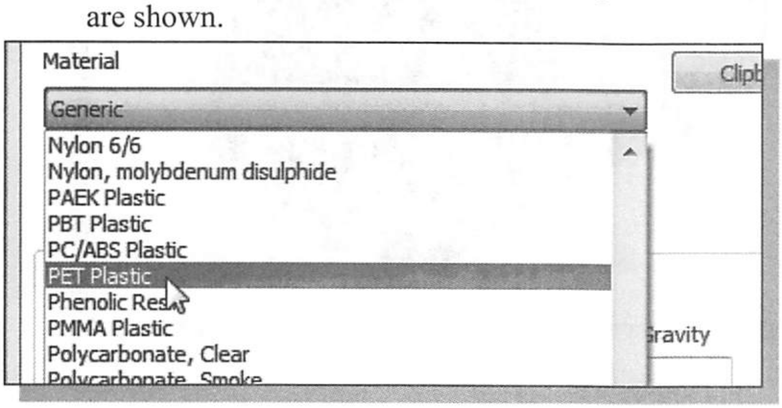

1. In the *browser*, **right--click** once on the ***part name*** to bring up the option menu; then pick **iProperties** in the *pop-up* menu.

2. On your own, look at the different information listed in the *iProperties* dialog box.

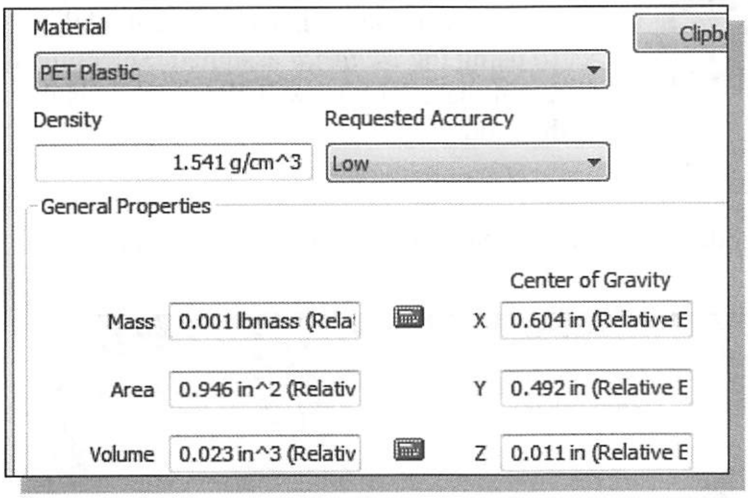
3. Click on the **Physical** tab, this is the page that contains the physical properties of the selected model.




- Note that the *Material* option is not assigned, and none of the physical properties are shown.

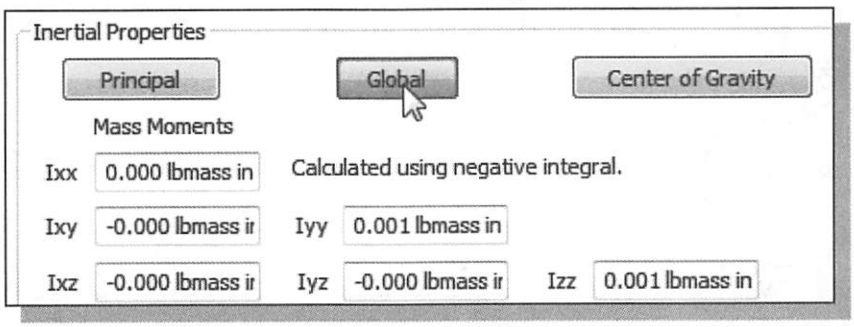


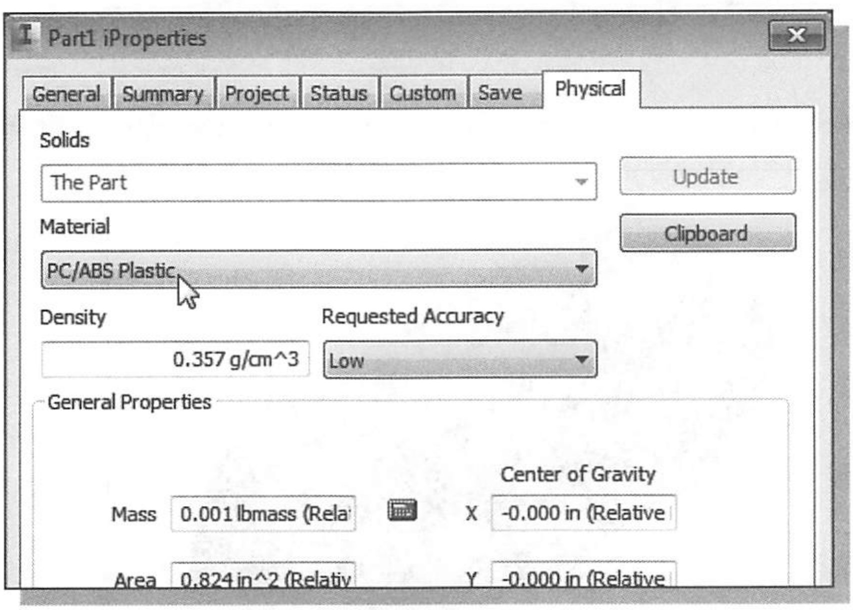

4. Click the down-arrow in the *Material* option to display the material list, and select **PET Plastic** as shown.



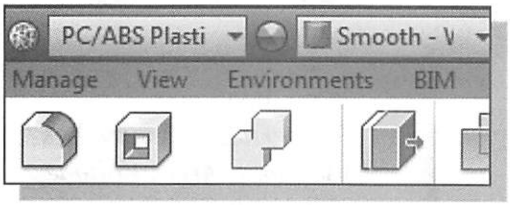

❖ Note the *General Properties* area now has the *Mass*, *Area*, *Volume* and *Center of Gravity* information of the model, based on the density of the selected material.

5. Note the *Mass Moments Inertia* of the design, with respect to the different axes, are also available.



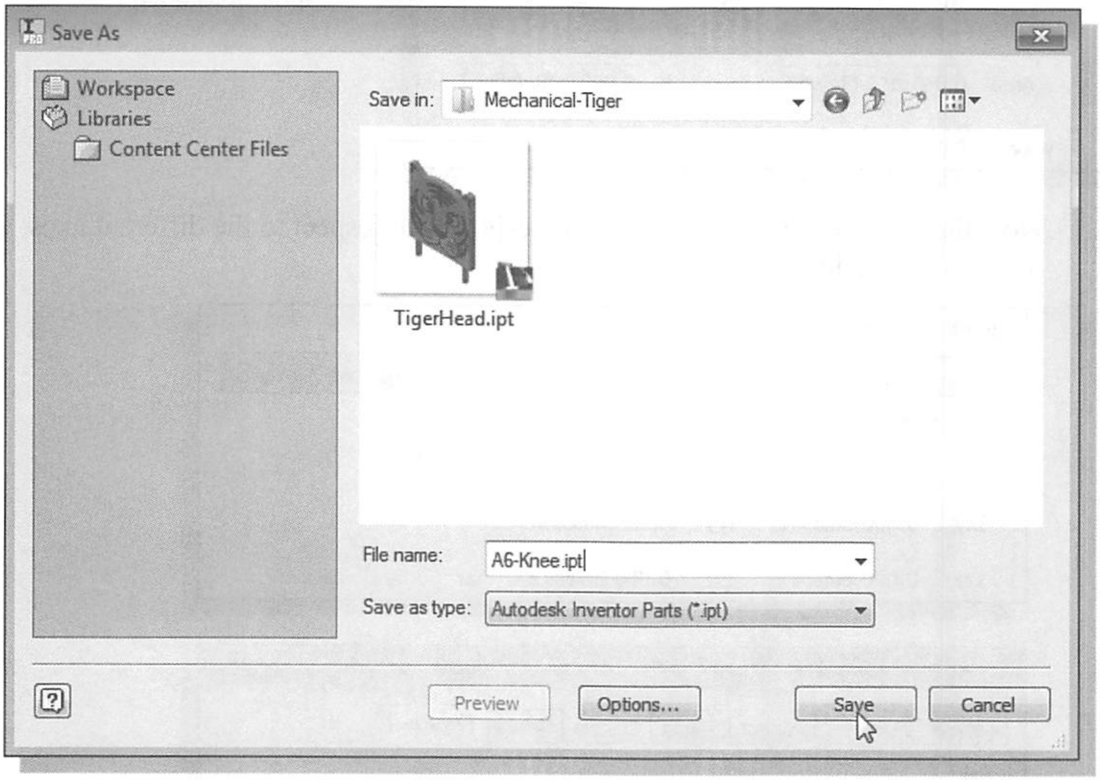



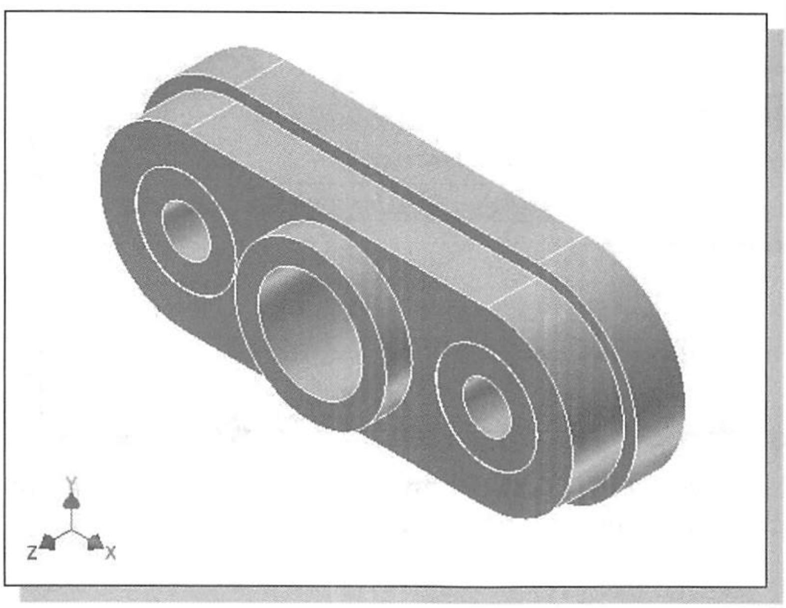

6. On your own, select **PC_ABS Plastic** as the *Material* type and compare the differences in using the different materials.




❖ Also note the default display of the model is set to using the *Material* assigned. Selecting a different material type will change the display of the model.

7. On your own, save the model in the *Mechanical-Tiger* project folder as **A6-Knee.ipt**.



## Review Questions:

1. What are stored in the *Autodesk Inventor History Tree*?
2. When extruding, what is the difference between *Distance* and *Through All*?
3. Describe the *history-based part modification* approach.
4. What determines how a model reacts when other features in the model change?
5. Describe the steps to rename existing features.
6. Describe two methods available in *Autodesk Inventor* to *modify the dimension values* of parametric sketches.
7. Create *History Tree sketches* showing the steps you plan to use to create the two models shown in the next pages:

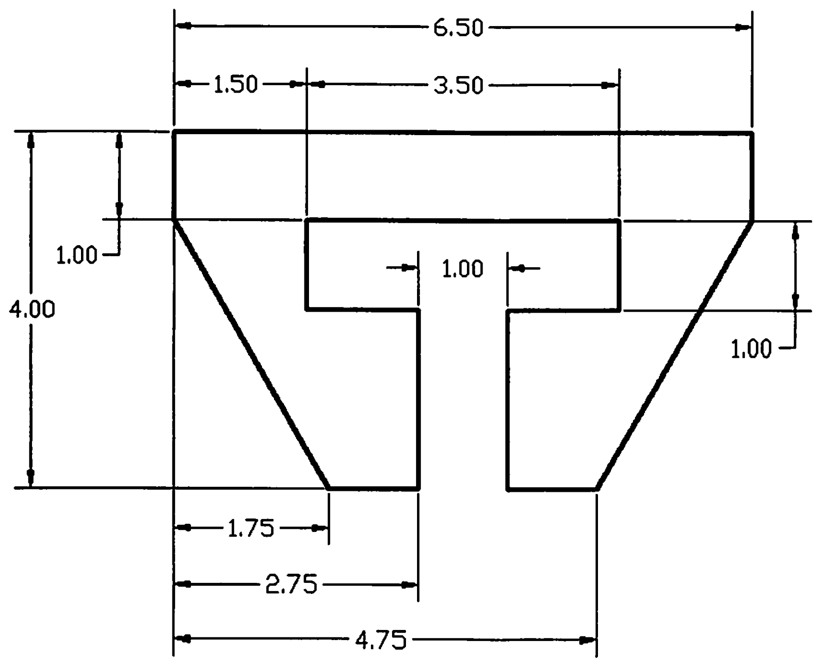
Ex.3)

Ex.4)

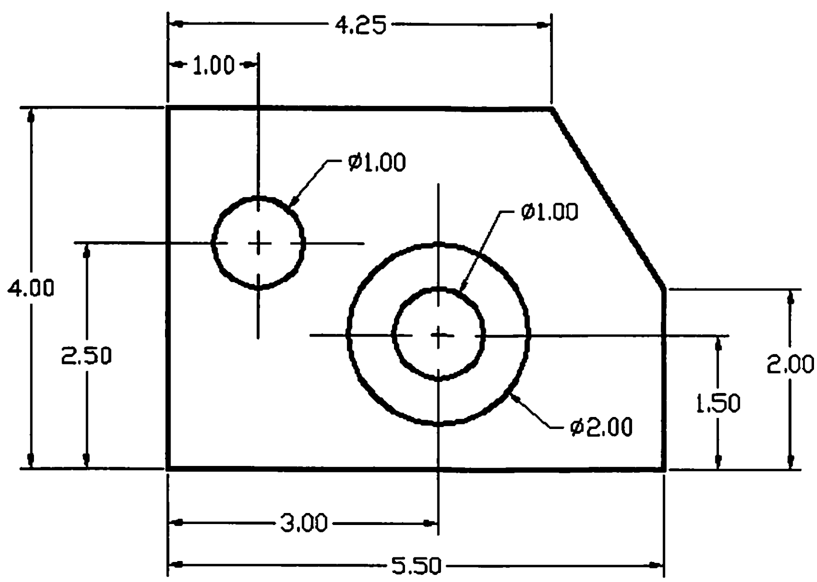
## Exercises:

(All dimensions are in inches.)

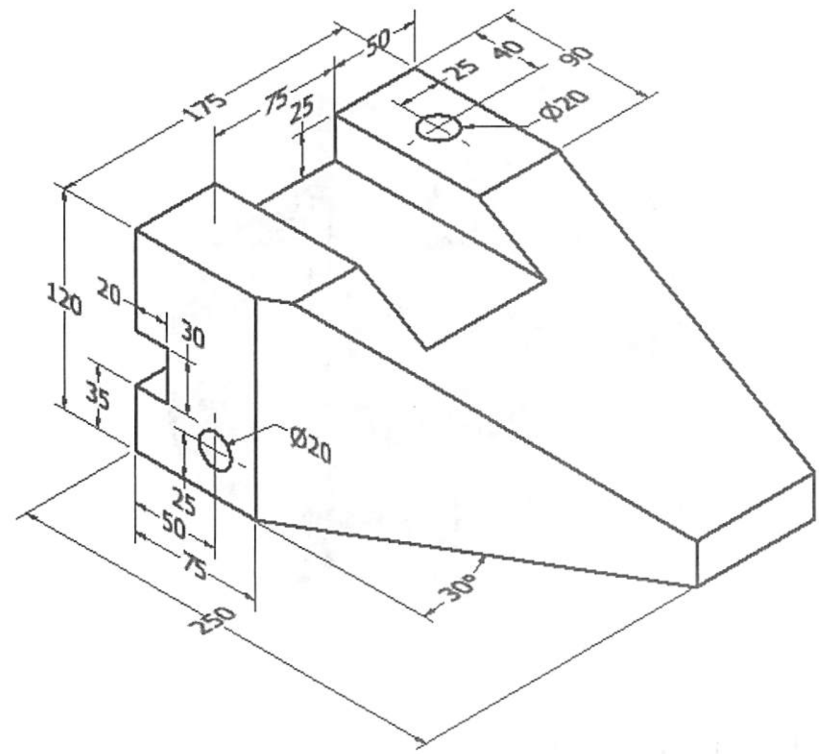
1. Latch Clip (thickness: 0.25 inches. Material: Cast Iron. Mass and Volume =?)



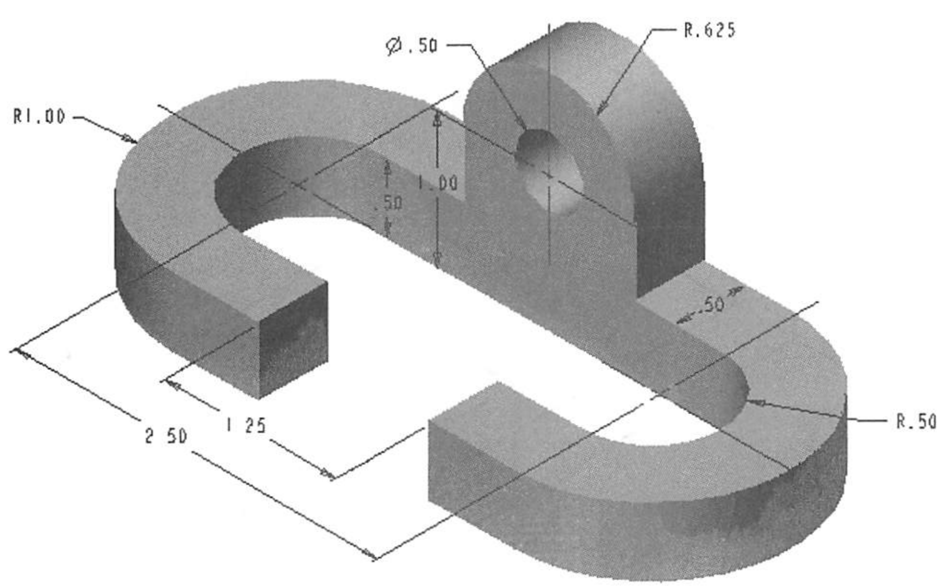

2. Guide Plate (thickness: 0.25 inches. Boss height 0.125 inches.)



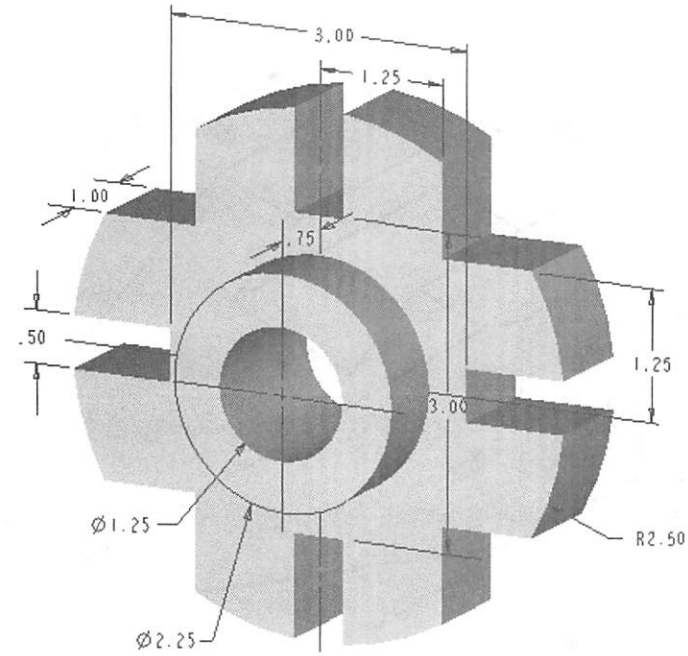
3. Angle Slider (Dimensions are in Millimeters)




4. Coupling Base (Dimensions are in inches.)



5. Indexing Guide (Dimensions are in inches.)




6. L-Bracket (Dimensions are in inches.)