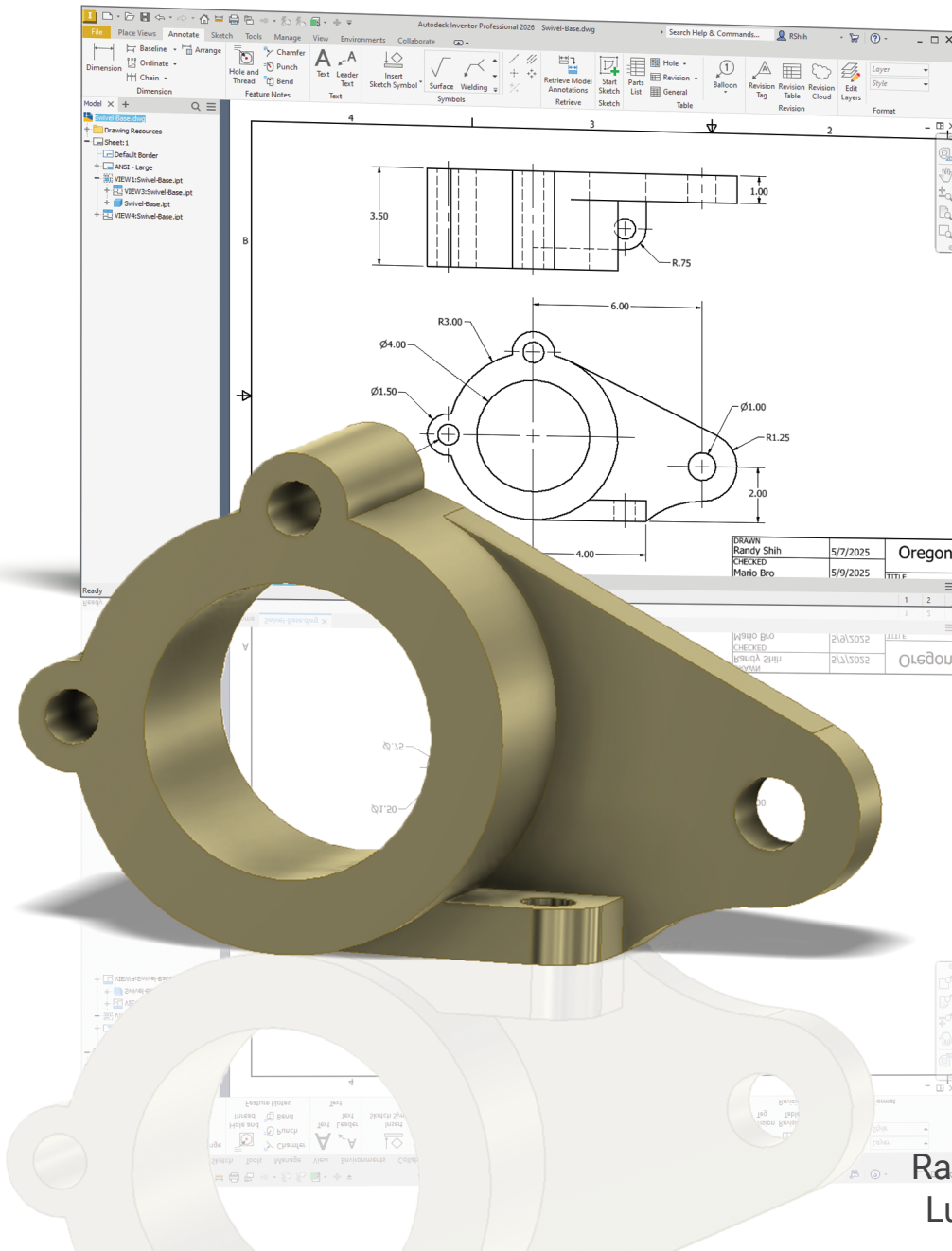


Parametric Modeling with Autodesk® Inventor® 2026

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Randy H. Shih
Luke Jumper



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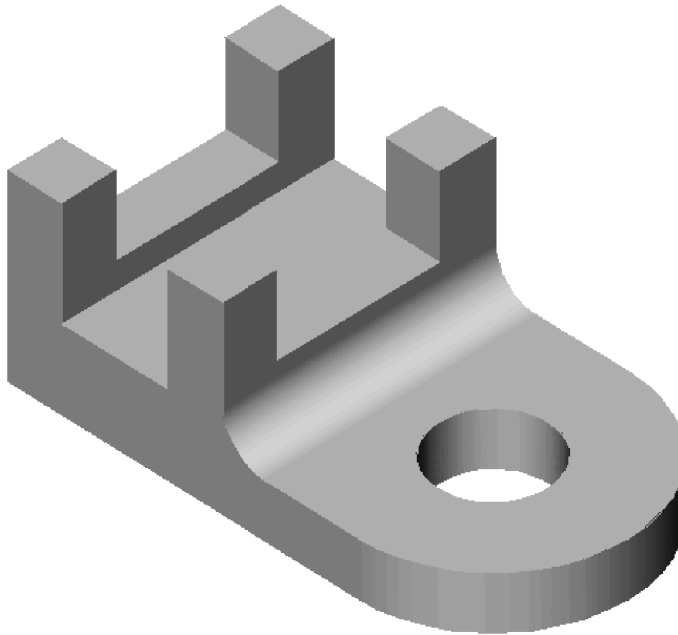
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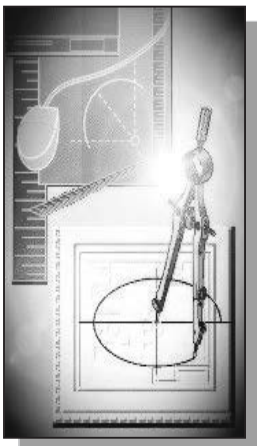
Chapter 4

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



Autodesk Inventor Certified User Exam Objectives Coverage

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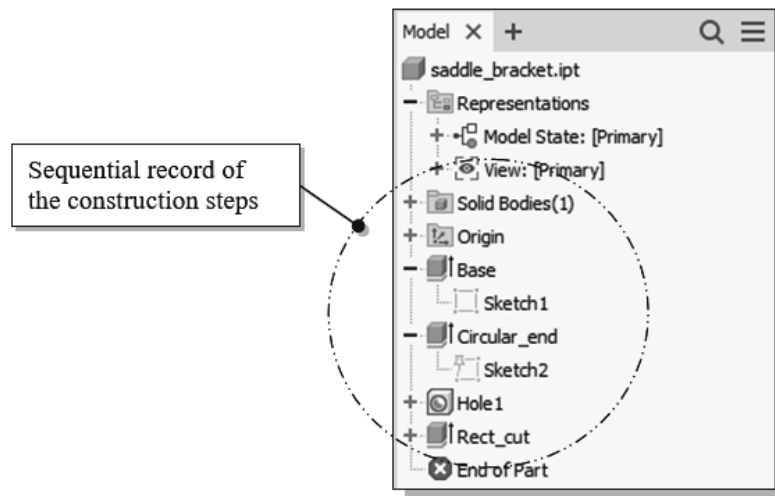
Section 4: Parts

Objectives: Creating parts, Work Features, Pattern Features, Part Properties.

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Introduction

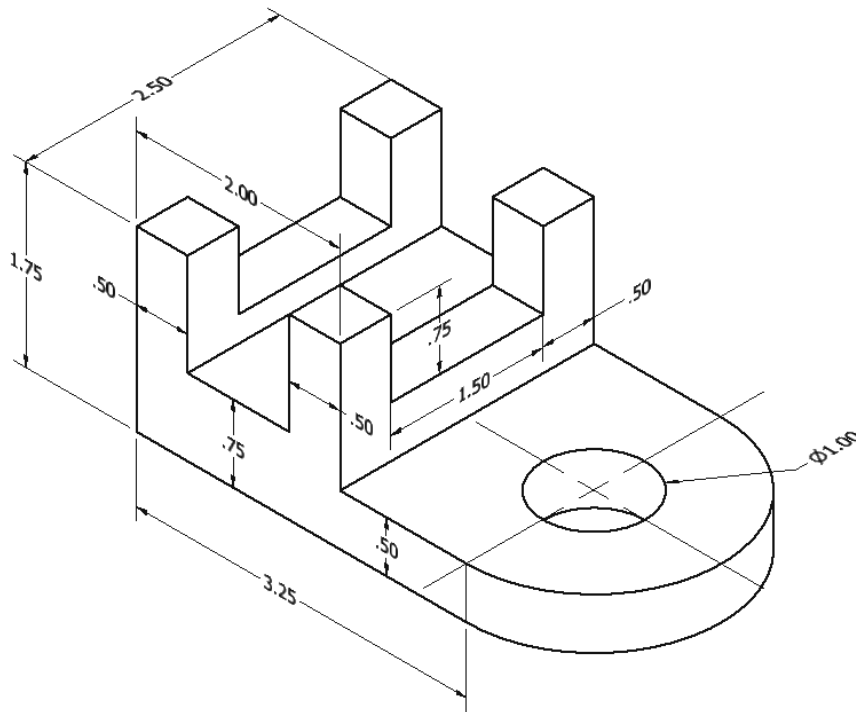
In Autodesk Inventor, the **design intents** are embedded into features in the **history tree**. The structure of the model history tree resembles that of a **CSG binary tree**. A CSG binary tree contains only *Boolean relations*, while the **Autodesk Inventor history tree** contains all features, including *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 in **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.

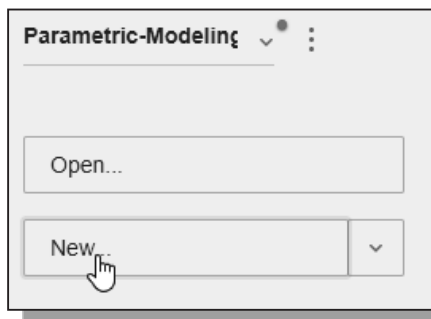
The Saddle Bracket Design



- ❖ 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? Would you organize the features differently if additional fillets were to be added in the design? 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.

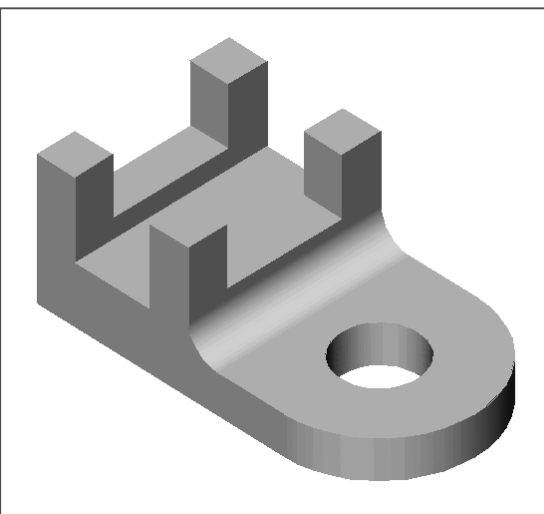
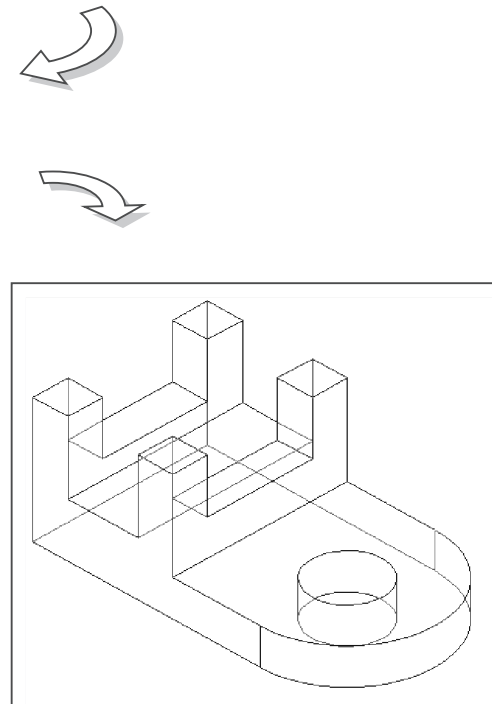
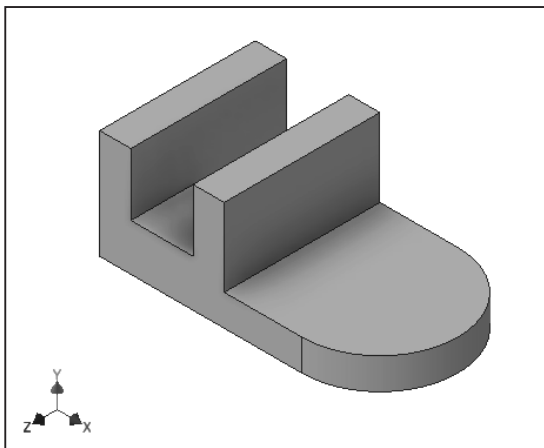
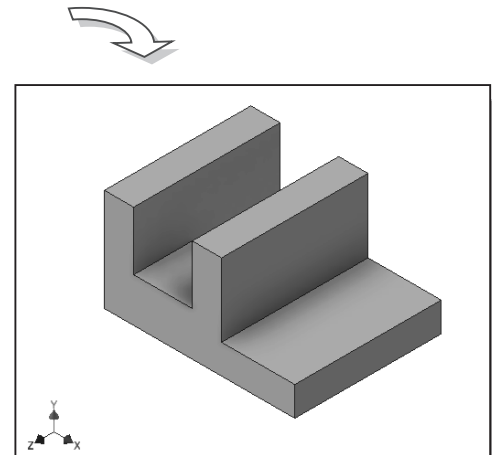
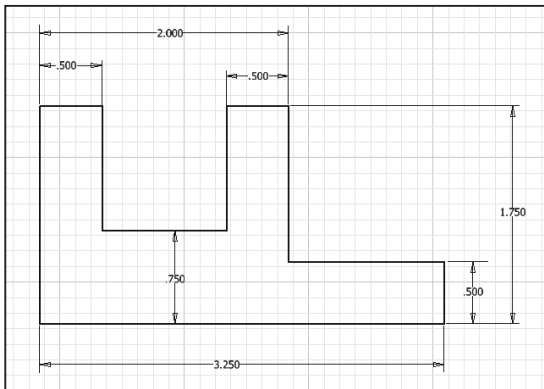
Starting Autodesk Inventor

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.

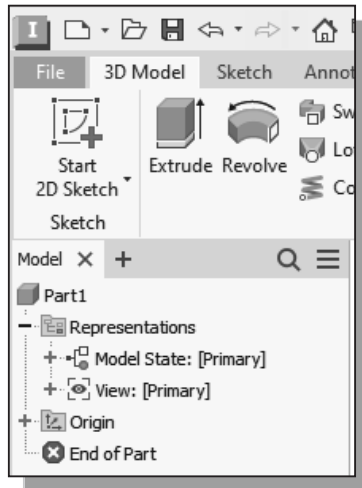


2. Once the program is loaded into memory, select the **New File** icon with a single click of the left-mouse-button as shown.
3. Select the **en-US->English** tab, and in the *Part template* area select **Standard(in).ipt**.
4. Click **Create** in the *New File* dialog box to accept the selected settings to start a new model.

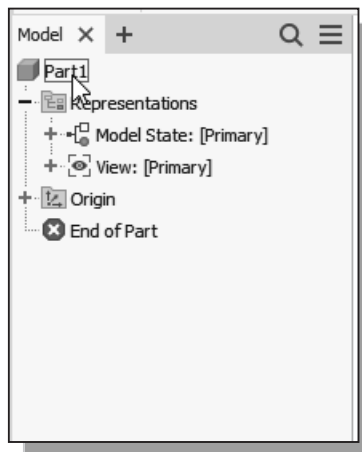
Modeling Strategy



The Autodesk Inventor Browser

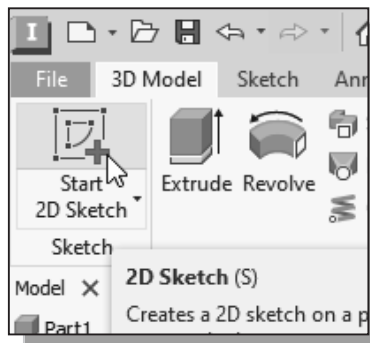


- In the Autodesk Inventor screen layout, the **browser** is located to the left of the graphics window. Autodesk Inventor can be used for part modeling, assembly modeling, part drawings, and assembly presentation. The **browser** window provides a visual structure of the features, constraints, and attributes 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.



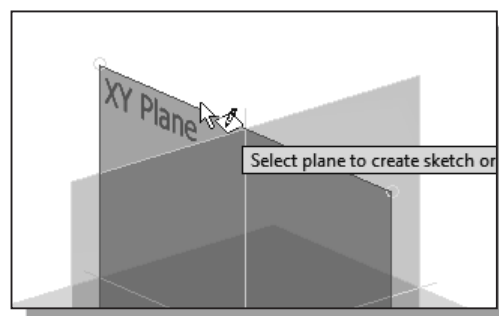
- The first item displayed in the **browser** is the name of the part, which is also the file name. By default, the name “Part1” is used when we first started Autodesk Inventor. The **browser** can also be used to modify parts and assemblies by moving, deleting, or renaming items within the hierarchy. 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.

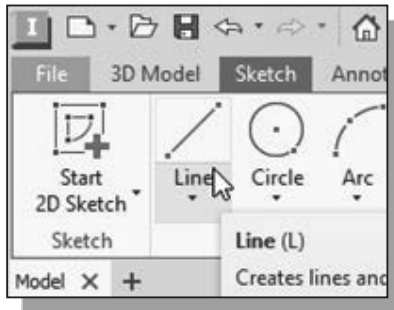
Creating the Base Feature



1. Move the graphics cursor to the **Start 2D Sketch** icon in the **Sketch toolbar** under the **3D Model tab**. 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.

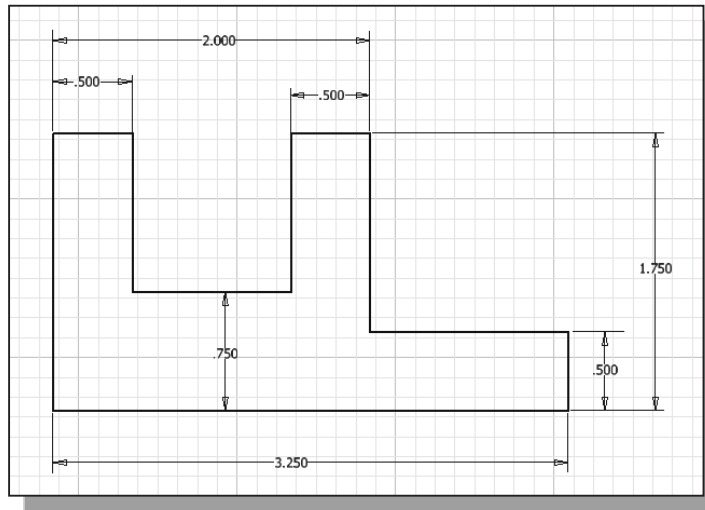
2. Move the cursor over the edge of the **XY Plane** in the graphics area. When the **XY Plane** is highlighted, click once with the **left-mouse-button** to select the **Plane** as the sketch plane for the new sketch.



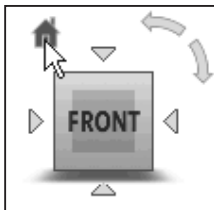


3. Select the **Line** icon by clicking once with the left-mouse-button; this will activate the **Line** command.

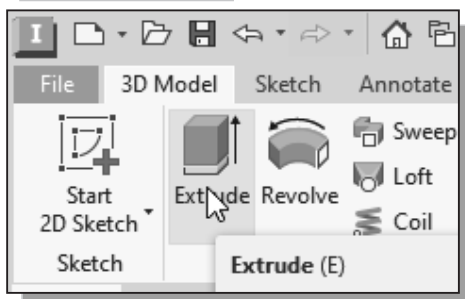
4. On your own, create and adjust the geometry by adding and modifying dimensions as shown below.



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

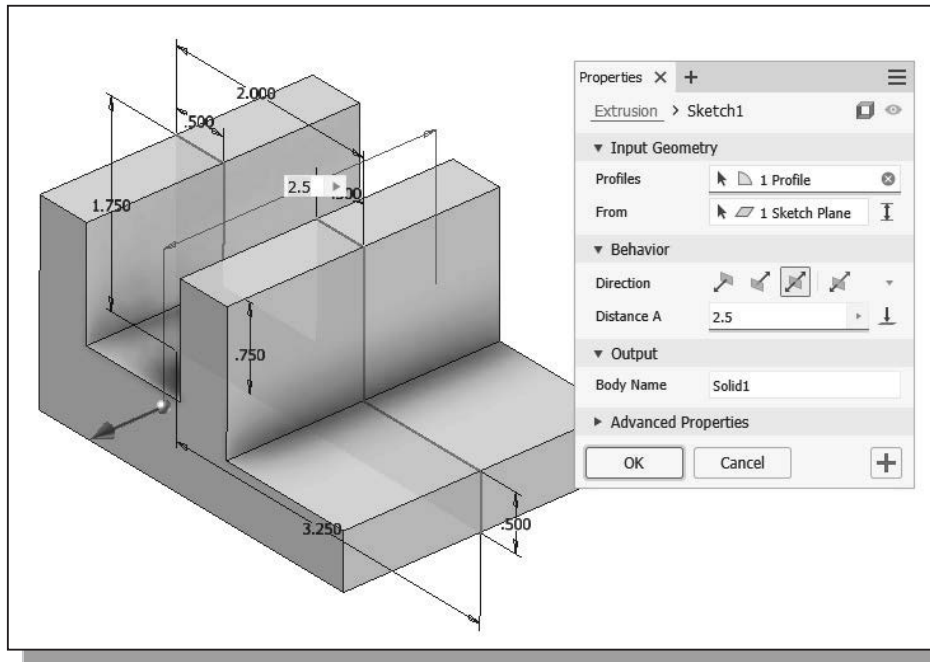


6. On your own, use the dynamic viewing functions to view the sketch. Click the home view icon to change the display to the *isometric* view before proceeding to the next step.

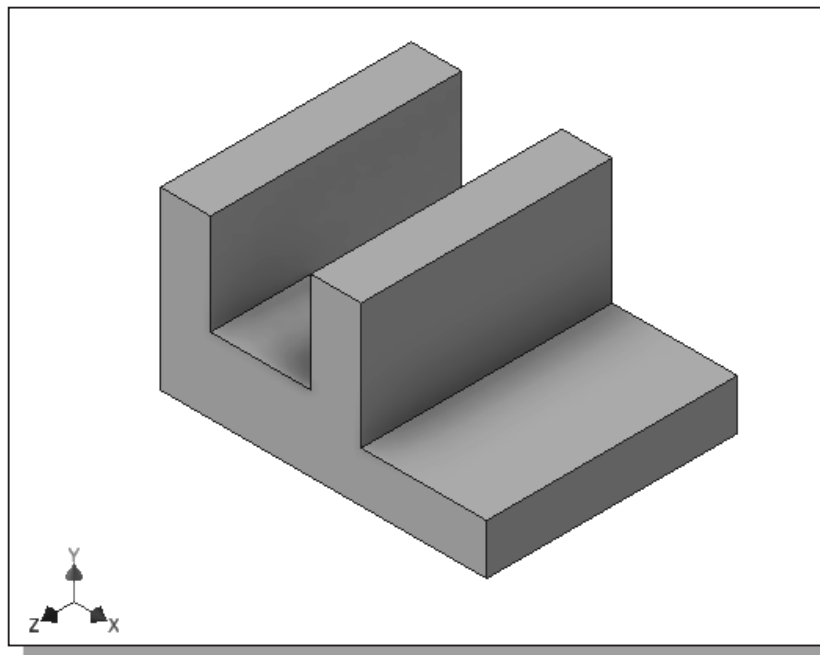


7. In the *Sketch* toolbar under the *3D Model* tab, select the **Extrude** command by left-clicking on the icon.

8. In the *Distance* option box, enter **2.5** as the total extrusion distance.
9. In the *Extrude* pop-up window, left-click once on the **Symmetric** icon. The **Symmetric** option allows us to extrude in both directions of the sketched profile.

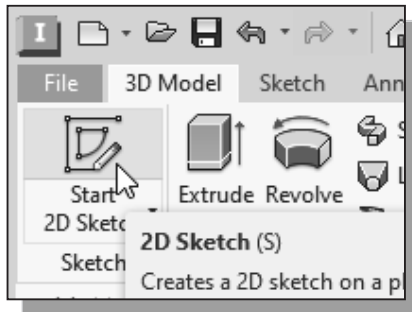


10. Click on the **OK** button to accept the settings and create the base feature.

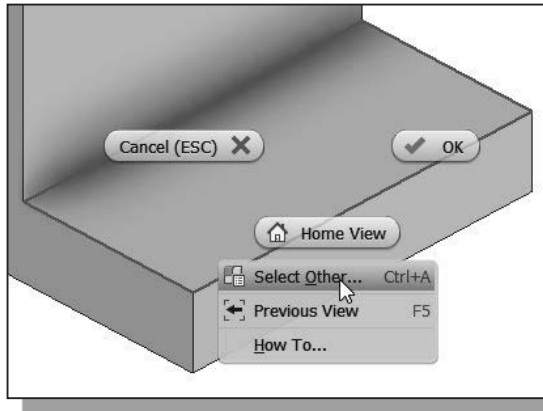


- On your own, use the *Dynamic Viewing* functions to view the 3D model. Also notice the extrusion feature is added to the *Model Tree* in the *browser* area.

Adding the Second Solid Feature

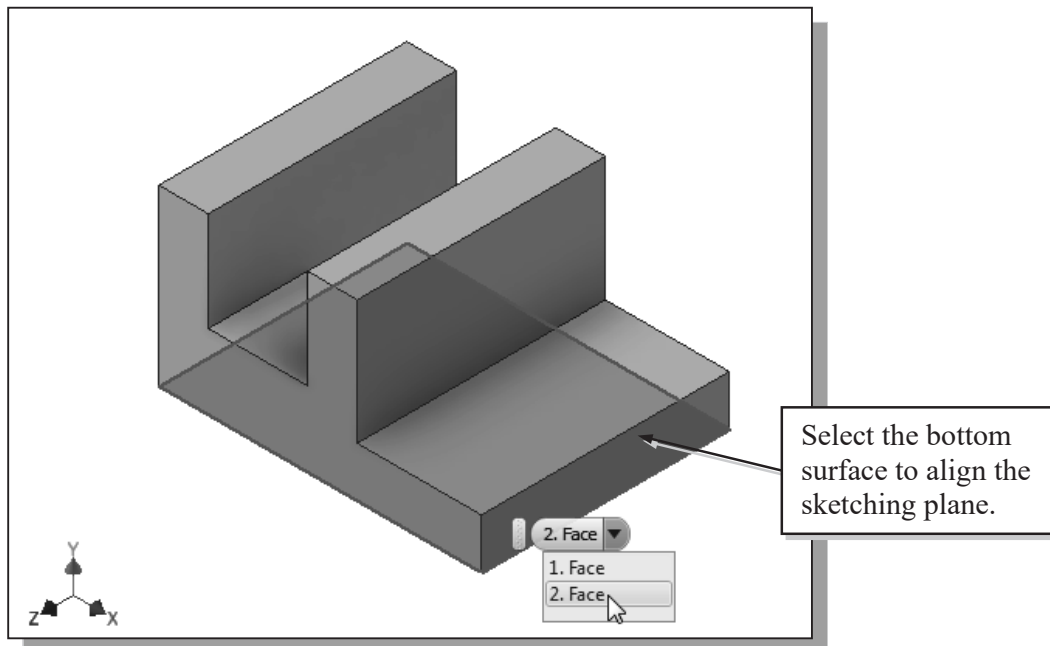


1. In the *Sketch toolbar* under the *3D Model tab* select the **Start 2D Sketch** command by left-clicking once on the icon.



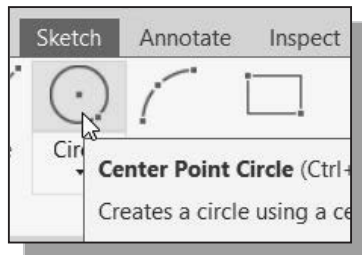
2. In the *Status Bar* area, the message “*Select plane to create sketch or an existing sketch to edit.*” is displayed. Move the cursor inside the upper horizontal face of the 3D object as shown below.
3. Click once with the **right-mouse-button** to bring up the option menu and choose **Select Other** to switch to the next feasible choice.

4. On your own, click on the down arrow to examine all possible surface selections.

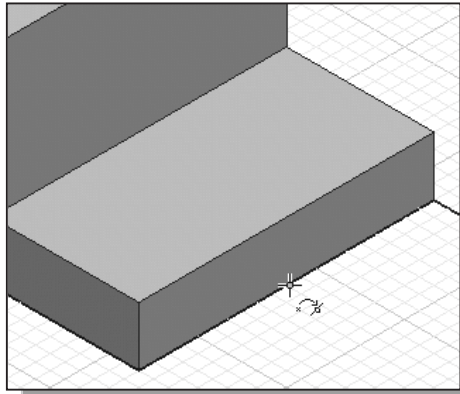


5. Select the **bottom horizontal face** of the solid model when it is highlighted as shown in the above figure.

Creating a 2D Sketch

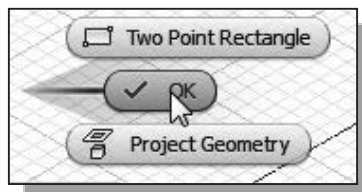
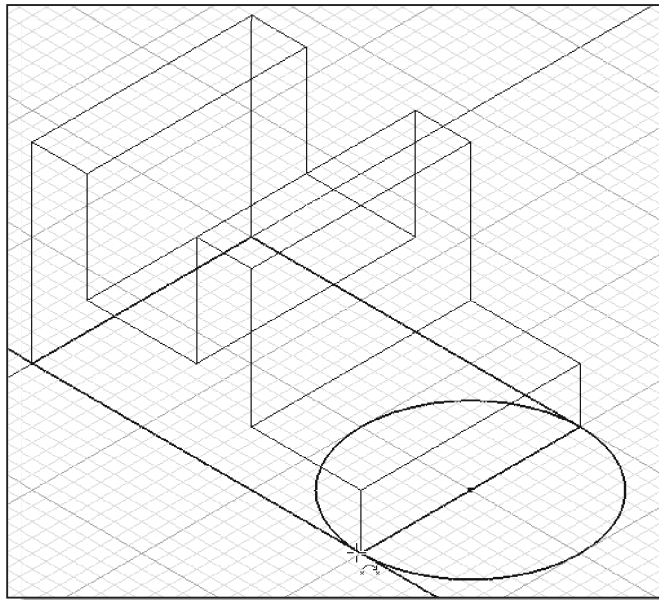


1. Select the **Center Point Circle** command by clicking once with the left-mouse-button on the icon in the *Sketch* tab.



- We will align the center of the circle to the midpoint of the base feature.
2. On your own, use the snap to midpoint option to pick the midpoint of the edge when the midpoint is displayed with GREEN color as shown in the figure. (Hit [F6] to set the display orientation if necessary.)

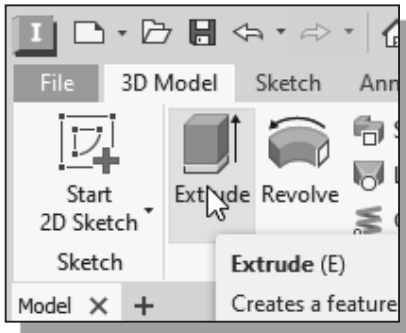
3. Select the front corner of the base feature to create a circle as shown below.



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



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

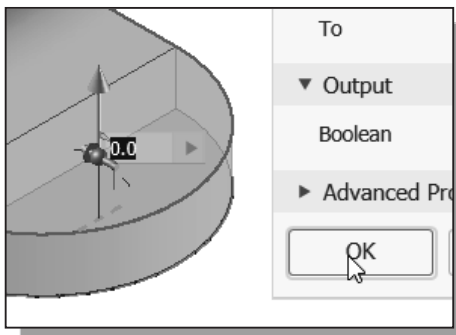
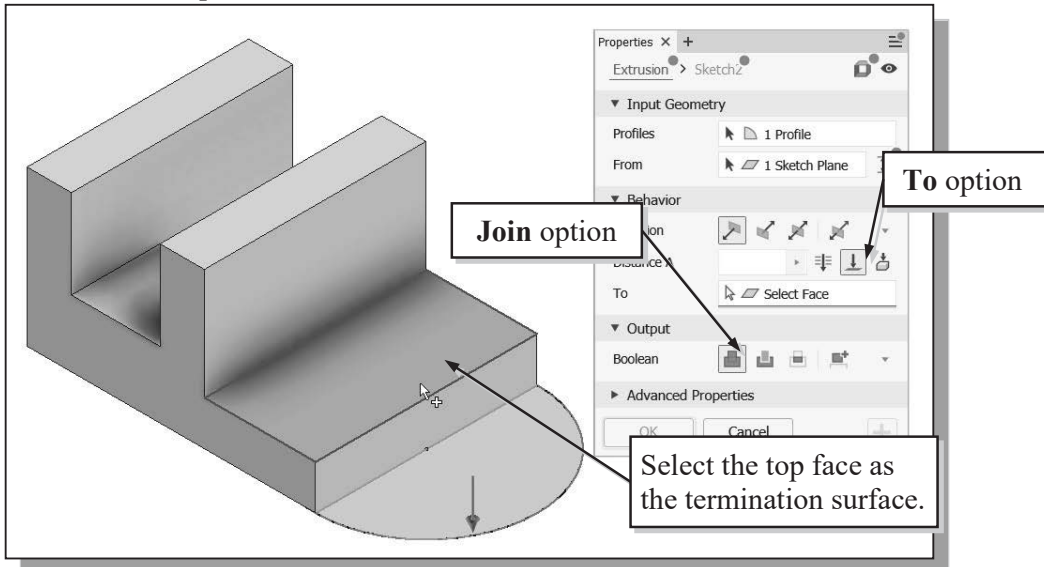


6. In the *Features* toolbar (the toolbar that is located to the left side of the graphics window), select the **Extrude** command by clicking the left-mouse-button on the icon.
7. Move the cursor to the outside **semi-circle** we just created and left-click once to select the region as the **profile** to be extruded.

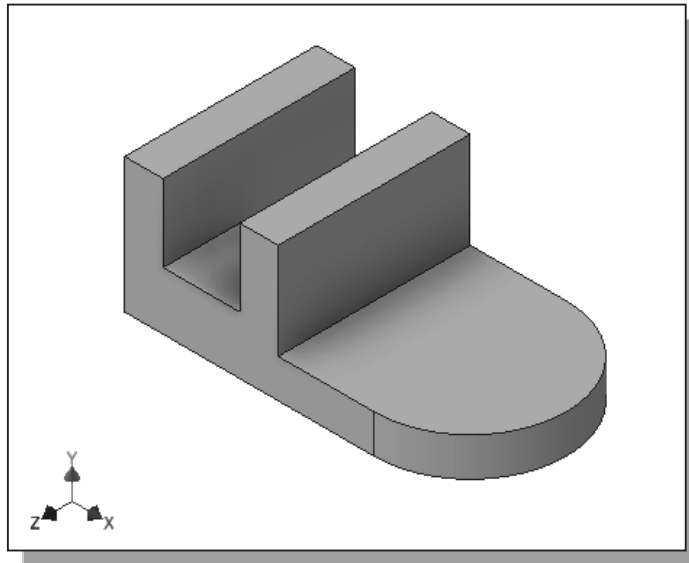
8. In the *Extrude* pop-up control, set the operation option to **Join**.

9. Also set the *Distance* option to **To Selected Face** as shown below.

10. Select the top face of the base feature as the termination surface for the extrusion.

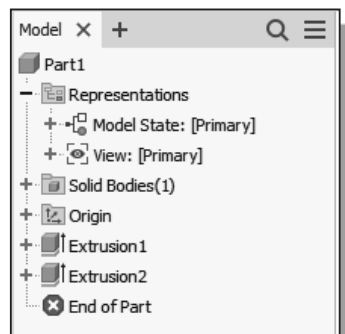


11. Click on the **OK** button to proceed with the *Join* operation.

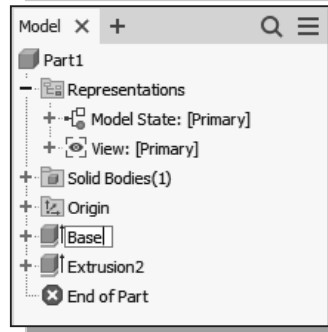


Renaming the Part Features

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.



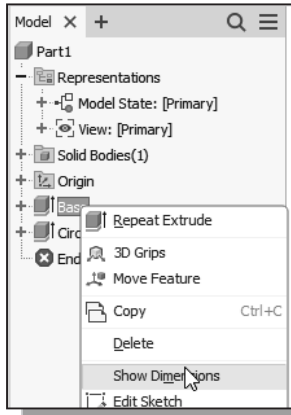
1. Select the first extruded feature in the *model browser* area by left-clicking once on the name of the feature, **Extrusion1**. Notice the selected feature is highlighted in the graphics window.



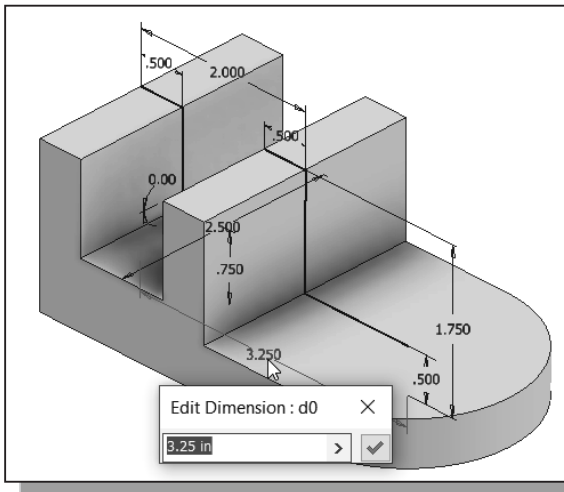
2. Left-click again on the feature name to enter the *Edit* mode as shown.
3. Enter **Base** as the new name for the first extruded feature.
4. On your own, rename the second extruded feature to **Circular_End**.

Adjusting the Width of the Base Feature

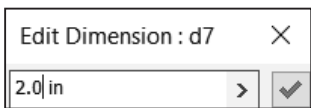
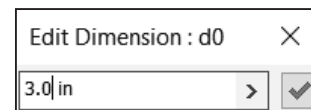
One of the main advantages of parametric modeling is the ease of performing part modifications at any time in the design process. Part modifications can be done through accessing the features in the history tree. Autodesk Inventor remembers the history of a part, including all the rules that were used to create it, so that changes can be made to any operation that was performed to create the part. For our *Saddle Bracket* design, we will reduce the size of the base feature from 3.25 inches to 3.0 inches, and the extrusion distance to 2.0 inches.



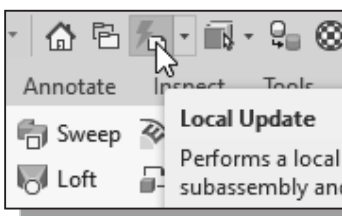
1. Select the first extruded feature, **Base**, in the *browser* area. Notice the selected feature is highlighted in the graphics window.
2. Inside the *browser* area, **right-mouse-click** on the first extruded feature to bring up the option menu and select the **Show Dimensions** option in the pop-up menu.



3. All dimensions used to create the **Base** feature are displayed on the screen. Select the overall width of the **Base** feature, the **3.25** dimension value, by double-clicking on the dimension text as shown.
4. Enter **3.0** in the *Edit Dimension* window.

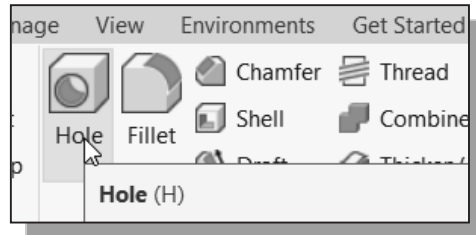


5. On your own, repeat the above steps and modify the extruded distance from **2.5** to **2.0**.

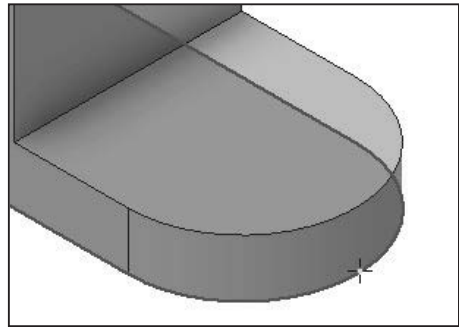


6. Click **Local Update** in the *Quick Access Toolbar*.
 - Note that Autodesk Inventor updates the model by re-linking all elements used to create the model. Any problems or conflicts that occur will also be displayed during the updating process.

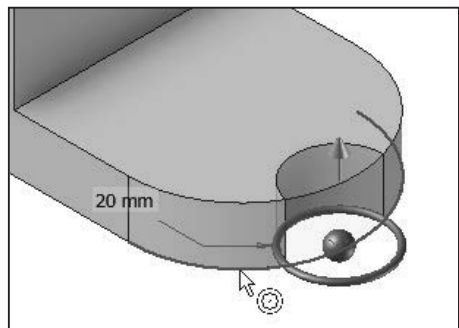
Adding a Placed Feature



1. In the *Sketch tab*, select the **Hole** command by left-clicking on the icon.

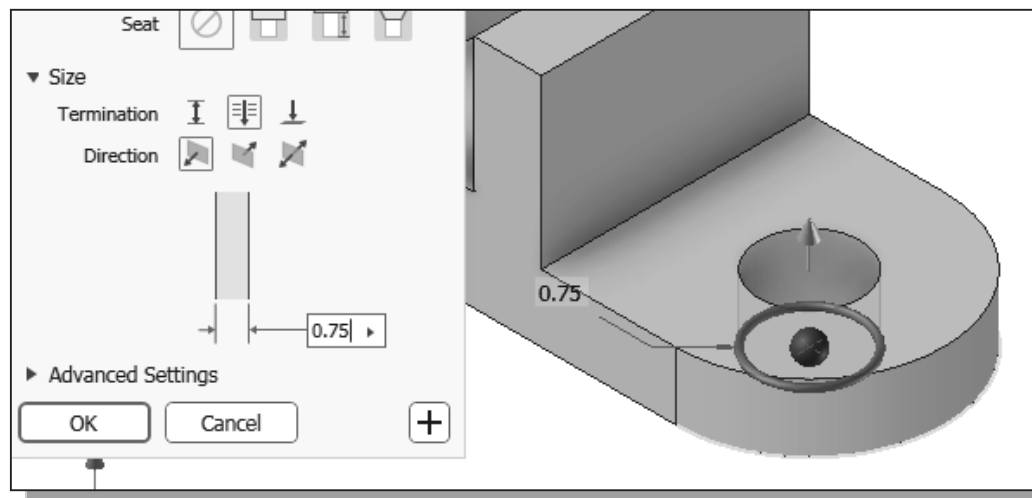


2. Pick the **bottom plane** of the solid model as the placement plane as shown.

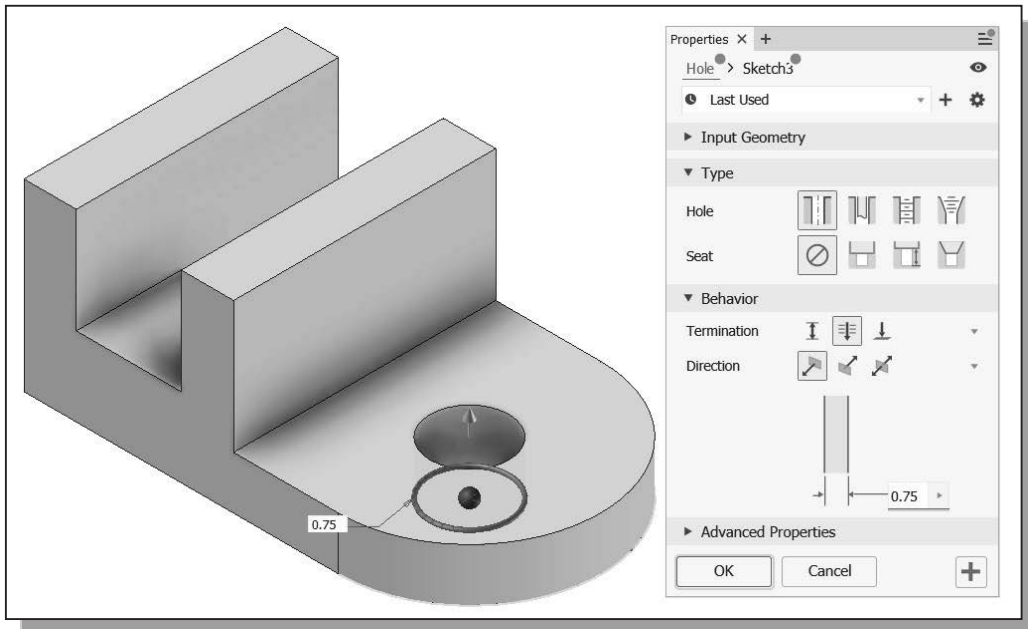


3. Pick the **bottom arc** and notice the *concentric reference symbol* appears indicating the center of the hole will be aligned to the center of the selected arc.

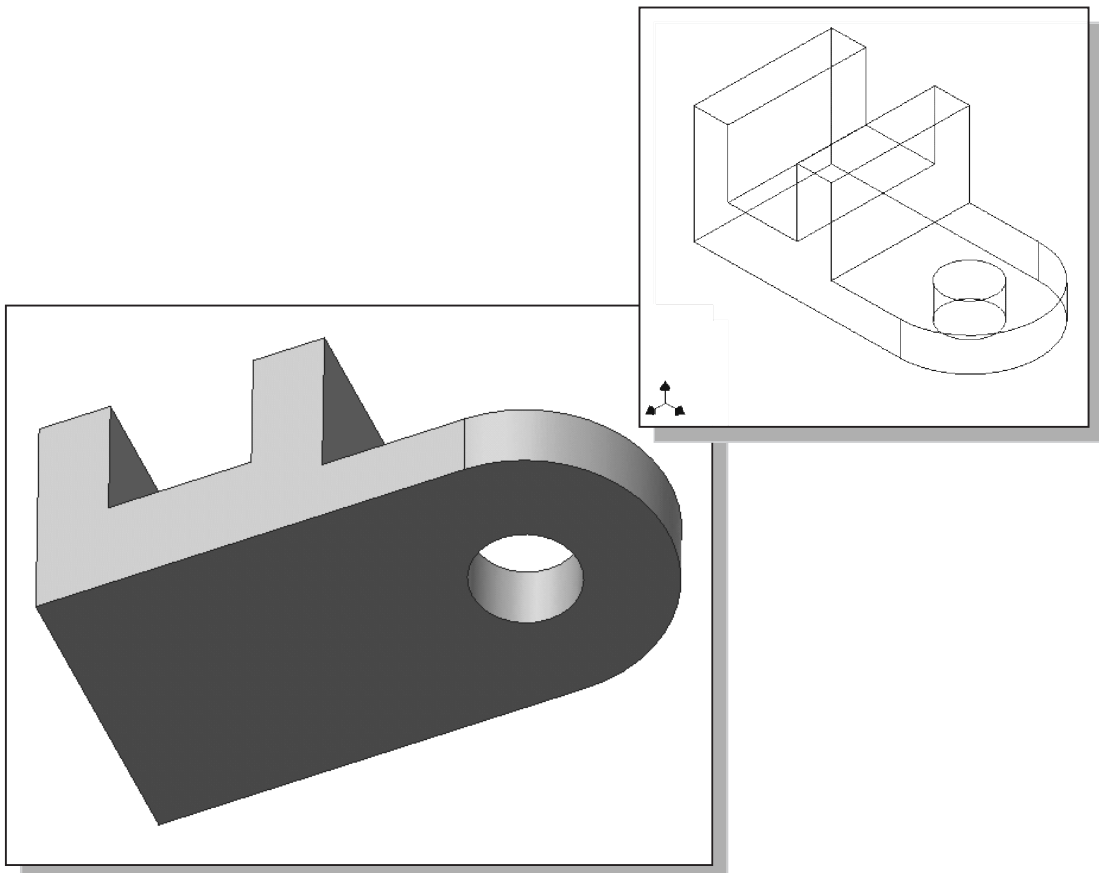
4. Set the hole diameter to **0.75 in** as shown.



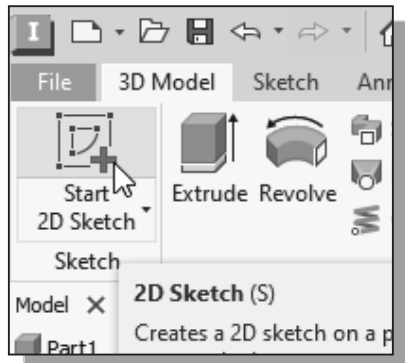
5. Set the termination option to **Through All** as shown.



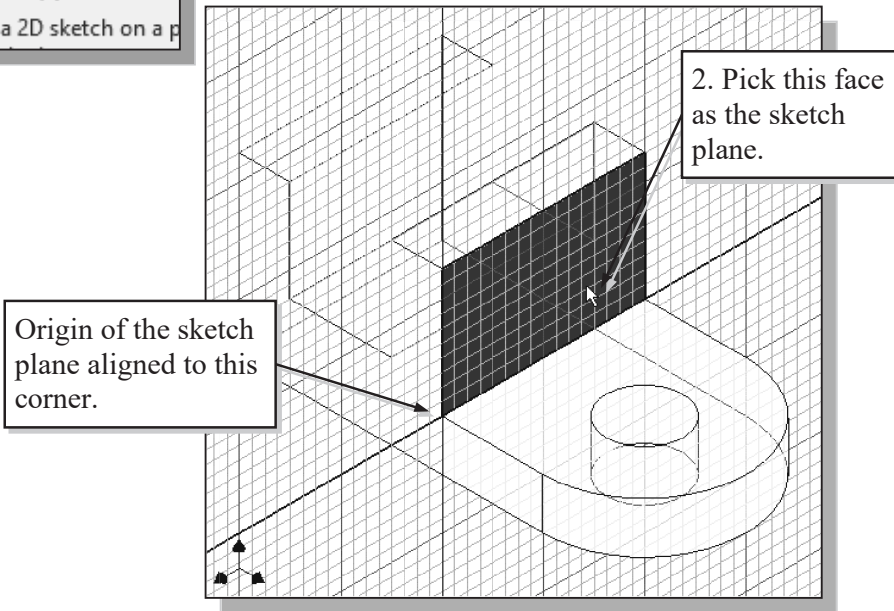
6. Click **OK** to accept the settings and create the *Hole* feature.



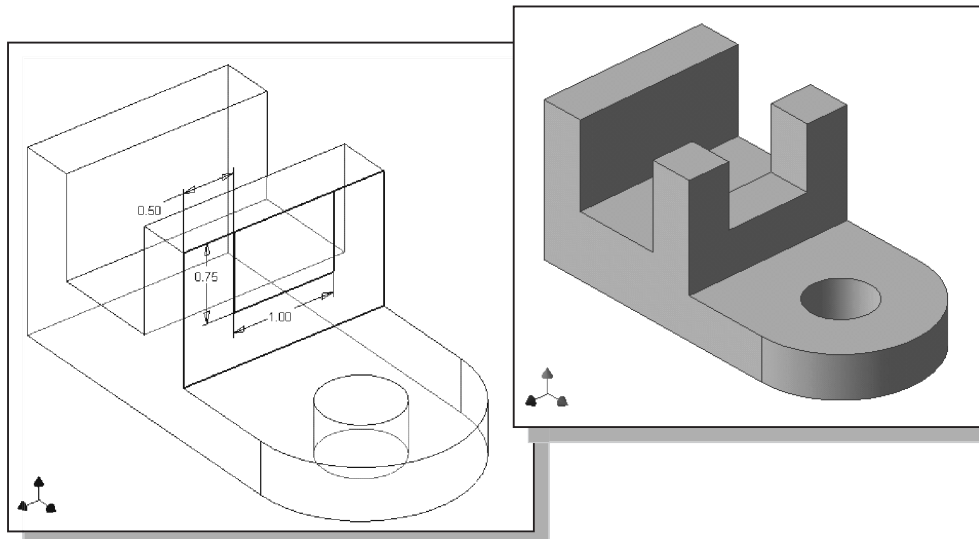
Creating a Rectangular Cut Feature



1. In the *Sketch toolbar* under the *3D Model tab* select the **Start 2D Sketch** command by left-clicking once on the icon.
2. Pick the **vertical face** of the solid as shown. (Note the alignment of the origin of the sketch plane.)

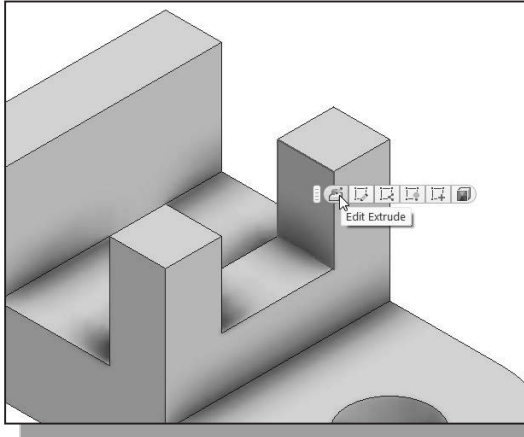


- On your own, create a rectangular cut (**1.0 x 0.75**) feature (**To Next** option) as shown and rename the feature to **Rect_Cut**.

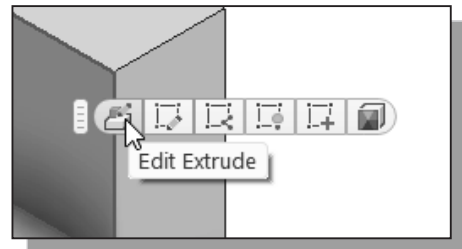


History-Based Part Modifications

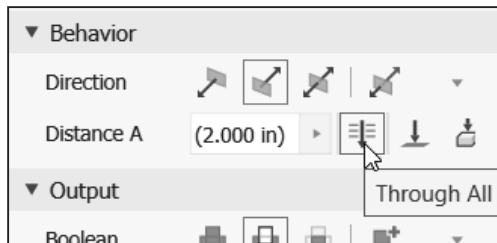
Autodesk Inventor uses the *history-based part modification* approach, which enables us to make modifications to the appropriate features and re-link the rest of the history tree without having to reconstruct the model from scratch. We can think of it as going back in time and modifying some aspects 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 depth of the rectangular cutout.



1. In the *Graphics* window, select the last cut feature, **Rect_Cut**, on the inside surface of the feature as shown.
2. Select **Edit Feature** in the pop-up menu. Notice the *Extrude* dialog box appears on the screen.

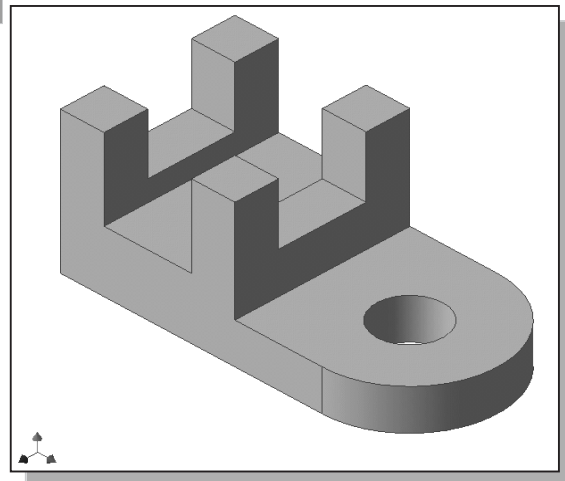


- Note that the feature pop-up options are also available through the Model History Tree, as shown on the next page.



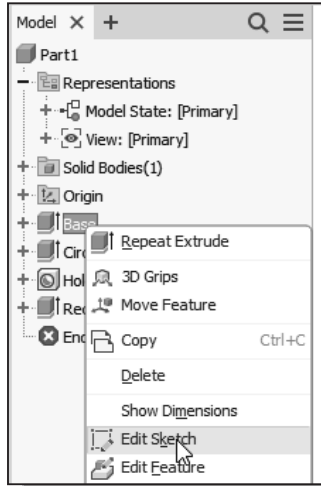
3. In the *Extrude* dialog box, set the termination *Extents* to the **Through All** option.
4. Click on the **OK** button to accept the settings.

- As can be seen, the history-based modification approach is very straight forward, and it only took a few seconds to adjust the cut feature to the **Through All** option.

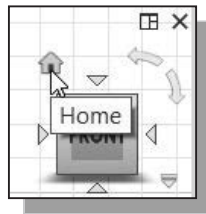


A Design Change

Engineering designs usually go through many revisions and changes. Autodesk Inventor provides an assortment of tools to handle design changes quickly and effectively. We will demonstrate some of the tools available by changing the **Base** feature of the design.

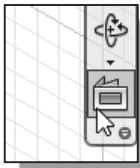


1. In the *browser* window, select the **Base** feature by left-clicking once on the name of the feature.
2. In the *browser*, right-click once on the **Base** feature to bring up the option menu; then pick **Edit Sketch** in the pop-up menu.



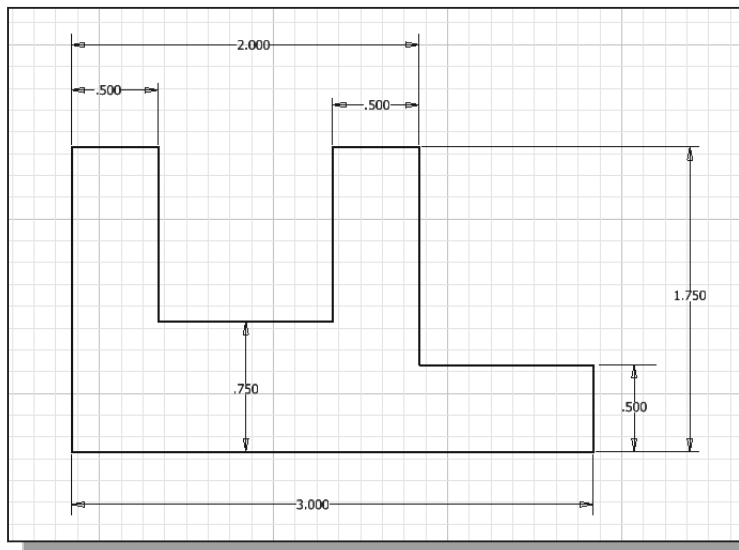
3. Click **Home** to reset the display to *Isometric*.

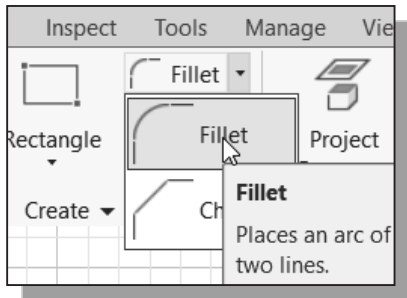
❖ Autodesk Inventor will now display the original 2D sketch of the selected feature in the graphics window. We have literally gone back to the point where we first created the 2D sketch. Notice the feature being modified is also highlighted in the desktop *browser*.



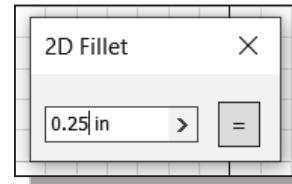
4. Click on the **Look At** icon in the *Standard* toolbar area.
 - The **Look At** command automatically aligns the *sketch plane* of a selected entity to the screen.

5. Select any line segment of the 2D sketch to reset the display to align to the 2D sketch.

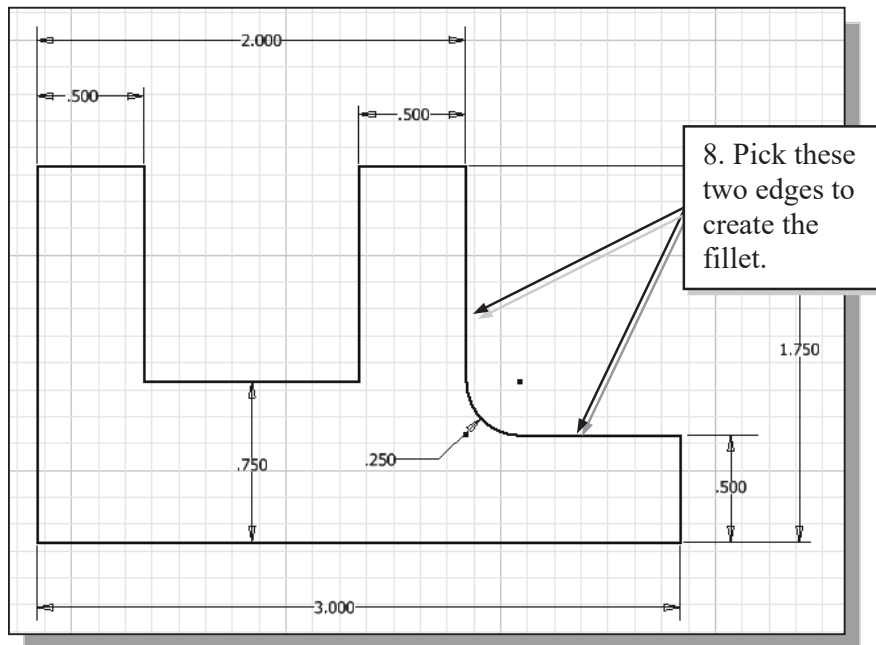




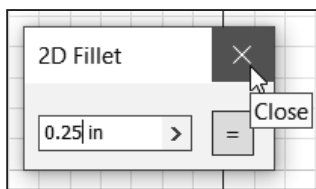
6. Select the **Fillet** command in the *2D Sketch* toolbar.
7. In the graphics window, enter **0.25** as the new radius of the fillet.



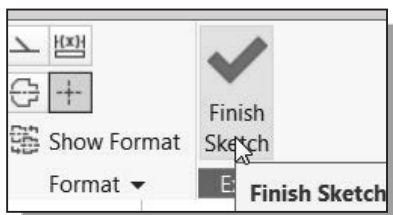
8. Select the two edges as shown to create the fillet.



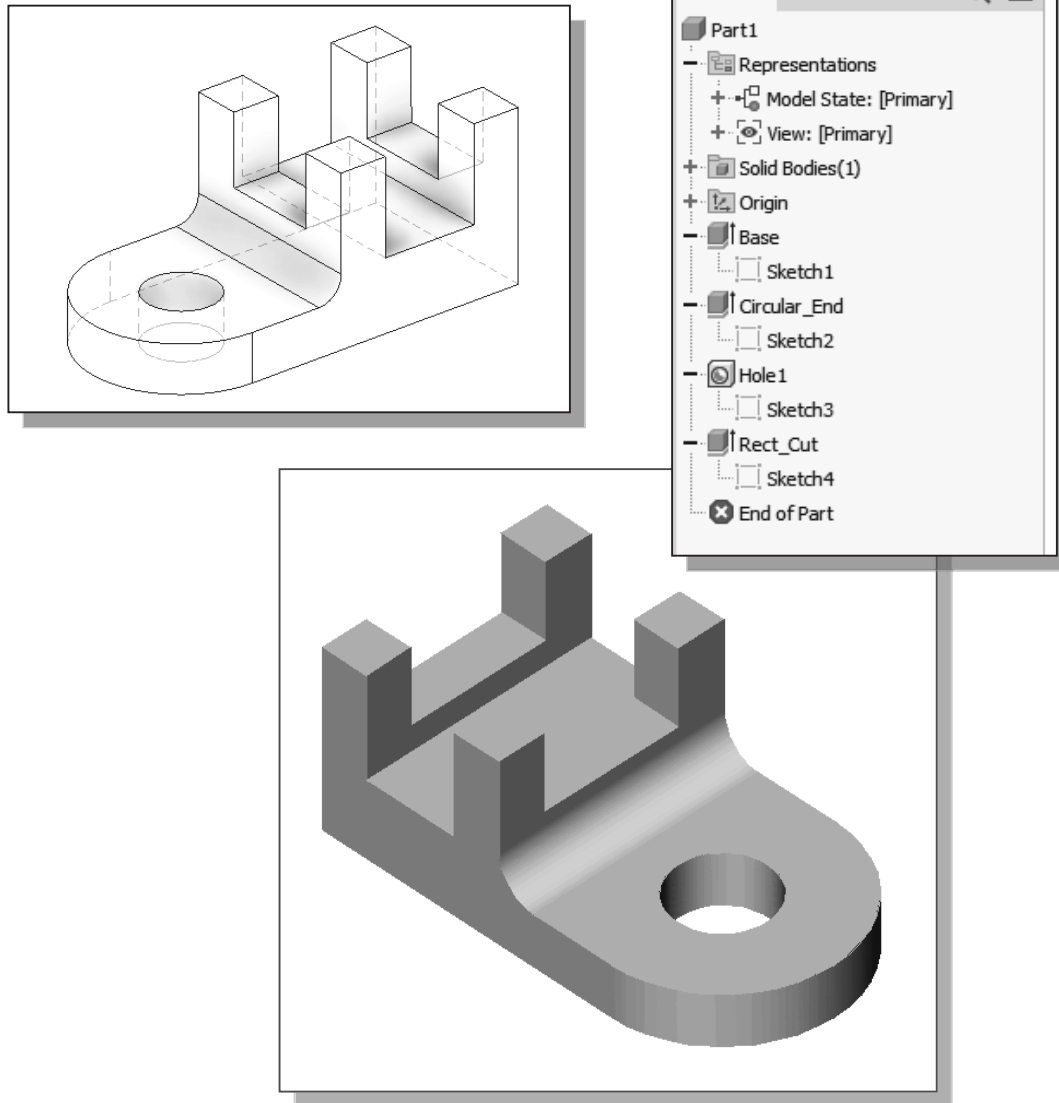
- Note that the fillet is created automatically with the dimension attached. The attached dimension can also be modified through the history tree.



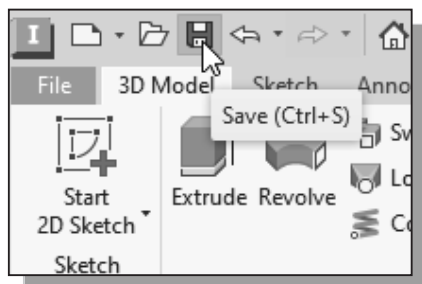
9. Click on the [X] icon in the *2D Fillet* window to end the Fillet command.



10. Select **Finish Sketch** in the *Ribbon* toolbar to end the Sketch option.



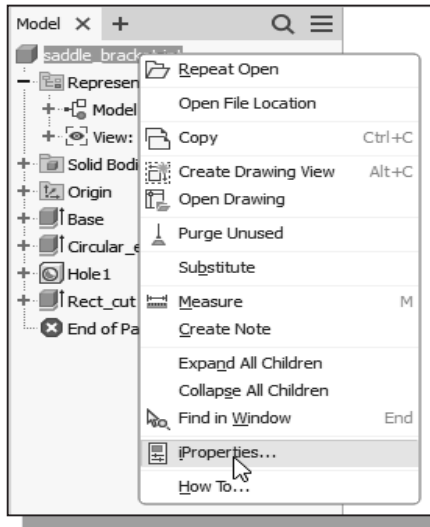
- ❖ In a typical design process, the initial design will undergo many analyses, testing, and reviews. The *history-based part modification* approach is an extremely powerful tool that enables us to quickly update the design. At the same time, it is quite clear that **PLANNING AHEAD** is also important in doing feature-based modeling.



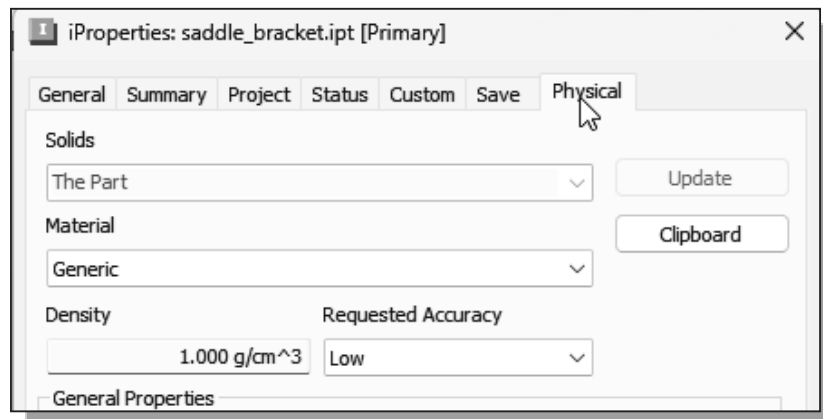
11. On your own, create the Chapter4 folder and save the model as ***Saddle_Bracket.ipt***.

Assigning and Calculating the Associated Physical Properties

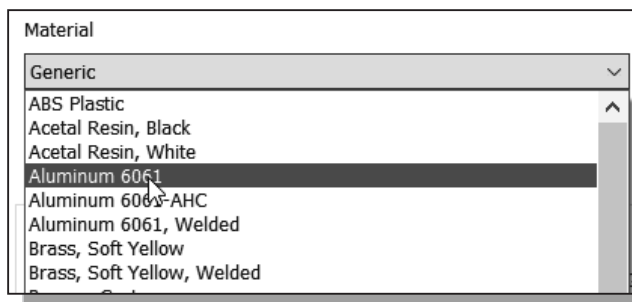
Autodesk Inventor models have properties called **iProperties**. The *iProperties* can be used to create reports and update assembly bills of materials, drawing parts lists, and other information. With *iProperties*, we can also set and calculate physical properties for a part or assembly using the material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



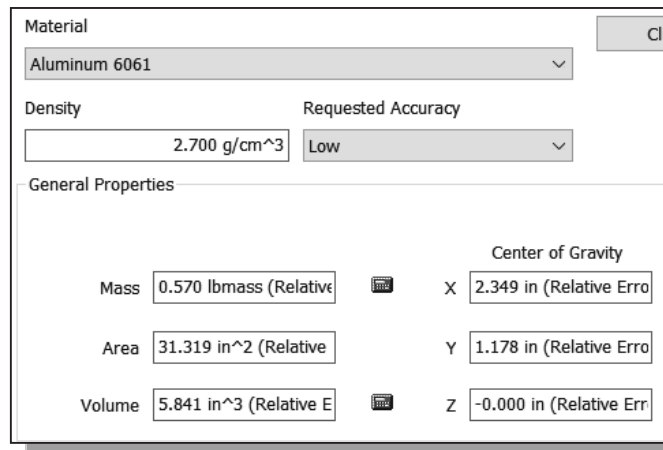
1. In the *browser*, **right-click** once on the *part name* to bring up the option menu; then pick **iProperties** in the *pop-up* menu.
2. On your own, look at the different information listed in the *iProperties* dialog box.
3. Click on the **Physical** tab; this is the page that contains the physical properties of the selected model.



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

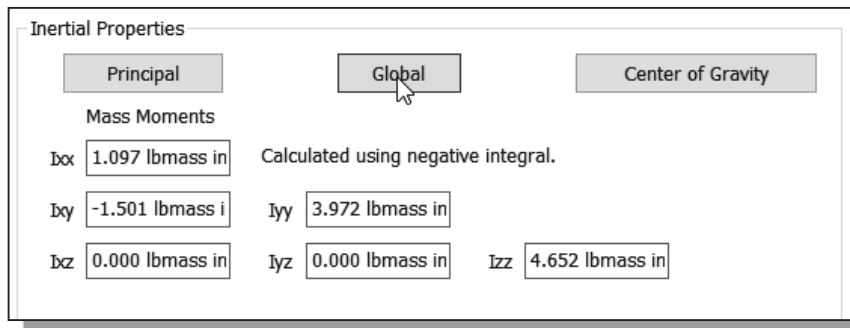


4. Click the down-arrow in the *Material* option to display the material list, and select **Aluminum-6061** as shown.

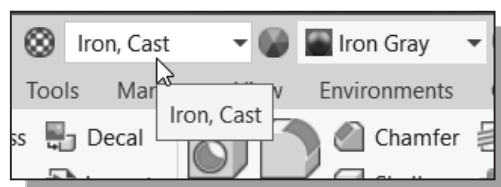
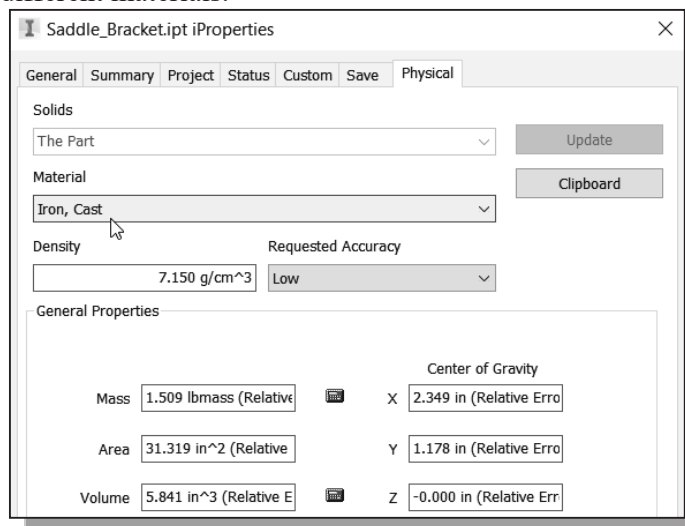


- ❖ Note the *General Properties* area now has the *Mass*, *Area*, *Volume* and the *Center of Gravity* information of the model based on the density of the selected material.

5. Click on the **Global** button to display the *Mass Moments Inertial* of the design.



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



- ❖ Also note the *Material* can be assigned through the quick access menu as shown.

Review Questions: (Time: 30 minutes.)

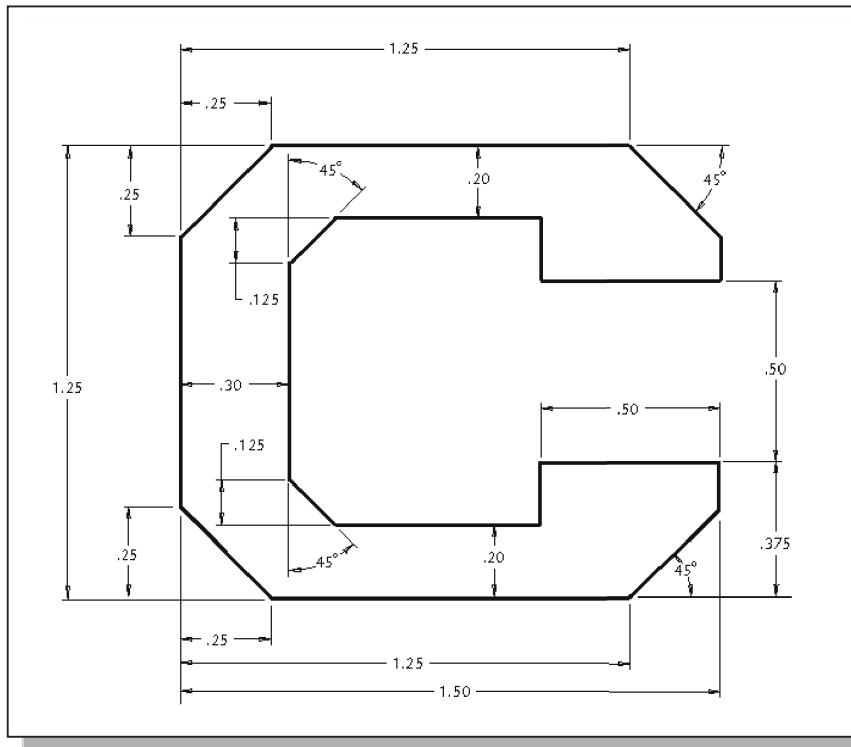
1. What are stored in the Autodesk Inventor *History Tree*?
2. When extruding, what is the difference between *Distance* and *Through All*?
3. Describe the *history-based part modification* approach.
4. What determines how a model reacts when other features in the model change?
5. Describe the steps 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 on the next page:

Ex.1)

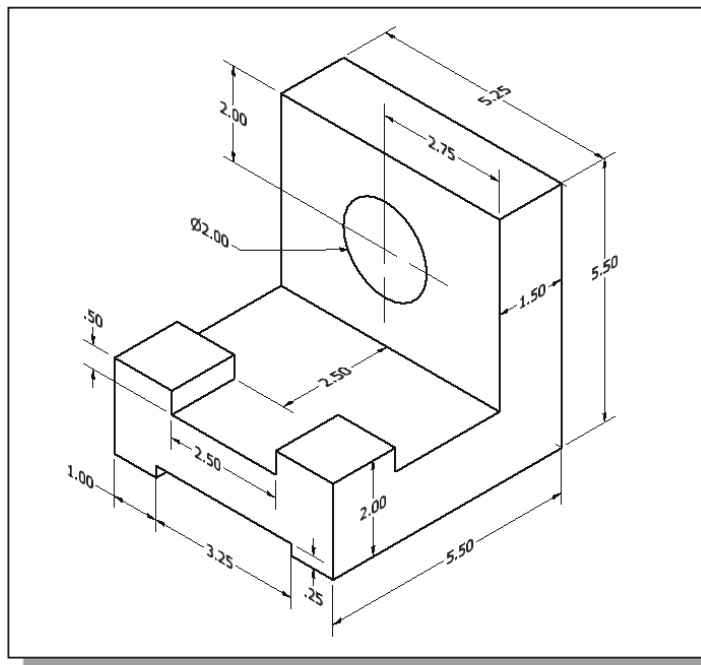
Ex.2)

Exercises: Create and save the exercises in the Chapter4 folder.
(Time: 180 minutes.)

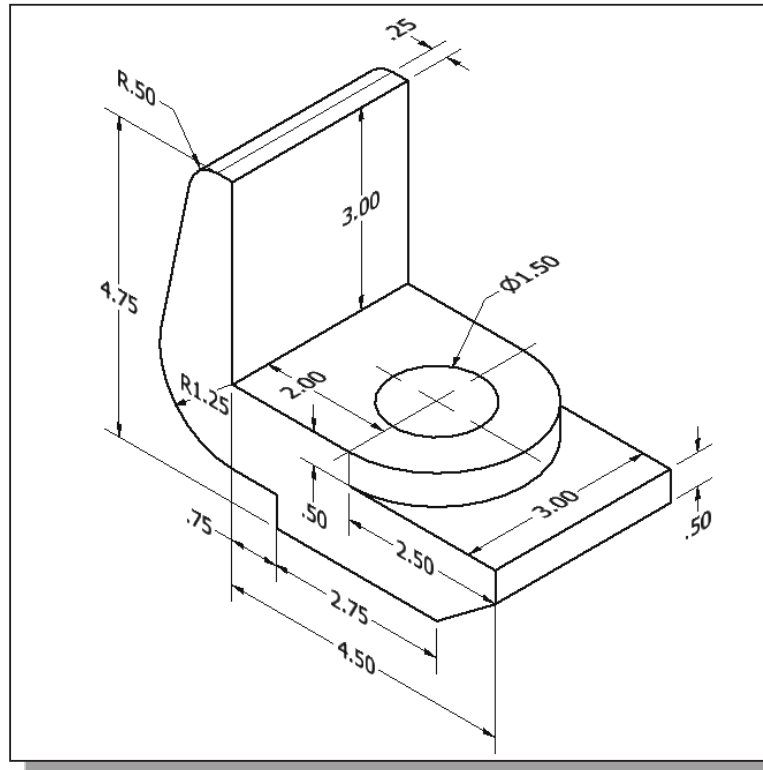
1. **C-Clip** (Dimensions are in inches. Plate thickness: **0.25 inches**.)



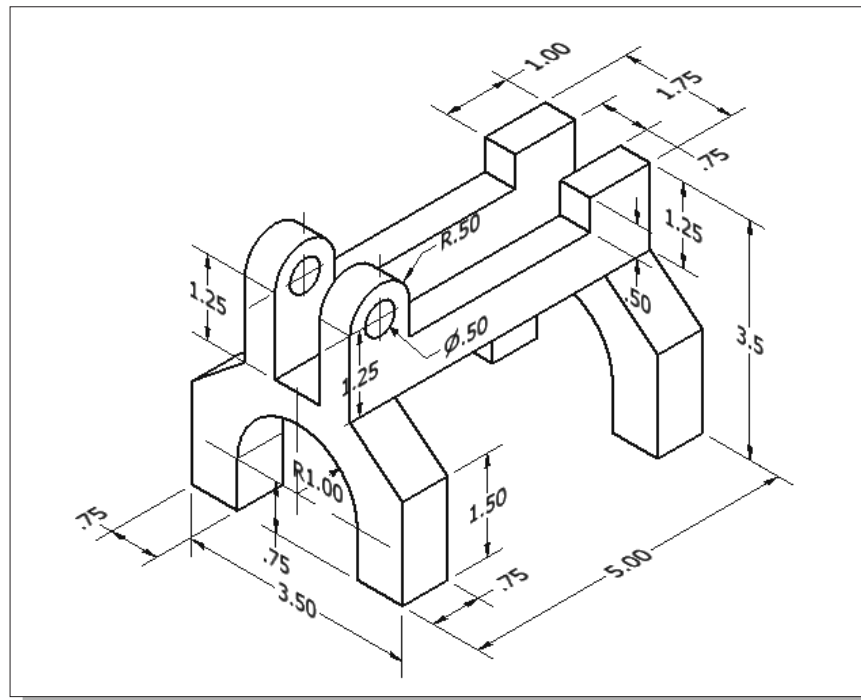
- ## 2. Tube Mount (Dimensions are in inches.)



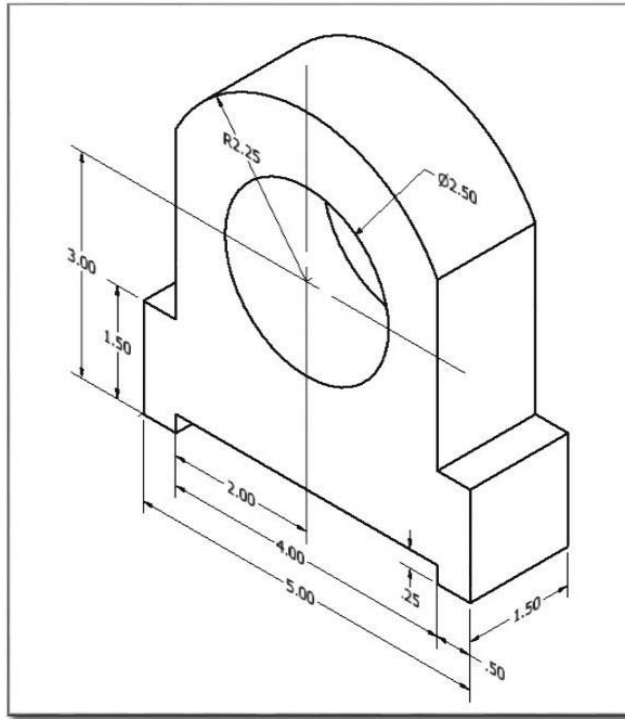
3. **Hanger Jaw** (Dimensions are in inches. Volume =?)



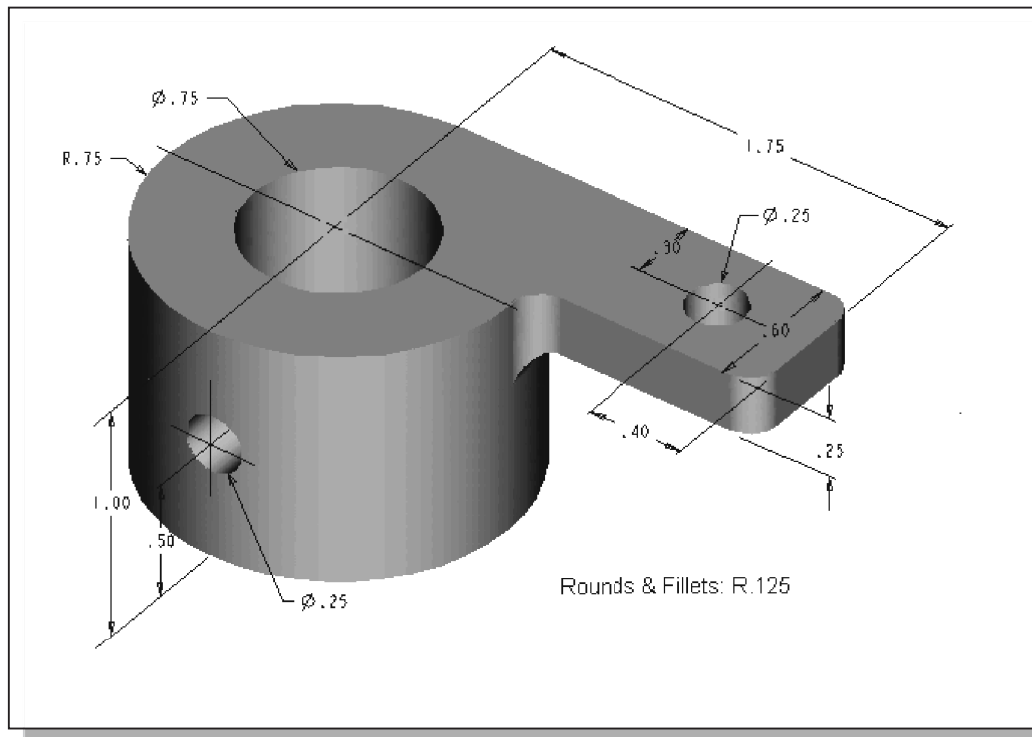
4. **Transfer Fork** (Dimensions are in inches. Material: **Cast Iron**. Volume =?)



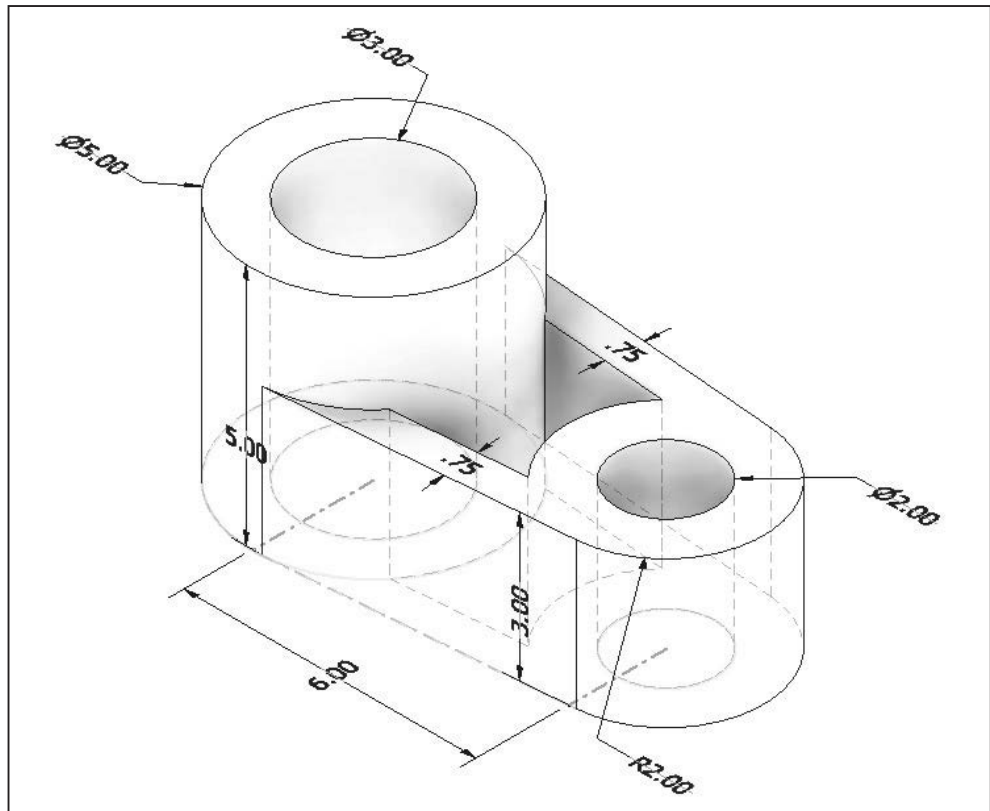
5. **Guide Slider** (Material: **Cast Iron**. Weight and Volume =?)



6. **Shaft Guide** (Material: **Aluminum-6061**. Mass and Volume =?)



7. **Idler Mount** (Dimensions are in inches. Center to center distance: 6.0)



Notes: