

# SOLIDWORKS® 2026

## Basic Tools

Getting Started with Parts,  
Assemblies and Drawings



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Visit the following websites to learn more about this book:



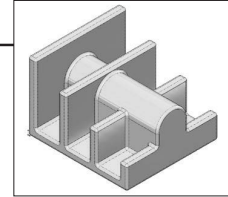
[amazon.com](https://www.amazon.com)

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[BARNES & NOBLE](https://www.barnesandnoble.com)

# CHAPTER 3

## Basic Solid Modeling — Extrude Options



### Basic Solid Modeling Extrude Options

Upon successful completion of this lesson, you will be able to:

- \* Sketch on planes and/or planar surfaces
- \* Use the sketch tools to construct geometry
- \* Add the geometric relations or constraints
- \* Add/modify dimensions
- \* Explore the different extrude options

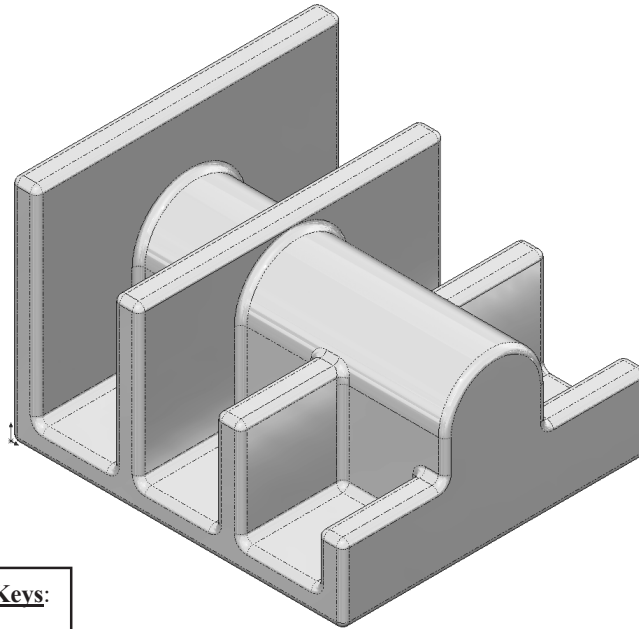
The following 5 basic steps will be demonstrated throughout this exercise:

- \* Select the sketch plane (or sketch face)
- \* Activate Sketch toolbar
- \* Sketch the profile using the sketch tools
- \* Define the profile with dimensions or relations
- \* Extrude the profile

Be sure to review the self-test questionnaires at the end of the lesson prior to moving to the next chapter.

# Basic Solid Modeling

## Extrude Options



### View Orientation Hot Keys:

Ctrl + 1 = Front View  
Ctrl + 2 = Back View  
Ctrl + 3 = Left View  
Ctrl + 4 = Right View  
Ctrl + 5 = Top View  
Ctrl + 6 = Bottom View  
Ctrl + 7 = Isometric View  
Ctrl + 8 = Normal To  
Selection

Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

### Tools Needed:



Insert Sketch



Line



Circle



Add Geometric  
Relations



Dimension



Sketch Fillet



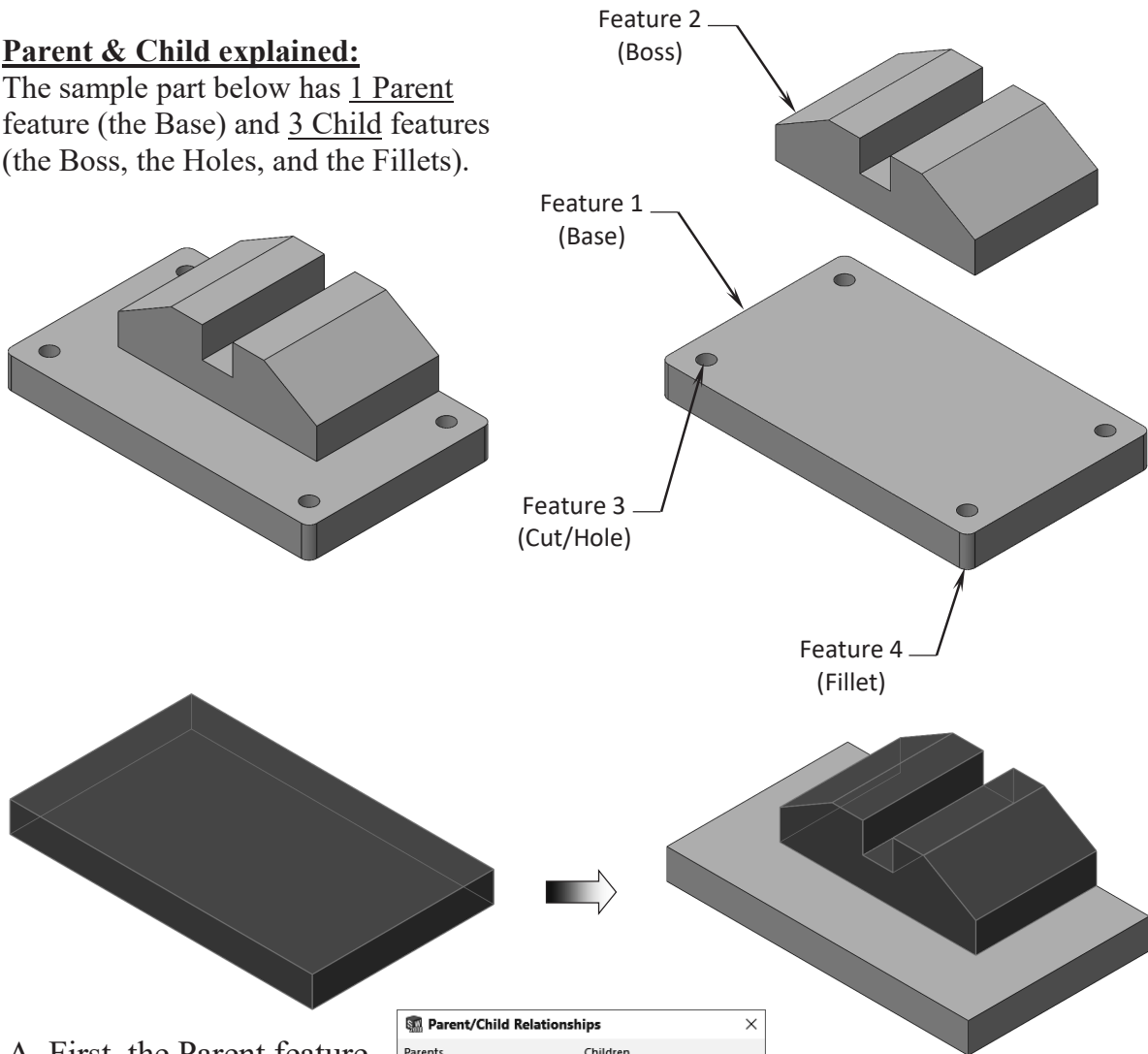
Trim Entities



Boss / Base  
Extrude

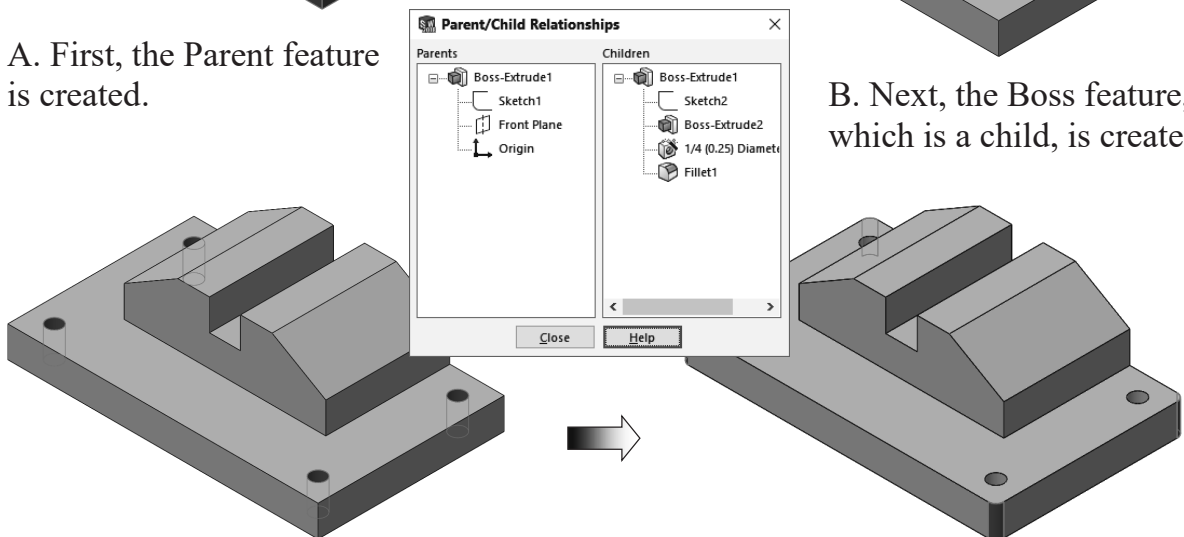
**Parent & Child explained:**

The sample part below has 1 Parent feature (the Base) and 3 Child features (the Boss, the Holes, and the Fillets).



A. First, the Parent feature is created.

B. Next, the Boss feature, which is a child, is created.

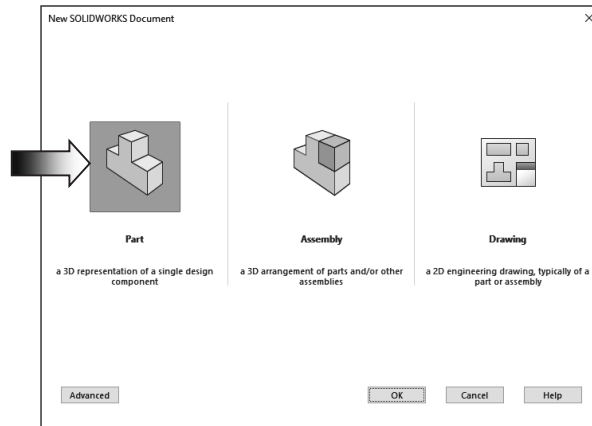


C. The features that remove material such as Extruded Cuts or Holes are created next.

D. Finally, the Fillets and Chamfers features are added last.

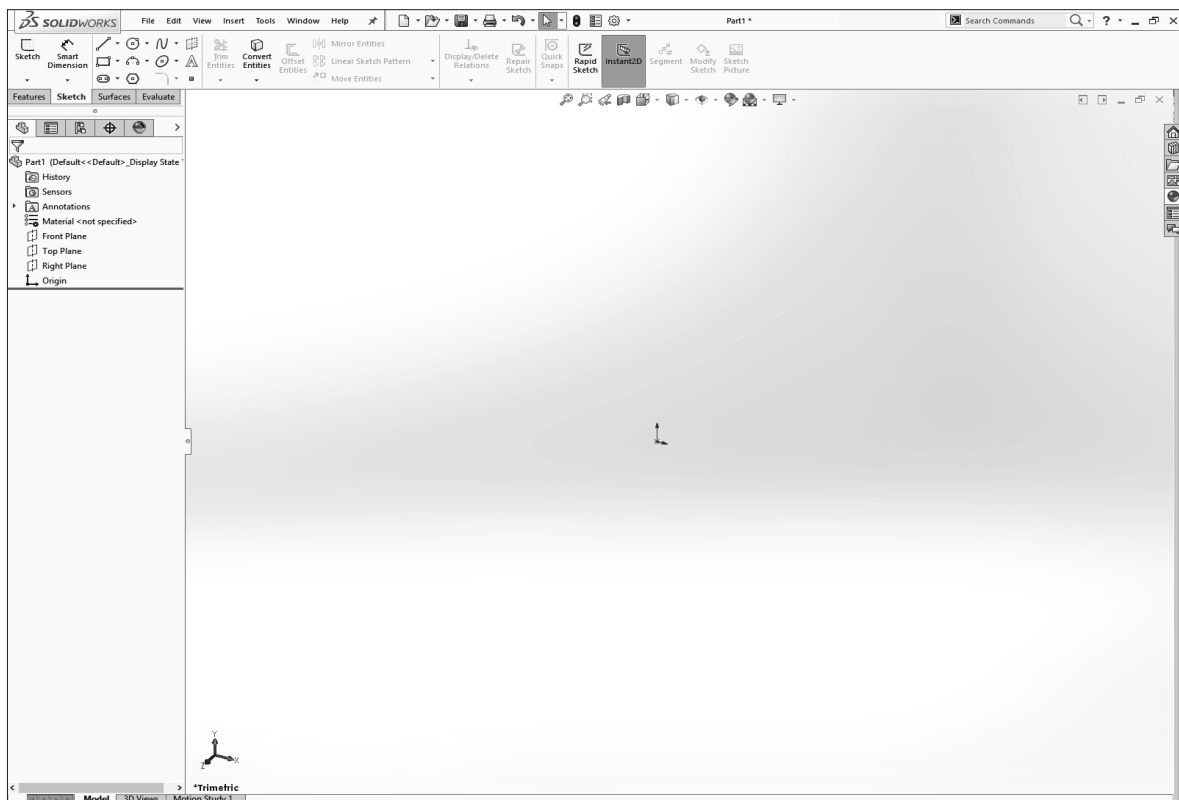
## 1. Starting a new Part:

From the **File** menu, select **New / Part**, or click the **New** icon.



Select the **Part** template from either the Templates or Tutorial folders.

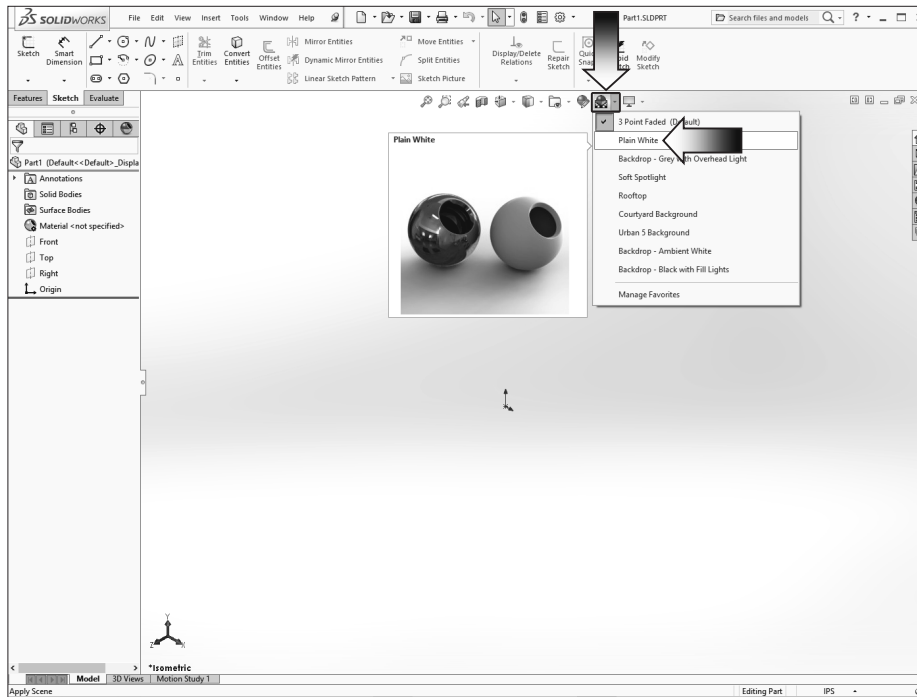
Click **OK**. A new part template is open.



## 2. Changing the Scene:

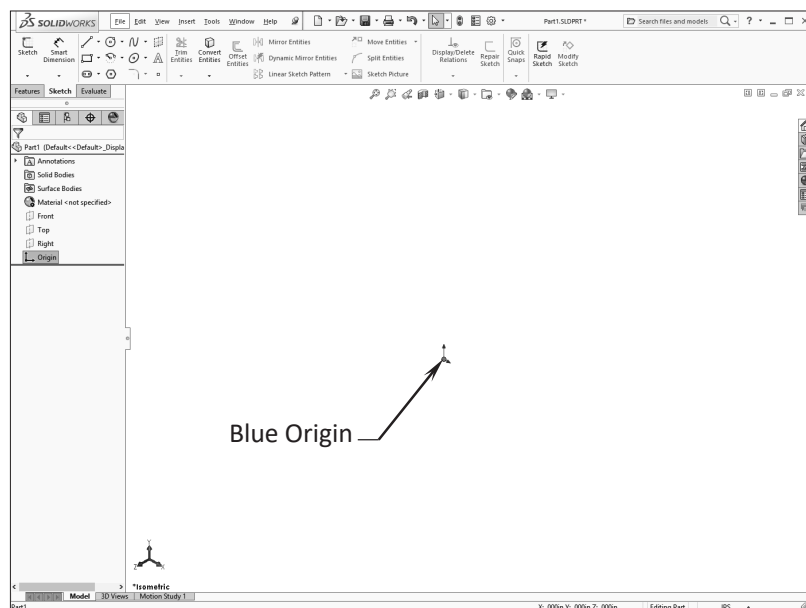
From the View (Heads-up) toolbar, click the **Apply Scene** button (arrow) and select the **Plain White** option (arrow).

By changing the scene color to **Plain White**, we can see the colors of the sketch entities and their dimensions a little better.



To show the Origin, click the **View, Hide /Show** drop-down menu and select **Origins**.

The Blue color Origin is the Zero position of the part and the Red Origin is the Zero position of the sketch.



### 3. Starting a new Sketch:

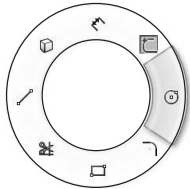
Select the Front plane from the Feature-Manager tree and click the **Sketch** drop-down arrow and select the **Sketch** command to start a new sketch.

A sketch is usually created first, relations and dimensions are added after, and then it gets extruded into a 3D feature.

From the CommandManager toolbar, select the **Line**



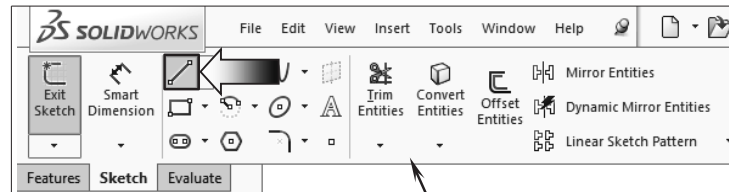
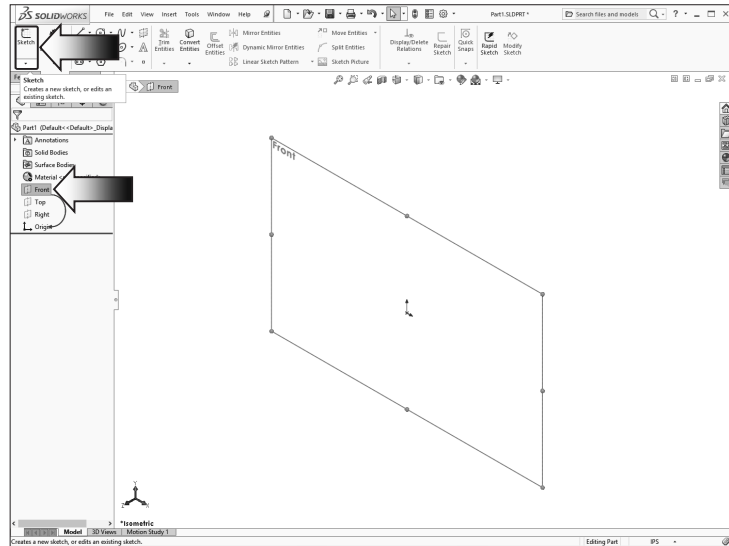
command.



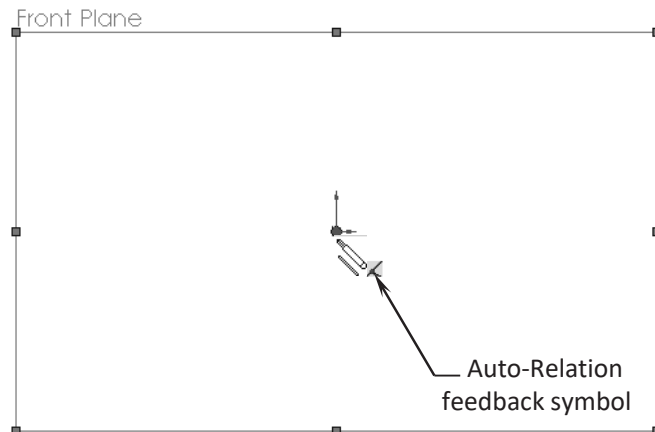
Mouse-Gesture

#### **OPTIONAL:**

*Right-Drag to display the Mouse-Gesture wheel and select the Line command from it. (See the Introduction section, page XVIII for details on customizing the Mouse Gesture.)*



CommandManager



Hover the mouse cursor over the Origin point; a yellow feedback symbol appears to indicate a relation (Coincident) is going to be added automatically to the 1<sup>st</sup> endpoint of the line. This endpoint will be locked at the zero position.



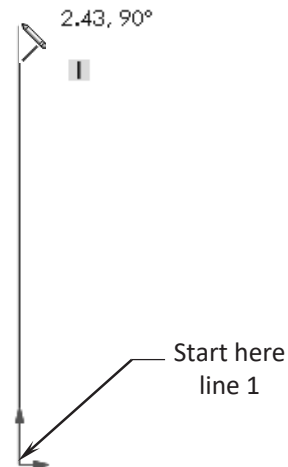
**Note:** There are 3 different ways to use the line tool: **Single Line**, **Multiple Lines** (connecting), and **Line-to-Arc** (transitioning from a line to a tangent arc).

The following steps are examples to demonstrate how to use all embedded functions of the Line command. We will delete the example and go back to the lesson on page 3-8.

A. To sketch a single line, use the **click + hold + drag** method:

With the Line tool already selected, click on the Origin point to start the line (press and hold the mouse button), drag the cursor upward, look for the **vertical relation** symbol, and release the mouse button.

Only 1 line is drawn; the mouse cursor is disconnected from the line.

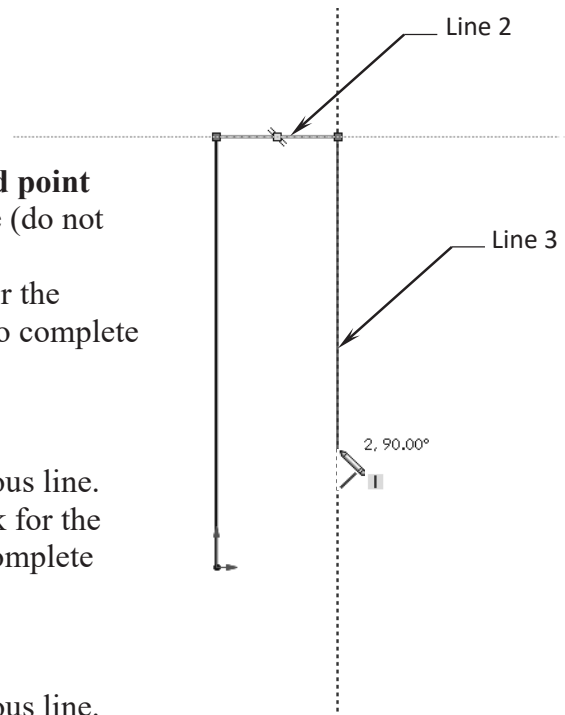


B. To sketch multiple lines, use the **click + release + click** method.

Position the mouse cursor at the **top end point** of the first line and click to start the line (do not hold the mouse button this time). Move the cursor to the right and look for the **Horizontal relation** symbol and click to complete the line.

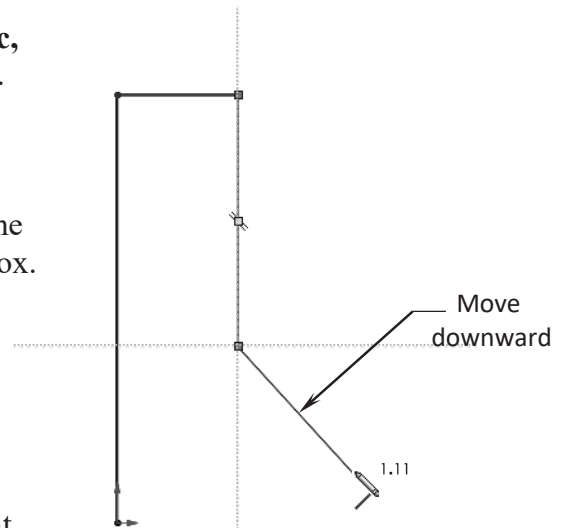
The next line is connecting to the previous line. Move the mouse cursor downward, look for the **vertical relation** symbol and click to complete the line.

The next line is connecting to the previous line.



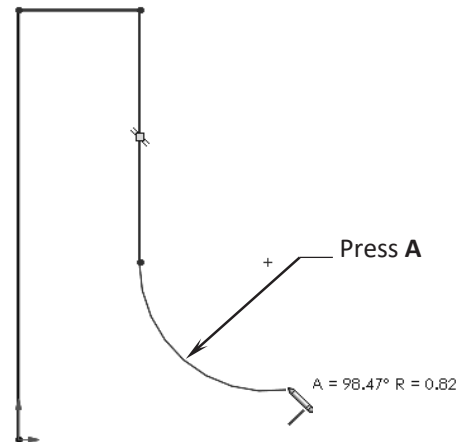
C. To change from a **Line** to a **Tangent Arc**, use the **click + release + press A** method.

Start a new line at the bottom of the 3<sup>rd</sup> line and move the mouse cursor outward approx. as shown.



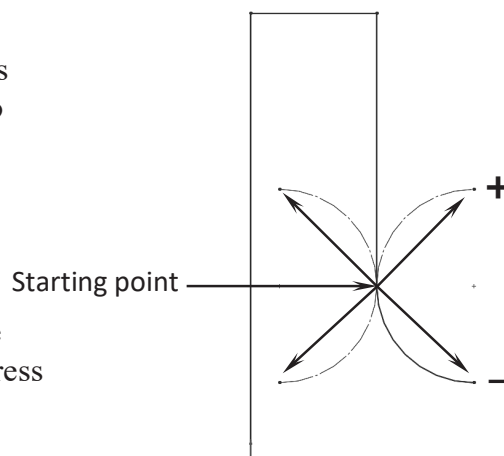
D. Press once on the **A** key and then move the cursor slightly to the left or to the right. The line is switched to a **Tangent Arc**.

Move the cursor back and forth, up and down to see how the Tangent Arc maintains its tangent relation with the line that it is connecting to.



Based on the starting point, the arc changes its direction from negative to positive when the cursor moves pass its zero point.

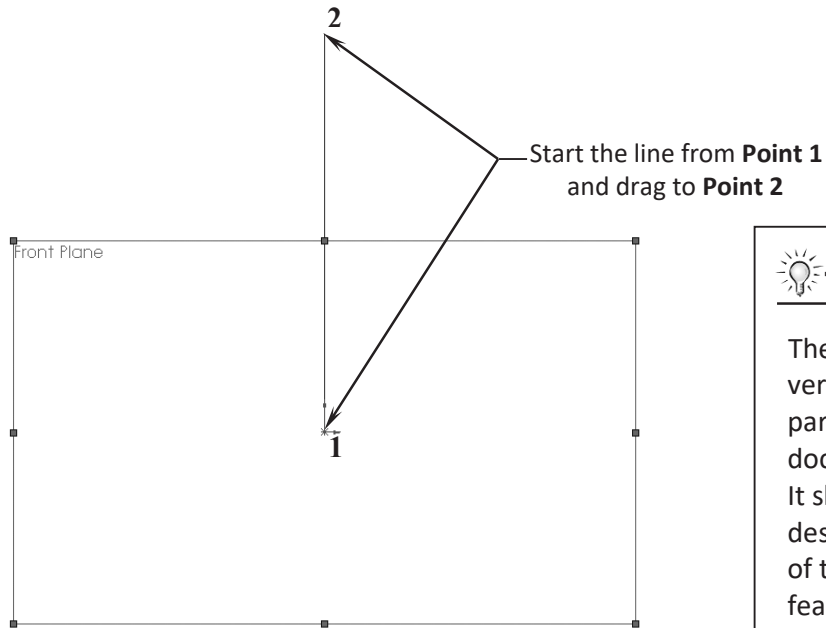
Move the mouse cursor towards the lower right side and click to complete the arc.



Delete all sketch entities before moving on to step number 4 (press Control+A then push Delete).

#### 4. Using the Click + Hold + Drag technique:

Select the **Line** command. Click the Origin point and **hold** the mouse button to start the line at point 1, **drag upwards** to point 2, then release the mouse button.



#### The Base Sketch

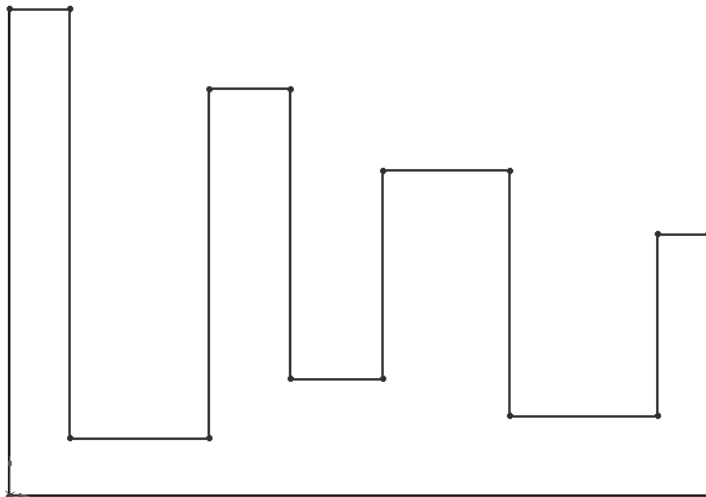
The Base Sketch is the very first sketch; it is the parent sketch of a part document.

It should primarily describe the basic shape of the part before other features can be added.



Continue adding other lines using the **Click-Hold-Drag** technique.

The relations like Horizontal and Vertical are added automatically to each sketch line. Other relations like Collinear and Equal are added manually.

The size and shape of the profile will be corrected in the next few steps.

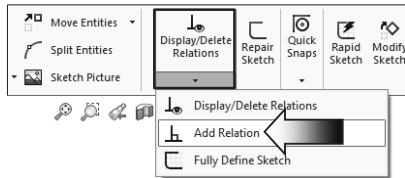


#### System Feedback

While sketching the lines, watch for the System Feedback Symbols such as  for Horizontal, and for  Vertical Auto Relations.


## 5. Adding Geometric Relations\*:

Click **Add Relation**  under Display/Delete Relations - OR - select **Tools / Relations / Add**.



Select the **4 lines** shown below.

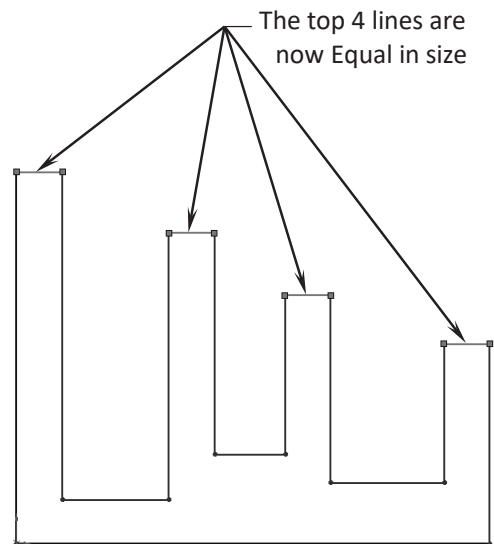
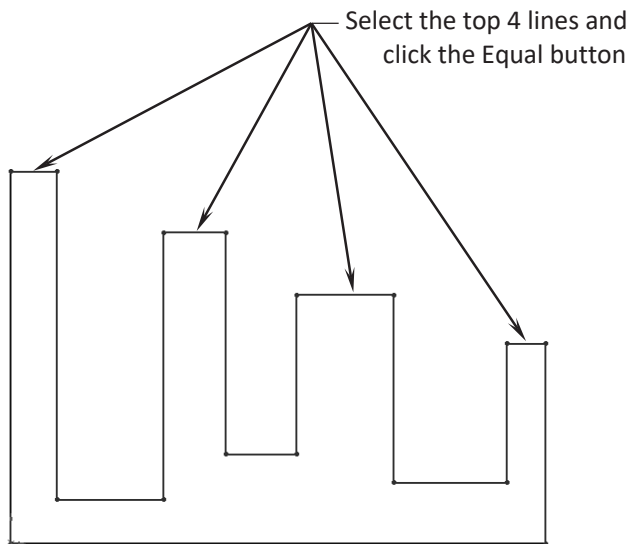
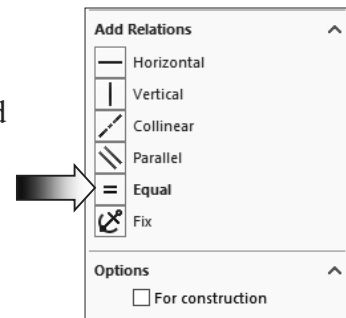
Click **Equal** from the Add Geometric Relation dialog box. This relation makes the length of the four selected lines equal.



### Equal Relations

Adding the EQUAL relations to these lines eliminates the need to dimension each line.

Geometric relations can be created manually or automatically. The next few steps in this chapter will demonstrate how geometric relations are added manually.



\* Geometric relations are one of the most powerful features in SOLIDWORKS. They are used in the sketch level to control the behaviors of the sketch entities when they are moved or rotated and to keep the associations between one another.

(When applying geometric relations between entities, one of them should be a 2D entity and the other can either be a 2D sketch entity, a model edge, a plane, an axis, or a curve.)

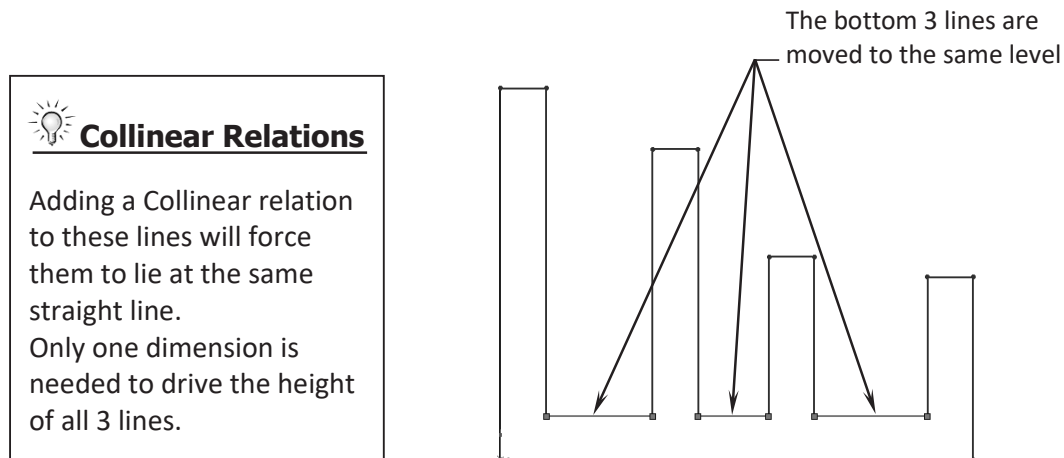
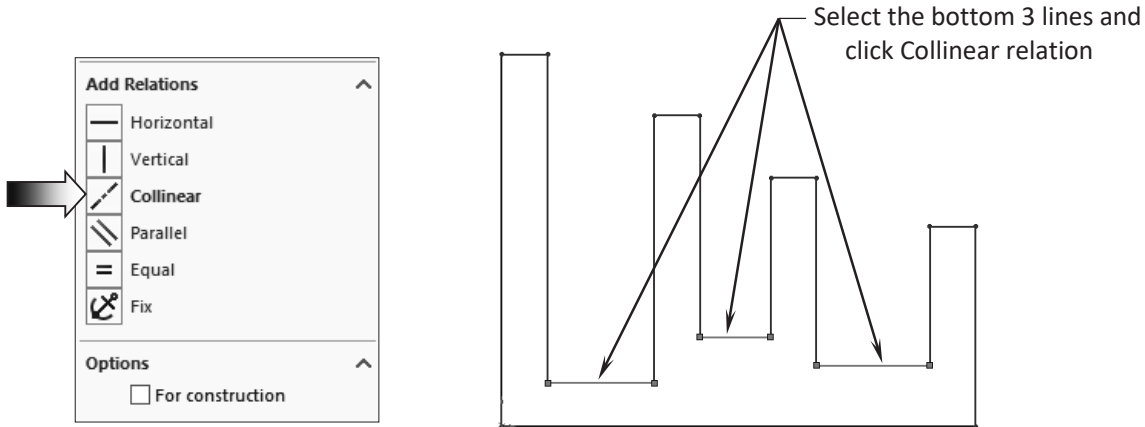
## 6. Adding a Collinear relation\*\*:

Select the **Add Relation**  command again.

Select the **3 lines** shown below.

Click **Collinear** from the Add Geometric Relations dialog box.

Click **OK** to accept the relation.

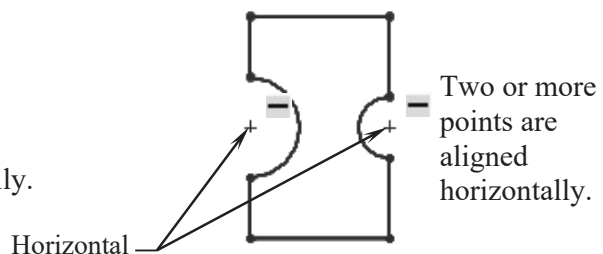
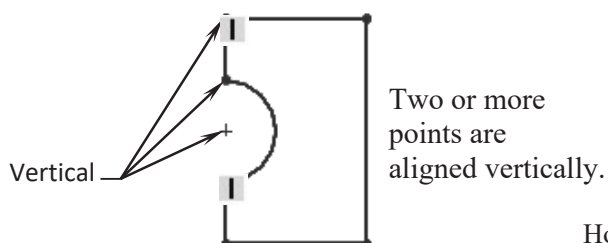
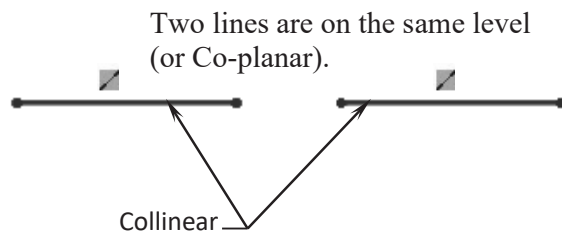
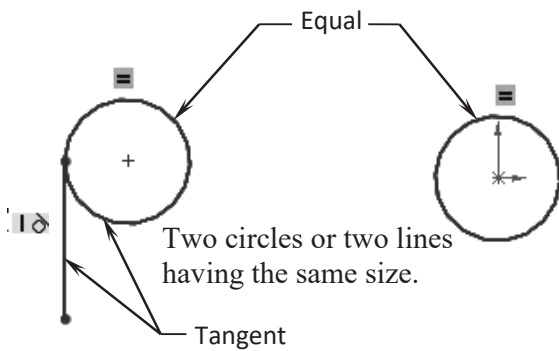
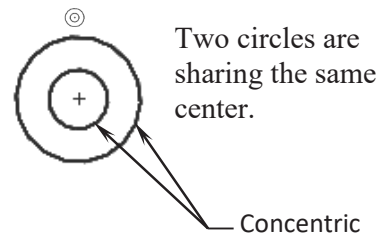
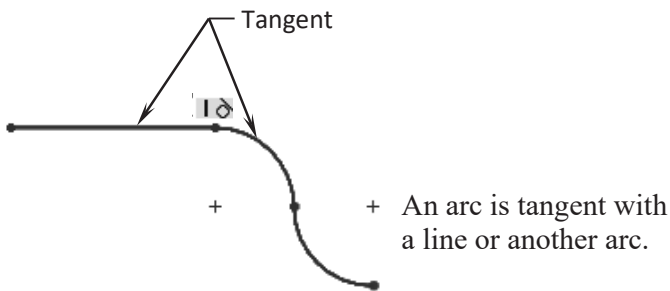
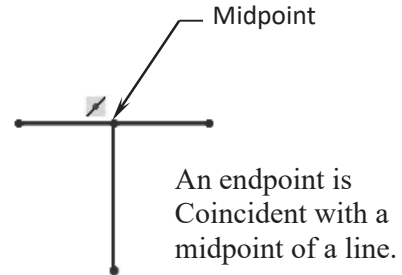
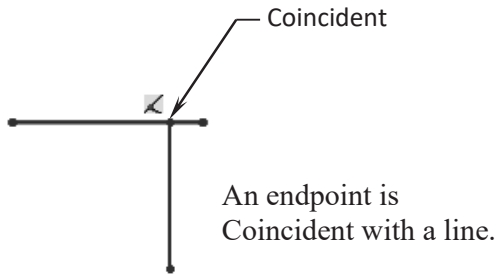


\*\* Collinear relations can be used to constrain the geometry as follows:


Collinear between a line and another line(s) (2D and 2D).

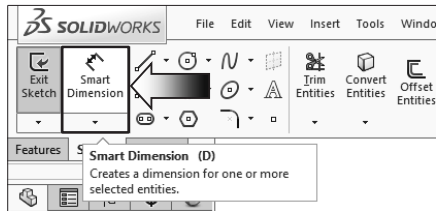
Collinear between a line(s) to a linear edge of a model (2D and 3D).

## Geometric Relation Examples



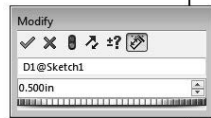
## 7. Adding the horizontal dimensions:

Select  from the Sketch toolbar - OR – press the letter D on the keyboard, and add the dimensions shown below (follow the 3 steps A, B and C).



A. Click line 1 →

B. Click line 2 →



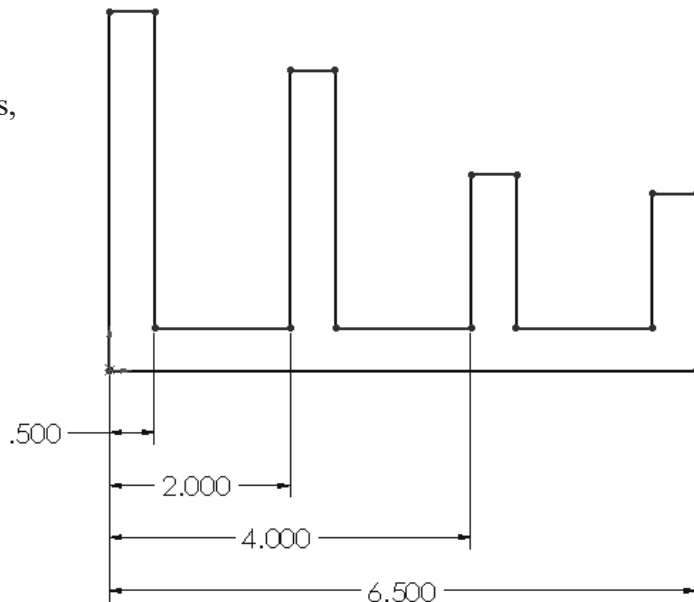
C. Place the dimension approximately here, type **.500** and press enter.

The Inch-Units is filled in automatically because it has been set previously to Inches, 3 decimal places.

Continue adding the horizontal dimensions as shown here.

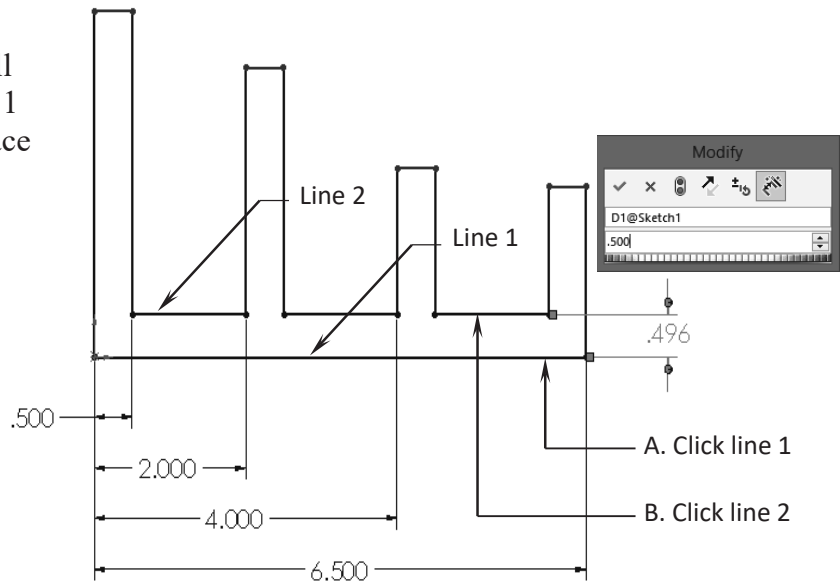
### **NOTE:**

*The color of the sketch lines changes from Blue to Black, to indicate that they have been constrained with dimensions.*

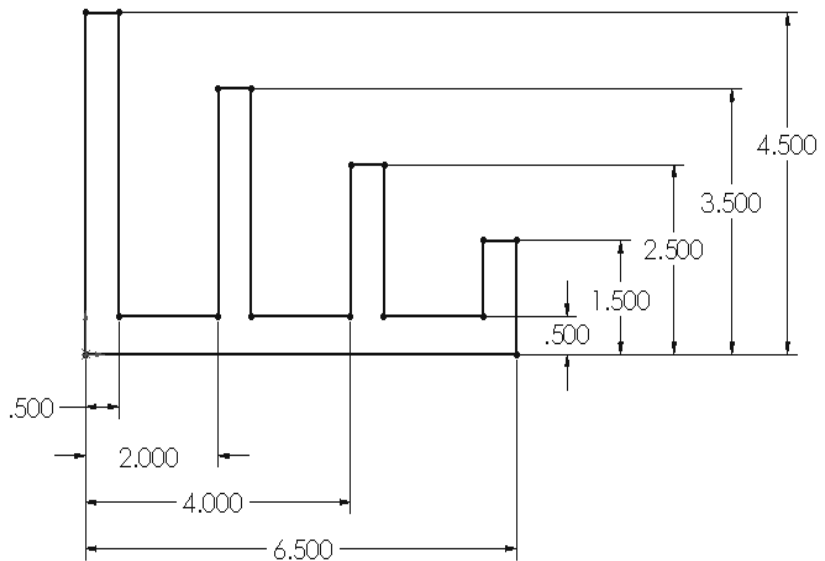


## 8. Adding the Vertical dimensions:

With the Smart-Dimension tool still selected, click line 1 and line 2, then place the dimension approximately as shown; change the value to **.500in.**



Continue adding other dimensions until the entire sketch turns into the Black color.



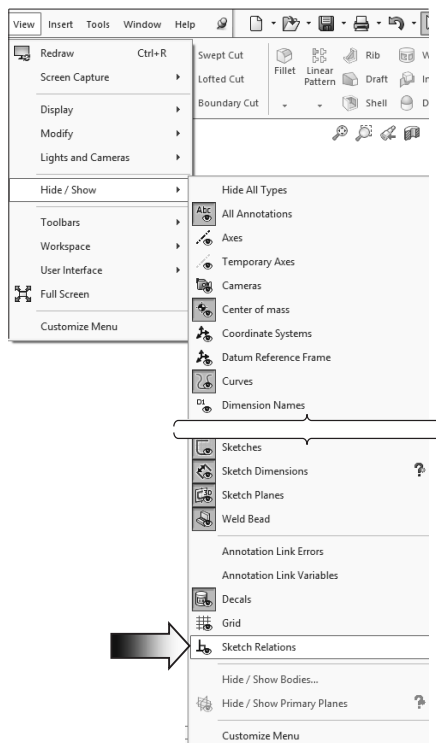
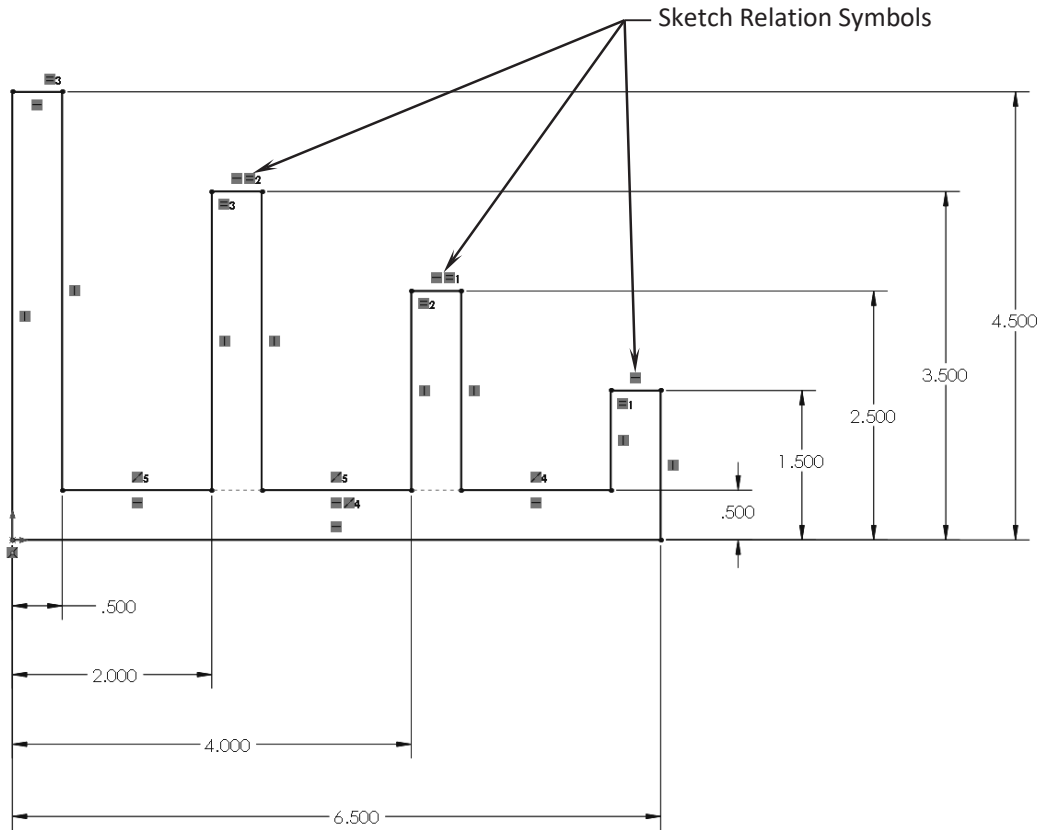
### The Status of a Sketch:

The current status of a sketch is displayed in the lower right corner of the screen.

**Fully Defined** = **Black**  
**Under Defined** = **Blue**  
**Over Defined** = **Red**







## 9. Hiding the Sketch Relation Symbols:

The Sketch Relation Symbols indicate which geometric relation a sketch entity has, but they get quite busy as shown.

To hide or show the Sketch Relation Symbols, go to the **View, Hide/Show** menus and click off the **Sketch Relations** option.

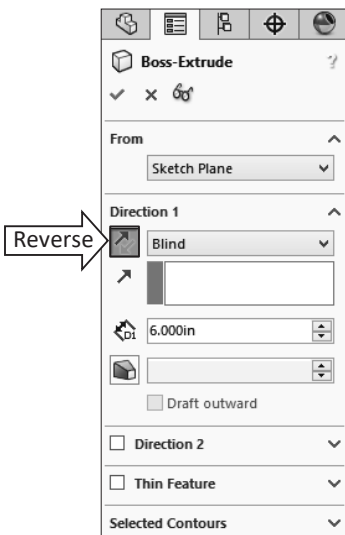
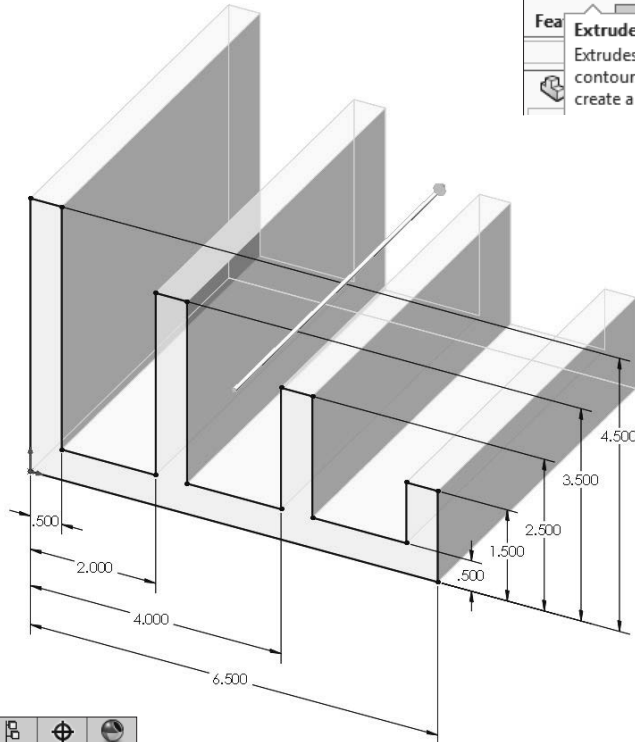
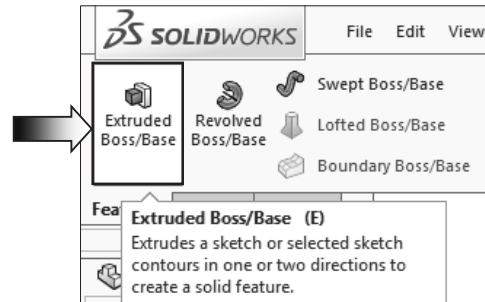
### Sketch Relation Symbols at a Glance

	Horizontal relation		Vertical relation
	Equal relation		Coincident relation
	Tangent relation		Collinear relation

## 10. Extruding the Base:

The **Extrude Boss/Base** command is used to define the characteristic of a 3D linear feature.

Switch to the **Features** tab and click  or select: **Insert** / **Boss-Base** / **Extrude**.



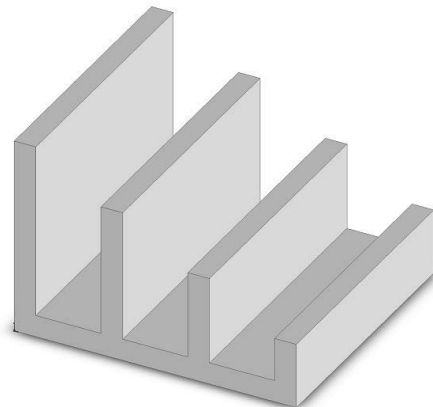
Select the following:

Direction: **Blind**.

Depth: **6.00in**.


Enabled **Reverse** direction (arrow).

Click **OK**.



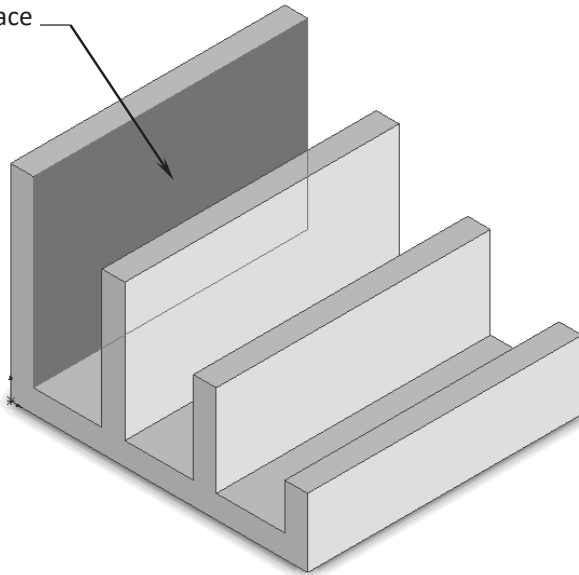
## 11. Sketching on a Planar Face:

Select the face as indicated below.

Click  or select **Insert/Sketch** and press the shortcut key **Ctrl+7** to change to the Isometric view.

Select the **Circle** command  from the Sketch Tools toolbar.

Select the  
Sketch Face



### Planar Surfaces

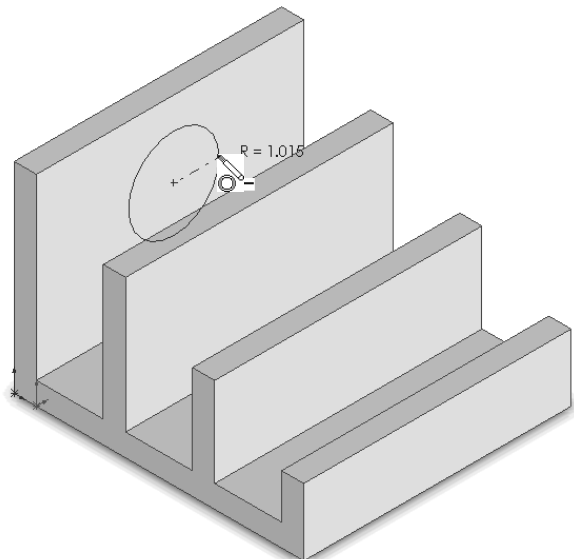
A planar surface of the model can also be used as a Sketch Plane.

The Sketch will then be extruded normal to the selected surface.


Position the mouse cursor near the center of the selected face, click and drag outward to make a circle.

While sketching the circle, the system displays the radius value next to the mouse cursor.

Dimensions are added after the profile is created.




## Select the **Smart Dimension**

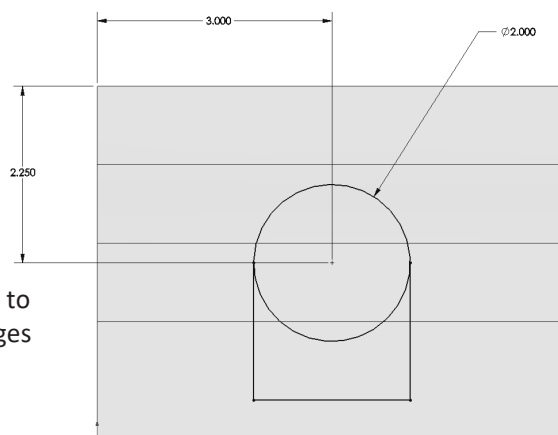
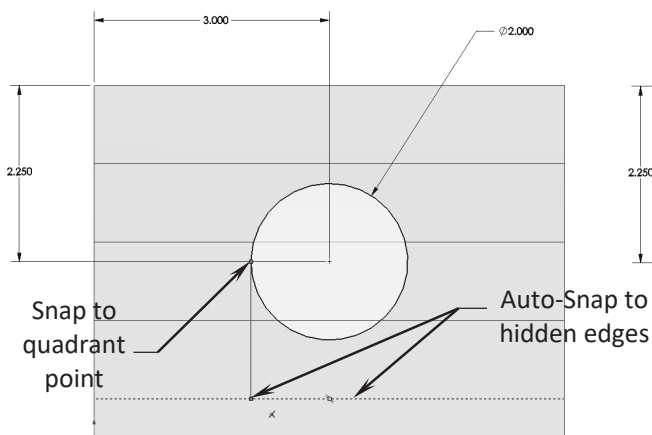
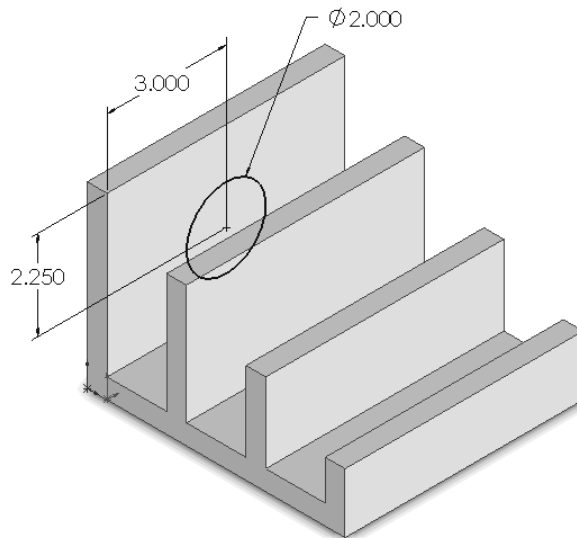
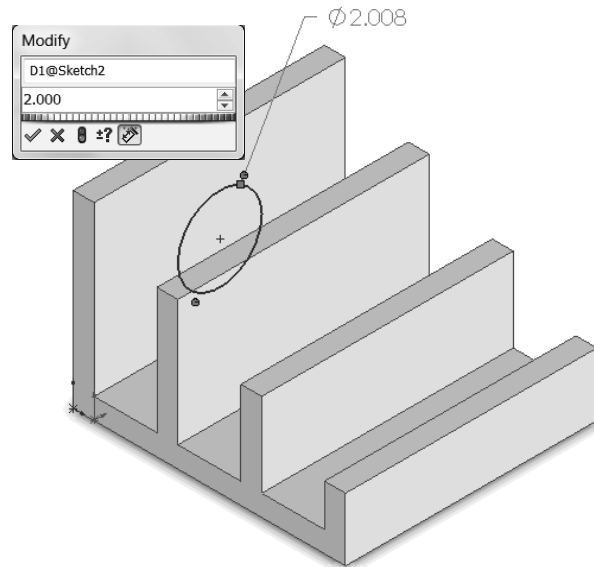
command  and add a diameter dimension to the circle. (Click on the edge of the circle and move the cursor outward at approximately 45 degrees, and click to place the dimension.)

To add the location dimensions, click the edge of the circle and the edge of the model, place the dimension, then correct the value.

Continue adding the location dimensions as shown to fully define the sketch.

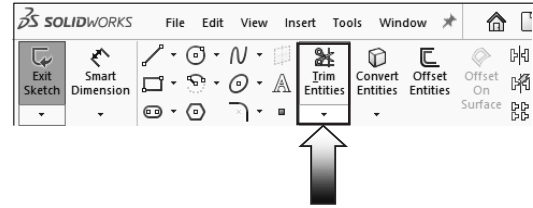
Select the Line command  and sketch the **3 lines** as shown below. Snap to the hidden edge of the model when it highlights.

The color of the sketch should change to black at this point (Fully Defined).

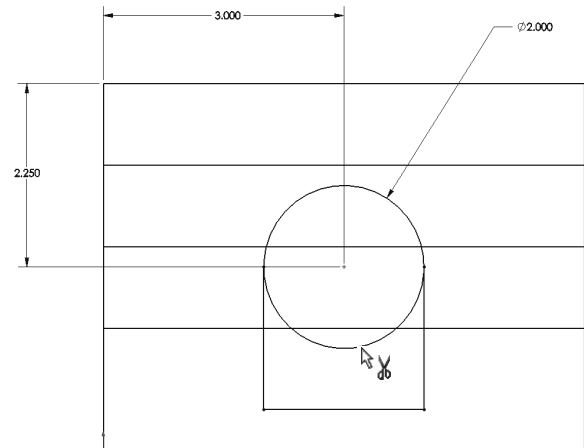
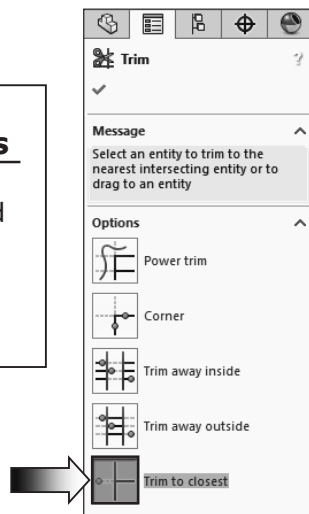
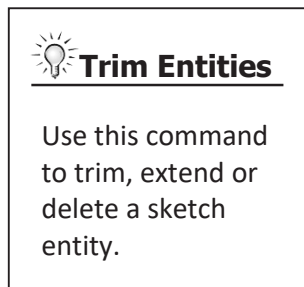


## 12. Using the Trim Entities command:

Select the **Trim Entities** command from the Sketch toolbar (arrow).



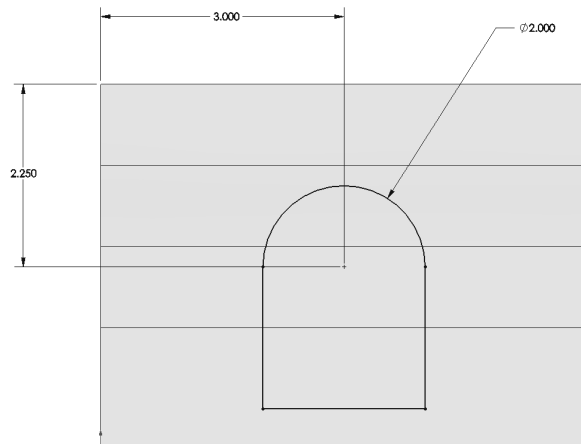
Click the **Trim to Closest** option (arrow). When the pointer is hovered over a sketch entity, this trim command will highlight the entity prior to trimming them to the next intersection.



*(The image is shown in wireframe for example only; remain in shaded mode.)*

Hover the pointer over the lower portion of the circle; the portion that is going to be trimmed will highlight. Click the mouse button to trim.

The bottom portion of the circle is trimmed, leaving the sketch as one-continuous-closed-profile, suitable to extrude and make a 3D feature.



Next, we are going to look at some of the extrude options available in SOLIDWORKS.

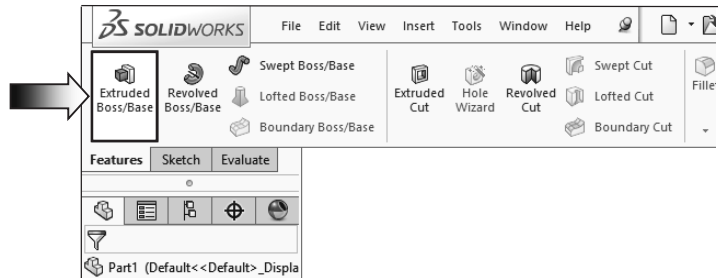
### 13. Extruding a Boss:

Switch to the **Features** tab and click **Extruded Boss/Base**  or select: **Insert / Boss-Base / Extrude**.



#### Extrude Options...

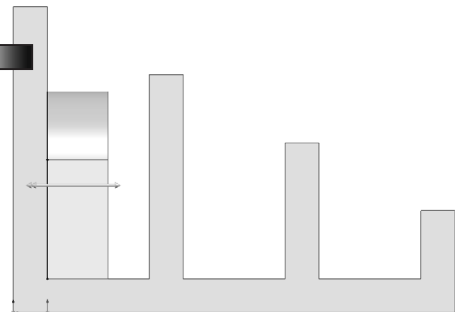
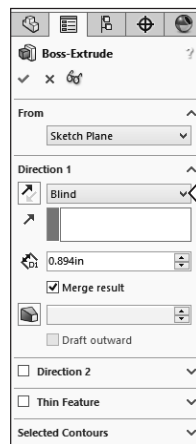
Explore each extrude option to see the different results. Press Undo to go back to the original state after each one.



#### A Using the Blind option:

When extruding with the Blind option, the following conditions are required:

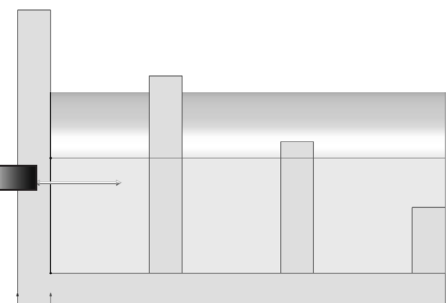
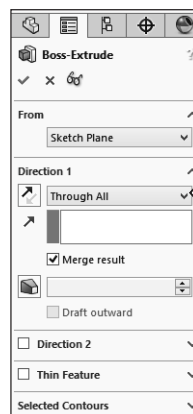
- \* Direction
- \* Depth dimension



**Blind**  
Condition

#### B Using the Through All option:

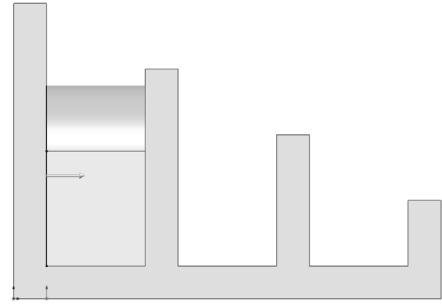
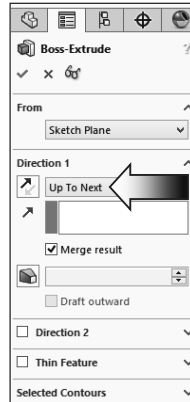
When the Through All option is selected, the system automatically extrudes the sketch to the length of the part, normal to the sketch plane.



**Through All**  
Condition

### C Using the Up To Next option:

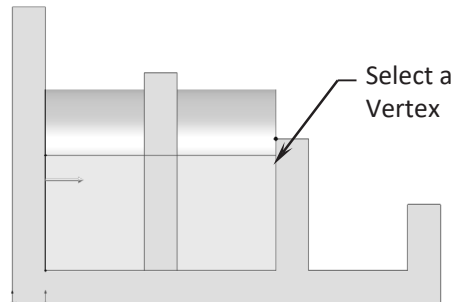
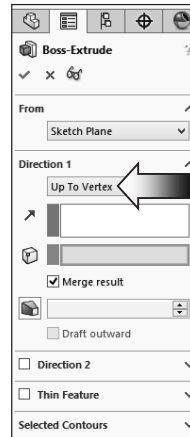
With the Up To Next option selected, the system extrudes the sketch to the very next group of surface(s), and blends it to match the geometry of the surface(s).



**Up To Next**  
Condition

### D Using the Up To Vertex option:

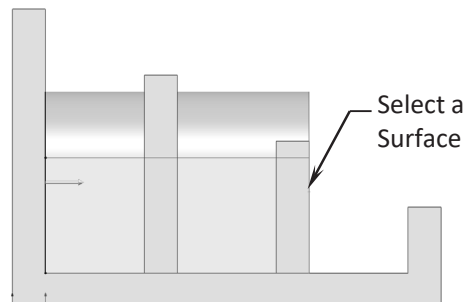
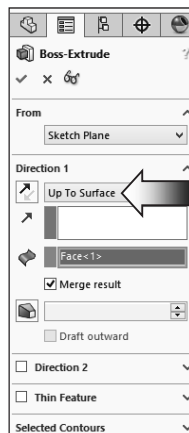
This option extrudes the sketch from its plane to a vertex, specified by the user, to define its depth.



**Up To Vertex**  
Condition

### E Using the Up To Surface option:

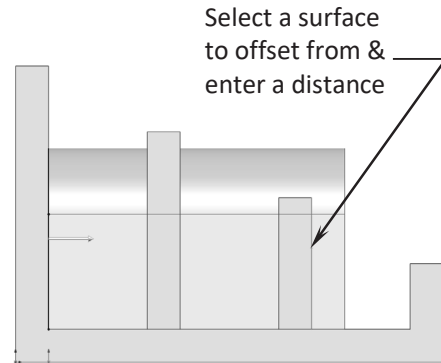
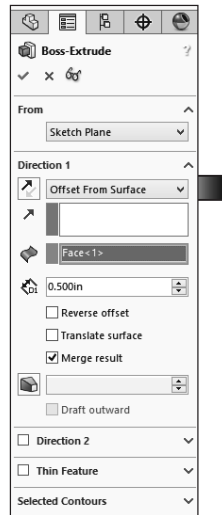
This option extrudes the sketch from its plane to a single surface to define its depth.



**Up To Surface**  
Condition

## F Using the Offset From Surface option:

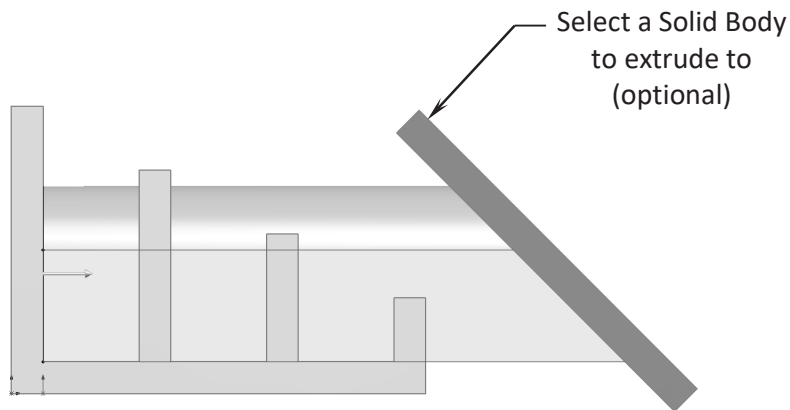
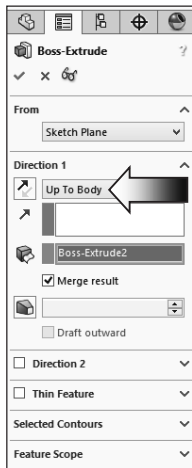
This option extrudes the sketch from its plane to a selected face, and then offsets at a distance specified by the user.



**Offset From Surface**  
Condition

## G Using the Up To Body option (optional):

This option extrudes the sketch from its sketch plane and blends into a selected body.



**Up To Body**  
Condition

The Up To Body option can also be used in assemblies or multi-body parts.

The Up To Body option works with either a solid body or a surface body.

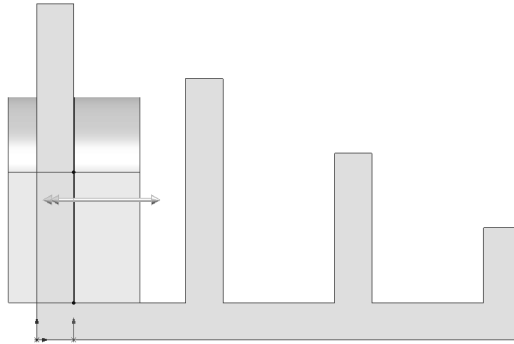
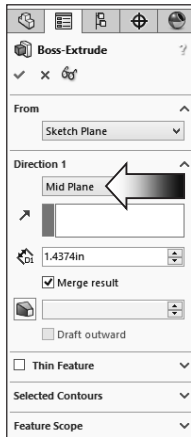
It is also useful when making extrusions in an assembly to extend a sketch to an uneven surface.



## H Using the Mid Plane option:

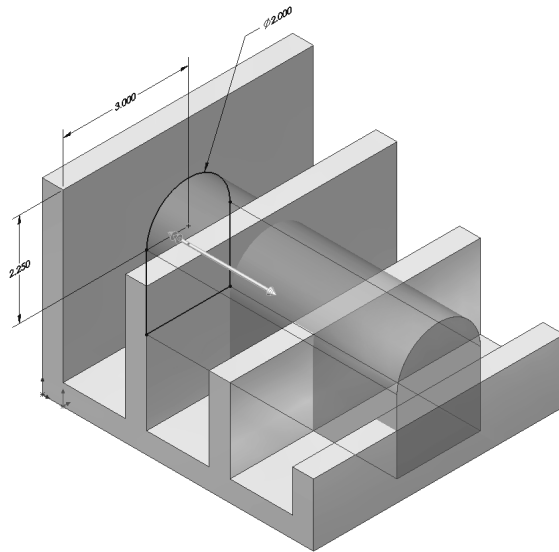
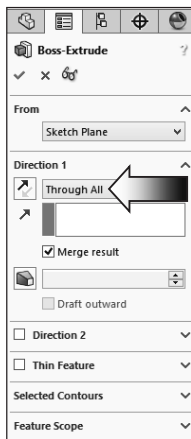
This option extrudes the sketch from its plane equally in both directions.

Enter the Total Depth dimension when using the Mid-Plane option.



**Mid Plane**  
Condition

After you are done with exploring all the extrude options, change the Direction 1 to **Through All**.

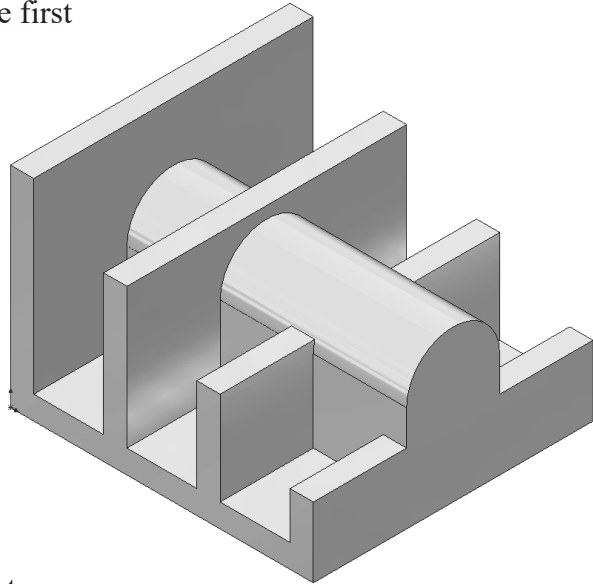


Click **OK**.

The system extrudes the sketch to the outermost surface as the result of the Through All end condition.

The overlapped material between the first and the second extruded features is removed automatically.

If the Merge Result checkbox is cleared, all interferences will be kept and the model becomes a multibody part.



## 14. Using the Hole Wizard:

The Hole Wizard tool is used to create customized holes of various types such as Counterbore, Countersink, Drill hole, Straight tap, Tapered tap, Legacy hole, etc.

First, you select a standard, hole type, and hole size, then you position the hole using either 2D sketch or 3D sketch placement.

Click **Hole Wizard** on the **Features** tab.

Select the following:

Hole Type: **Counterbore**

Standard: **ANSI Inch**

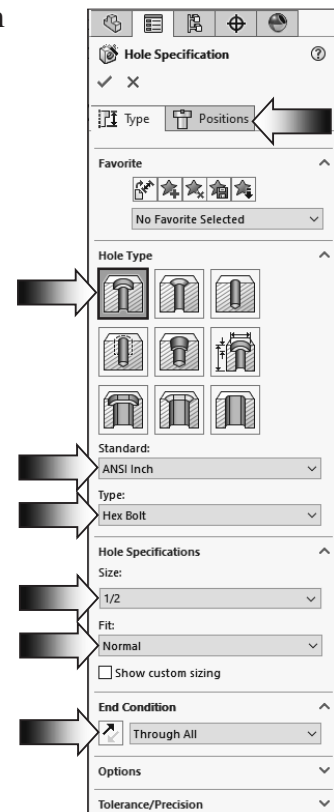
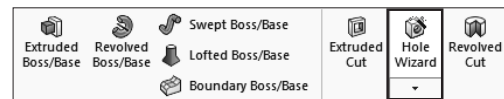
Type: **Hex Bolt**

Size: **1/2**

Fit: **Normal**

End Condition: **Through All**

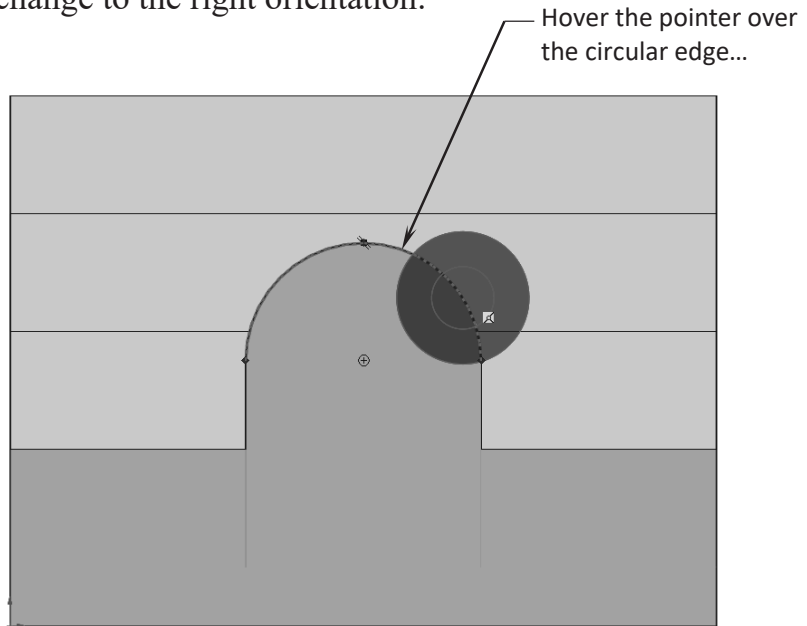
Click the **Positions** tab on the upper left of the tree.



Press **Control+4** to change to the right orientation.

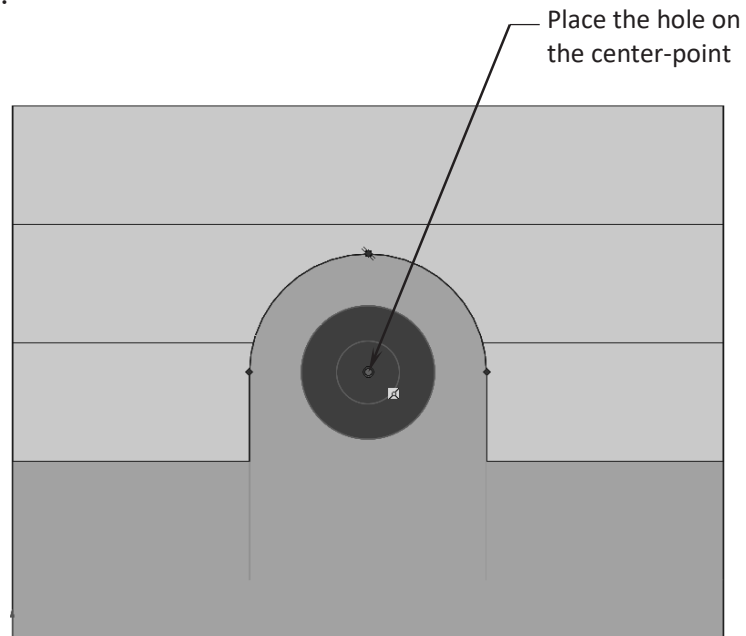
Hover the mouse cursor over the circular edge as indicated, to wake up the center of the cylindrical feature.

Move the mouse cursor to the center point and click to place the counterbore hole.



This shortcut eliminates the need for creating the concentric relation between the counterbore hole and the cylindrical feature.

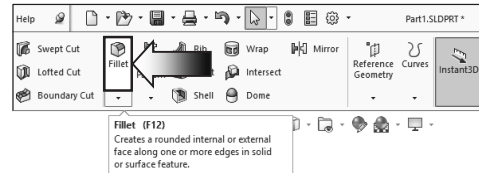
The system adds a **coincident** relation between the center of the counterbore hole and the center of the cylindrical feature.



Click **OK** to accept the hole and close out of the Hole Wizard command.

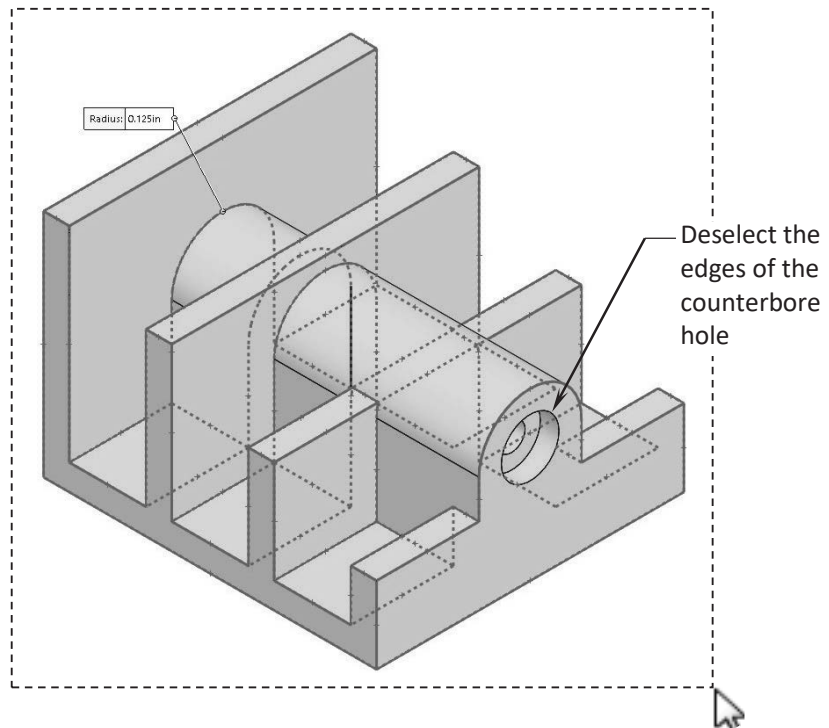
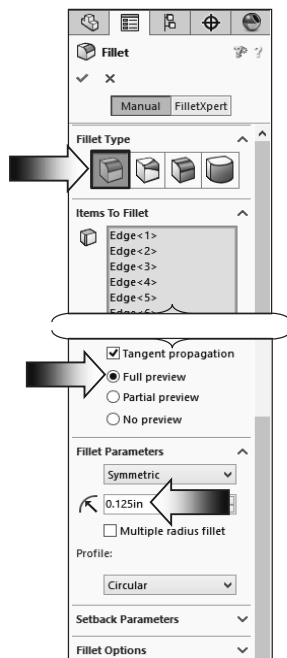
## 15. Adding the model fillets:

Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, select groups of faces, edges, or edge loops.



The **radius** value stays in effect until you change it. Therefore, you can select any number of edges or faces in the same operation.

Click **Fillet**  or select **Insert / Features / Fillet/Round**.



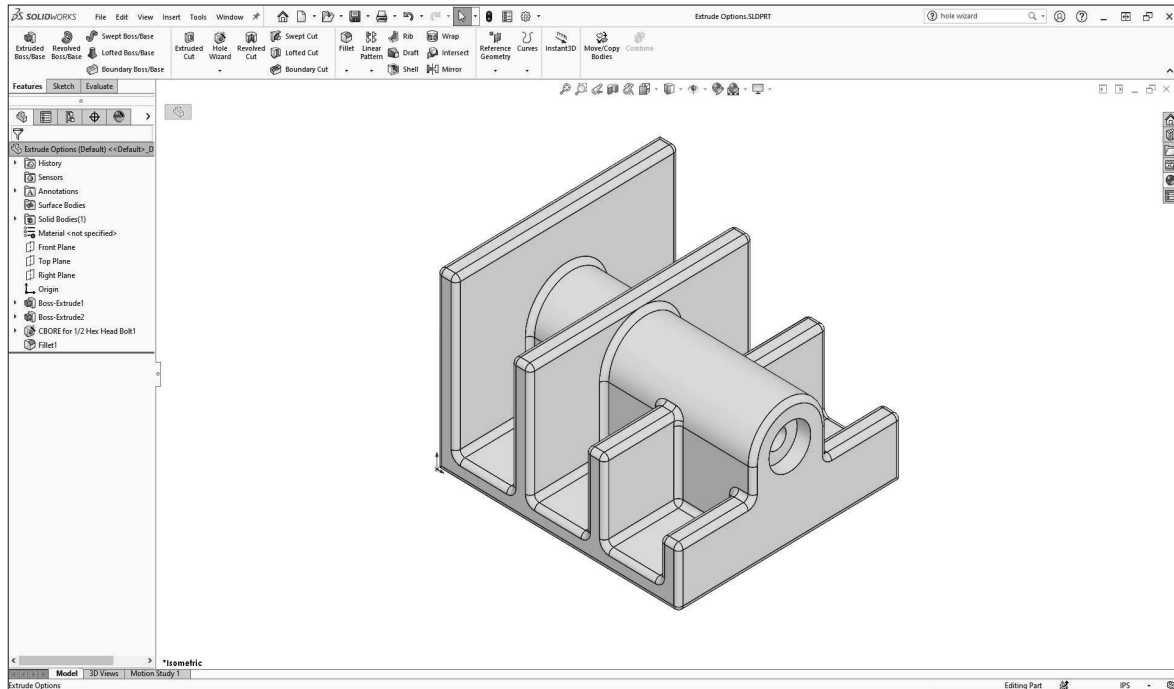
Select the **Constant Size Fillet** button (arrow).

Either "drag-select" to highlight all edges of the model, or press the shortcut key **Control+A** (select all). Deselect the edges of the counterbore hole (front & back).

Enter **.125in.** for radius size.

Enable the **Full Preview** checkbox.

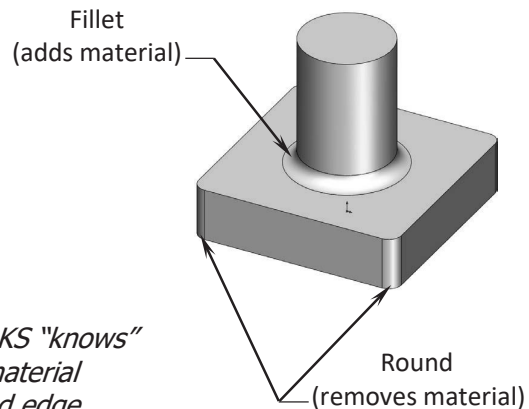
Click **OK**.



In the Training Files folder, in the Built Parts folder, you will also find copies of the parts, assemblies, and drawings that were created for cross referencing or reviewing purposes.

**\* Fillets and Rounds:**

*Using the same Fillet command, SOLIDWORKS "knows" whether to add material (Fillet) or remove material (Round) to the faces adjacent to the selected edge.*



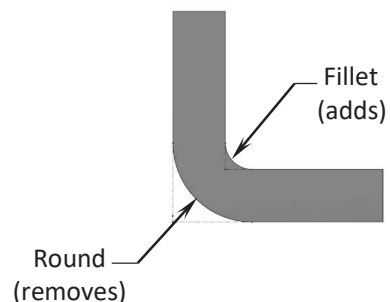
**16. Saving your work:**

Select **File / Save As**.

Change the file type to **Part** file (.sldprt).

Enter **Extrude Options** for the name of the file.

Click **Save**.



## Questions for Review

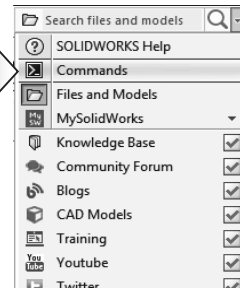
1. To open a new sketch, first you must select a plane from the FeatureManager tree.
  - a. True
  - b. False
2. Geometric relations can be used only at the assembly level, not at the part level.
  - a. True
  - b. False
3. The current status of a sketch is displayed in the lower right area of the screen as Under defined, Fully defined, or Over defined.
  - a. True
  - b. False
4. Once a feature is extruded, its extruded direction cannot be changed.
  - a. True
  - b. False
5. A planar face can also be used as a sketch plane.
  - a. True
  - b. False
6. The Equal relation only works for Lines, not Circles or Arcs.
  - a. True
  - b. False
7. After a dimension is created, its value cannot be changed.
  - a. True
  - b. False
8. When the UP TO SURFACE option is selected, you must choose a surface as the end-condition to extrude up to.
  - a. True
  - b. False
9. UP TO VERTEX is not a valid Extrude option.
  - a. True
  - b. False

1. TRUE  
2. FALSE  
3. TRUE  
4. FALSE  
5. TRUE  
6. FALSE  
7. FALSE  
8. TRUE  
9. FALSE

## Using the Search Commands:

The Search Commands lets you find and run commands from SOLIDWORKS Search or locate commands in the user interface.

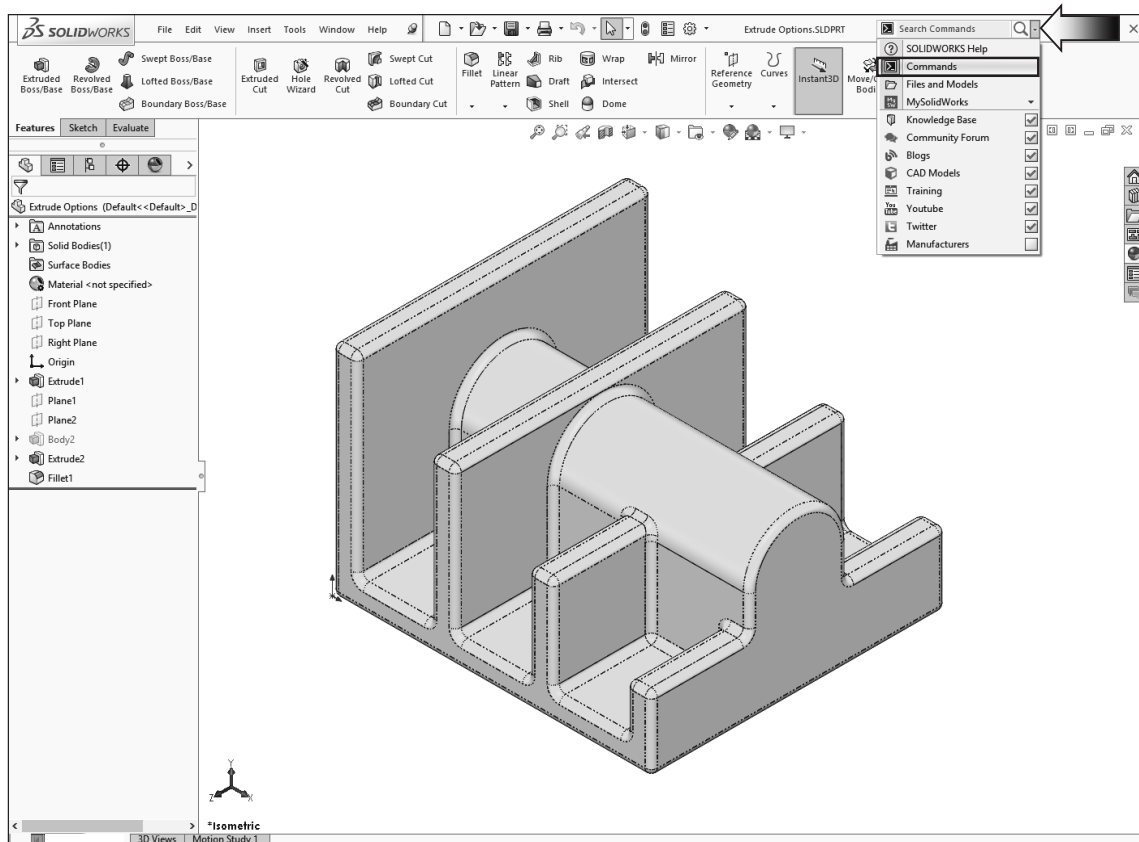
These features make it easy to find and run any SOLIDWORKS command:



The results are filtered as you type and typically find the command that you need within a few keystrokes.

When you run a command from the results list for a query, Search Commands remembers that command and places it at the top of the results list when you type the same query again.

Search shortcuts let you assign simple and familiar keystroke sequences to commands you use more regularly.




Expand the search next to the magnifying glass  to see the search options.

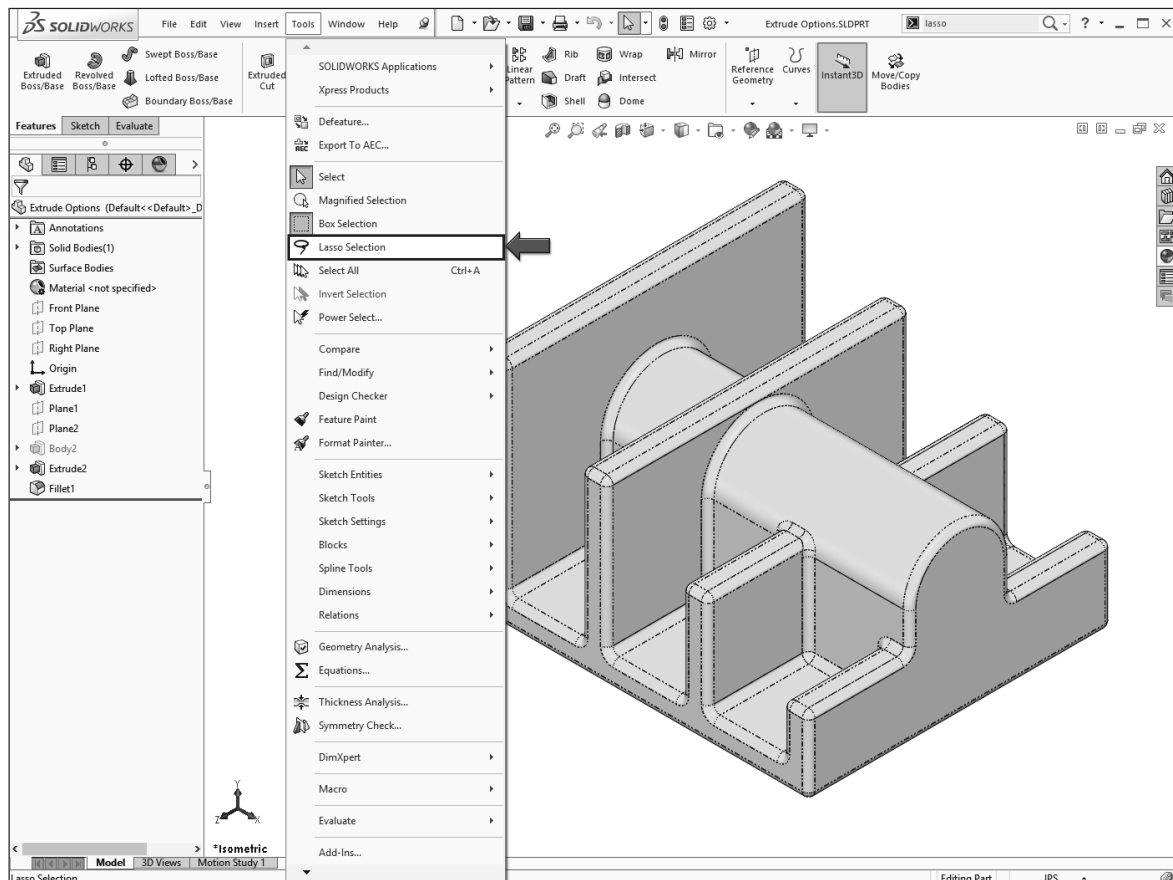
## 1. Search Commands in Features Mode:

The example below shows how you might use Search Commands to find and run the **Lasso Selection** command in Feature Mode.

While the part is still open, start typing the command **Lasso Selection** in Search Commands. As soon as you type the first few letters of the word Lasso, the results list displays only those commands that include the character sequence "lasso," and **Lasso Selection** appears near the top of the results list.



Click **Show Command Location** ; a **red arrow** indicates the command in the user interface (do not move the mouse while the system is searching).

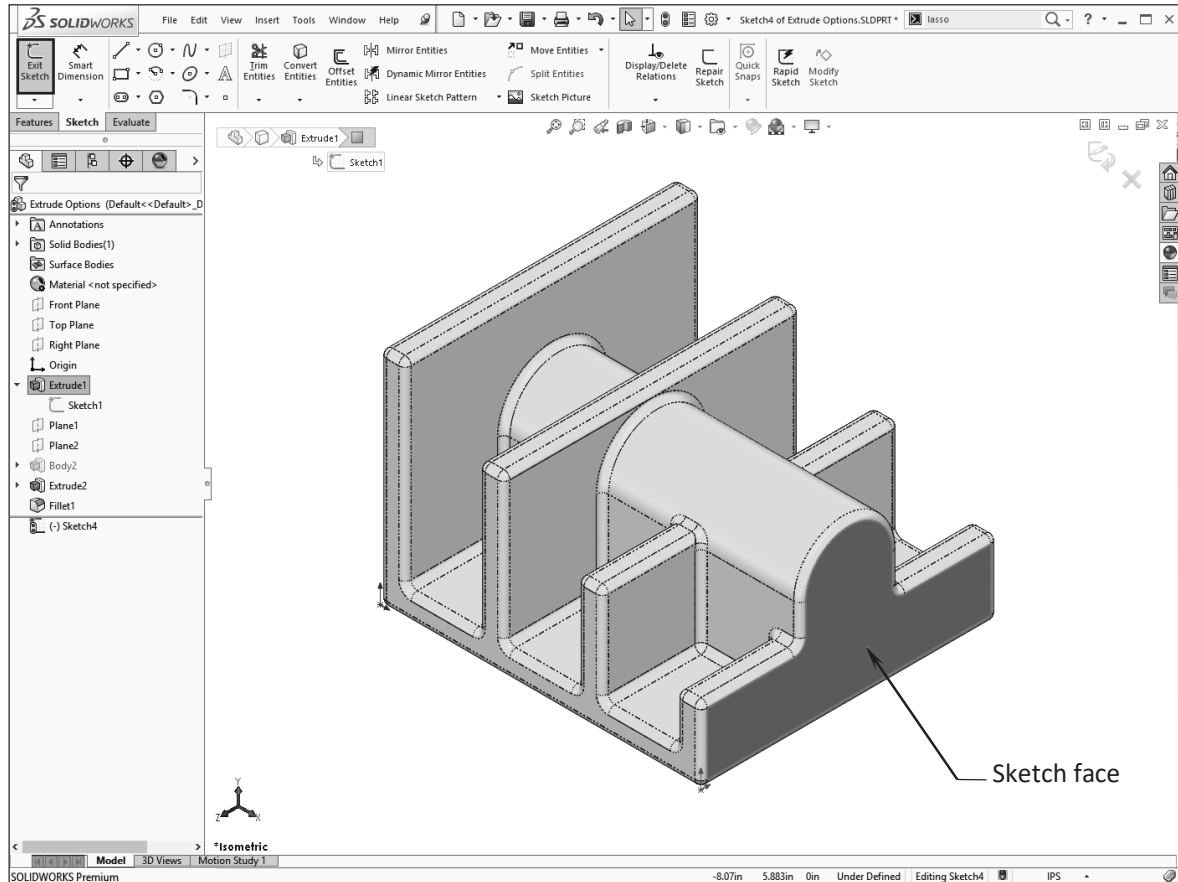




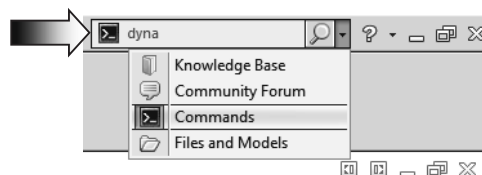
## 2. Search Commands in Sketch Mode:


The example below shows how you might use Search Commands to find and run the **Dynamic Mirror** command in Sketch Mode.

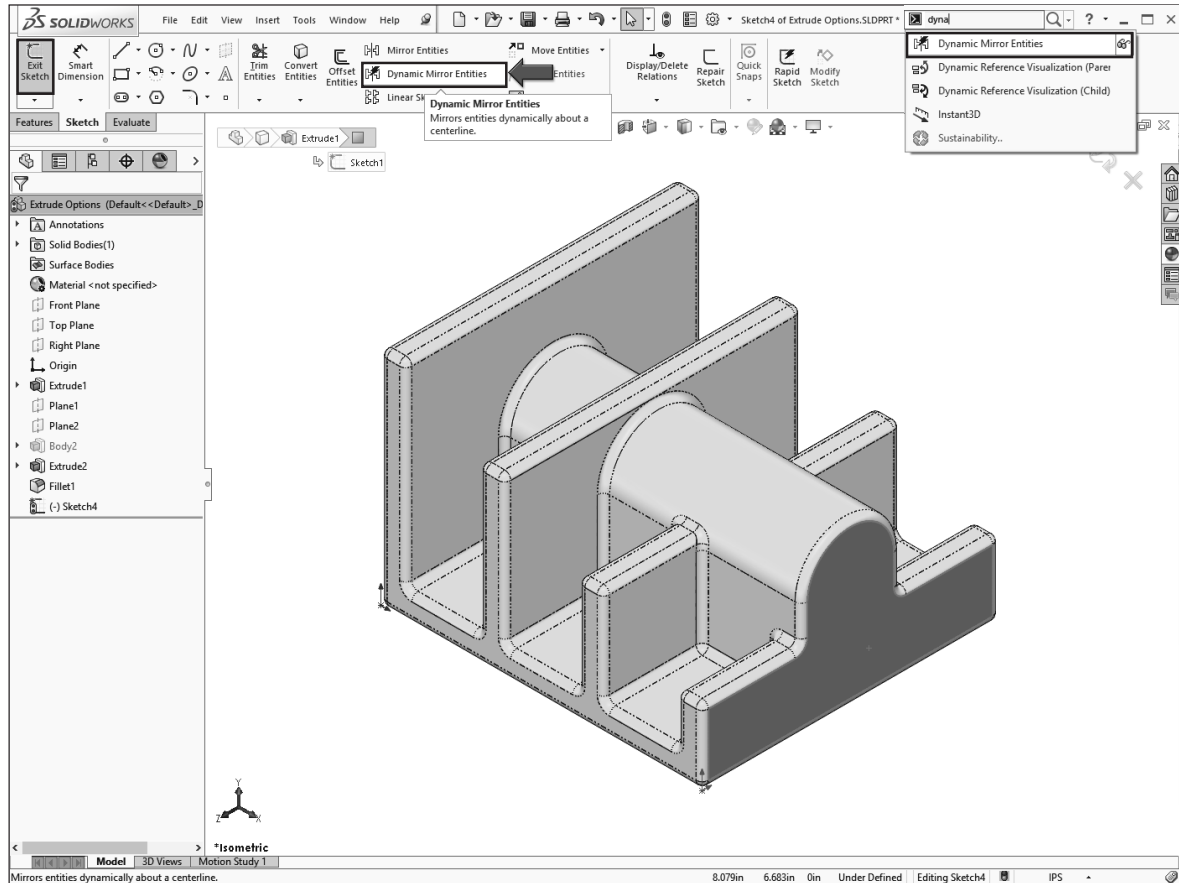
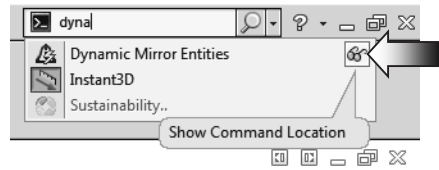
Using the same part, open a **new sketch** on the side face of the model as noted.



Start typing the command **Dynamic Mirror** in Search Commands. As soon as you type the first few letters of the word Dynamic, the results list displays only those commands that include the character sequence "**dyna**," and **Dynamic command** appears near the top of the results list.



Click **Show Command Location** ; a red arrow indicates the command in the user interface (you will cancel the search operation if the mouse is moved during this time).



Additionally, a Search Shortcut can be assigned to any command to help find it more quickly (see Customize Keyboard in the SOLIDWORKS Help for more info):

1. Click **Tools / Customize**, and select the **Keyboard** tab.
2. Navigate to the command to which you want to assign a search shortcut.
3. In the Search Shortcut column for the command, type the shortcut letter you want to use, and then click OK.

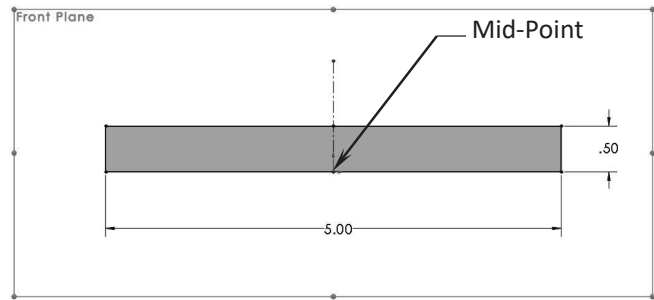
**Save** and close all documents.

## **Exercise 1: Extrude Boss & Hole Wizard**

**NOTE:** The exercise gives you the opportunity to apply what you have learned from the lesson. There will be enough instruction provided for you to create the model but some of the steps may require you to plan ahead of time on how you should constrain the geometry, such as use only geometric relations, or use only dimensions, or use both.

1. Select the **Front plane** and open a new sketch.

Sketch a **Rectangle** and add the dimensions and relations needed to fully define the sketch.

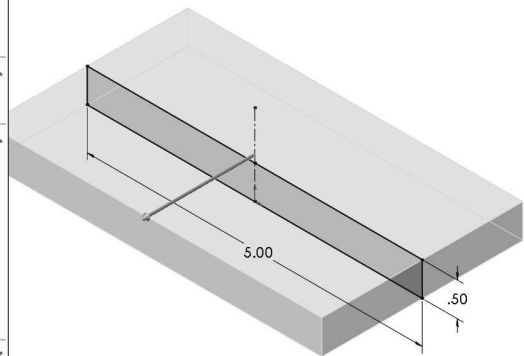
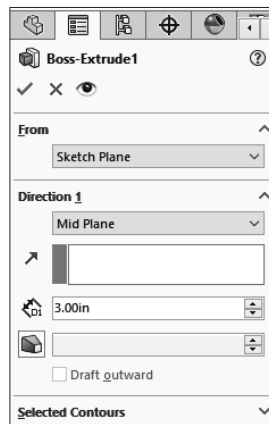


2. Change to the **Features** tab and click **Extruded Boss Base**.

Select **Mid-plane** for Direction 1.

Enter **3.00in.** for Depth.

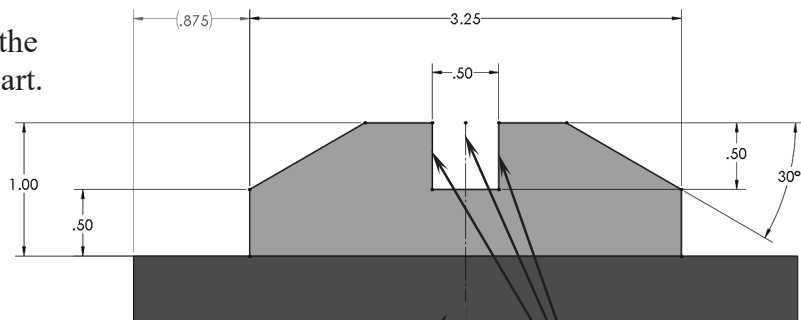
Click **OK**.



3. Open a new sketch on the face in the back of the part.

Sketch the profile shown.

Add the dimensions and relations as indicated to fully define the sketch.



Sketch on the face  
in the back

Symmetric relation  
between 3 lines

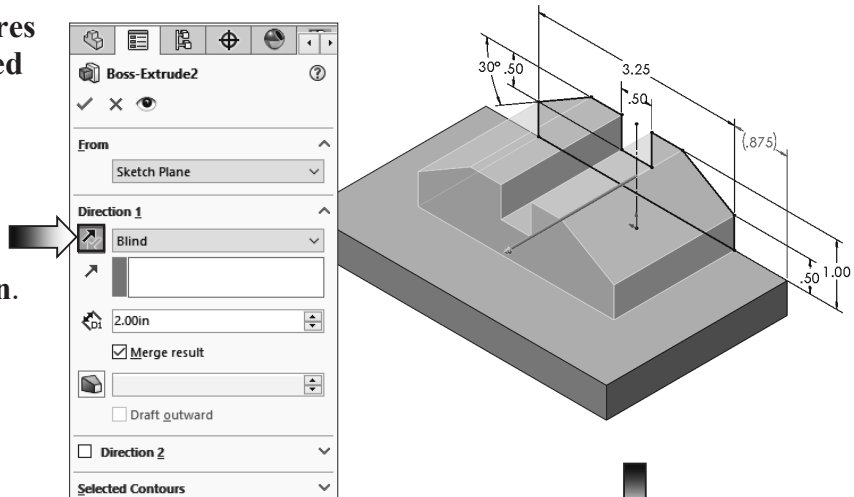
#### 4. Change to the Features tab and press **Extruded Boss Base**.

For Direction 1, select the **Blind** option.

For Depth, enter **2.00in**.

Click **Reverse** (arrow) to extrude towards the front.

Click **OK**.

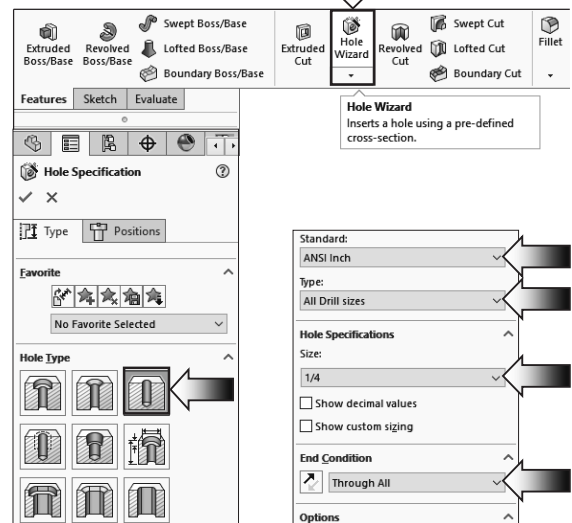


#### 5. Adding the Holes:

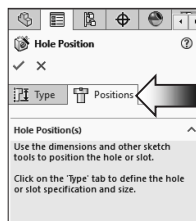
Click the **Hole Wizard** command on the Features tab.

Set/select the following:

- \* Hole Type: **Hole**
- \* Standard: **ANSI Inch**
- \* Type: **All Drill Sizes**
- \* Size: **1/4**
- \* End Condition: **Through All**

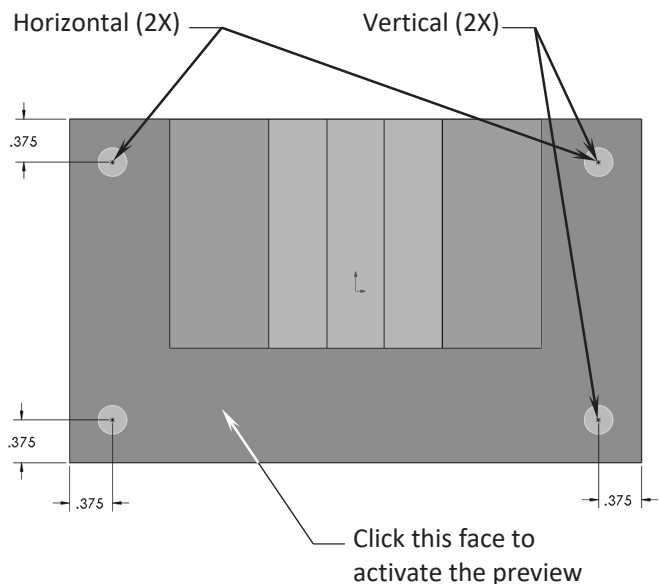


Click the **Position** tab (arrow).



Select the face as indicated to activate the preview graphics.

Place 4 holes and add the dimensions/relations as noted to fully define the sketch.



Click **OK** to accept and exit the Hole Wizard command.

## 6. Adding Fillets:

Click the **Fillet** command on the Features tab.

The **Constant Size** (arrow) option should be selected by default.

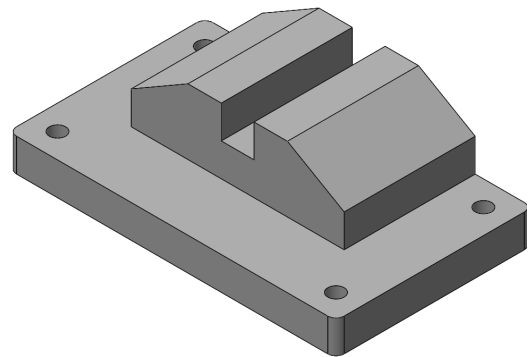
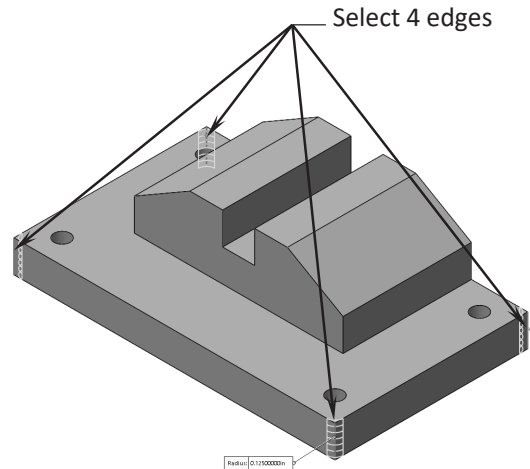
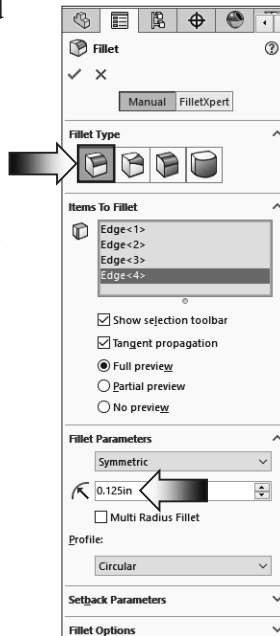
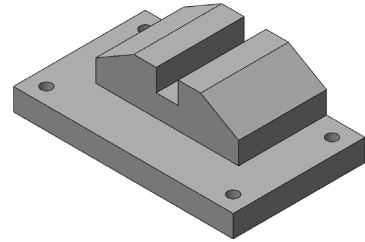
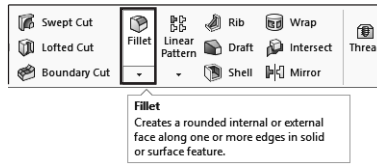
For Radius size, enter **.125in**.

For Items to Fillet, select the **4 vertical edges** as indicated.

Enable the **Full Preview** if needed.

Click **OK**.

The fillets are added to the selected edges.



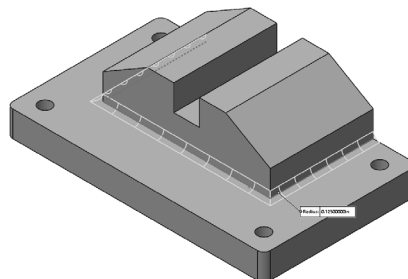
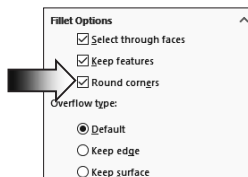
## 7. Saving your work:

Select **File, Save As**.

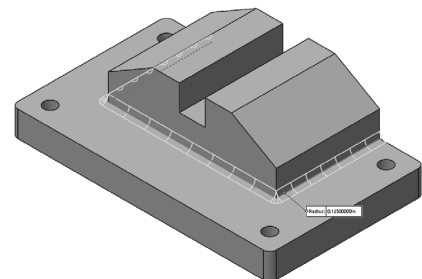
Enter **Extrudes\_Exe1** for the file name.

Click **Save**.

(Try out the option **Round Corners**.)



Without Round Corners

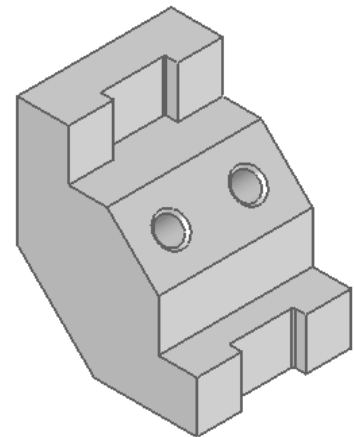
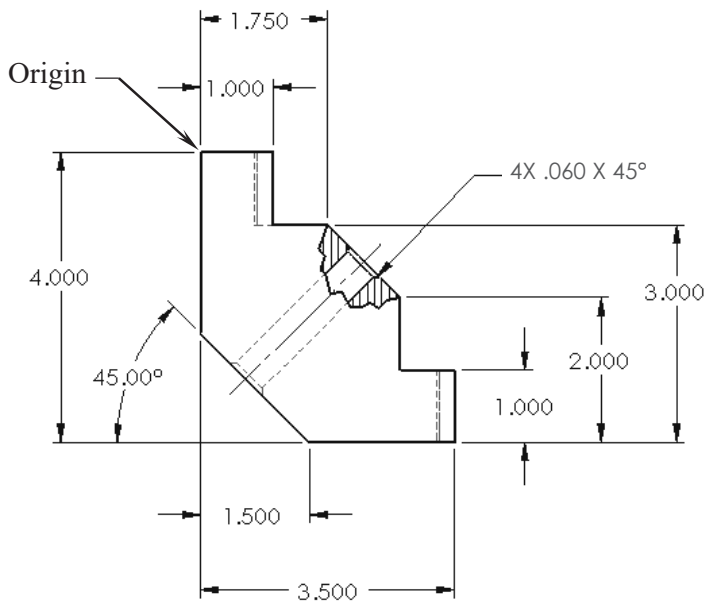
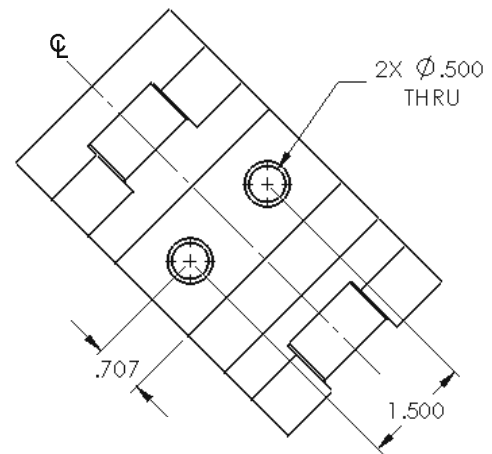
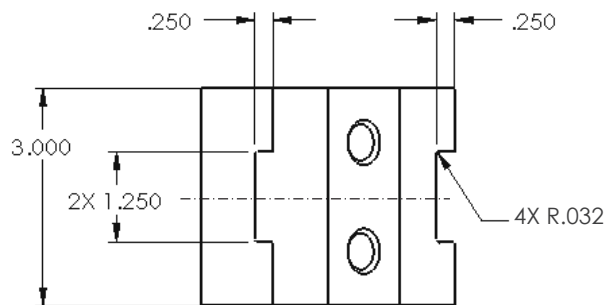


With Round Corners

## Exercise 2: Extrude Boss & Extrude Cut

**NOTE:** In an exercise, there will be less step-by-step instruction than those in the lessons which will give you a chance to apply what you have learned from the previous lesson and create the model on your own.

1. Dimensions are in inches, 3 decimal places.
2. Use Mid-Plane end condition for the Base feature.
3. The part is symmetrical about the Front plane.
4. Use the instructions on the following pages if needed.



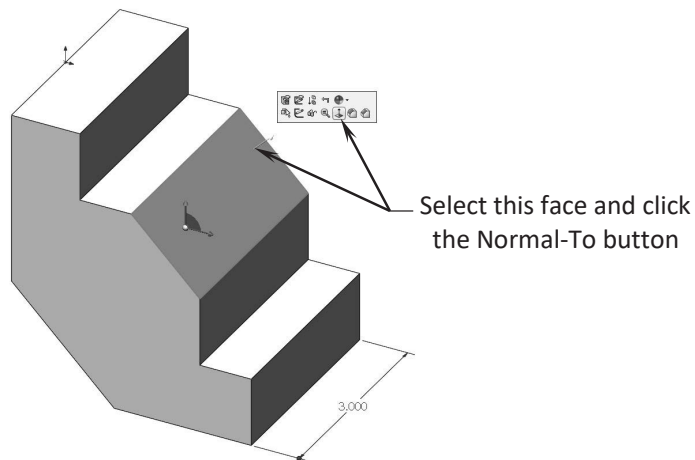


## 2. Adding the through holes:

Select the face as indicated and click the Normal-To button.

This command rotates the part normal to the screen.

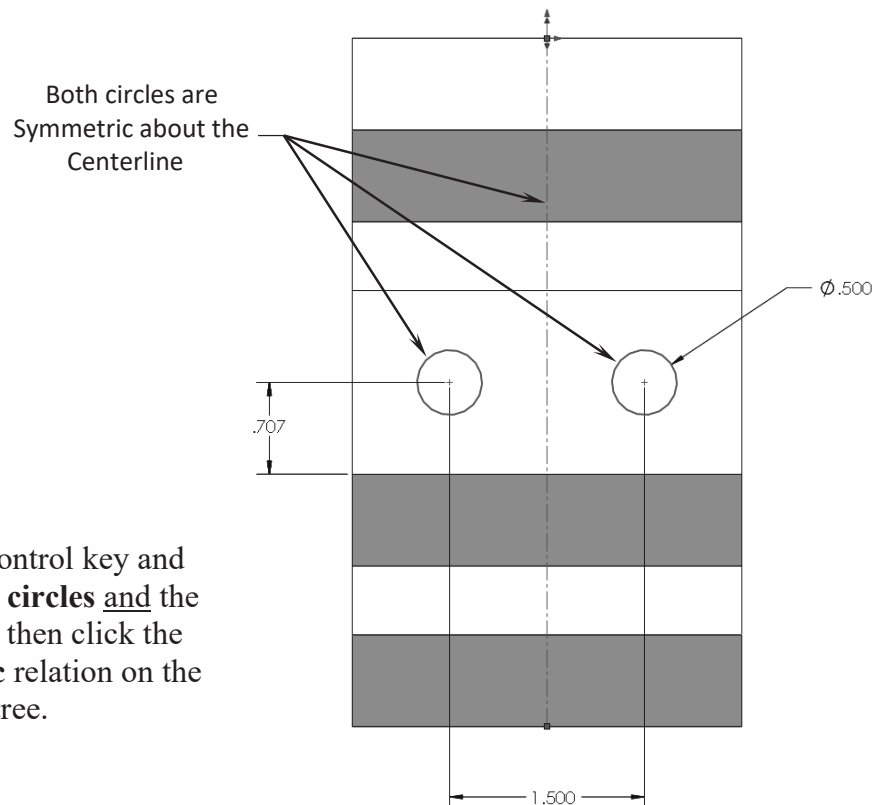
The hotkey for this command is **Ctrl+8**.



Open a **new sketch** and draw a **centerline** that starts from the origin point.

Sketch **2 circles** on either side of the centerline.

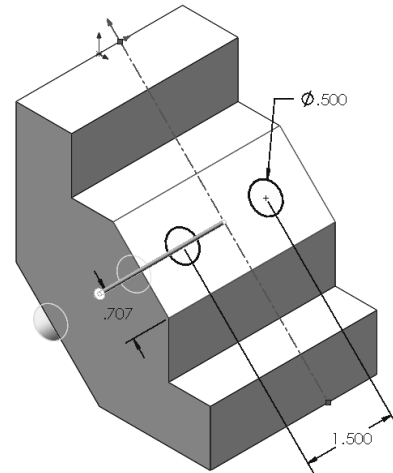
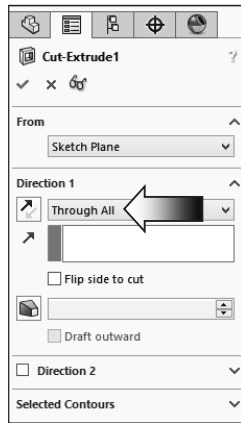
Add the diameter and location dimensions shown. Push Escape when done.



Hold the Control key and select **both circles** and the **centerline**, then click the **Symmetric** relation on the properties tree.



Create an extruded cut using the **Through-All** end condition.



### 3. Adding the upper cut:

Select the upper face and click the Sketch button to open a **new sketch**.

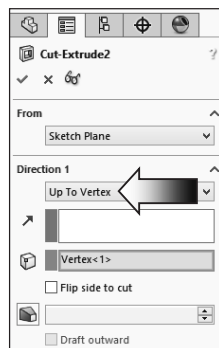
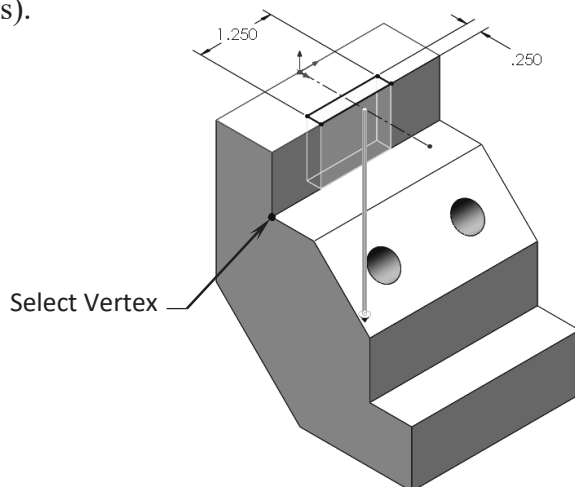
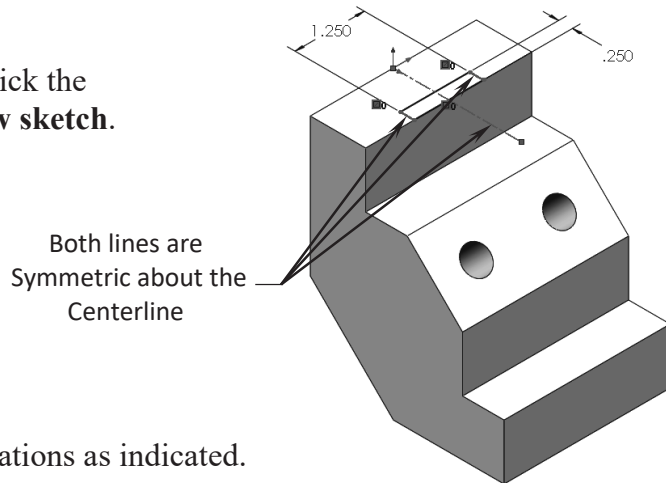
Sketch a **centerline** that starts at the Origin.

Sketch a **rectangle** as shown.

Add the dimensions and relations as indicated.

Create an **extruded cut** using the **Up-To-Vertex** condition (up-to-surface also works).

Select the **Vertex** indicated.



Click **OK**.

#### 4. Adding the lower cut:

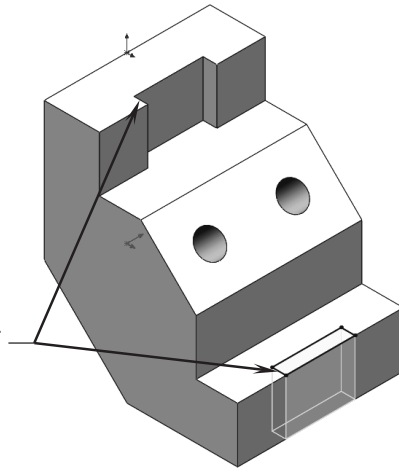
Select the lower face of the part and open a **new sketch**.

Sketch a **rectangle** on this face.

Add a **Collinear** and an **Equal** relation to the **lines** and the **edges** as noted.

Extrude a cut using the **Through All** condition.

The line is Collinear and Equal with the edge on both sides



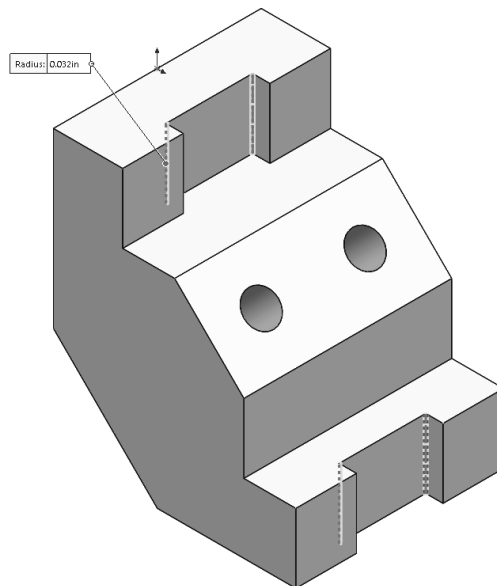
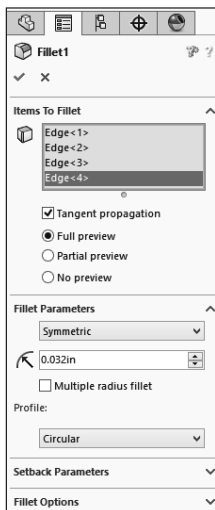
#### 5. Adding fillets:

Select the **Fillet** command from the Features toolbar.

Enter **.032in.** for radius size.

Select the **4 vertical edges** on the inside of the 2 cuts.

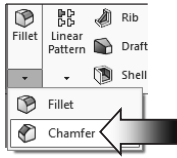
Keep all other options in their default settings.



Click **OK**.

## 6. Adding chamfers:

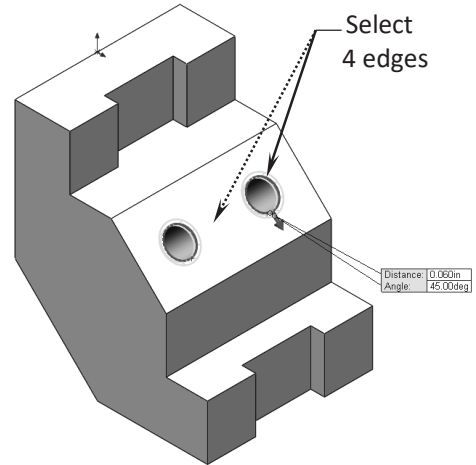
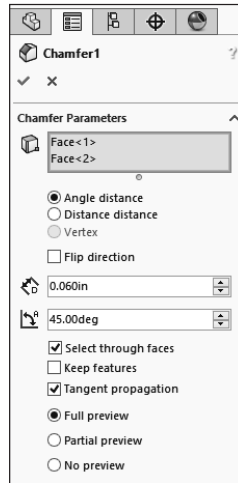
Click **Chamfer** under the Fillet drop-down.



Enter **.060in.** for depth.

Select the **4 circular edges** of the 2 holes (both front and back).

Click **OK**.



## 7. Saving your work:

Click **File / Save As**.

Enter **Extrudes\_Exe2** for the file name.

Select a location to save the file.

Click **Save**.

