

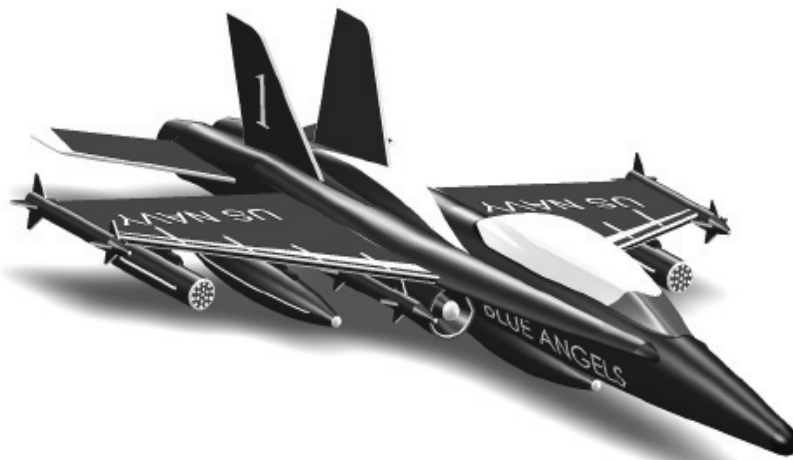
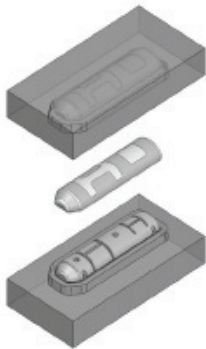
Inside:

SUPPLEMENTAL  
FILES ON CD



# SolidWorks 2011

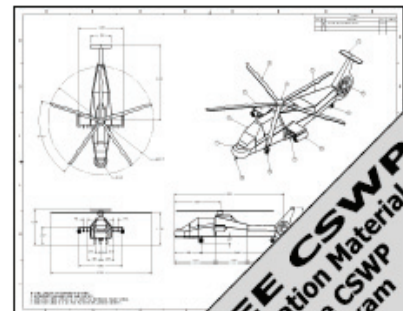
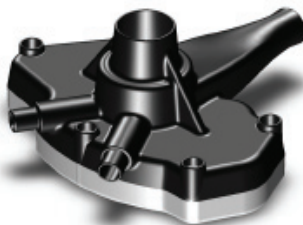
## Part I - Basic Tools



### Introductory Level Tutorials

#### Parts, Assemblies and Drawings

Written by: Sr. Certified SolidWorks Instructor  
Paul Tran, CSWP, CSWI



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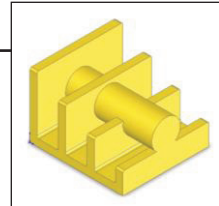
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# CHAPTER 3

## Basic Solid Modeling

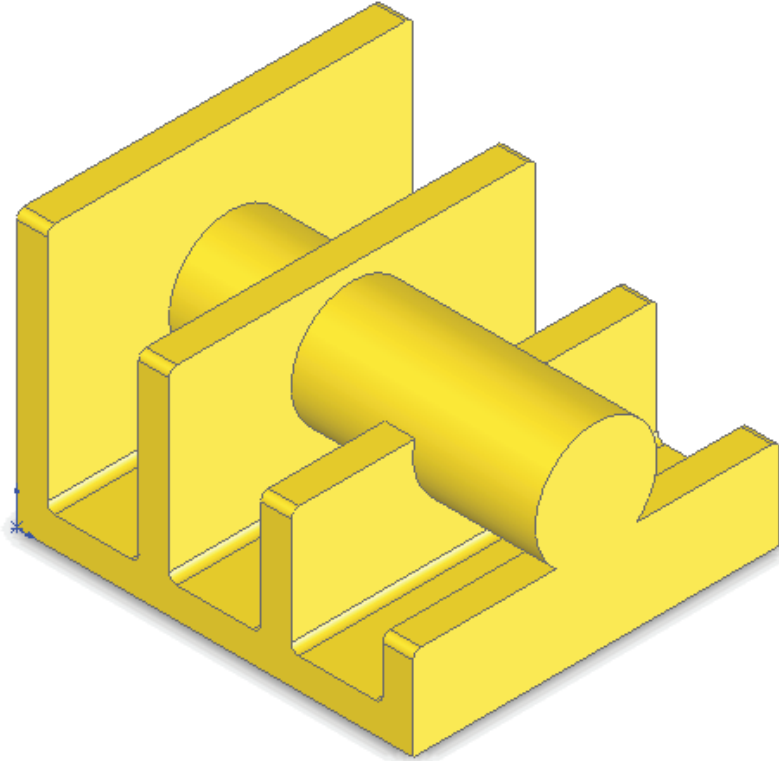


### Basic Solid Modeling Extrude Options

- Upon successful completion of this lesson, you will be able to:
  - \* Sketch on planes and/or planar surfaces.
  - \* Use sketch tools to construct geometry.
  - \* Add 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.
  - \* Activate Sketch pencil  .
  - \* Sketch the profile using the sketch tools    .
  - \* Define the profile with dimensions  or relations  .
  - \* Extrude the profile  .
  
- Be sure to review self-test questionnaires at the end of the lesson, prior to going to the next chapter.

# Basic Solid Modeling

## Extrude Options



Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

### Tools Needed:



Insert Sketch



Line



Circle



Add Geometric Relations



Dimension



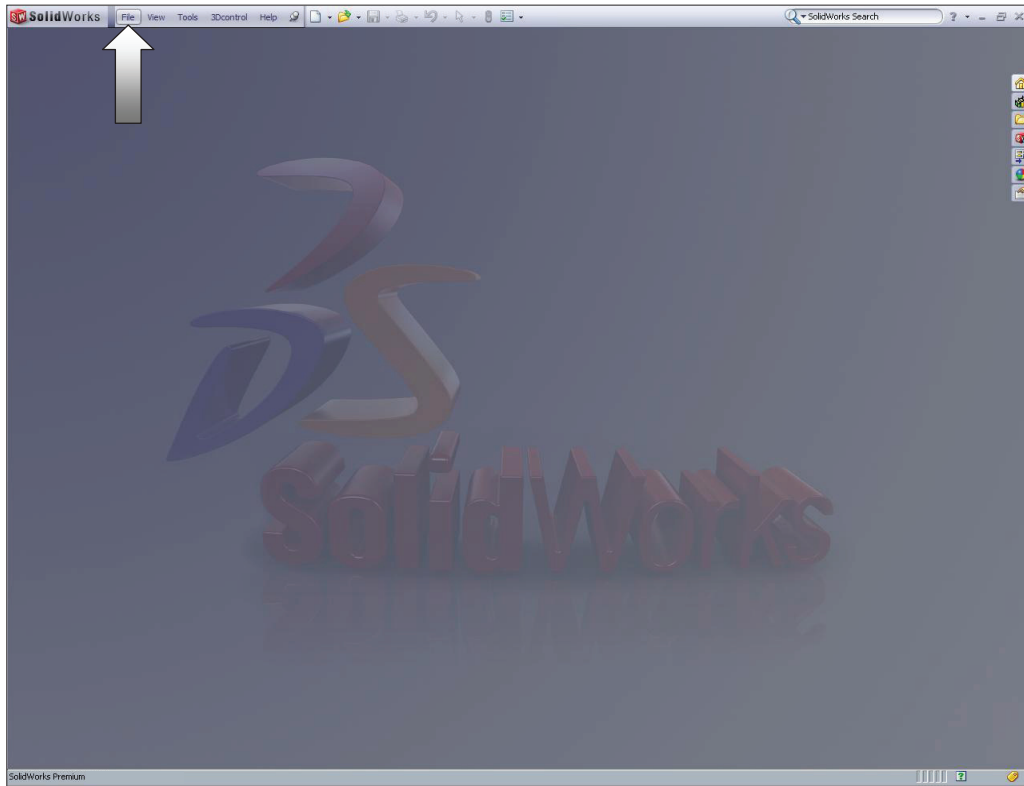
Sketch Fillet



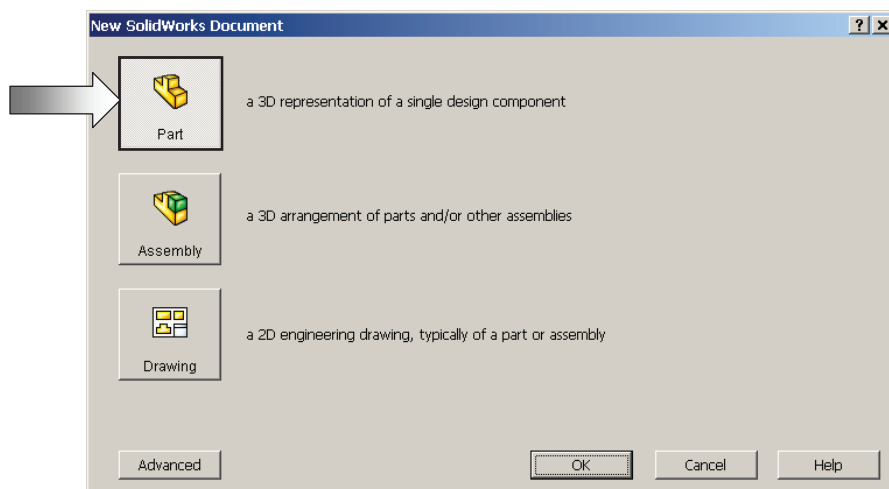
Boss / Base Extrude

## 1. Starting a new Part:

- From the **File** menu, select **New / Part**.

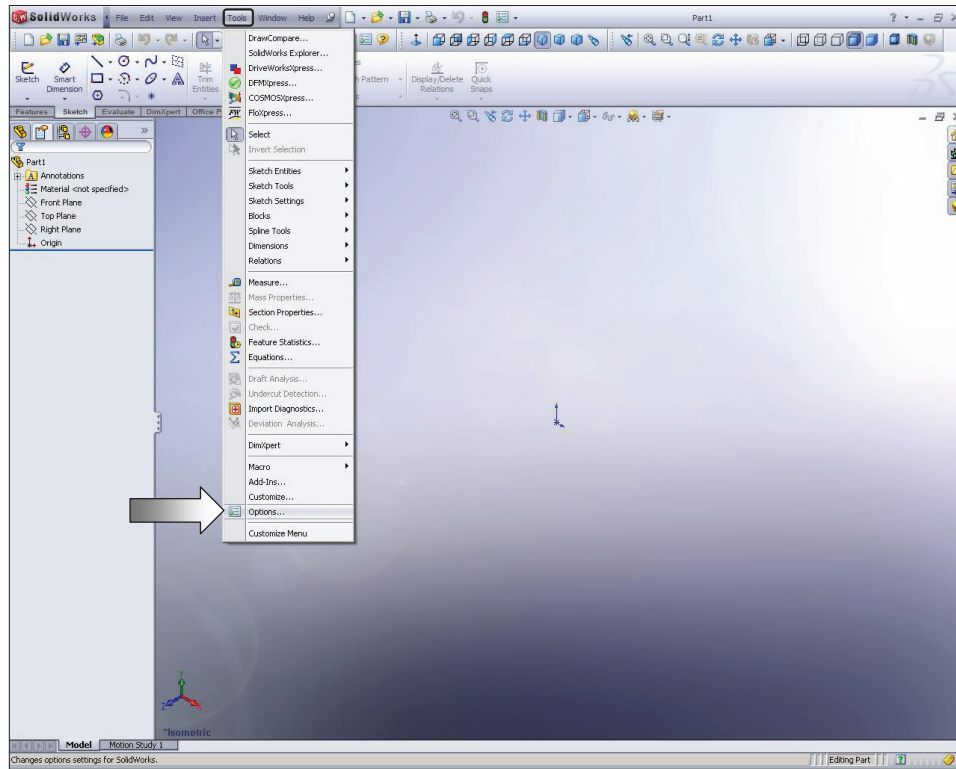


- Select the **Part** template from either the Templates or Tutorial folders.



- Click **OK** ; a new part template is opened.

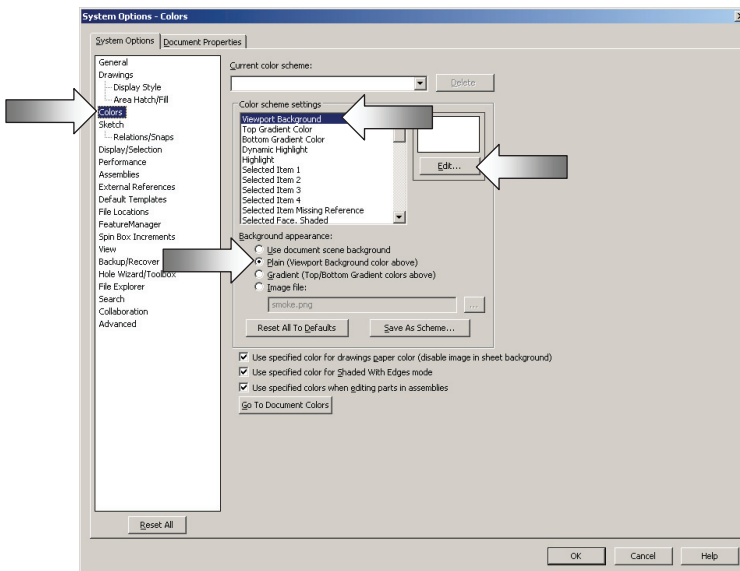
- The next step is to set up your system options.



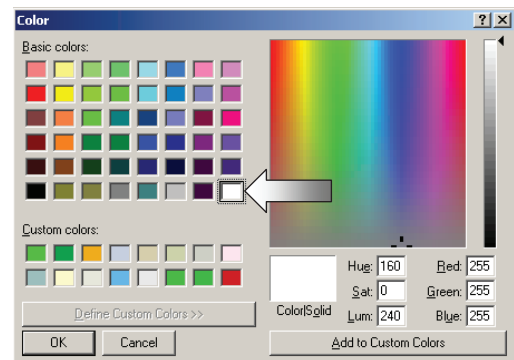
## 2. Changing the Background color:

- Select **Tools / Options / Colors**.


- Under **Background Appearance**, select **Plain** (Viewport Background Color), and choose the White color if not yet selected (arrows).

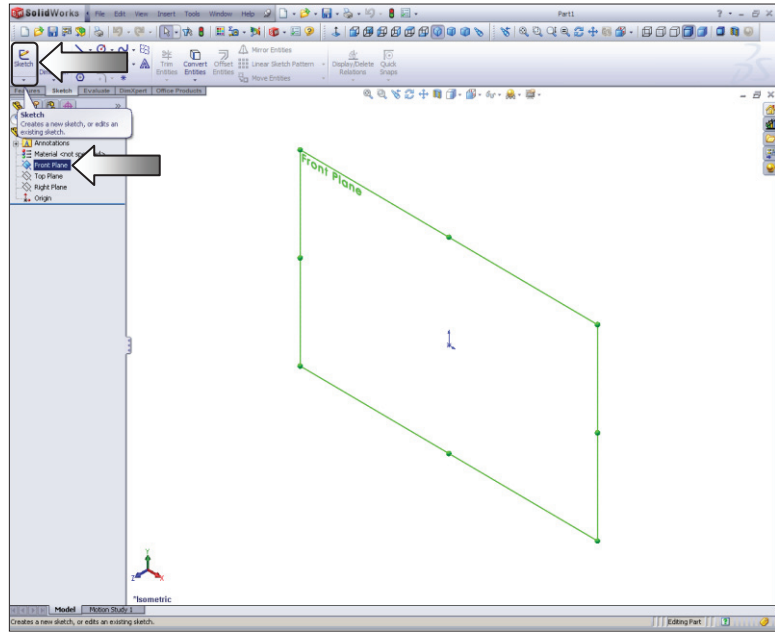



- This will change your background color to **white**.

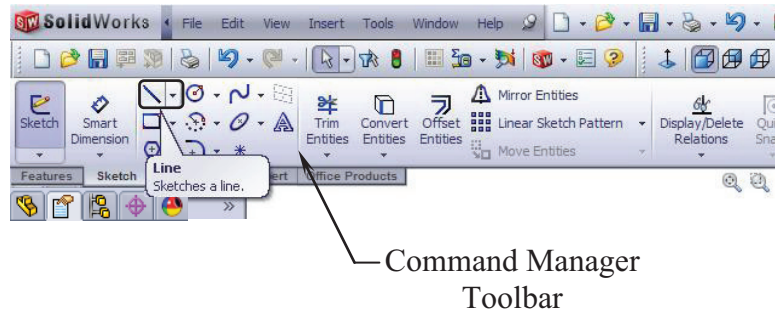


### 3. Starting a new Sketch:

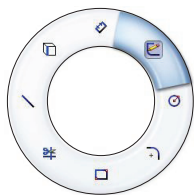
- Select the Front plane from the FeatureManager tree and then click the Pencil icon  to open a new sketch.



- From the Command Manager toolbar, select the Line  command.



Command Manager Toolbar

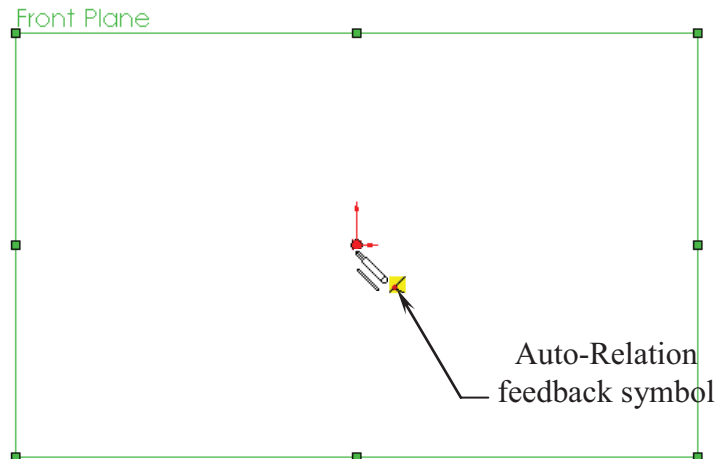


Mouse Gesture

**OPTION:**

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

- Position the mouse cursor at the Origin point, a yellow feedback symbol pops up to indicate that a relation (Coincident) is going to be added automatically to the 1<sup>st</sup> endpoint of the line.

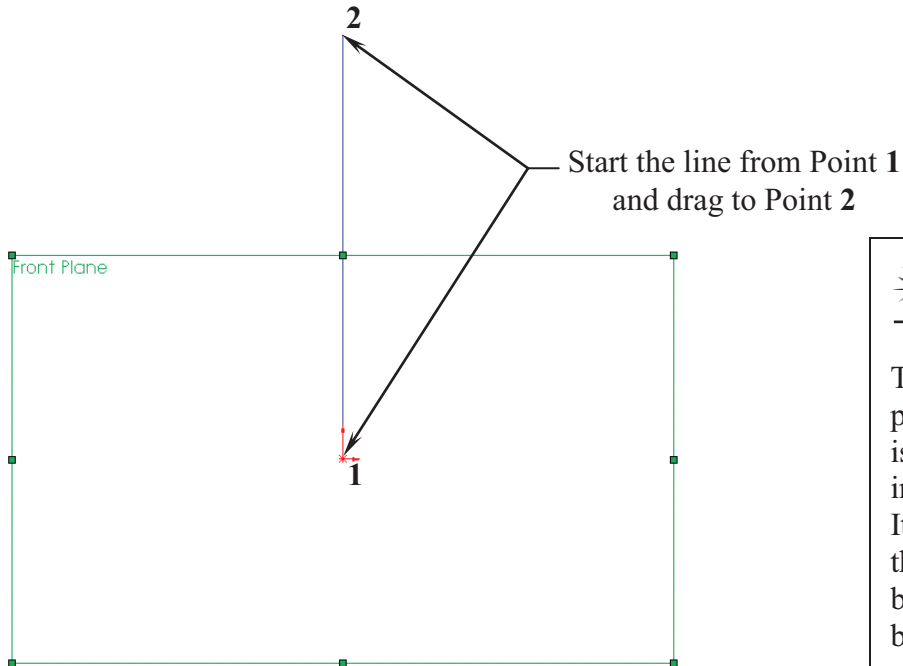


Auto-Relation feedback symbol



#### 4. Using the line command:

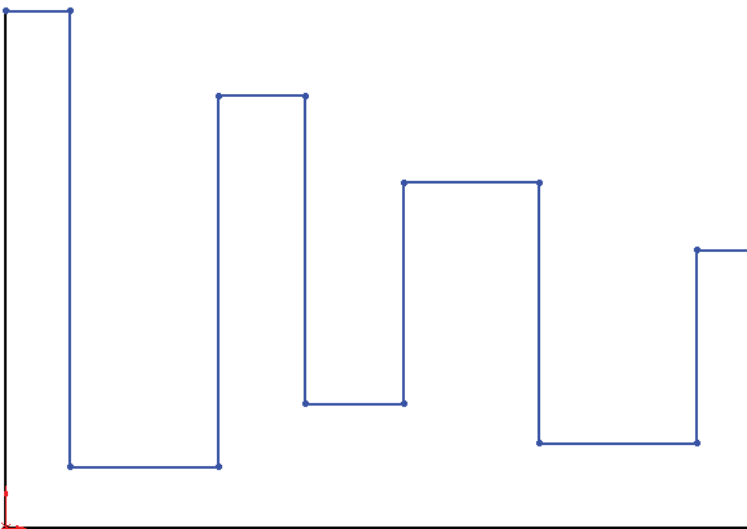
- Click on the Origin point and **hold** the mouse button to start the line at point 1, **drag upwards** to point 2, and then release the mouse button.





#### The Base Sketch

The Base Sketch is the parent sketch of a part and is also the very first sketch in a part document. It should mostly describe the basic shape of the part, before other features can be added on to it.

- Continue sketching the other lines using the **Click-Hold-Drag** technique.
- The size and shape of the profile will be corrected in the next few steps.




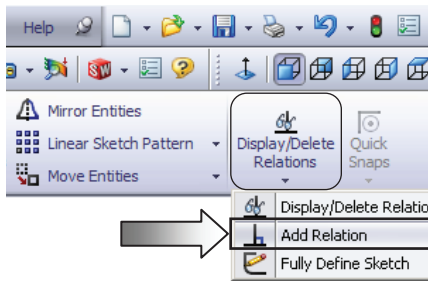
#### System Feedback

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




## 5. Adding Geometric Relations\*:

- Click  under Display/Delete Relations - OR - select **Tools / Relations / Add**.
- Select the 4 lines as shown below.
- Click **EQUAL** from the Add Geometric Relation dialog box.



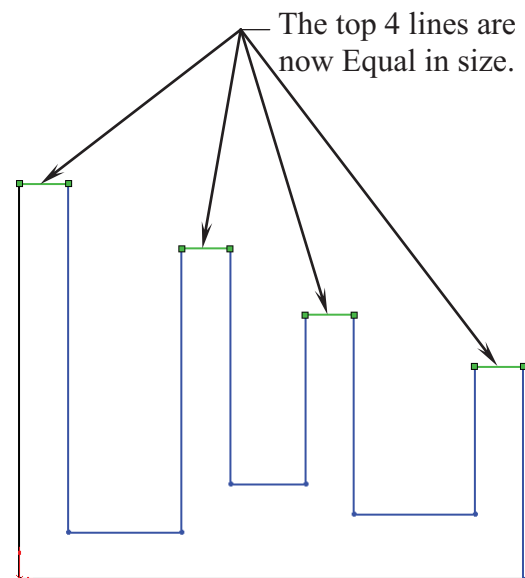
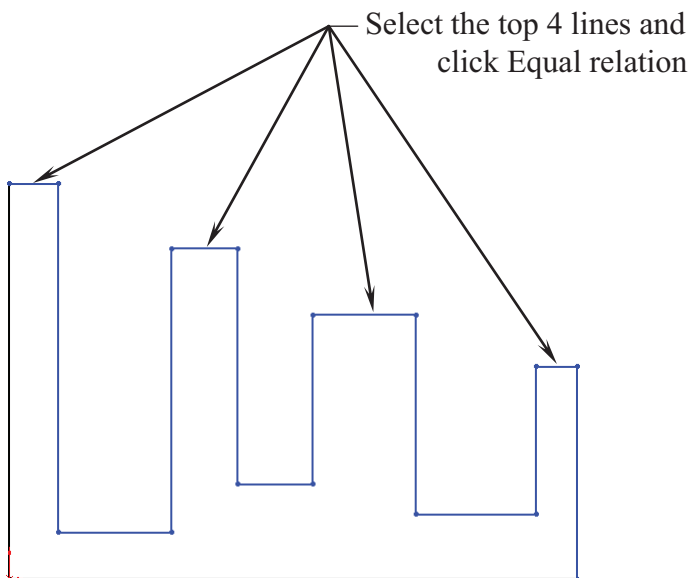
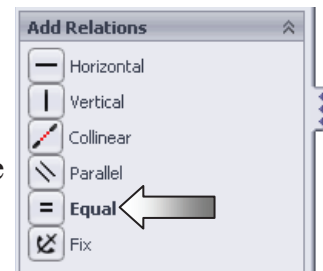
\* Geometric relations are one of the most powerful features in SolidWorks. They're used in the sketch level to control the behaviors of the sketch entities and to keep them associated with one another. When applying geometric relations between the items, one of the items should be a 2D entity and the other item can either be a 2D sketch entity, or a model edge, a plane, an axis, or a curve, etc.





### Equal Relations

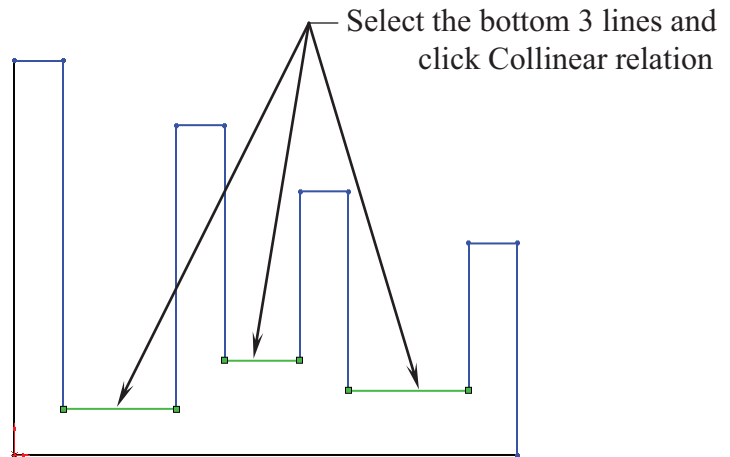
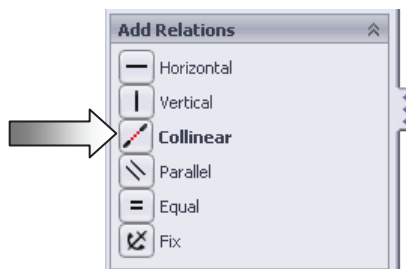
Adding the EQUAL relations to these lines eliminates the need to dimension each line.


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



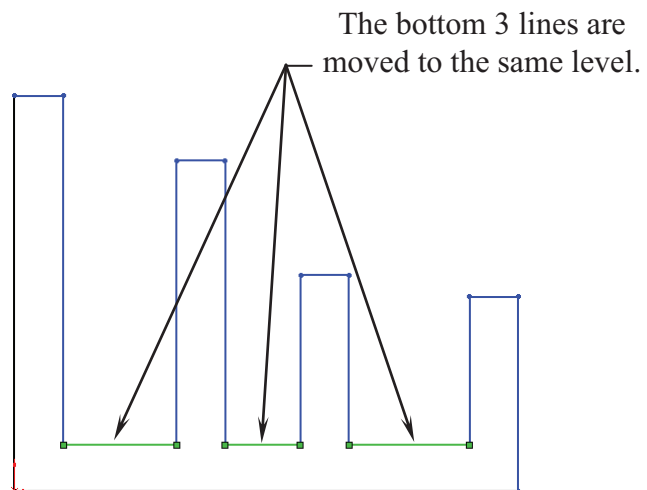
## 6. Adding a Collinear relation\*\*:

- Select the Add Relation  command again.
- Select the 3 lines as shown below.
- Click **COLLINEAR** from the Add Geometric Relations dialog box.
- Click **OK** .



 **Collinear Relations**

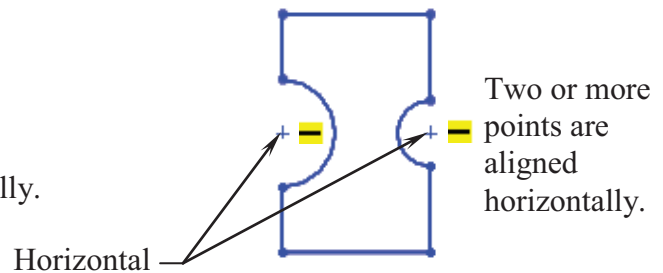
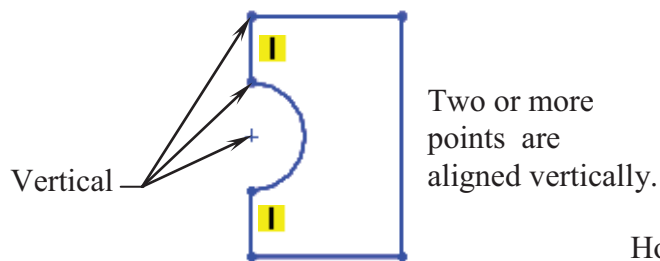
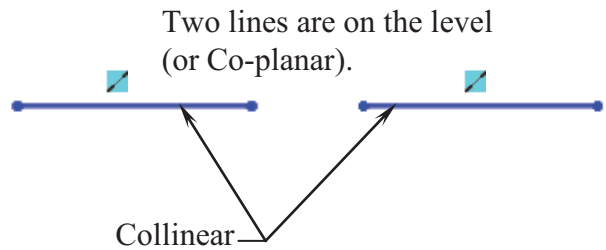
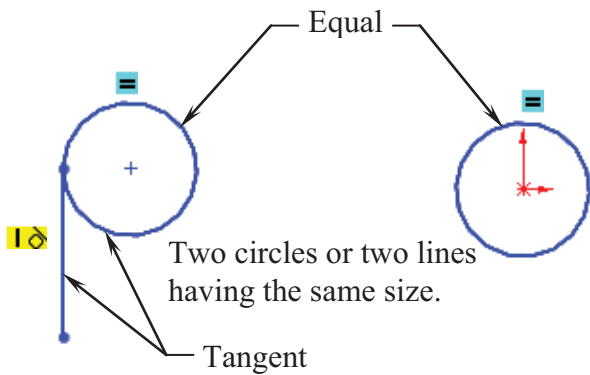
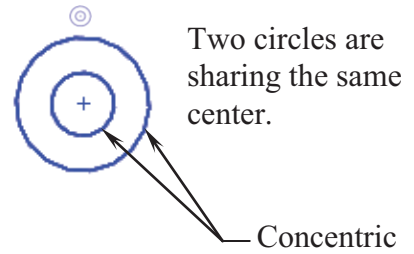
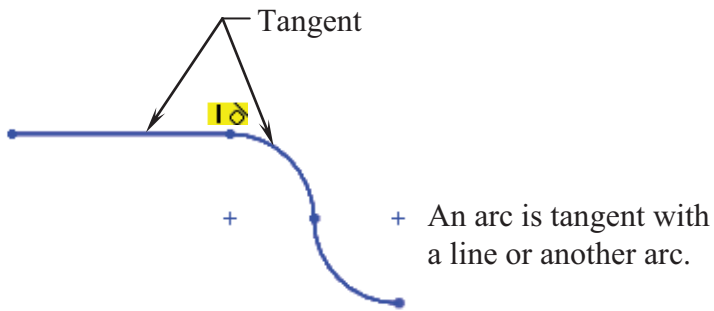
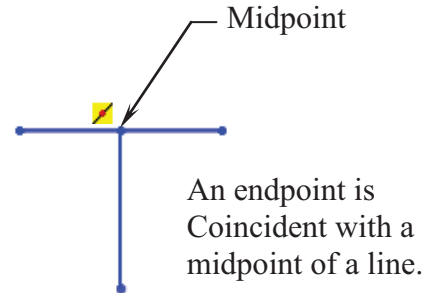
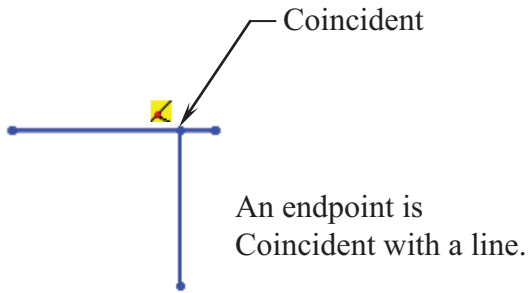
Adding a Collinear relation to these lines puts them on the same height level; only one dimension is needed to drive the height of all 3 lines.




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

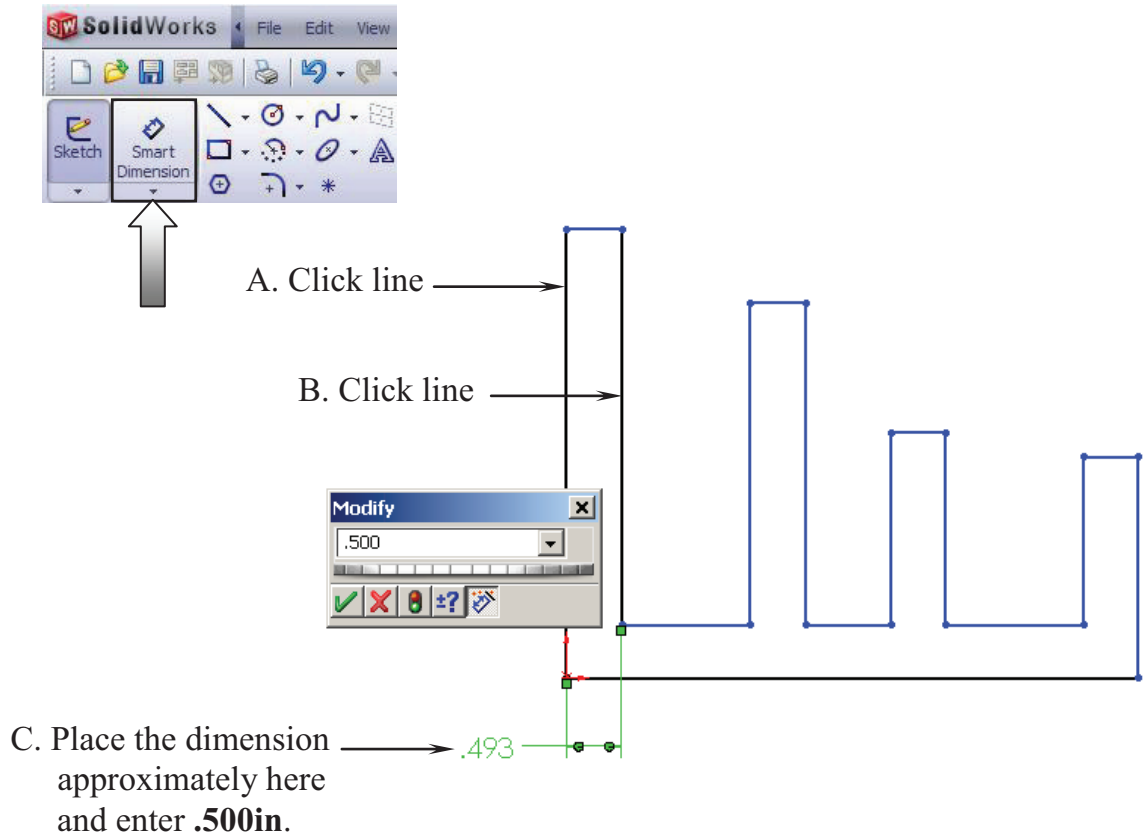
- Collinear a line to another line(s).
- Collinear a line(s) to an edge of a model.

## Geometric Relations Examples



## 7. Adding the horizontal dimensions:

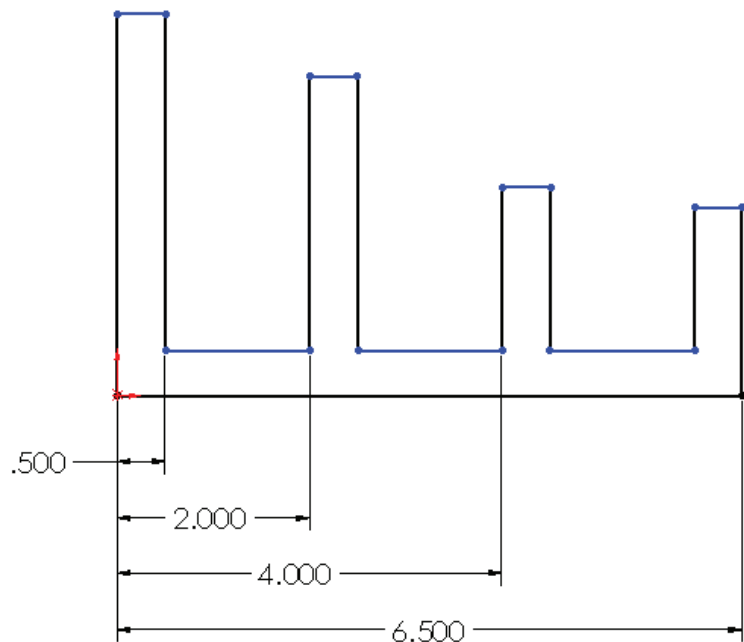
- Select  from the Sketch toolbar - OR - select **Insert / Dimension**, and add the dimensions as shown below (follow the 3 steps A, B and C).



- Continue adding the horizontal dimensions as shown here.

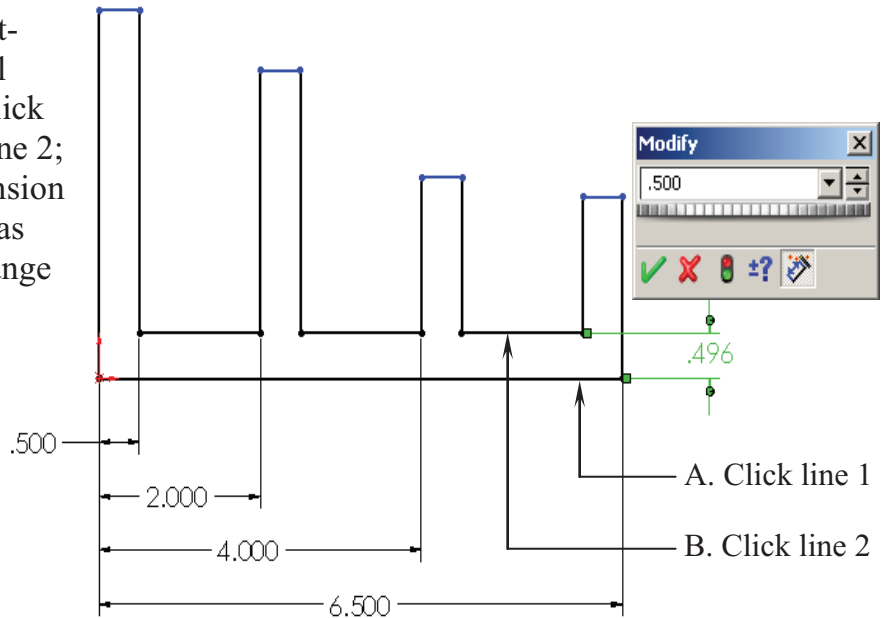
***NOTE:***

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

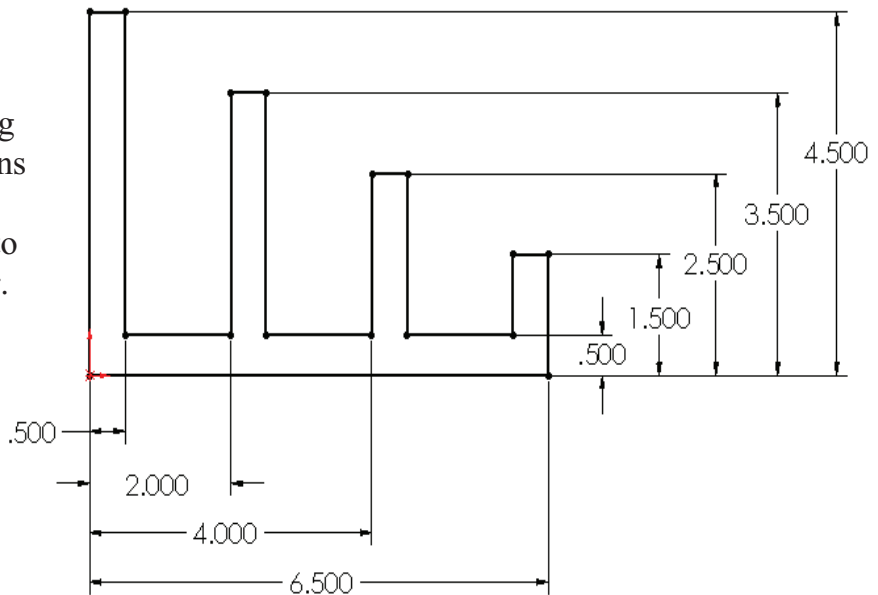


## 8. Adding the Vertical dimensions:

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



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

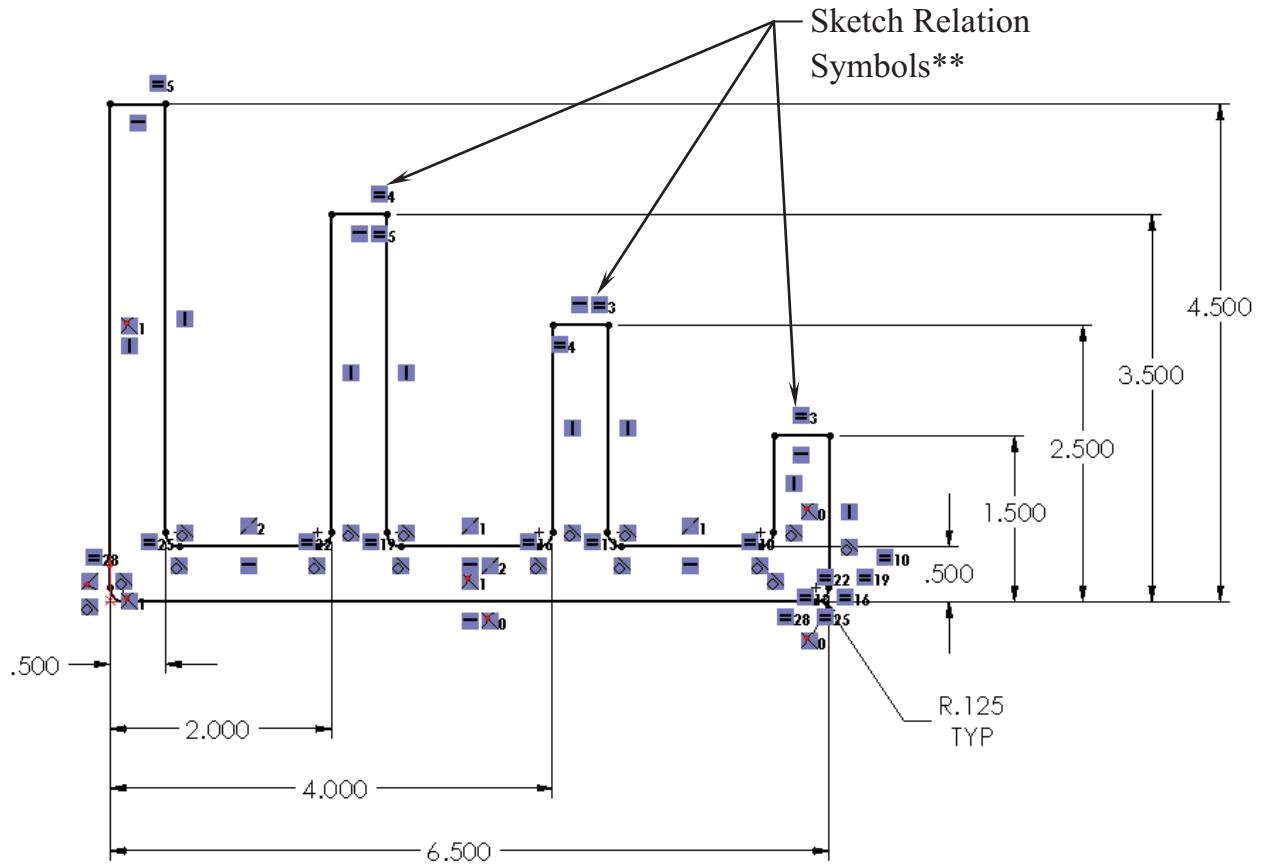


### The Status of a Sketch:

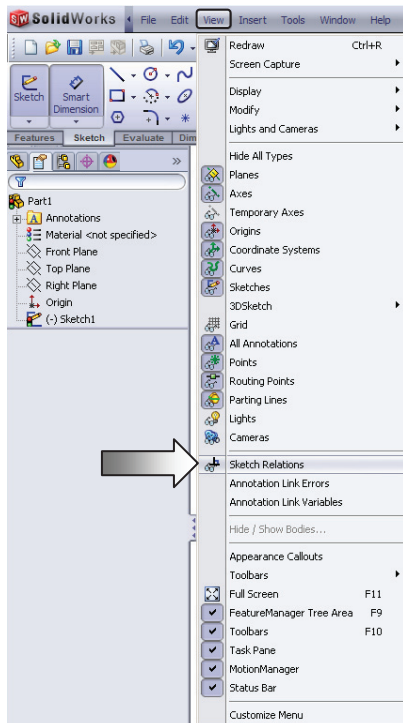
The current status of a sketch is displayed in the lower right corner of the screen.

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





### 9. Hiding the Sketch Relation Symbols:




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

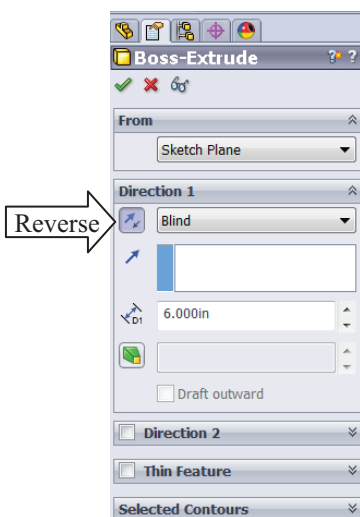
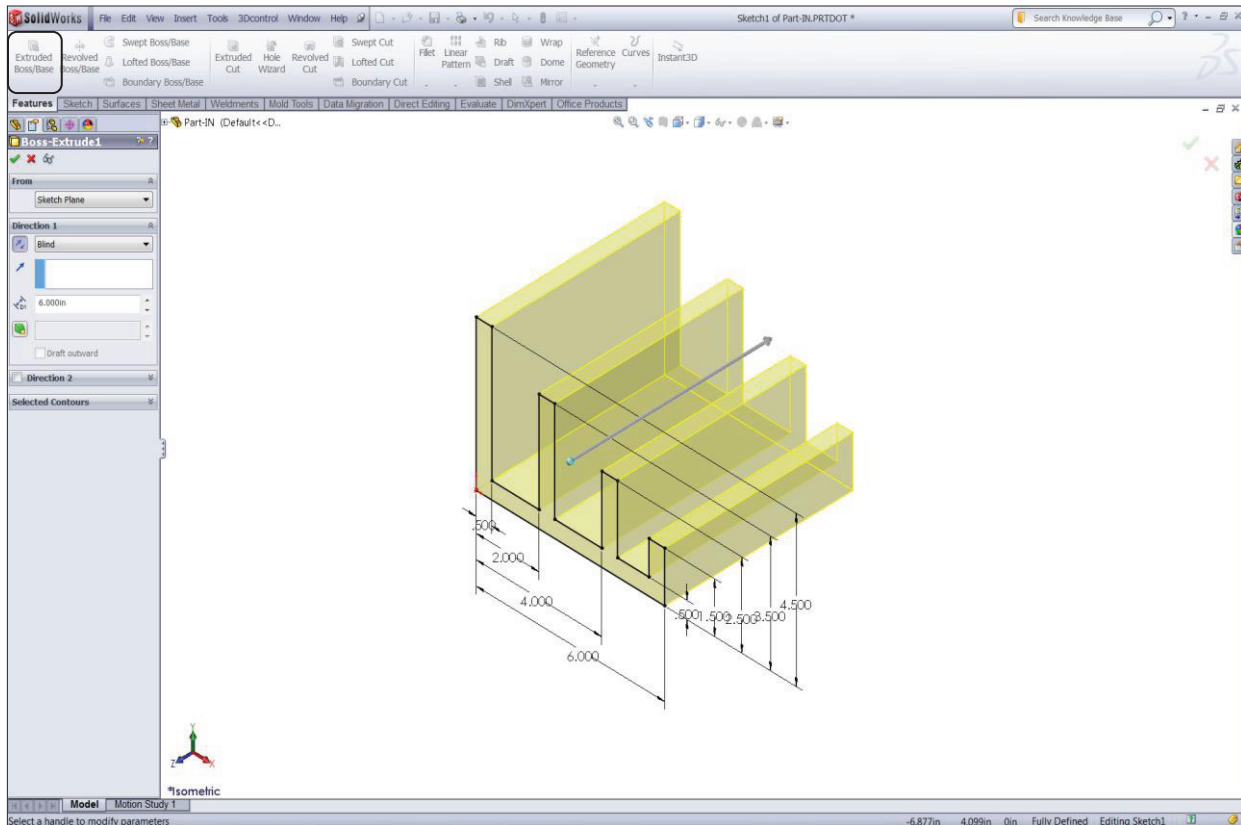
- To hide or show the Sketch Relation Symbols, go to the **View** menu and uncheck the **Sketch Relations** option.


#### Sketch Relation Symbols at a Glance

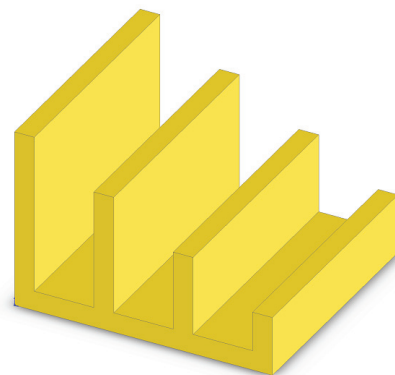
	Horizontal relation		Vertical relation
	Equal relation		Coincident relation
	Tangent relation		Collinear relation

## 10. Extruding the Base:

- The **Extrude Boss/Base** command is used to define the characteristic of a 3D linear feature.
- Click  from the Features toolbar - OR- select **Insert / Boss Base / Extrude**.





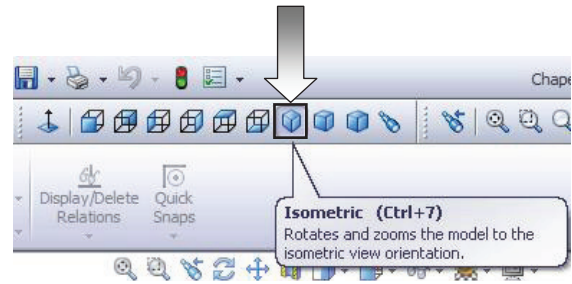
- Set the following:
- Direction: **Blind**.
- Depth: **6.00 in**.
- Enabled **Reverse** direction.
- Click **OK** .





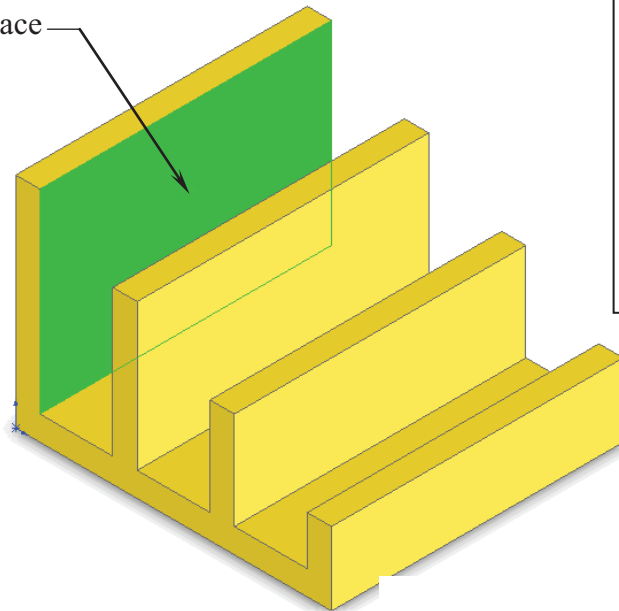
## 11. Sketching on a Planar Face:

- Select the face as indicated.
- Click  or select **Insert/Sketch**.
- Click  from the Sketch Tools toolbar
- OR - select **Tools / Sketch Entity / Circle**.



(From the View toolbar above the CommandManager, click the Isometric icon or press the shortcut keys **Ctrl+7**).

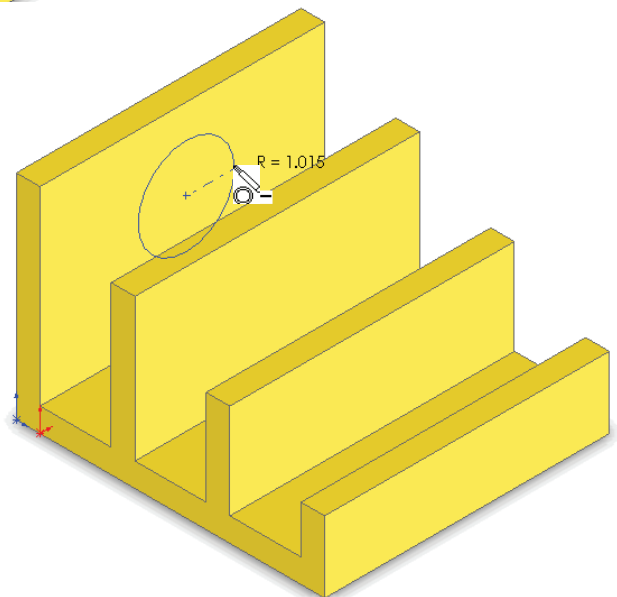
Select the Sketch Face




### **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.

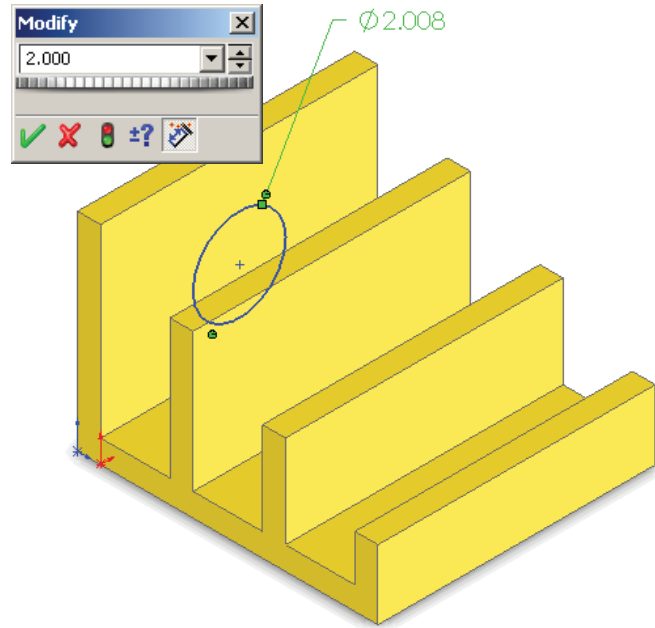
- Position the mouse cursor on the selected face, click and drag outward to draw a circle.
- While sketching the circle, the system displays the radius value next to the mouse cursor.



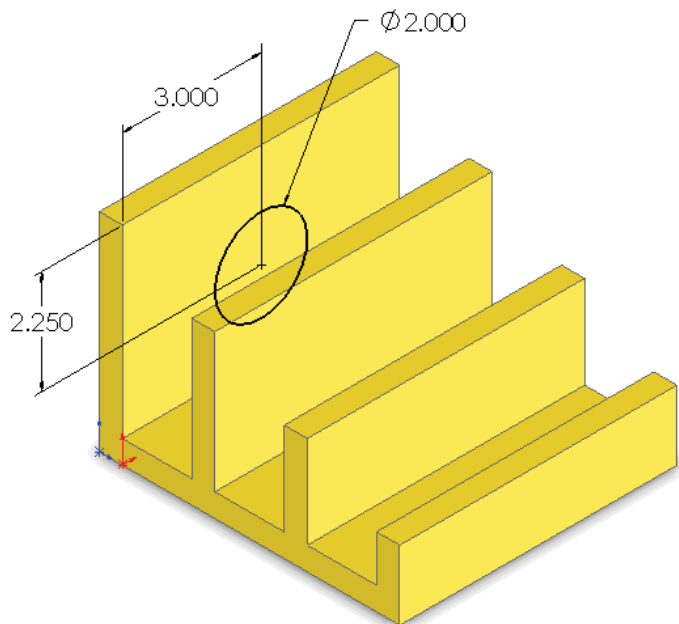
- Select the **Smart Dimension**

command  and add a diameter dimension to the circle.

(Click on the circle and move the mouse cursor outward, at approximately 45 degrees, and place it).




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

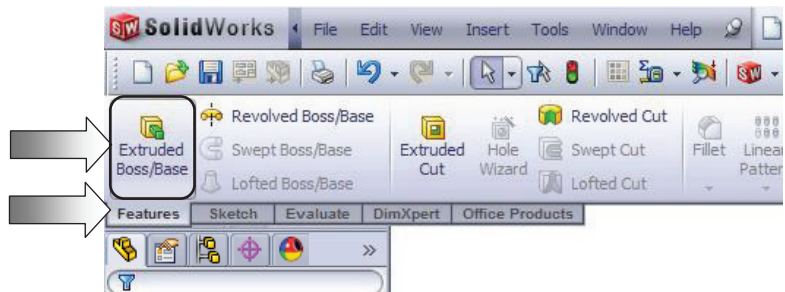


## 12. Extruding a Boss:

- Click  or select **Insert / Boss-Base / Extrude**.

 **Extrude Options...**

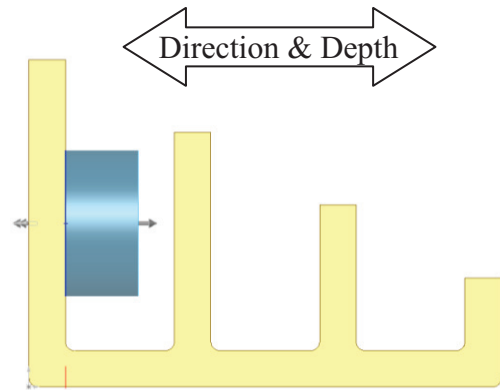
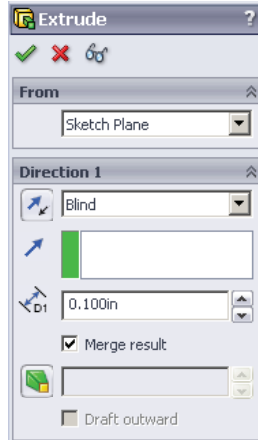
Explore each extrude option to see the different results. Press Undo to go back to the original stage.



**A Using the Blind option:**

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

- \* Direction
- \* Depth dimension

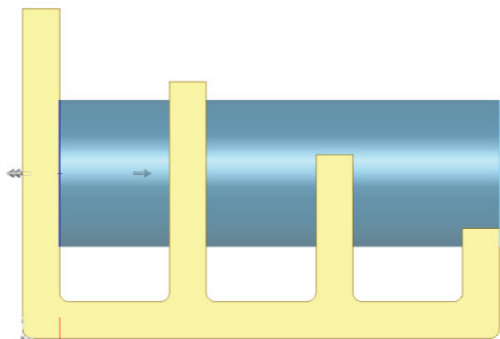
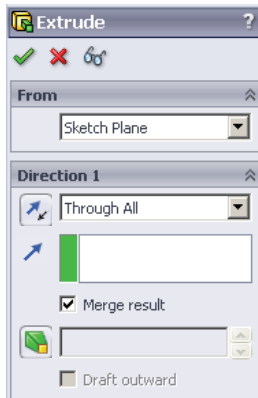


**Blind**  
Condition

- Drag the direction arrow on the preview graphics to define the direction, then enter a depth dimension to try out the Blind option.

**B Using the Through All option:**

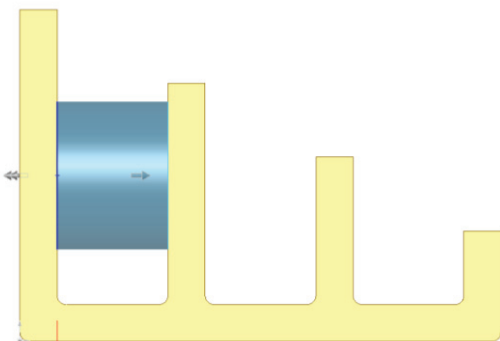
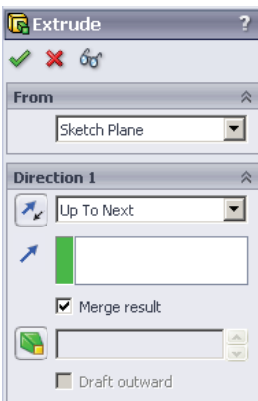
- When the Through All option is selected, the system automatically extrudes the sketch to the length of the part normal to the sketch plane.



**Through All**  
Condition

**C Using the Up To Next option:**

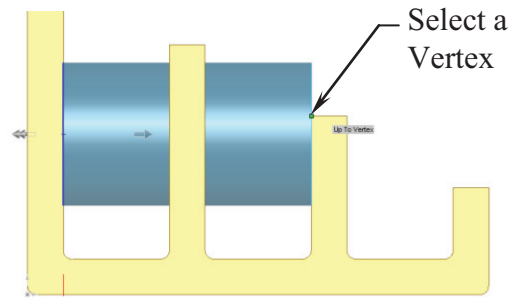
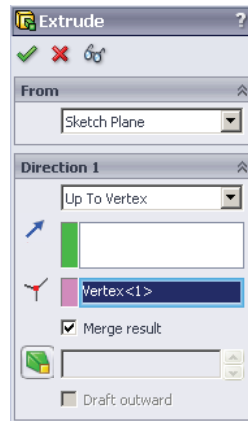
- With the Up To Next option, the system extrudes the sketch to the very next surface, and blends it to match the shape of the surface whether it is a planar or curved surface.



**Up To Next**  
Condition

**D Using the Up To Vertex option:**

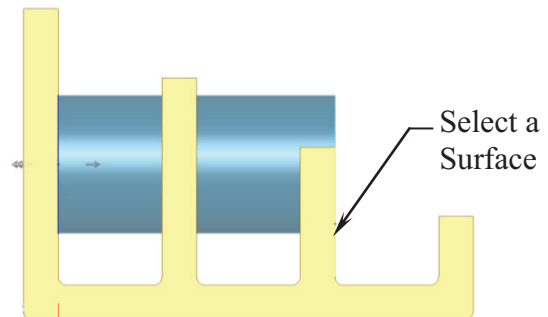
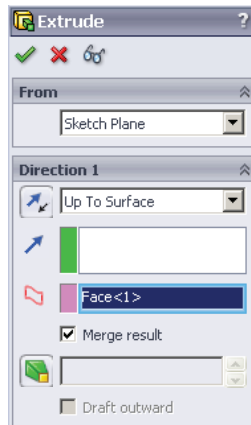
- This option extrudes the sketch from its plane to a vertex, specified by the user, to define its depth.



**Up To Vertex**  
Condition

**E Using the Up To Surface option:**

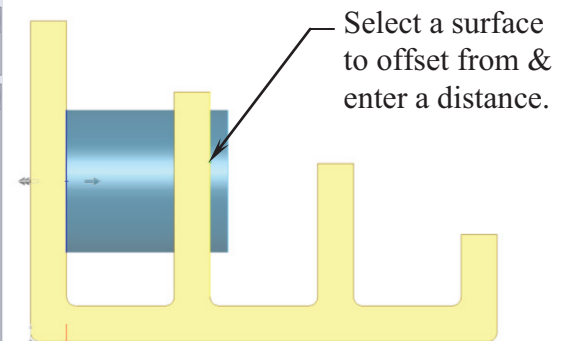
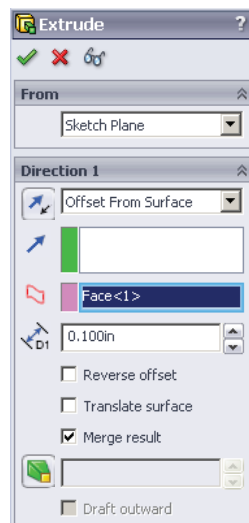
- This option extrudes the sketch from its plane to a selected surface or a face to define its depth.



**Up To Surface**  
Condition

**F Using the Offset From Surface option:**

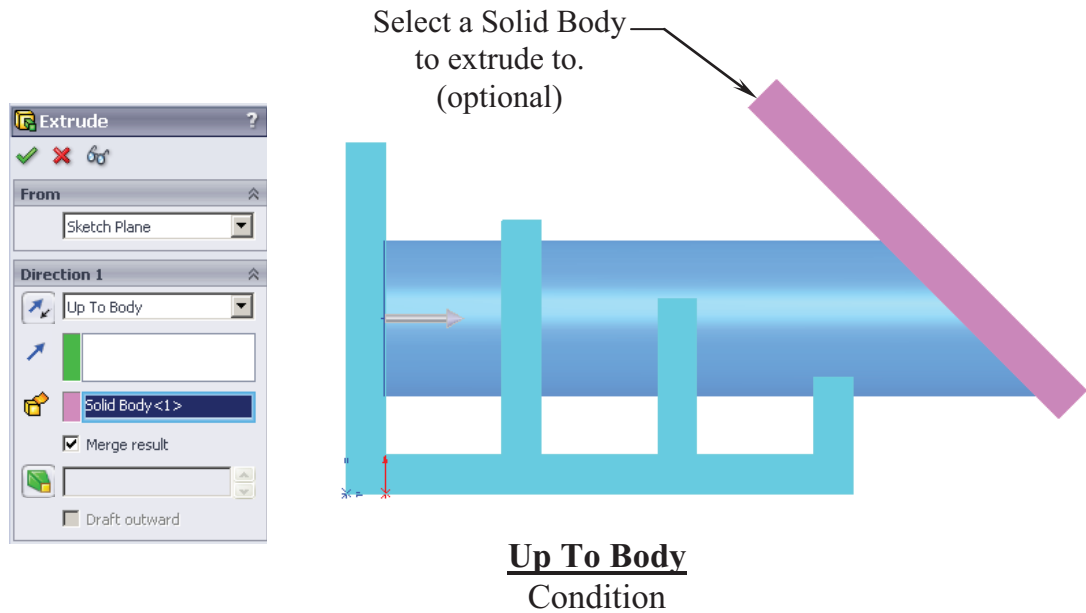
- This option extrudes the sketch from its plane to a specified distance from the selected face or surface.



**Offset From Surface**  
Condition

### G Using the Up To Body option:

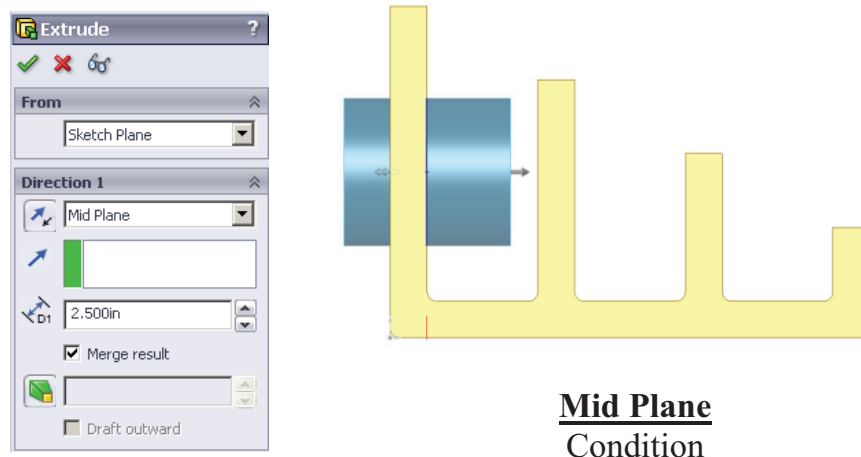
- This option extrudes the sketch from its sketch plane to a specified body.



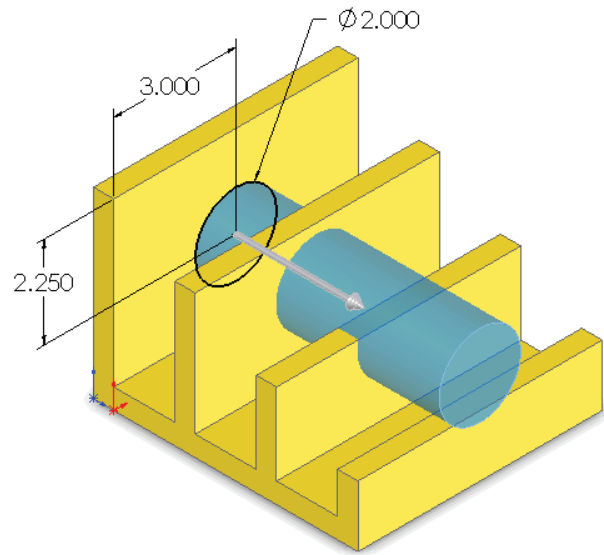
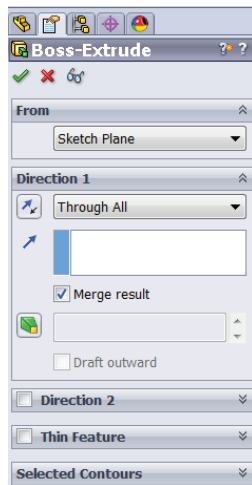
- The Up To Body option can also be used in assemblies or multi-body parts.

### 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.



- After you are done with exploring all the extrude options, change the final condition to: **Through All**

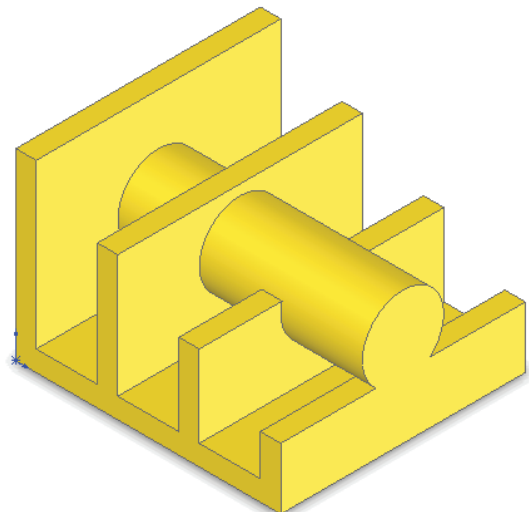


- Click OK .

- The system extrudes the circle to the outer most surface as the result of the Through All end condition.

***Extrude summary:***

- \* *The Extrude Boss/Base command is used to define the characteristic of a 3D feature.*
- \* *A sketch can be extruded in both directions at the same time, from its sketch plane.*
- \* *A sketch can also be extruded as a solid or a thin feature.*

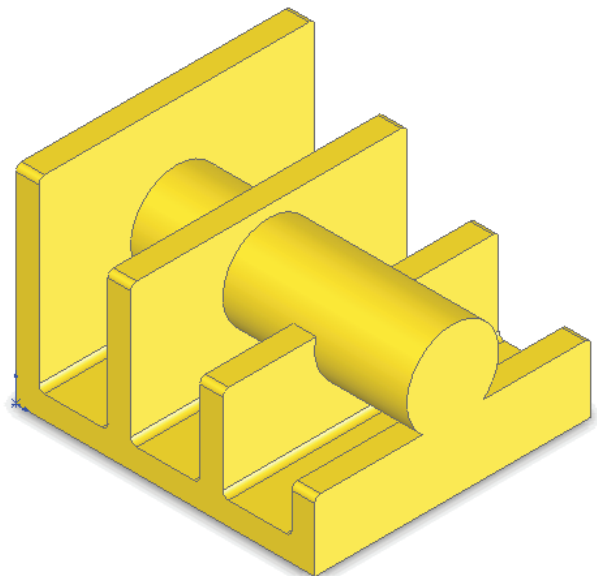
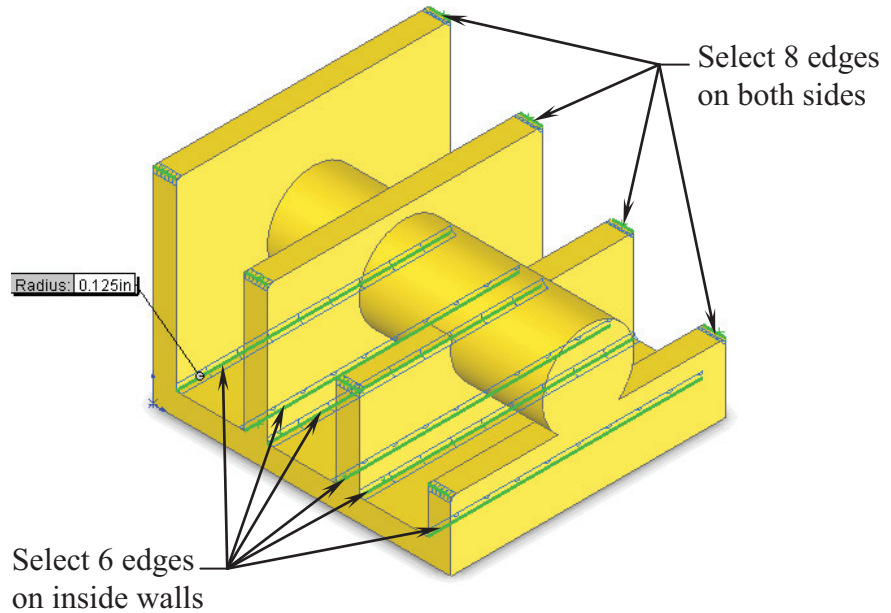
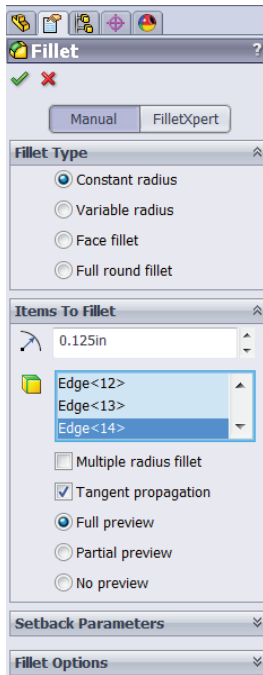
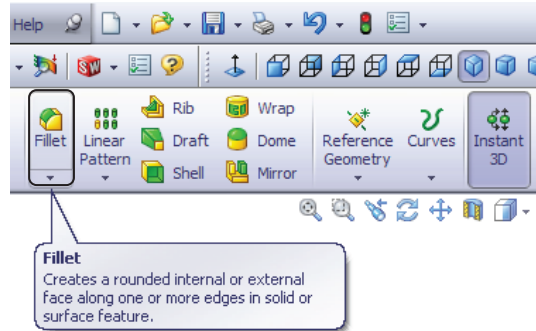


### 13. Adding the model fillets\*:

- Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, select sets 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  or select **Insert / Features / Fillet/Round**.



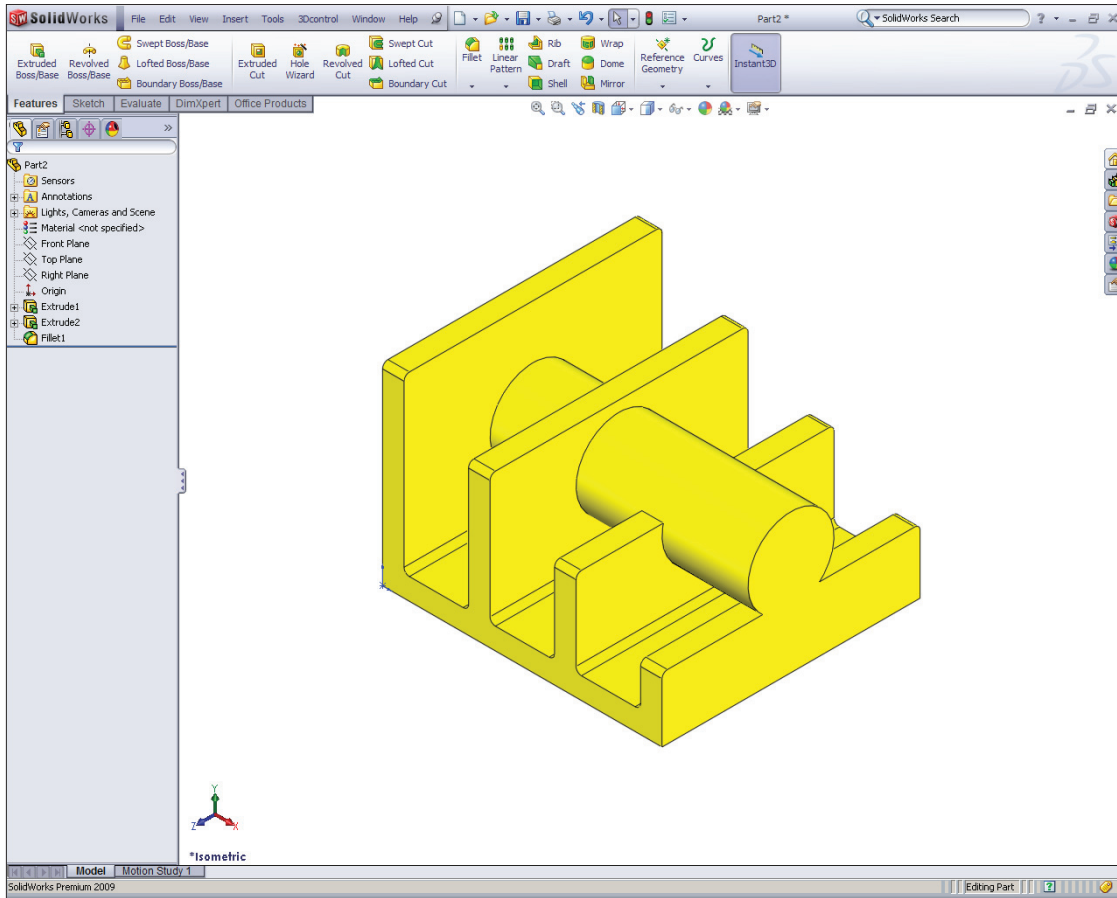
- Enter **.125 in.** for radius value.

- Select the edges as indicated to add the fillets.

- Click **OK** .

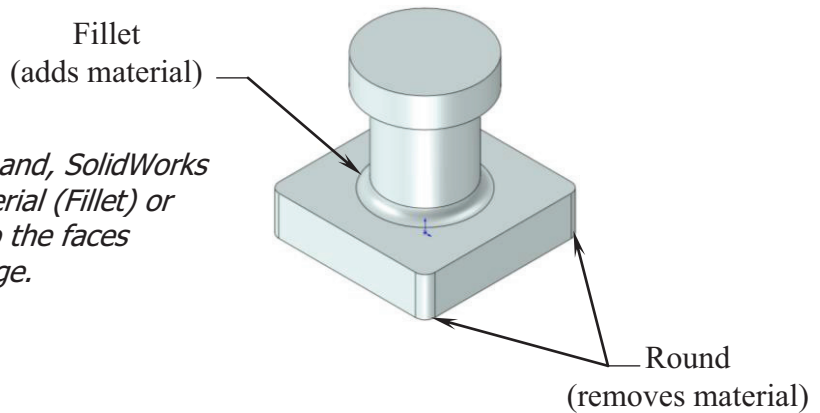


- The resulting fillets.



**\* 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.*



**14. Saving your work:**

- Select **File / Save As**.
- Enter **Extrude Options** for file name.
- Click **Save**.

# Questions for Review

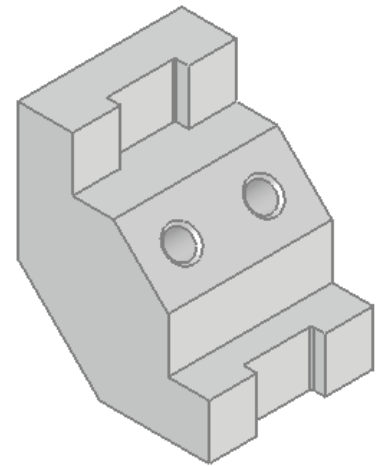
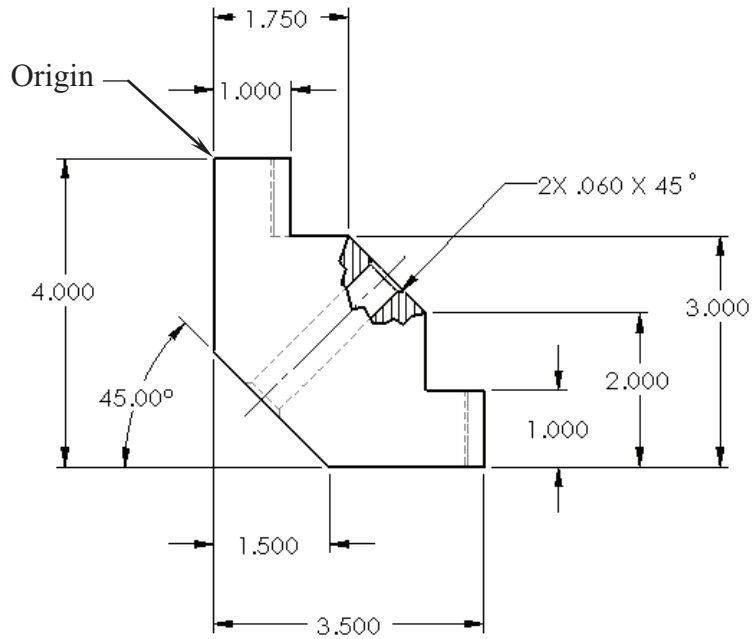
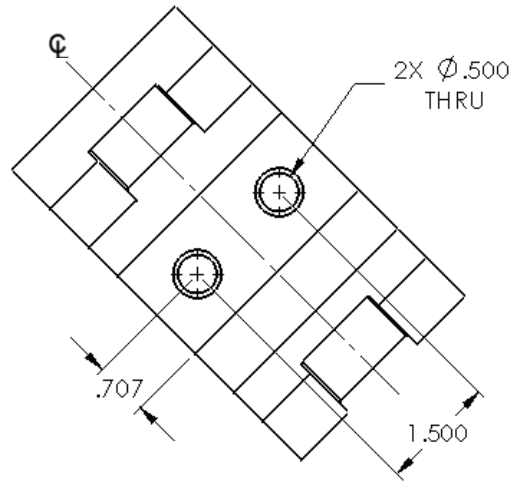
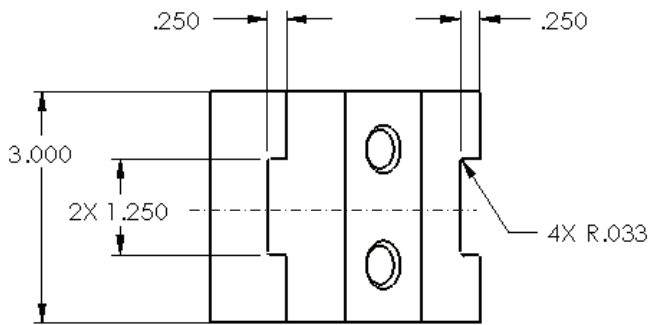
## Basic Solid Modeling

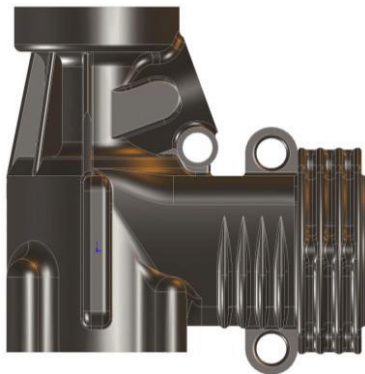
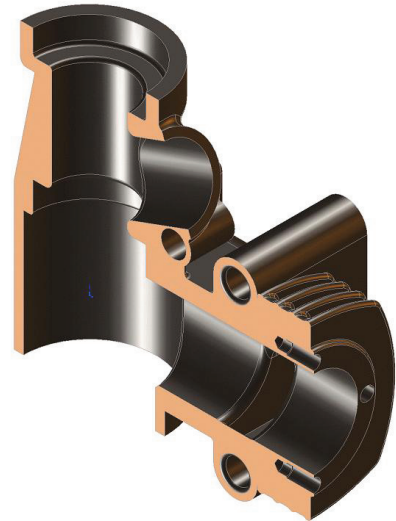
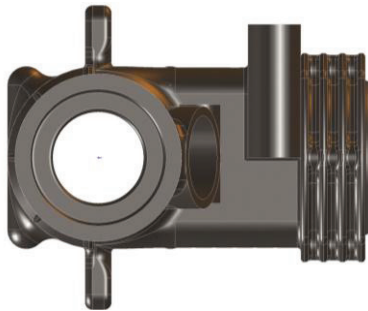
1. To open a new sketch, first you must select a plane from the FeatureManager tree.
  - a. True
  - b. False
2. Geometric relations can be used only in the assembly environments.
  - a. True
  - b. False
3. The current status of a sketch is displayed in the lower right area of the screen as:  
Under defined, Fully defined, or Over defined.
  - a. True
  - b. False
4. Once a feature is extruded, its extrude 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. 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 have to choose a surface as an 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

**Exercise: Extrude Boss & Extrude Cut.**

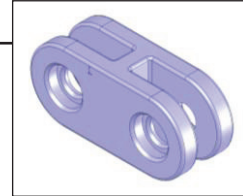
1. Dimensions are in inches, 3 decimal places.
2. Use Mid-Plane end condition for the Base feature.
3. The part is symmetrical about the Front plane.
4. Save your work as: **Exe1-Extrudes.**





# CHAPTER 4

## Basic Solid Modeling



### Basic Solid Modeling Extrude & Revolve

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

\* Perform basic modeling techniques.

\* Sketch on planar surfaces  .

\* Add dimensions  .

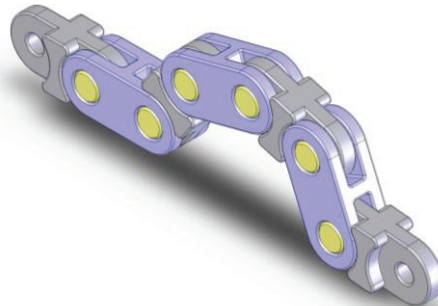
\* Add geometric relations or constraints  .

\* Base / Boss extrudes  .

\* Cut extrudes  .

\* Create revolved features  .

\* Create Fillets  and Chamfers  .



- The components will also be used later in an assembly to demonstrate how they can be copied several times and constrained, to form a new assembly, check for interferences and dynamic motion of an assembly.

- Be sure to review self-test questionnaires at the end of the lesson, prior to going to the next chapter. They will help to see if you have gained enough knowledge that are required in the next chapters.