SolidWorks 2010
Part I - Basic Tools

Introductory Level Tutorials
Parts, Assemblies and Drawings

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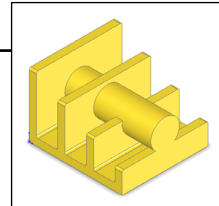
CHAPTER 3

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
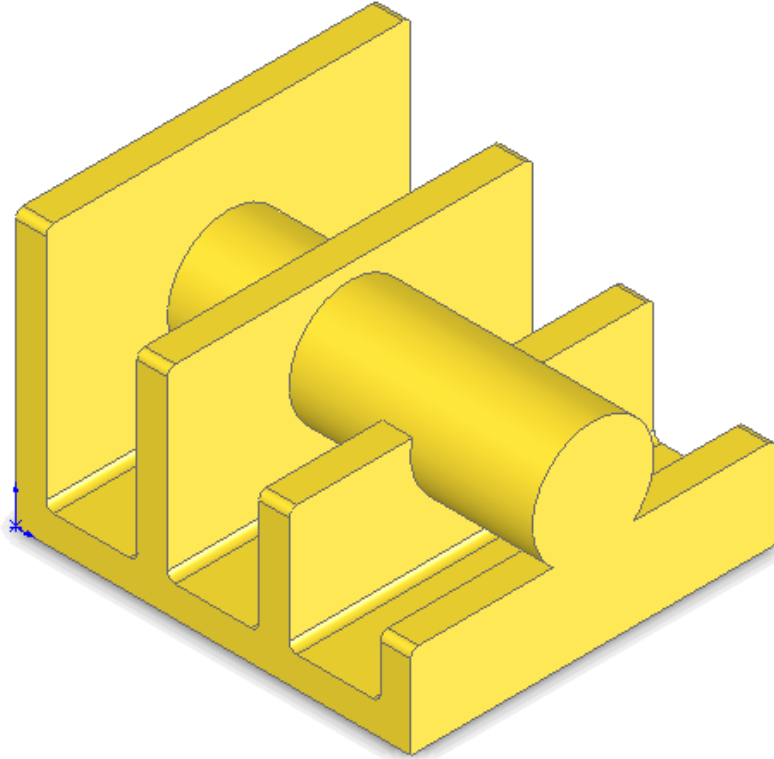
Dimensioning Standards: **ANSI**	
Units: **INCHES** – 3 Decimals	

Tools Needed:

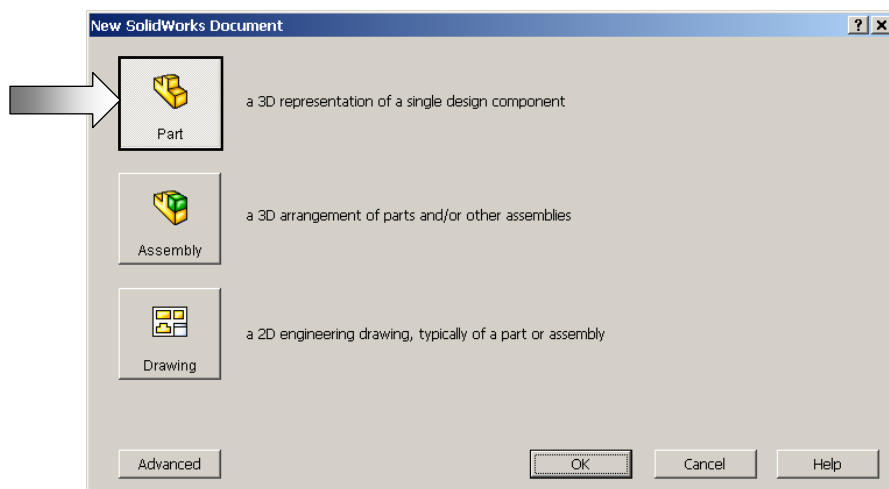|		
Insert Sketch	Line	Circle
Add Geometric Relations	Dimension	Sketch Fillet
	Boss / Base Extrude	

1. Starting a new Part:

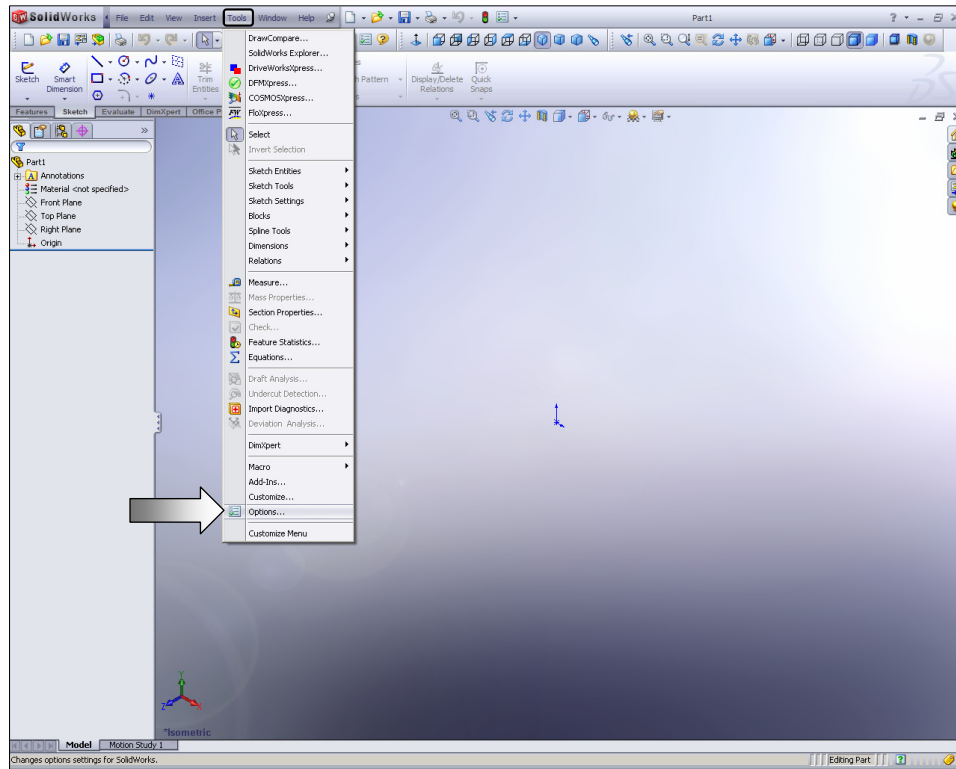- From the **File** menu, select **New / Part**.

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

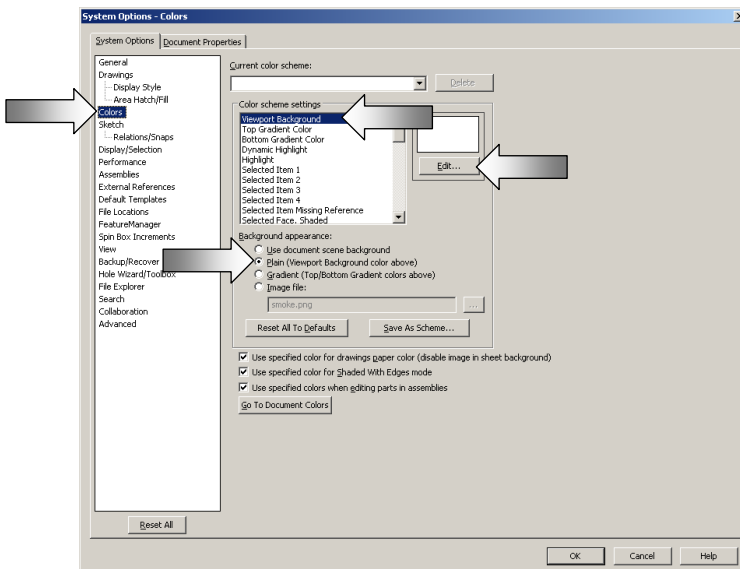- Click **OK** [OK] ; a new part template is opened.

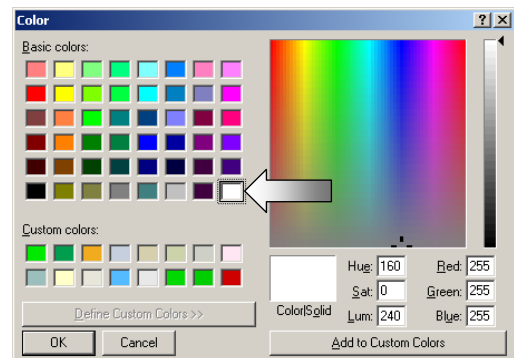- The next step is to set up your system options.

2. Changing the Background color:

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

- Under **Background Appearance**, select **Plain** (Viewport Background Color), and choose the <u>White</u> color if not yet selected (arrows).
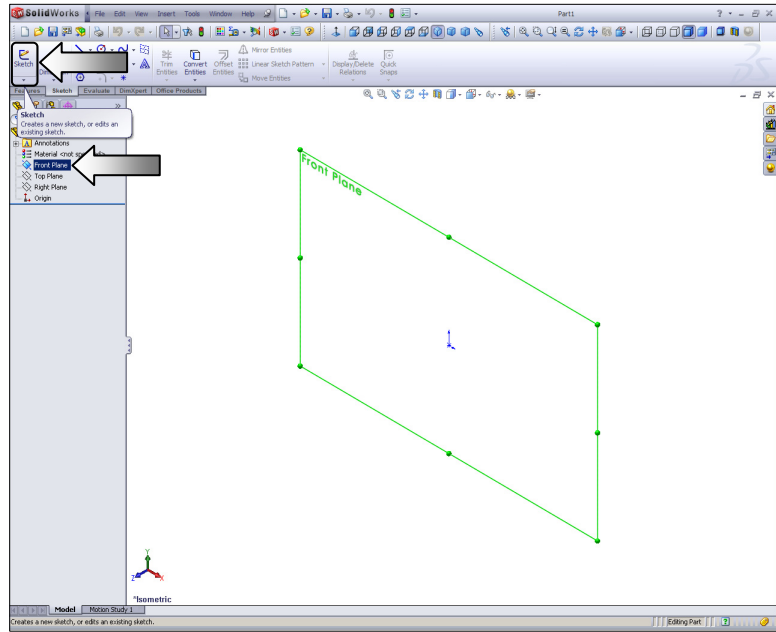
- This will change your background **Color to white.**
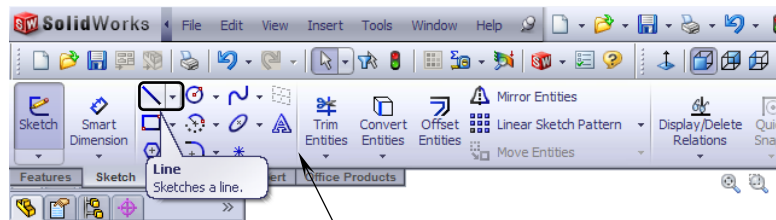
3. Starting a new Sketch:

- Select the Front plane from the FeatureManager tree and then click the Pencil

icon ![pencil icon] to open a new sketch.

- From the Command-Manager toolbar, select the

Line ![line icon] command.

Command Manager Toolbar
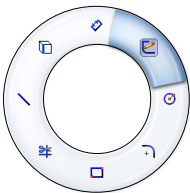
Mouse Gesture

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) will be added automatically to the sketch entity.

Front Plane

Auto-Relation feedback symbol

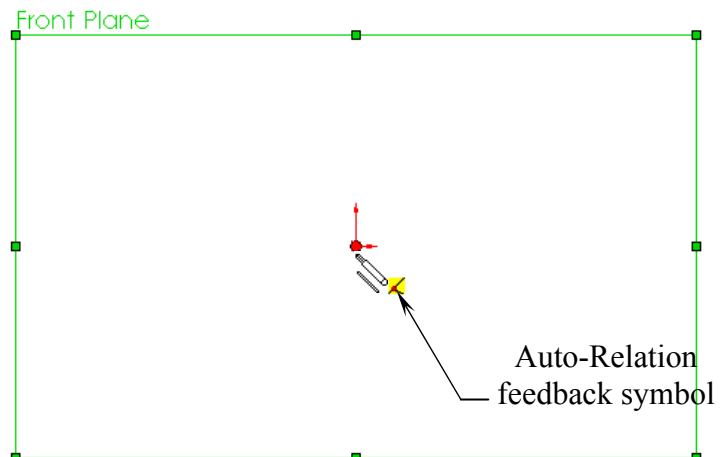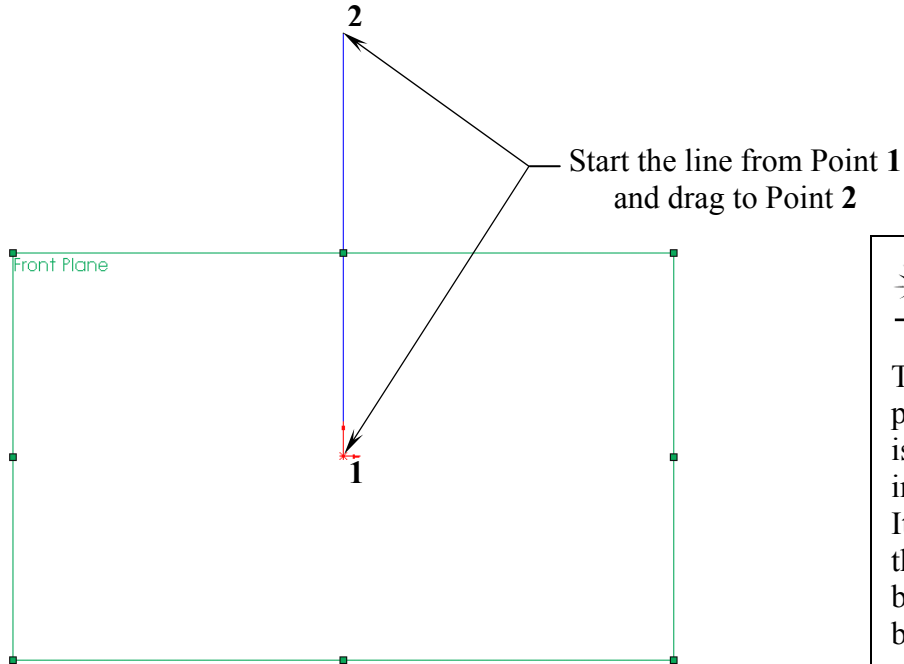## 4. Sketching the first profile:

- 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.

2

Start the line from Point **1**
and drag to Point **2**

Front Plane

1

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.
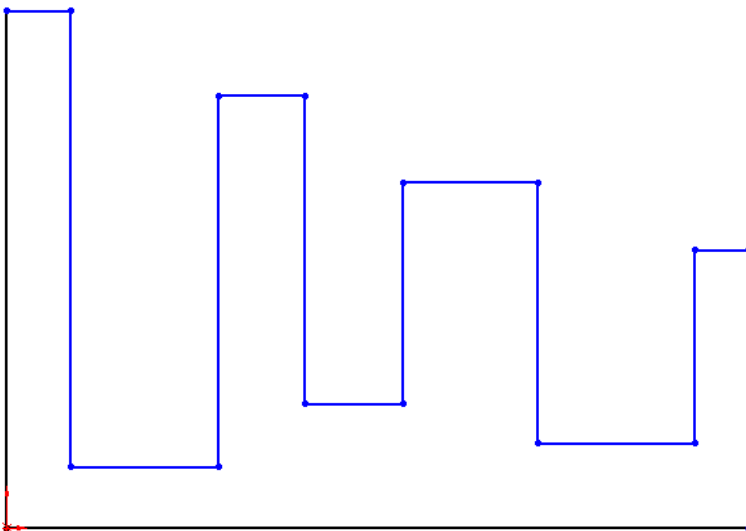
System Feedback

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

5. Adding Geometric Relations*:
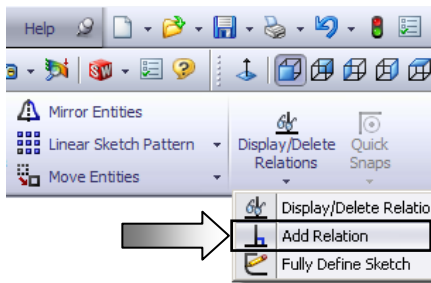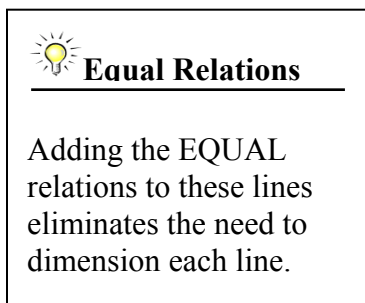
- Click under Display/Delete Relations - OR - select **Tools / Relations / Add**.

- 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.

Select the top 4 lines and click Equal relation

The top 4 lines are now Equal in size.
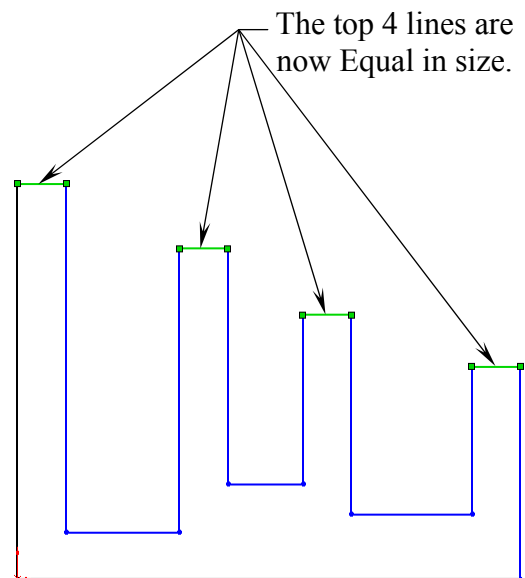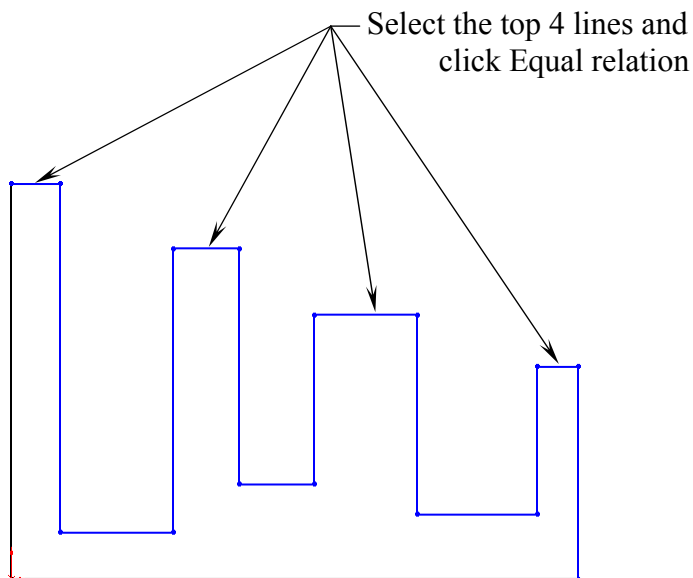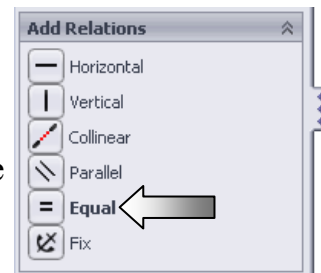
6. Adding a Collinear relation**:

- Select the Add Relation [icon] command again.

- Select the 3 lines as shown below.

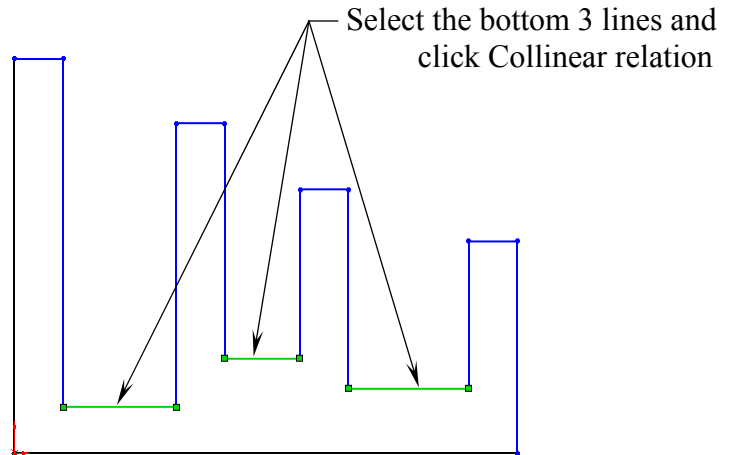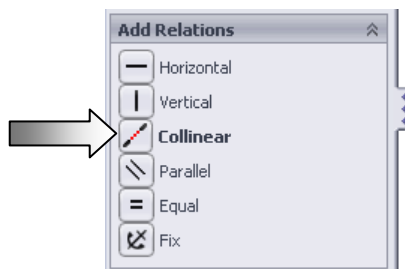- Click **COLLINEAR** from the Add Geometric Relations dialog box.

- Click **OK** [icon].

Select the bottom 3 lines and click Collinear relation

Add Relations
- Horizontal
- Vertical
- **Collinear**
- Parallel
- Equal
- Fix

The bottom 3 lines are moved to the same level.

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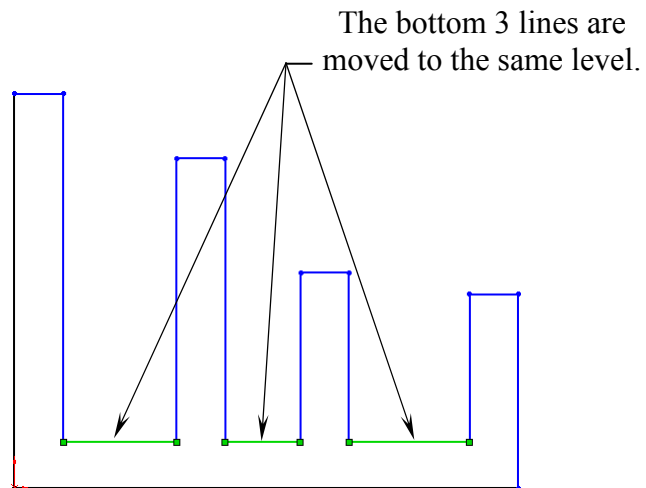
** The Collinear relations can be used to constrain the geometry as follows:

- Collinear a line to another line(s).

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

7. Adding the horizontal dimensions:

- Select [icon] from the Sketch toolbar - OR - select **Insert / Dimension**, and add the dimensions as shown below (follow the 3 steps).
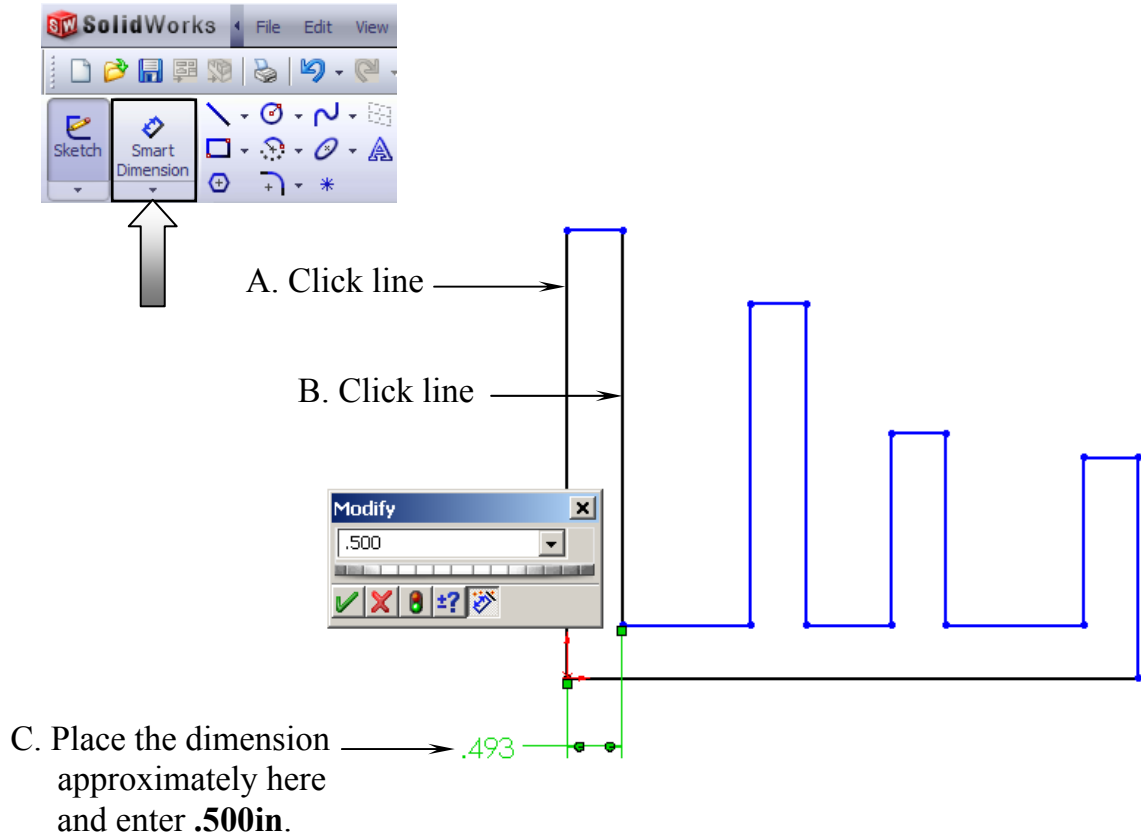
A. Click line

B. Click line

Modify
.500

C. Place the dimension approximately here and enter **.500in**.

- 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.

.500

2.000

4.000

6.500
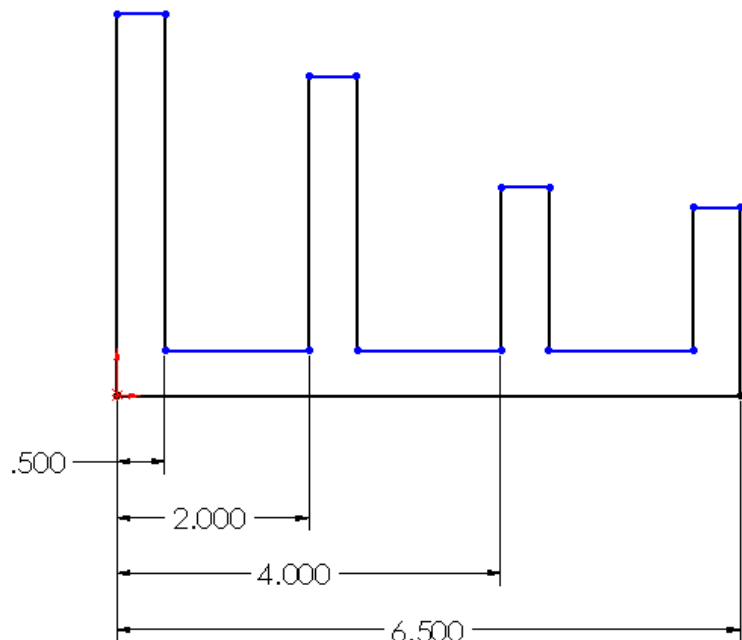
8. Adding the Vertical dimensions:

- 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 **.
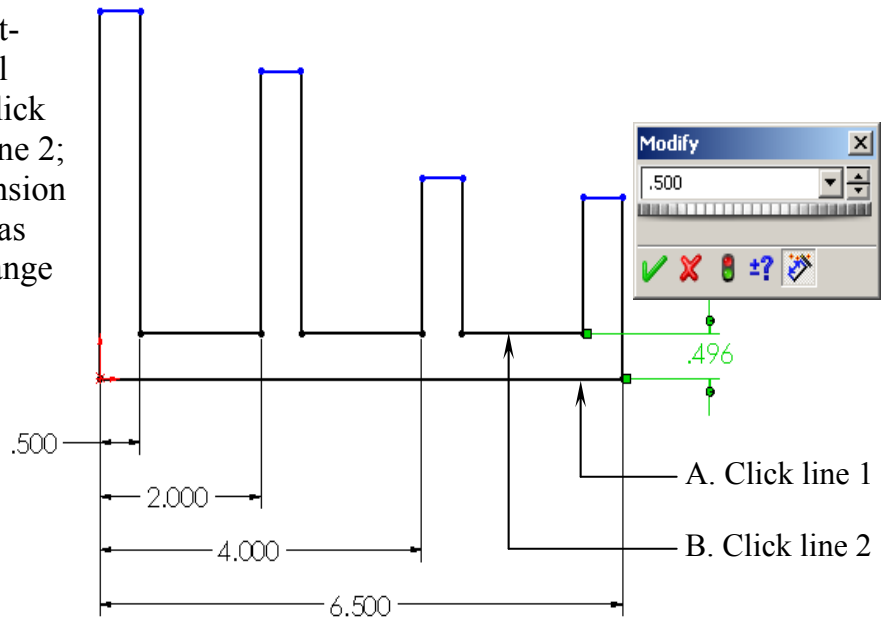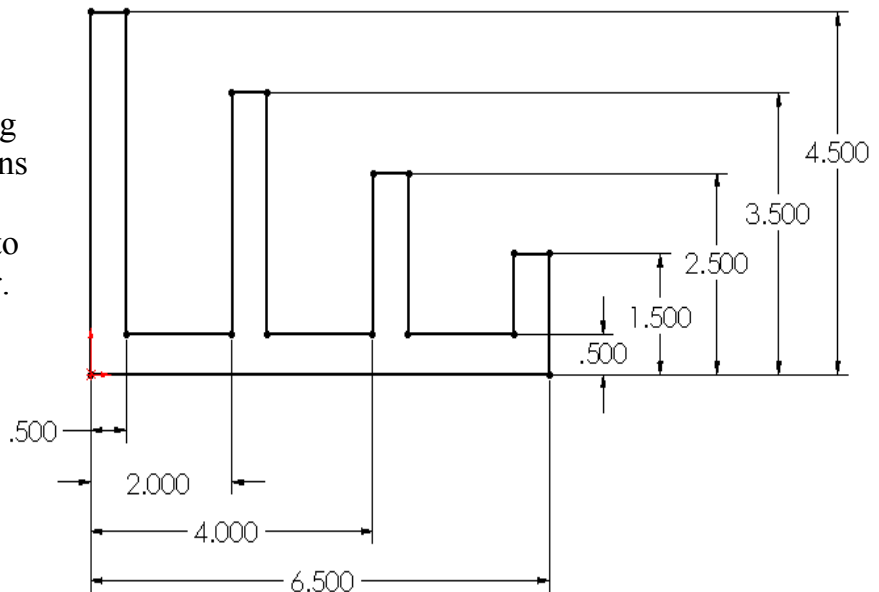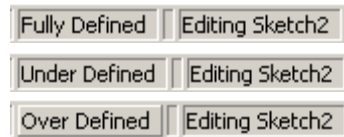500 in**.

Modify
.500

.496

.500

2.000

4.000

6.500

A. Click line 1

B. Click line 2

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

4.500

3.500

2.500

1.500

.500

.500

2.000

4.000

6.500

The Status of a Sketch:

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

Fully Defined	=	**Black**	Fully Defined	Editing Sketch2
Under Defined	=	**Blue**	Under Defined	Editing Sketch2
Over Defined	=	**Red**	Over Defined	Editing Sketch2

Sketch Relation

4.500

3.500

2.500

1.500

.500

R.125
TYP

.500

2.000

4.000

6.500
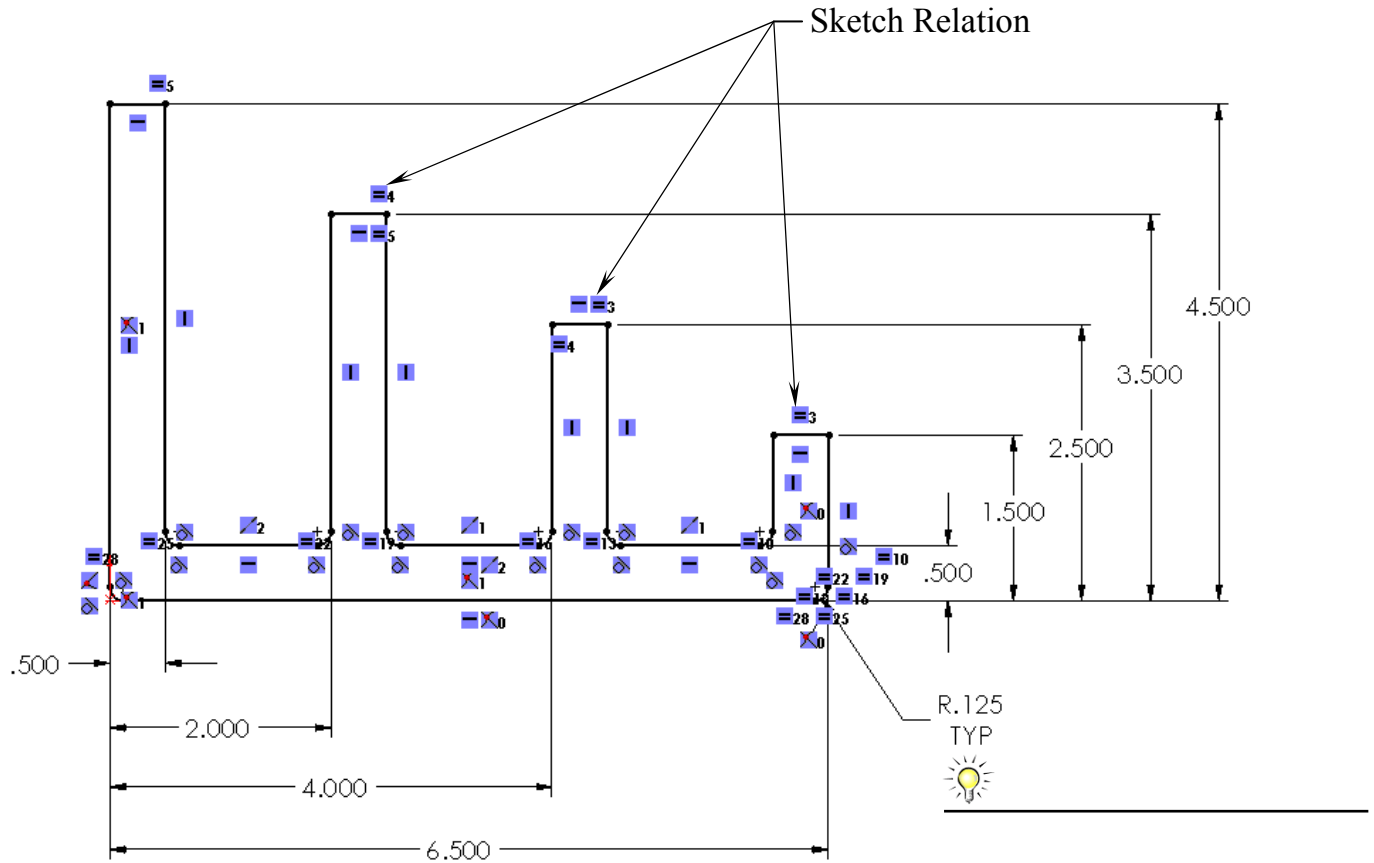
9. Hiding the Sketch Relation Symbols:

- 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.
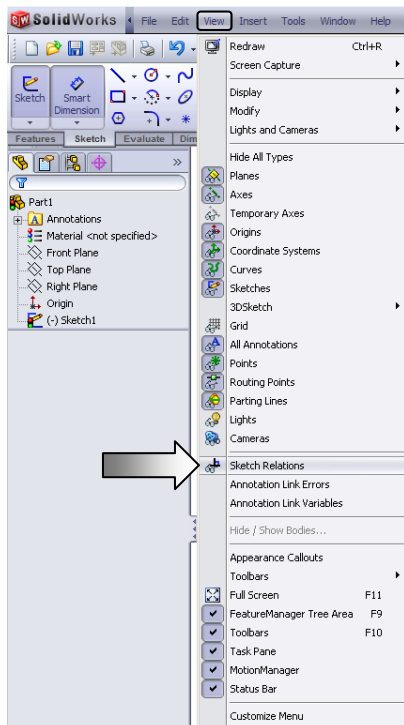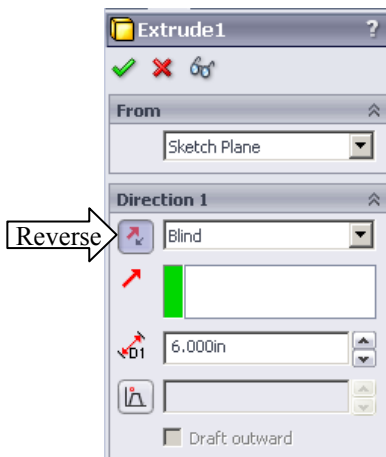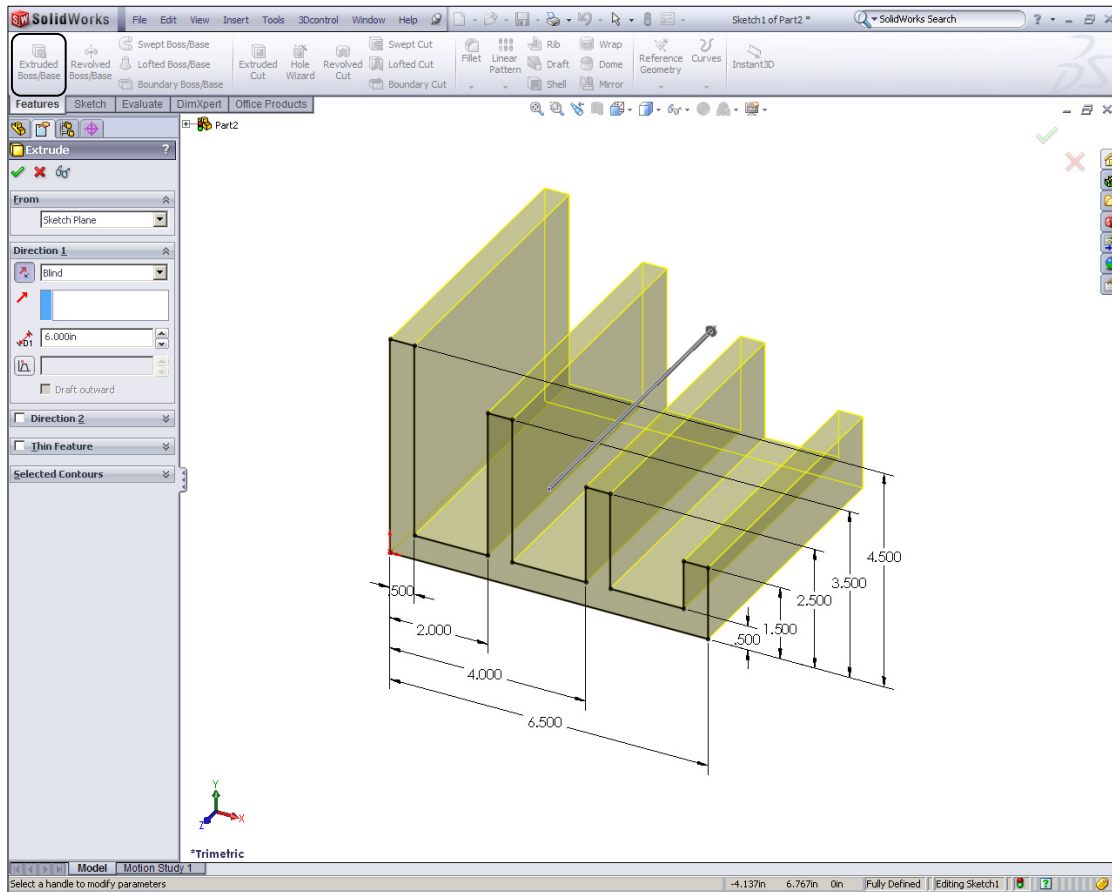

Sketch Relation Symbols at a Glance

|			
▬	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 [icon] from the Features toolbar - OR- select **Insert / Boss Base / Extrude**.
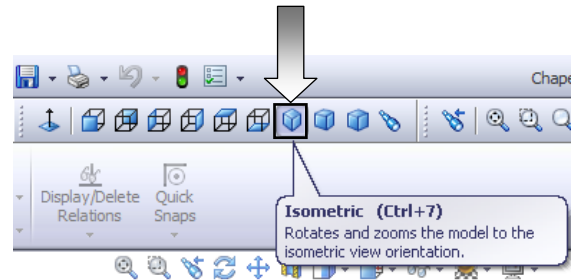

- Set Direction 1 to **Blind**.

- Set Depth to **6.00 in**.

- Click the **Reverse** button.

- Click **OK** ✅.
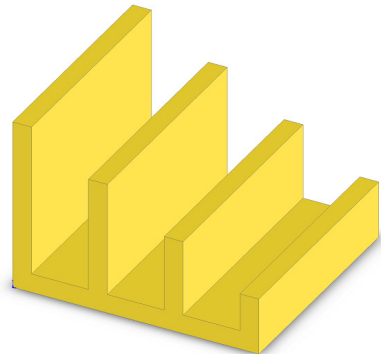
11. Sketching on a Planar Face:

- Select the face as indicated.

- Click ![pencil icon] or select **Insert/Sketch**.

- Click ![circle icon] 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**).

Isometric (Ctrl+7)
Rotates and zooms the model to the isometric view orientation.
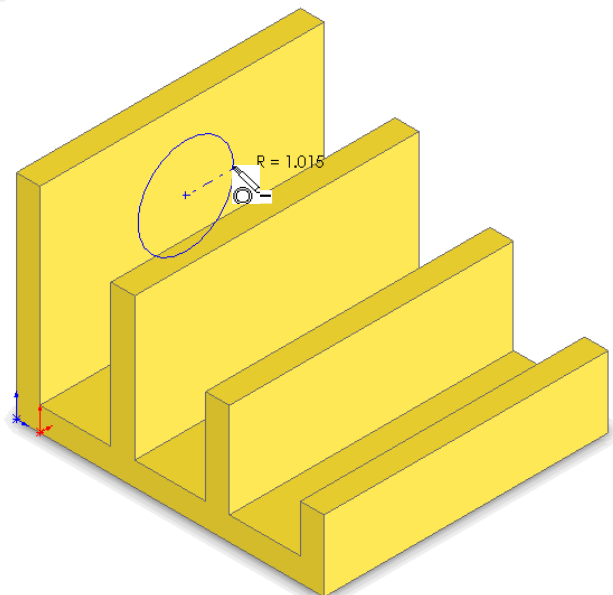
Select the Sketch Face
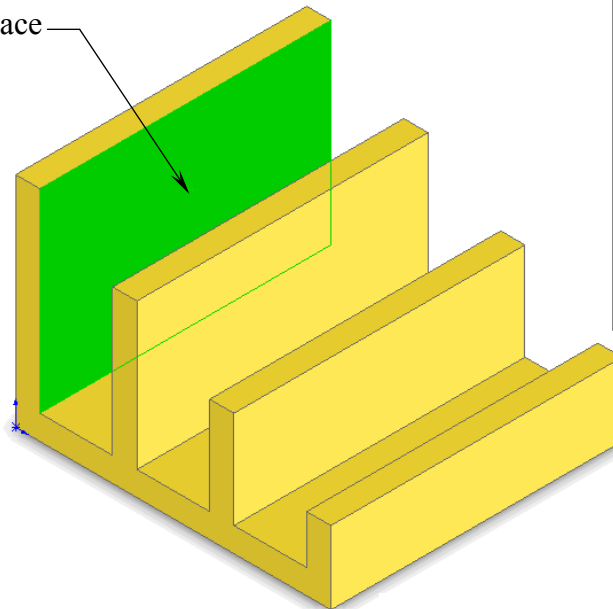
> ### 💡 Planar Surfaces
>
> - A planar surface of the model can also be used as a Sketch Plane.
>
> - The Sketch will then be extruded normal to the selected surface.

- 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.

R = 1.015

- Select the **Smart Dimension**

command [icon] 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).

Modify
2.000

Ø2.008
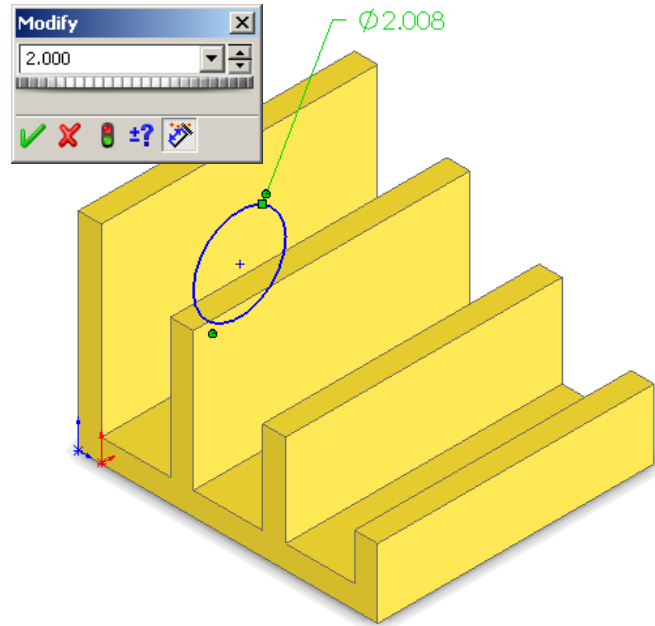
- Continue adding the
location dimensions
as shown, to fully
define the sketch.
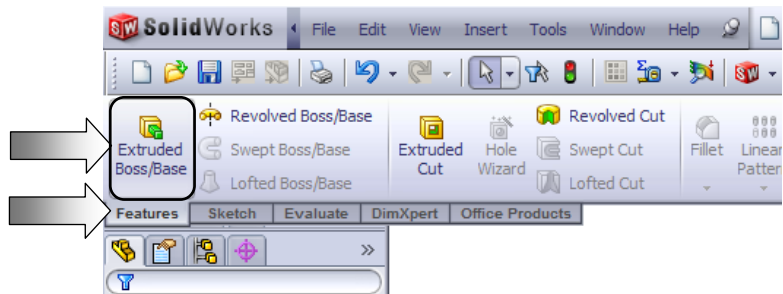
Ø2.000

3.000

2.250
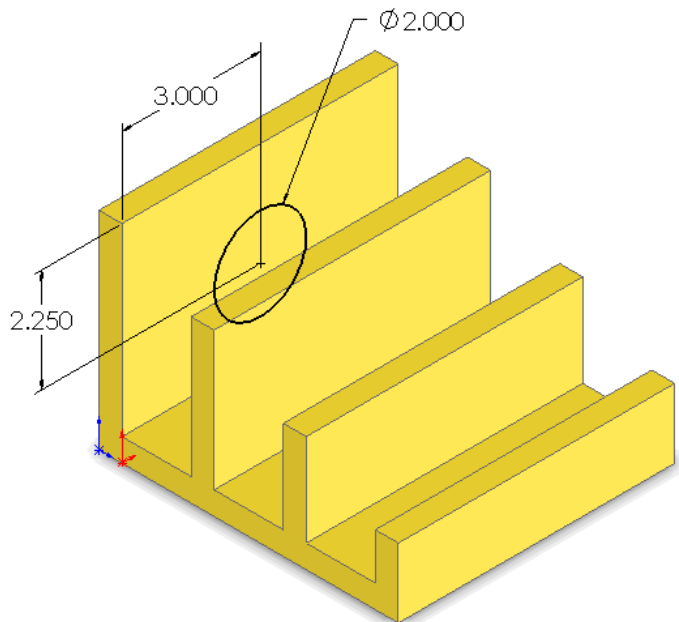
12. Extruding a Boss:

- Click [icon] 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.

SolidWorks   File   Edit   View   Insert   Tools   Window   Help

Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base

Extruded Cut   Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut

Fillet   Linear Patter

Features   Sketch   Evaluate   DimXpert   Office Products

(A) **Using the Blind option:**

- When extruding with the Blind option, the following conditions are required:
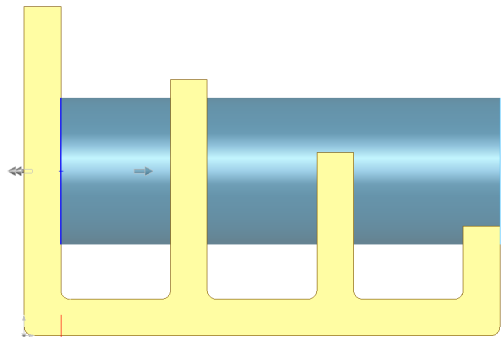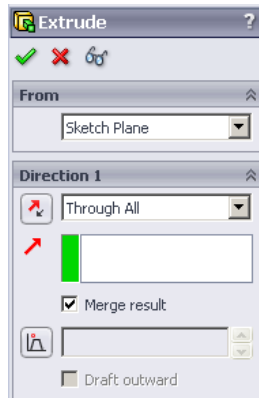
   * Direction
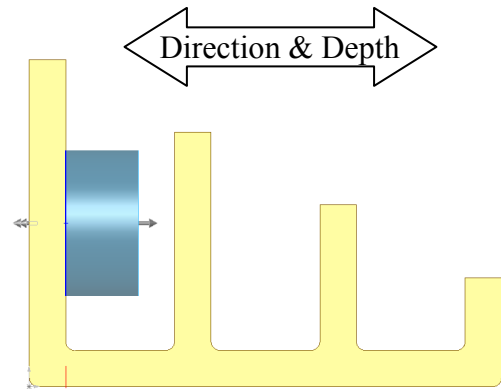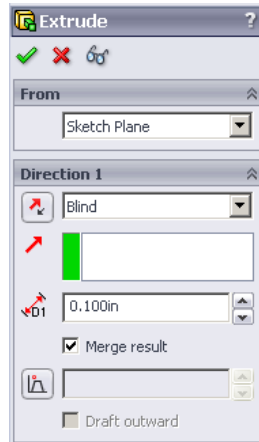
   * Depth dimension

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

Direction & Depth

Blind
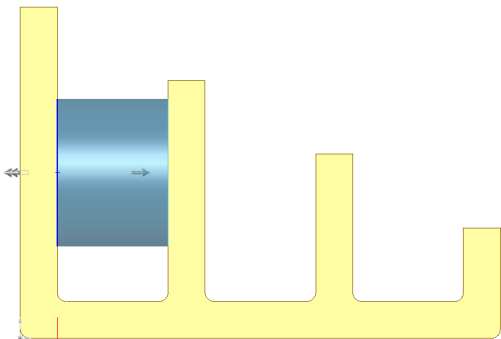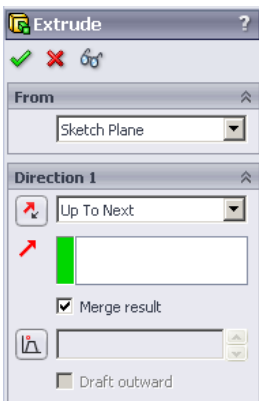Condition

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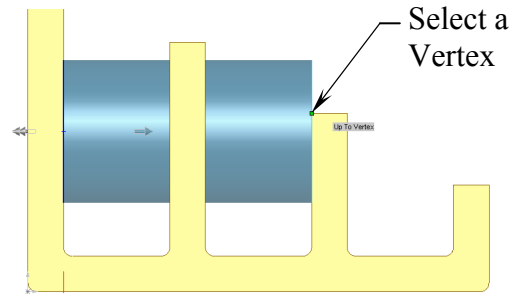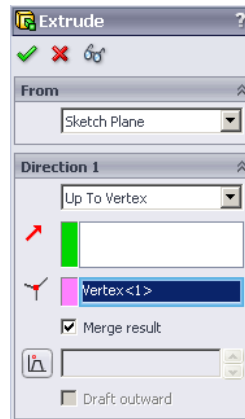
(C) **Using the Up To Next option:**

- With the Up To Next option, 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
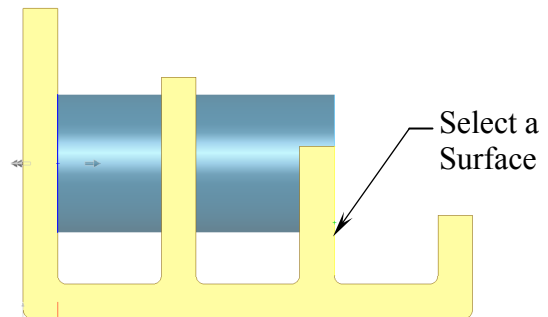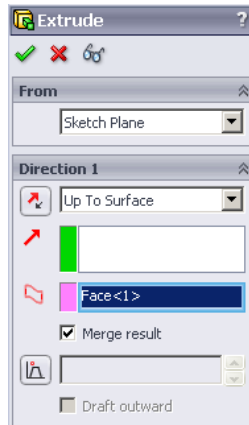Condition

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.

Select a Vertex

Up To Vertex
Condition

E **Using the Up To Surface option:**
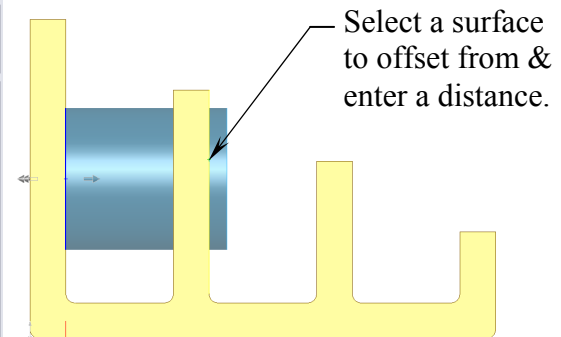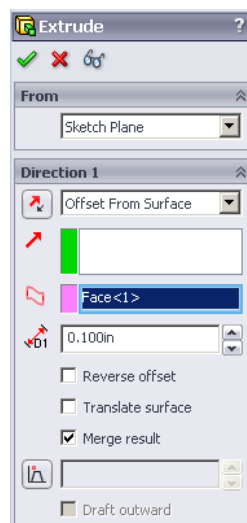
- This option extrudes the sketch from its plane to a selected surface or a face to define its depth.

Select a Surface

**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.

Select a surface to offset from & enter a distance.

Offset From Surface
Condition

(G) Using the Up To Body option:

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

Select a Solid Body
to extrude to.

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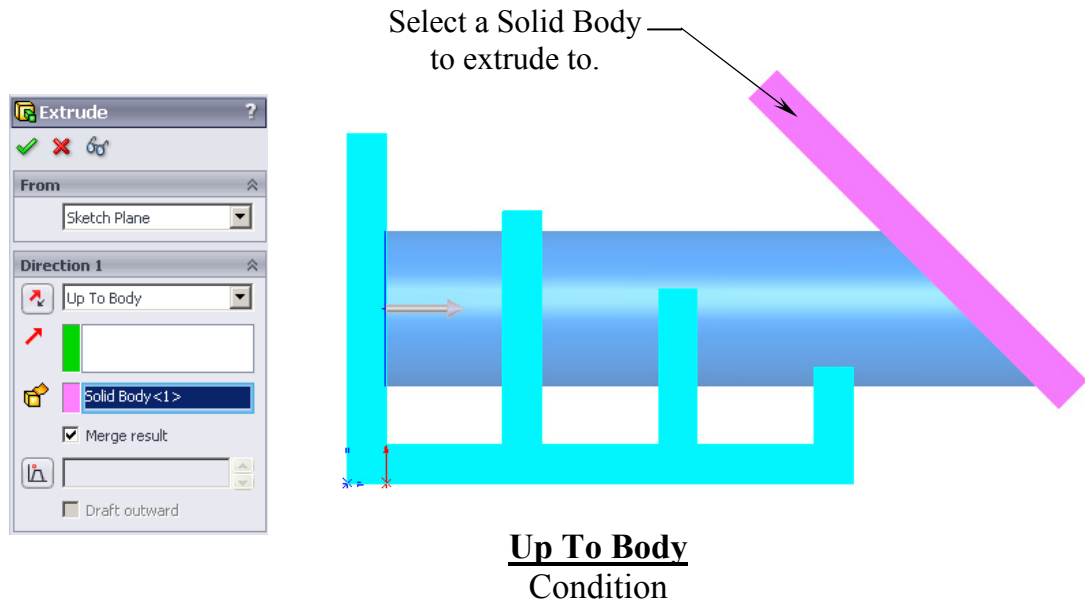
- The Up To Body option can also be used in assemblies or multi-body parts.

(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.

Mid Plane
Condition
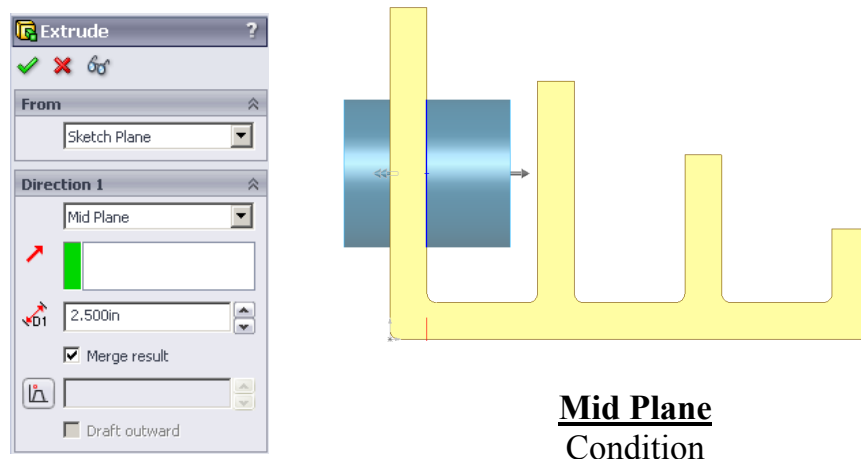
- After you are finished with exploring all the extrude options, change the final condition to: **Through All**
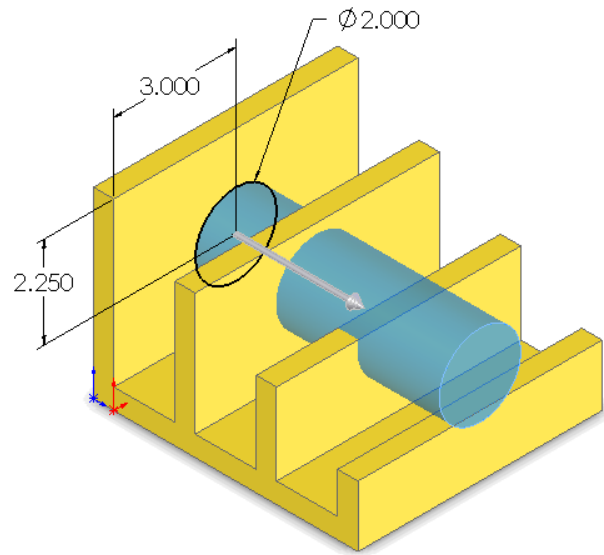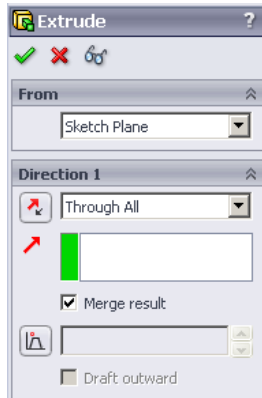
- Click O**K** ✅.

- 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.
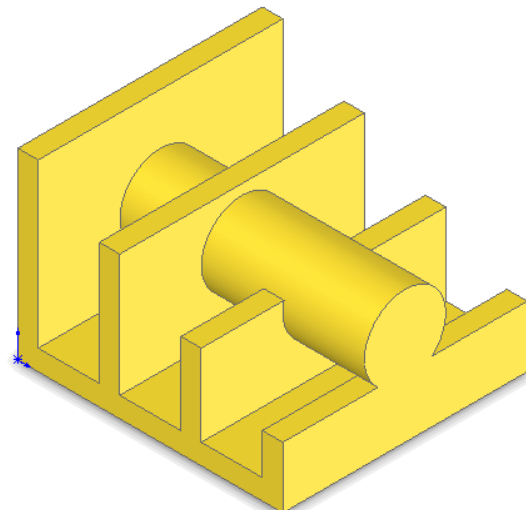
13. Adding the model fillets*:

- 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.

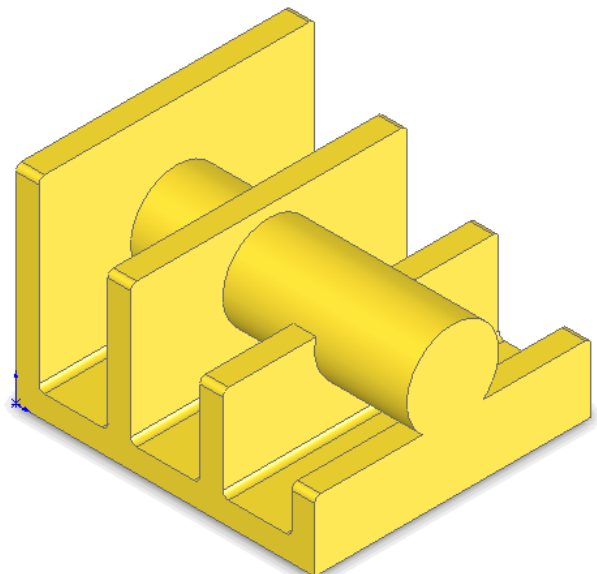- The **radius** value stays in effect until you change it. Therefore, you can select any number of edges or faces in the same operation.

- Click [icon] or select **Insert / Features / Fillet/Round**.

Select 8 edges on both sides

Radius: 0.125in

Select 6 edges on inside walls
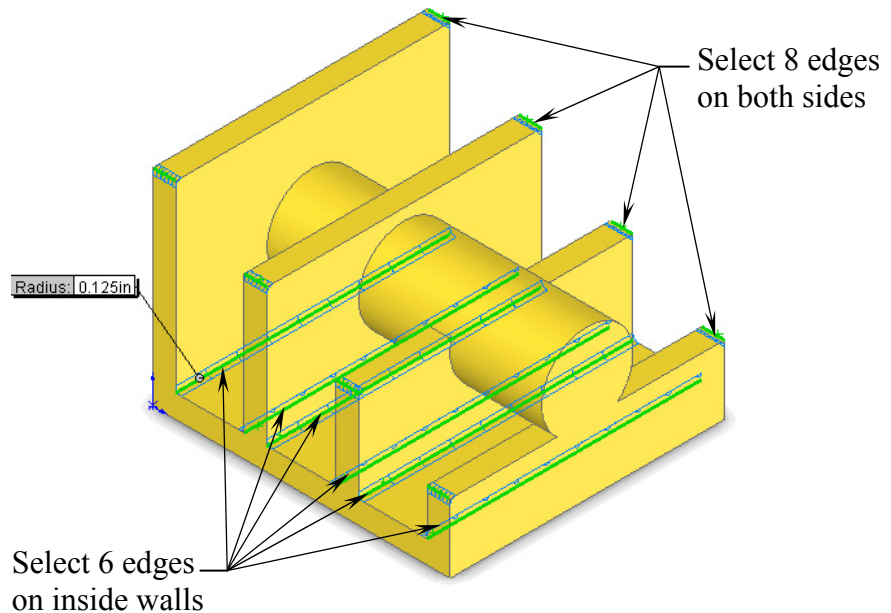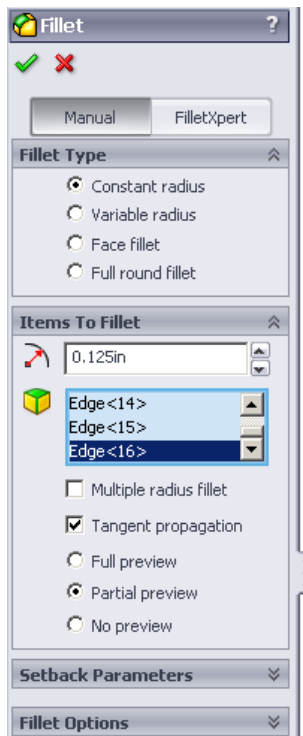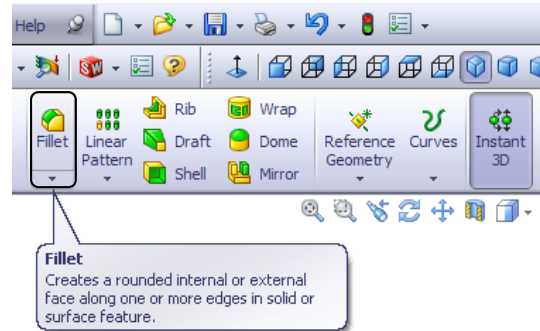
- Enter **.125 in**. for Radius Value.

- Select the edges as indicated to add the fillets.

- Click **OK** ✅.

- The resulting fillets.
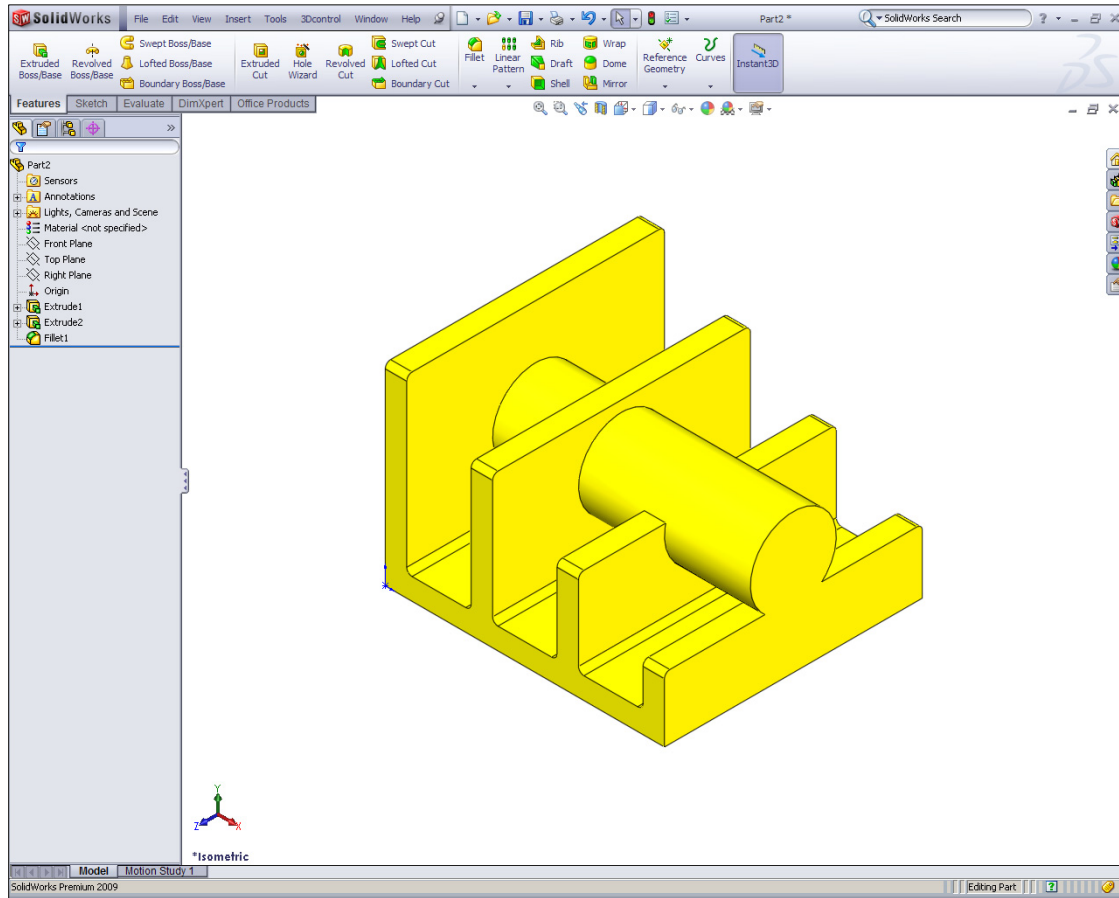
* Fillets and Rounds:

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

Fillet
(adds material)

Round
(removes material)
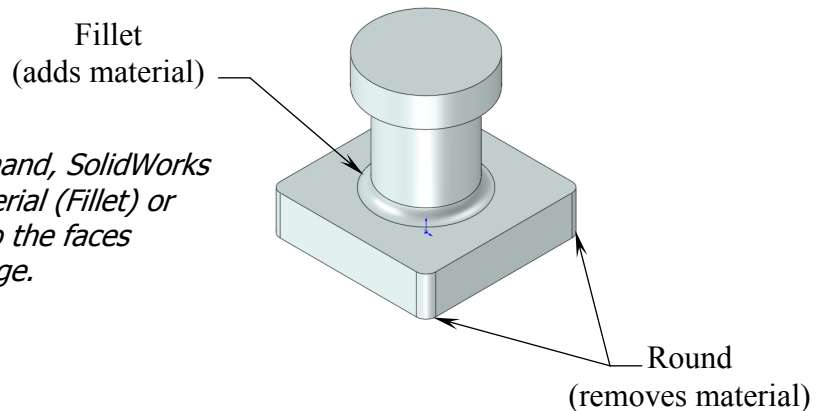
14. Saving your work:

- Select **File / Save As**.

- Enter **Extrude Options** for file name.

- Click **Save**.

Questions for Review

Basic Solid Modeling

1. To open a new sketch, you must select a plane from the FeatureManager tree first.
    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. Once 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

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

.250   .250

3.000

2X 1.250

4X R.033

2X Ø.500
THRU

(.707)

1.500

1.750

1.000

2X .060 X 45°

4.000

45.00°

3.000

2.000

1.000

1.500

3.500