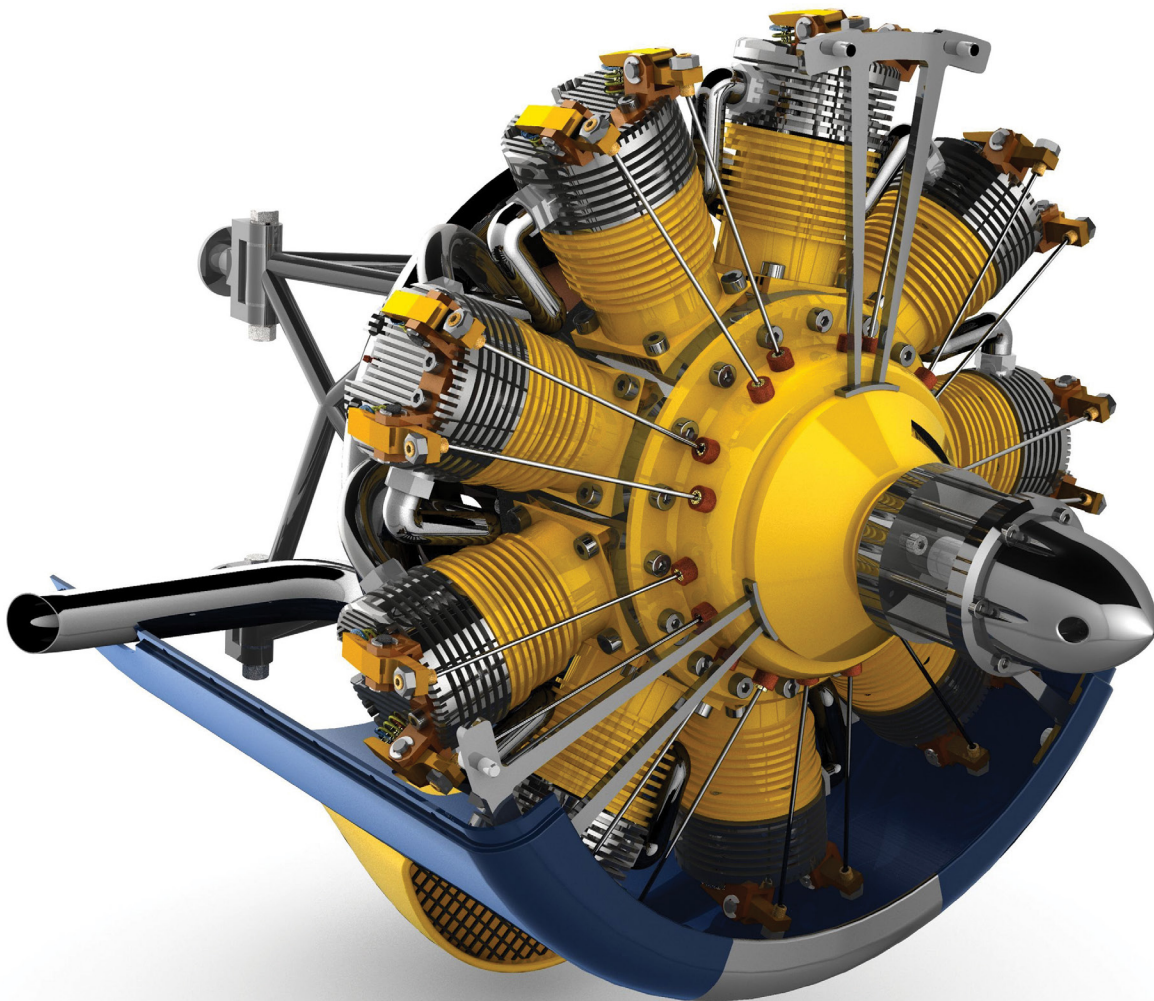# SOLIDWORKS® 2018
Basic Tools

Getting Started with Parts,
Assemblies and Drawings

Paul Tran CSWE, CSWI

Visit the following websites to learn more about this book:
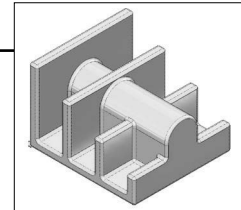
SDC **Publications** amazon.com Google books BARNES&NOBLE

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.

    * Activate Sketch pencil .

    * 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
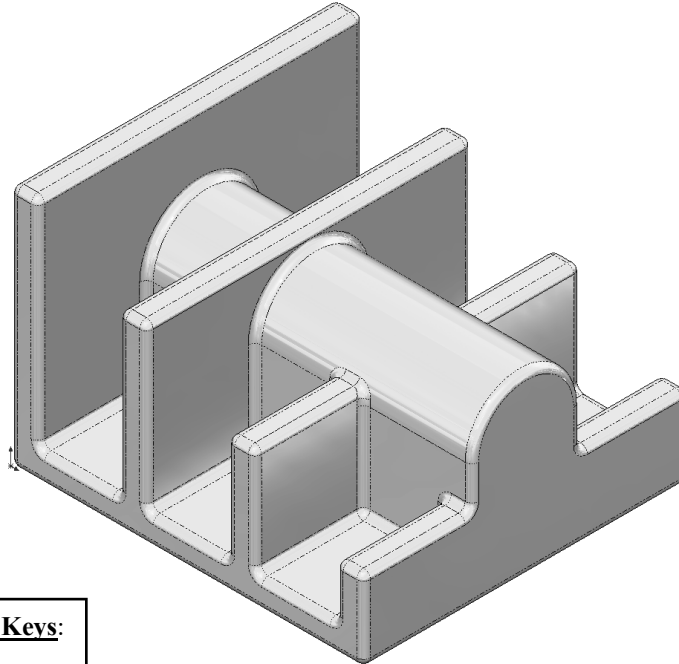
Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

Tools Needed:

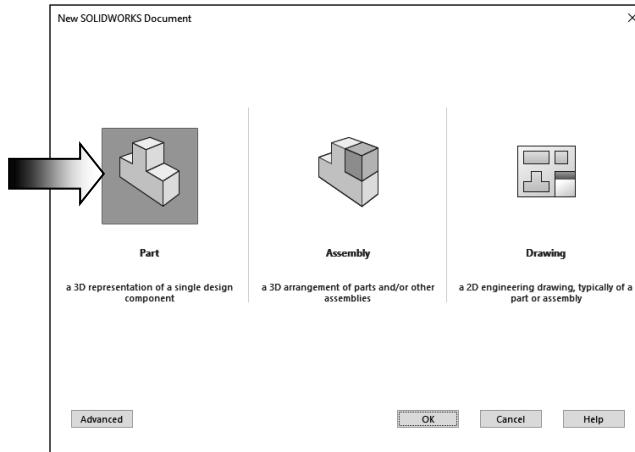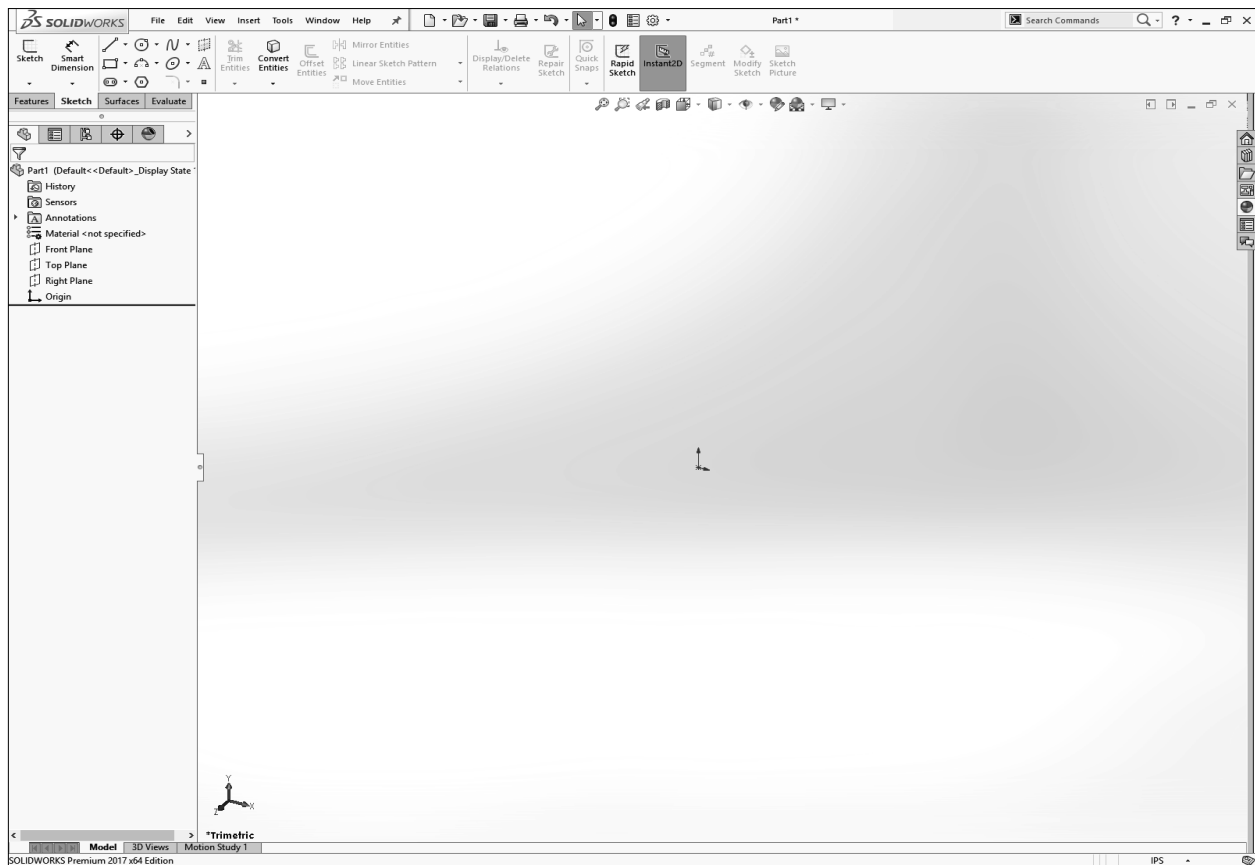|		
Insert Sketch	Line	Circle
Add Geometric Relations	Dimension	Sketch Fillet
Trim Entities	Boss / Base Extrude	

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 opened.

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 better see the colors of the sketch entities and their dimensions.
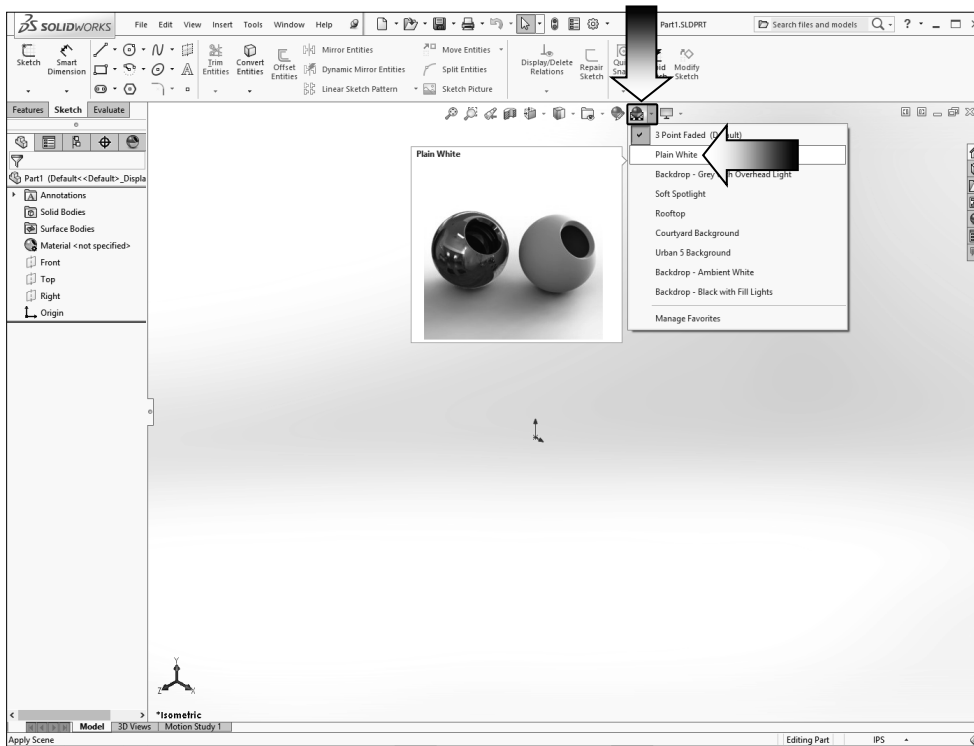
- To show the Origin, click the **View** dropdown menu and select **Origins**.

- The Blue Origin is the Zero position of the part and the Red Origin is the Zero position of a sketch.
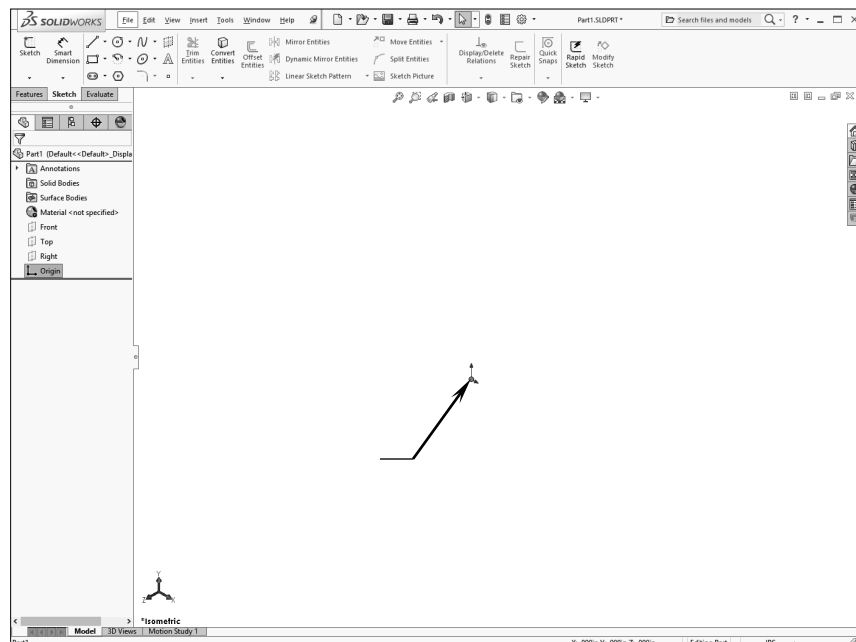
3. Starting a new Sketch:

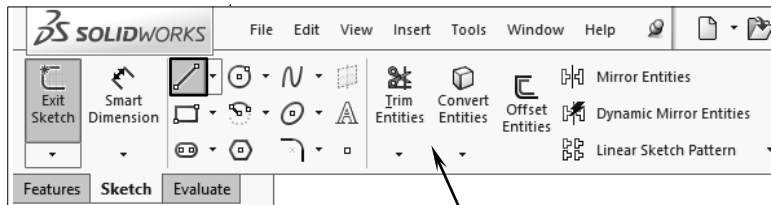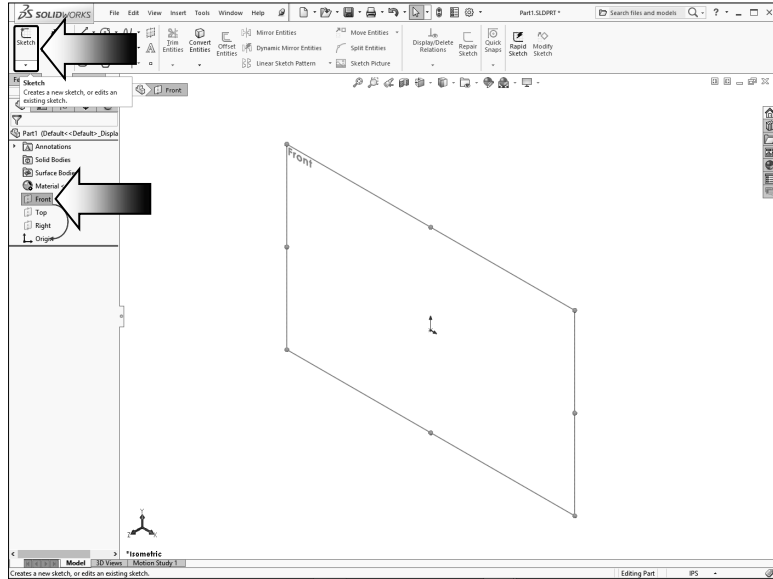- Select the <u>Front</u> plane from the Feature-Manager tree and click the **Pencil** icon [Sketch icon] to start a new sketch.

- A sketch is normally created first, relations and dimensions are added after, and then it gets extruded into a 3D feature.
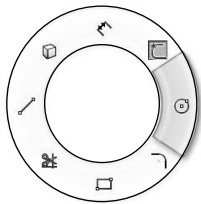
- From the Command-Manager toolbar, select the **Line** [Line icon] command.

CommandManager

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.)

- 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 1st endpoint of the line. This endpoint will be locked at the zero position.
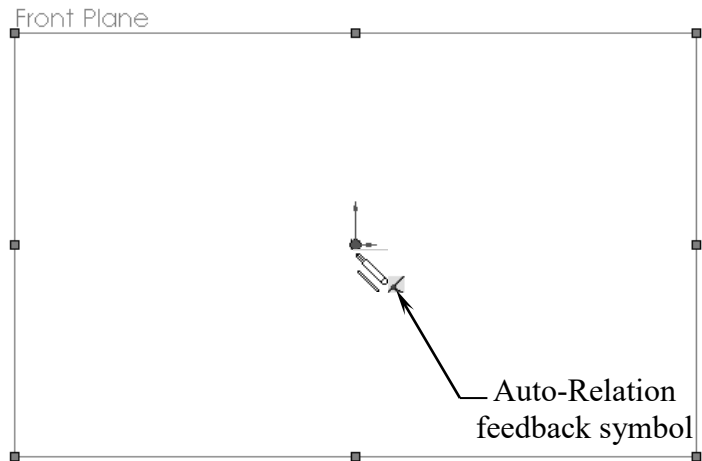
Auto-Relation feedback symbol

4. Using the Click + Hold + Drag technique:

- Click at 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.

2

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

Front Plane

1
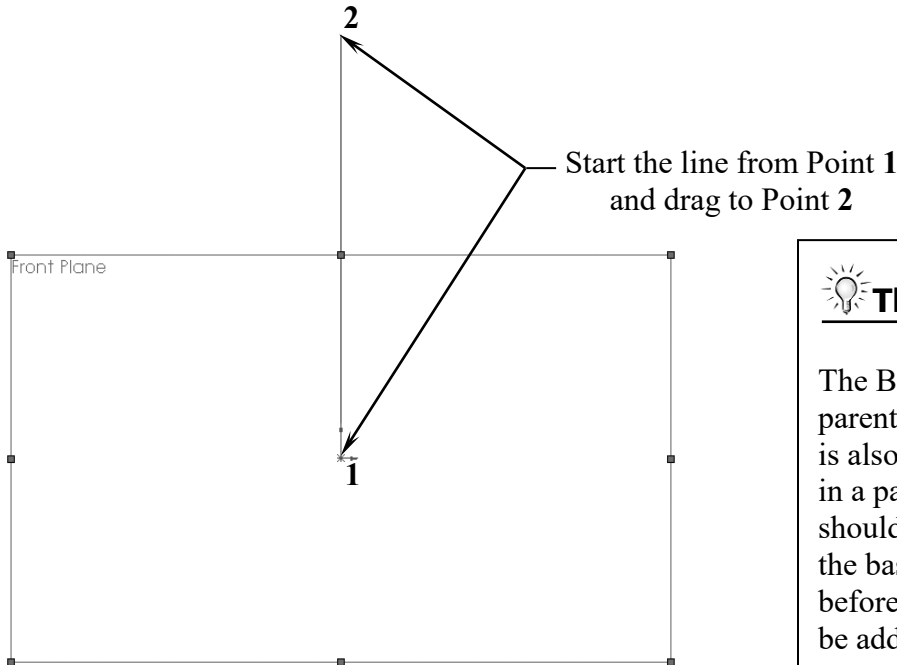
┌─────────────────────────────┐
│ 💡 **The Base Sketch**   │
│   │
│ The Base Sketch is the   │
│ parent sketch of a part and │
│ is also the very first sketch│
│ in a part document. It   │
│ should primarily describe   │
│ the basic shape of the part │
│ before other features can   │
│ be added.   │
└─────────────────────────────┘

- 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.
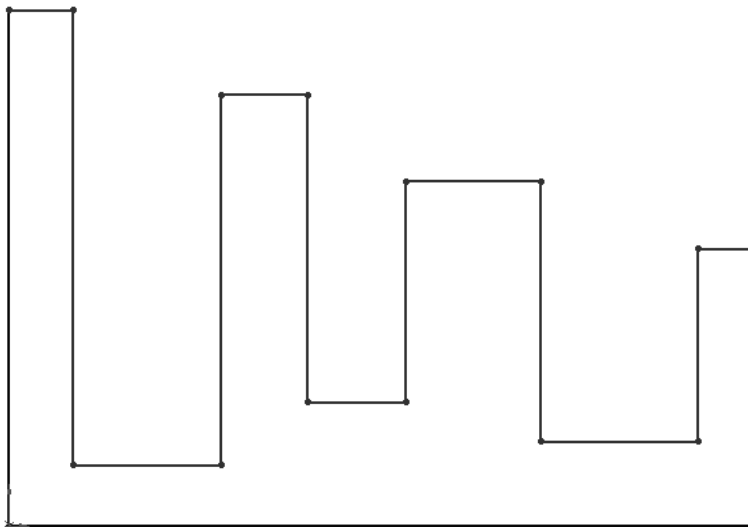
┌─────────────────────────────┐
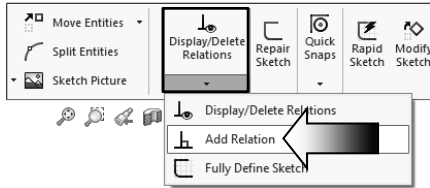│ 💡 **System Feedback**   │
│   │
│ While sketching the   │
│ lines, watch for the   │
│ System Feedback   │
│ Symbols such as ▬   │
│ for Horizontal, and   │
│ for ▮ Vertical Auto   │
│ Relations.   │
└─────────────────────────────┘

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 two selected lines equal.

* 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 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 or a model edge, a plane, an axis, or a curve, etc.
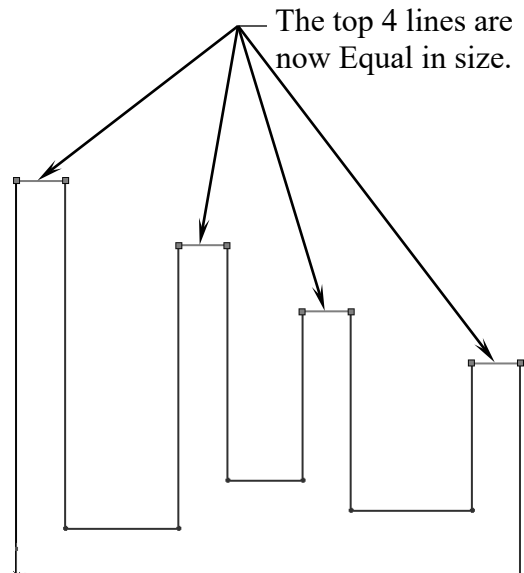
Equal Relations

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

Geometric relations can be created manually or automatically. The next few steps in this chapter will demonstrate how geometric relations are added manually.
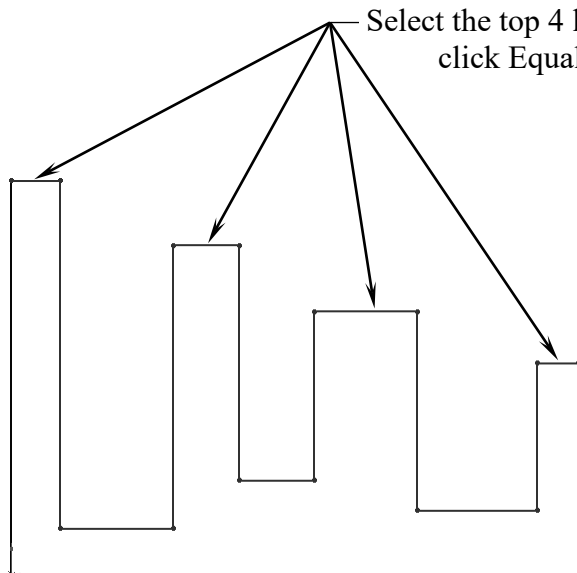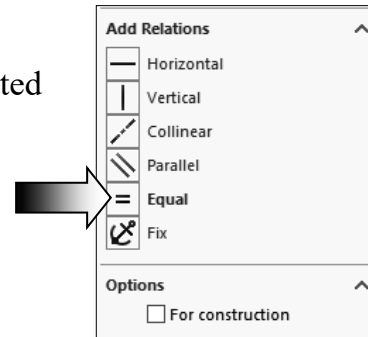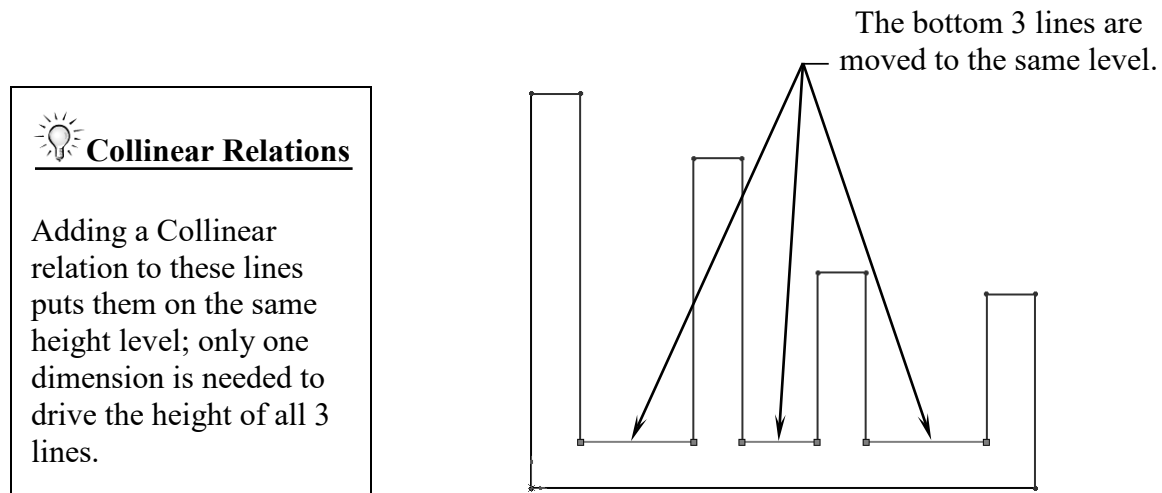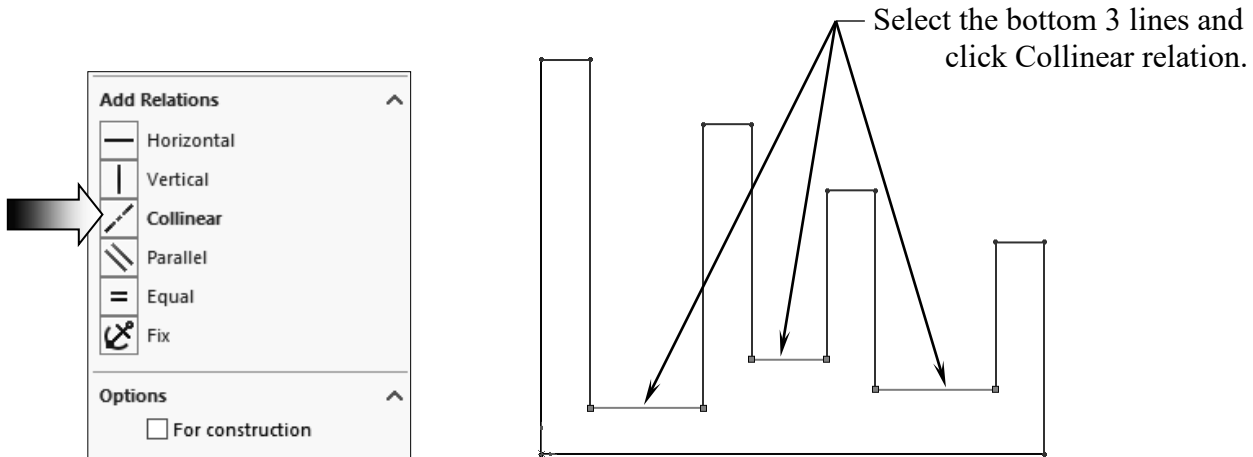
Select the top 4 lines and click Equal relation.

The top 4 lines are now Equal in size.

6. Adding a Collinear relation**:

- Select the **Add Relation** ⬛ command again.

- Select the 3 lines as shown below.

- Click **Collinear** from the Add Geometric Relations dialog box.
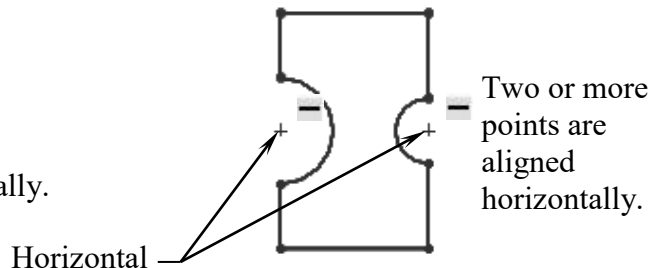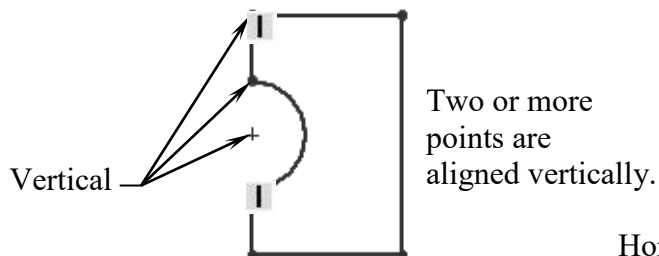
- Click **OK** ✅.

Select the bottom 3 lines and click Collinear relation.

Add Relations

— Horizontal

| Vertical

╱ Collinear

╲ Parallel

= Equal

⚓ Fix

Options

☐ For construction

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.

** 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 Relations Examples

Coincident

An endpoint is
Coincident with a line.

Midpoint

An endpoint is
Coincident with a
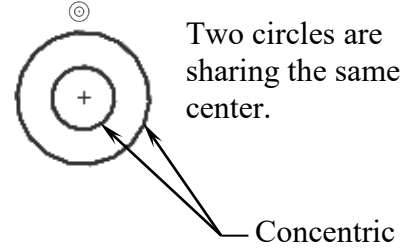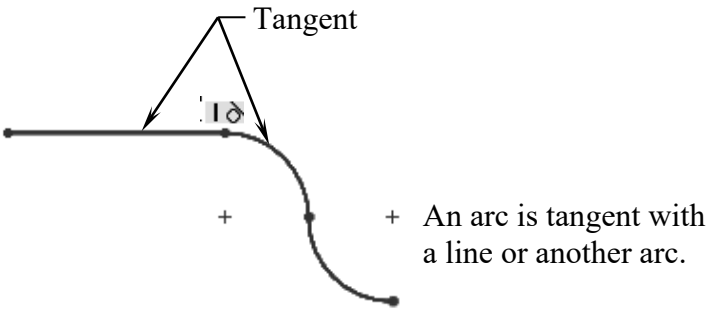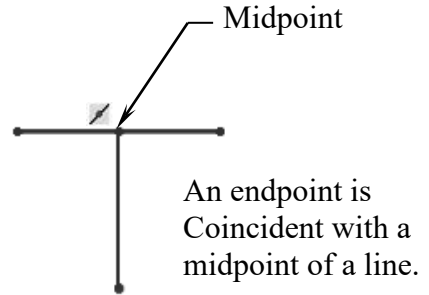midpoint of a line.

Tangent

An arc is tangent with
a line or another arc.

Two circles are
sharing the same
center.

Concentric

Equal

Two circles or two lines
having the same size.

Tangent

Two lines are on the same level
(or Co-planar).

Collinear

Vertical

Two or more
points are
aligned vertically.

Horizontal

Two or more
points are
aligned
horizontally.

7. Adding the horizontal dimensions:

- Select [icon] from the Sketch toolbar - OR - select **Insert / Dimension**, 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.
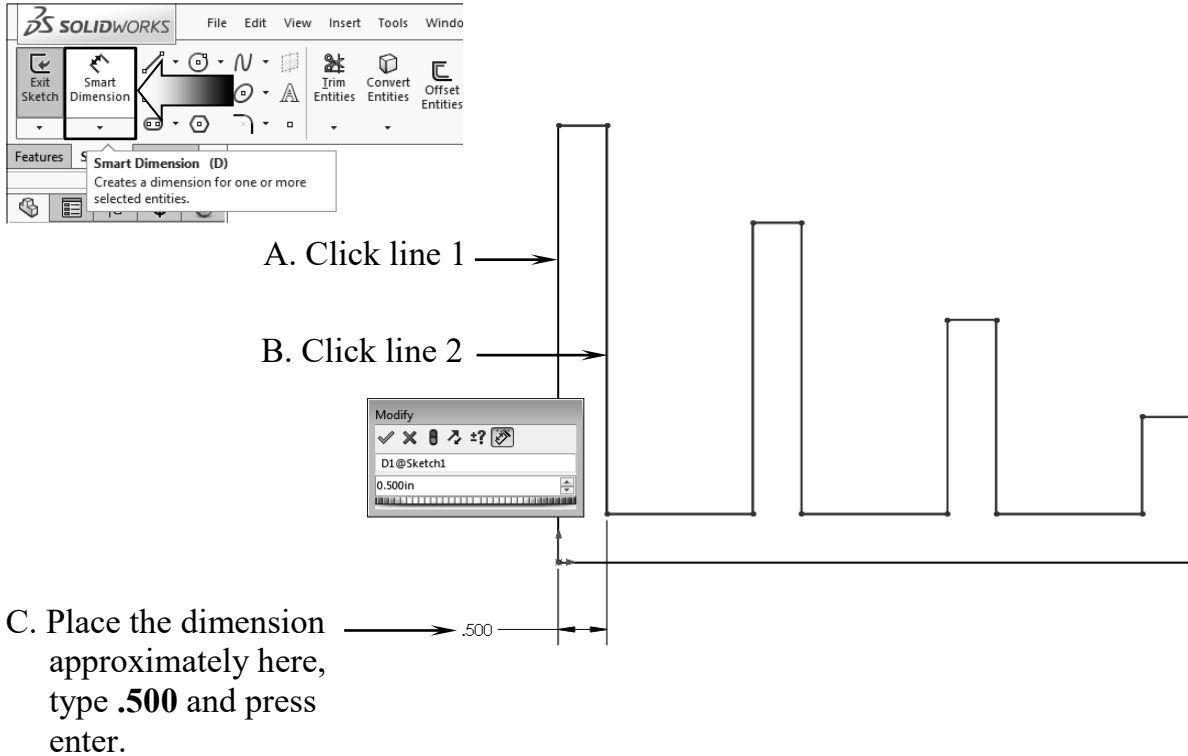
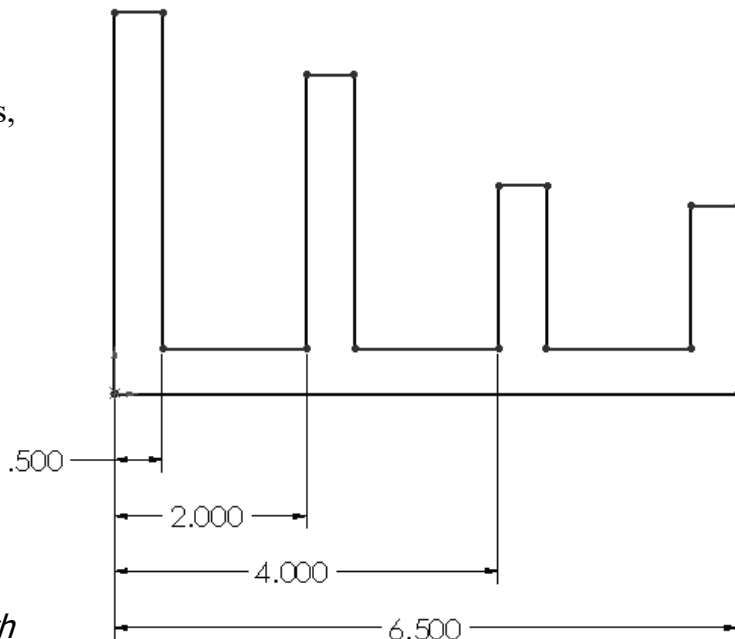- The Inch-Units is filled in automatically because it has been set previously to Inches, 3 decimal places.

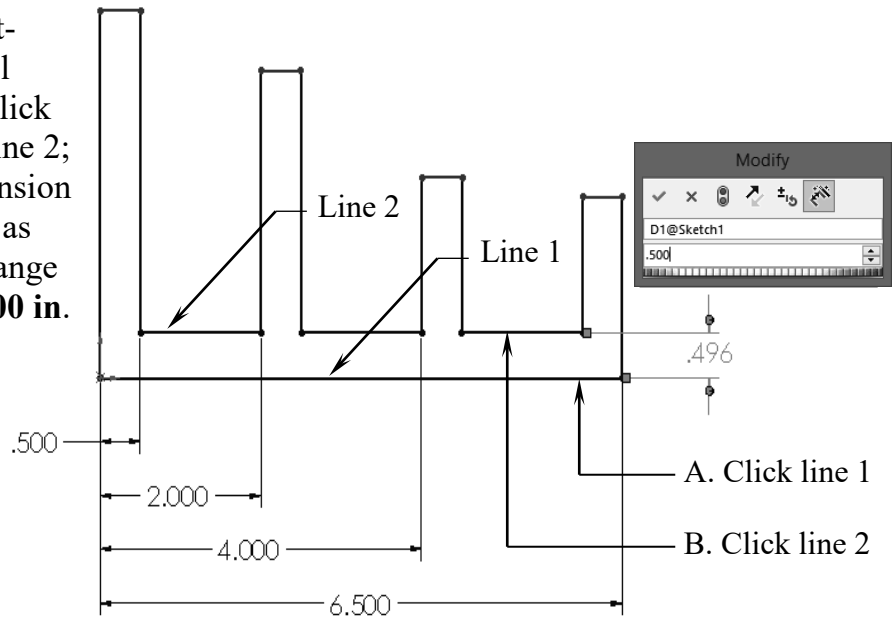- Continue adding the horizontal dimensions as shown here.

NOTE:
The color of the sketch lines changes from Blue to Black, to indicate that they have been constrained with dimensions.
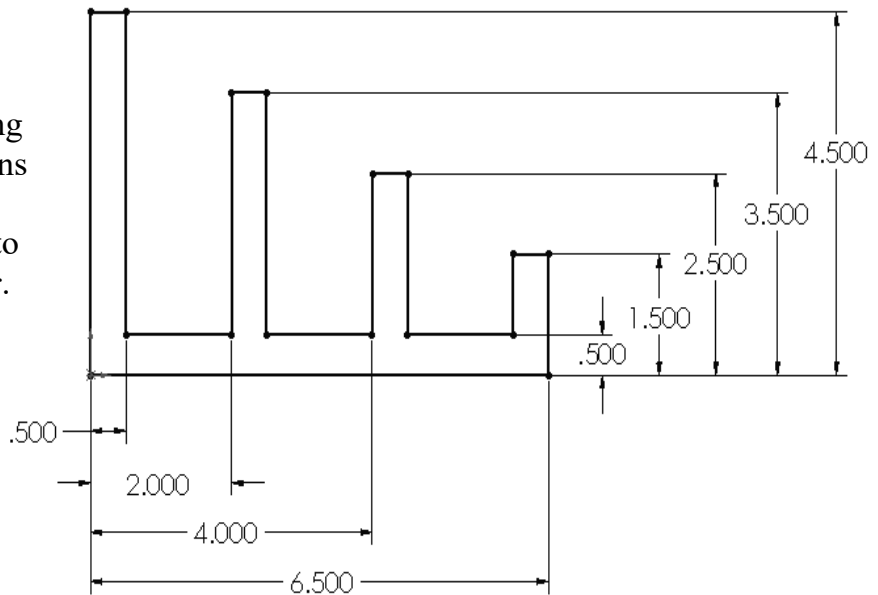
.500

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**.
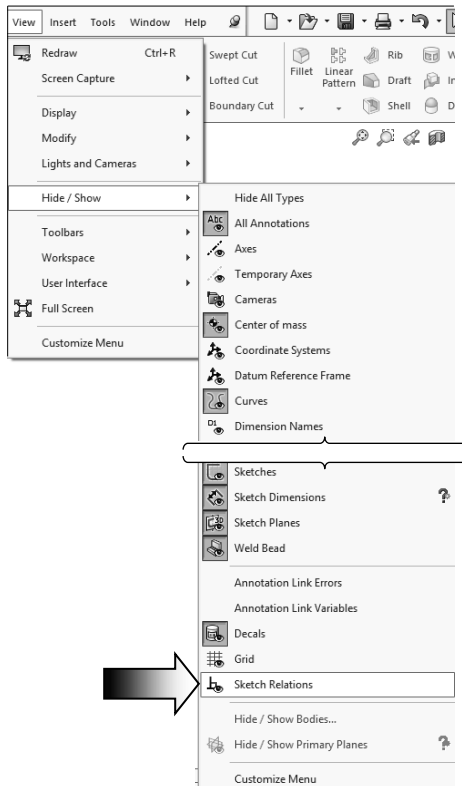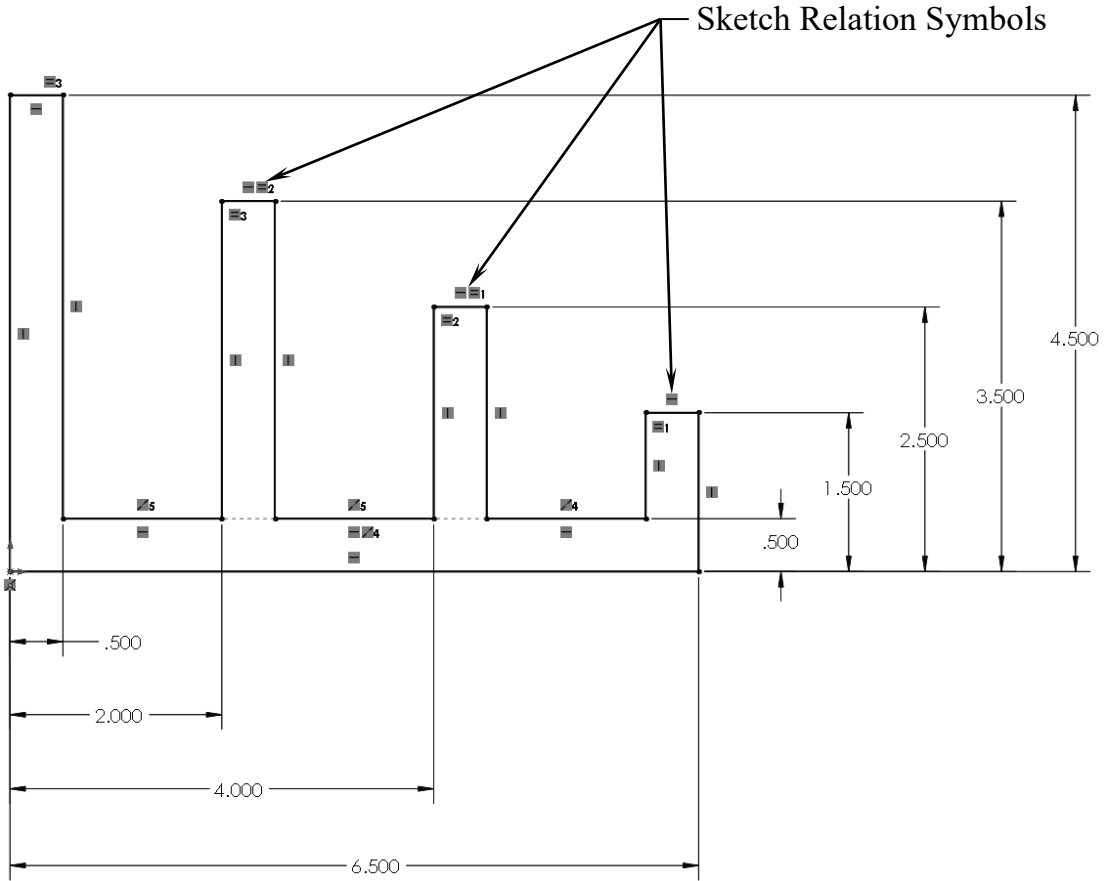
Line 2

Line 1

Modify

D1@Sketch1
.500

.496

A. Click line 1

B. Click line 2

.500

2.000

4.000

6.500

- 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
Under Defined	=	**Blue**	Under Defined
Over Defined	=	**Red**	Over Defined

Sketch Relation Symbols

4.500

3.500

2.500

1.500

.500

.500

2.000

4.000

6.500

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** menu and Click off the **Sketch Relations** option.

View	Insert	Tools	Window	Help

Redraw   Ctrl+R
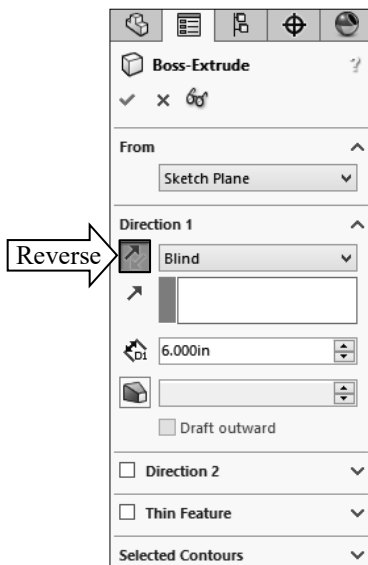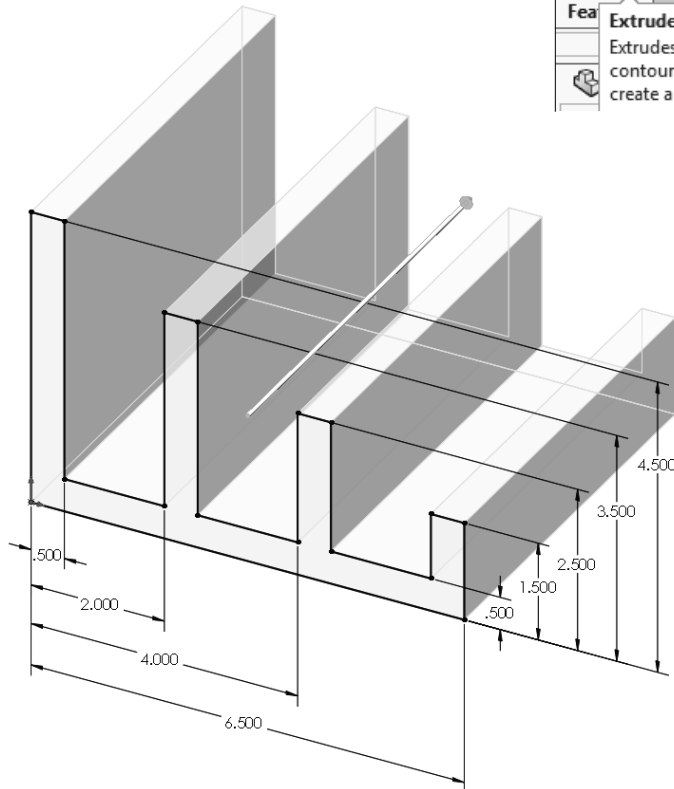
Screen Capture

Display

Modify

Lights and Cameras

Hide / Show

Toolbars

Workspace

User Interface

Full Screen

Customize Menu

Hide All Types

All Annotations

Axes

Temporary Axes

Cameras

Center of mass

Coordinate Systems

Datum Reference Frame

Curves

Dimension Names

Sketches

Sketch Dimensions

Sketch Planes

Weld Bead

Annotation Link Errors

Annotation Link Variables

Decals

Grid

Sketch Relations

Hide / Show Bodies...

Hide / Show Primary Planes

Customize Menu

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 [Extruded Boss/Base] from the Features toolbar OR select **Insert / Boss Base / Extrude**.

Extruded Boss/Base (E)
Extrudes a sketch or selected sketch contours in one or two directions to create a solid feature.
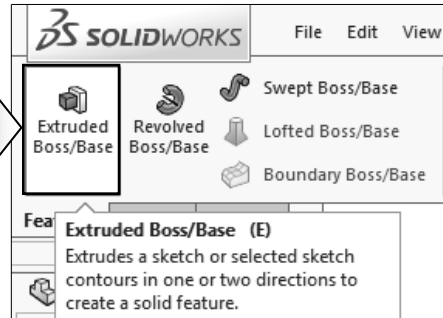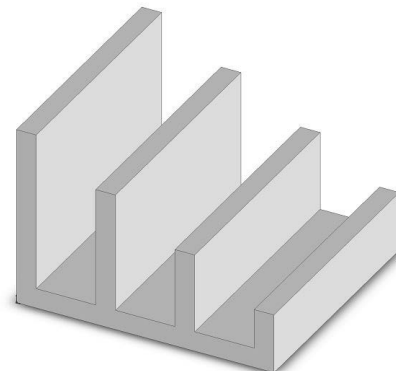
- Set the following:

- Direction: **Blind**.

- Depth: **6.00 in**.
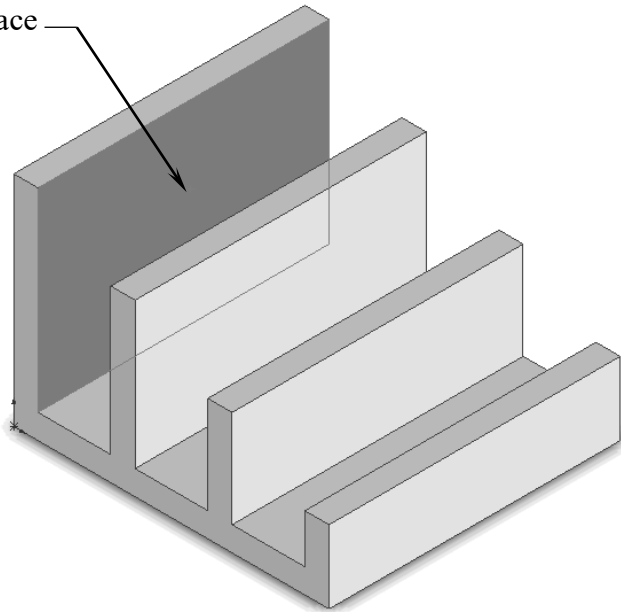
- Enabled **Reverse** direction.

- Click **OK** ✓.

11. Sketching on a Planar Face:

- Select the face as indicated.

- Click [Sketch] or select **Insert/Sketch** and press the shortcut keys **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, and click and drag outward to draw a circle.

R = 1.015

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

- Dimensions are added after the profile is created.
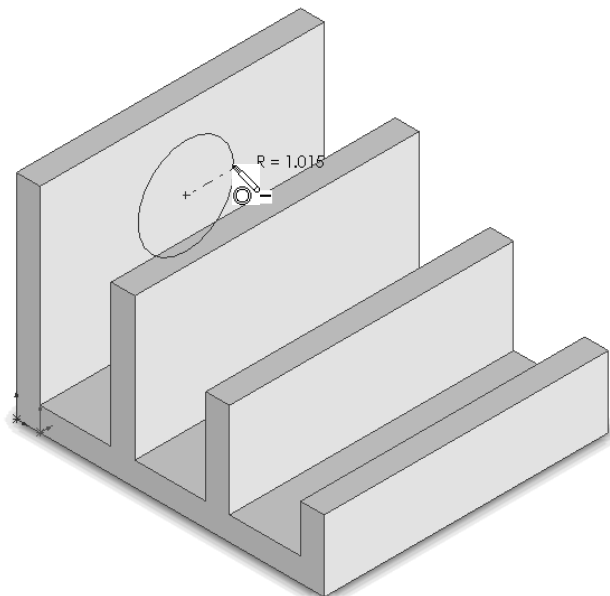
- Select the **Smart Dimension**
command [Smart Dimension] and add a
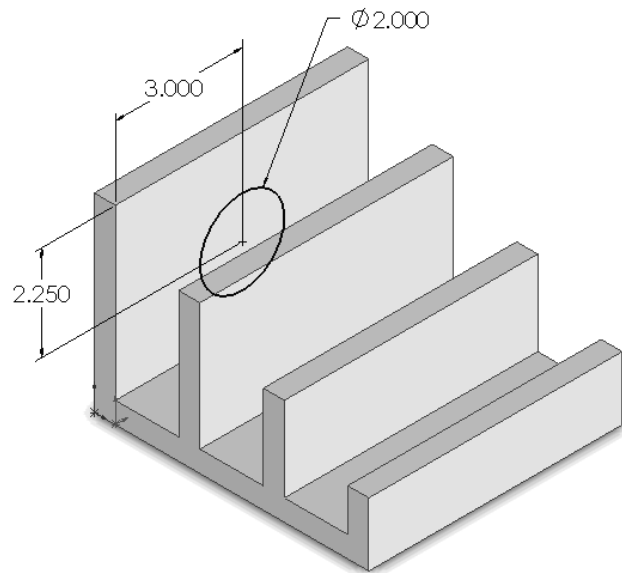diameter dimension to the
circle.

(Click on the circle and move
the mouse cursor outward at
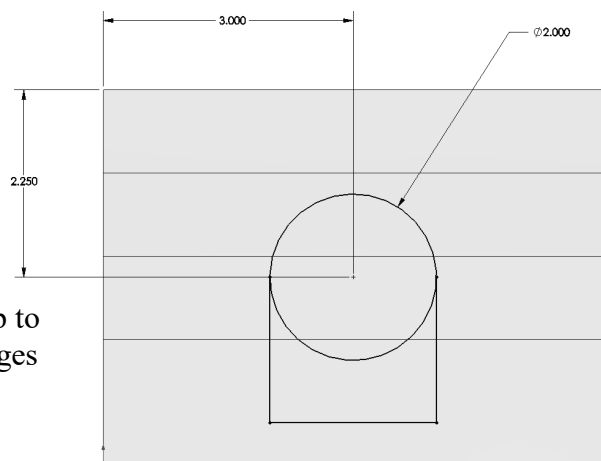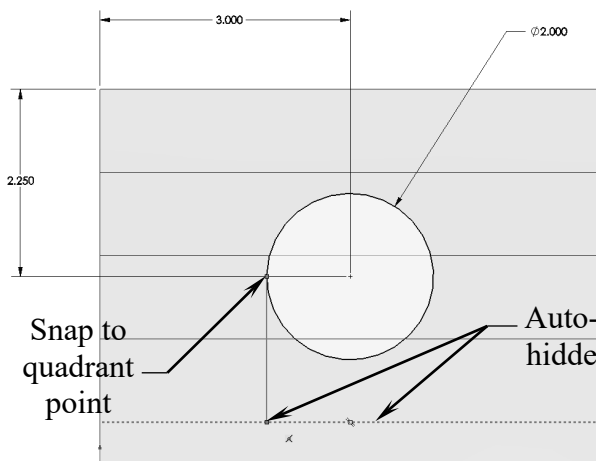approximately 45 degrees, and
place the dimension.)
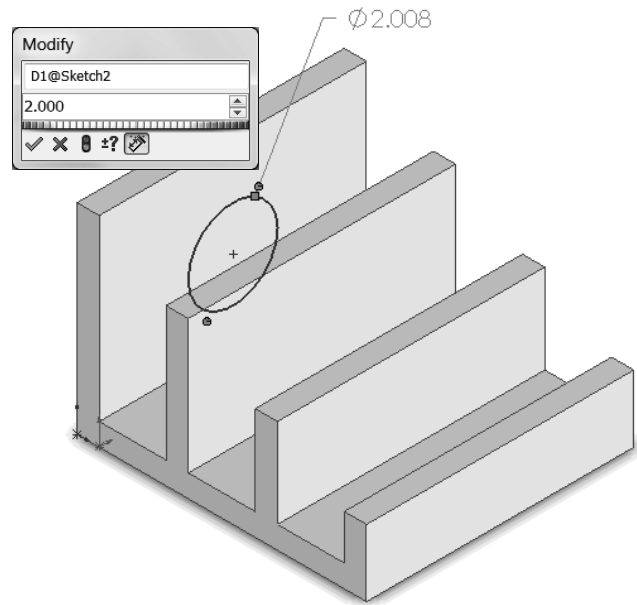
- To add the location dimensions
click the edge of the circle and
the edge of the model, place
the dimension, then correct
the value.

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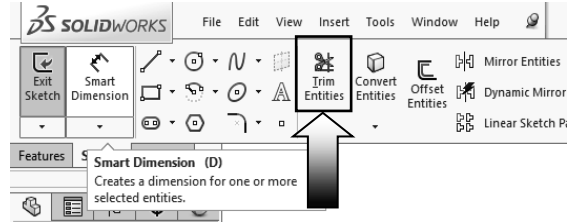
- The color of the sketch should
change to black at this point (Fully
Defined).

Ø2.008

Modify

D1@Sketch2

2.000

Ø2.000

3.000

2.250

Ø2.000

3.000

2.250

Ø2.000

3.000

2.250

Snap to
quadrant
point

Auto-Snap to
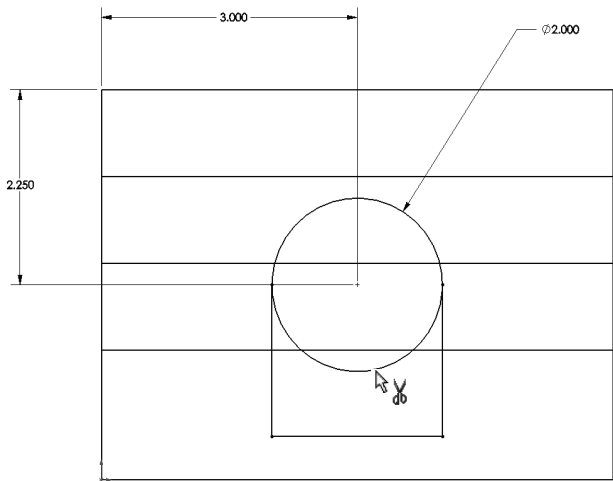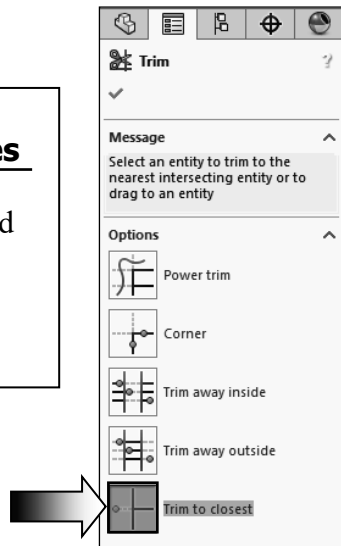hidden edges

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 the entities, this trim command highlights the entities prior to trimming to the next intersection.
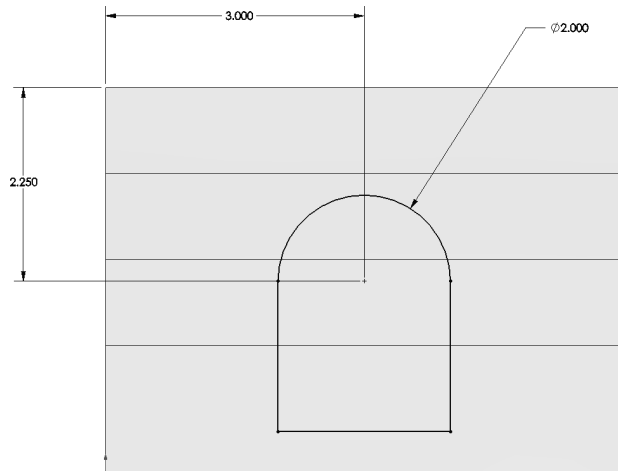
> 💡 **Trim Entities**
>
> Use this command to trim, extend or delete a sketch entity.

- Hover the pointer over the lower portion of the circle, the portion that is going to be trimmed-off lights up. 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 into a feature.
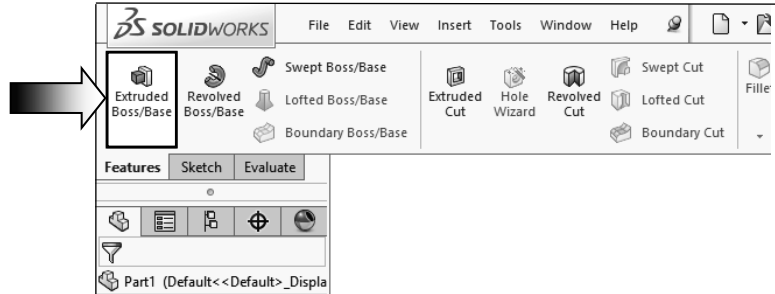
- Next, we are going to look at some of the extrude options available in SOLIDWORKS.

13. Extruding a Boss:

- Switch to the Feature toolbar and click ⬚ or select:
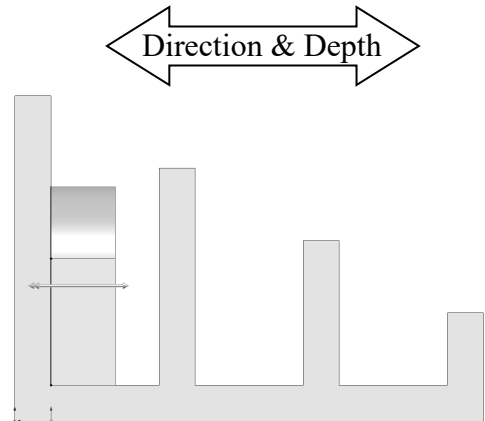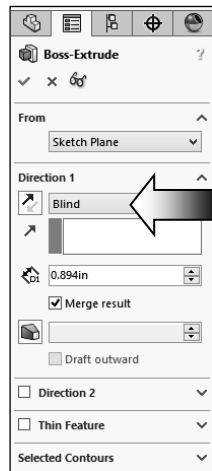**Insert / Boss-Base / Extrude**.

💡 **Extrude Options...**

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

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

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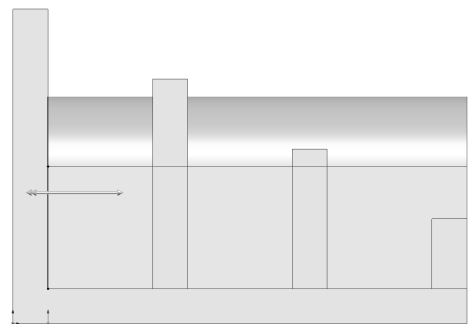
* Direction

* Depth dimension
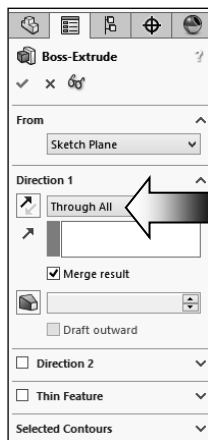
- Drag the direction arrow on the preview graphics to define the direction, then enter a dimension for the extrude depth.

Direction & Depth

Blind
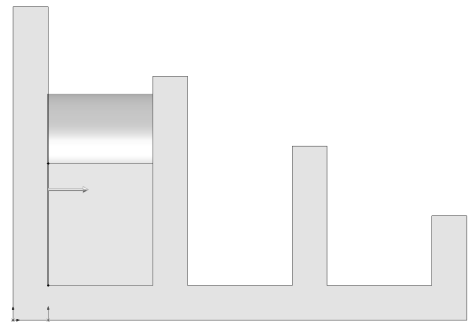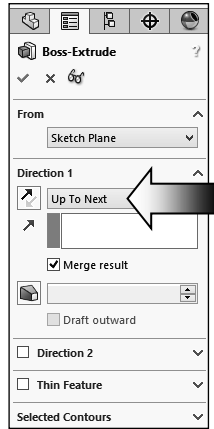Condition

(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

Ⓒ Using the Up To Next option:

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

Up To Next
Condition

Ⓓ 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
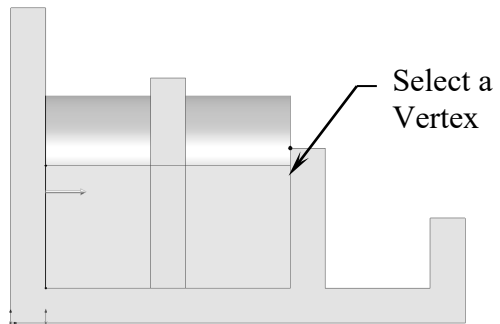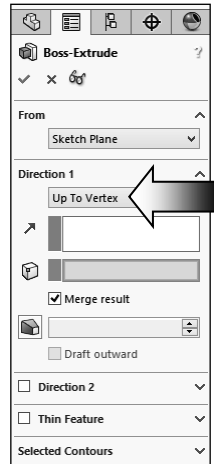
Up To Vertex
Condition

Ⓔ Using the Up To Surface option:

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

Select a Surface

Up To Surface
Condition

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

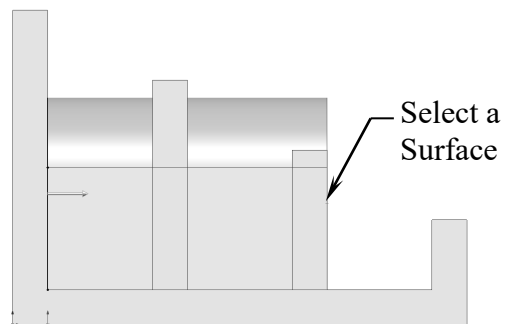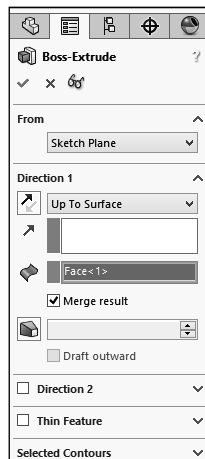- This option extrudes the sketch from its plane to a selected face, then offsets at a specified distance.

Select a surface to offset from & enter a distance.
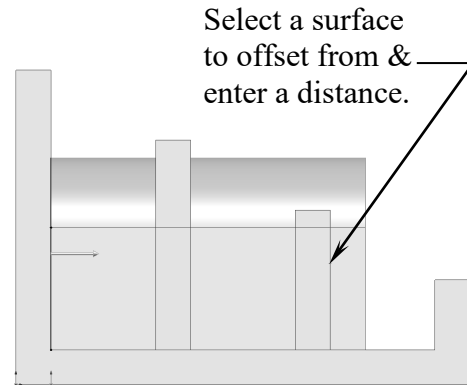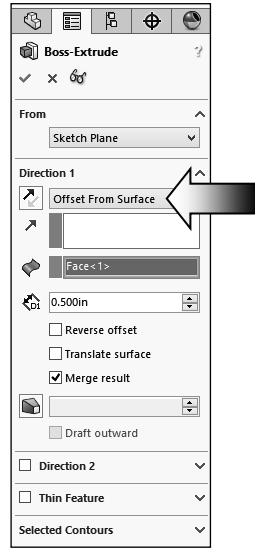
From Surface
Condition

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 (optional).
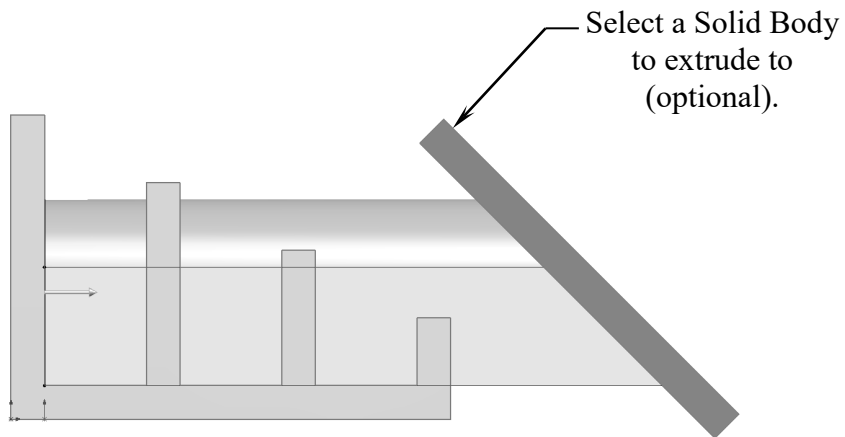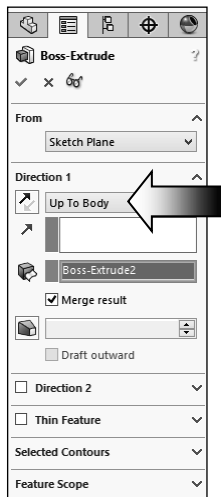
Up To Body
Condition

- 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.
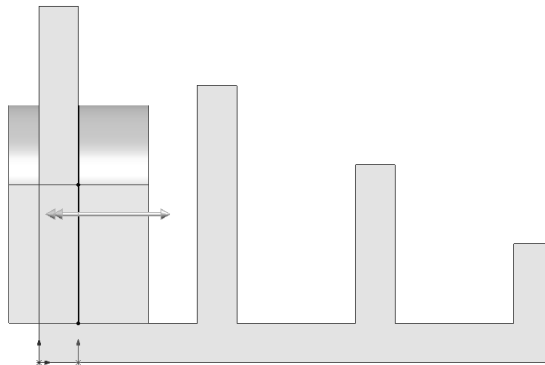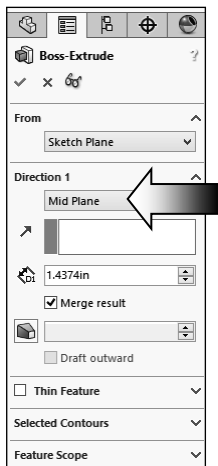
Mid Plane
Condition

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

- Click O**K** ✅.

- The system extrudes the circle 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.

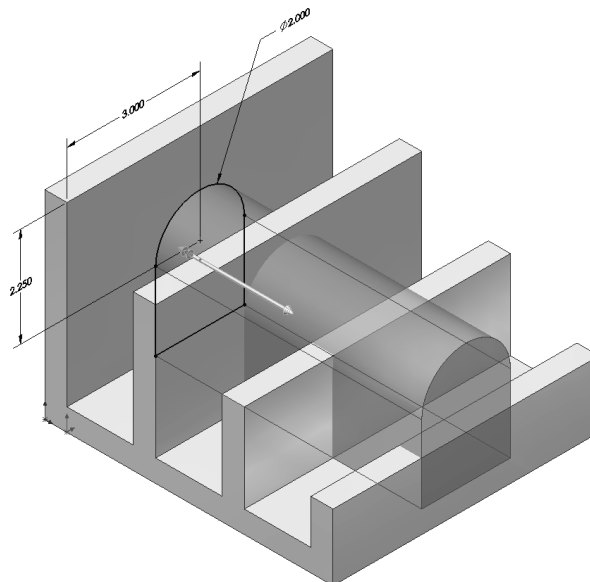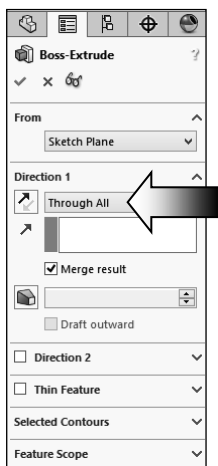- Unless the Merge Result checkbox is cleared, all interferences will be detected and removed.
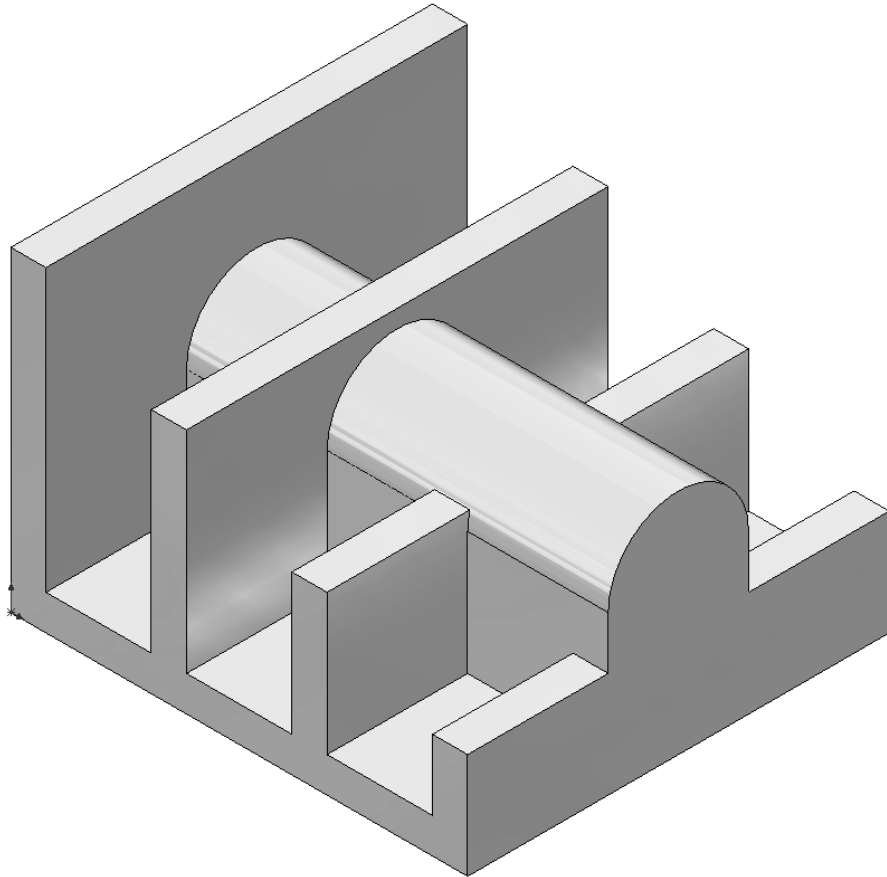
Extrude summary:

* The Extrude Boss/Base command is used to add thickness to a sketch and 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.

14. Adding the model fillets by Lasso*:

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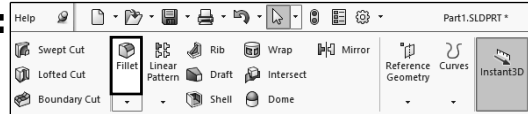
- The **radius** value stays in effect until you change it.
  Therefore, you can select any number of edges or faces in the same operation.

- Click [Fillet] or select **Insert / Features / Fillet/Round**.
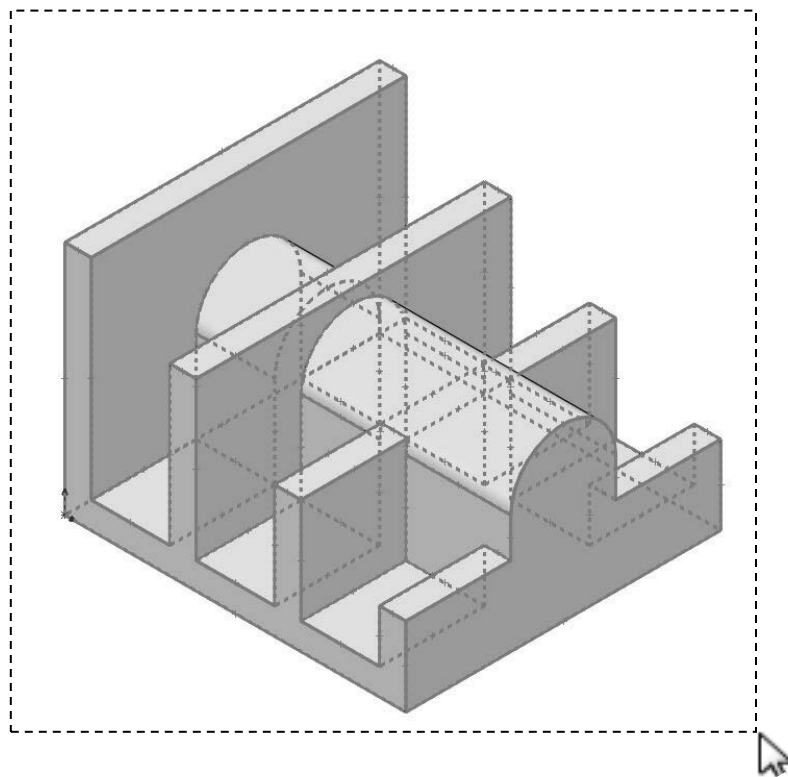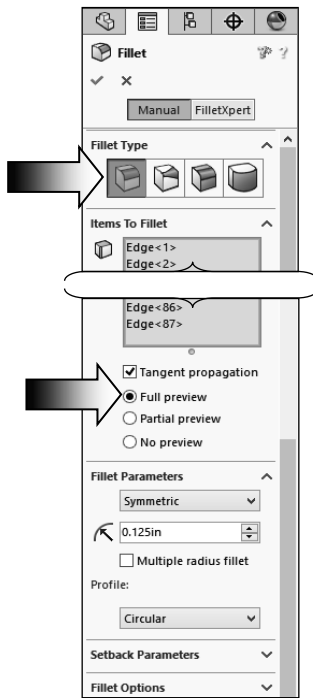
- Select the **Constant Size Fillet** button (Arrow).

- Either "drag-select" to highlight all edges of the model or press the shortcut key **Control+A** (select all).

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

- Enable the **Full Preview** checkbox.

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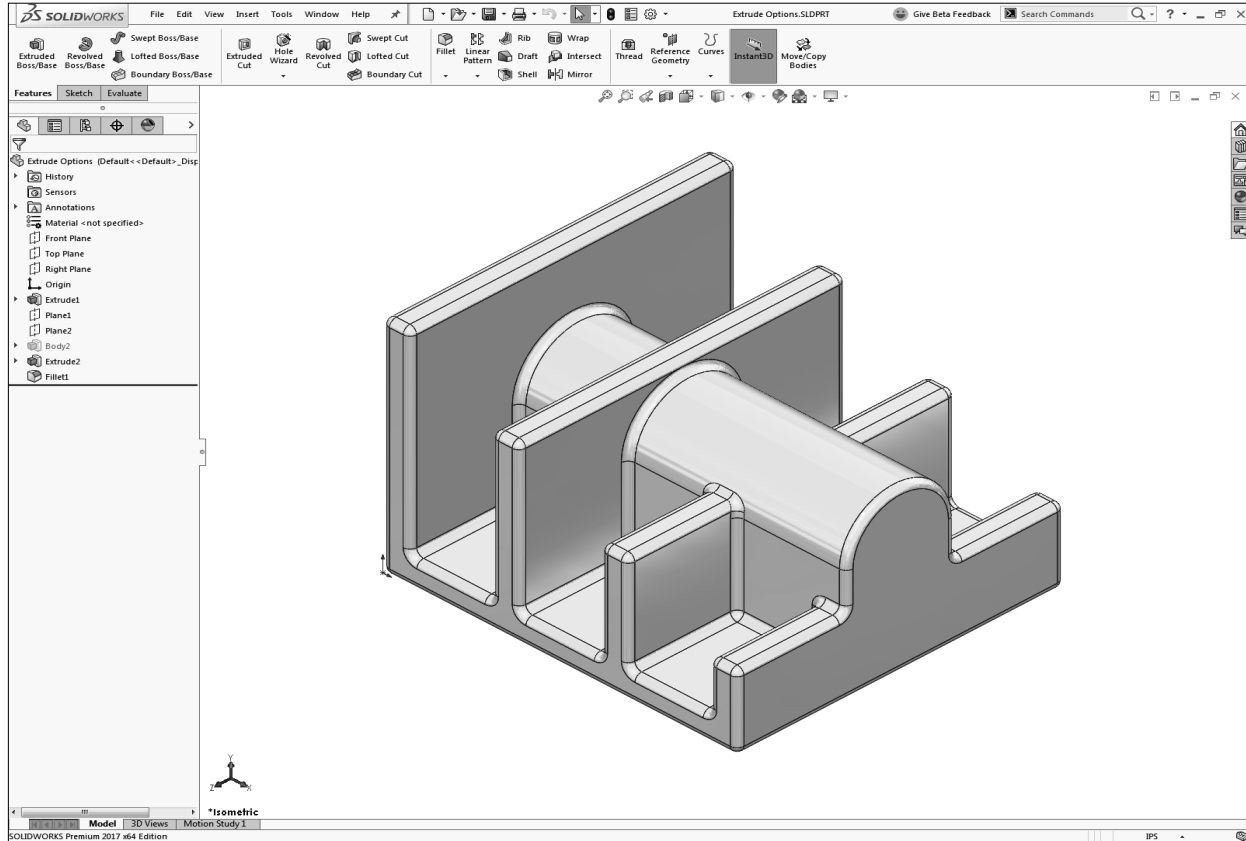
In the Training Files folder, in the Built Parts folder you will also find copies of the parts, assemblies, and drawings that were created for cross referencing or reviewing purposes.

* 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.

Fillet
(adds material)

Round
(removes material)
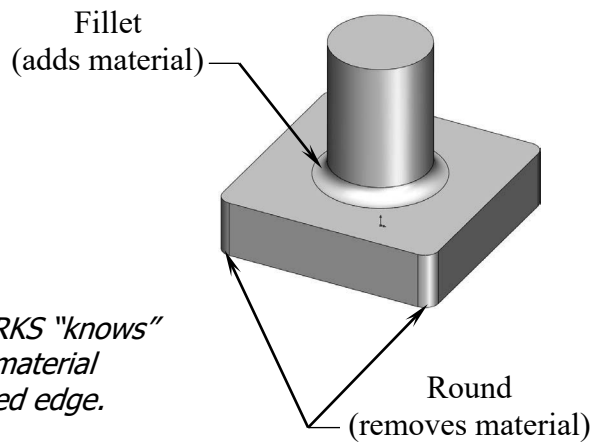
15. Saving your work:

- Select **File / Save As**.

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

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

- Click **Save**.

Fillet

Round

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.
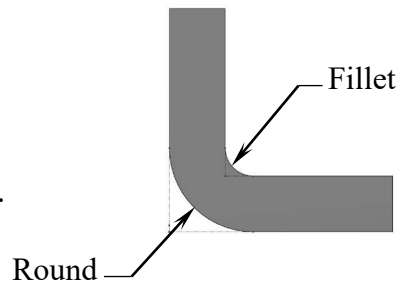   a. True
    b. False

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. 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 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

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 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 regularly.
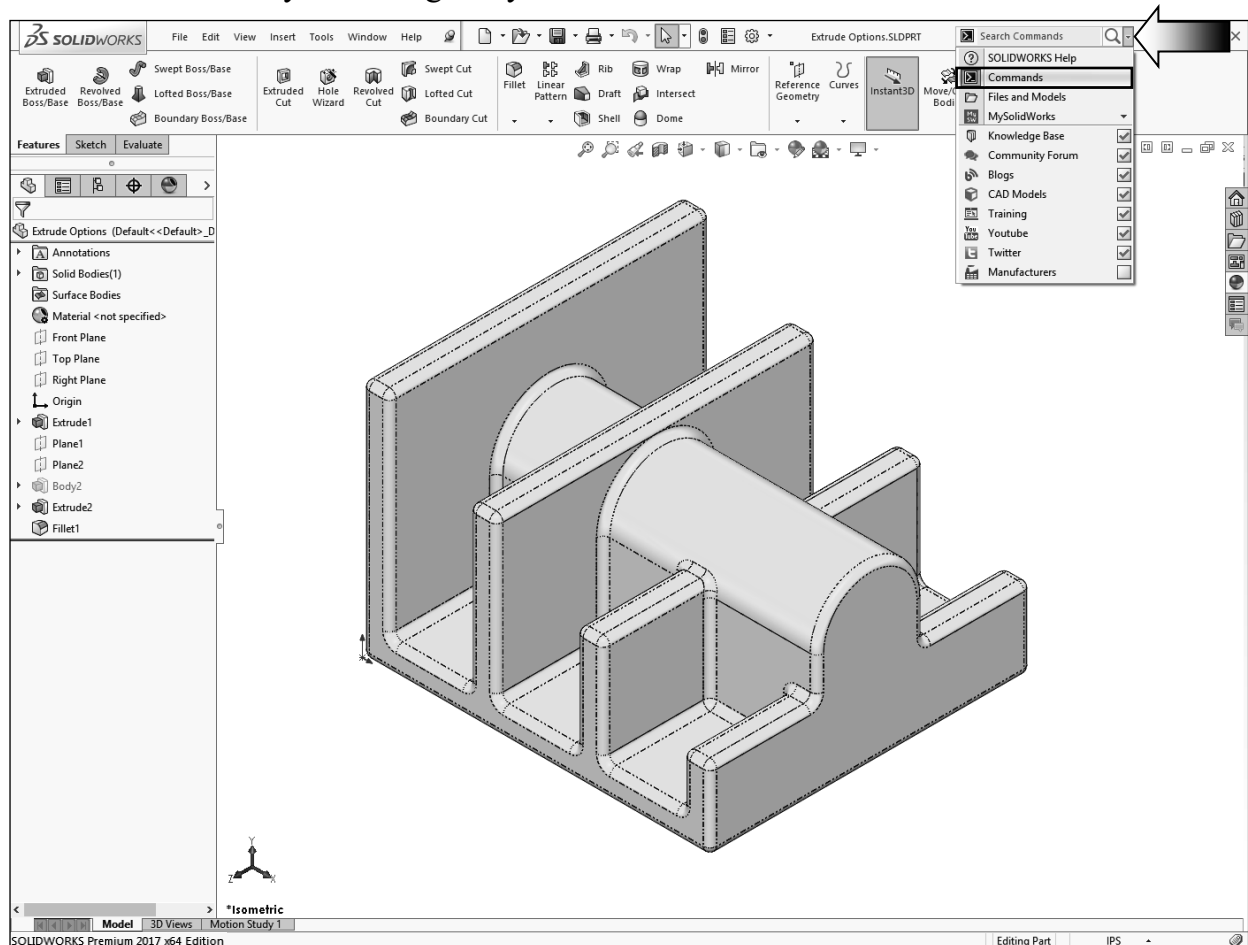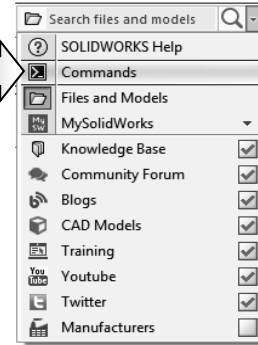
- Click the drop down arrow to see the search options (arrow).

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 the Feature Mode.

- With the part 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.
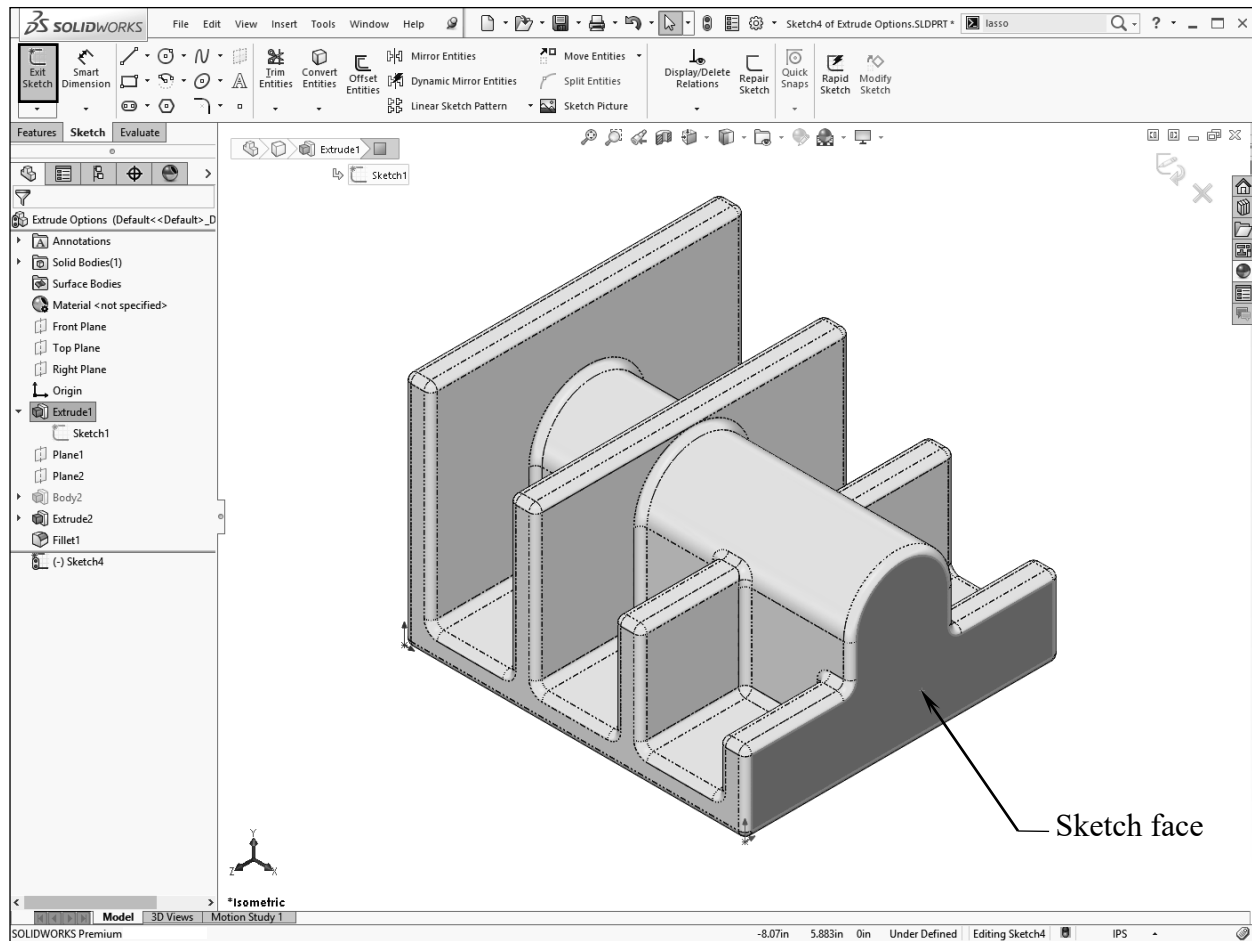

- Click **Show Command Location** 🔍 ; a **red arrow** indicates the command in the user interface.
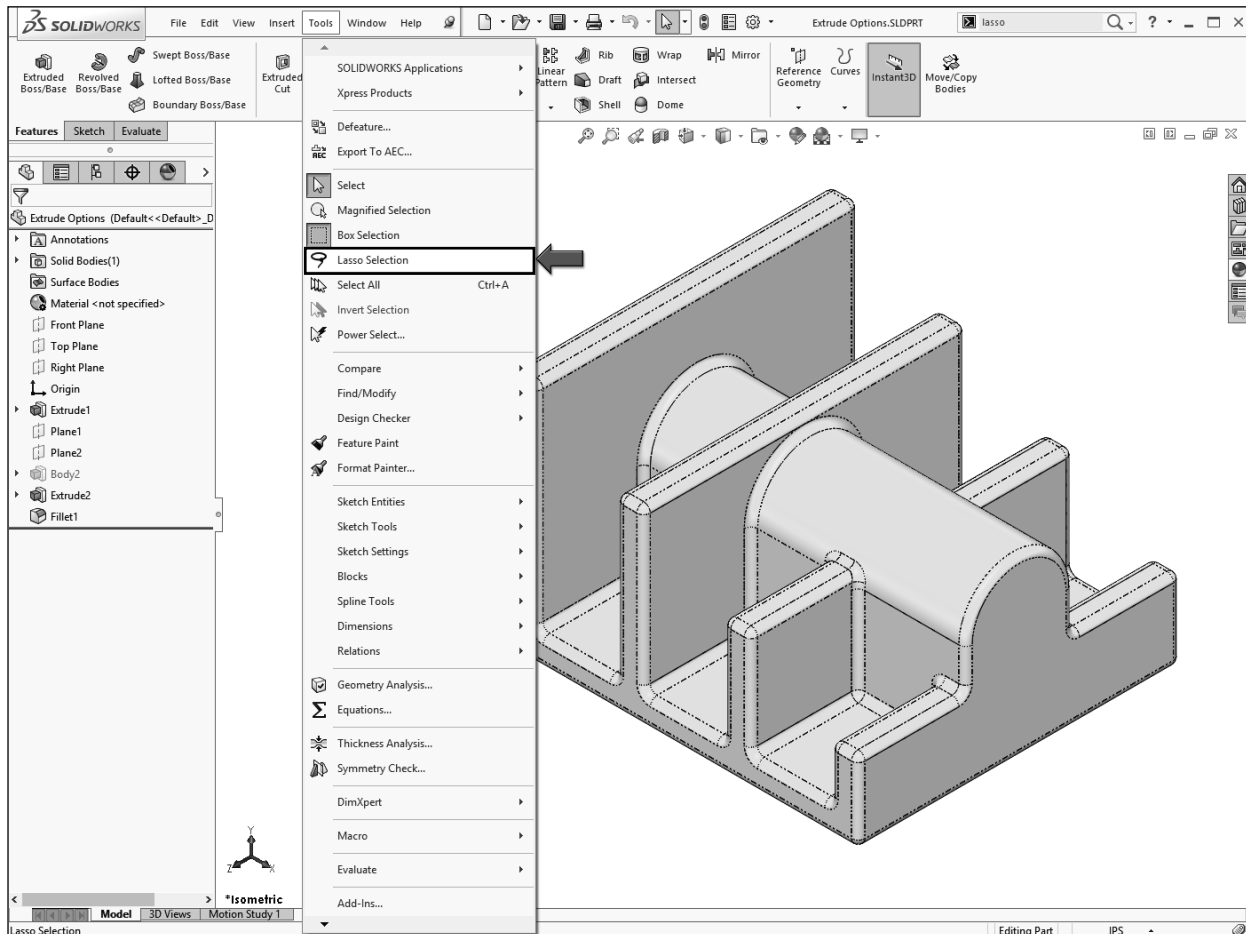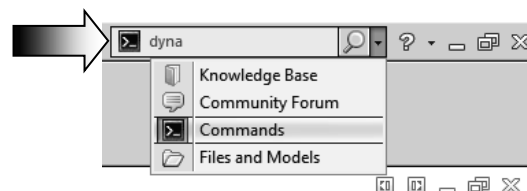
2. Search Commands in Sketch Mode:

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

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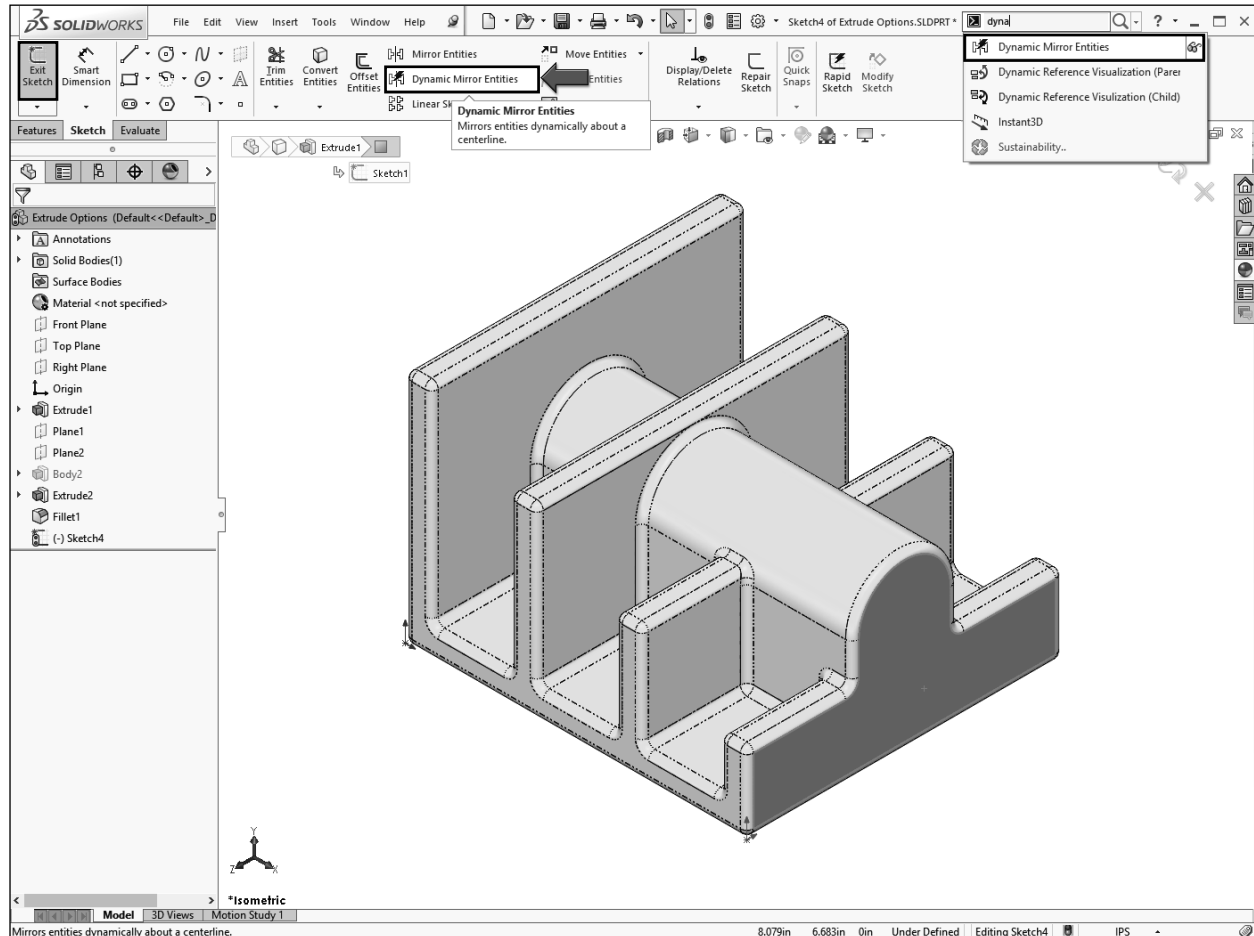

Sketch face

- 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** [icon]; a red arrow indicates the command in the user interface.


- 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, then click OK.

- **Save** and close all documents.

Exercise: 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 in the previous lesson to build 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.
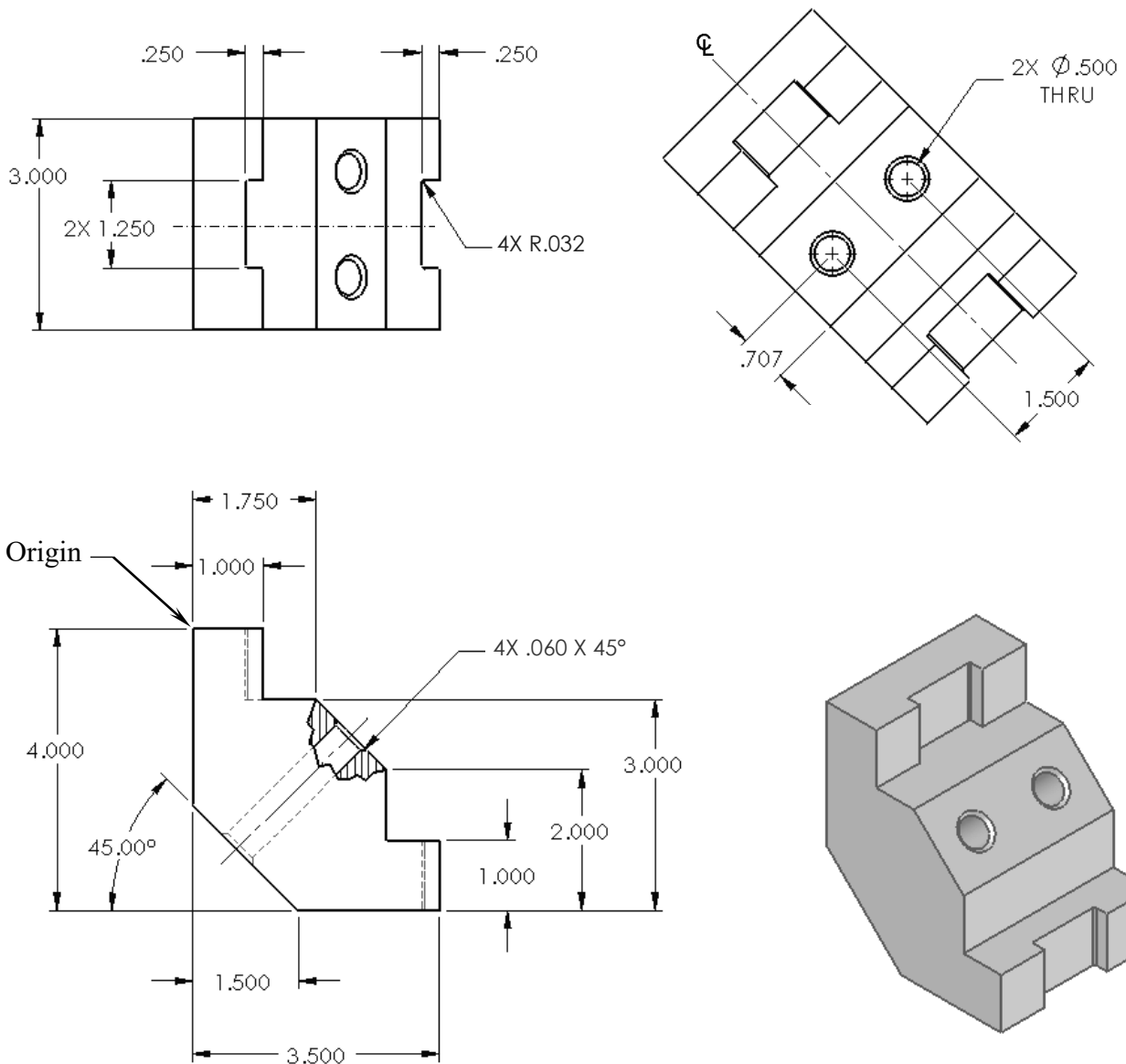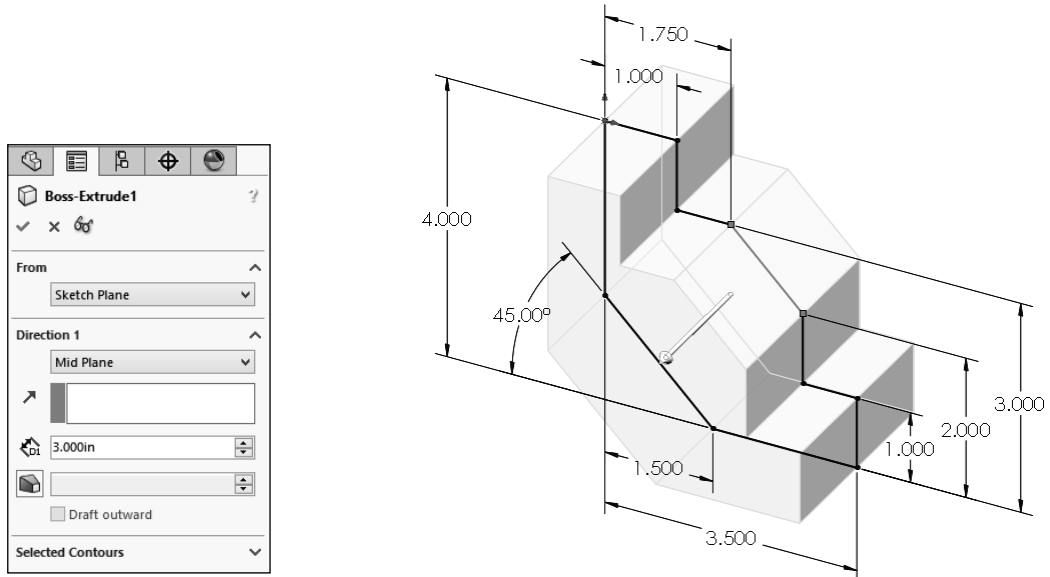
1. Starting with the base sketch:

- Select the <u>Front</u> plane and open a new sketch.

- Starting at the top left corner, using the line command, sketch the profile below.
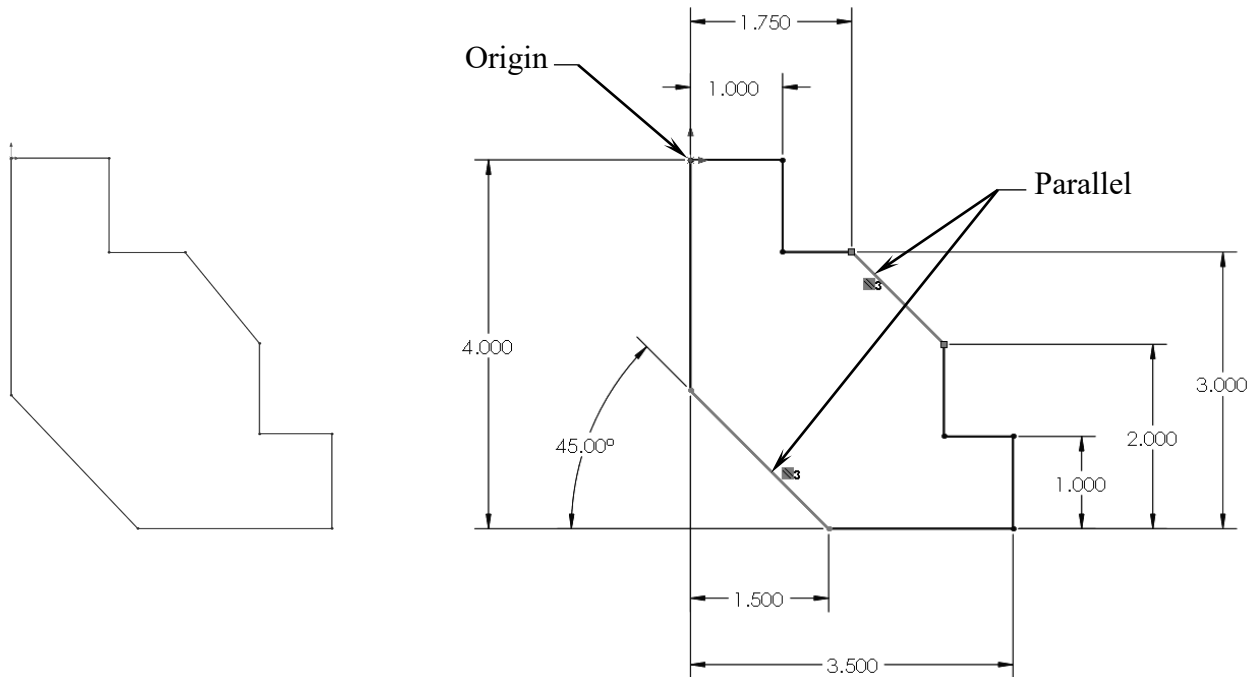
- Add the dimensions shown.

- Add the Parallel relation to fully define the sketch.

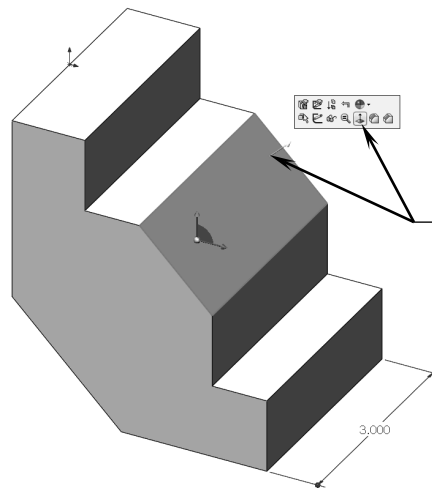- Extrude Boss/Base with **Mid Plane** and **3.000"** in depth.

2. Adding the through holes:

- Select the <u>face</u> as indicated and click the Normal-To button.

- This command rotates the part normal to the screen.

Select this face and click the Normal-To button
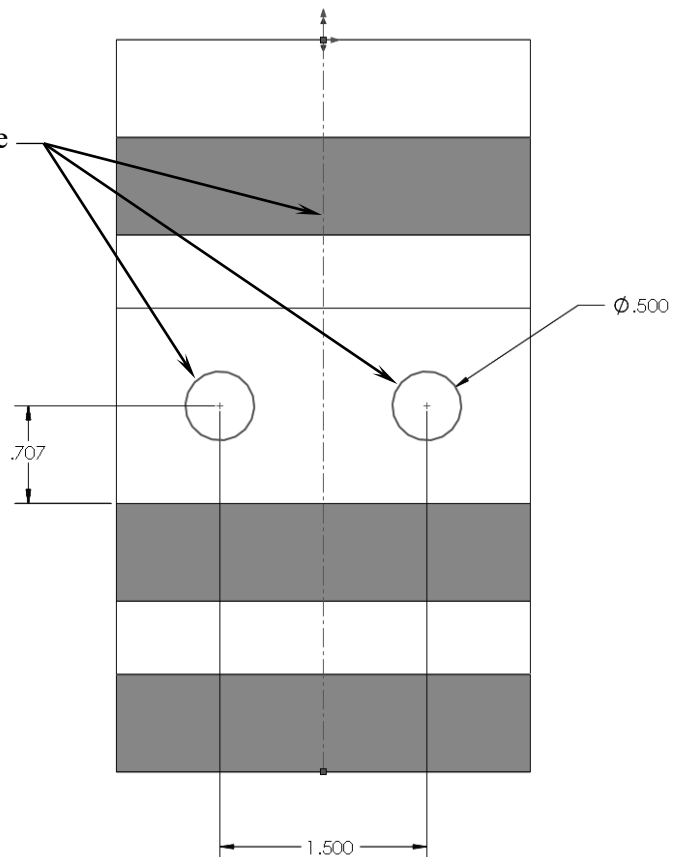
- The hot-key for this command is **Ctrl + 8**.

3.000

- 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.

Both circles are Symmetric about the Centerline

Ø.500

.707

- Hold the Control key and select both circles <u>and</u> the centerline, then click the Symmetric relation on the properties tree.
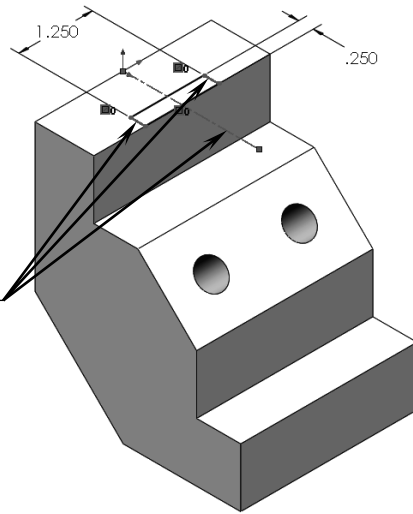
1.500

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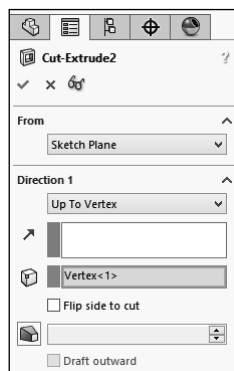 cut using the **Through-
  All** condition.

Ø.500

.707

1.500
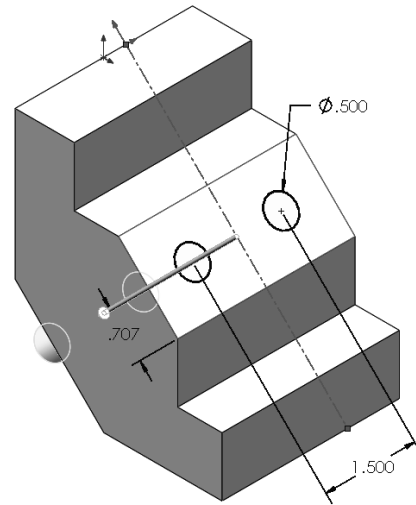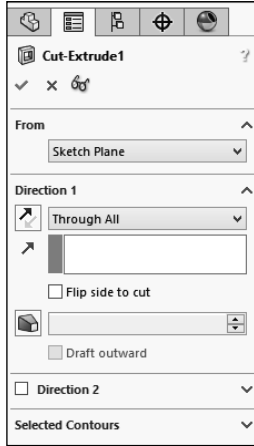
3. Adding the upper cut:

1.250

.250

- Select the <u>upper face</u> and click the
  Sketch pencil to open a new sketch.

- Sketch a centerline
  that starts at the
  Origin.

Both lines are
Symmetric about the
Centerline

- 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).

1.250

.250

- Select the Vertex indicated.
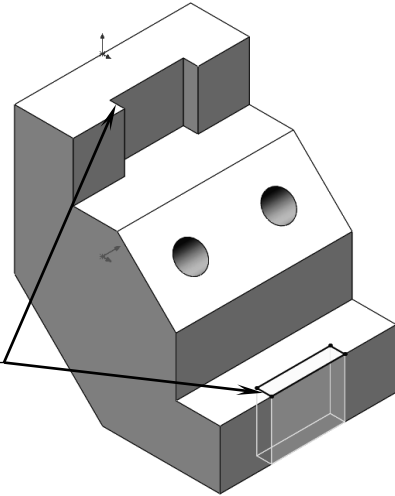
Select Vertex
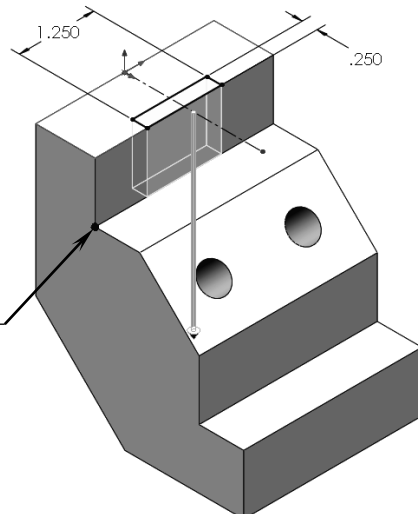
- Click **OK**.

4. Adding the lower cut:

- Select the <u>lower face</u> of the part and open a new sketch.

- Sketch a rectangle on this face.

- Add a Collinear <u>and</u> an Equal relation to the lines and the edges as noted.

The line is Collinear <u>and</u> Equal with the edge on both sides.

- Extrude a cut using the **Through All** condition.
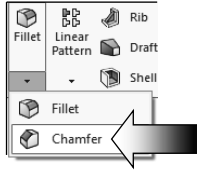
5. Adding fillets:

- 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.

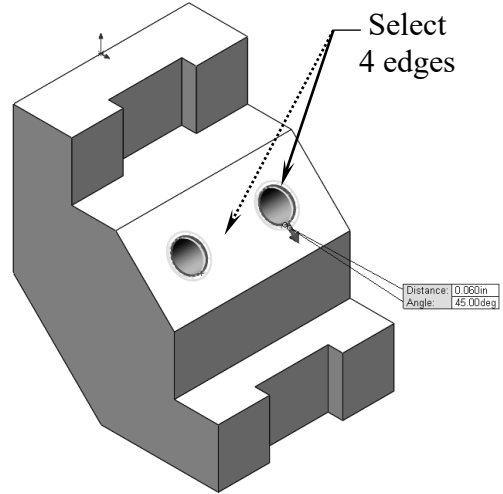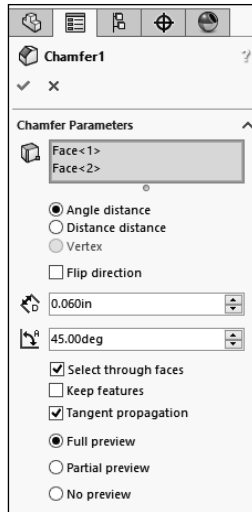- Keep all other options at their default settings.
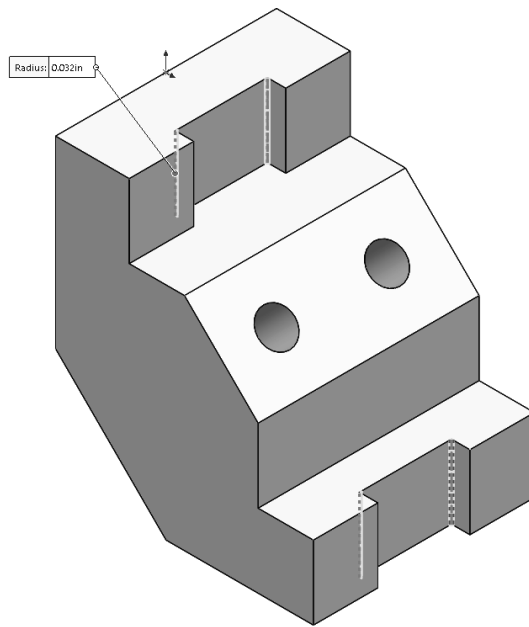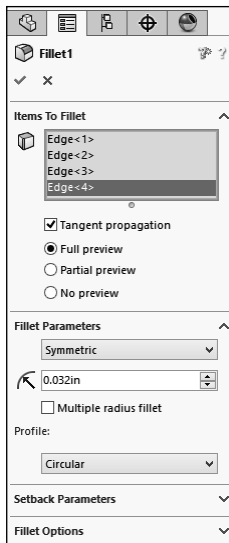
- Click **OK**.

6. Adding chamfers:

- Click **Chamfer** under the Fillet button.

- Enter **.060** for depth.

- Select the 4 circular edges of the 2 holes.

- Click **OK**.

Select 4 edges

7. Saving your work:

- Click **File / Save As**.

- Enter **Extrudes_Exe1** for the file name.

- Select a location to save the file.

- Click **Save**.