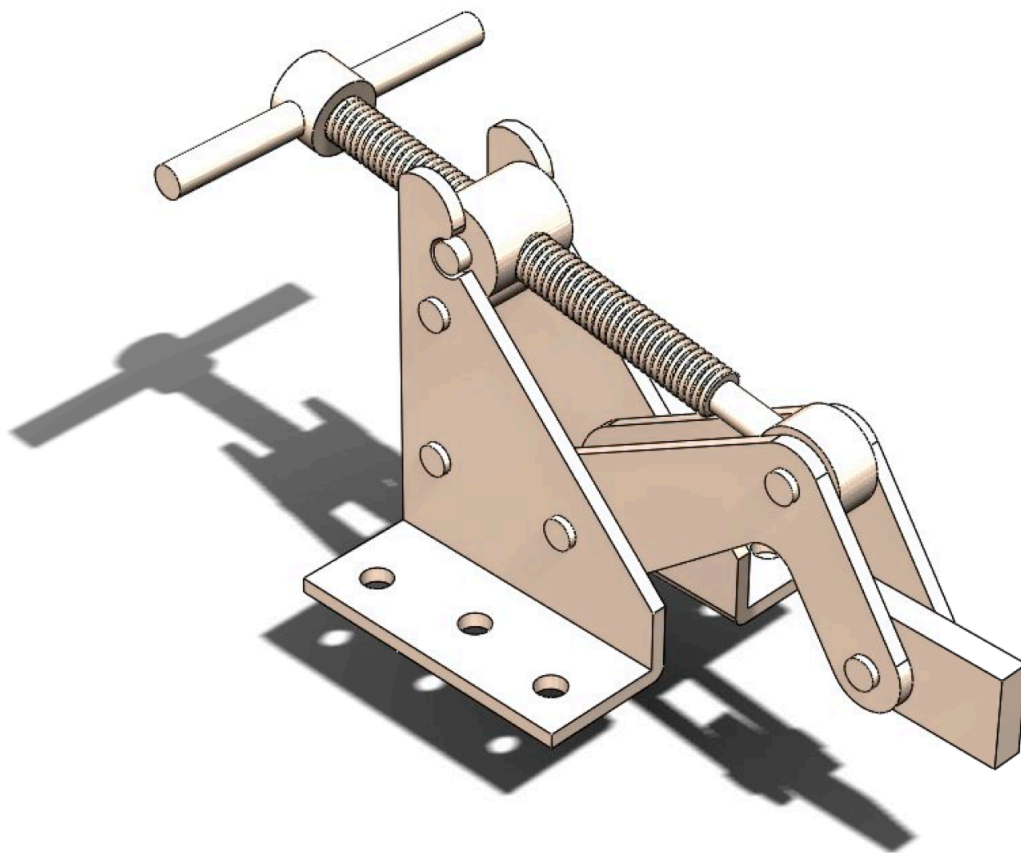


Part and Assembly Modeling

with SOLIDWORKS 2015

Huei-Huang Lee



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Preface

Use of This Book

This workbook is an introductory tutorial to geometric modelings using **SOLIDWORKS 2015**. It is not intended to be a comprehensive guide to parts and assembly modelings. It is prepared mainly for those students who have no experience in **SOLIDWORKS** geometric modeling, but want to acquire some. I provide this workbook to the students in my classroom and require them to complete the exercises in 3-4 weeks, to make them feel more comfortable when working on advanced capabilities of **SOLIDWORKS**, such as **Simulation**, **Motion**.

Companion Webpage

A webpage is maintained for this book:

http://myweb.ncku.edu.tw/~hhlee/Myweb_at_NCKU/SWG2015.html

The webpage contains links to following resources: (a) videos that demonstrate the steps of each section in this book, and (b) the finished **SOLIDWORKS** files of each section. (c) This book, in PDF format.

As for the finished files, if everything works smoothly, you may not need them at all. Every model can be built from scratch by following the steps in the book. I provide these files just in case you need them. For example, when you run into trouble and you don't want to redo it from the beginning, you may find these files useful. Or you may happen to have trouble following the steps in the book, you can then look up the details in these files.

Notations

Chapters and sections are numbered in a traditional way. Each section is further divided into subsections. For example, the first subsection of the second section of Chapter 3 is denoted as "3.2-1." Textboxes in a subsection are ordered with numbers, each of which is enclosed by a pair of square brackets (e.g., [4]). We refer to that textbox as "3.2-1[4]." When referring to a textbox from the same subsection, we drop the subsection identifier. For example, we simply write "[4]." Notations used in this book are summarized as follows (for further illustration, see page 4):

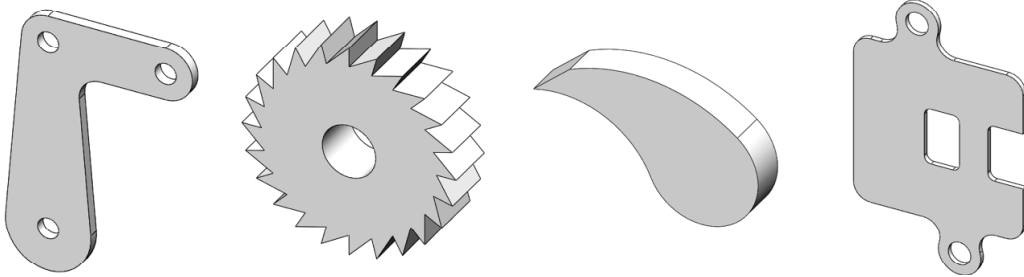
3.2-1	Numbers after a hyphen are subsection numbers.
[1], [2], ...	Numbers with square brackets are textbox numbers.
SOLIDWORKS	SOLIDWORKS terms are boldfaced.
(Round-cornered textboxes)	A round-cornered textbox indicates some mouse or keyboard actions are needed.
(Sharp-cornered textboxes)	A sharp-cornered textbox is used for commentary only; no mouse or keyboard actions are needed in that step.
#	A symbol # is used to indicate the last textbox of a subsection.

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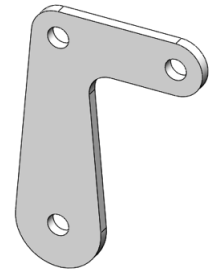
Chapter I

Sketching

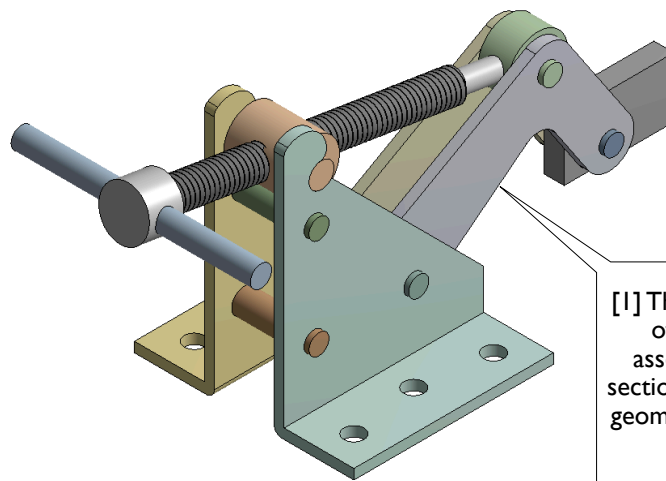


Section I.I

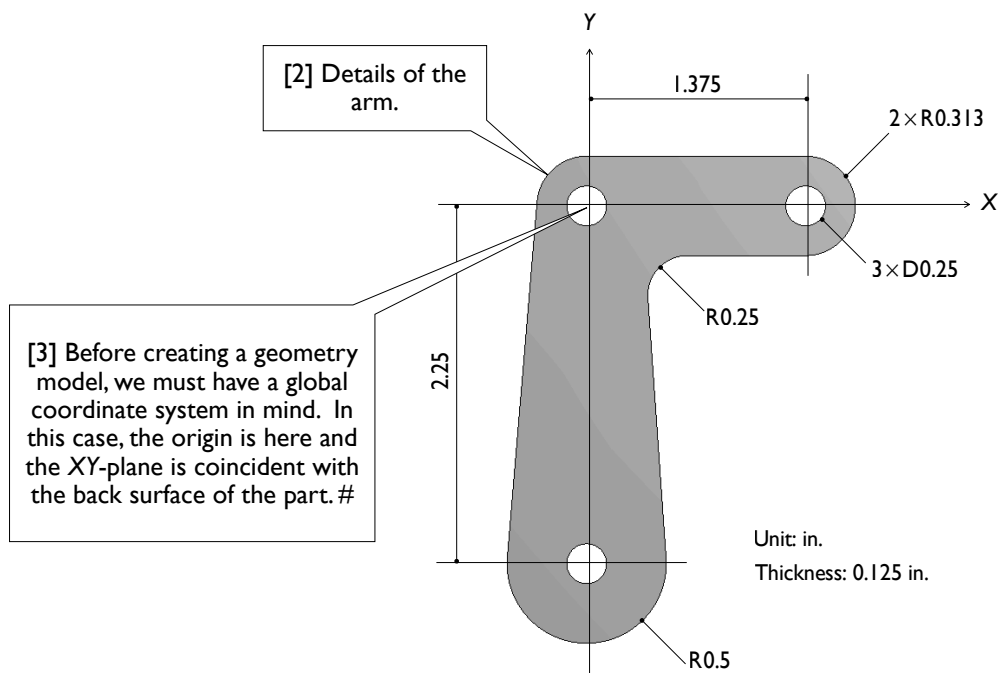
Arm



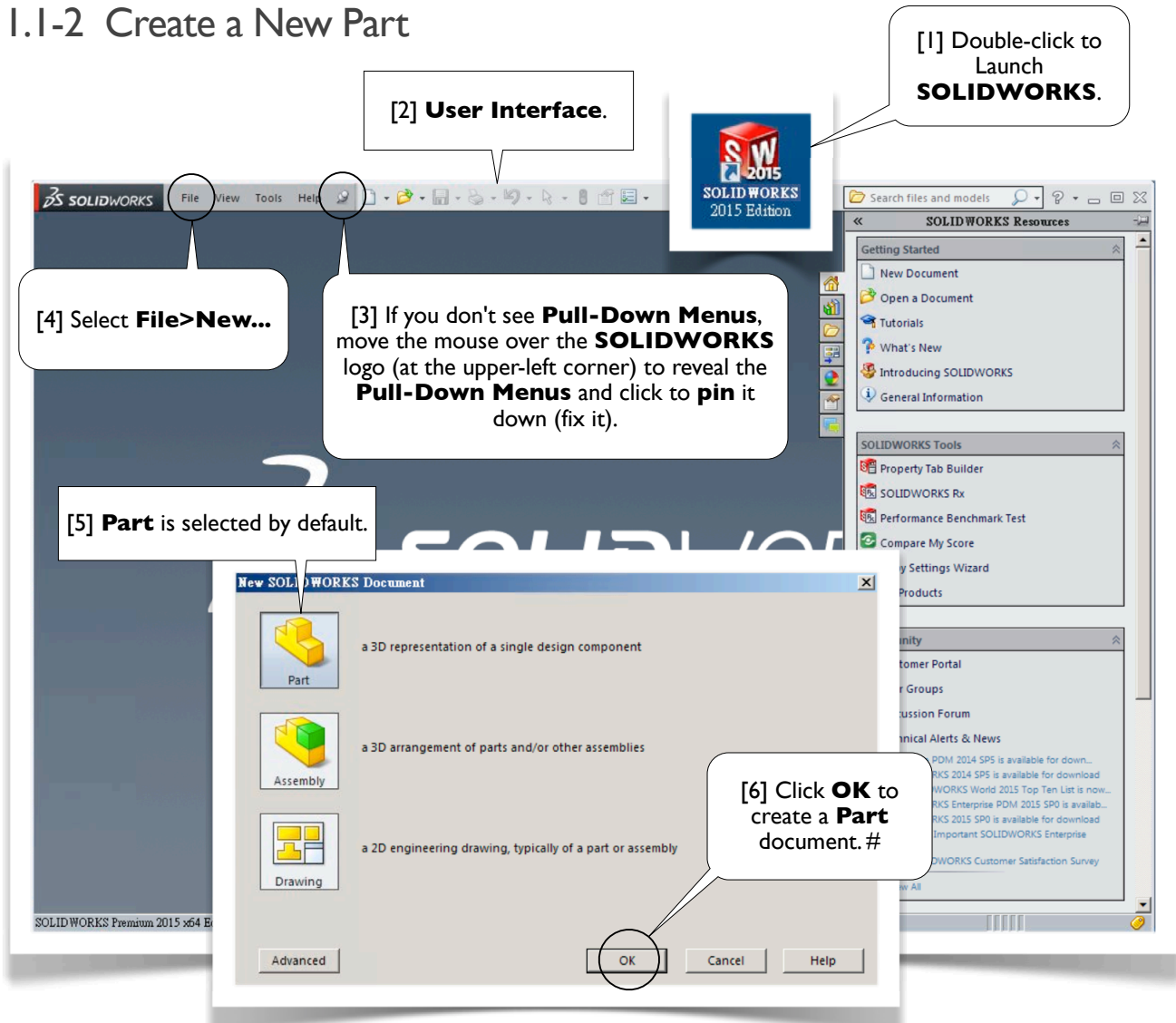
I.I-1 About the Arm



[1] The arm is a part of a clamping assembly. In this section, we'll create a geometric model for the arm.



1.1-2 Create a New Part



About the **Textboxes**

1. Within each subsection (e.g., 1.1-2), textboxes are ordered with numbers, each of which is enclosed by a pair of square brackets (e.g., [1]). When you read the contents of a subsection, please follow the order of the textboxes.
2. The textbox numbers are also used as reference numbers. Inside a subsection, we simply refer to a textbox by its number (e.g., [1]). From other subsections, we refer to a textbox by its subsection identifier and the textbox number (e.g., 1.1-2[1]).
3. A textbox is either round-cornered (e.g., [1, 3, 4, 6]) or sharp-cornered (e.g., [2, 5]). A round-cornered textbox indicates that **mouse or keyboard actions** are needed in that step. A sharp-cornered textbox is used for commentary only; i.e., mouse or keyboard actions are not needed in that step.
4. A symbol # is used to indicate the last textbox of a subsection [6], so that you don't leave out any textboxes.

SOLIDWORKS Terms

In this book, terms used in **SOLIDWORKS** are boldfaced (e.g., **Part** in [5, 6]) to facilitate the readability.

1.1-3 Set Up Units

[1] Select **Tools>Options...** which is near the bottom of the menu.

[7] The **Options** command is also available here.

[2] Click **Document Properties** tab.

[3] Select **Units**.

[4] Select **IPS**.

[5] Select **.123** (three decimal places).

[6] Click **OK**.

[8] The units also can be set up from here. #

The **Document Properties - Units** dialog box shows the following settings:

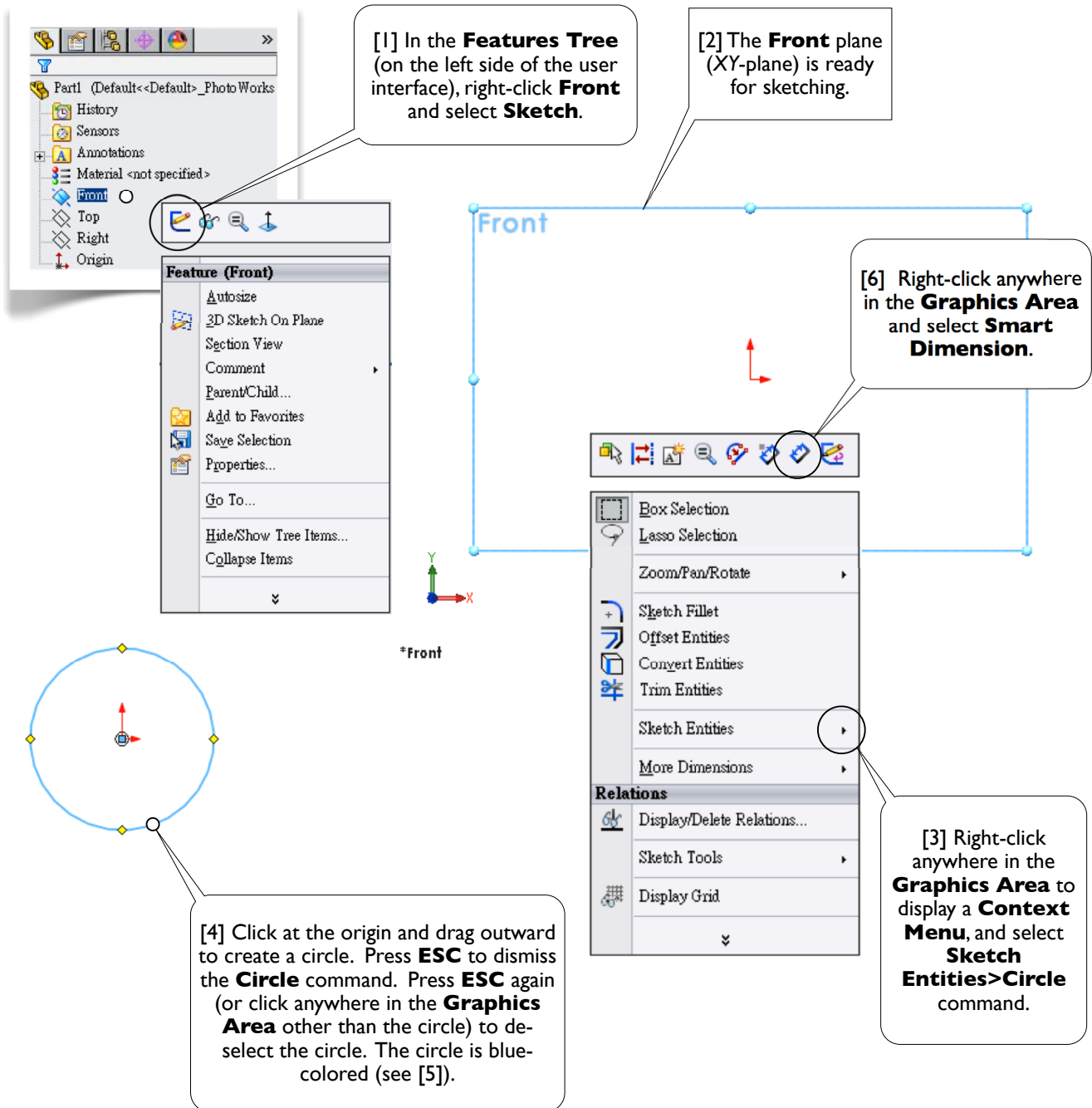
Type	Unit	Decimals	Fractions	More
Basic Units				
Length	inches	.123		
Dual Dimension Length	inches	.123		
Angle	degrees	.12		
Mass/Section Properties				
Length	inches	.12		
Mass	pounds			
Per Unit Volume	inches ³			
Motion Units				
Time	second	.12		
Force	pound-force	.12		
Power	watt	.12		
Energy	BTU	.12		

Decimal rounding options:

- ☒ Round half away from zero
- ☐ Round half towards zero
- ☐ Round half to even
- ☐ Truncate without rounding

☒ Only apply rounding method to dimensions

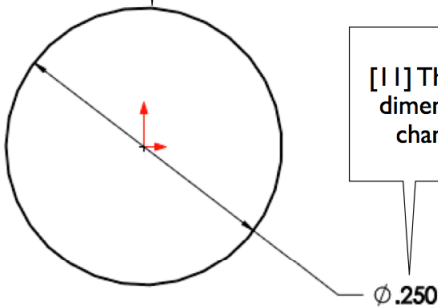
I.I-4 Draw a Circle



[5] Color Codes of Sketch Entities

A sketch entity is blue-colored (either light-blue or dark-blue) when it is not yet well-defined [4]. A well-defined entity (i.e., fixed in the space) becomes black (e.g., [7], next page). When over-defined, an entity becomes red.

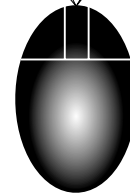
[7] Click the circle and move lower-rightward to create a diameter; type 0.25 (in) for the diameter. The circle now turns black (fixed). Use mouse functions to zoom in/out [8] or pan the sketch [9]. Drag the dimension to a location like this. Finally, press **ESC** to dismiss the **Smart Dimension**.



[11] The font size of the dimension text can be changed (see [12]).

[8] Scrolling the **Mouse Wheel** allows you to zoom in/out the sketch.

[9] Dragging the mouse with **Control-Middle-Button** allows you to pan the sketch.



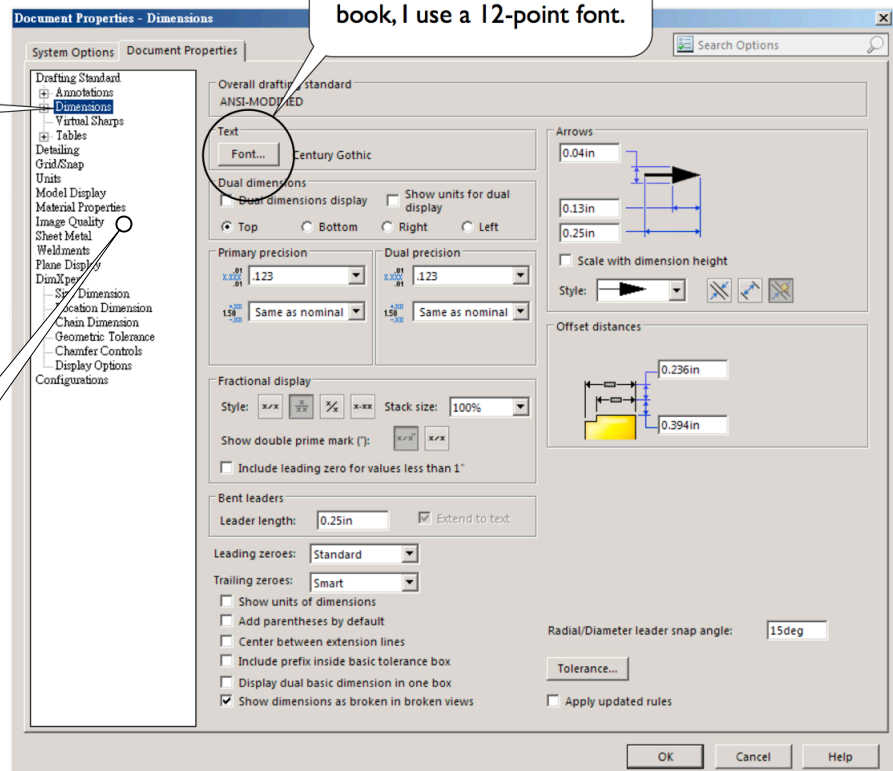
[10] If you made a mistake, you always can **Undo** the mistake.



[12] To change the font size of dimension texts, select **Dimension** in the **Document Properties** (I.I-3[1, 2], page 5).

[13] Click **Font...** and then select a font size. In this book, I use a 12-point font.

[14] **Image Quality** can be used to improve the smoothness of the model. #

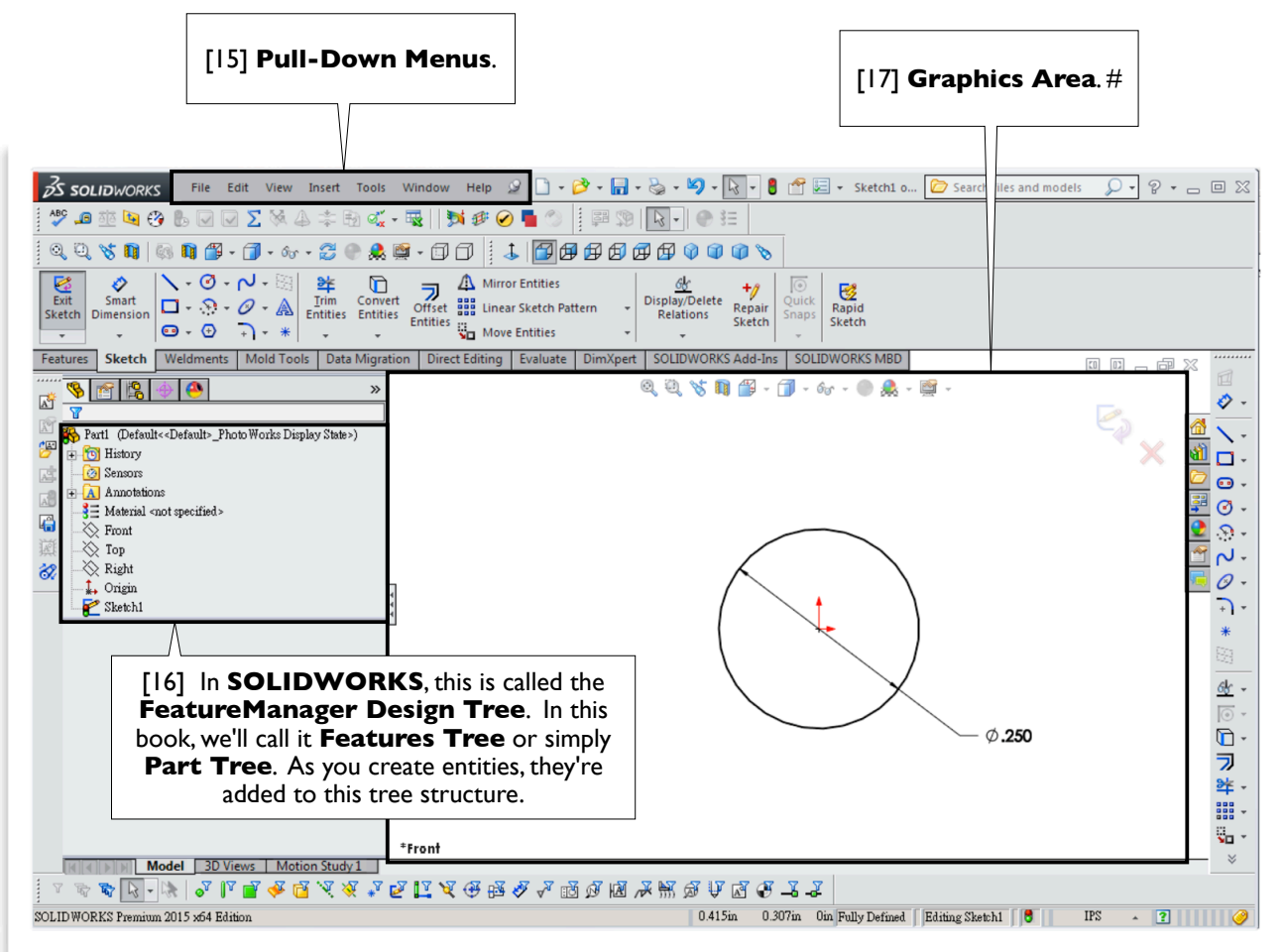


SOLIDWORKS Commands

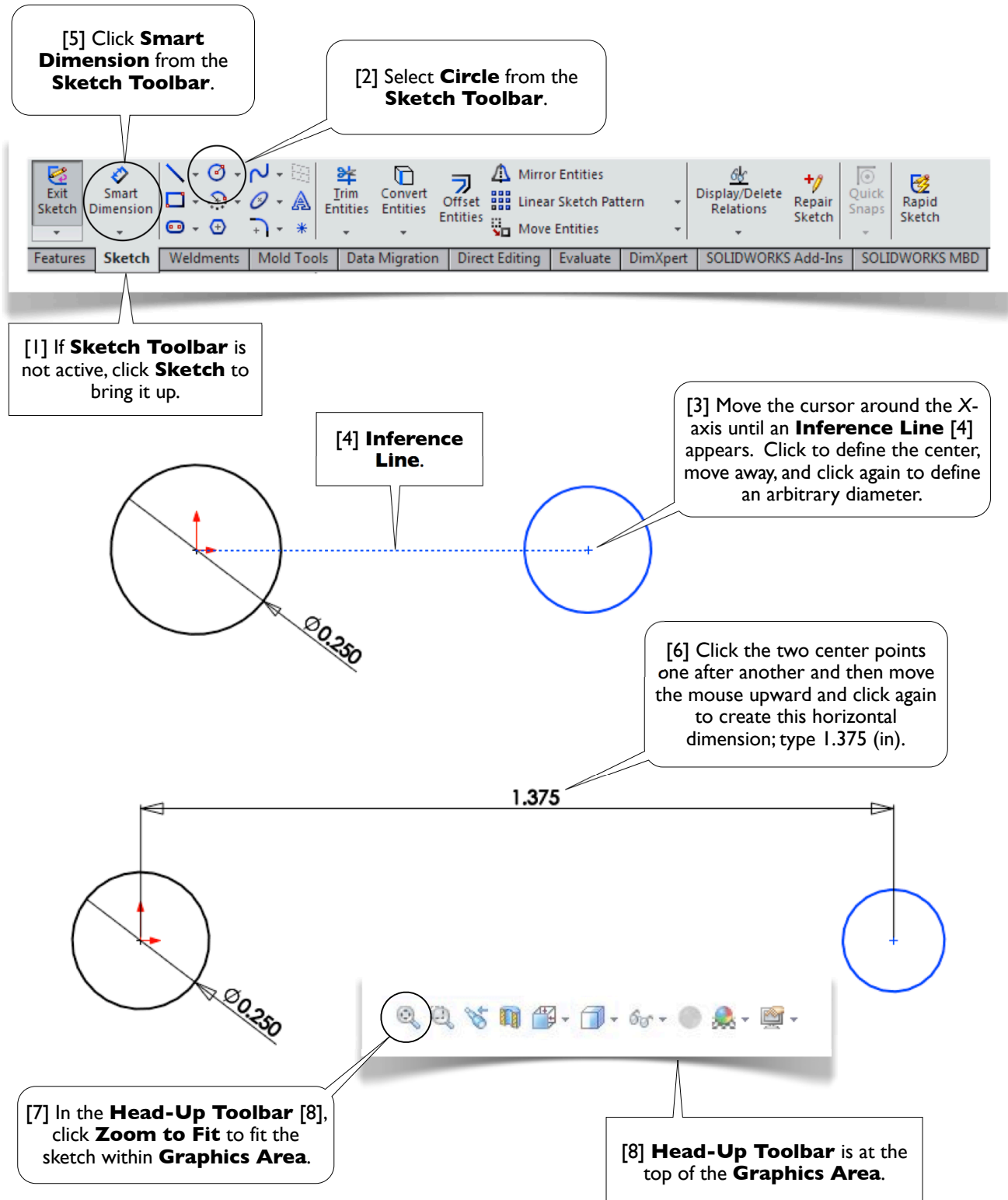
There are literally hundreds of **SOLIDWORKS** commands (tools). All commands can be found in the **Pull-Down Menus** [15]. Nevertheless, the most intuitive way to issue a command is through a context-sensitive menu, or simply called **Context Menu** [1, 3, 6] (page 6). To issue a command with a **Context Menu**, you right-click an object on either the **Part Tree** [16] or the **Graphics Area** [17]. The commands available in a **Context Menu** depend on the kind of object you're working on (that's why it is called a context-sensitive menu). In step [1] (page 6), the object you were working on is the **Front** plane; in steps [3, 6] (page 6), the object you were working on is the **Graphics Area**.

After you accumulate some experiences, you may find that a more convenient way to issue a command is simply clicking a command on a **Toolbar** (e.g., [10], last page). In this book, we roughly follow these rules to issue a command:

1. As novices, we issue a command through a **Context Menu**, because it is the most intuitive way.
2. If a command is not available with a **Context Menu**, we select it from the **Pull-Down Menus**, because it is the most comprehensive way (i.e., all commands can be found there).
3. As we accumulate experiences, we begin to issue a command by clicking a button in a **Toolbar**, because it is the most convenient way.



I.I-5 Draw Another Circle



[16] Select **Add Relation** from the **Context Menu** again.

[9] Press **ESC** to dismiss **Smart Dimension**. Select **Add Relation** from the **Context Menu** (you may need to expand the **Context Menu**).

[20] Click **OK** to dismiss the **Property Box**.

[17] In the **Graphics Area**, click the two circles' centers.

[19] A **Horizontal** relation is added, in addition to the existing **Distance** relation.

[18] Click **Horizontal** to make the two points align horizontally.

[14] Click **OK** to dismiss the **Property Box**. The **Part Tree** re-appears.

[10] A **Property Box** appears in place of the **Part Tree**.

[11] In the **Graphics Area**, click the two circles one after the other. Note that their names appear here.

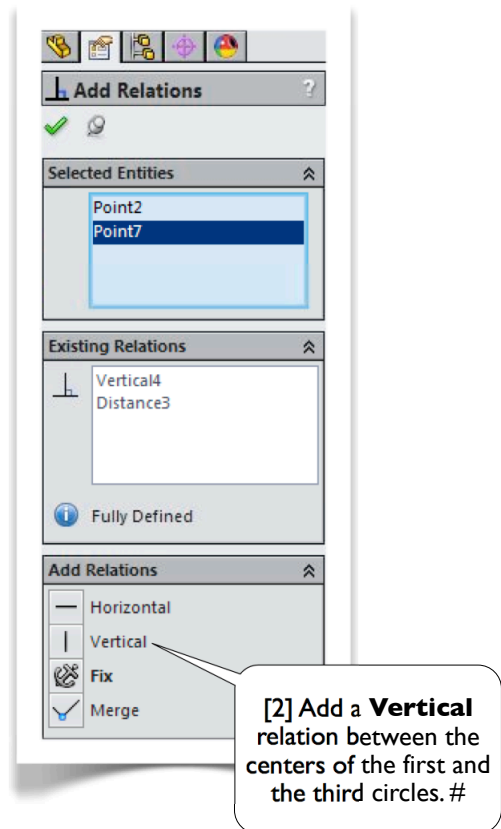
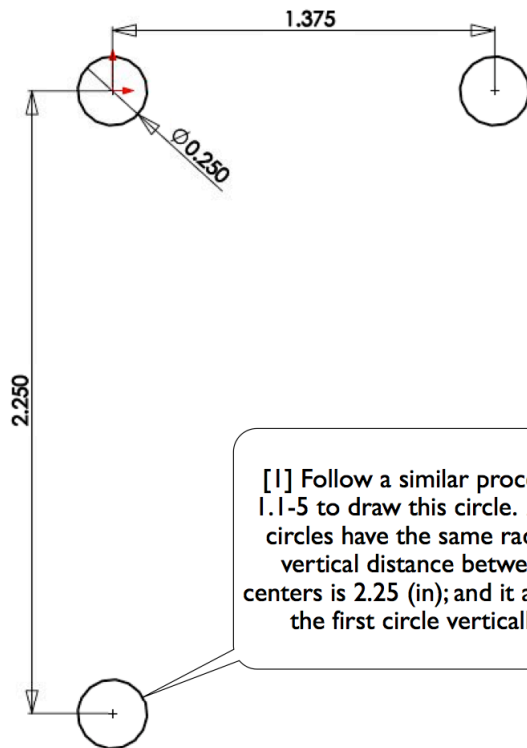
[13] A relation between the two entities is added.

[12] Click **Equal** to make their sizes equal.

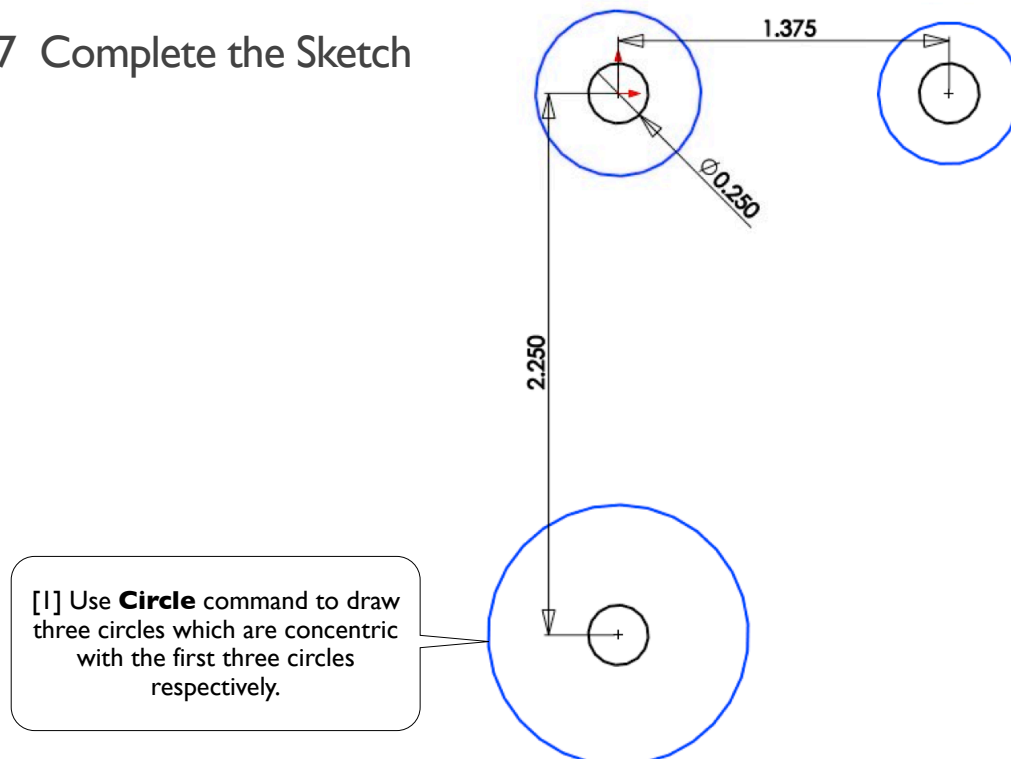
[15] De-select the two circles (press **ESC** or click anywhere on the **Graphics Area**). Now, the two circles have the same radius. The second circle is still blue-colored, meaning that it is not well-defined yet. We now impose another relation.

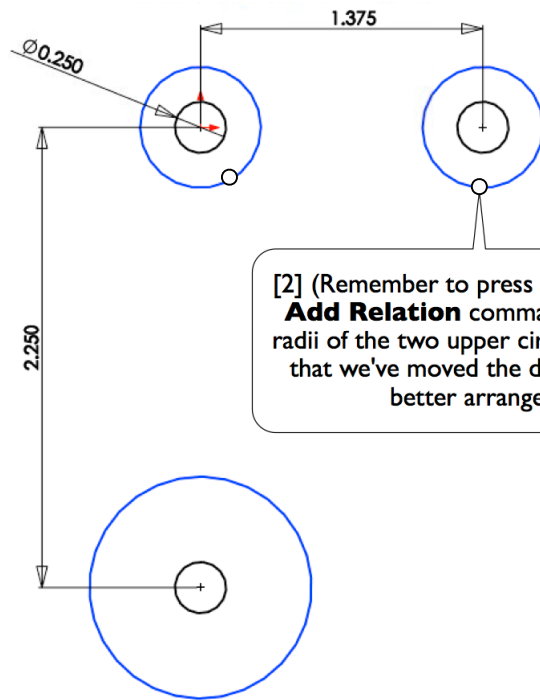
[21] Press **ESC** to de-select the two points. Now, the second circle becomes black (fixed) too. #

I.I-6 Draw the Third Circle

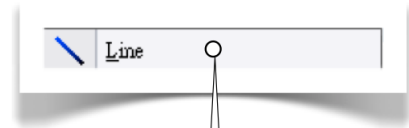


I.I-7 Complete the Sketch



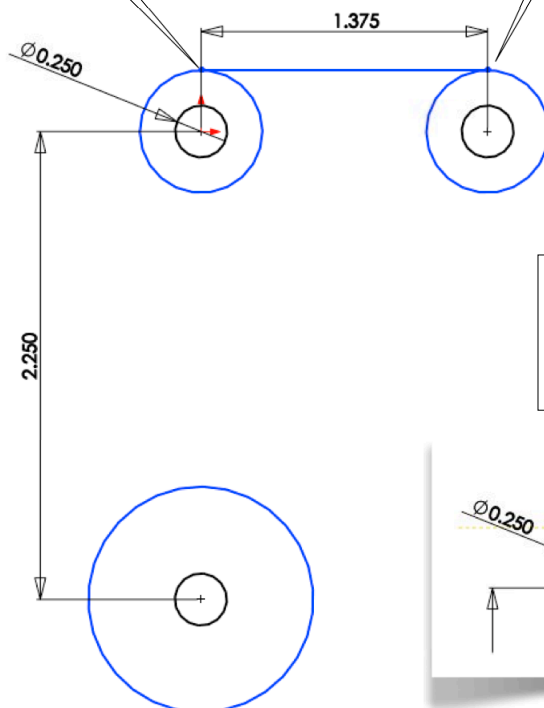


[2] (Remember to press **ESC** twice.) Use **Add Relation** command to make the radii of the two upper circles equal. Note that we've moved the dimensions for a better arrangement.



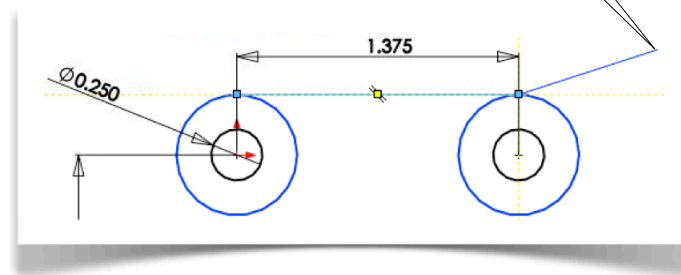
[3] Select **Sketch Entities>Line** command from the **Context Menu**. Before right-clicking to pop-up the **Context Menu**, make sure no command is active (if so, press **ESC** to dismiss it) and no sketch entity is selected (if so, press **ESC** to de-select it).

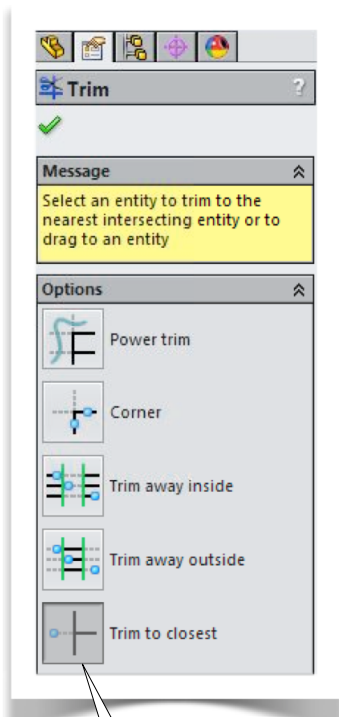
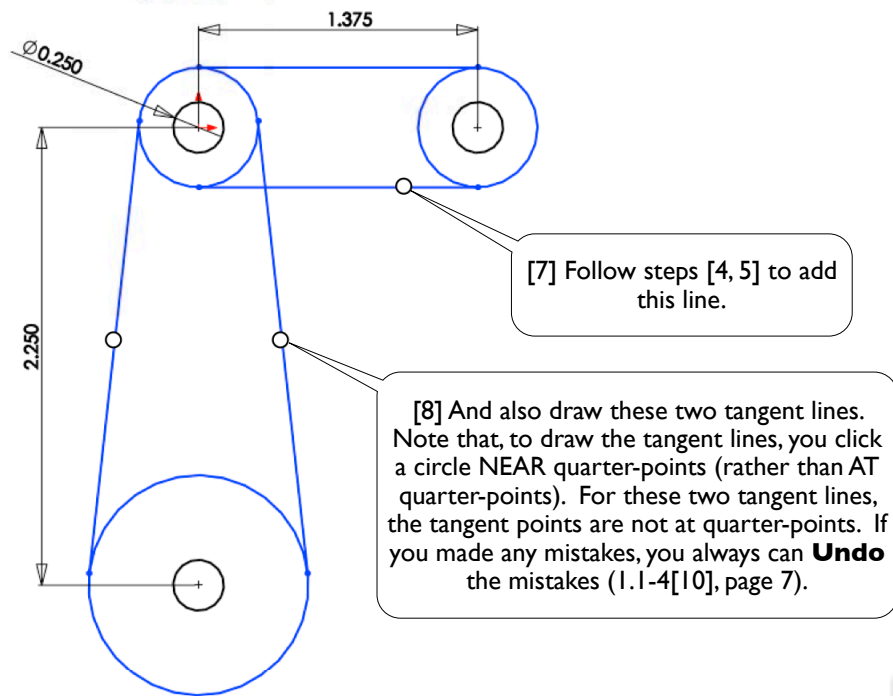
[4] Click the upper quarter-point of this circle...



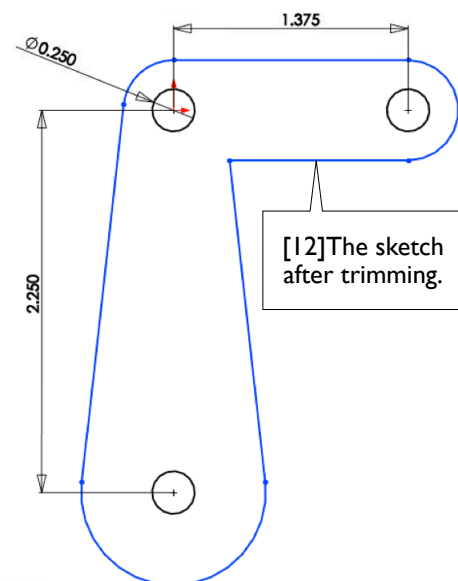
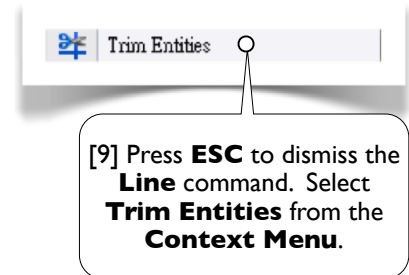
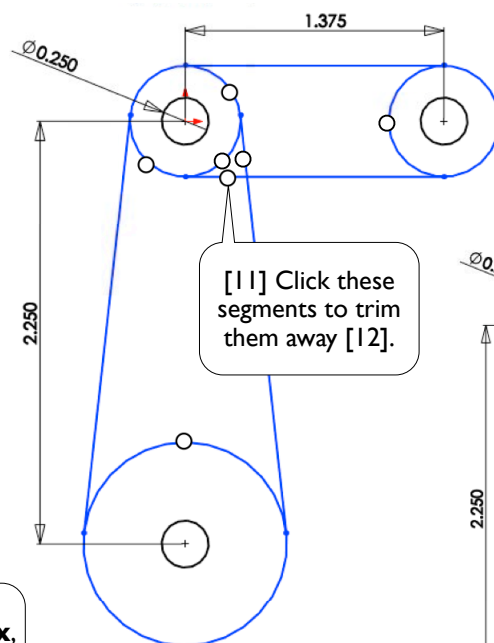
[5] And click the upper quarter-point of this circle. Double-click to end the line drawing [6].

[6] The **Line** command can be used to draw multiple line segments. To end a session of line-drawing without dismiss the **Line** command, simply double-click.





[10] In the **Property Box**, select **Trim to closest**.



[14] The radius of this arc is automatically adjusted to agree with the relation [2] (page 12).

[16] All sketch entities are fixed (black-colored) now.

[13] Use **Smart Dimension** (1.1-5[5], page 9) to specify this radius (0.313 in).

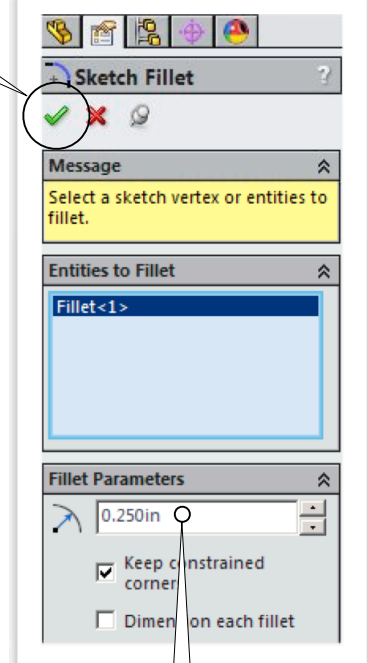
[18] Select this vertex.

[15] And specify this radius (0.5 in).



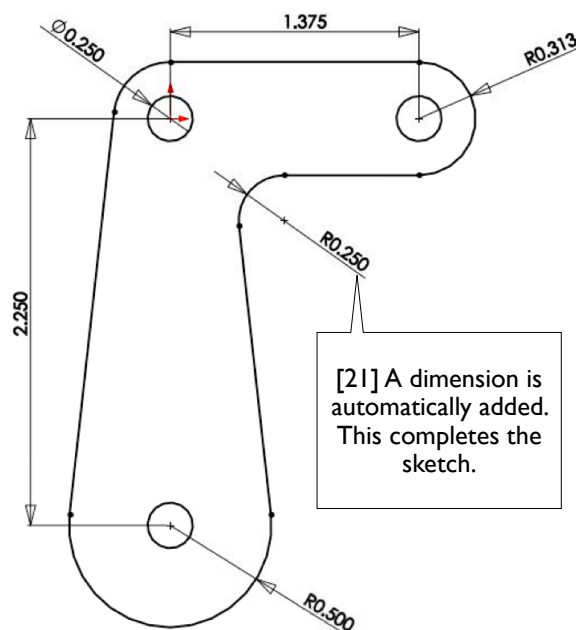
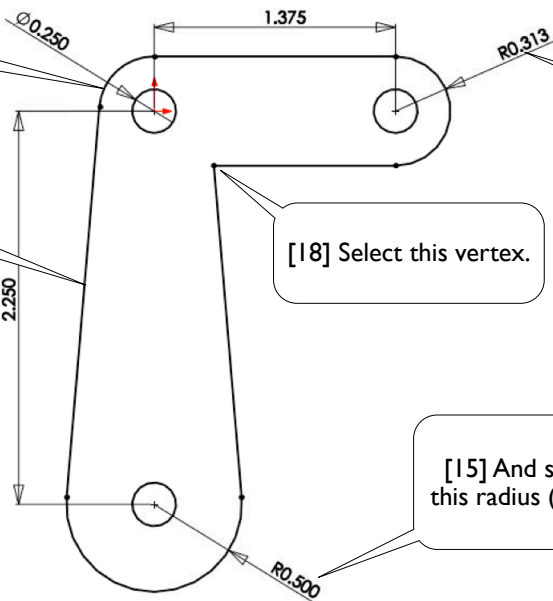
[17] Press **ESC** to dismiss **Smart Dimension**. Select **Sketch Fillet** from the **Context Menu**.

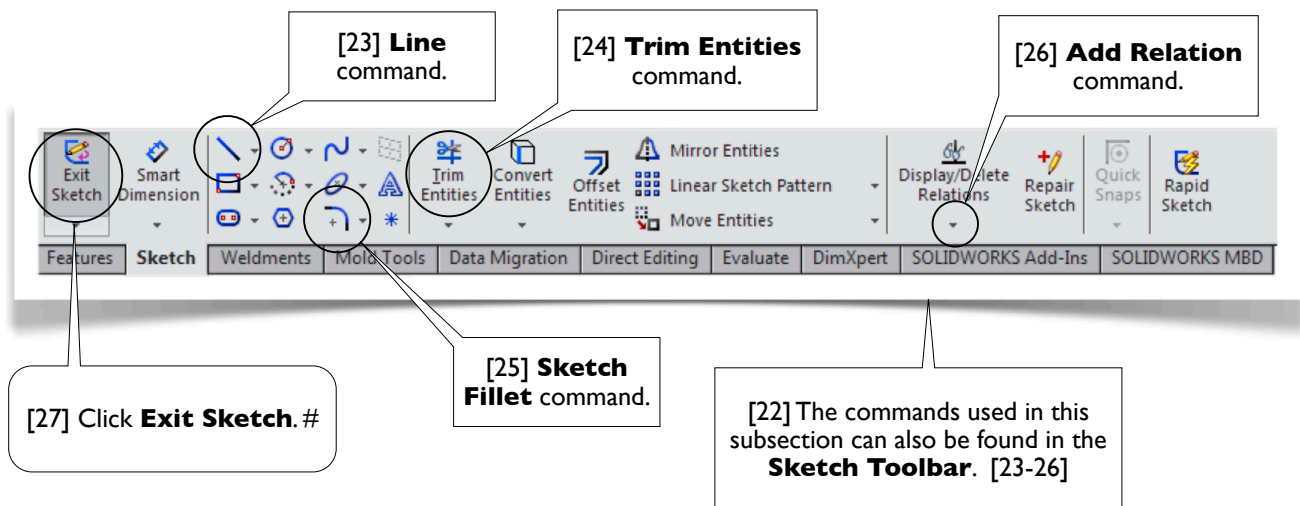
[20] Click **OK** to accept the properties.



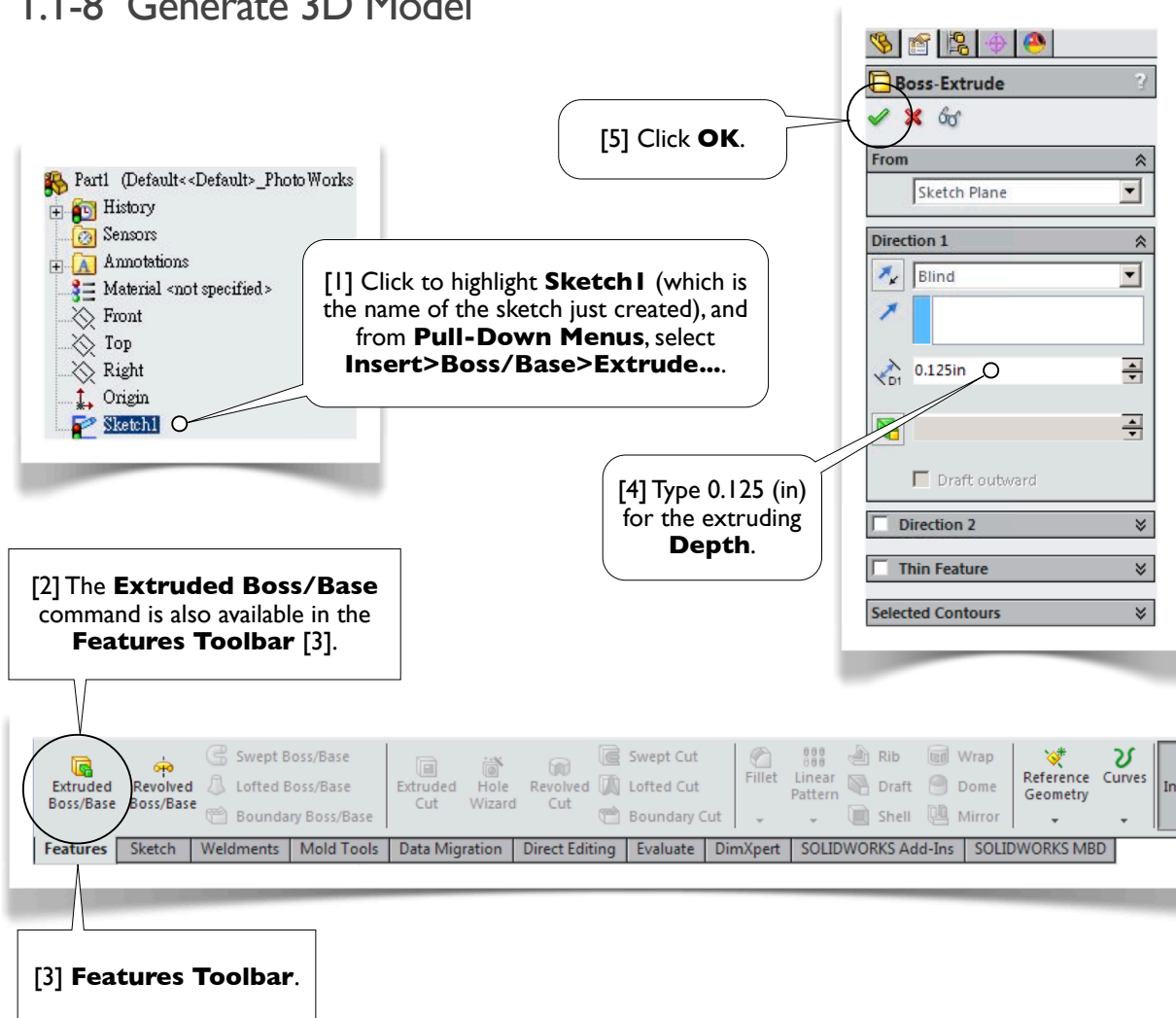
[21] A dimension is automatically added. This completes the sketch.

[19] Type 0.25 (in) for the **Fillet Radius**.

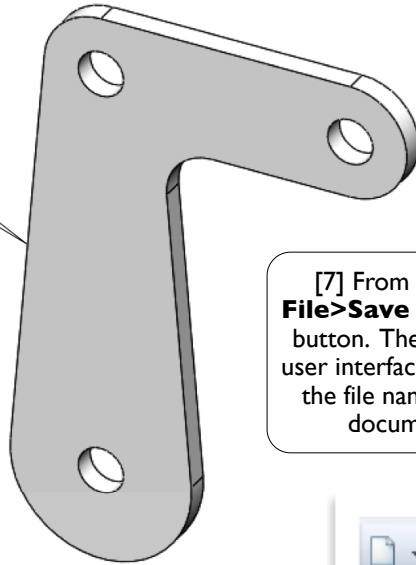




1.1-8 Generate 3D Model



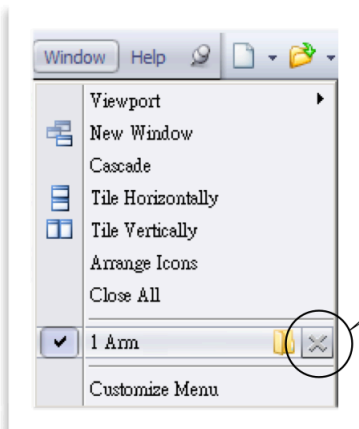
[6] The finished 3D model.



[7] From **Pull-Down Menu**, select **File>Save** or, on the **Toolbar**, click **Save** button. The **Toolbar** is on the top of the user interface. Save this part document with the file name **Arm**. The full name of the document is **Arm.SLDPRT.#**



I.I-9 Wrap Up

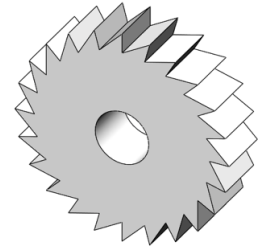


[1] Select **File>Close** from the **Pull-Down Menu** to close the part document. Or, you may select **Window** menu and click here.

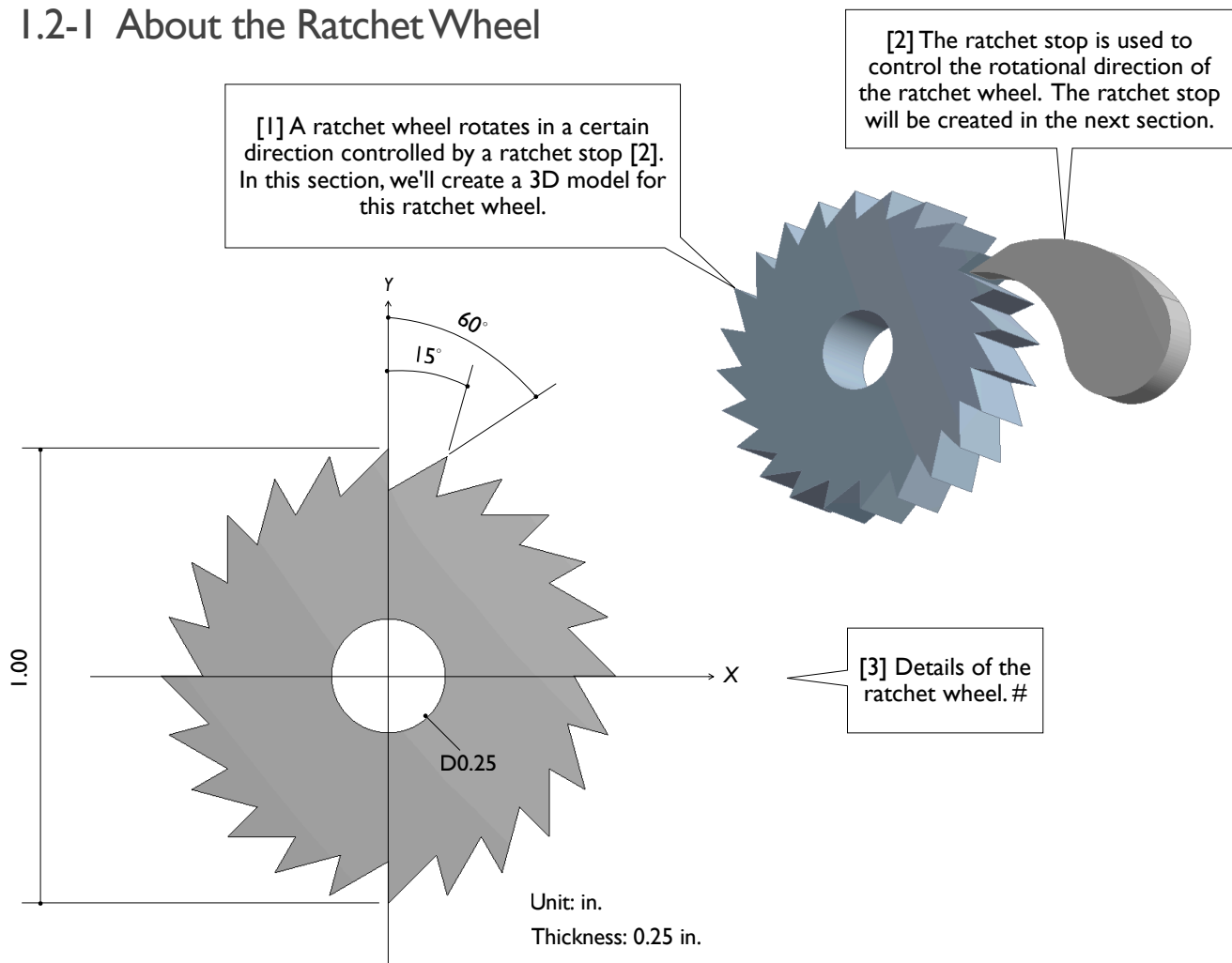
[2] Select **File>Exit** from **Pull-Down Menu** to quit **SOLIDWORKS.#**

Section 1.2

Ratchet Wheel



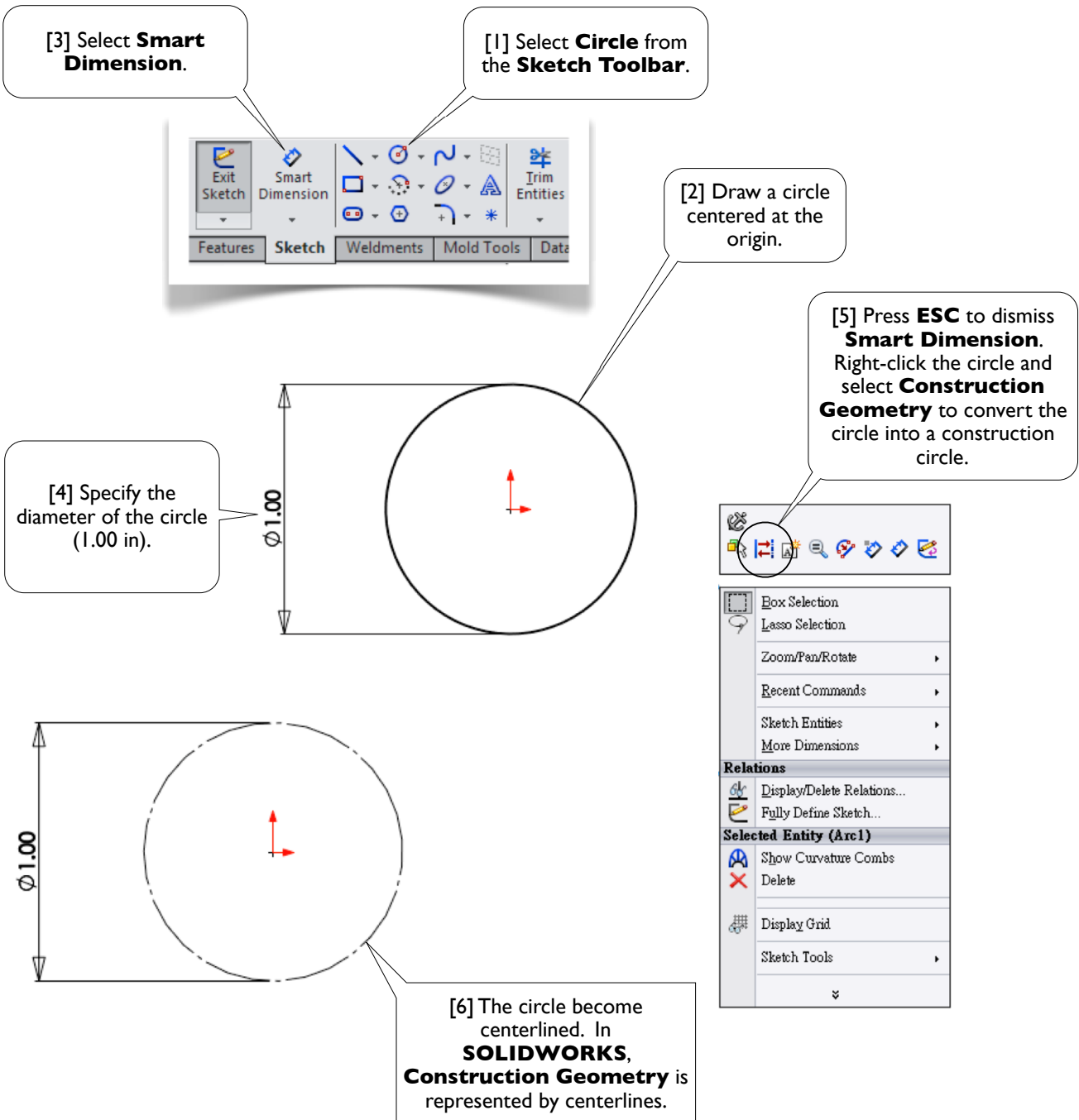
1.2-1 About the Ratchet Wheel



1.2-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part (1.1-2[1, 4-6], page 4). Set up **IPS** unit system with 2 decimal places for the length unit (1.1-3, page 5). Start a sketch on **Front** plane (1.1-4[1, 2], page 6). #

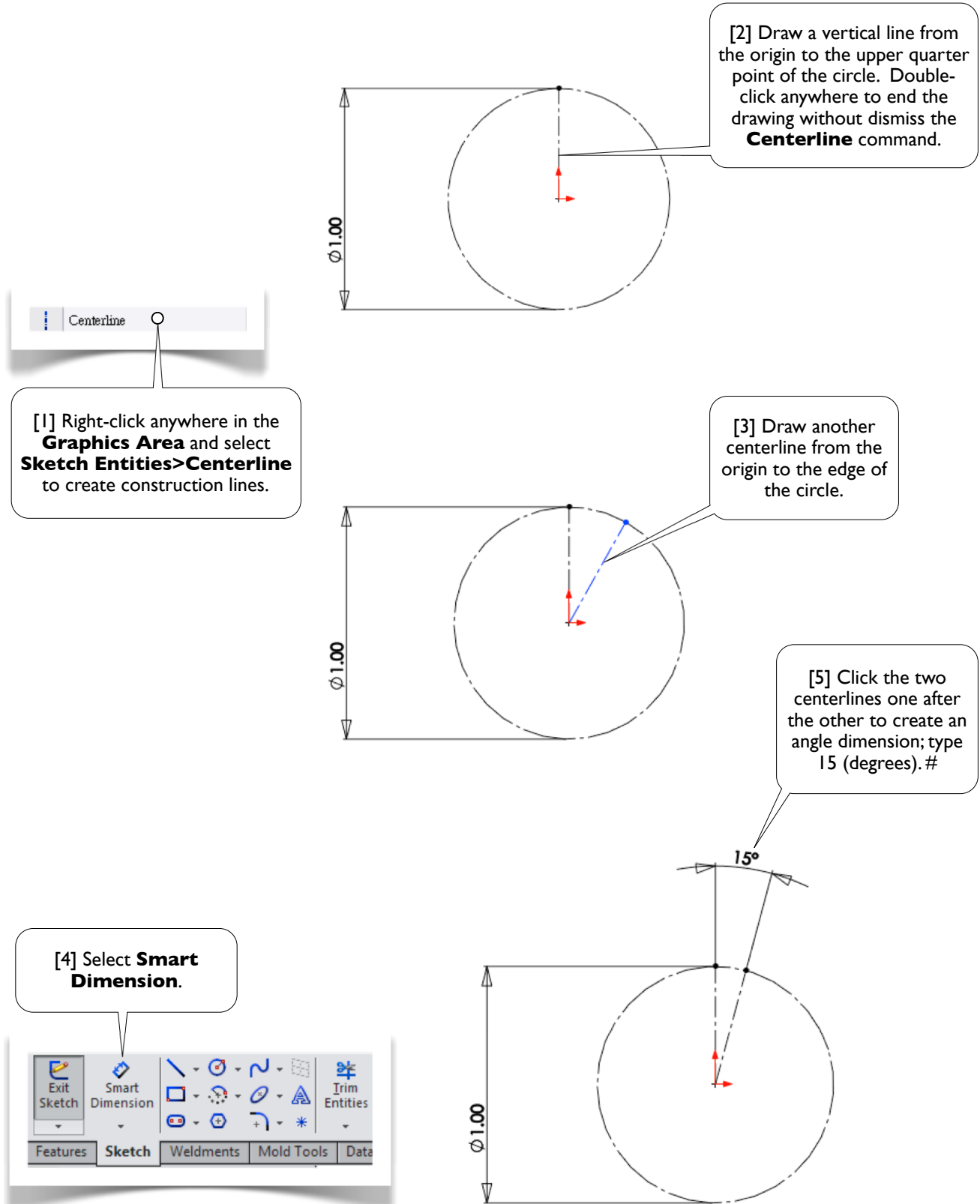
1.2-3 Draw a Construction Circle



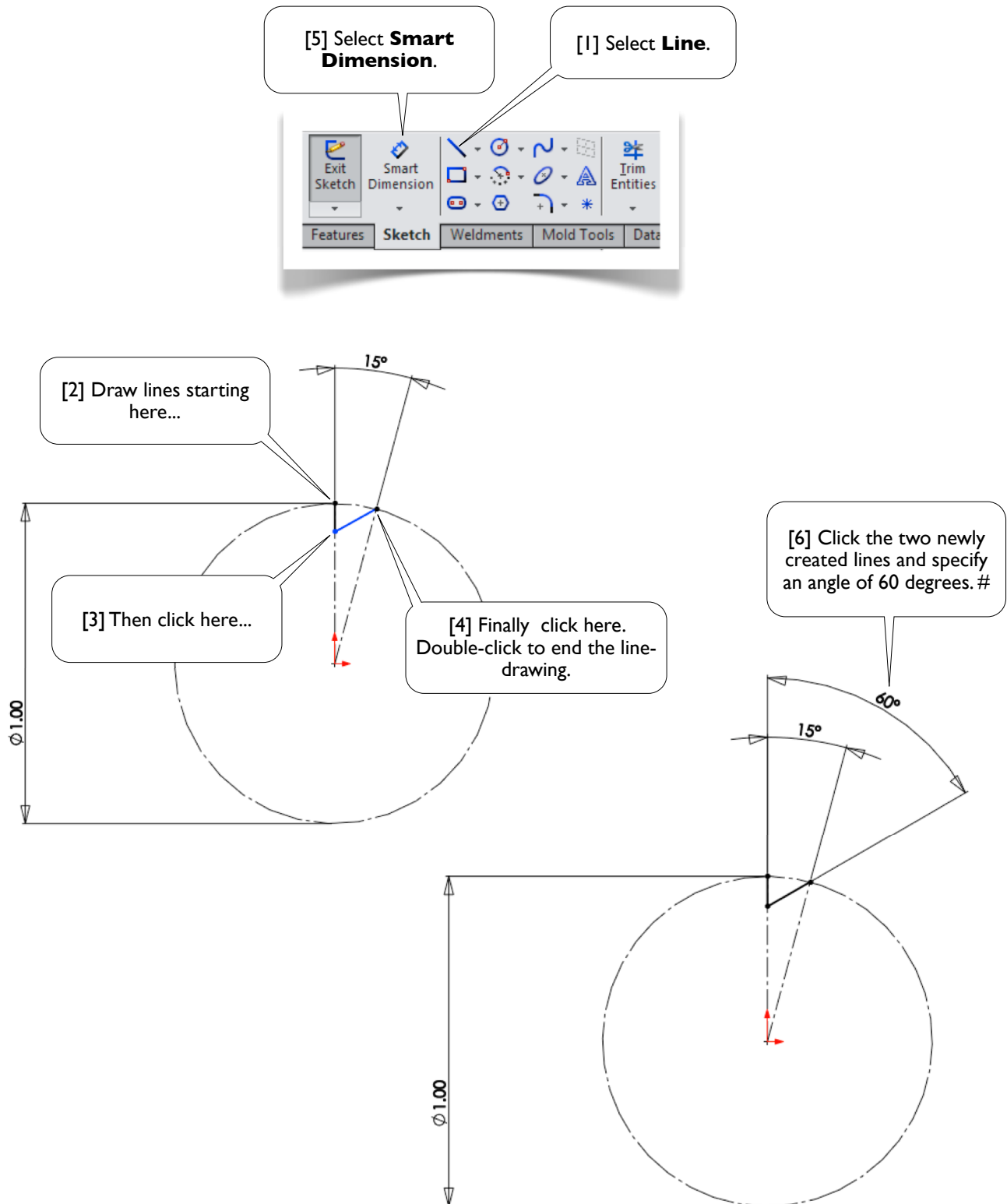
[7] **Construction Geometry**

Frequently used **Construction Geometries** include construction lines and construction circles. A construction line can be finite length or infinite length. A **Construction Geometry** is used for reference only, it is not a geometric entity. #

1.2-4 Draw Construction Lines



1.2-5 Draw a Tooth



1.2-6 Duplicate the Tooth

[4] Click **OK**.

[1] From **Pull-Down Menus**, select **Tools>Sketch Tools>Circular Pattern**. And select the centerlined circle (to define the pattern direction).

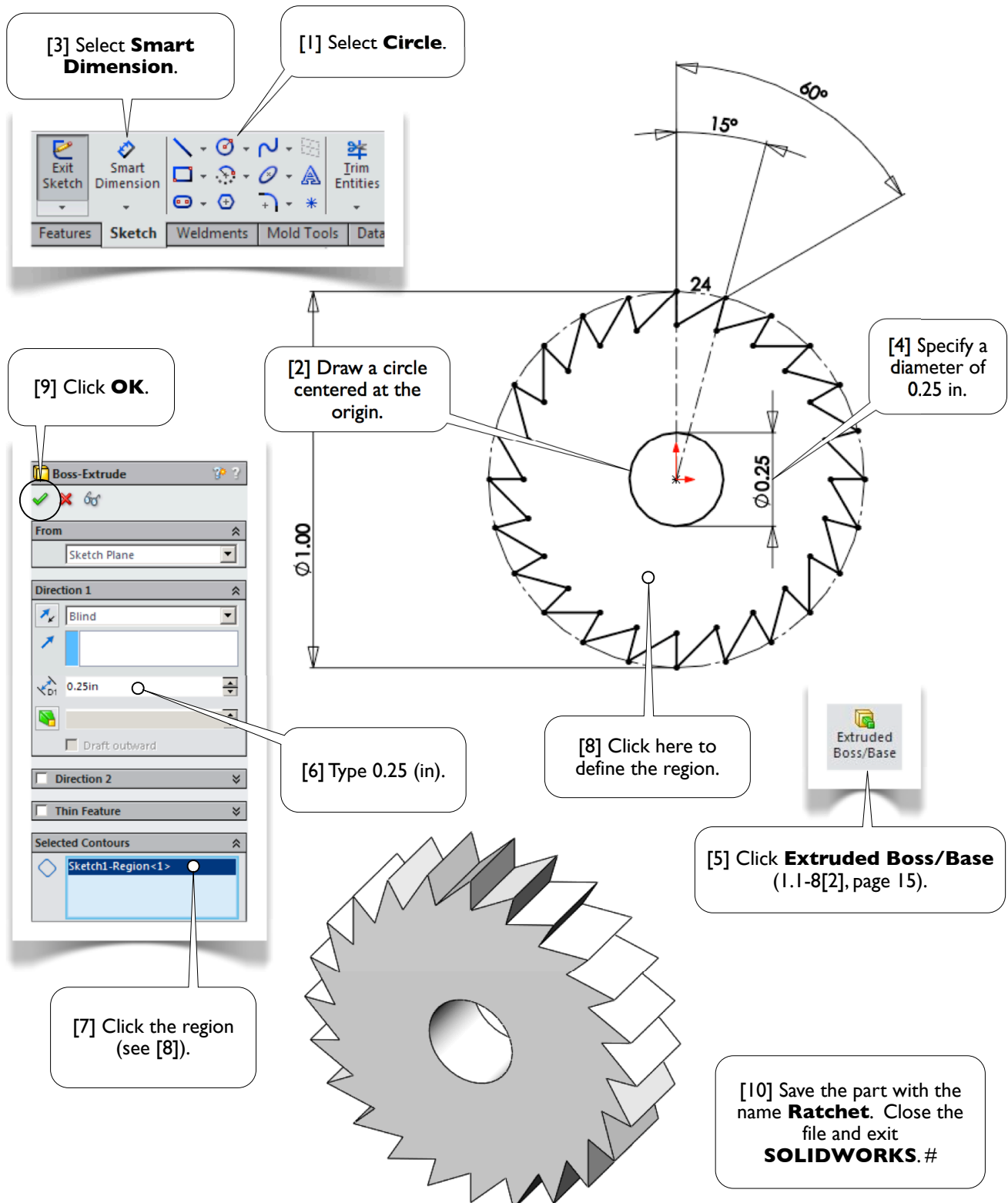
[2] Type 24 for **Number of Instances**.

[3] Right-click this box and select **Clear Selections** from the **Context Menu** and then select the two line segments created in 1.2-5 (last page) for **Entities to Pattern**.

[5] The **Circular Sketch Pattern** command is also available by clicking the arrow next to **Linear Sketch Pattern**. #

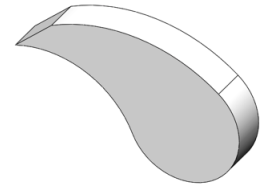


1.2-7 Finished the Sketch and Generate 3D Model

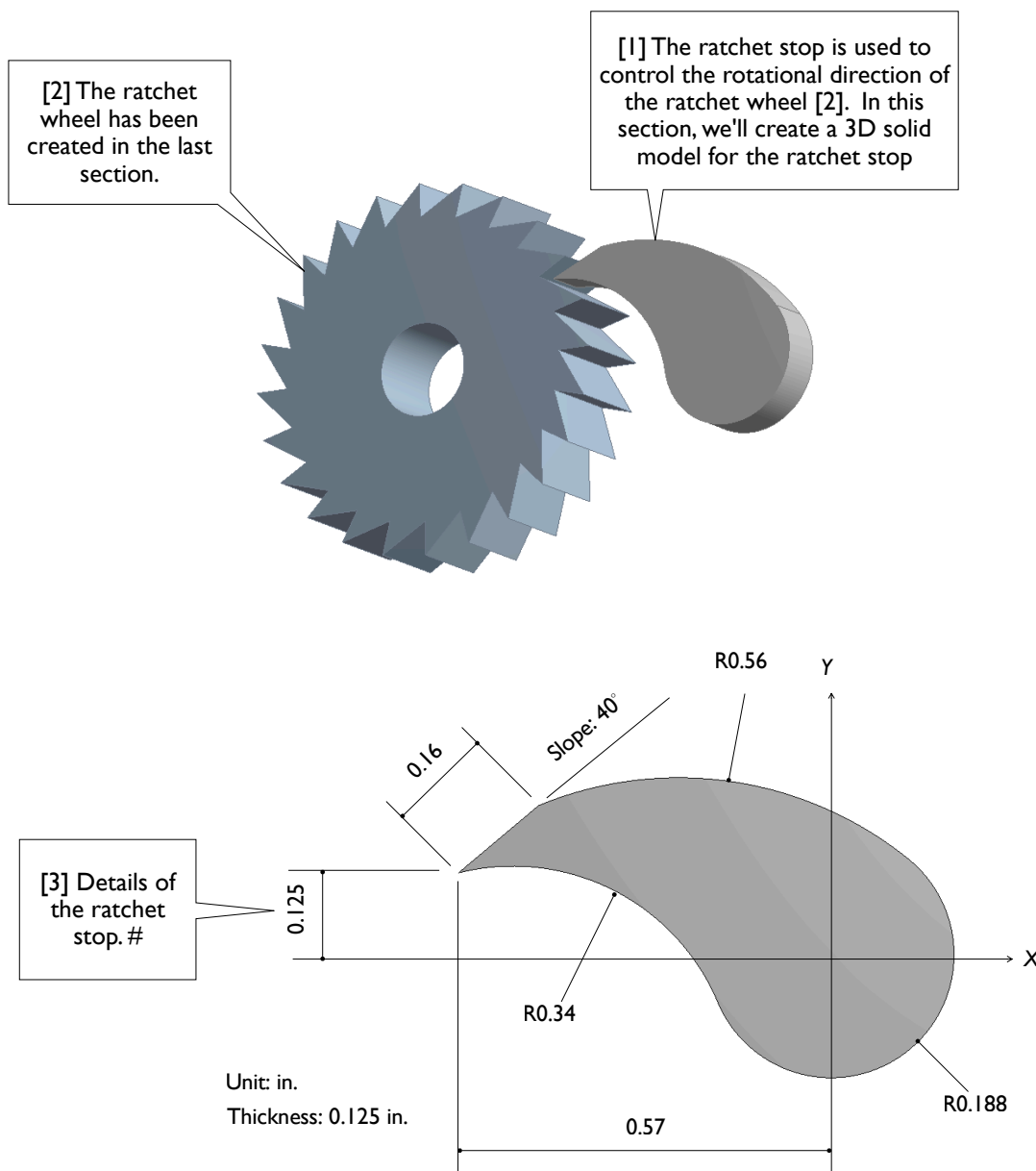


Section 1.3

Ratchet Stop



1.3-1 About the **Ratchet Stop**

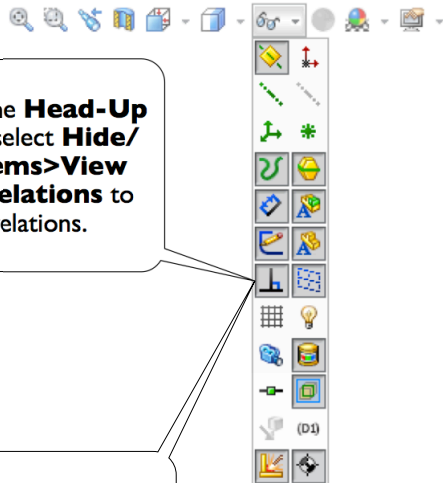


[7] Then define another end point. Double-click to end the drawing without dismiss **Tangent Arc** command.

[6] Click this end point of the existing arc...

[9] Then define another end. Press **ESC** to dismiss the **Tangent Arc**.

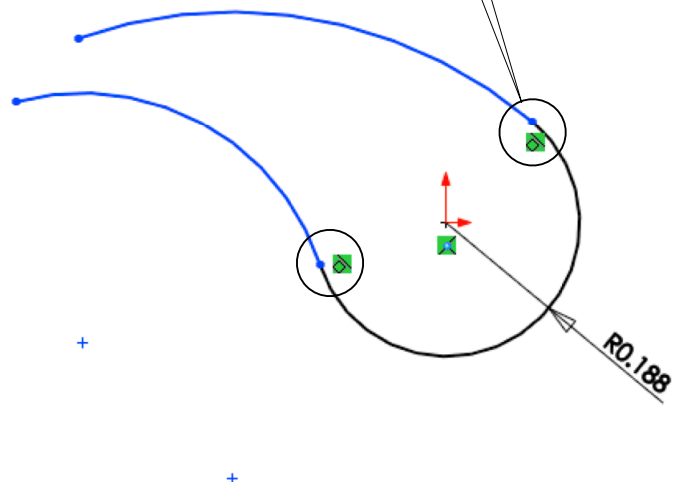
[8] Click another end point of the first arc...

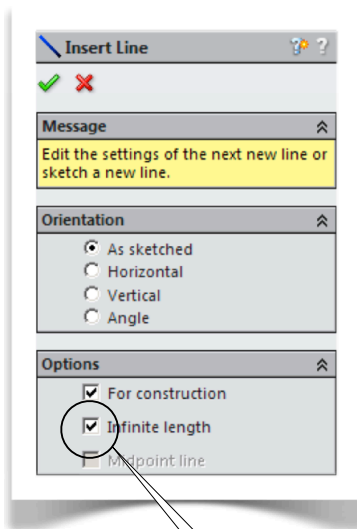


[10] From the **Head-Up Toolbar**, select **Hide/Show Items>View Sketch Relations** to show relations.

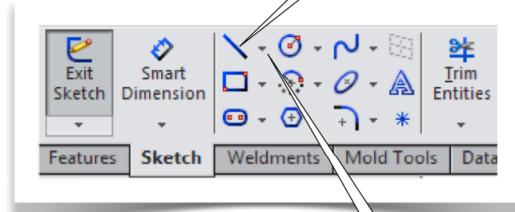
[12] Select **View Sketch Relations** again to hide the relations.

[11] A **Tangent** symbol appears next to each tangent point.





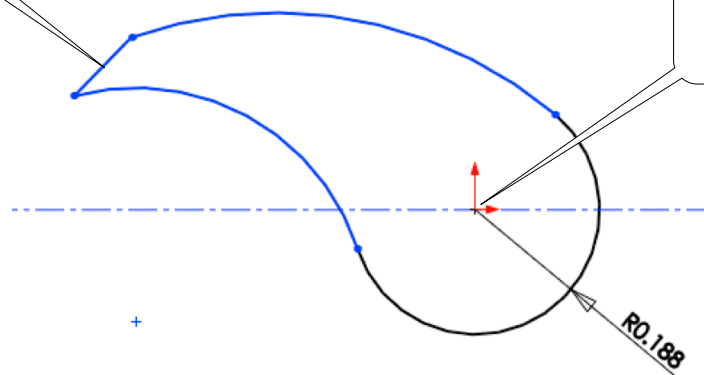
[16] In the **Property Box**, select **Infinite length**.



[13] Select **Line** command.

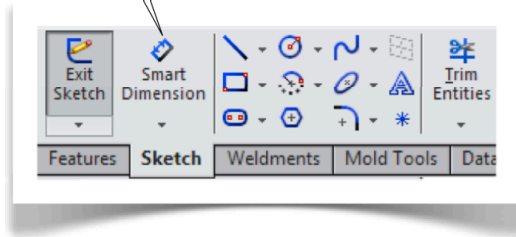
[15] Click the small triangle next to **Line** command and select **Centerline**.

[14] Draw this line.

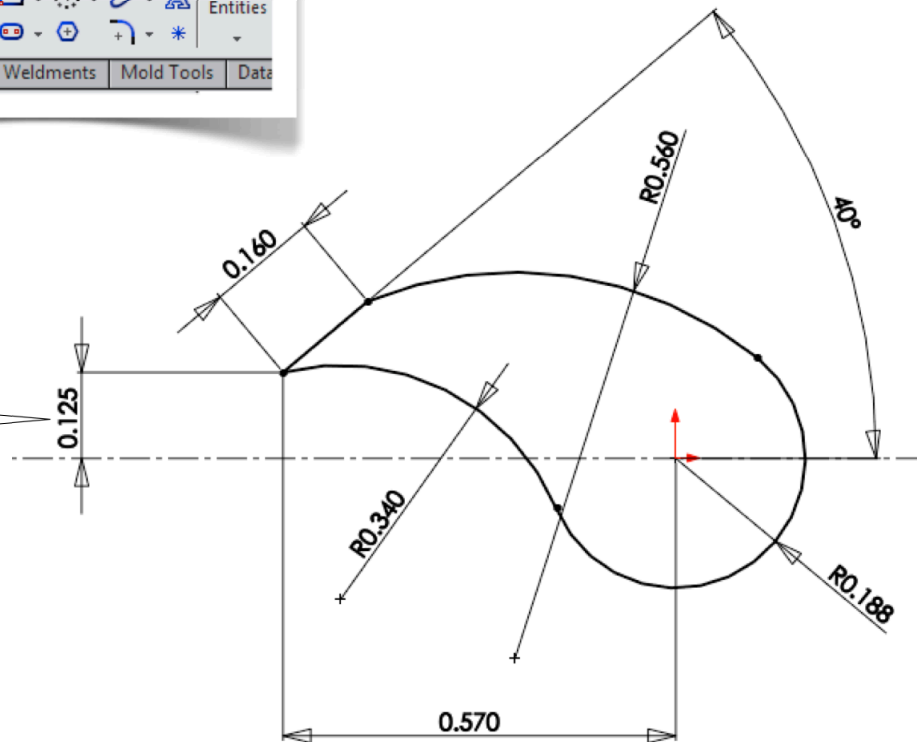


[17] Click the origin and then click any horizontal point to create a construction line of infinite length.

[18] Select **Smart Dimension**.

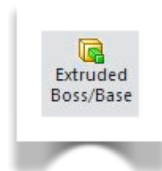
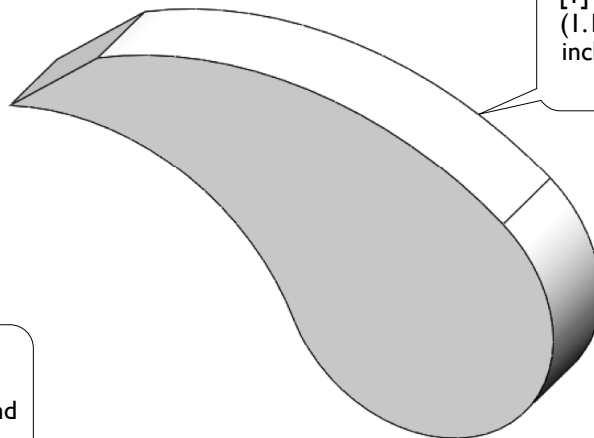


[19] Finish up the sketch by specify the rest of the dimensions. All entities must be black-colored. #



1.3-4 Generate 3D Model

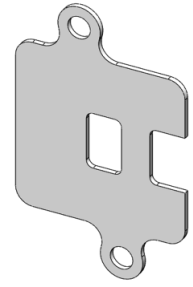
[1] **Extrude** the sketch (1.1-8[2], page 15) 0.125 inches to create this 3D model.



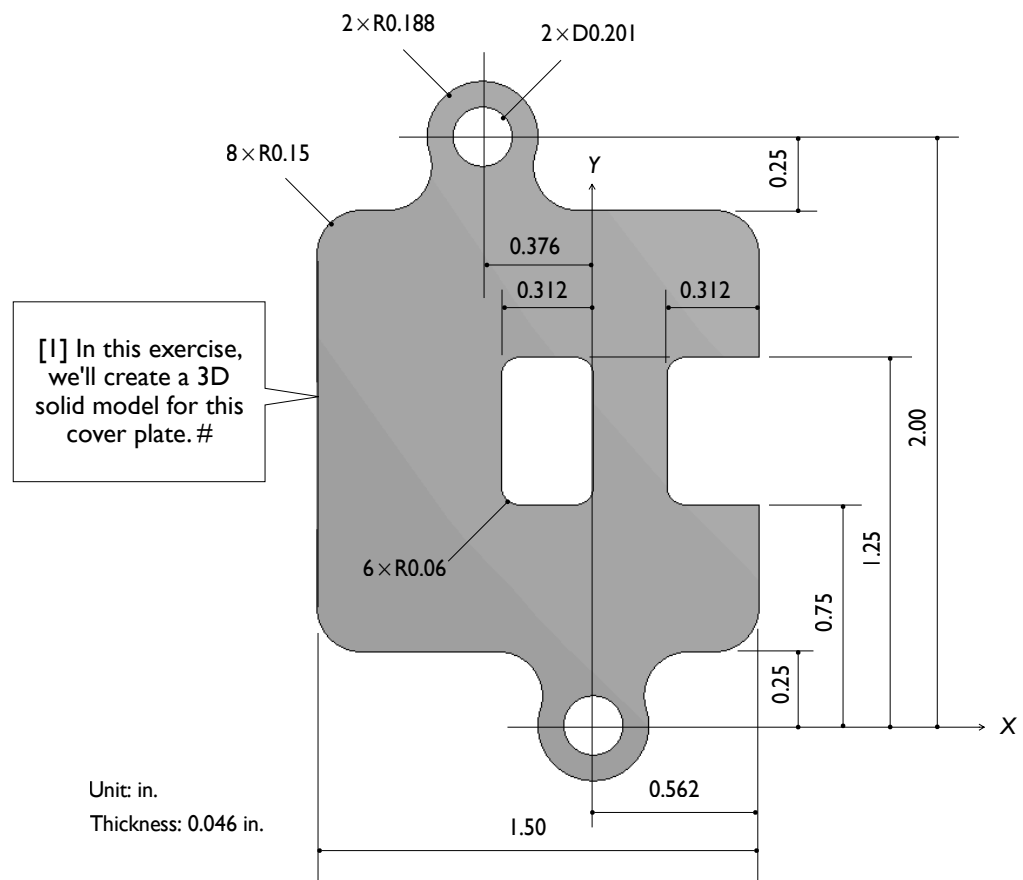
[2] Save the part with the name **Stop**. Close the file and exit **SOLIDWORKS**. #

Section 1.4

Cover Plate



1.4-1 About the Cover Plate

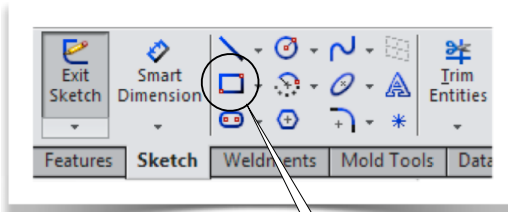


1.4-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part (1.1-2, page 4). Set up **IPS** unit system with 3 decimal places for the length unit (1.1-3, page 5). Create a sketch on **Front** plane (1.1-4[1, 2], page 6). #

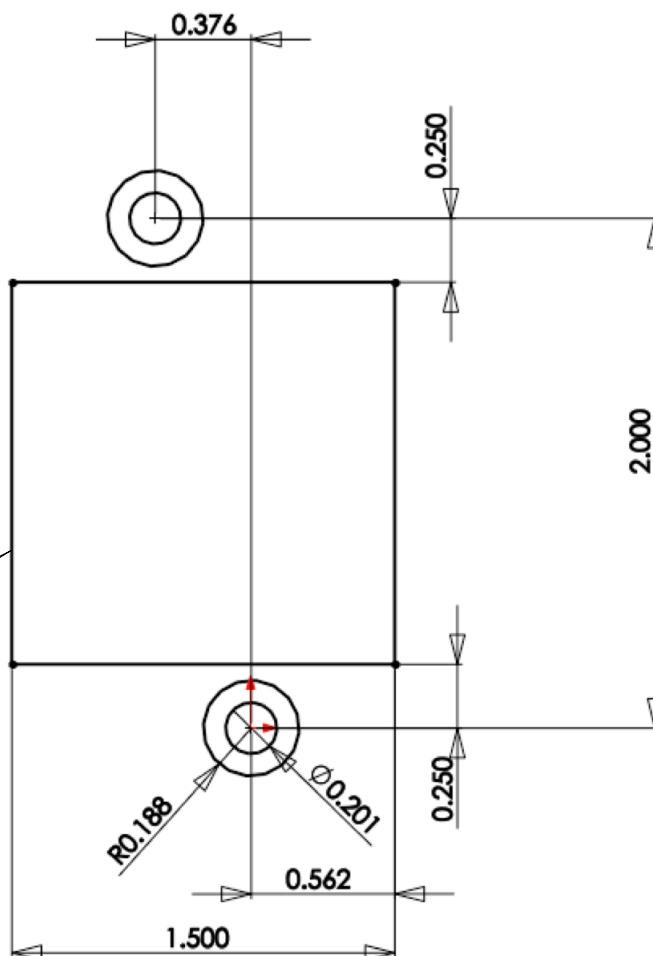


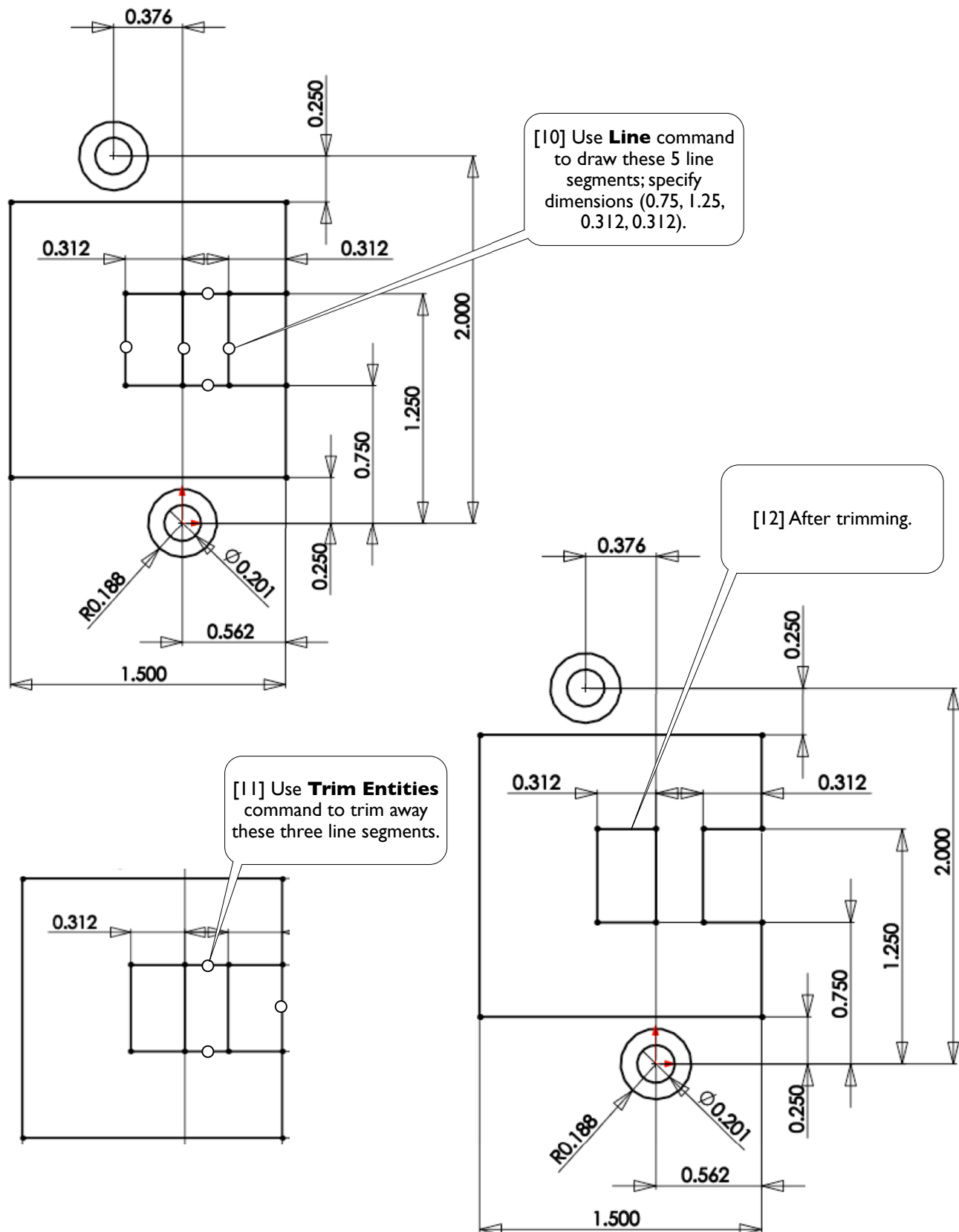
[7] Select **Sketch Entities>Corner Rectangle** from the **Context Menu**.

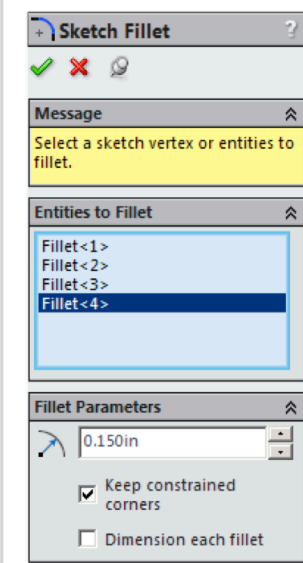
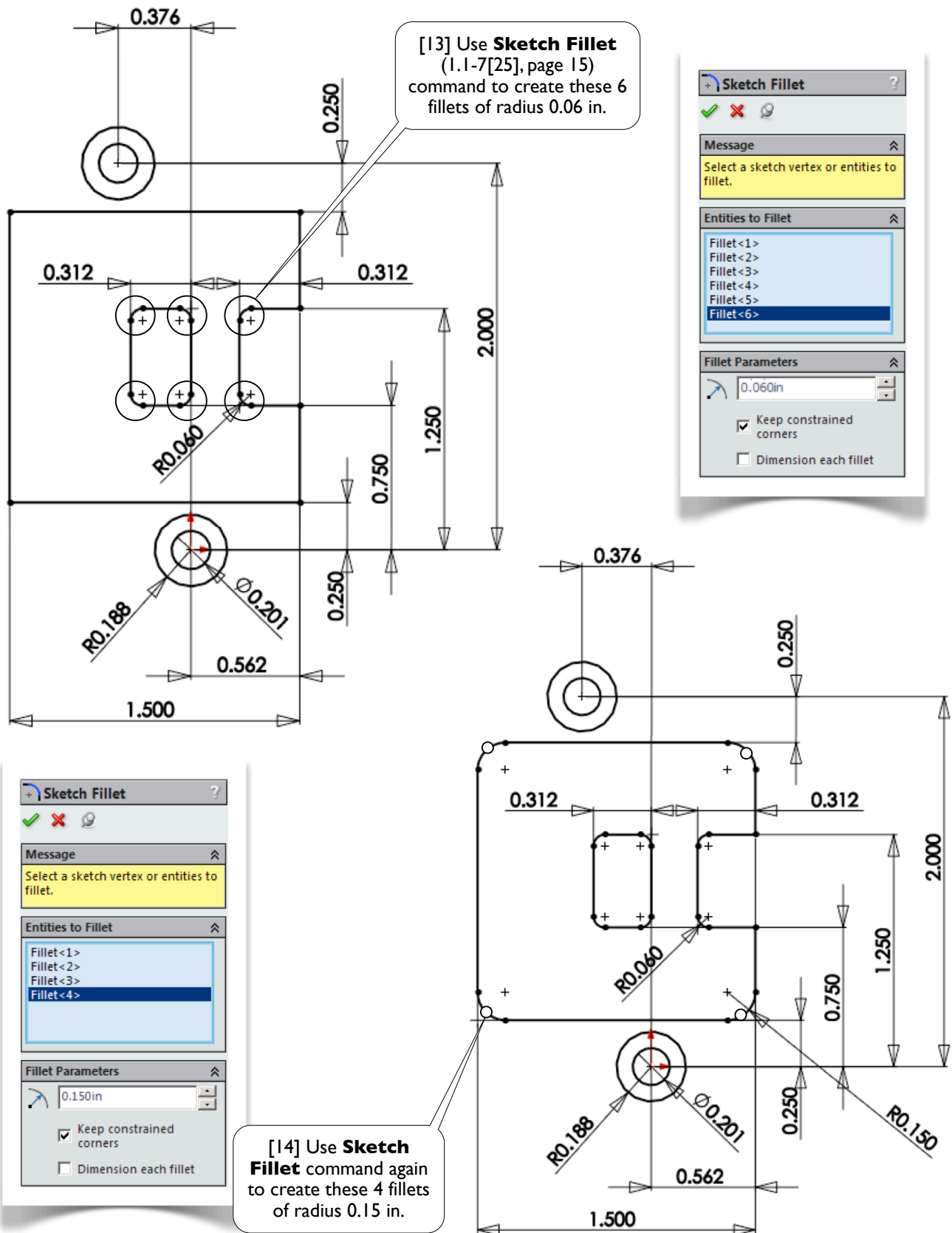


[8] The **Corner Rectangle** is also available in the **Sketch Toolbar**.

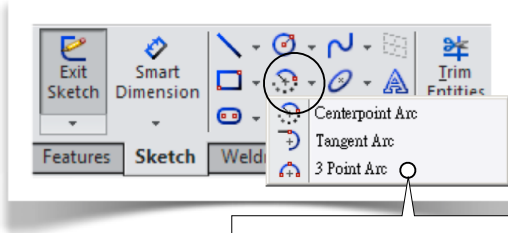
[9] Draw a rectangle and specify dimensions (1.5, 0.25, 0.25, 0.562).



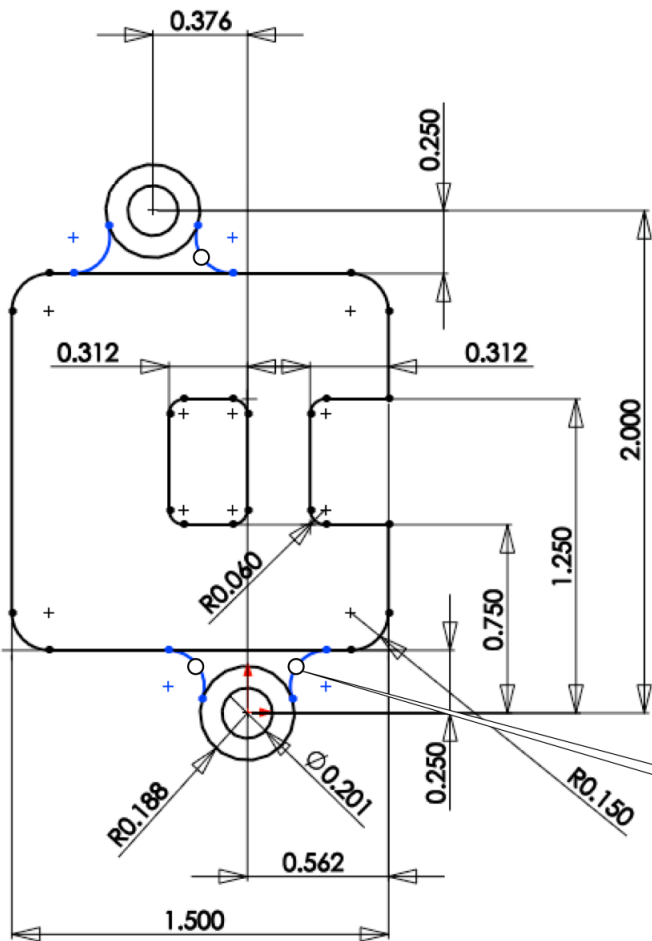
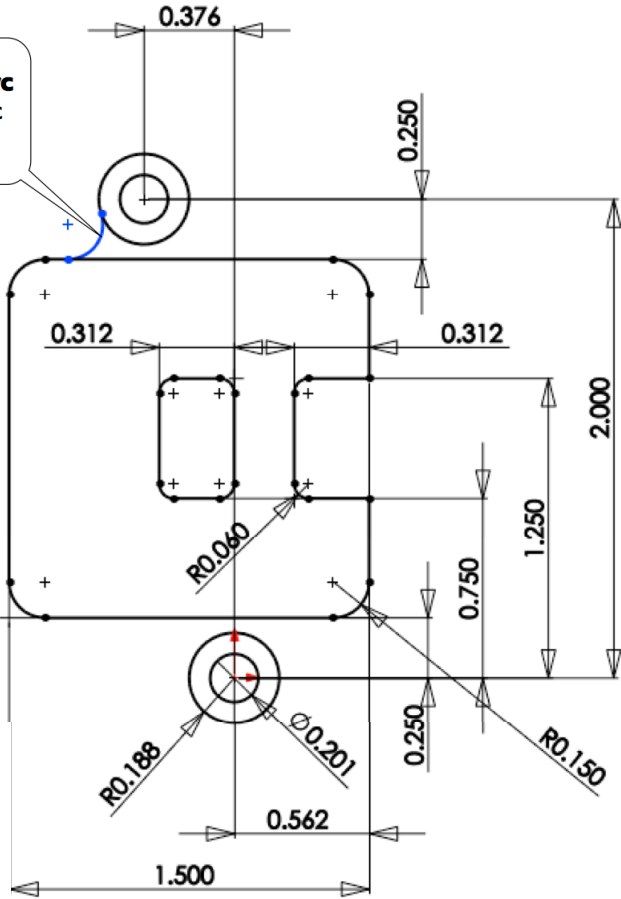




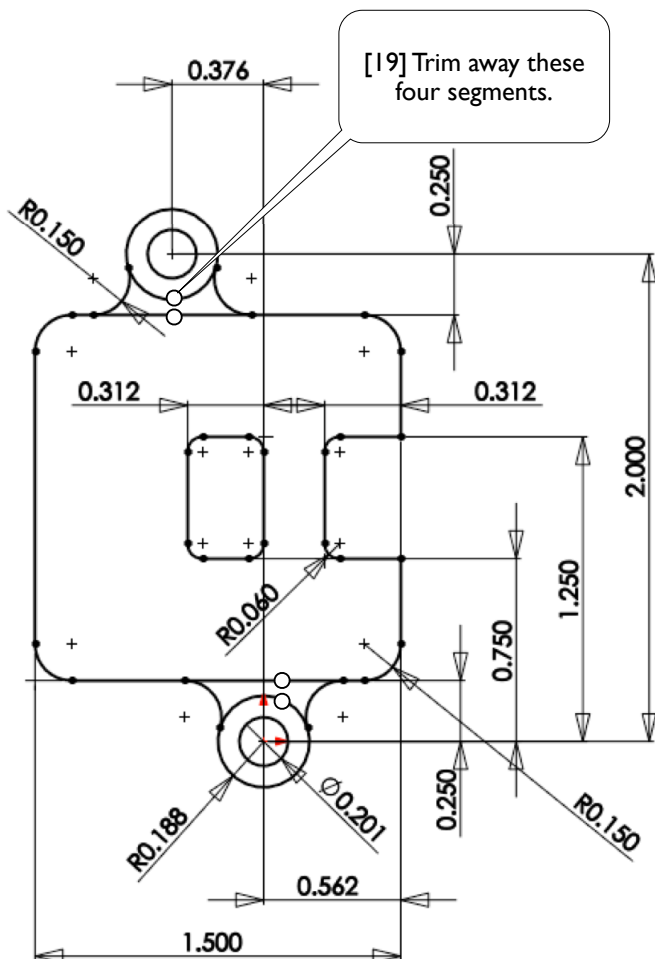
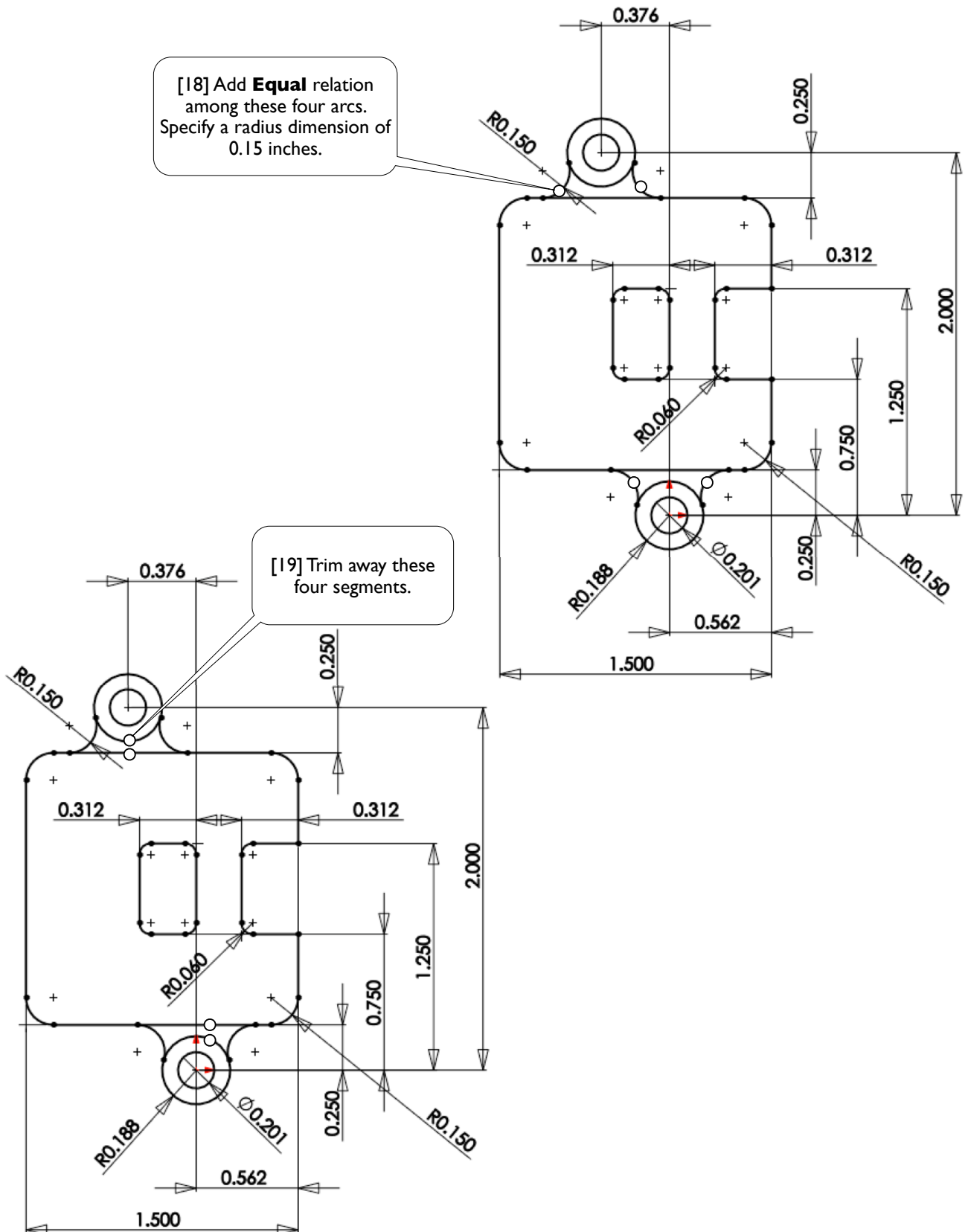
[15] Select **Sketch Entities>3 Point Arc** from the **Context Menu** and draw an arc like this. Add two **Tangent** relations.

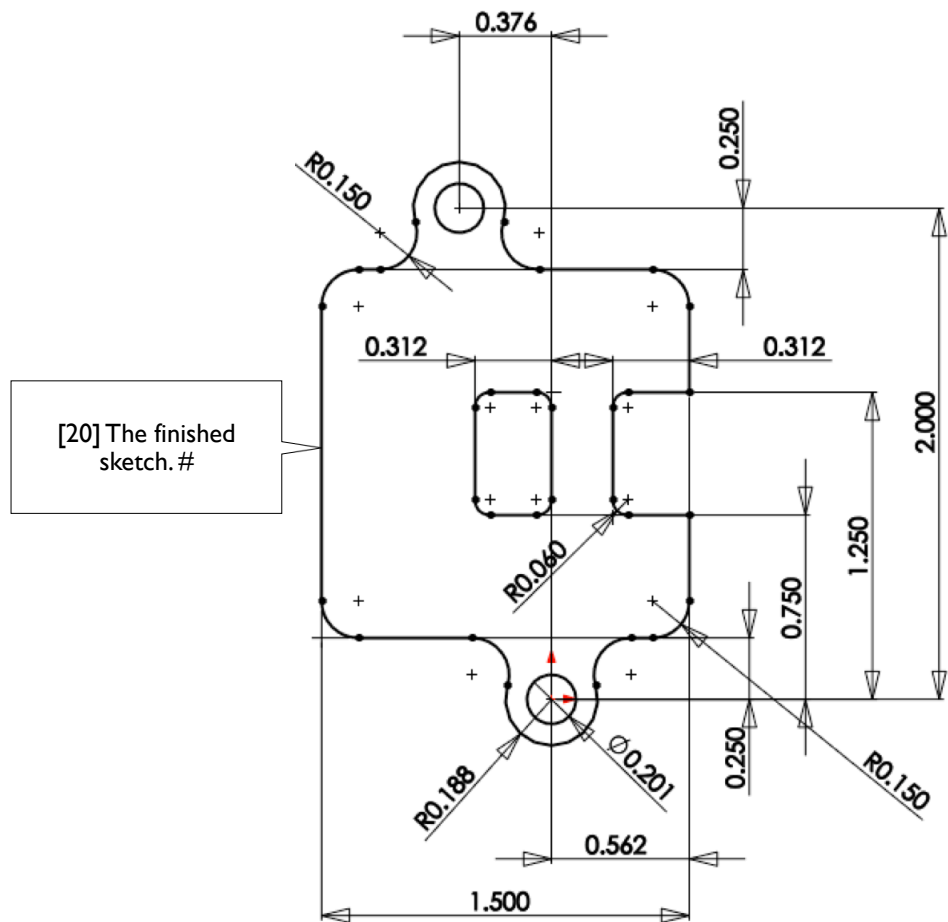


[16] The **3 Point Arc** command is also available in the **Sketch Toolbar**.

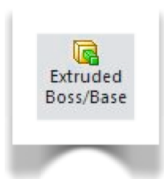


[17] Repeat step [15] three more times.





I.4-4 Generate 3D Model

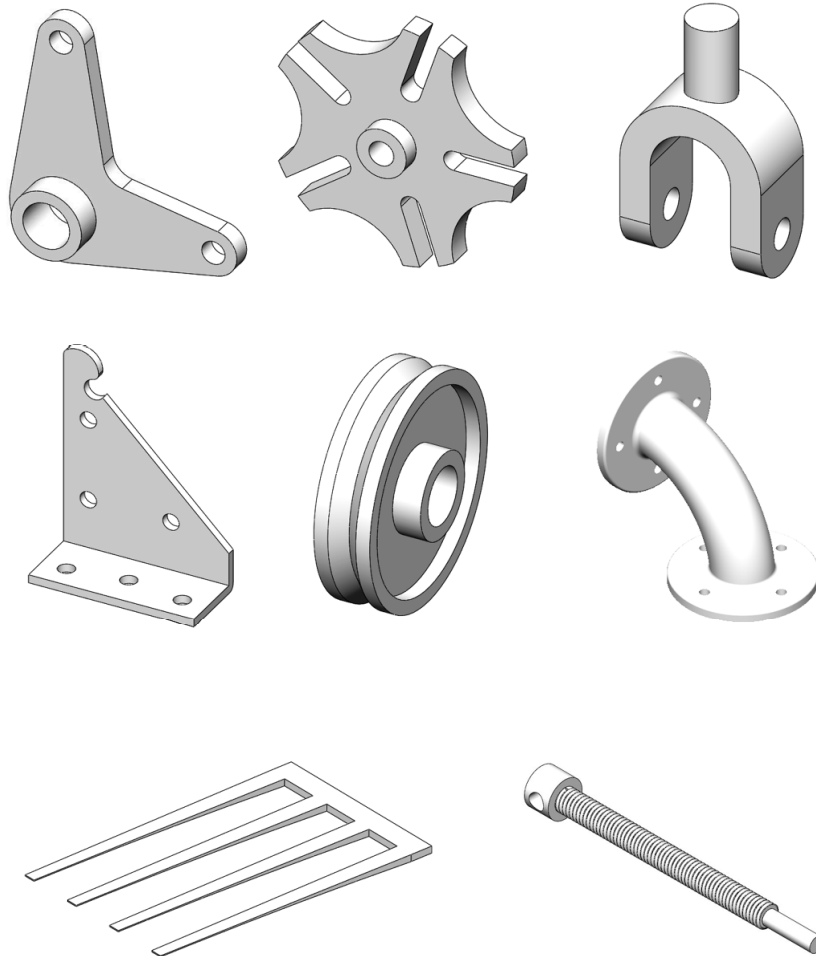


[1] **Extrude** the sketch 0.046 inches to create this 3D model.

[2] Save the part with the file name **Cover**. Close the file and exit **SOLIDWORKS**. #

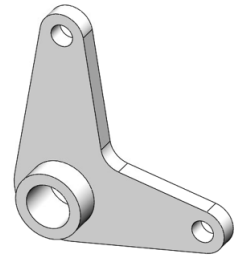
Chapter 2

Part Modeling



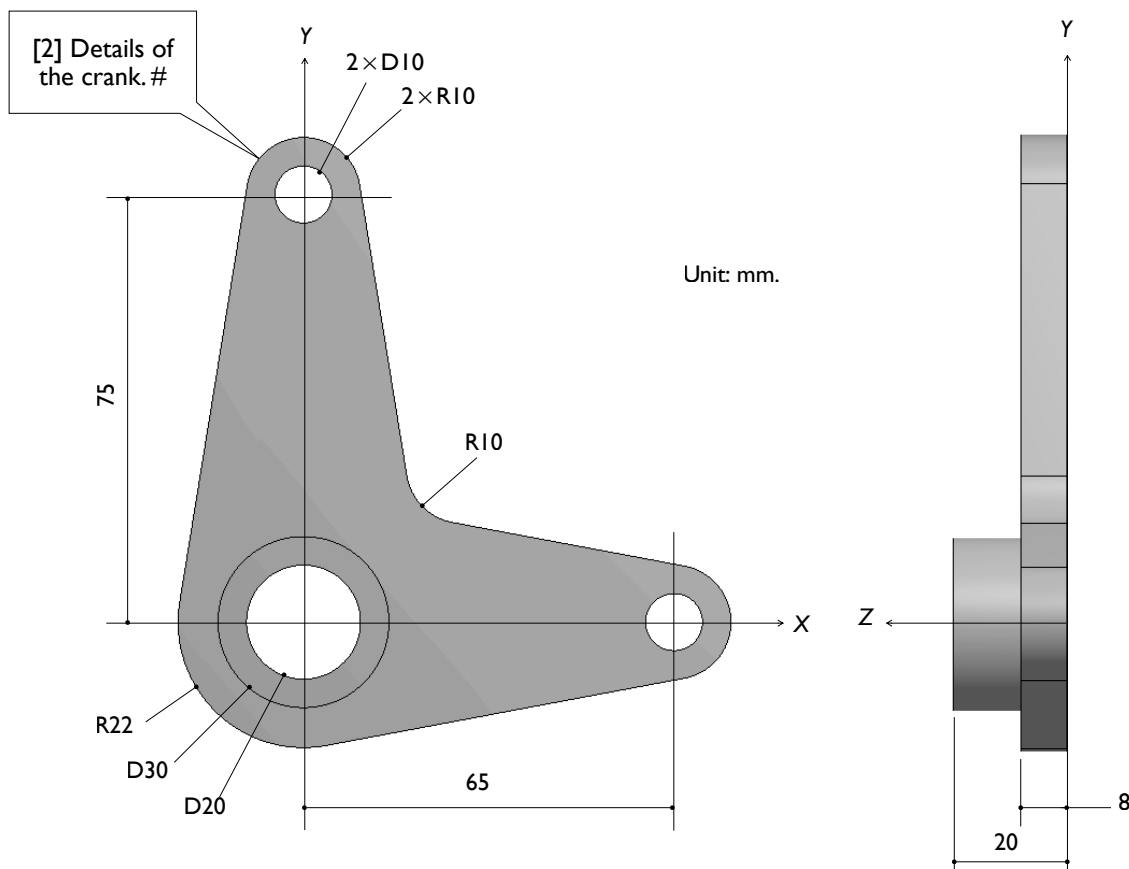
Section 2.1

Crank



2.1-1 About the Crank

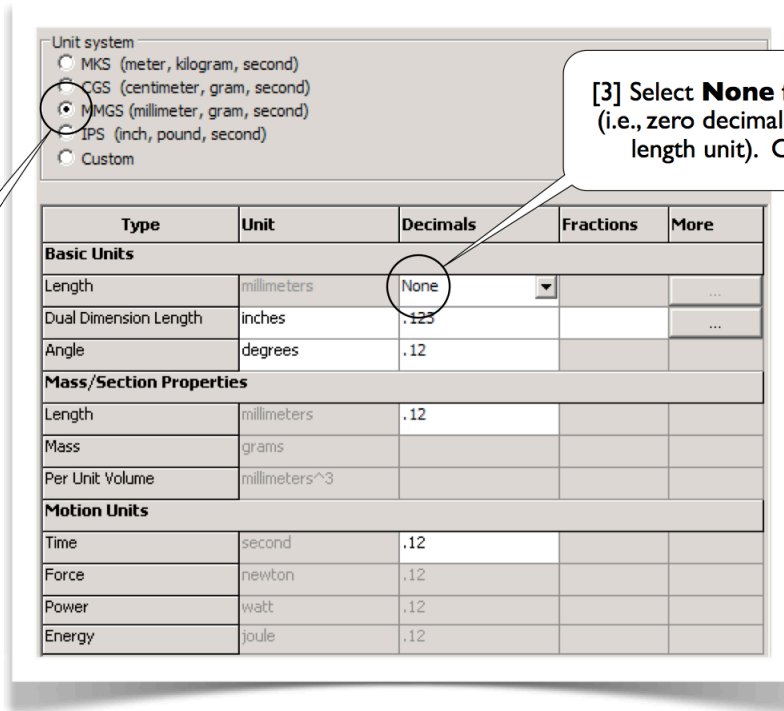
[1] In this exercise, we'll create a 3D solid model for a crank [2]. The model can be viewed as a series of three two-step operations; each involves drawing a sketch on a plane and then extruding the sketch. The material of the body is either added to or cut from the existing body.



2.1-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part.

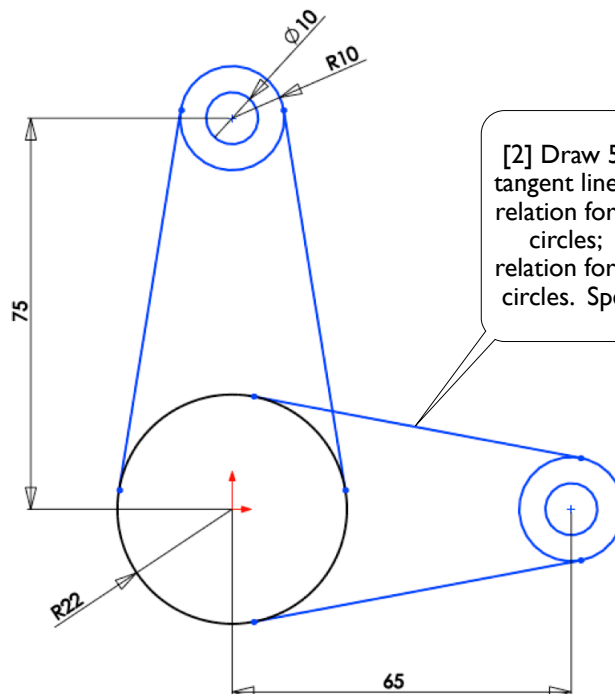
[2] In the **Options**, select **MMGS** as unit system.



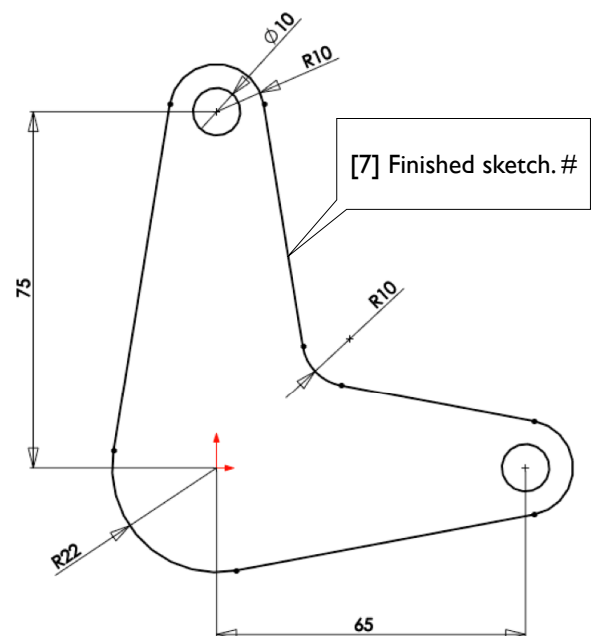
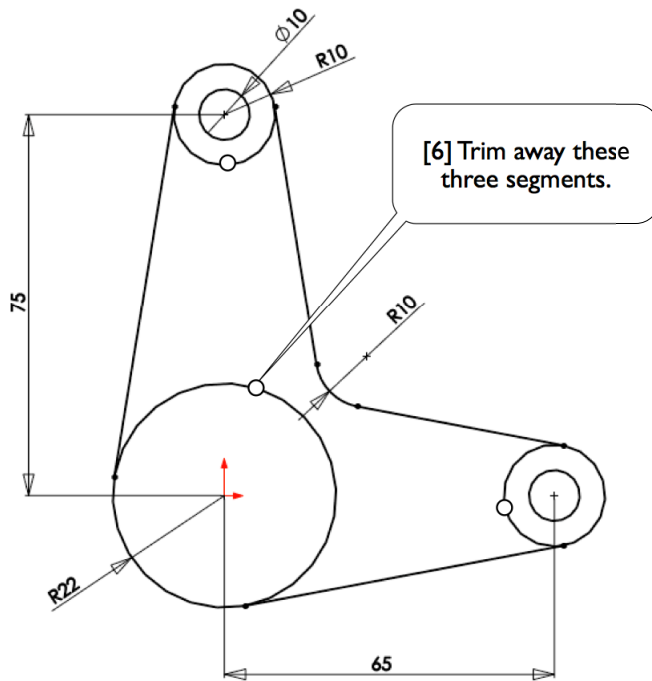
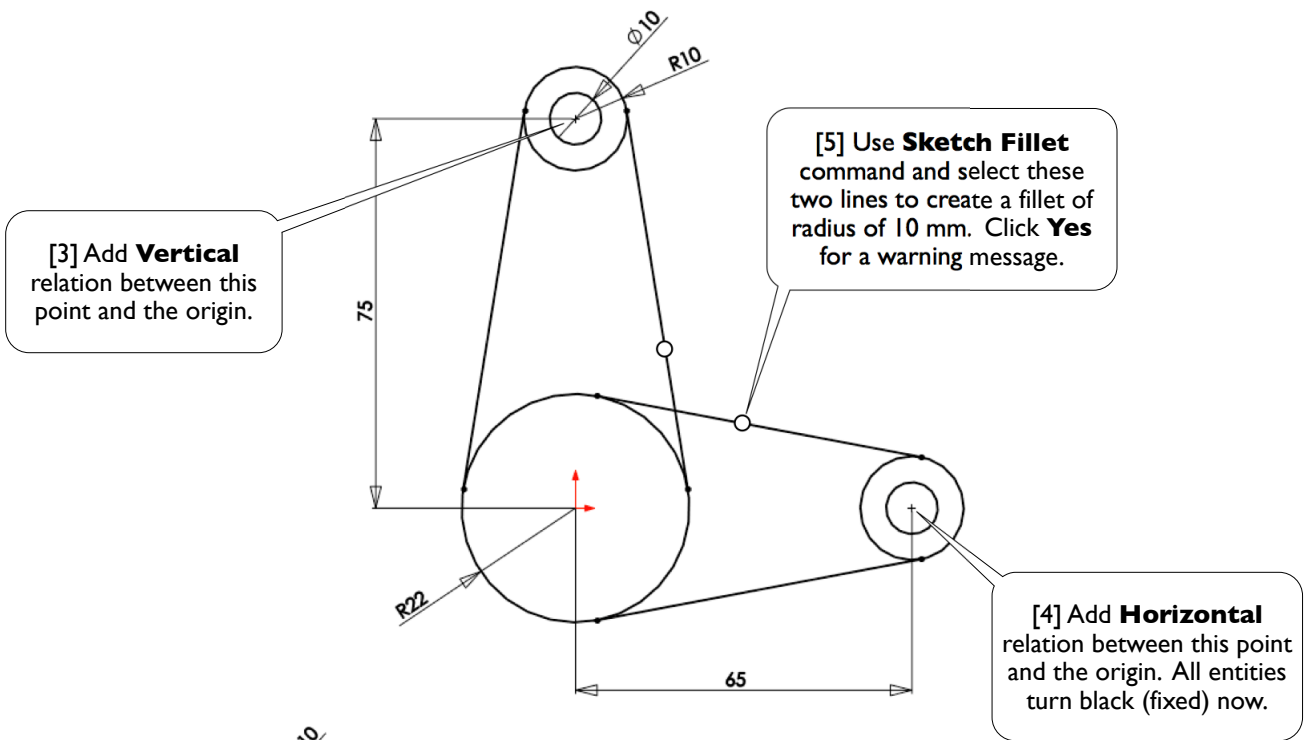
[3] Select **None** for **Decimals** (i.e., zero decimal places for the length unit). Click **OK**. #

2.1-3 Draw a Sketch for the Base Body

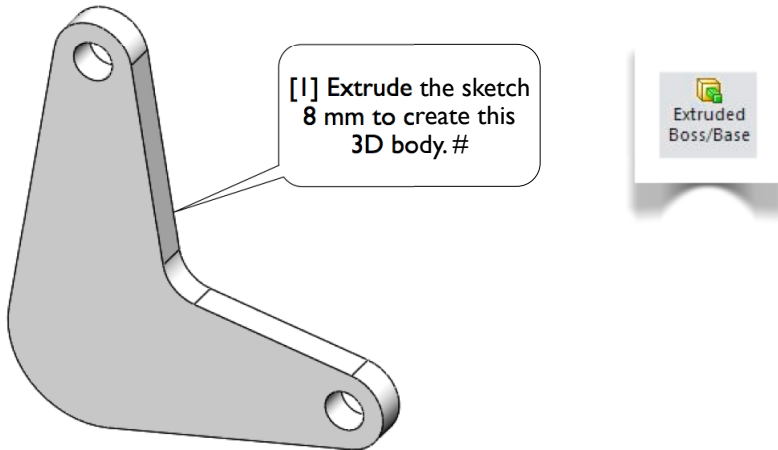
[1] Create a sketch on **Front** plane.



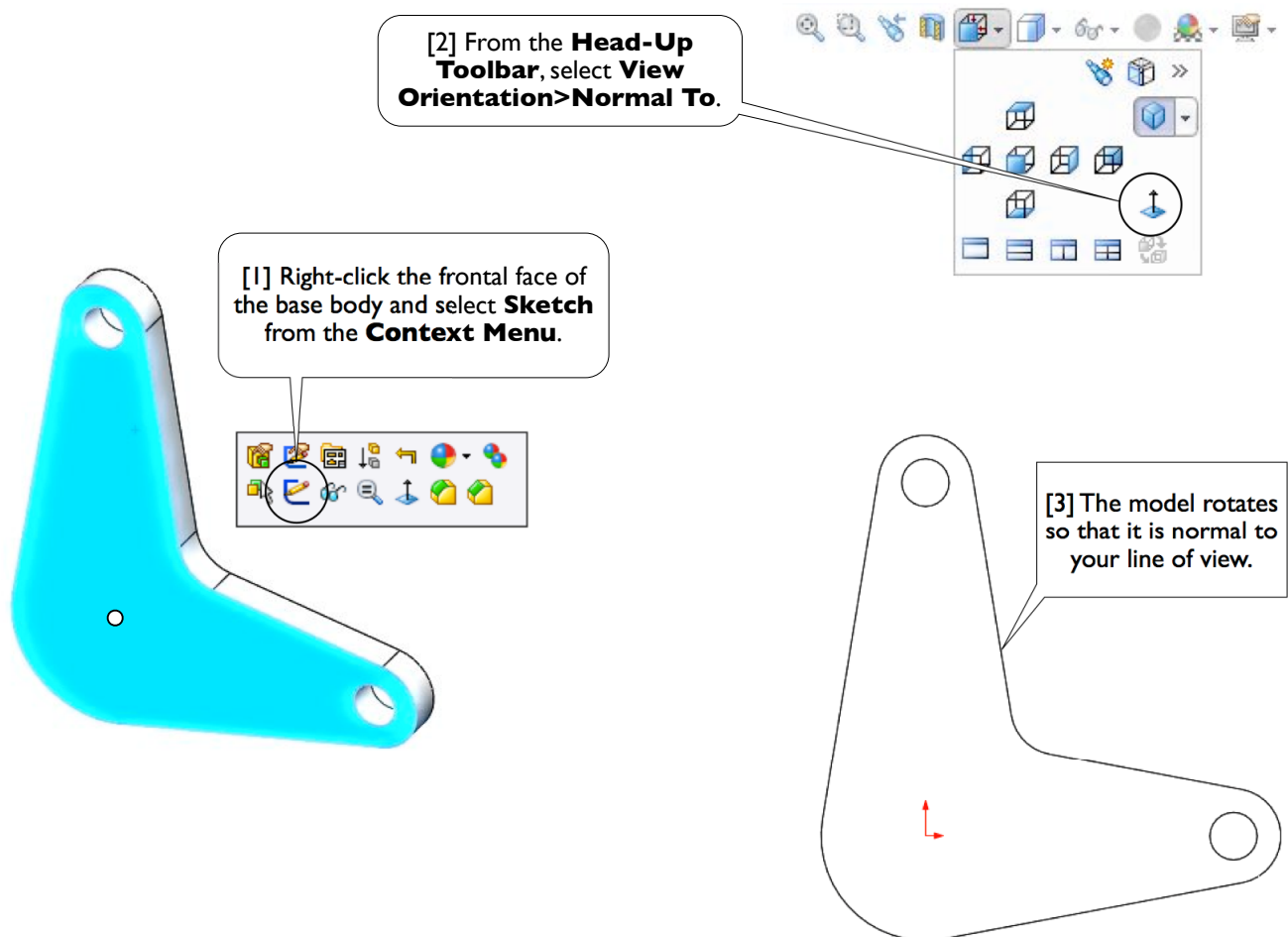
[2] Draw 5 circles and four tangent lines. Add an **Equal** relation for the two smallest circles; add an **Equal** relation for the two medium circles. Specify dimensions.



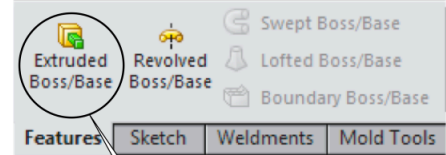
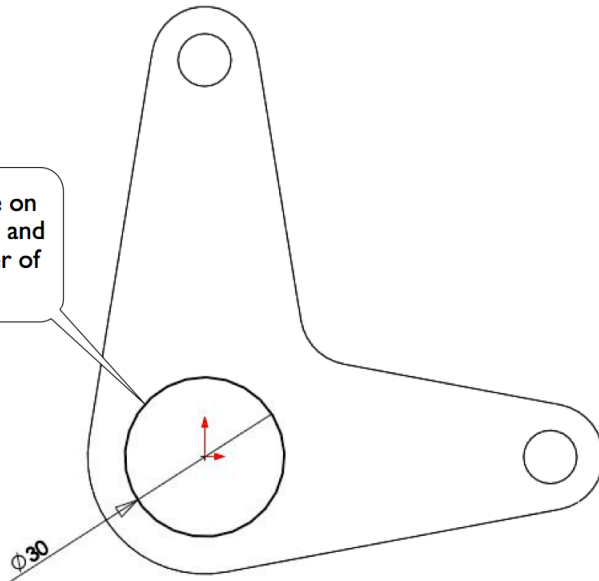
2.1-4 Extrude the Sketch to Create the Base Body



2.1-5 Add Features to the Base Body

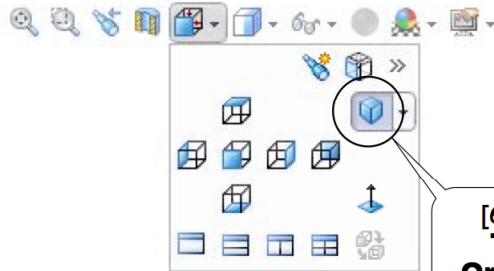


[4] Draw a circle on the plane like this and specify a diameter of 30 mm.



[5] **Extrude** the sketch 12 mm.

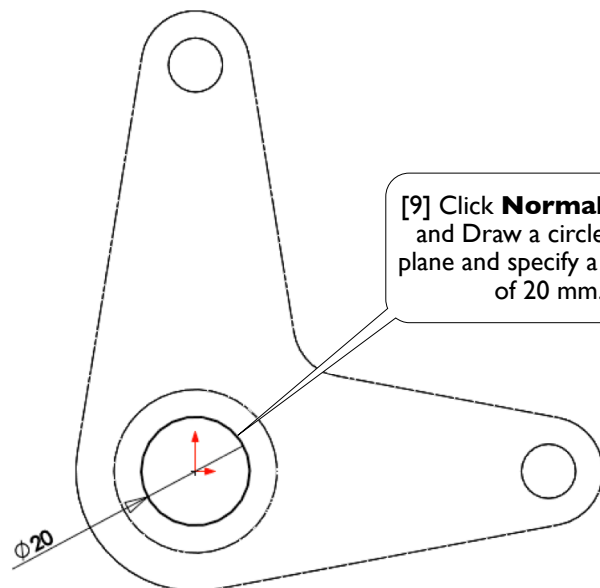
[10] The **Normal To** is also available in the **Standard Views Toolbar**.



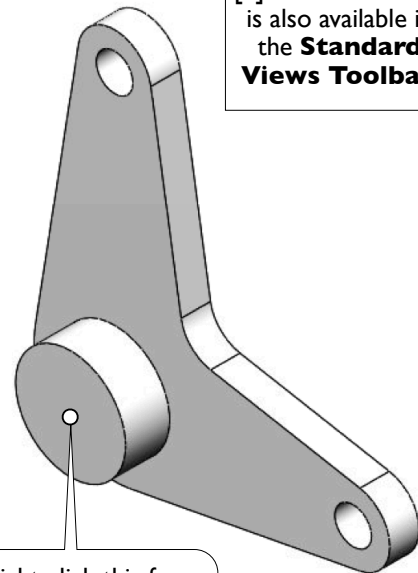
[6] From the **Head-Up Toolbar**, select **View Orientation>Isometric**.

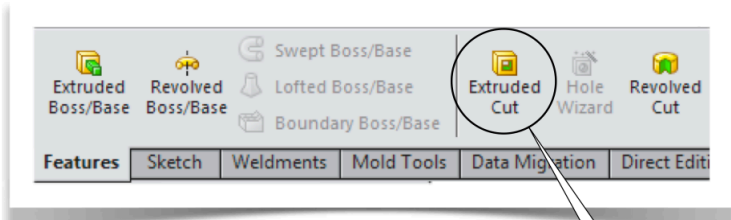
[7] The **Isometric** is also available in the **Standard Views Toolbar**.

[9] Click **Normal To** [10] and Draw a circle on the plane and specify a diameter of 20 mm.



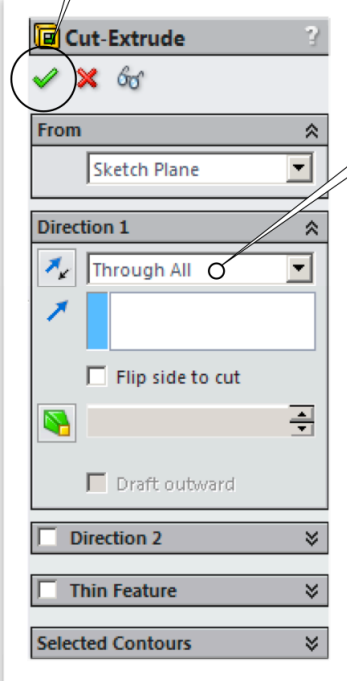
[8] Right-click this face and select **Sketch** from the **Context Menu** (see [1], last page)





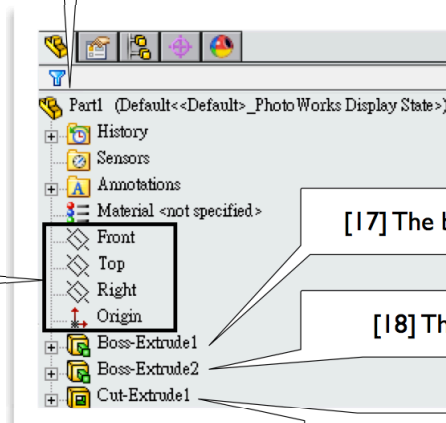
[13] Click **OK**.

[11] From the **Pull-Down Menus**, select **Insert>Cut>Extrude...** or, from **Features Toolbar**, select **Extruded Cut**.



[12] Select **Through All** for **End Condition**.

[15] The **Part Tree (Features Tree)**.



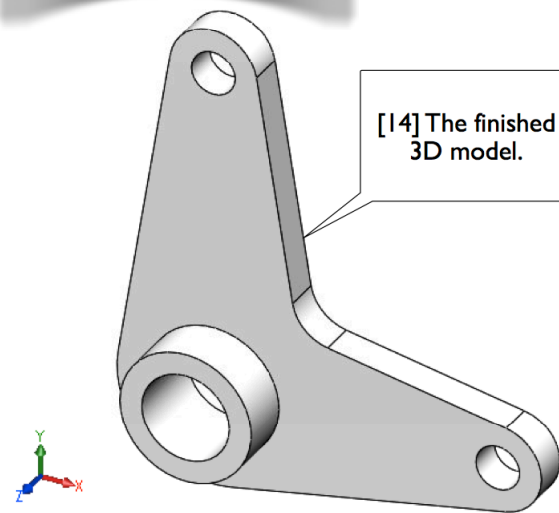
[16] The **reference geometries**.

[17] The base body.

[18] The boss.

[19] The hole.

[14] The finished 3D model.



*Trimetric

[20] A **part** consists of reference geometries, base body, and **features** on the base body. In this example, the features added to the base body are the boss and the hole. Note that it is equally good to treat the two small holes as features on the base body.

[21] Save the part with the file name **Crank**.
Exit **SOLIDWORKS**.#

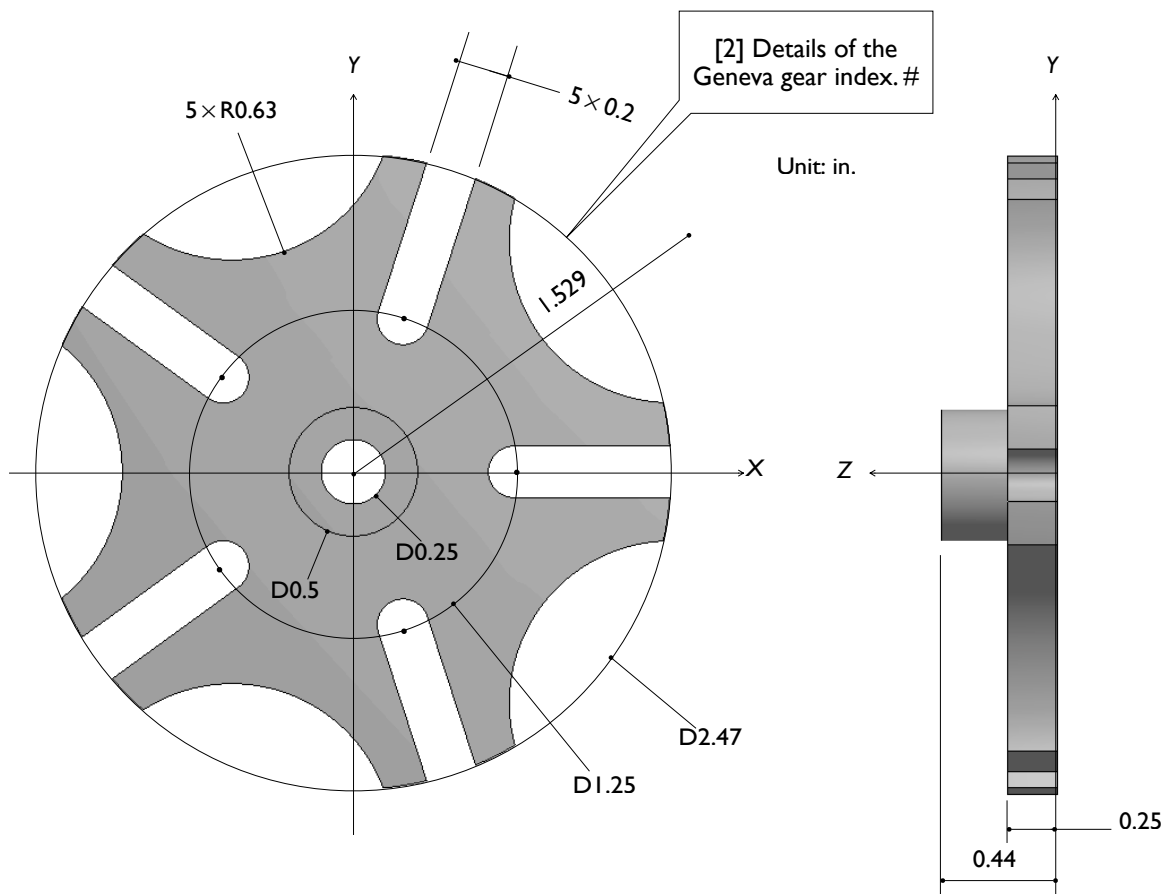
Section 2.2

Geneva Gear Index



2.2-1 About the Geneva Gear Index

[1] In this exercise, we'll create a 3D solid model for a Geneva gear index [2].

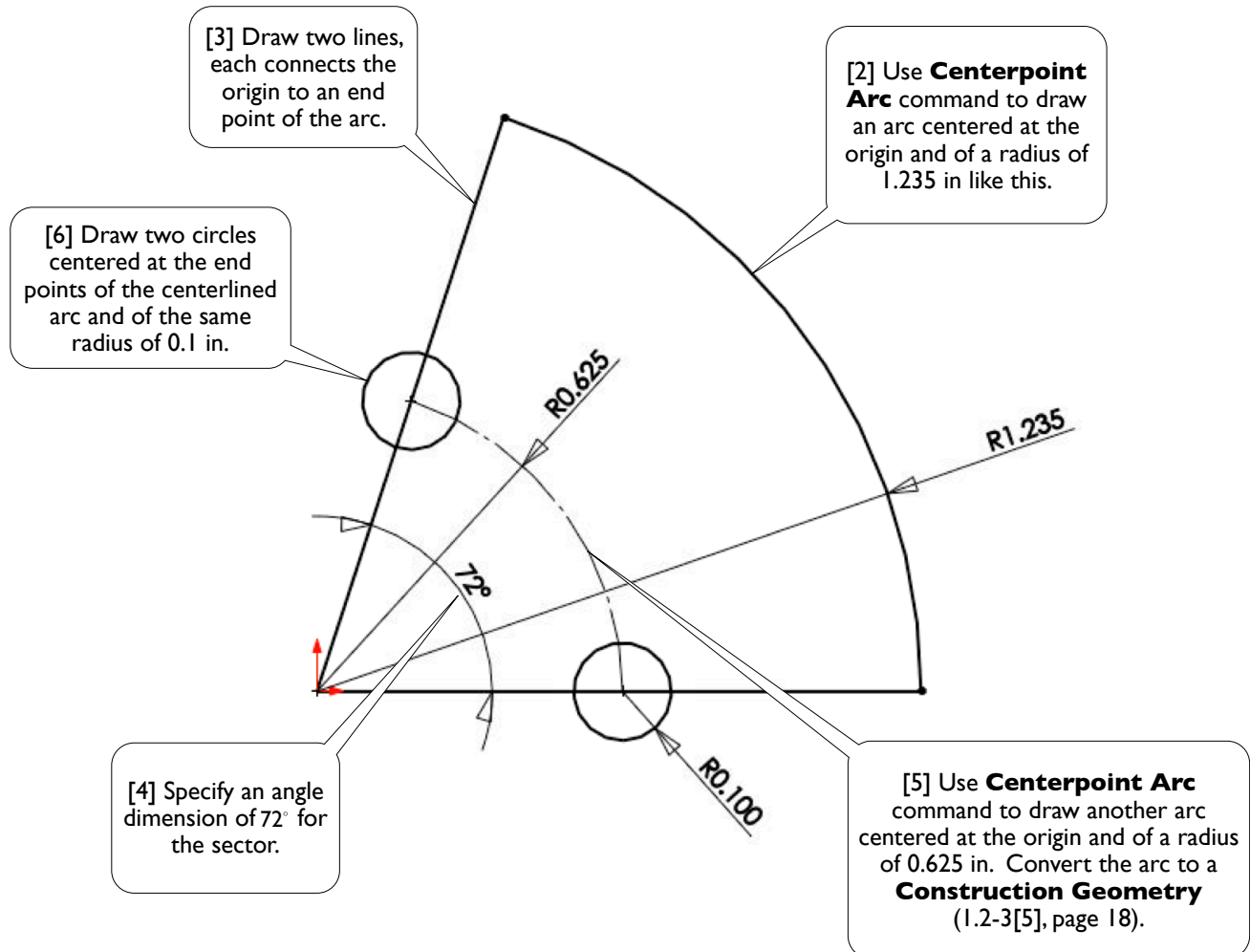


2.2-2 Start Up

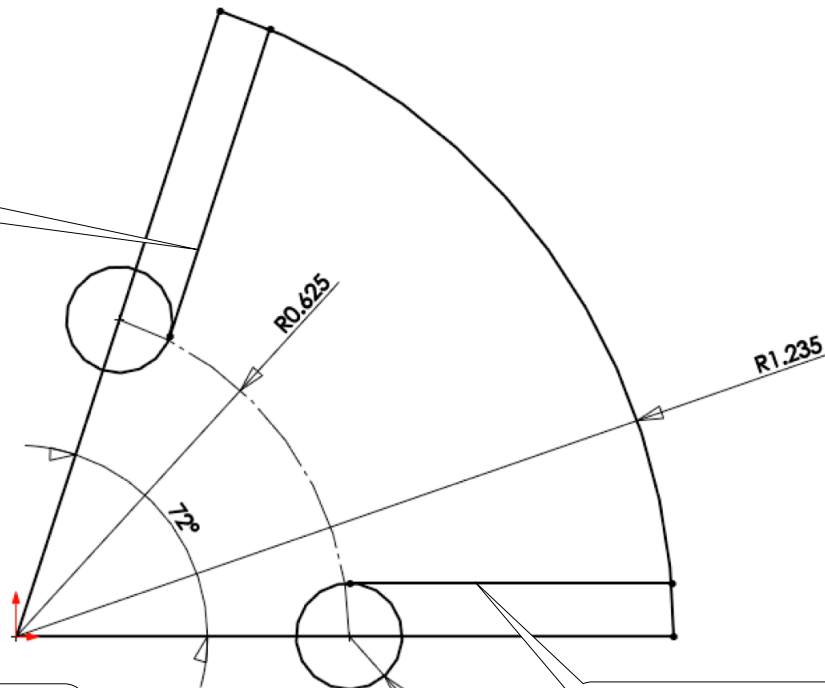
[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit. #

2.2-3 Draw a Sketch for 1/5 of the Gear Index

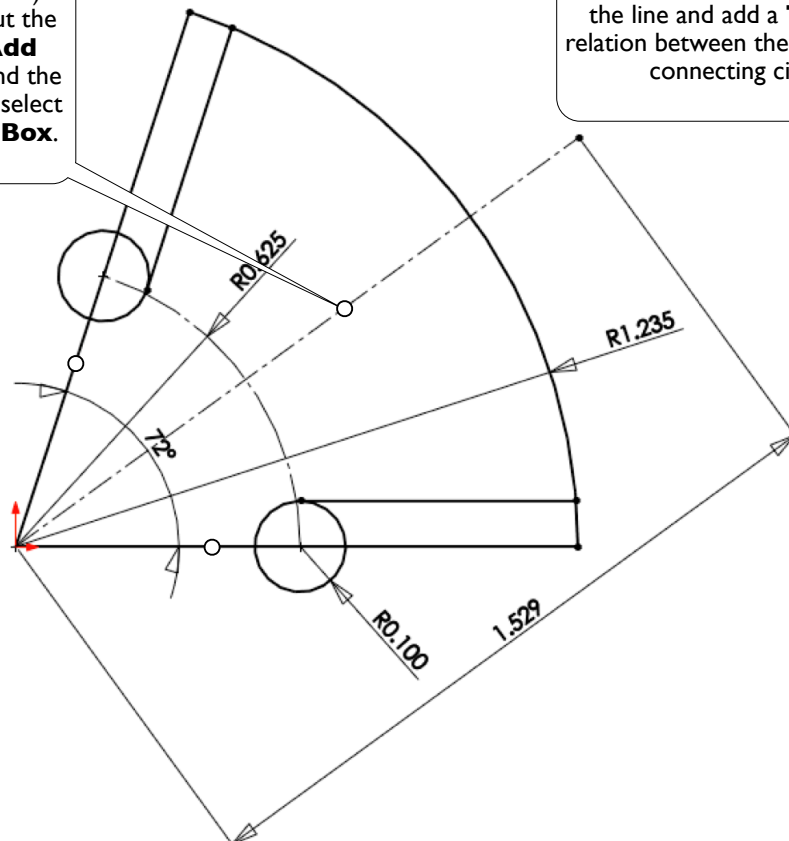
[1] Create a sketch on **Front** plane.



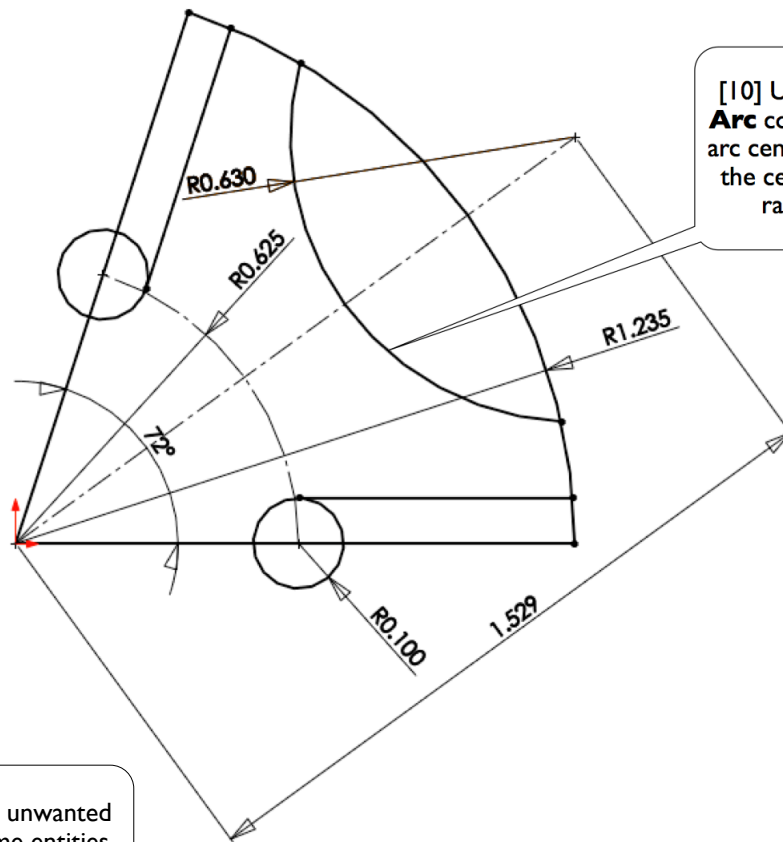
[8] Draw a line connecting the upper circle to the outer arc. Add a **Parallel** relation between the line and the line next to it. Add a **Tangent** relation between the line and the connecting circle.



[9] Use **Centerline** command to draw a centerline starting from the origin. Specify the length (1.529 in). Make the sector symmetric about the centerline. To do this, select **Add Relation**, click the centerline and the two edge lines of the sector, and select **Symmetric** in the **Property Box**.



[7] Draw a line connecting the lower circle to the outer arc. Add a **Horizontal** relation on the line and add a **Tangent** relation between the line and the connecting circle.

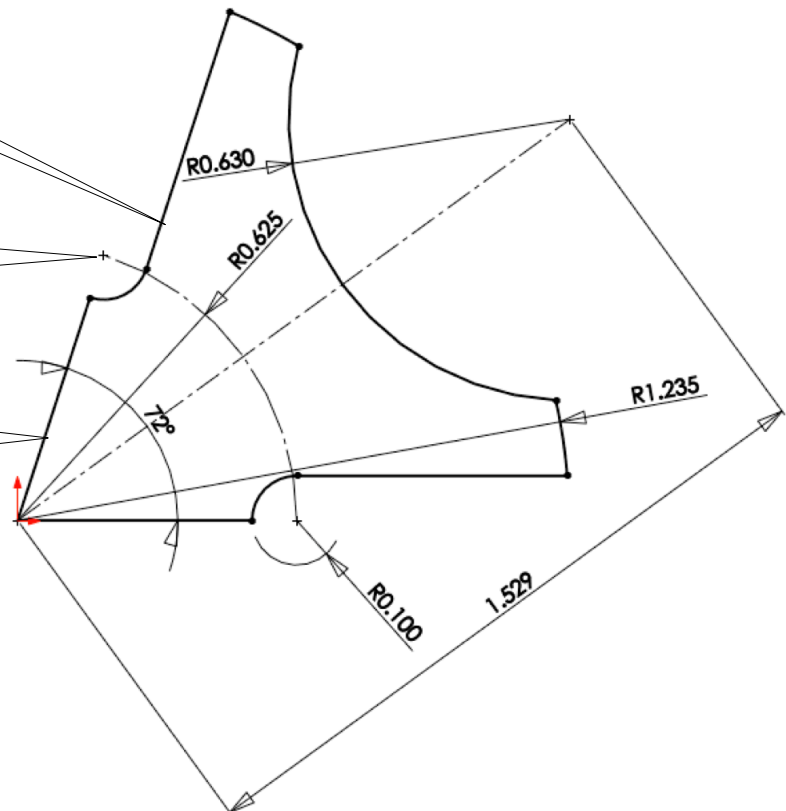


[10] Use **Centerpoint Arc** command to draw an arc centered at one end of the centerline. Specify a radius of 0.63 in.

[11] Trim away unwanted segments. Some entities turn back to blue color; add relations to fix them (see [12-14]).

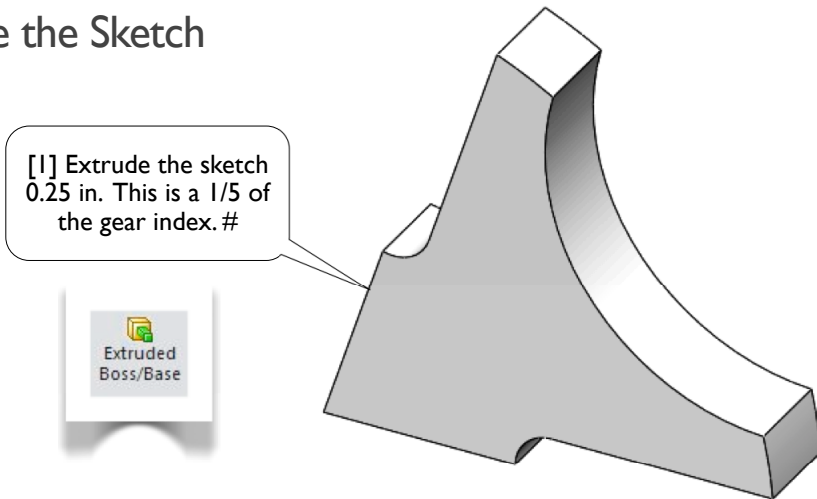
[12] Some entities turn back to blue, try to fix them by adding a **Coincident** relation for this point...

[13] And this line. Add another **Coincident** for the other side of the sector.

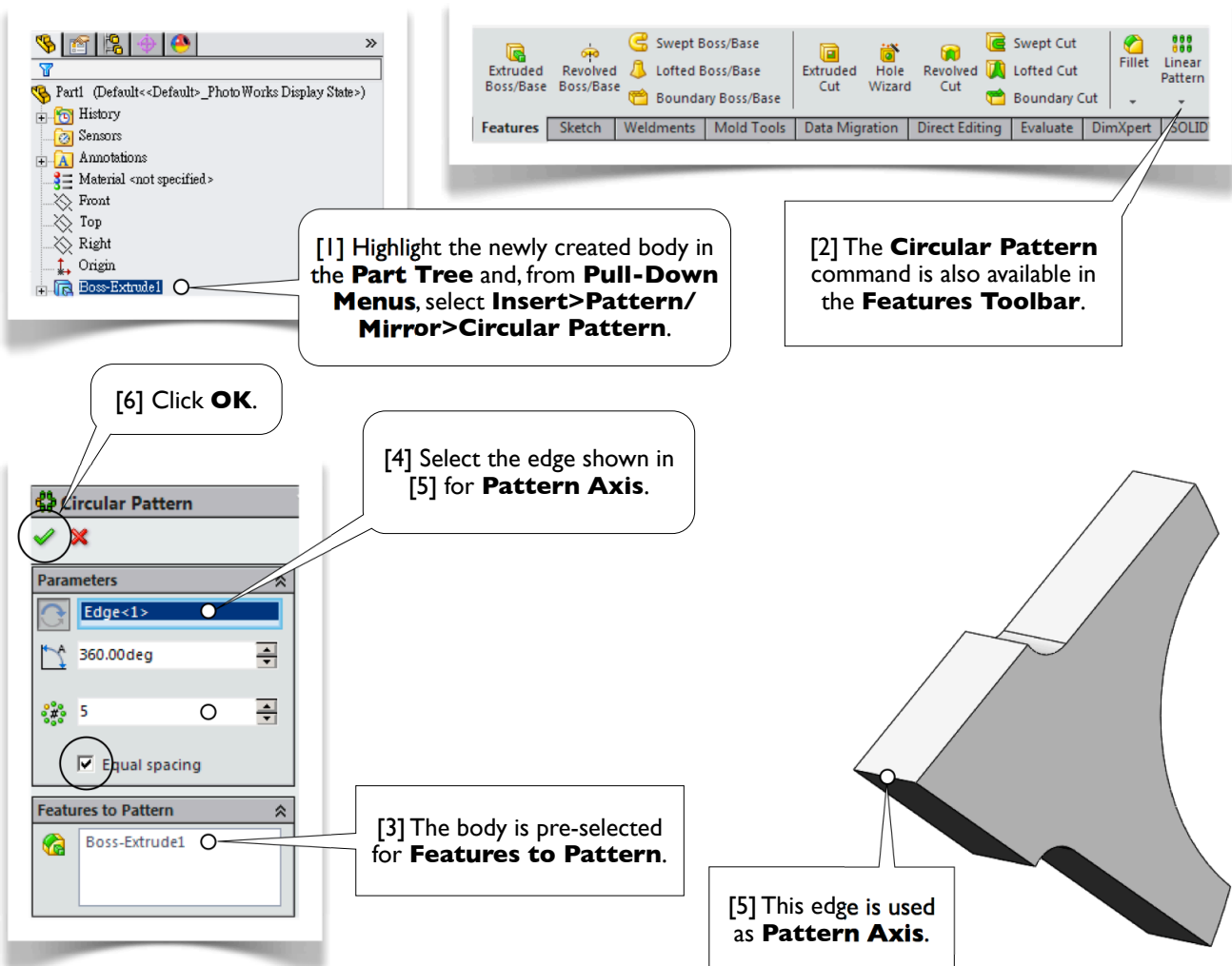


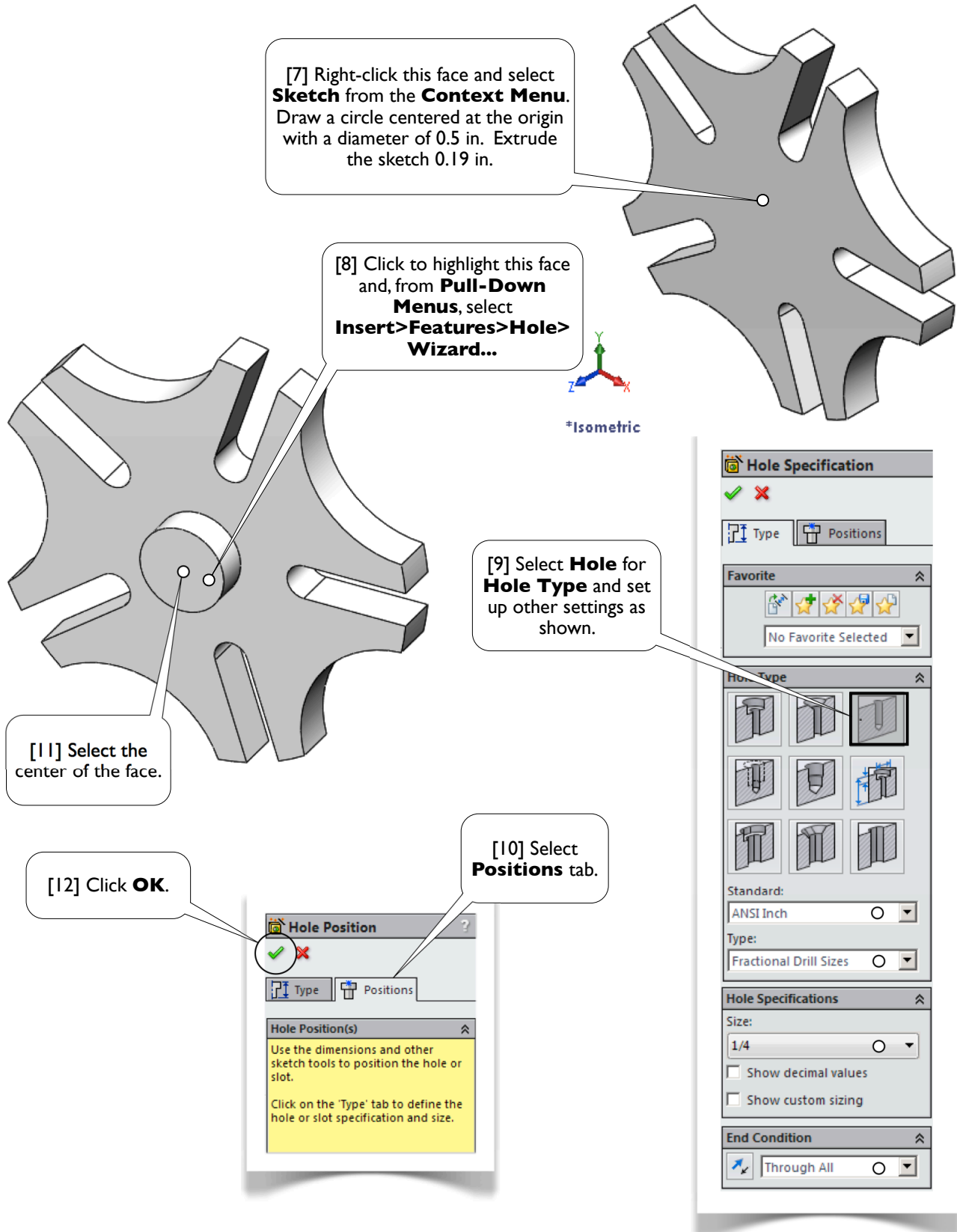
[14] If steps [12, 13] don't fix all the entities. Try to drag an unfixed entity to figure out what relations should be added. #

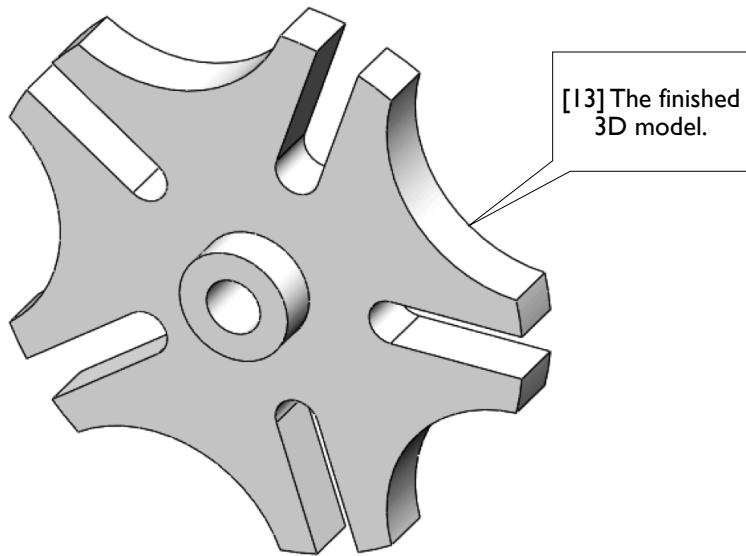
2.2-4 Extrude the Sketch



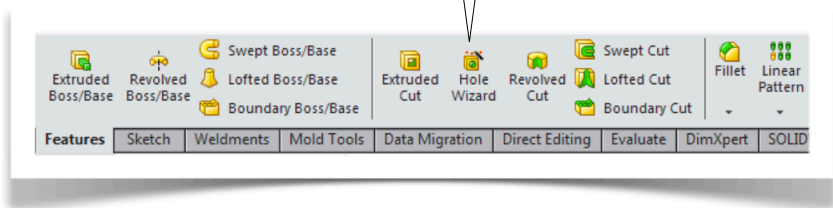
2.2-5 Complete the Full Model







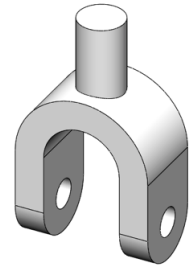
[14] The **Hole Wizard** command is also available in **Features Toolbar**.



[15] Save the part with the file name **Geneva**. Close the file and exit **SOLIDWORKS**. #

Section 2.3

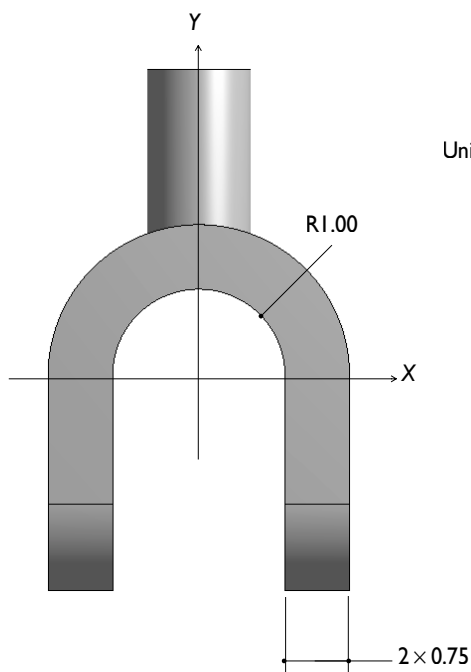
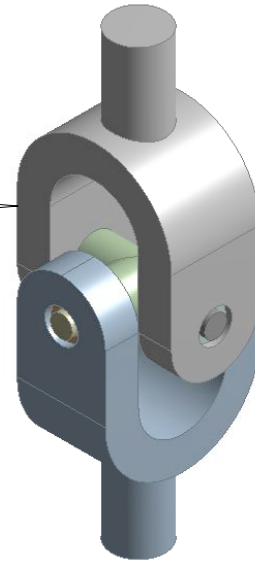
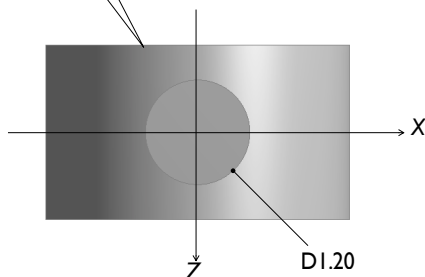
Yoke



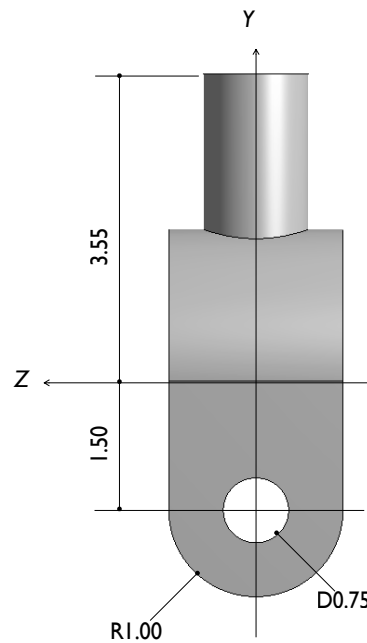
2.3-1 About the Yoke

[2] Details of the yoke. #

[1] The yoke is a part of a universal joint. In this exercise, we'll create a 3D solid model for the yoke.



Unit: in.



2.3-2 Start Up

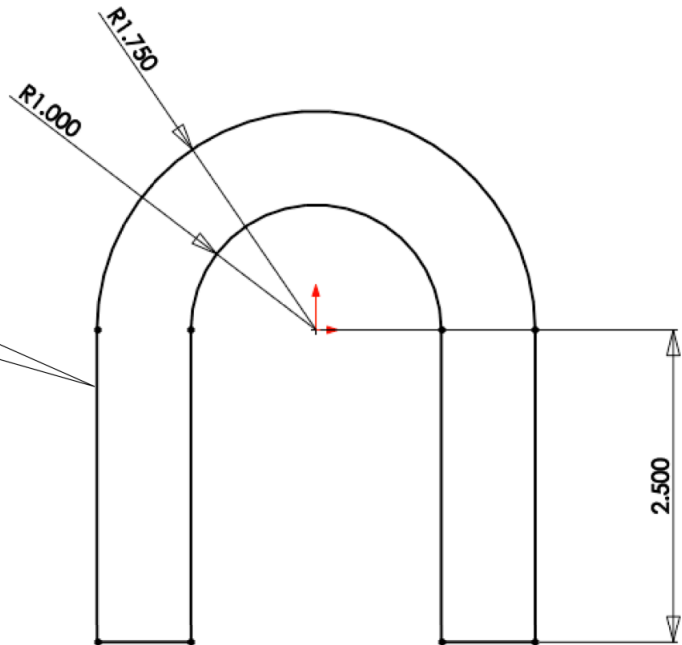
[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit. #

2.3-3 Create a Base Body

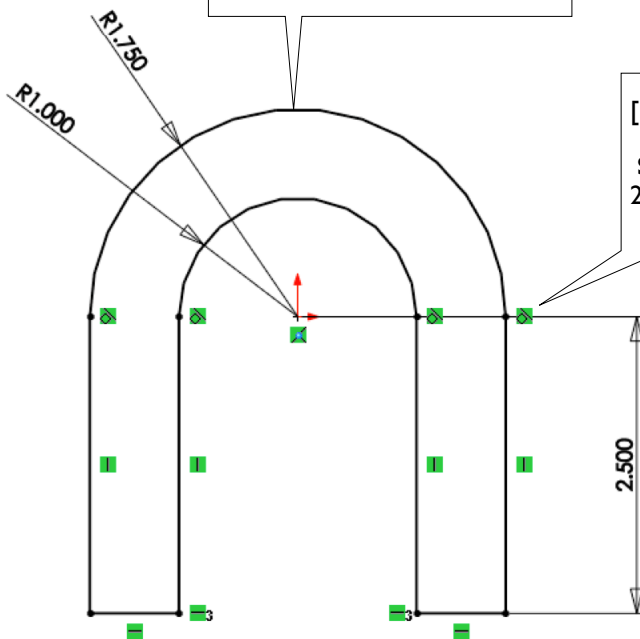
[1] Create a sketch on **Front** plane.

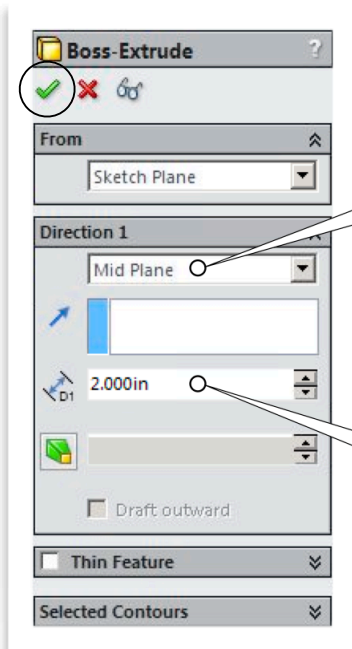
[2] Draw a sketch like this. If there are any blue entities (not well-defined), see [3, 4].

[4] Another way is to drag an unfixed entity to figure out what relations should be added.



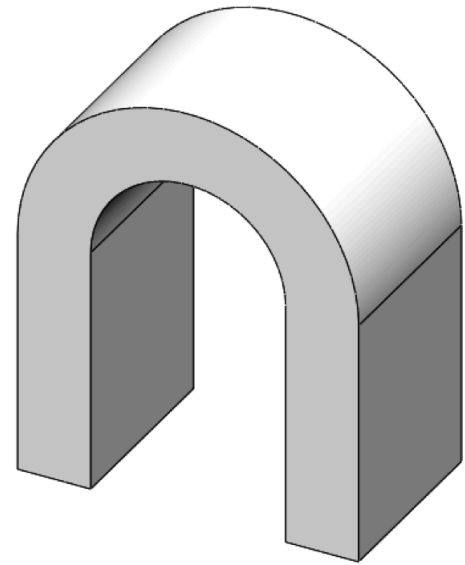
[3] If there are any blue entities, select **Hide/Show Items>View Sketch Relations** (1.3-3[10], page 25) to view all relations. Add relations to fix the entities.



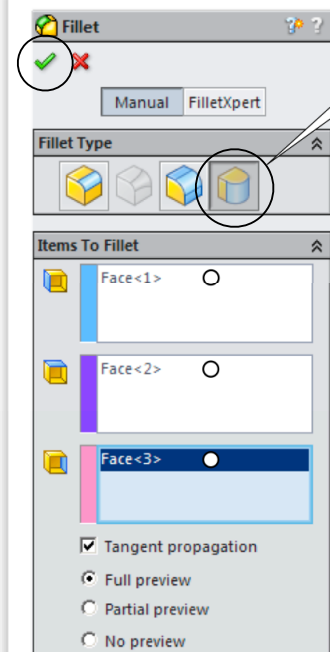


[6] Select **Mid Plane** so that the sketch extrudes both sides (2 inches is the total depth). #

[5] Extrude the sketch 2 in.



2.3-4 Create Rounds and Holes



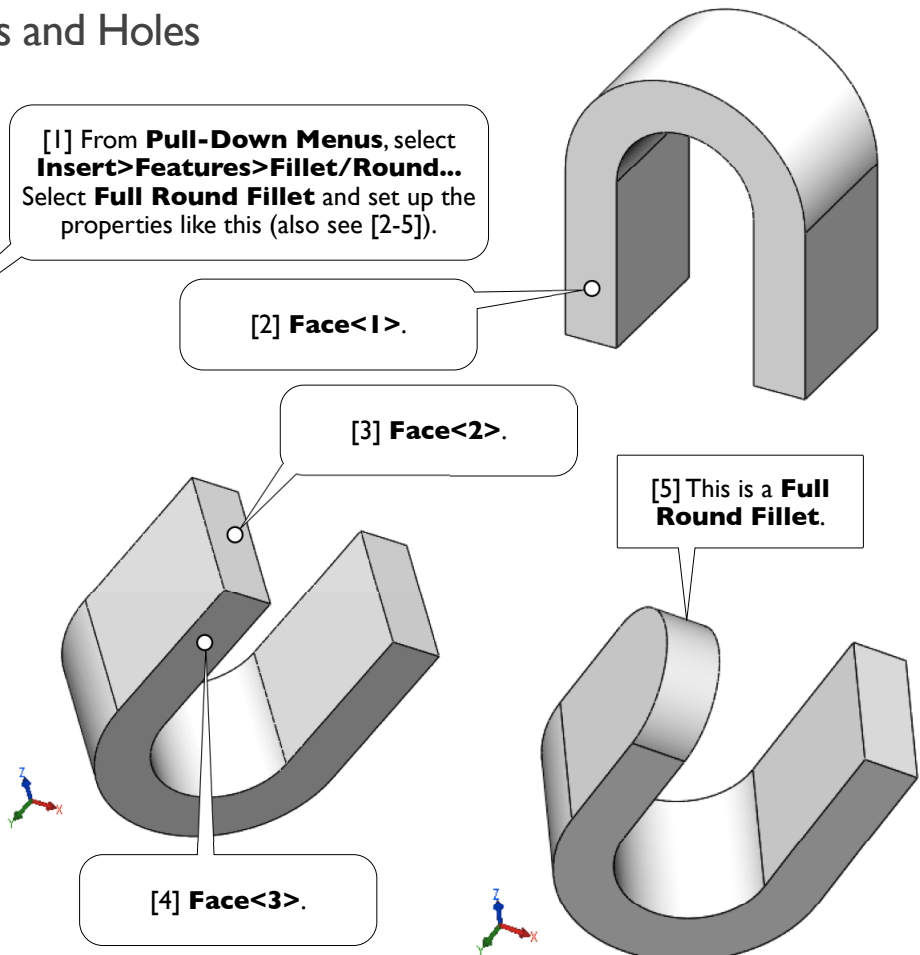
[1] From **Pull-Down Menus**, select **Insert>Features>Fillet/Round...** Select **Full Round Fillet** and set up the properties like this (also see [2-5]).

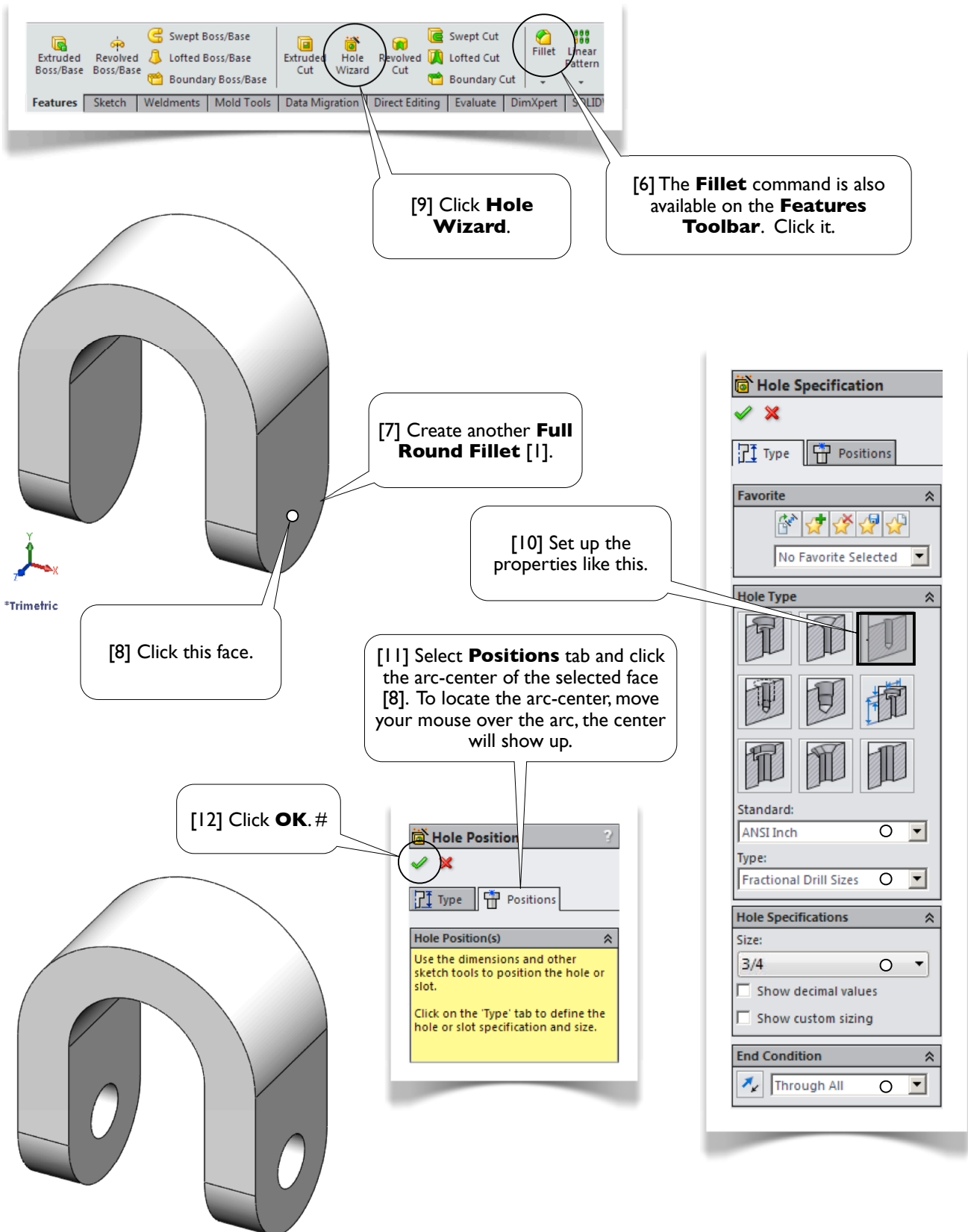
[2] **Face<1>**.

[3] **Face<2>**.

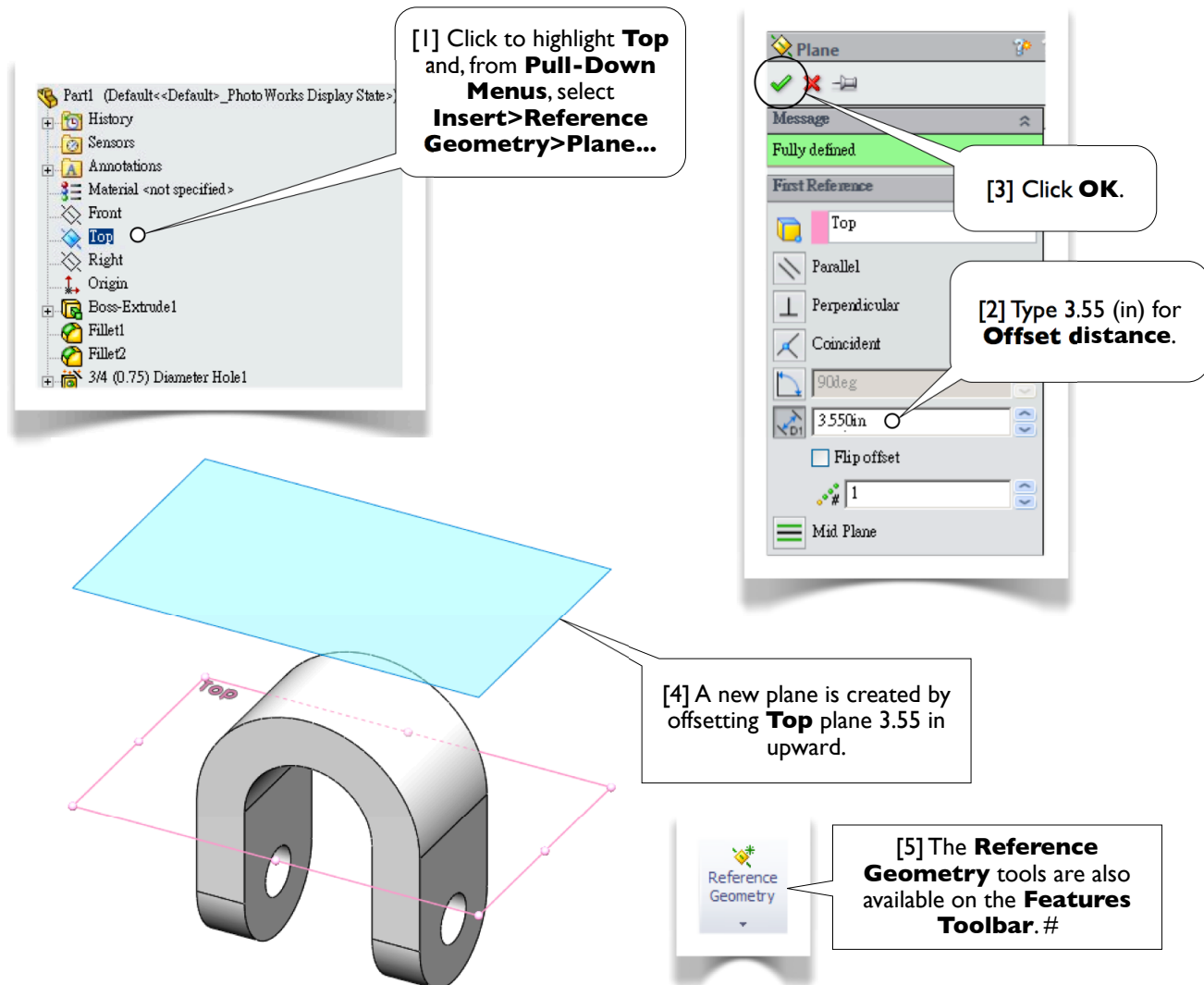
[4] **Face<3>**.

[5] This is a **Full Round Fillet**.

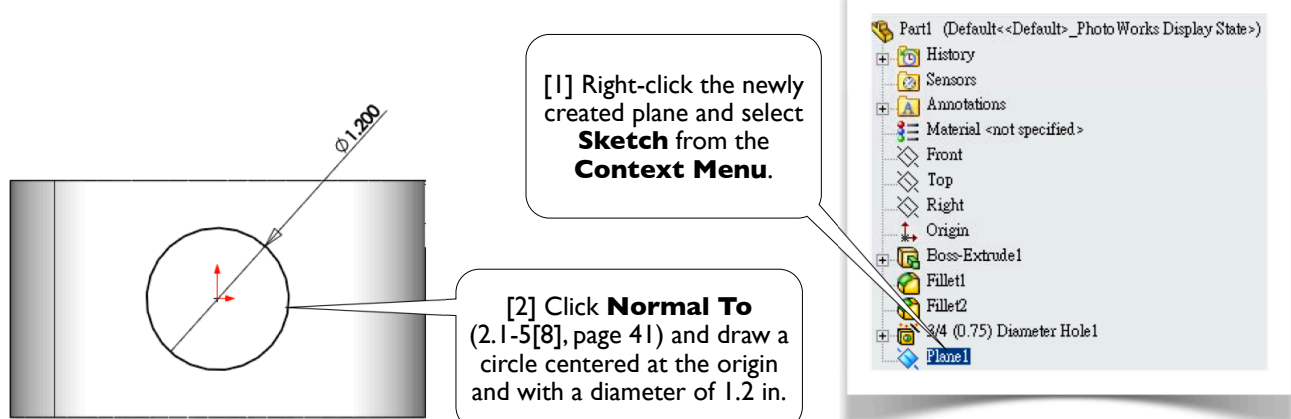


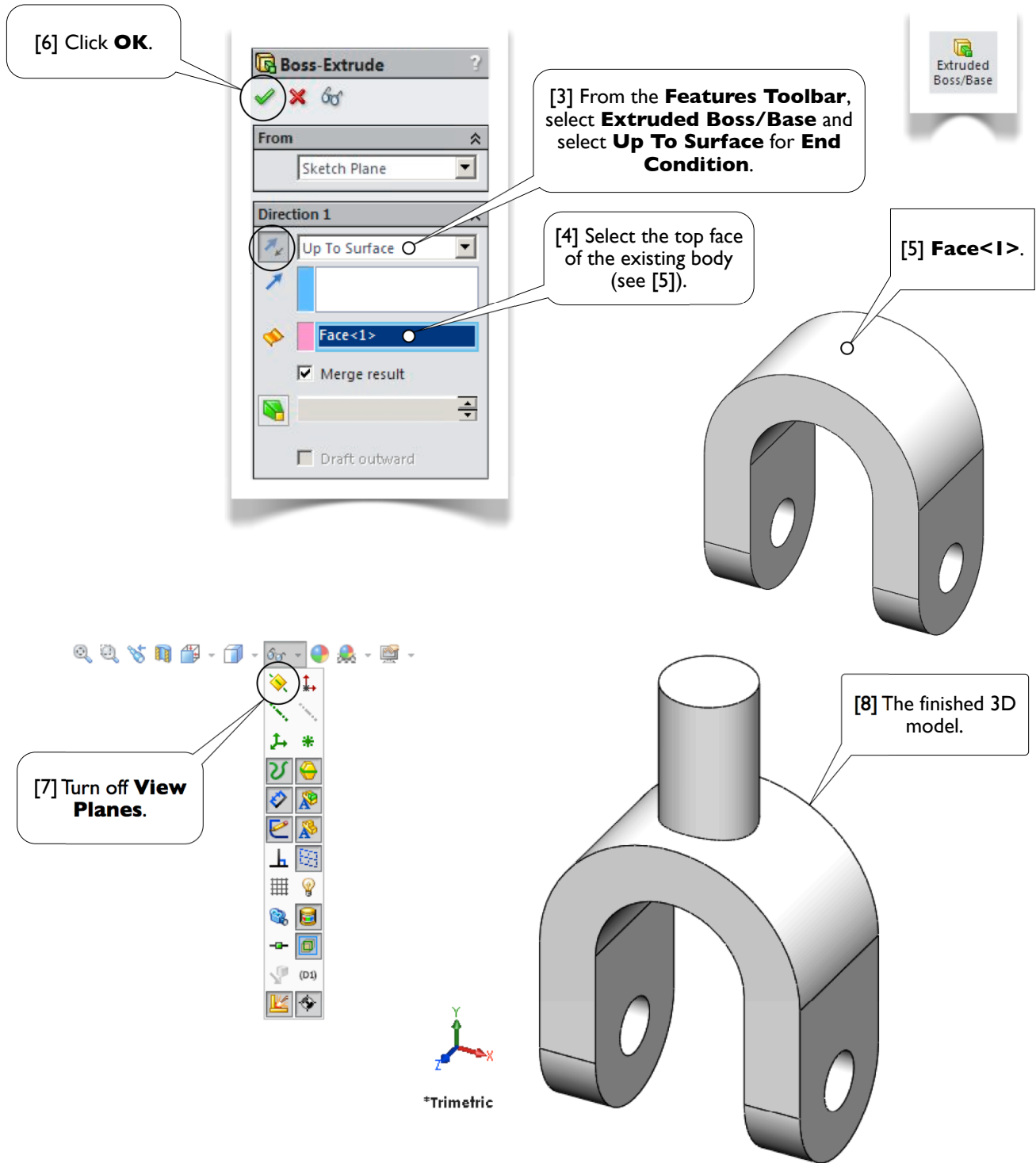


2.3-5 Create a Plane



2.3-6 Create the Shaft

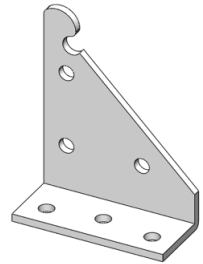




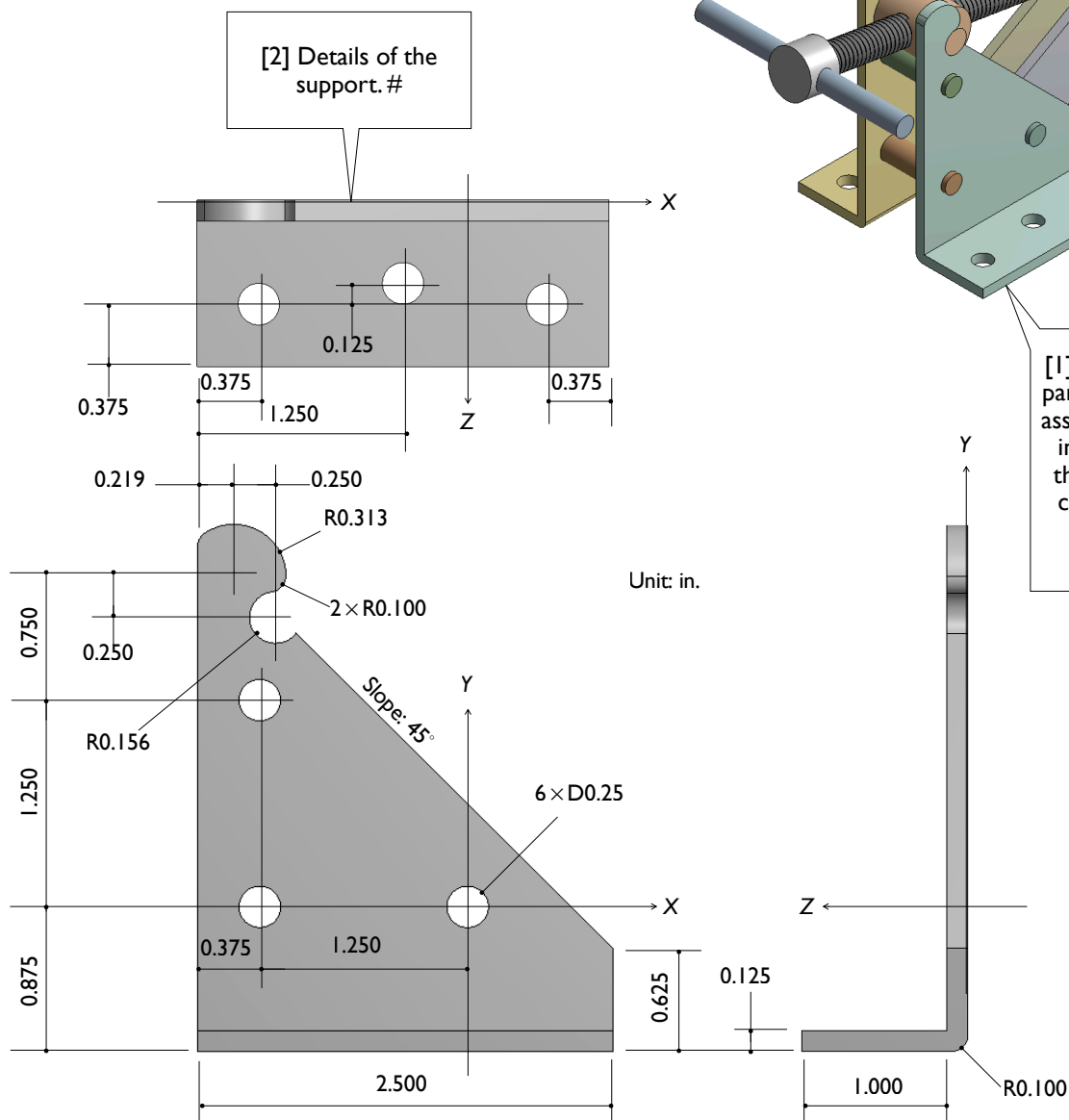
[9] Save the part with the file name **Yoke**. Close the file and exit **SOLIDWORKS**.#

Section 2.4

Support



2.4-1 About the Support



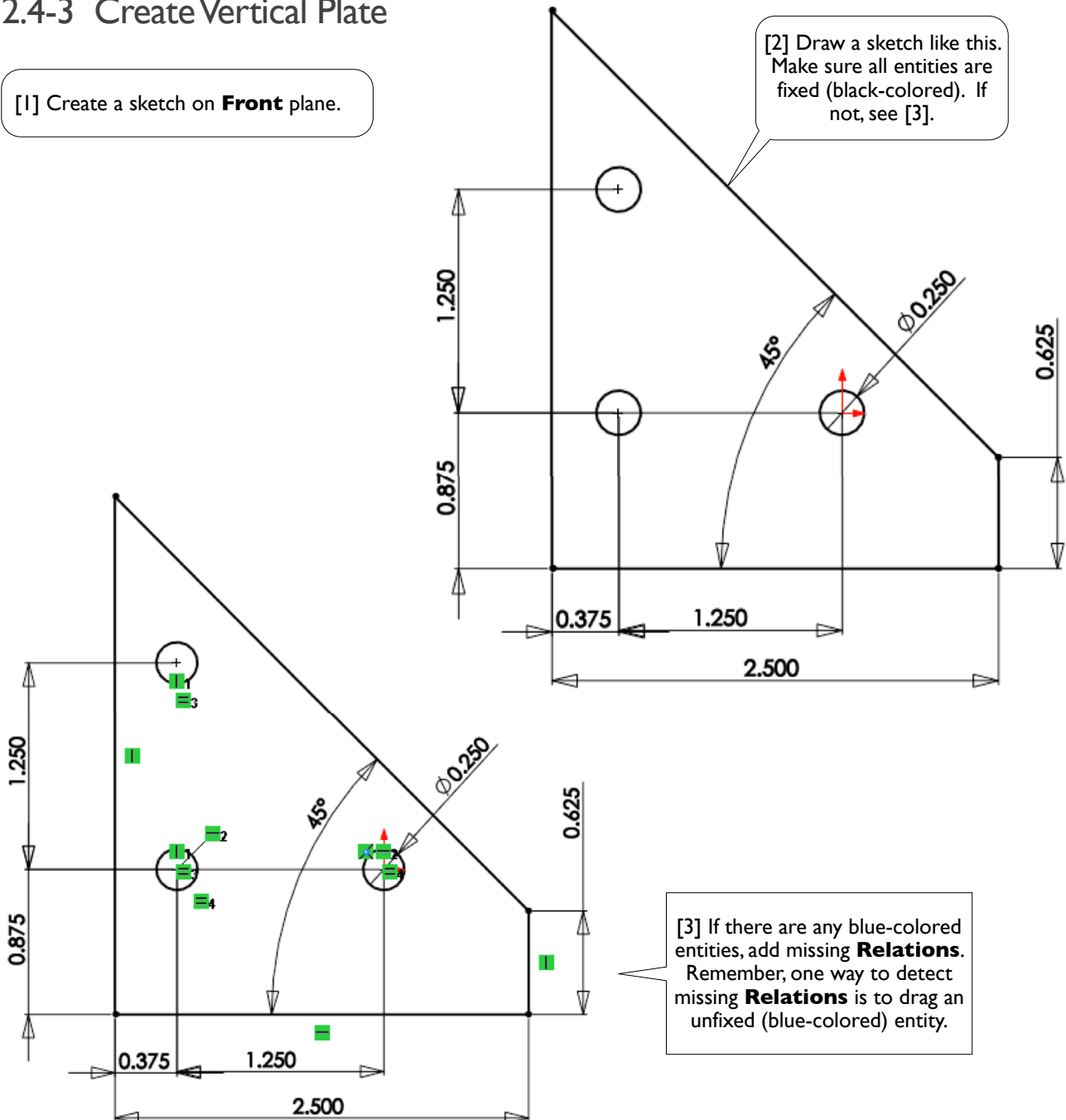
2.4-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system and with 3 decimal places for the length unit. #

2.4-3 Create Vertical Plate

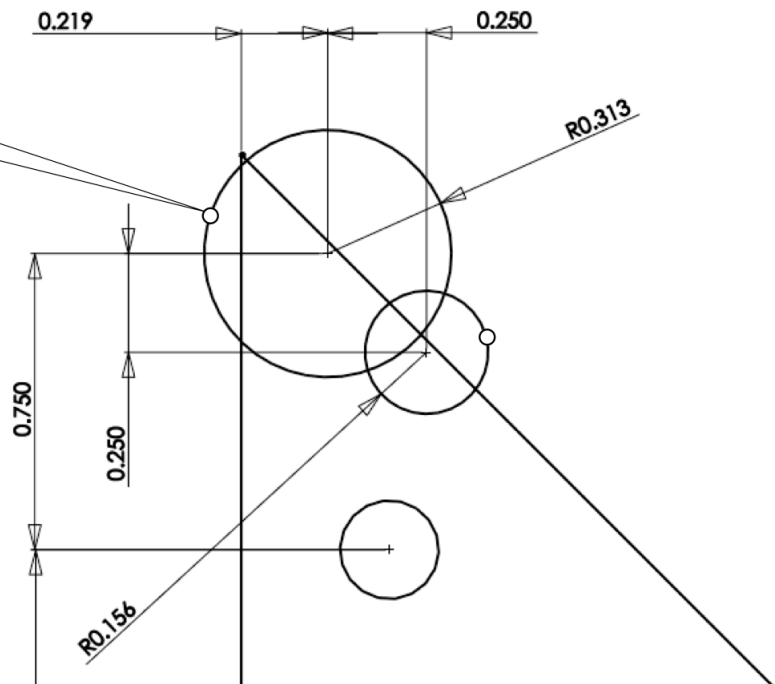
[1] Create a sketch on **Front** plane.

[2] Draw a sketch like this. Make sure all entities are fixed (black-colored). If not, see [3].

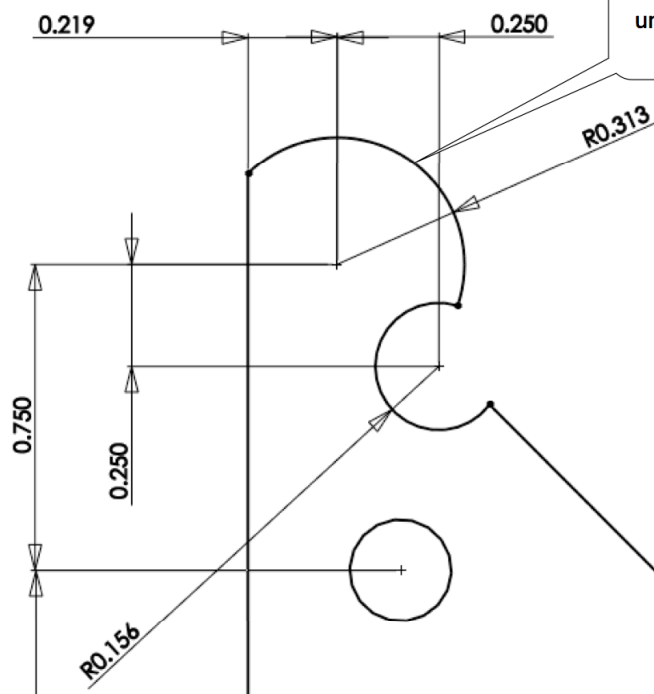


[3] If there are any blue-colored entities, add missing **Relations**. Remember, one way to detect missing **Relations** is to drag an unfixed (blue-colored) entity.

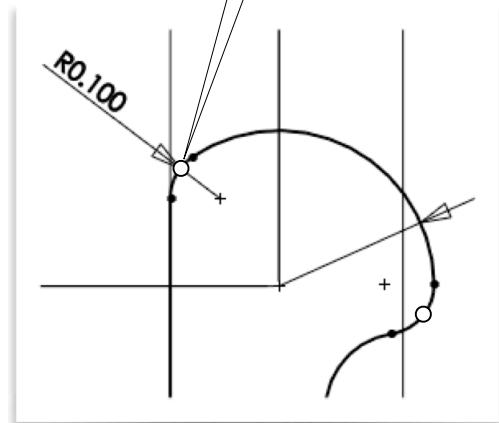
[4] Draw two circles like this. Make sure all entities are fixed.

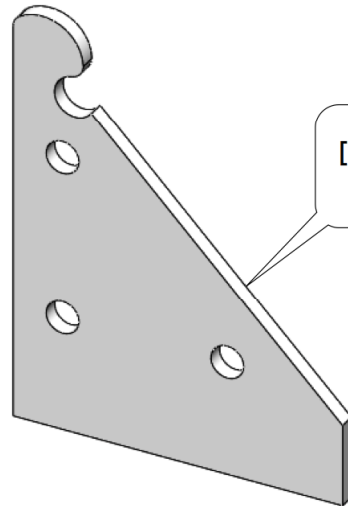


[5] Trim away unwanted segments.



[6] Use **Sketch Fillet** command to draw two fillets of the same radius of 0.1 inches. The two fillets may need to create separately.



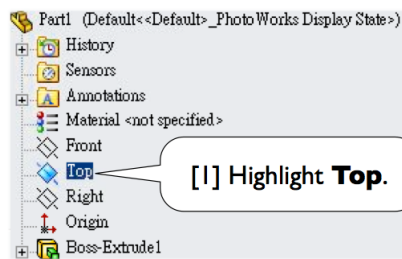
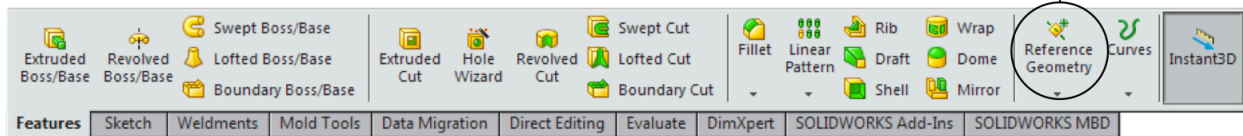


[7] Extrude the sketch
0.125 inches. #

Extruded
Boss/Base

2.4-4 Create Horizontal Plate

[2] Select **Reference
Geometry>Plane** from the
Features Toolbar.



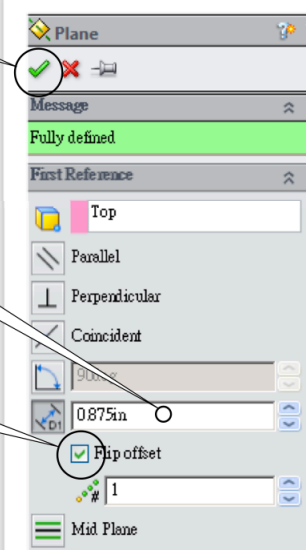
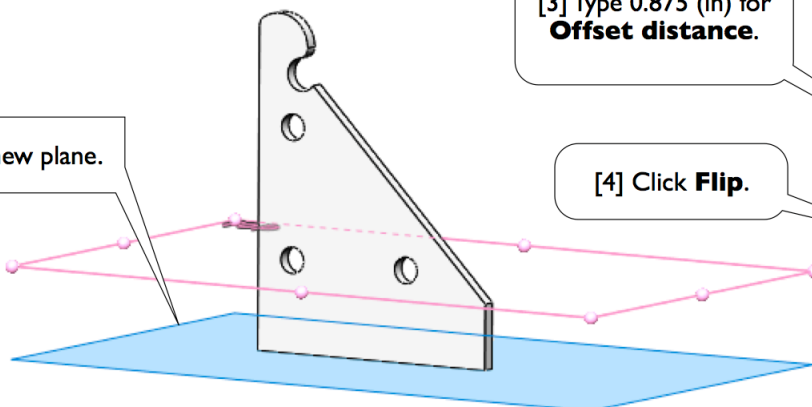
[1] Highlight **Top**.

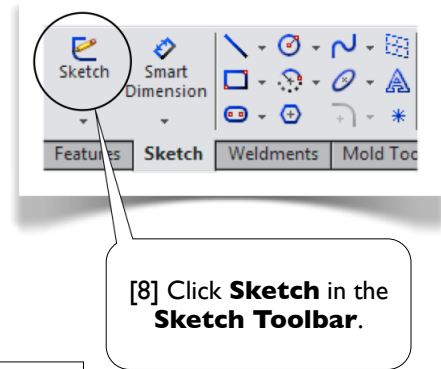
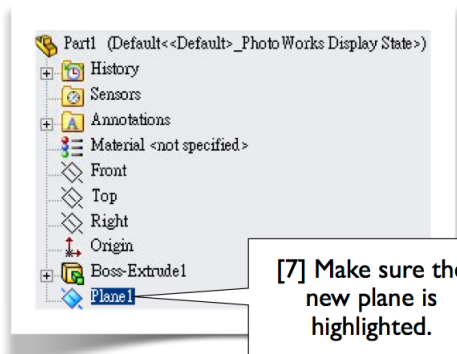
[6] Click **OK**.

[3] Type 0.875 (in) for
Offset distance.

[4] Click **Flip**.

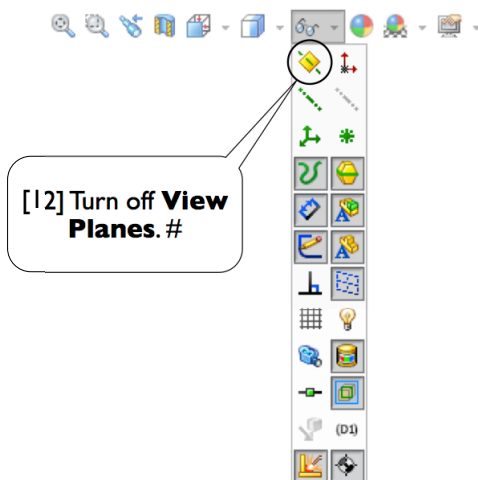
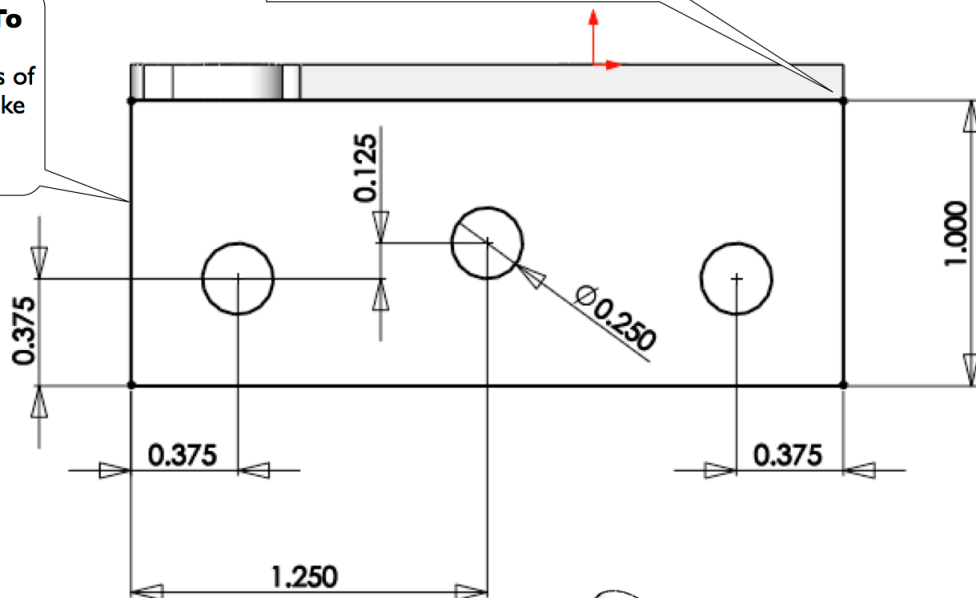
[5] The new plane.



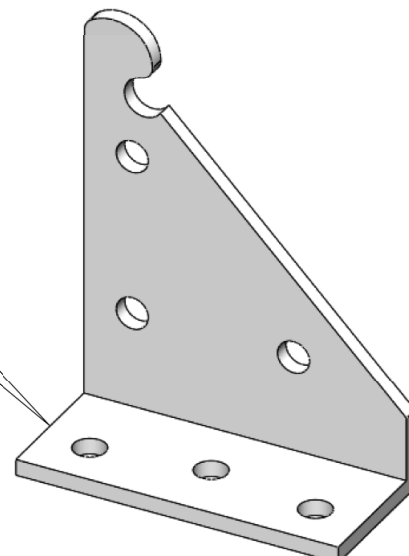
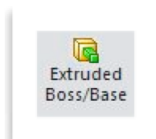


[9] Click **Normal To** and draw a sketch (including three circles of the same diameter) like this. Make sure all entities are fixed.

[10] Remember, one way to detect missing **Relations** is to drag an unfixed entity. You may need to add a **Coincident** relation here.



[11] Extrude (upward) 0.125 inches.



2.4-5 Create Fillet

[1] In the **Features Toolbar**, click **Fillet**.

[2] Select **Constant size**.

[3] Select the edge shown in [4].

[4] **Edge<1>**.

[5] Type 0.1 (in).

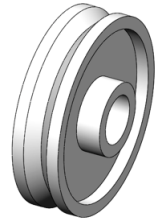
[6] Click **OK**.

[7] Save the part with the file name **Support**. Close the file and exit **SOLIDWORKS**.#

The diagram illustrates the process of creating a fillet on a 3D model of a bracket. The bracket consists of a vertical plate and a horizontal base, both featuring circular holes. A fillet is applied to the inner corner of the vertical plate. The Fillet dialog box is shown with the 'Constant size' option selected, 'Edge<1>' selected in the 'Items To Fillet' list, and the radius set to 0.100 in. The 'Profile' is set to 'Circular'.

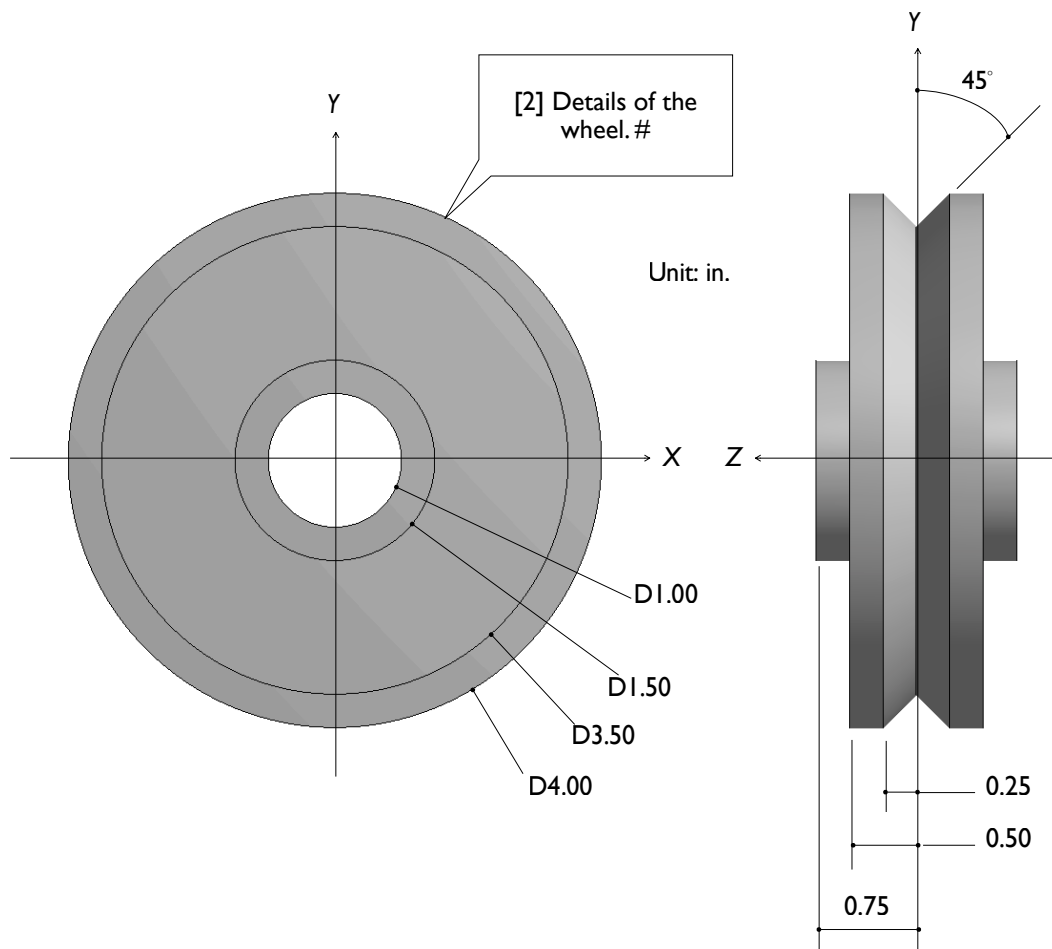
Section 2.5

Wheel



2.5-1 About the Wheel

[1] So far, we exclusively used **Extrude** command to create 3D solids. In this section, we introduce another command to create 3D solids: **Revolve**, which takes a sketch as the profile and revolves about an axis to create a 3D solid body. We'll create a 3D solid model for a wheel [2]. The wheel is axisymmetric. An axisymmetric body always can be created by drawing a **profile** then revolving about its **axis of revolution**.

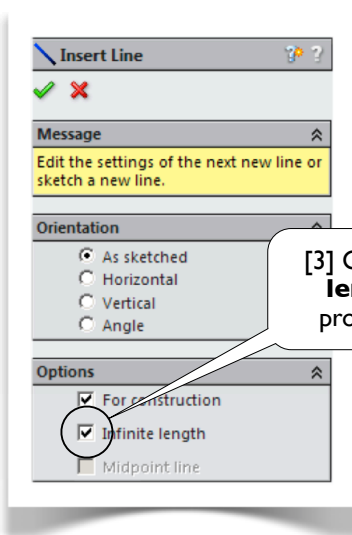


2.5-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 2 decimal places for the length unit. #

2.5-3 Create a Sketch for the Profile

[1] Create a sketch on **Front** plane.

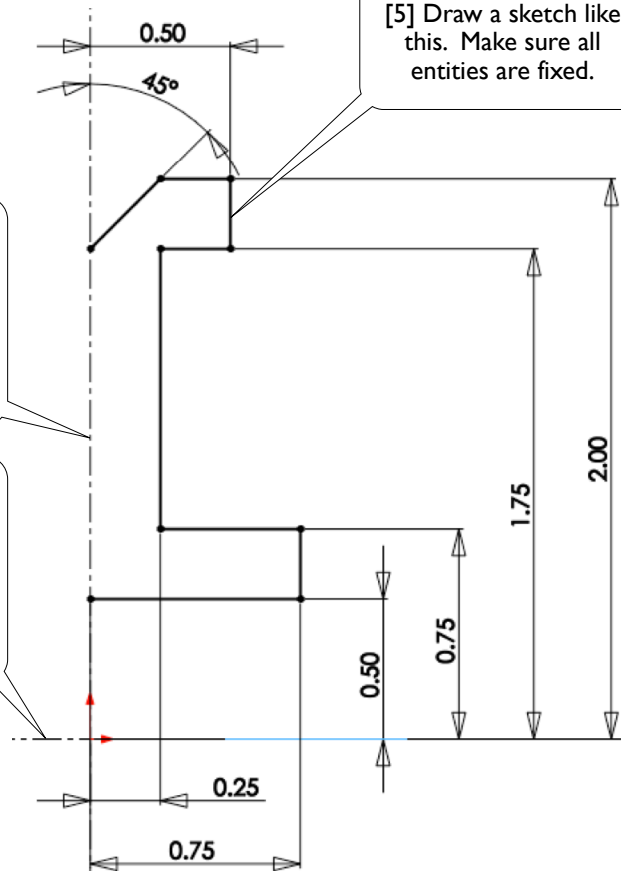


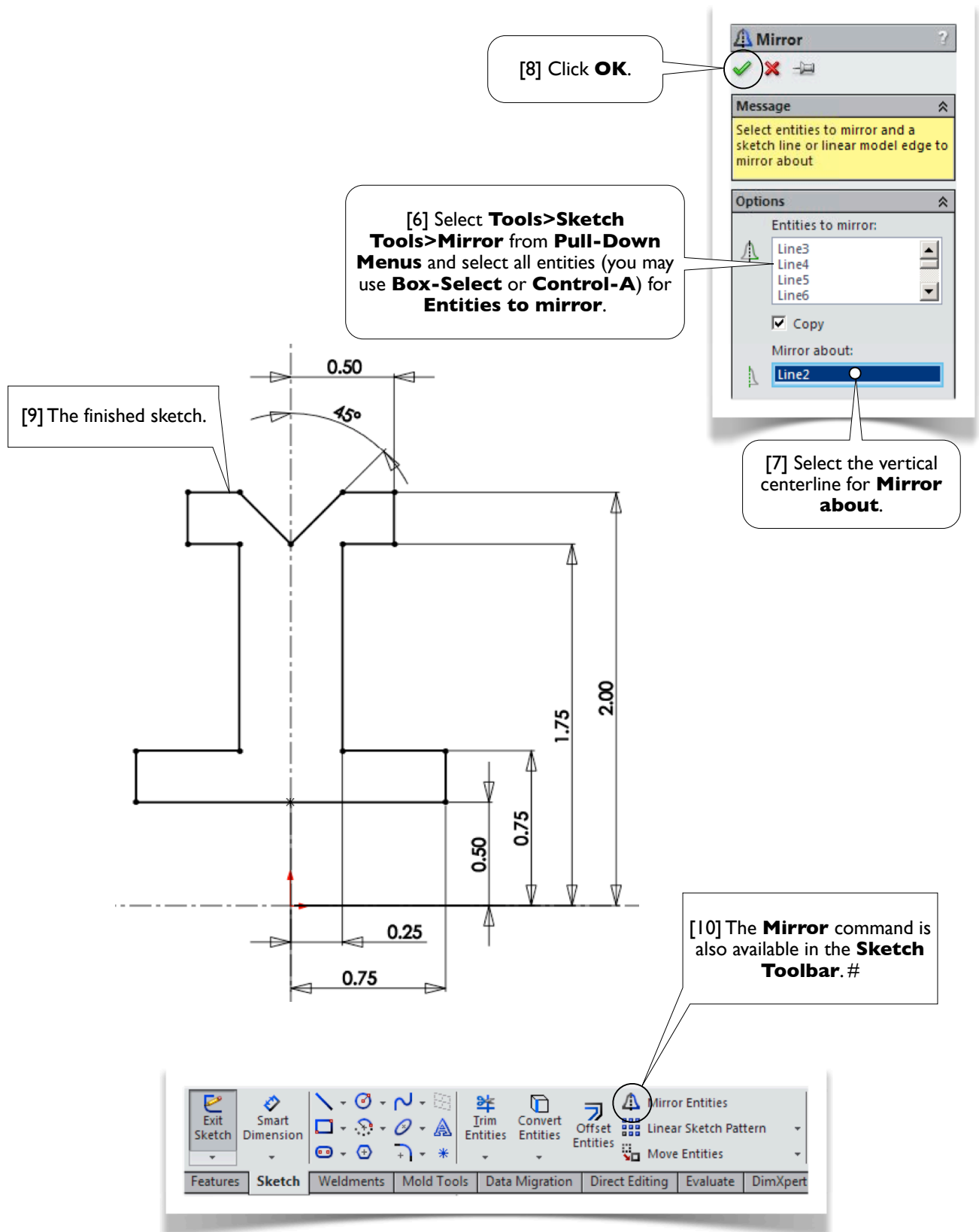
[3] Click **Infinite length** in the properties box.

[4] Draw a vertical centerline of infinite length passing through the origin. This centerline will be used as the axis of symmetry.

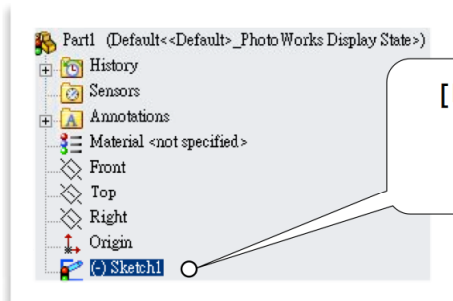
[2] Draw a horizontal centerline of infinite length (see [3]) passing through the origin. This centerline will be used as the axis of revolution.

[5] Draw a sketch like this. Make sure all entities are fixed.

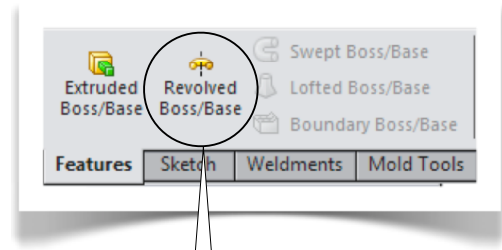




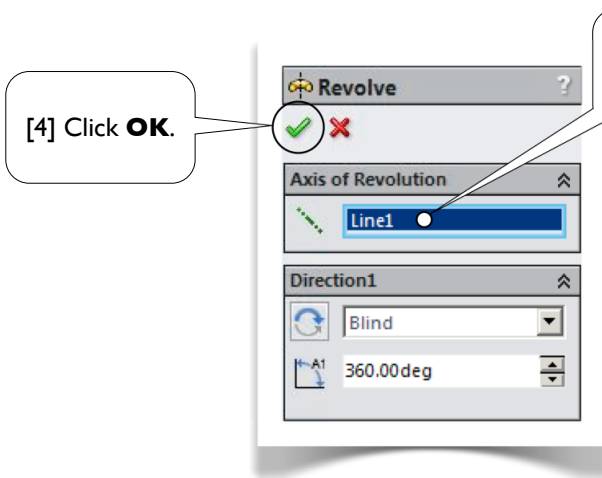
2.5-4 Revolve the Sketch



[1] While the sketch is highlighted, select **Insert>Boss/Base>Revolve...**



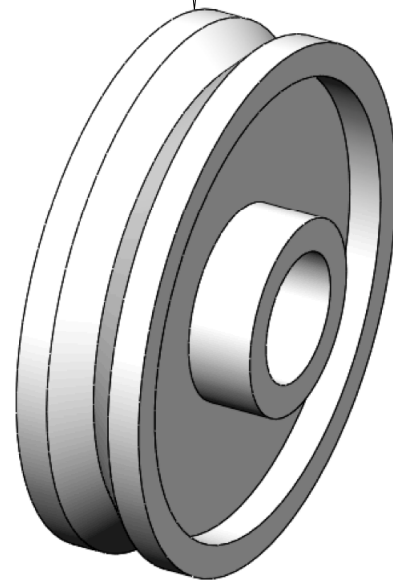
[2] The **Revolve** command is also available in the **Features Toolbar**.



[4] Click **OK**.

[3] Select the horizontal centerline.

[5] The finished 3D model.



[6] Save the part with the file name **Wheel**. Close the file and exit **SOLIDWORKS**.#

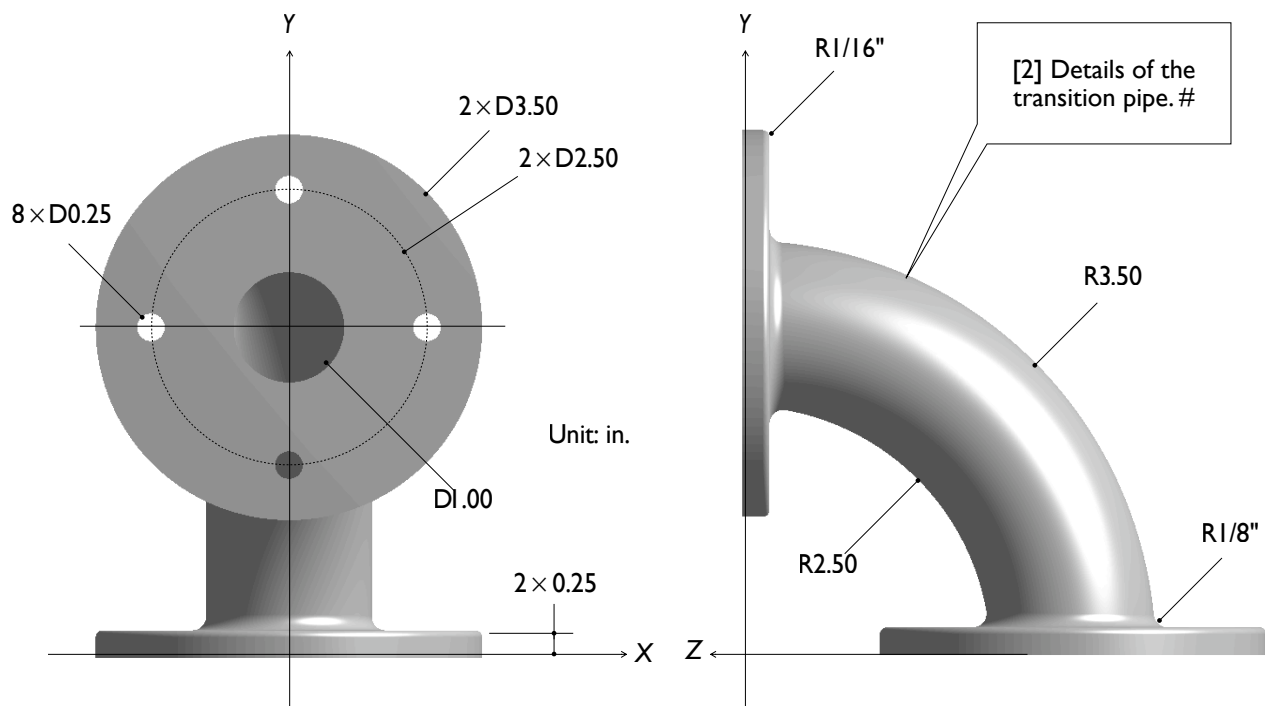
Section 2.6

Transition Pipe



2.6-1 About the Transition Pipe

[1] In this section, we introduce another command to create 3D solids: **Sweep**, which takes a sketch as the **path** and another sketch as the **profile**; the **profile** then "sweeps" along the **path** to create a 3D solid body. In this exercise, we'll create a 3D solid model for a transition pipe, which is used to connect two pipe segments.



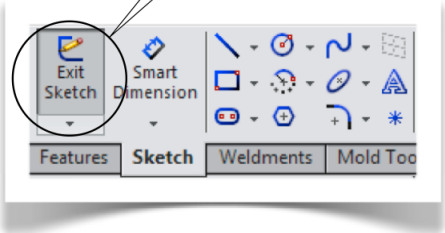
2.6-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 2 decimal places for the length unit. #

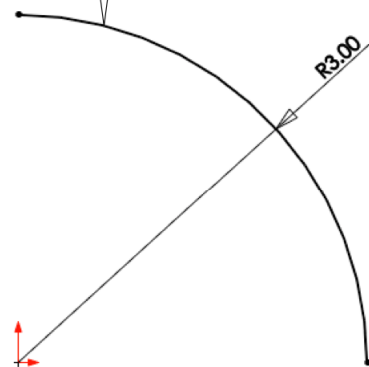
2.6-3 Create a Sketch for the **Path**

[1] Create a sketch on **Front** plane.

[3] Click **Exit Sketch** in the **Sketch Toolbar**. #

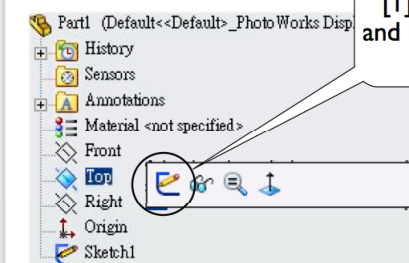


[2] Draw a sketch like this. This sketch will be used as a sweeping **path**. Note that, each end point aligns with the origin either vertically or horizontally.



2.6-4 Create a Sketch for the **Profile**

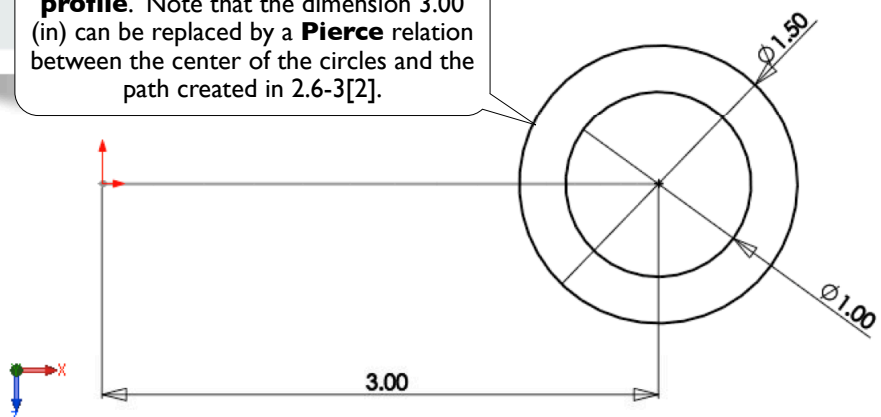
[1] Right-click **Top** plane and select **Sketch** to create a second sketch.



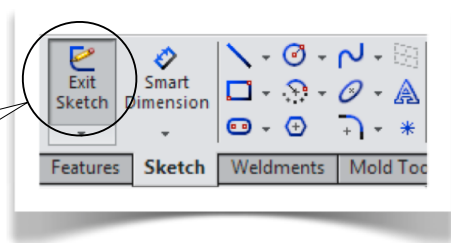
[2] In the **Standard Views** **Toolbar**, click **Normal To**.



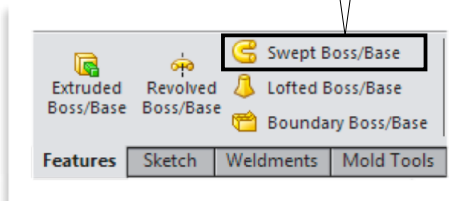
[3] Draw two concentric circles like this. This sketch will be used as a sweeping **profile**. Note that the dimension 3.00 (in) can be replaced by a **Pierce** relation between the center of the circles and the path created in 2.6-3[2].



[4] Click Exit Sketch in the **Sketch Toolbar**. #

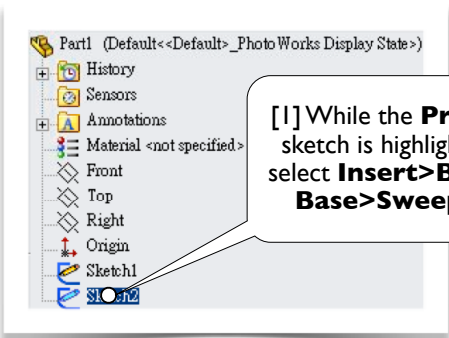


[2] The **Sweep** command is also available in the **Features Toolbar**.

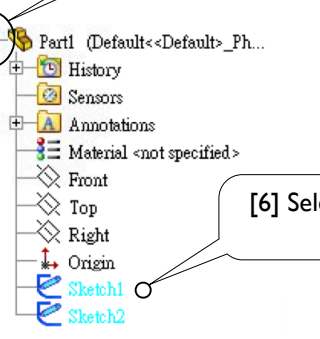


2.6-5 Create the Curved Pipe

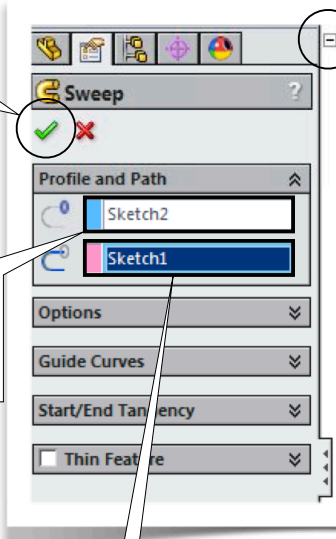
[1] While the **Profile** sketch is highlighted, select **Insert>Boss/Base>Sweep...**



[5] In the **Graphics Area**, click "+" sign to expand the **Part Tree**; the "+" sign becomes "-" sign.



[7] Click **OK**.

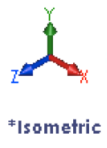
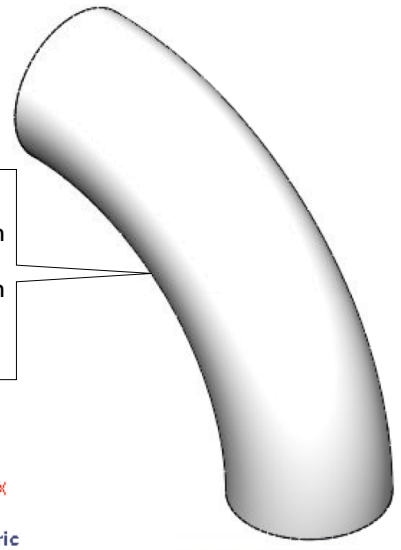


[3] The **profile** sketch (**Sketch2**) is pre-selected.

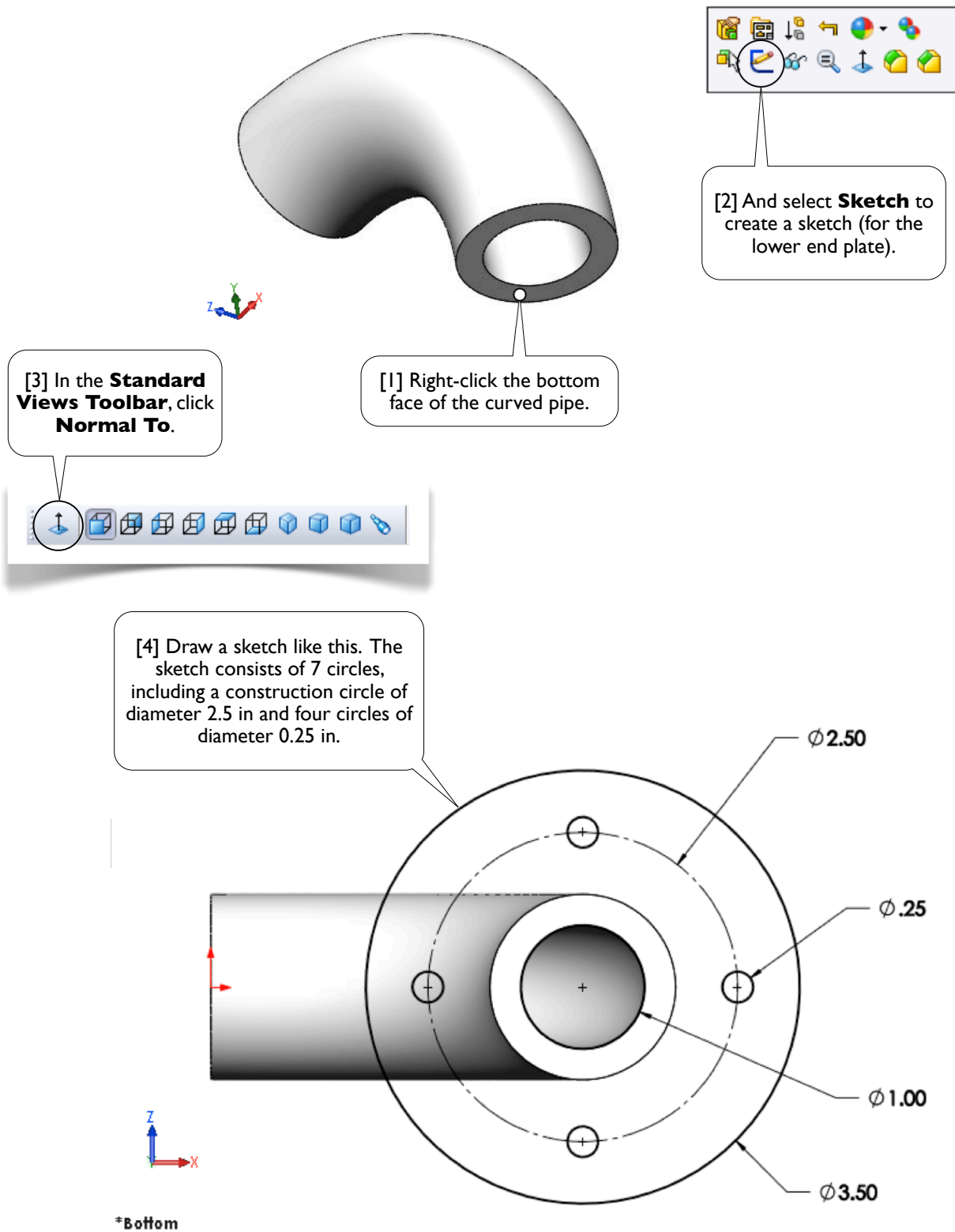
[6] Select the **Path** sketch (**Sketch1**).

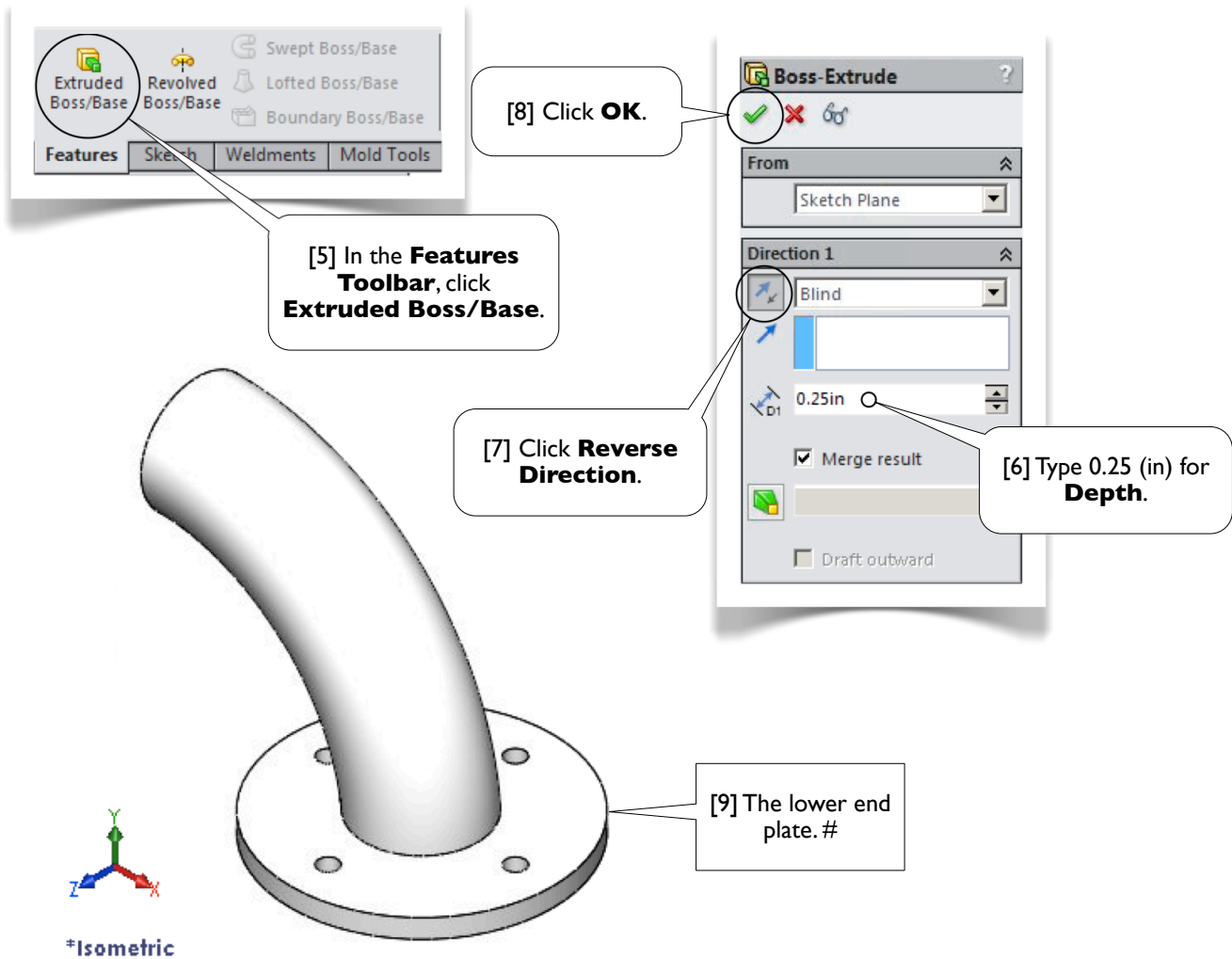
[4] Click to activate **Path** box.

[8] The curved pipe. Note that the curved pipe also can be created by **Revolving** the **Profile** 90 degrees with an axis coincident with the **Z-axis**. #



2.6-6 Create the Lower End Plate

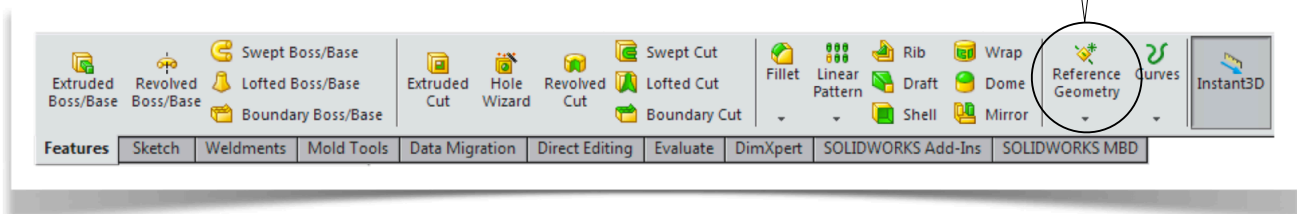


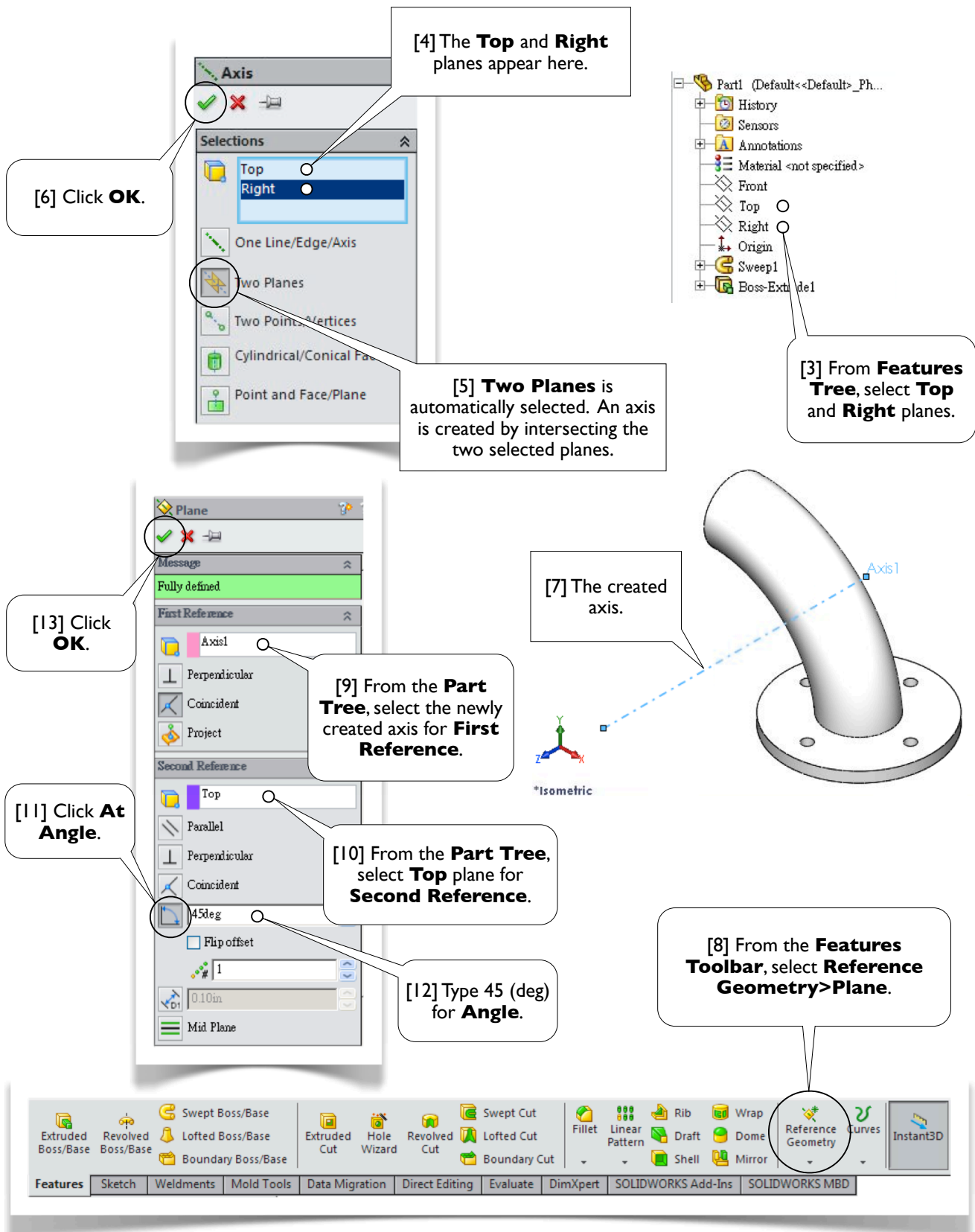


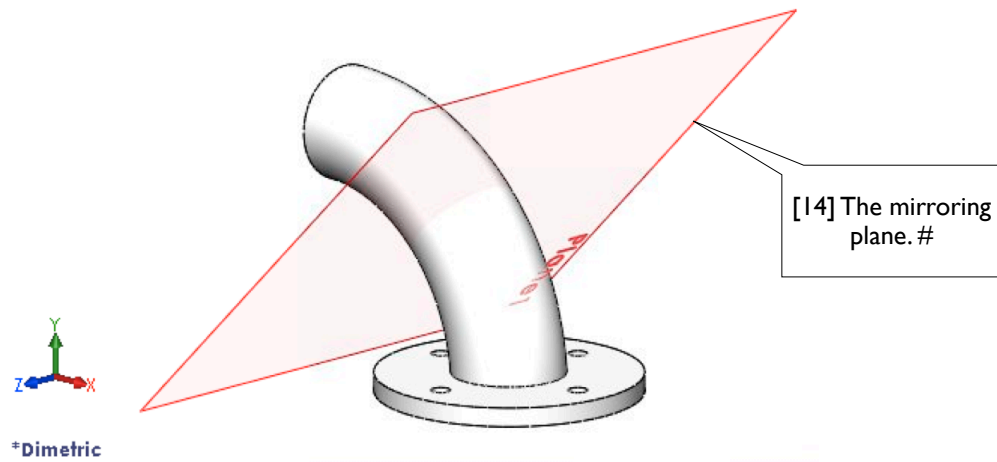
2.6-7 Create a Mirroring Plane

[1] Next, we want to create the upper end plate by using **Mirror** command. The mirroring plane will be created by rotating the **Top** plane 45 degrees about an axis coincident with the Z-axis. First, we create the axis.

[2] From **Features Toolbar**, select **Reference Geometry>Axis**.







2.6-8 Create the Upper End Plate

[1] Make sure the newly created plane is highlighted.

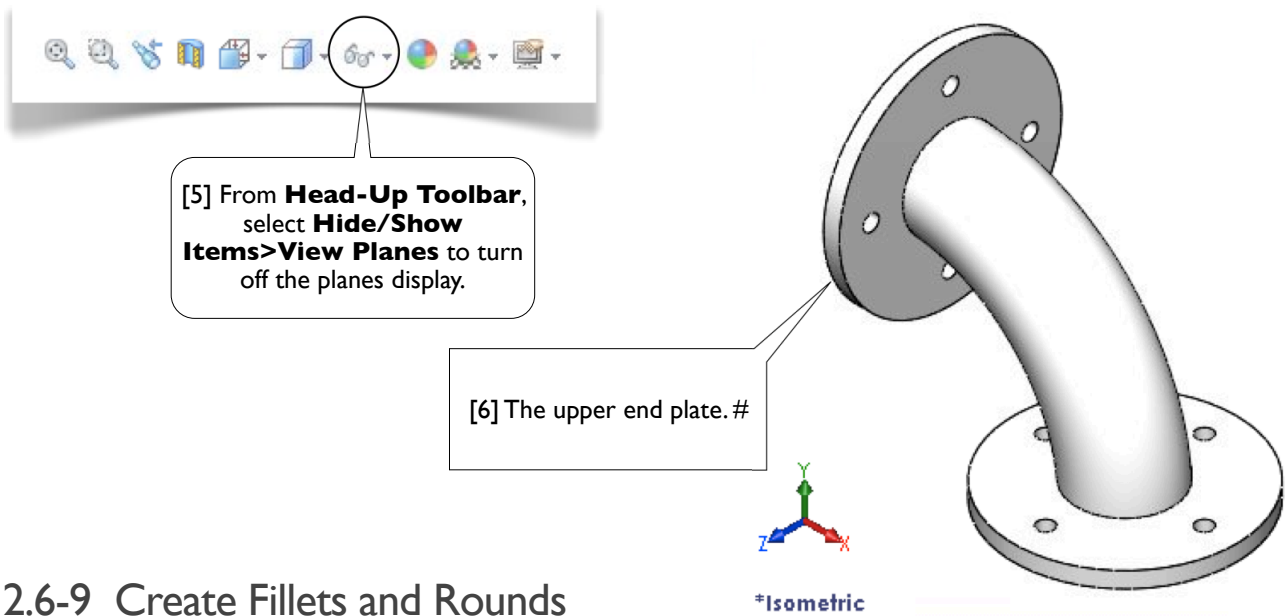
[2] In **Features Toolbar**, click **Mirror**.

[3] From the **Part Tree** (or from the **Graphics Area**) select the lower end plate (**Boss-Extrude1**).

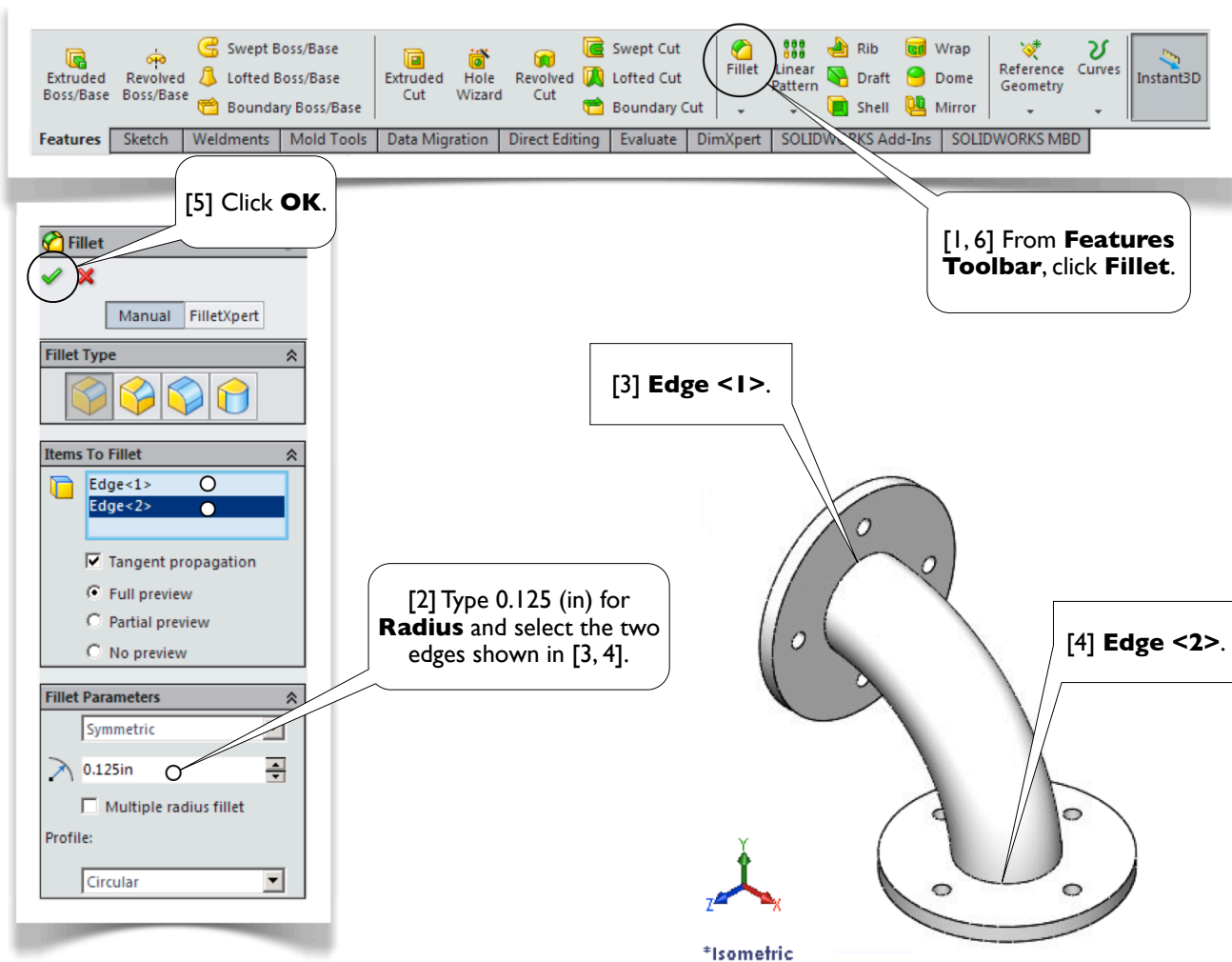
[4] Click **OK**.

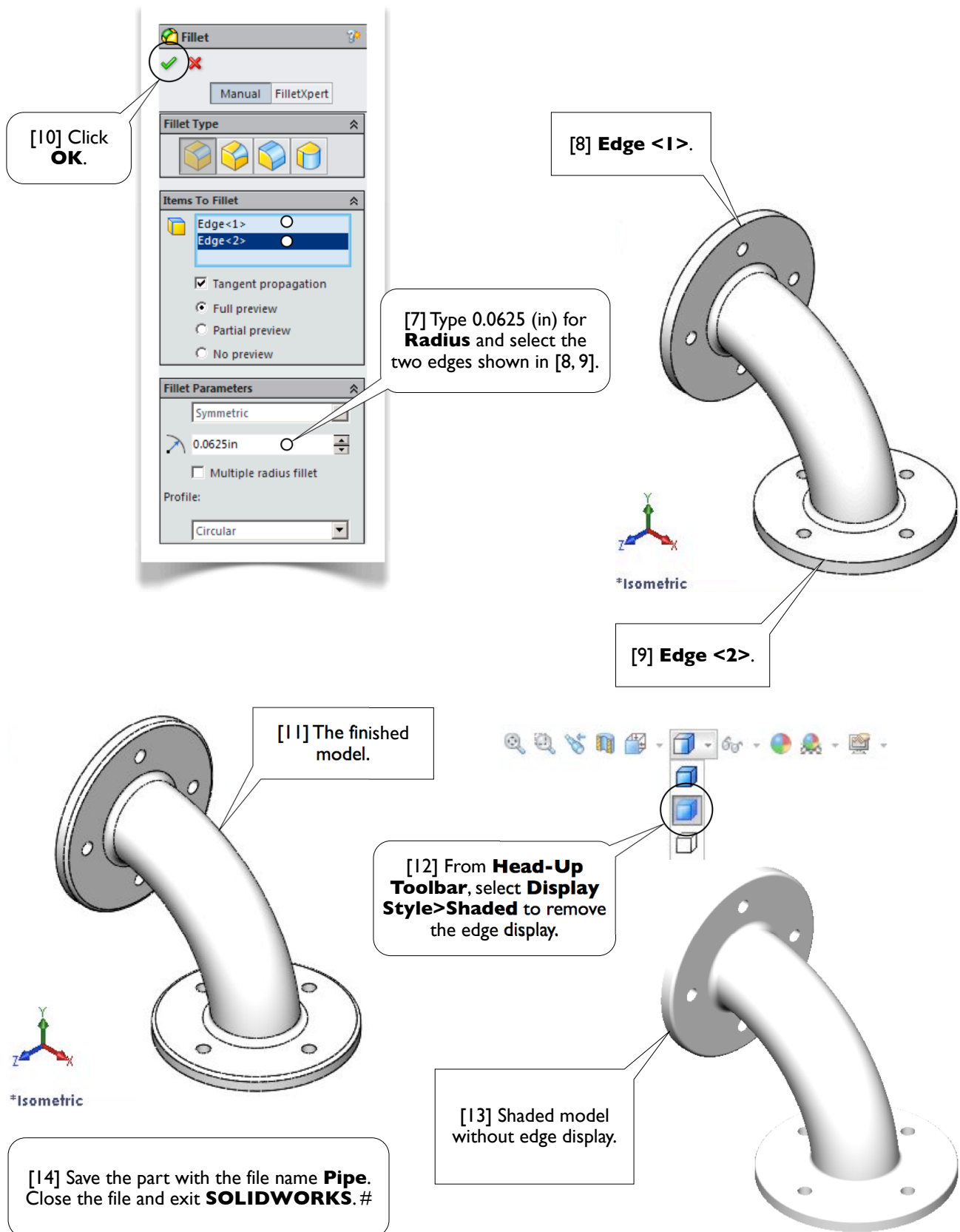
The **Mirror** dialog box shows:

- Mirror Face/Plane:** Plane1
- Features to Mirror:** Boss-Extrude1



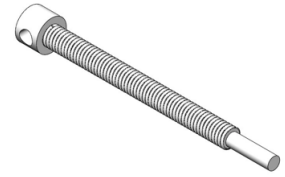
2.6-9 Create Fillets and Rounds





Section 2.7

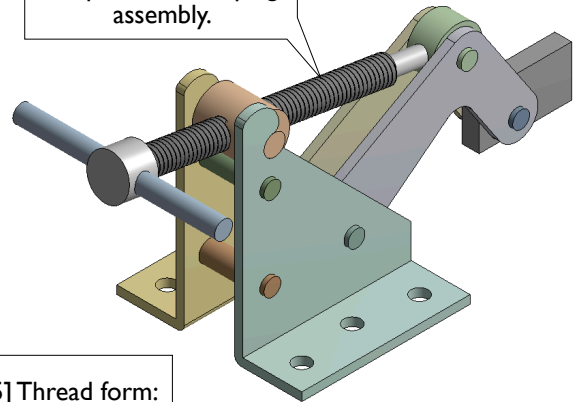
Threaded Shaft



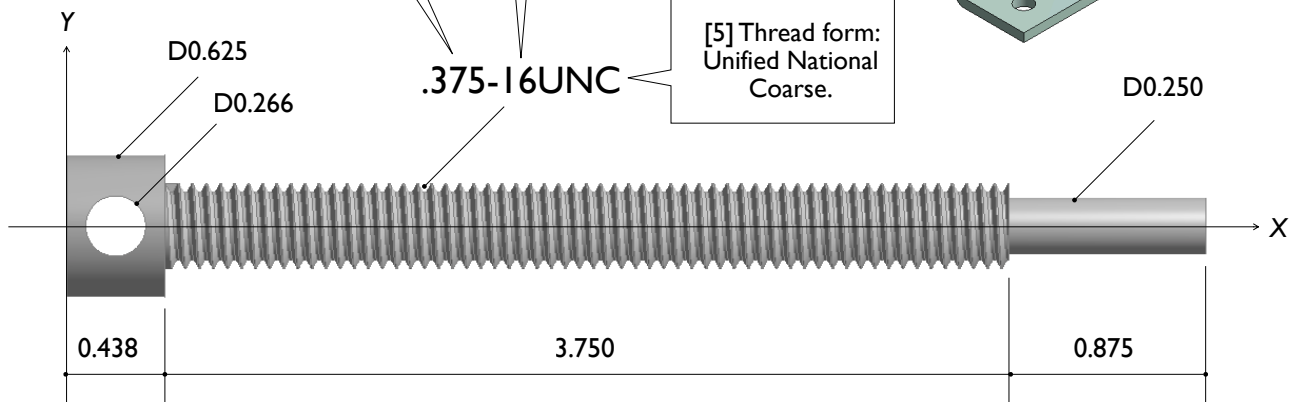
2.7-1 About the Threaded Shaft

[1] The threaded shaft is a part of the clamping mechanism mentioned in Sections 1.1 and 2.4 [2]. In this exercise, we will create a 3D solid model for the threaded shaft.

[2] The threaded shaft is a part of a clamping assembly.



Unit: in



[6] Details of the threads. #

$$d = 0.375 \text{ in}$$

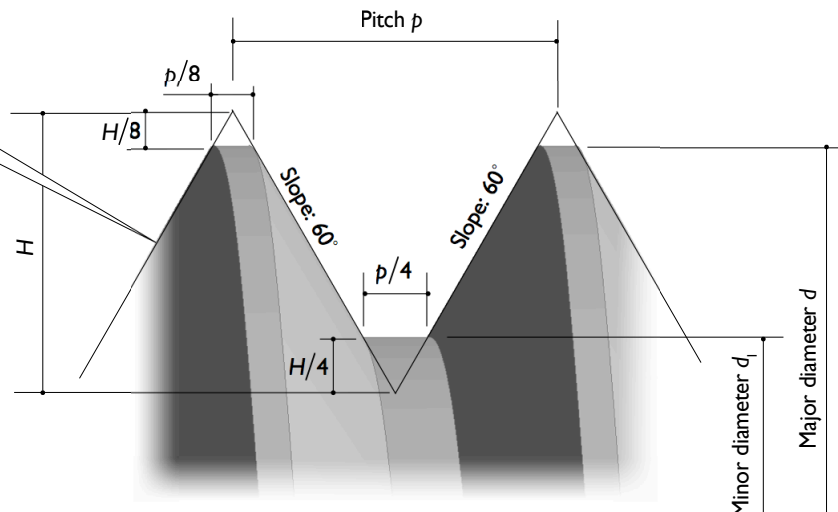
$$p = 0.0625 \text{ in}$$

$$H = (\sqrt{3}/2)p = 0.0541266 \text{ in}$$

$$d_1 = d - \frac{5H}{8} \times 2 = 0.307342 \text{ in}$$

$$\frac{p}{4} = 0.015625 \text{ in}$$

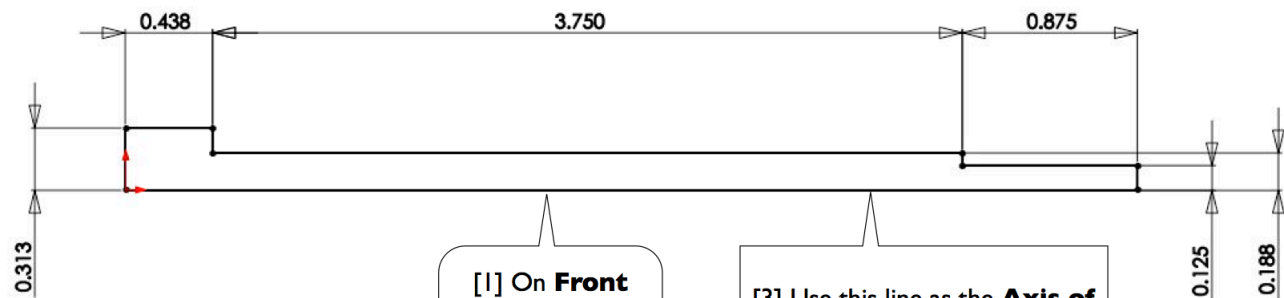
$$\frac{p}{8} = 0.0078125 \text{ in}$$



2.7-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit. #

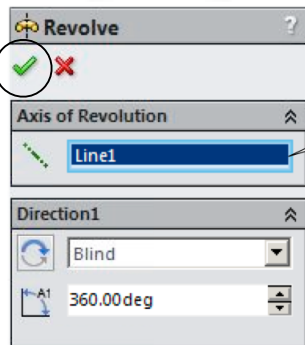
2.7-3 Create a Shaft Base



[1] On **Front** plane, draw a sketch like this.

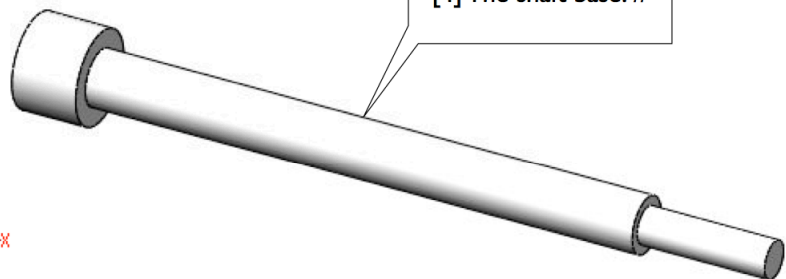
[3] Use this line as the **Axis of Revolution**.

Revolved Boss/Base



[2] **Revolve** (2.5-4[2], page 65) the sketch 360 degrees to create the shaft base. Use the bottom line of the sketch as the **Axis of Revolution** [3].

[4] The shaft base. #



*Trimetric

2.7-4 Create Threads

The diagram shows a 3D perspective view of a shaft with a threaded section of length 3.750. A coordinate system (X, Y, Z) is shown at the end of the shaft. Below the perspective view is a 2D cross-section sketch of the thread profile. The sketch shows a trapezoidal profile with a top width of 0.016 and a total height of 0.154. The flanks are at 60 degrees. A coordinate system (X, Y, Z) is also shown for the sketch.

[1] On the **Front** plane, draw a single line of length 3.75 inches like this. Remember to click **Exit Sketch**. This sketch will be used as the sweeping **Path**.

[2] On the **Front** plane, draw a sketch of trapezoid like this. Remember to click **Exit Sketch**. This sketch will be used as the sweeping **Profile**.

[3] With **Sketch3** (created in [2]) highlighted, from **Features Toolbar**, click **Swept Cut**.

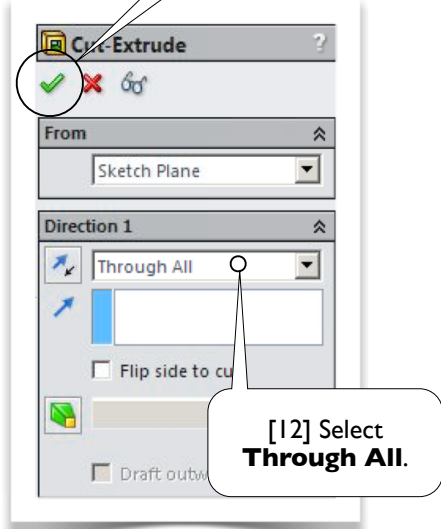
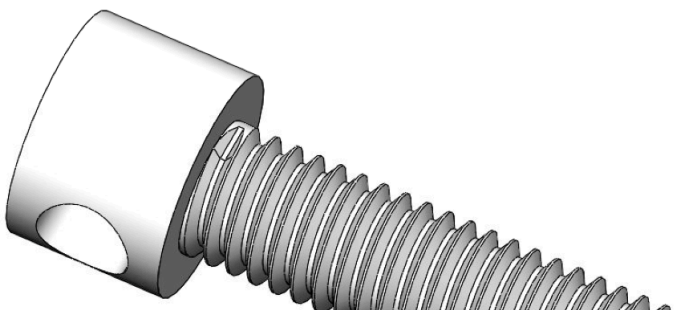
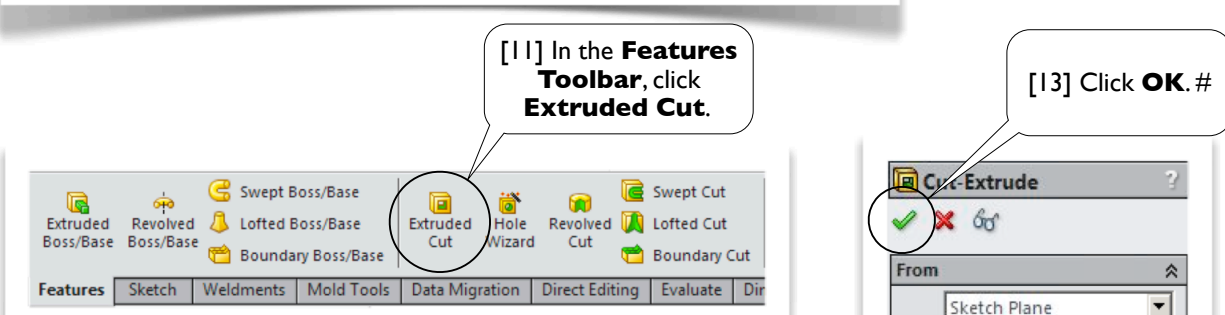
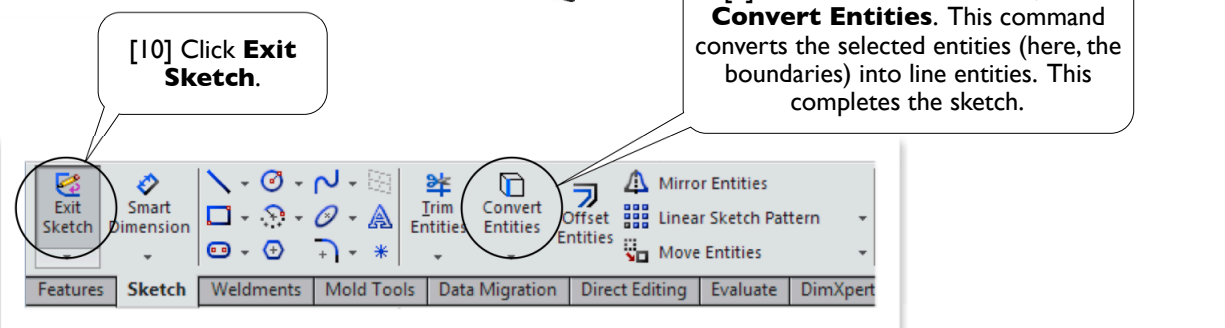
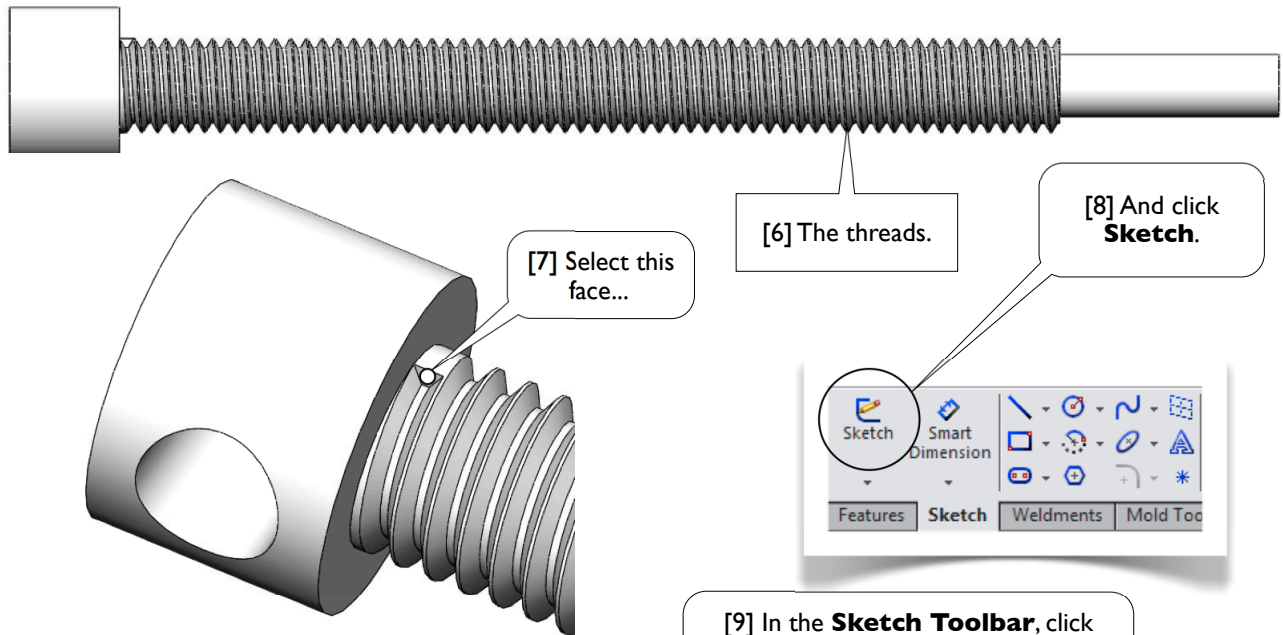
[4] **Sketch3** is used as **Profile**.

[5] From the **Part Tree**, select the **Sketch2** (created in [1]) for **Path** and set up other parameters like this. Note that the number of turns (60) is calculated by $3.75/0.0625$, where 0.0625 (in) is the thread pitch. Click **OK**.

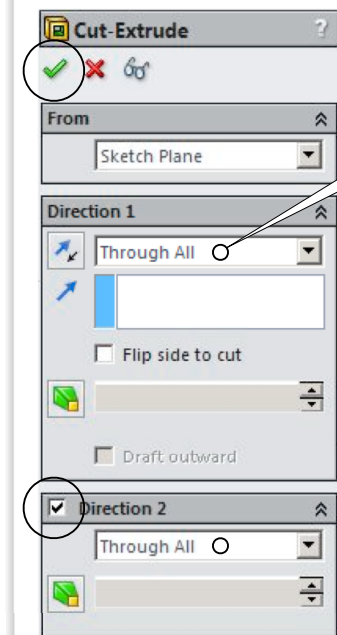
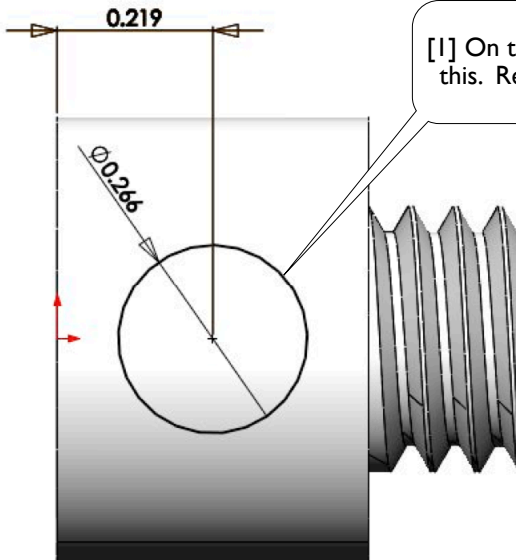
The **Features Toolbar** shows the **Swept Cut** button highlighted.

The **Cut-Sweep** dialog box shows the following settings:

- Profile and Path:**
 - Profile:** Sketch3
 - Path:** Sketch2
- Options:**
 - Orientation/twist type:** Twist Along Path
 - Define by:** Turns
 - Turns:** 60.000
 - Merge tangent faces:** ☐
 - Show preview:** ☒



2.7-5 Create a Hole



[2] With the sketch highlighted, from the **Features Toolbar**, select **Extruded Cut** and set up the parameters like this.



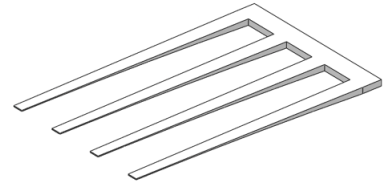
*Trimetric

[4] Save the part with the file name **Shaft**. Close the file and exit **SOLIDWORKS**. #

[3] The finished model.

Section 2.8

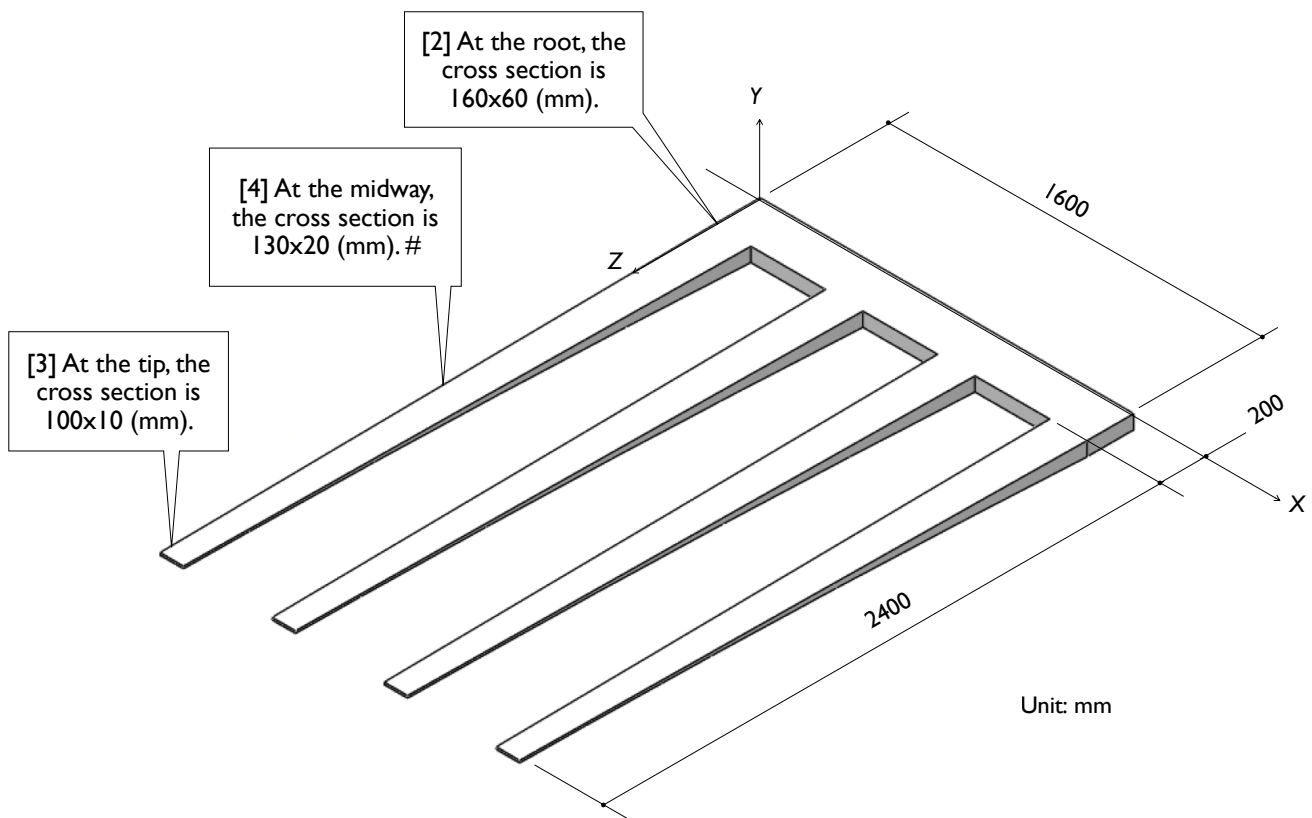
Lifting Fork



2.8-1 About the Lifting Fork

[1] The lifting fork is used in an **LCD** (liquid crystal display) manufacturing factory to handle glass panels. In this section, we will create a 3D solid model for the lifting fork.

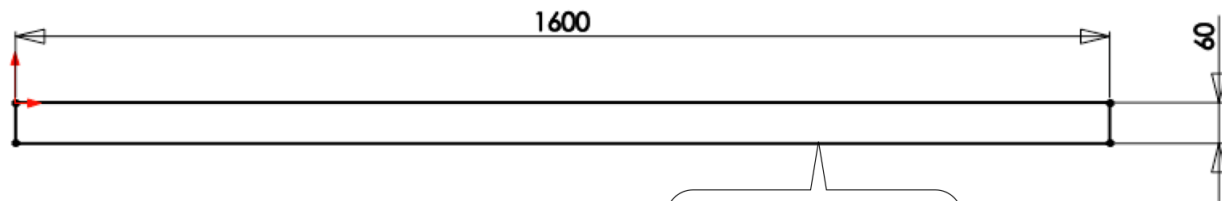
The cross sections of the prongs (fingers) are not uniform along the length [2, 3, 4]. The **Extrude** command or **Sweep** command can not be used to create the prongs. This exercise introduces a new command to create 3D solids: **Loft**, which takes a series of profiles and creates a 3D solid that fits through these profiles.



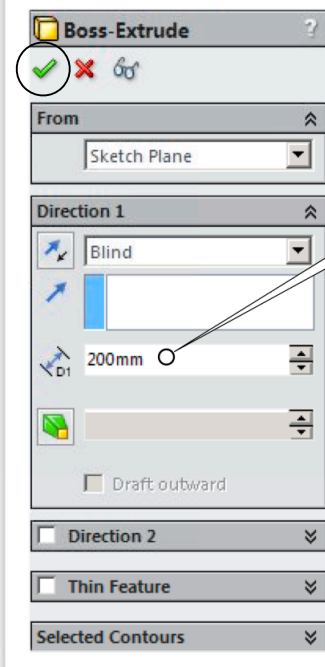
2.8-2 Start Up

[1] Launch **SOLIDWORKS** and create a new part. Set up **MMGS** unit system with zero decimal places for the length unit. #

2.8-3 Create a Transversal Beam

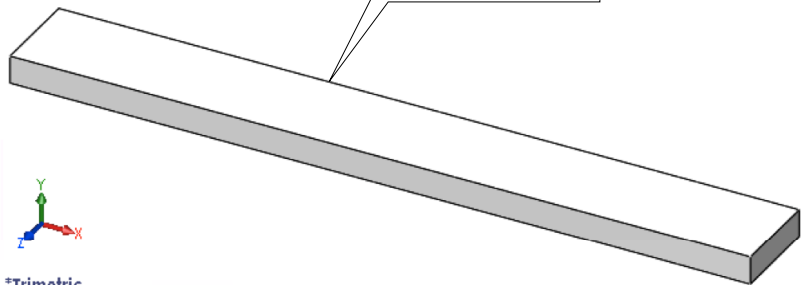


[1] On **Front** plane, draw a sketch like this.



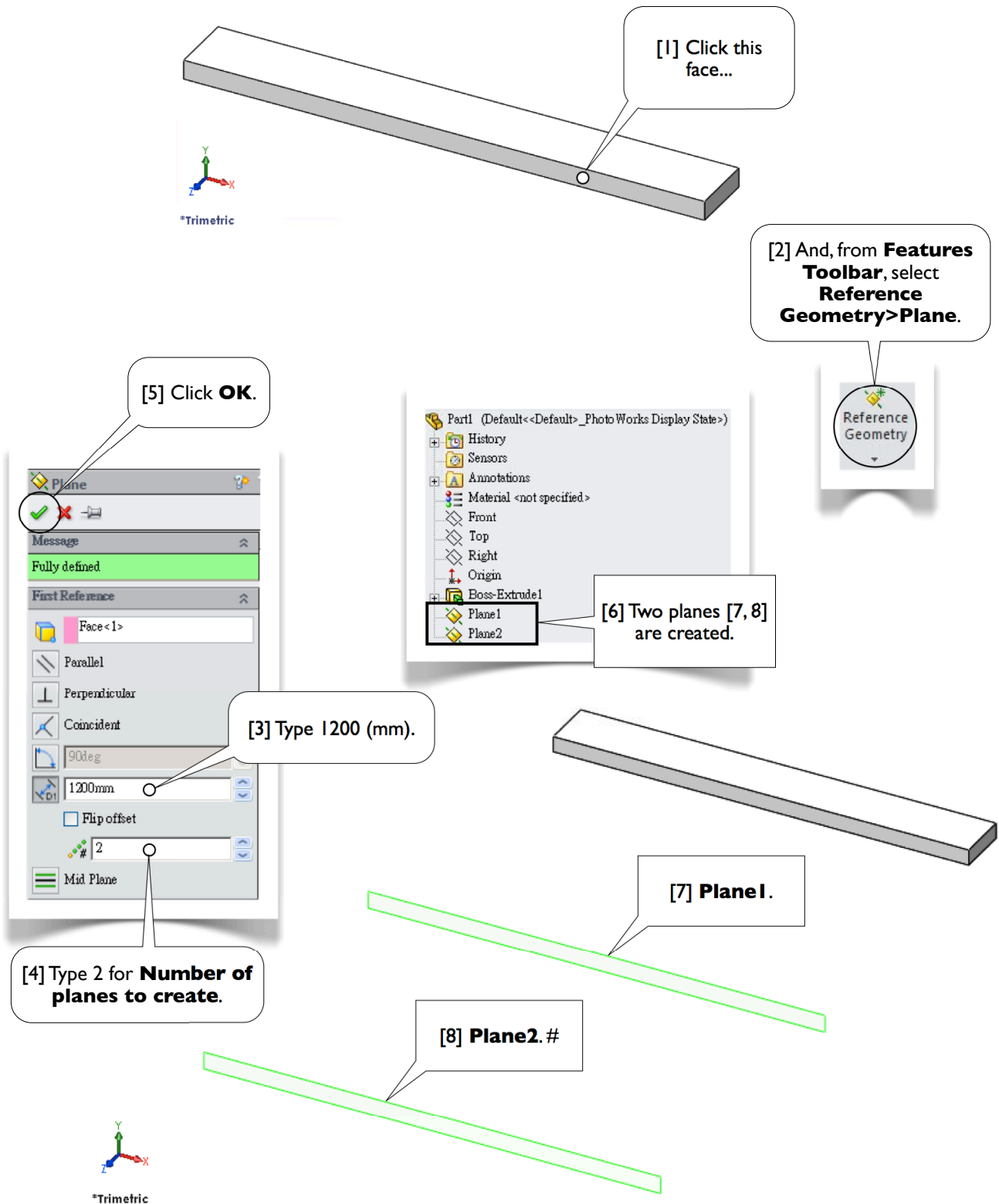
[2] **Extrude** the sketch 200 mm to create the transversal beam.

[3] The transversal beam. #

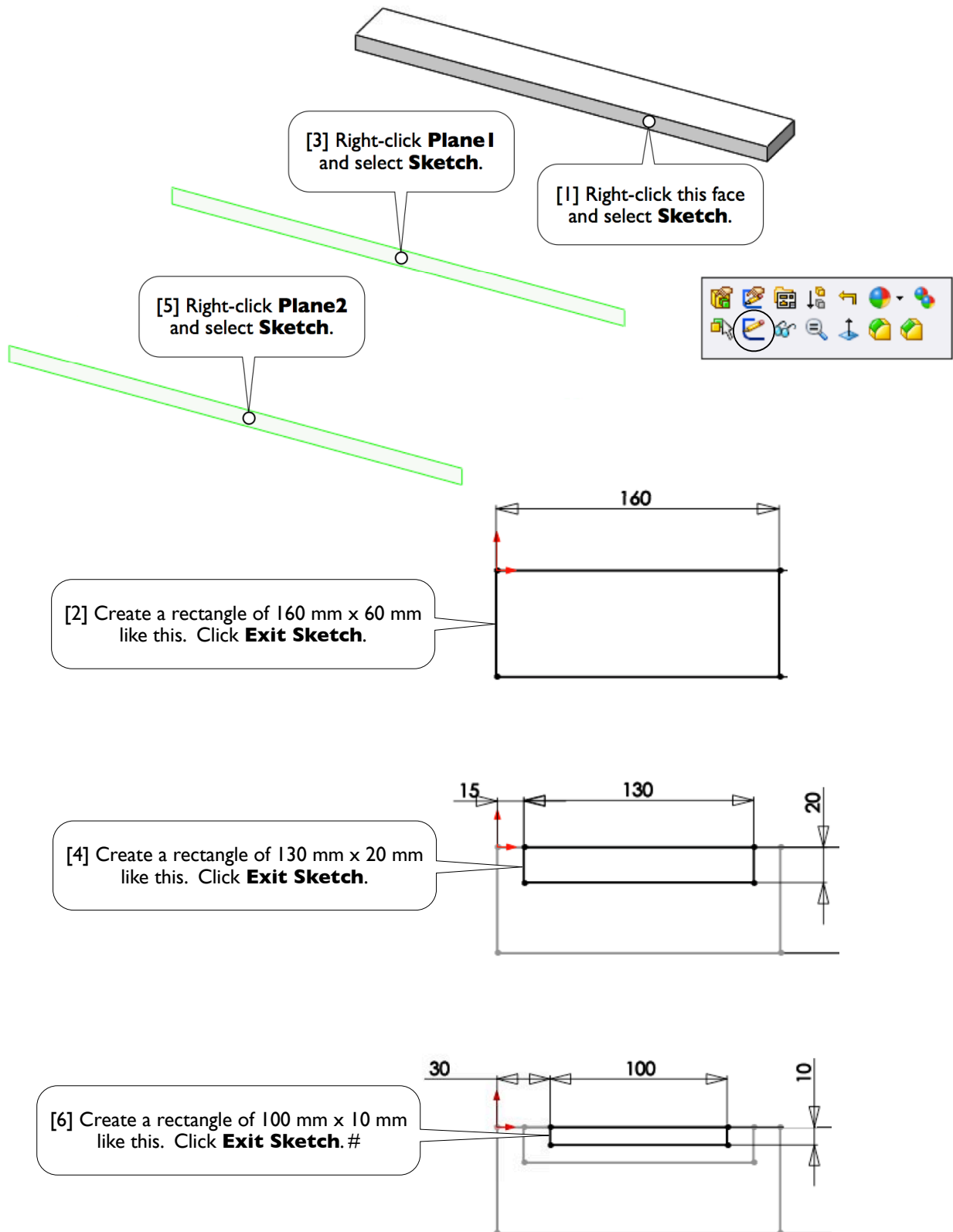


*Trimetric

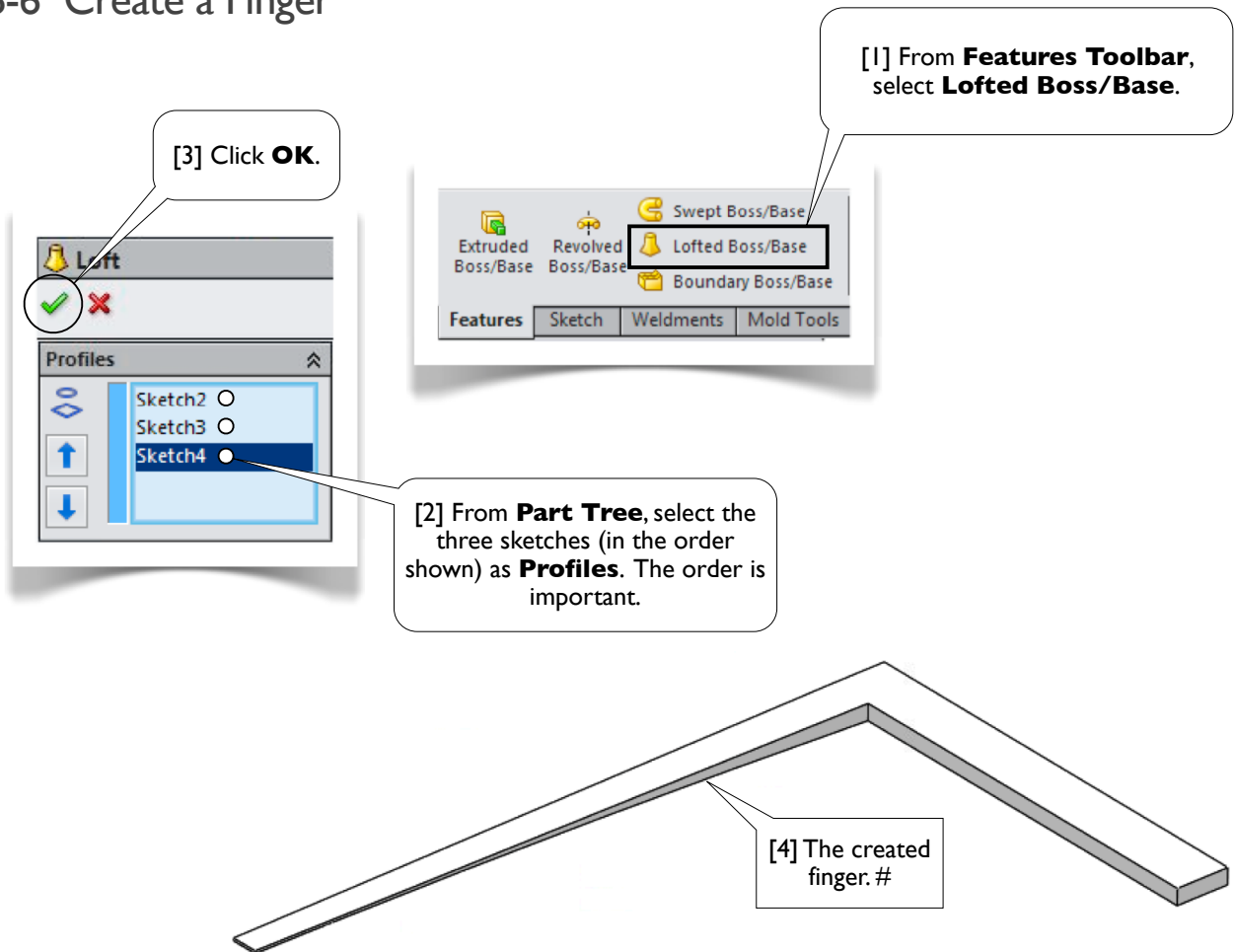
2.8-4 Create Two Planes



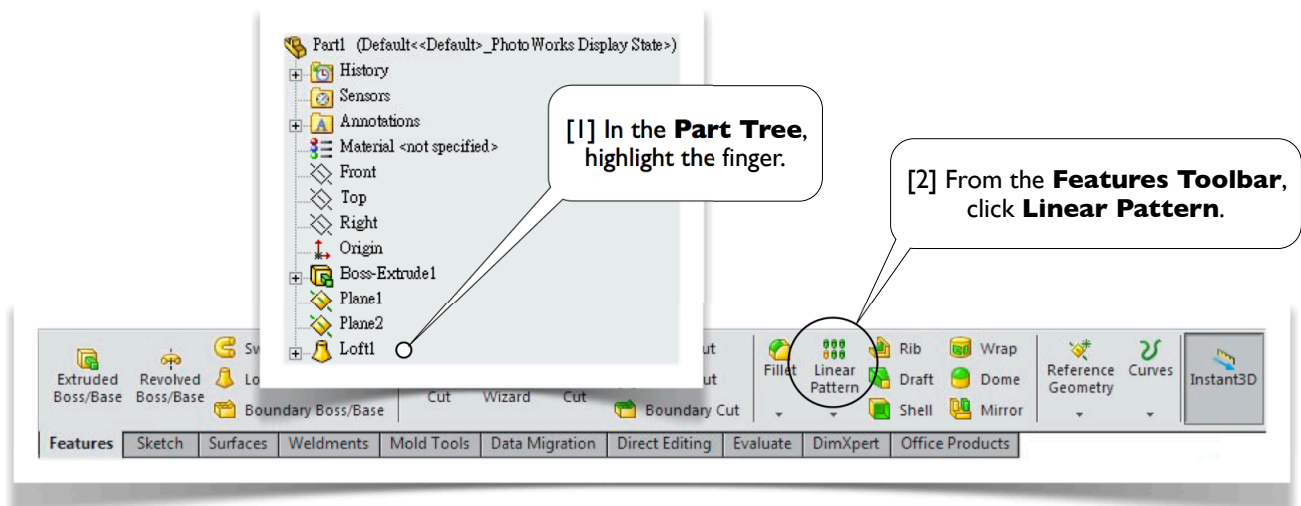
2.8-5 Sketch Three Profiles

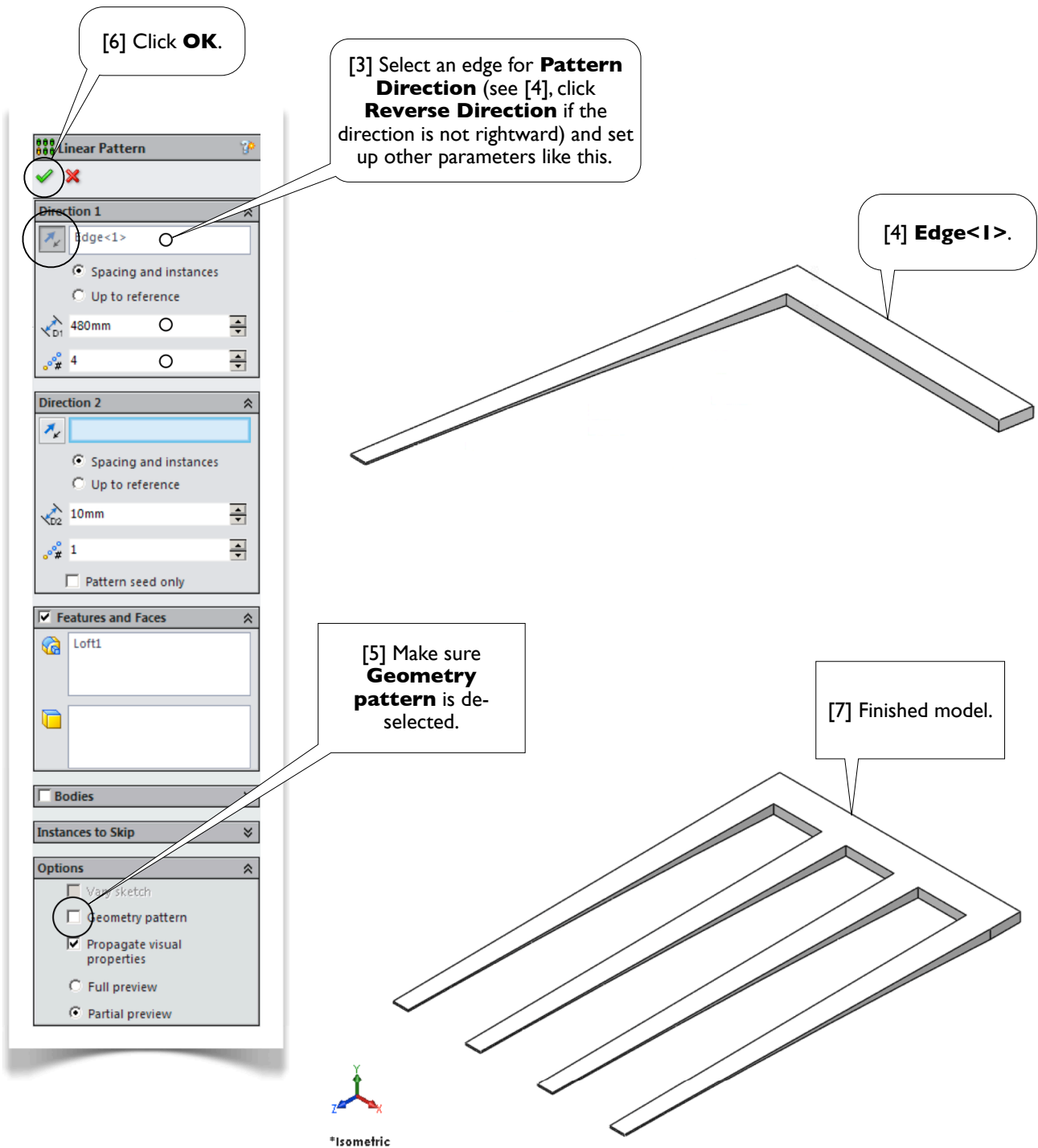


2.8-6 Create a Finger



2.8-7 Create the Other Fingers

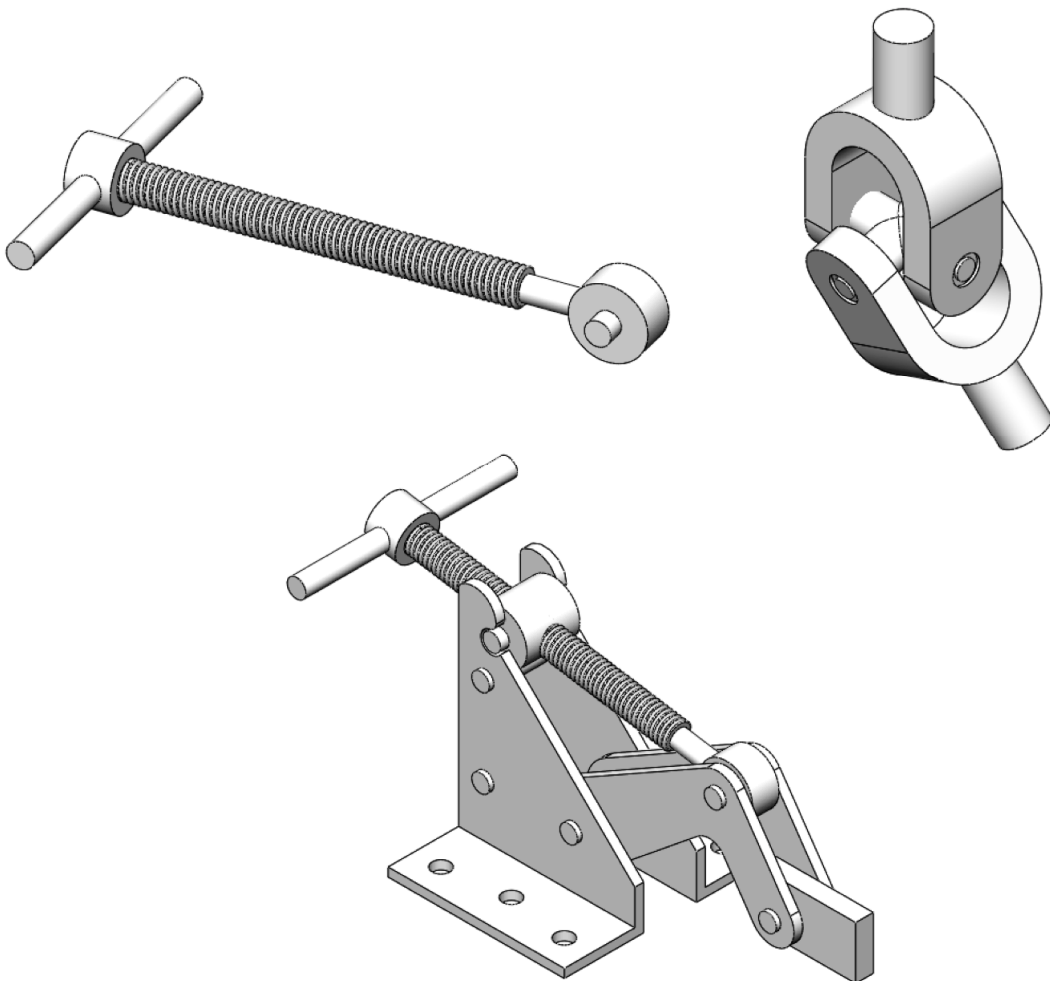




[8] Save the part with the file name **Fork**.
Close the file and exit **SOLIDWORKS**.#

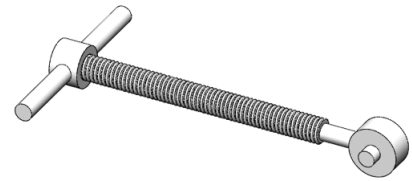
Chapter 3

Assembly Modeling



Section 3.1

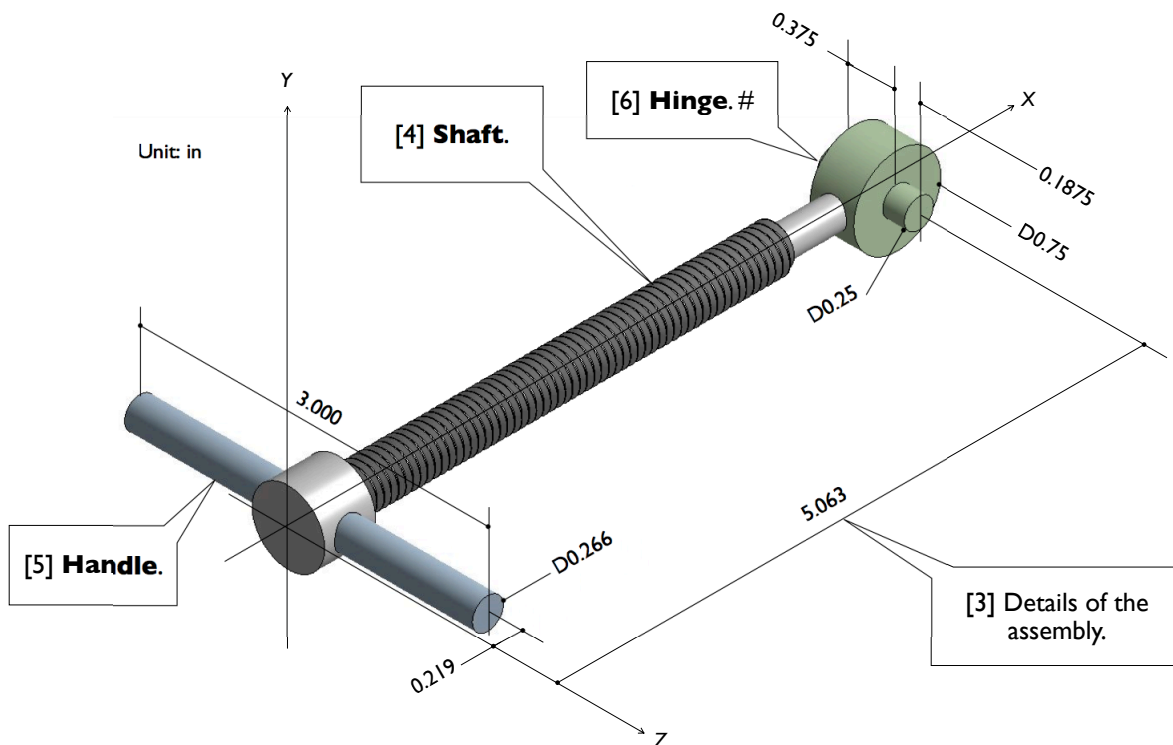
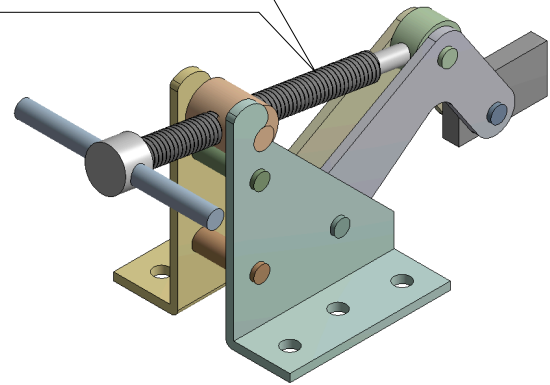
Shaft Assembly



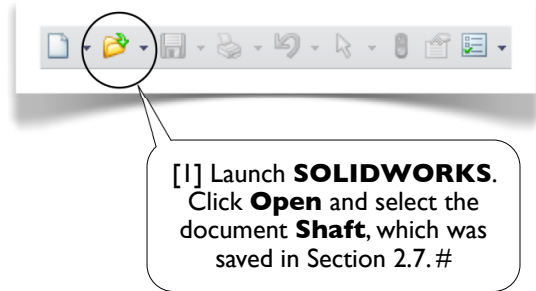
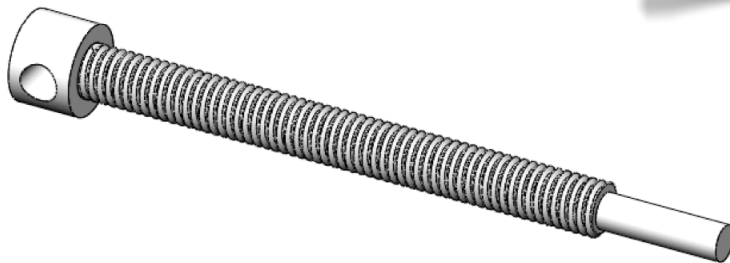
3.1-1 Introduction

[1] In this exercise, we'll create a shaft assembly [2, 3]. The assembly consists of three parts: the **Shaft** [4] created in Section 2.7, a **Handle** [5], and a **Hinge** [6]. We use a coordinate system for the assembly which is coincident with that of the part **Shaft**.

[2] The shaft assembly is a sub-assembly of the clamping mechanism.



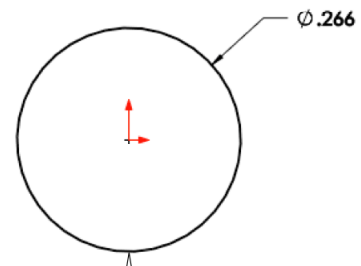
3.1-2 Open **Shaft**



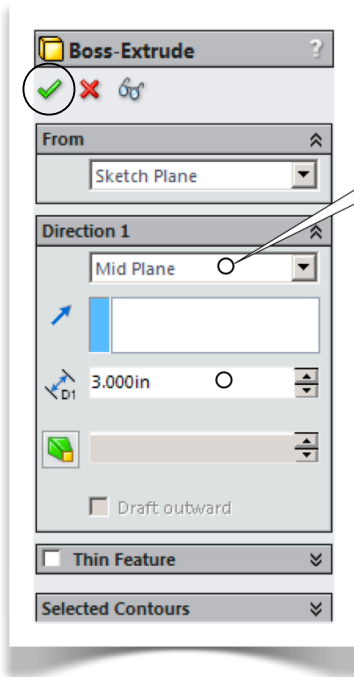
3.1-3 Create **Handle**



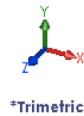
[1] Click **New** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.



[2] On **Front** plane, draw a circle centered at the origin like this.



[3] Extrude the sketch 3 in, using **Mid Plane** option.



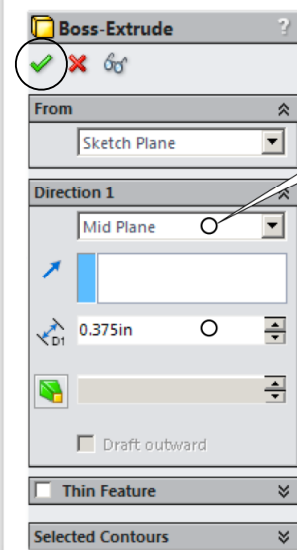
[4] Save the part with the file name **Handle**. #



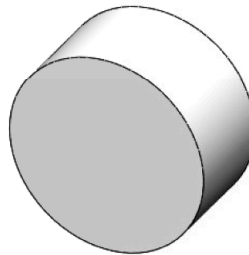
3.1-4 Create Hinge



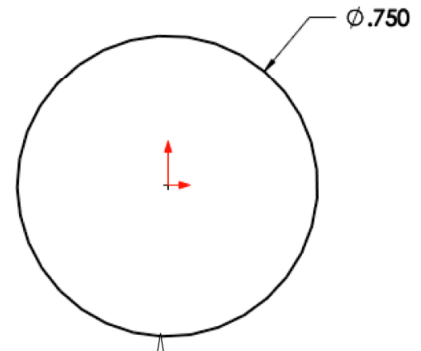
[1] Click **New** and create a new part.
Set up **IPS** unit system with 3 decimal
places for the length unit.



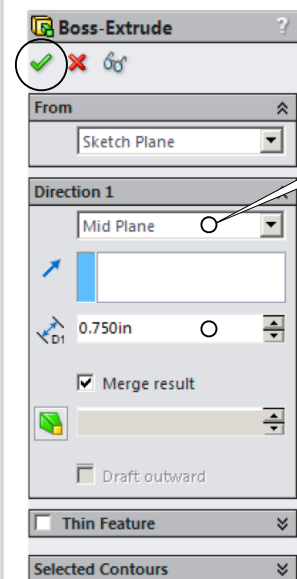
[3] Extrude the sketch 0.375
inches, using **Mid Plane**
option.



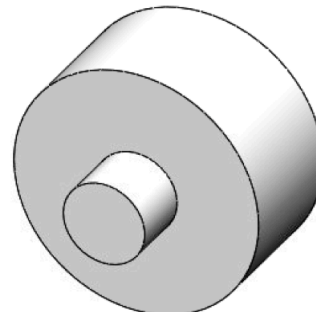
*Trimetric



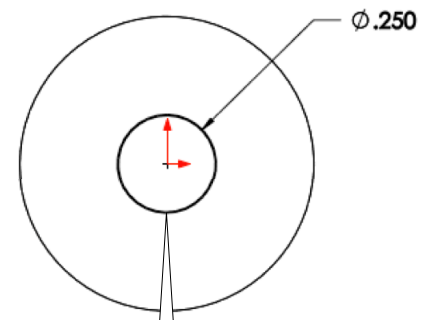
[2] On **Front** plane, draw a
circle centered at the origin
like this.



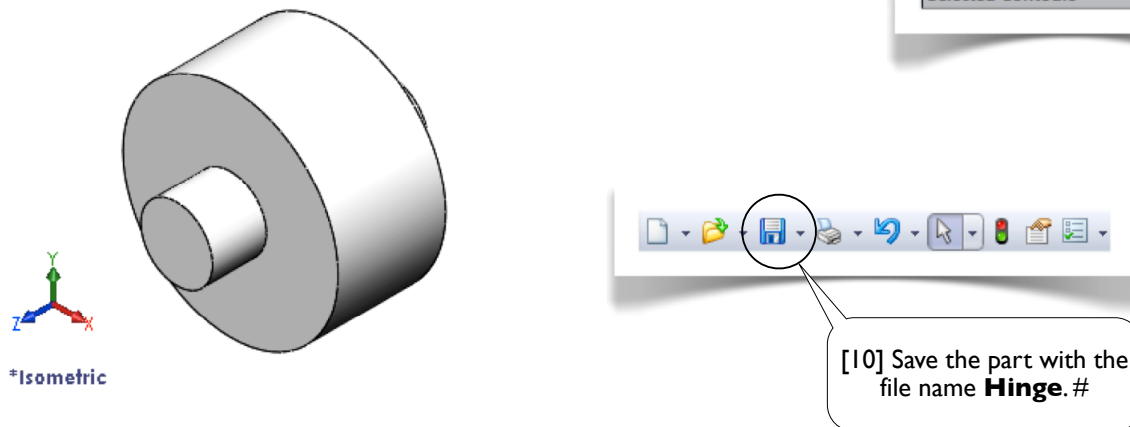
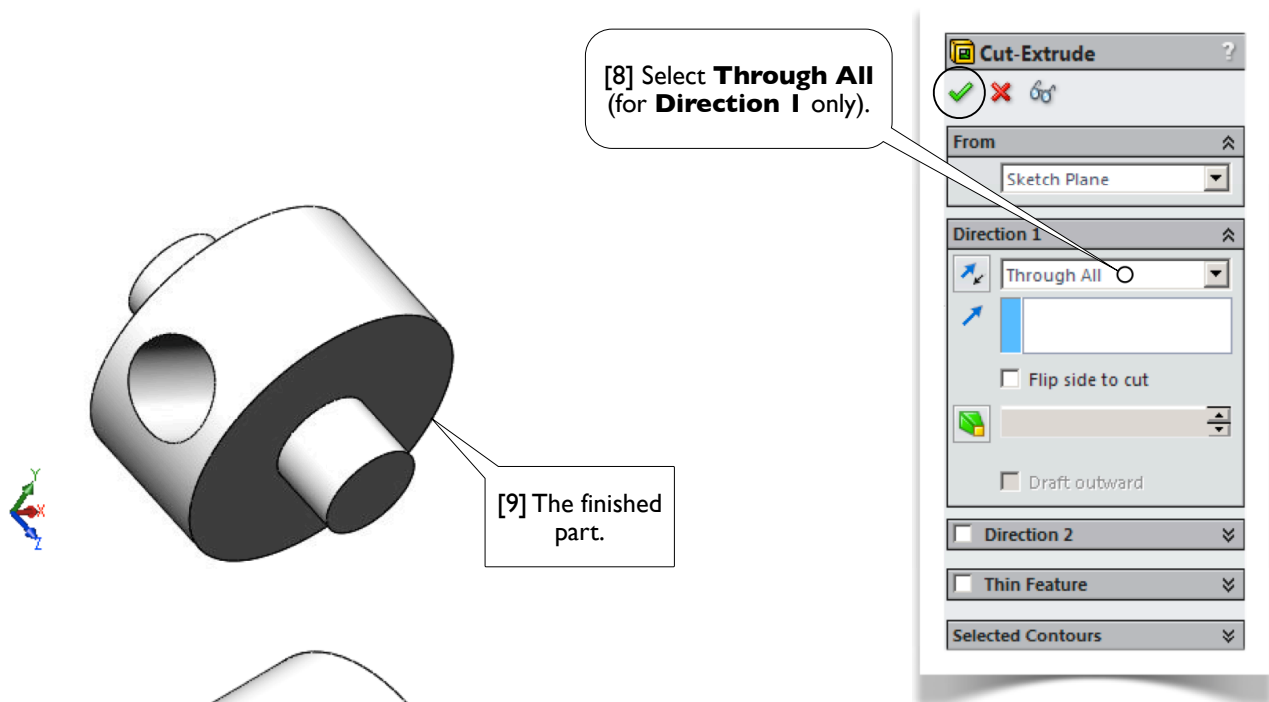
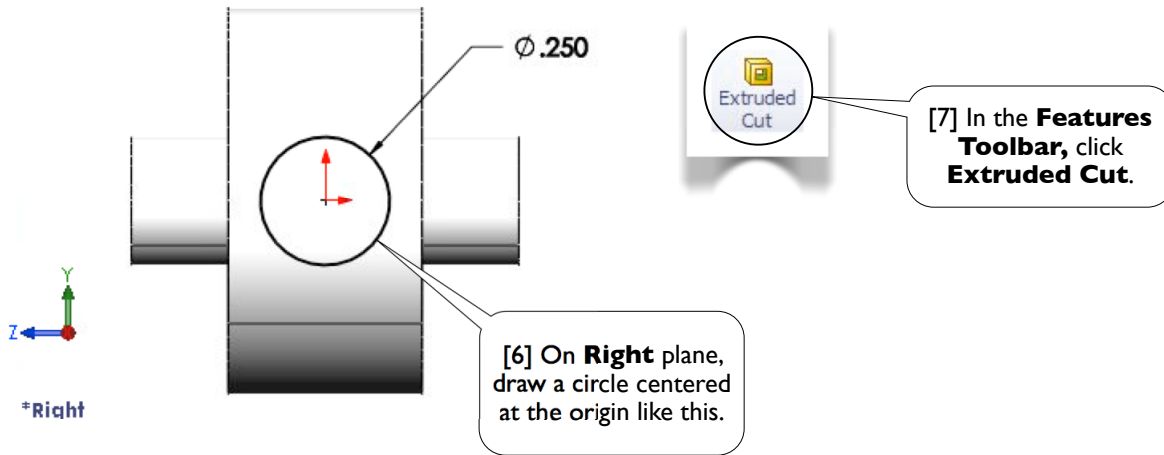
[5] Extrude the sketch 0.75
inches, using **Mid Plane**
option.



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[4] On **Front** plane, draw a
circle centered at the origin
like this.



3.1-5 Create a New Assembly

[1] If you pull down the **Window** menu, you can see that three **Part** documents are opened in the computer memory. We now create an assembly consisting of these three **Parts**.

[2] Click **New**.

[3] Select **Assembly**.

[4] Click **OK**.

[5] In the **Head-Up Toolbar**, turn on **View Origins**.

[6] This is the origin of the new assembly. We now insert the **Shaft** so that the part's coordinate system aligns with the assembly's coordinate system.

[7] In the **Property Box**, select **Shaft**.

New SOLIDWORKS Document

Part: a 3D representation of a single design component

Assembly: a 3D arrangement of parts and/or other assemblies

Drawing: a 2D engineering drawing, typically of a part or assembly

Advanced OK Cancel Help

Window Help

Viewport

New Window

Cascade

Tile Horizontally

Tile Vertically

Arrange Icons

Close All

1 Shaft

2 Handle

3 Hinge

Browse Open Documents... Ctrl-Tab

Customize Menu

Begin Assembly

Message

Select a component to insert, then place it in the graphics area or hit OK to locate it at the origin.

Or design top-down using a Layout with blocks. Parts may then be created from the blocks.

Create Layout

Part/Assembly to Insert

Open documents:

Handle

Hinge

Shaft

Browse...

Thumbnail Preview

Options

Start command when creating new assembly

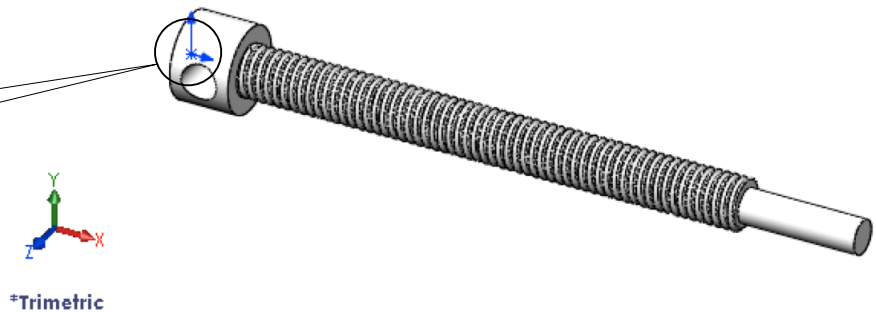
Graphics preview

Make virtual

Envelope

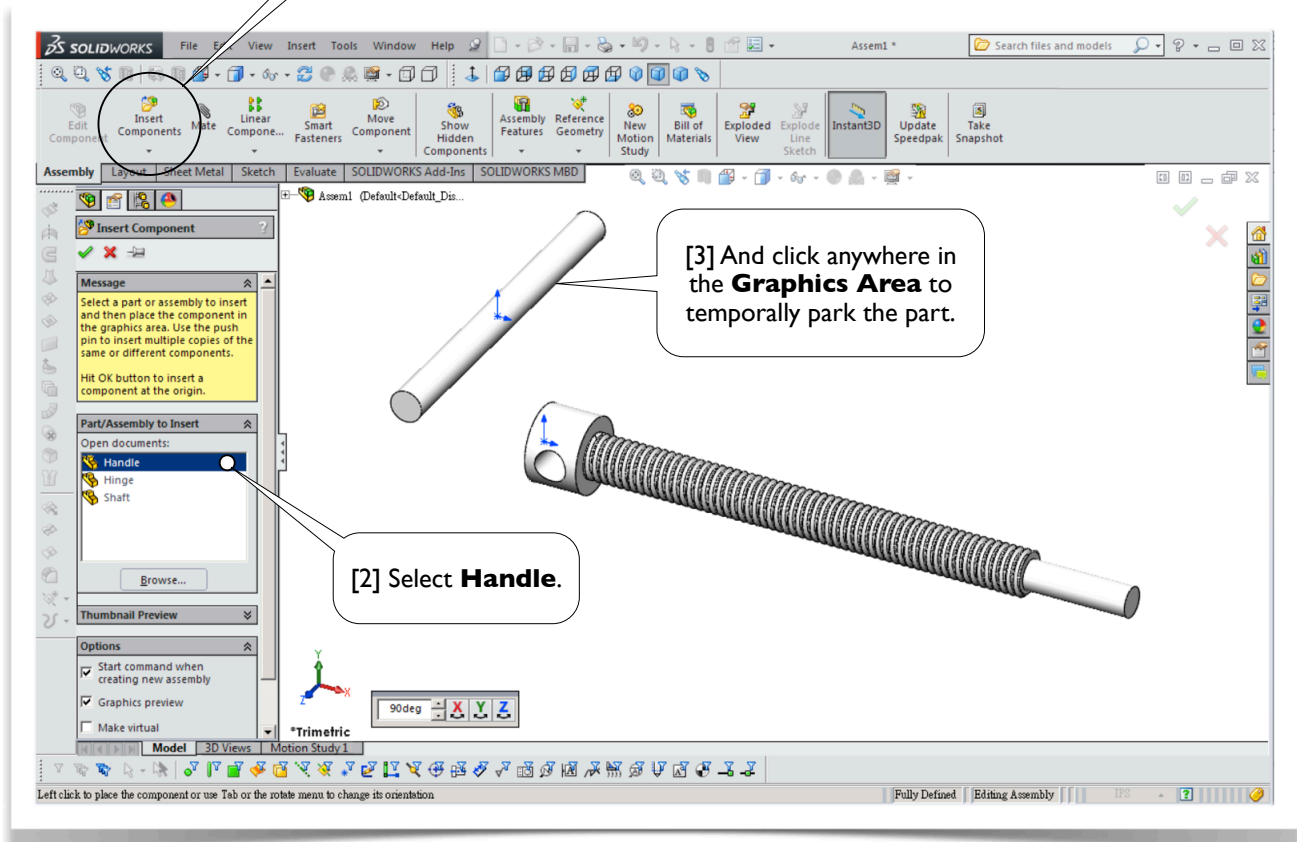
Show Rotate context toolbar

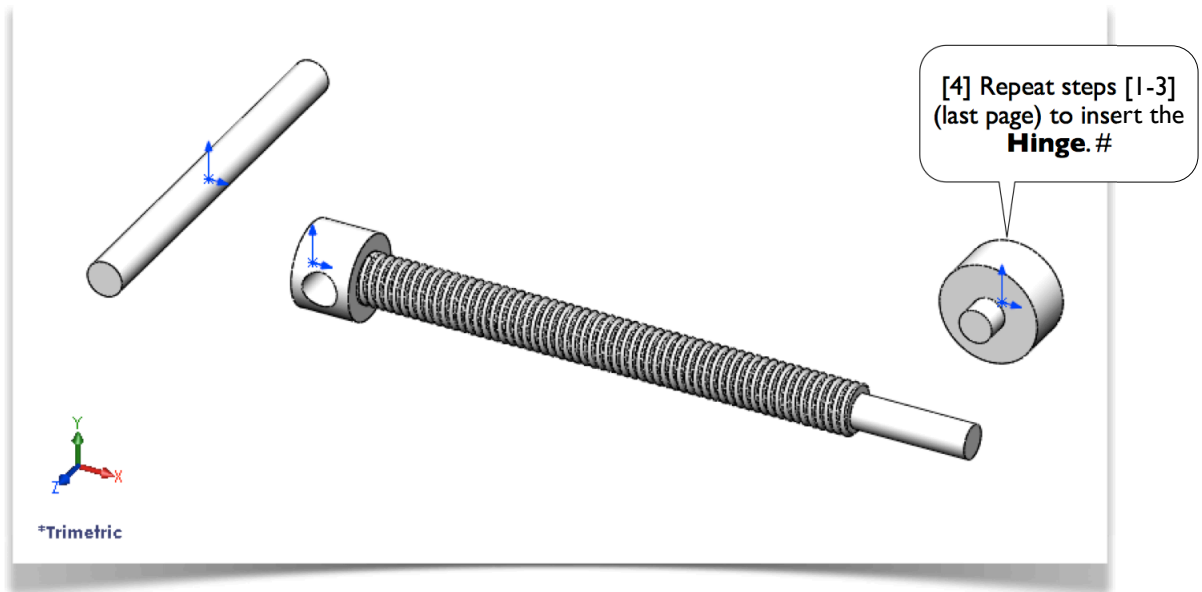
[8] Click the assembly's origin. The **Shaft** is inserted and its coordinate system aligns with the assembly's coordinate system. #



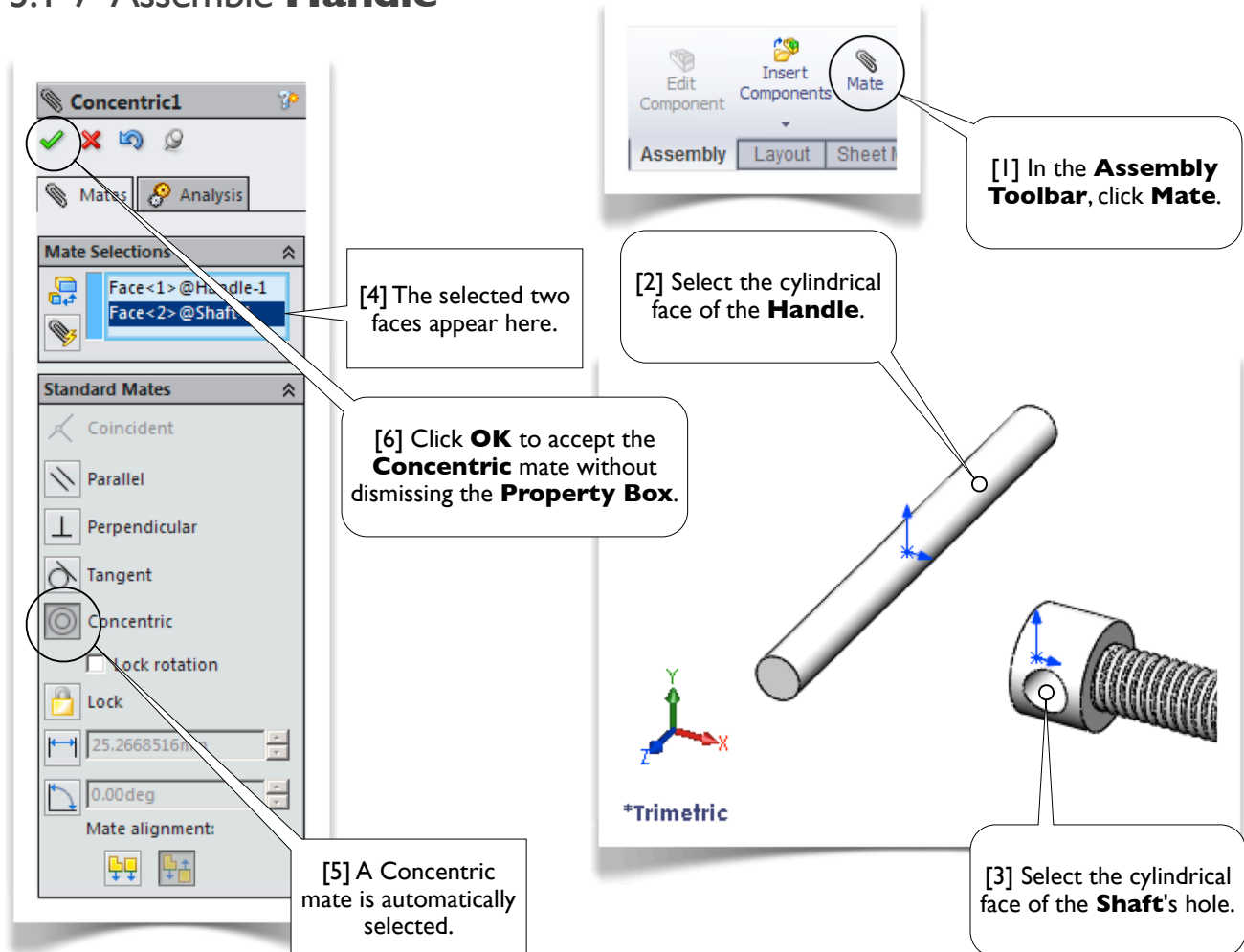
3.1-6 Insert the Other Components

[1] In the **Assembly Toolbar**, click **Insert Components**.

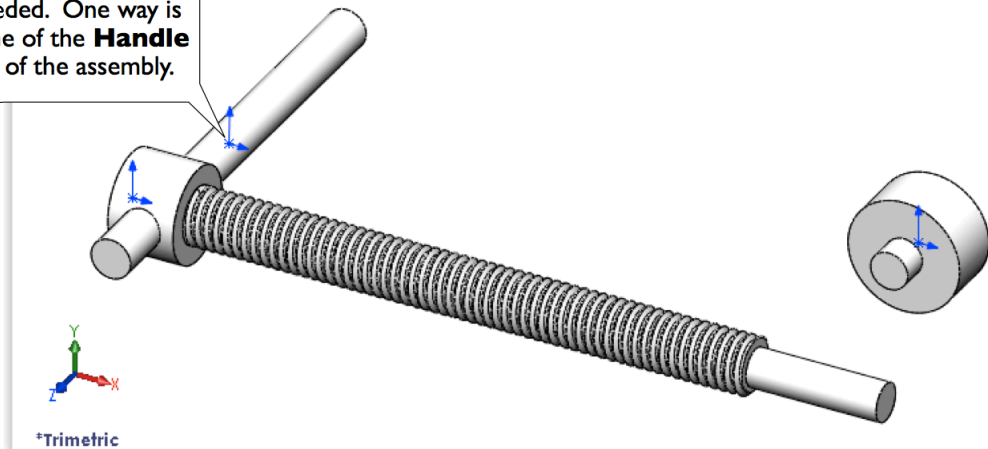




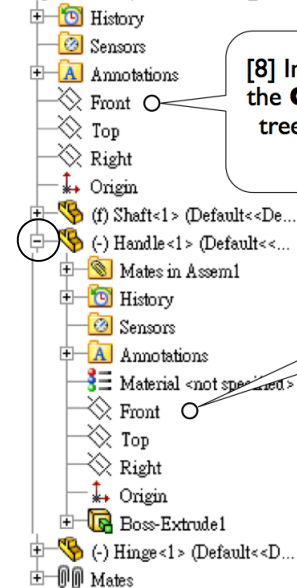
3.1-7 Assemble **Handle**



[7] The **Handle** is assembled into the hole of the **Shaft**. However, the **Handle** is not well positioned yet (you may move it using your mouse); an additional **Mate** is needed. One way is to align the **Front** plane of the **Handle** with the **Front** plane of the assembly.



Assem1 (Default<Default_Dis...

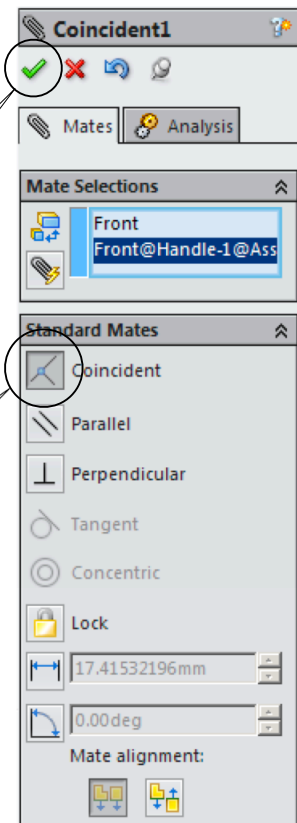


[8] In the **Part Tree** (which is in the **Graphics Area**), select **Front** plane of the assembly.

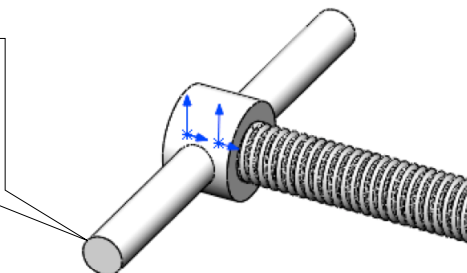
[9] Select **Front** plane of the **Handle**. Expand the tree if necessary.

[11] Click **OK** to accept the **Coincident** mate without dismissing the **Property Box**.

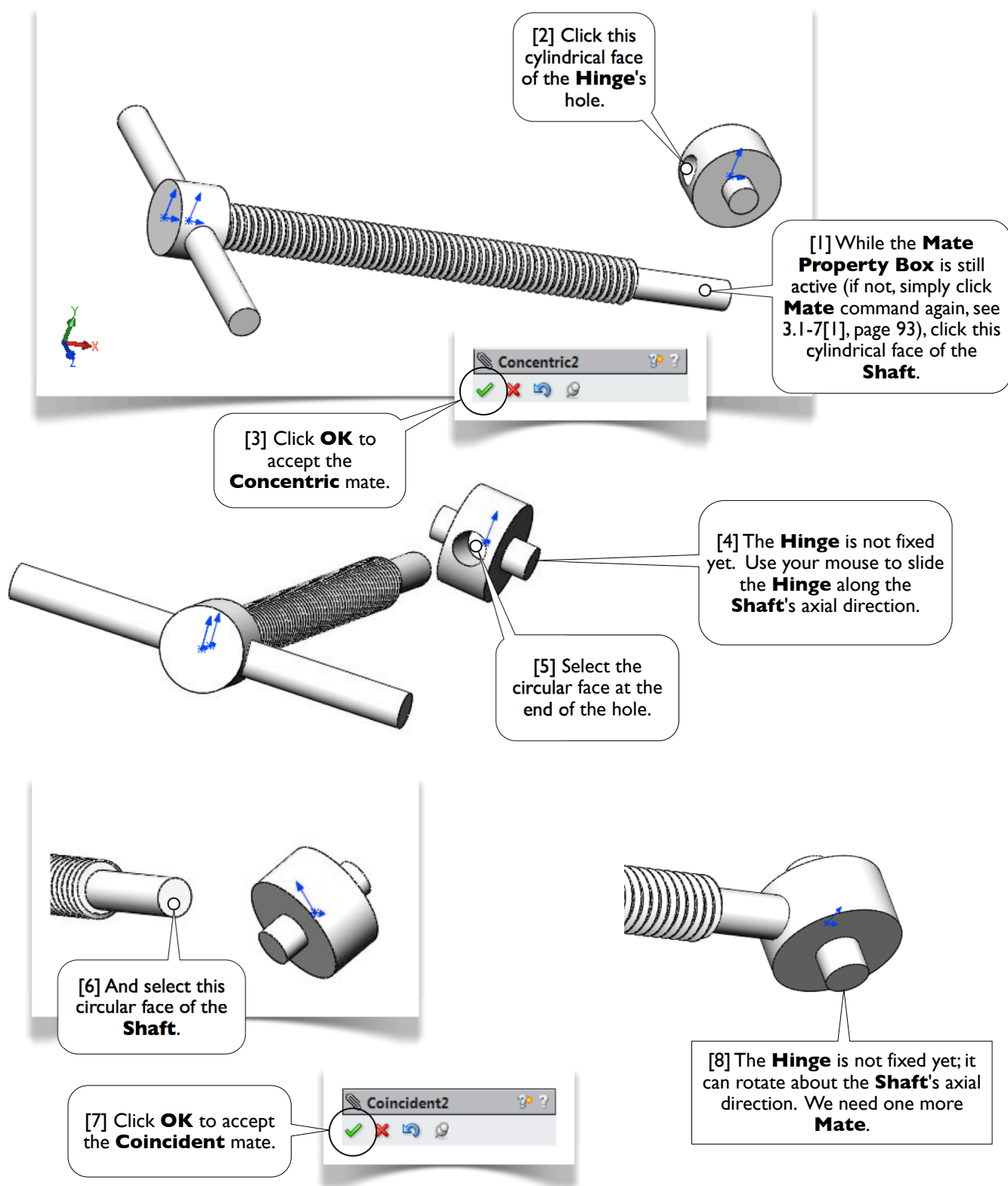
[10] A **Coincident** mate is automatically selected.

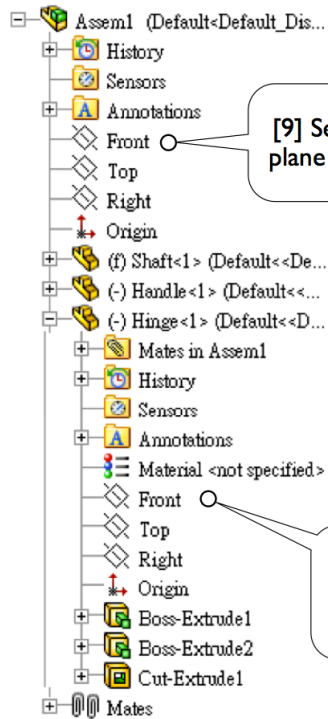


[12] This completes the assembly of the **Handle**. The handle still can rotate about its axis, however, we neglect this deficiency. #



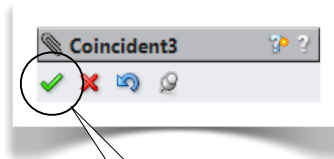
3.1-8 Assemble **Hinge**



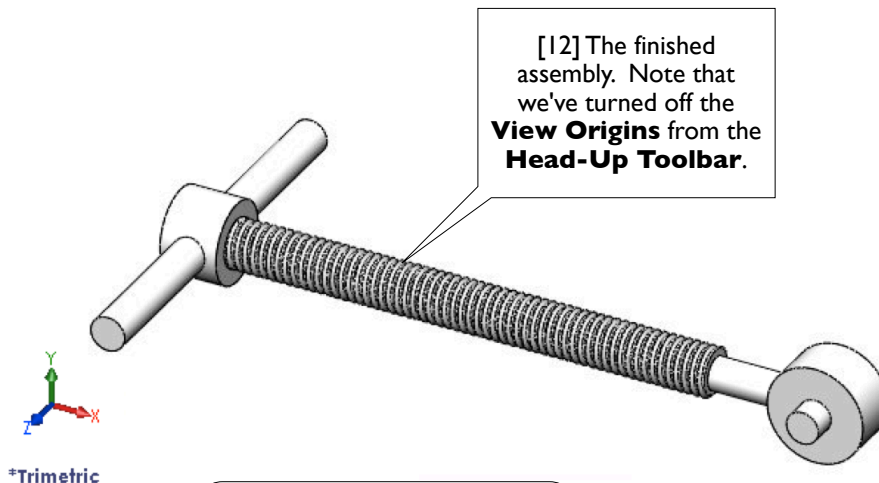


[9] Select the **Front** plane of the assembly.

[10] Select the **Front** plane of the **Hinge**. Expand the tree if necessary.



[11] Click **OK** to accept the Coincident mate. Click **OK** again to dismiss the **Mate** command.



[12] The finished assembly. Note that we've turned off the **View Origins** from the **Head-Up Toolbar**.

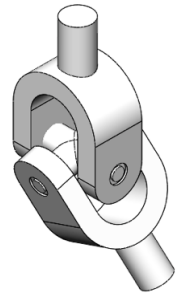
*Trimetric

[13] Save the assembly with the file name **ShaftAssembly**. The full name of the document is **ShaftAssembly.SLDASM**. Exit **SOLIDWORKS**. #



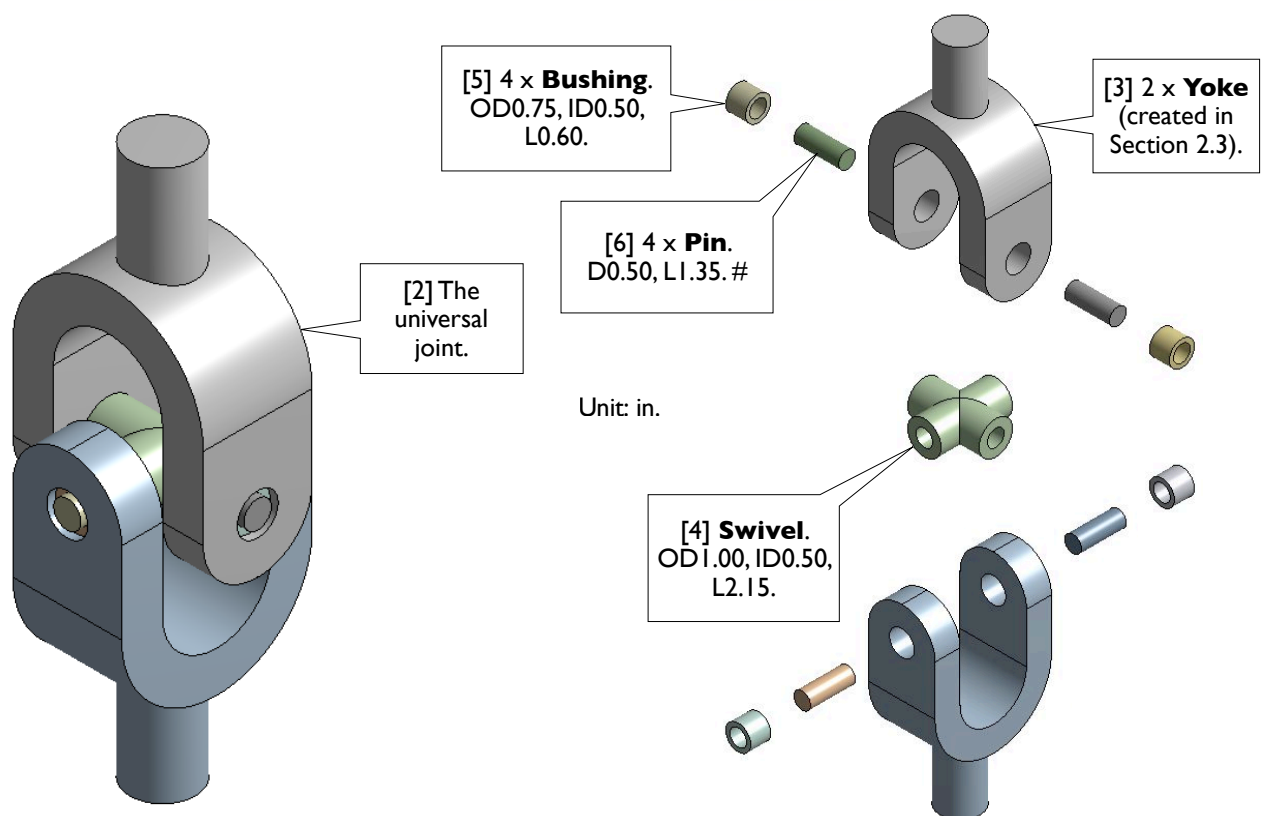
Section 3.2

Universal Joint

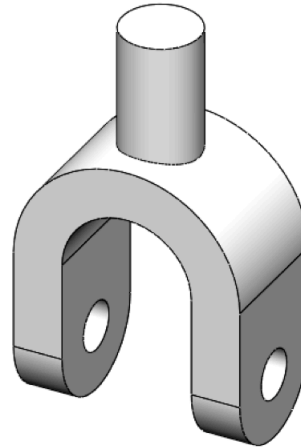
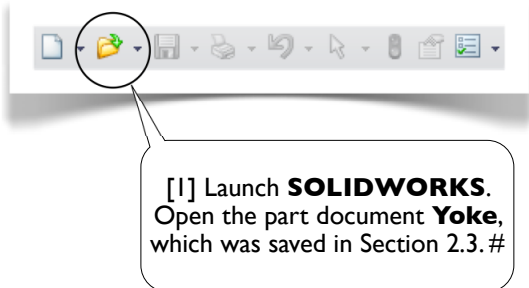


3.2-1 Introduction

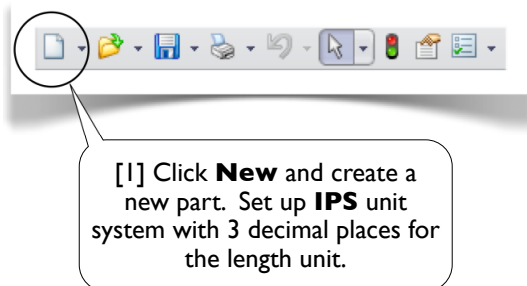
[1] In this exercise, we'll create a universal joint [2]. The assembly consists of four kinds of parts [3-6], of which the **Yoke** [3] was created in Section 2.3.



3.2-2 Open **Yoke**



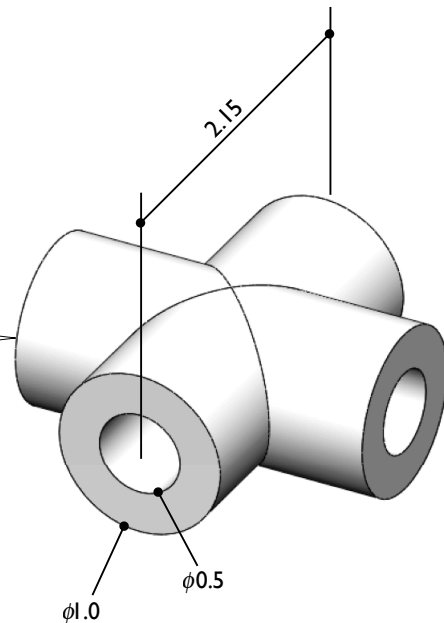
3.2-3 Create **Swivel**



[2] Create a 3D model like this. Use any coordinate system as your convenience. Save the part with the file name **Swivel**. #



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3.2-4 Create **Bushing**

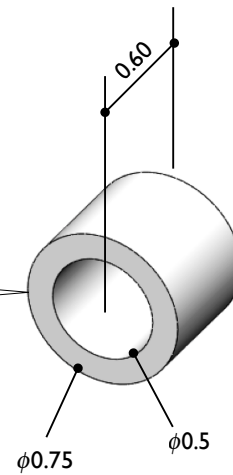


[1] Click **New** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

[2] Create a 3D model like this. Use any coordinate system as your convenience. Save the part with the file name **Bushing.#**



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3.2-5 Create **Pin**

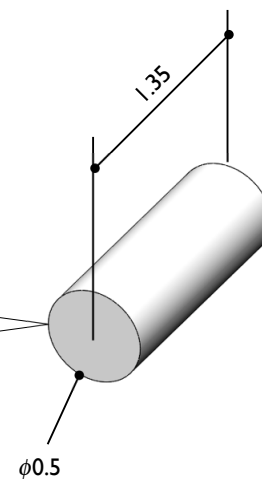


[1] Click **New** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

[2] Create a 3D model like this. Use any coordinate system as your convenience. Save the part with the file name **Pin.#**



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3.2-6 Create a New Assembly

[1] If you pull down the **Window** menu, you can see that four **Part** documents are opened in the computer memory. We now create an assembly which consists of these four kinds of **Parts**.

[2] Click **New**.

Window Help [New] [Open] [Save] [Print] [Undo] [Redo] [Zoom]

Viewport

- New Window
- Cascade
- Tile Horizontally
- Tile Vertically
- Arrange Icons
- Close All

1 Yoke

2 Swivel

3 Bushing

4 Pin

Browse Open Documents... Ctrl-Tab

Customize Menu

[3] Select **Assembly**.

New SOLIDWORKS Document

Part a 3D representation of a single design component

Assembly a 3D arrangement of parts and/or other assemblies

Drawing a 2D engineering drawing, typically of a part or assembly

Advanced OK Cancel Help

[4] Click **OK**.

Begin Assembly ?

Message

Select a component to insert, then place it in the graphics area or hit OK to locate it at the origin.

Or design top-down using a Layout with blocks. Parts may then be created from the blocks.

Create Layout

Part/Assembly to Insert

Open documents:

- Bushing
- Pin
- Swivel
- Yoke

Browse...

Thumbnail Preview

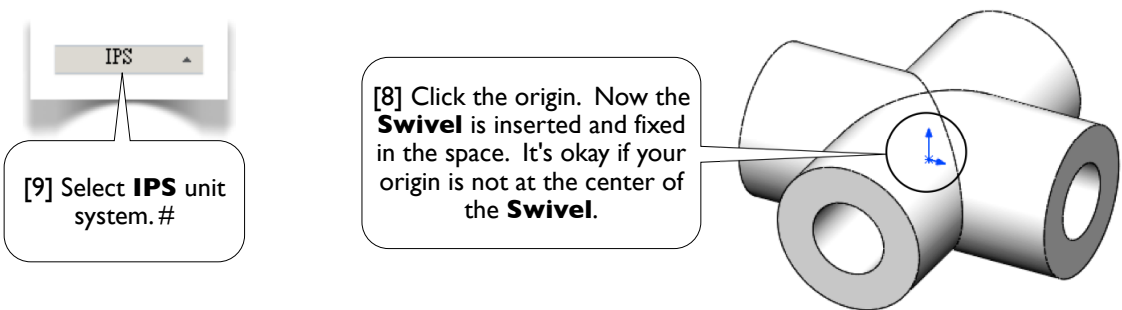
Options

- ☒ Start command when creating new assembly
- ☒ Graphics preview
- ☐ Make virtual
- ☐ Envelope
- ☒ Show Rotate context toolbar

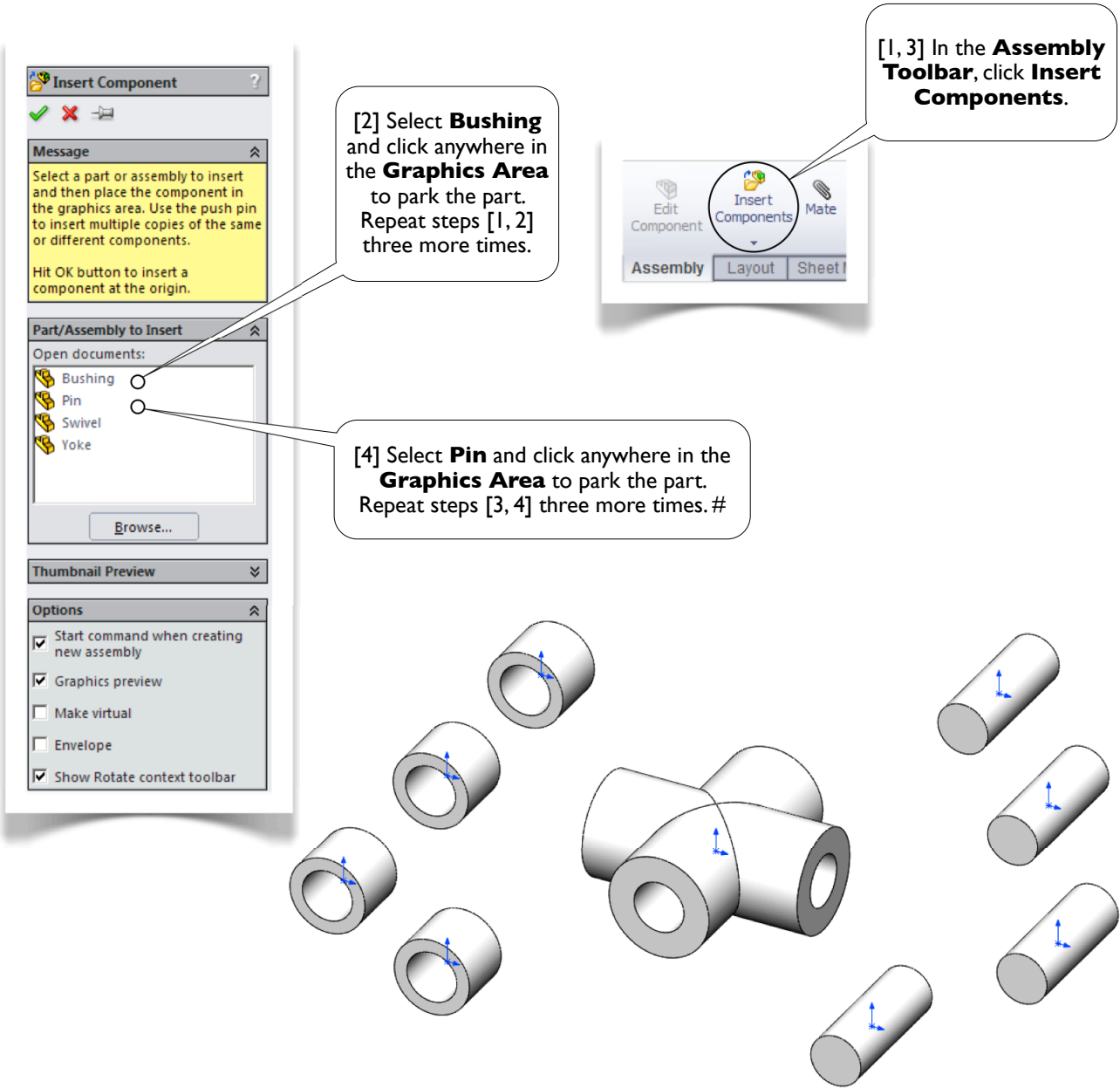
[5] In the **Head-Up Toolbar**, turn on **View Origins**.

[6] This is the origin of the new assembly. We now insert the **Swivel** so that the part's coordinate system aligns with the assembly's coordinate system.

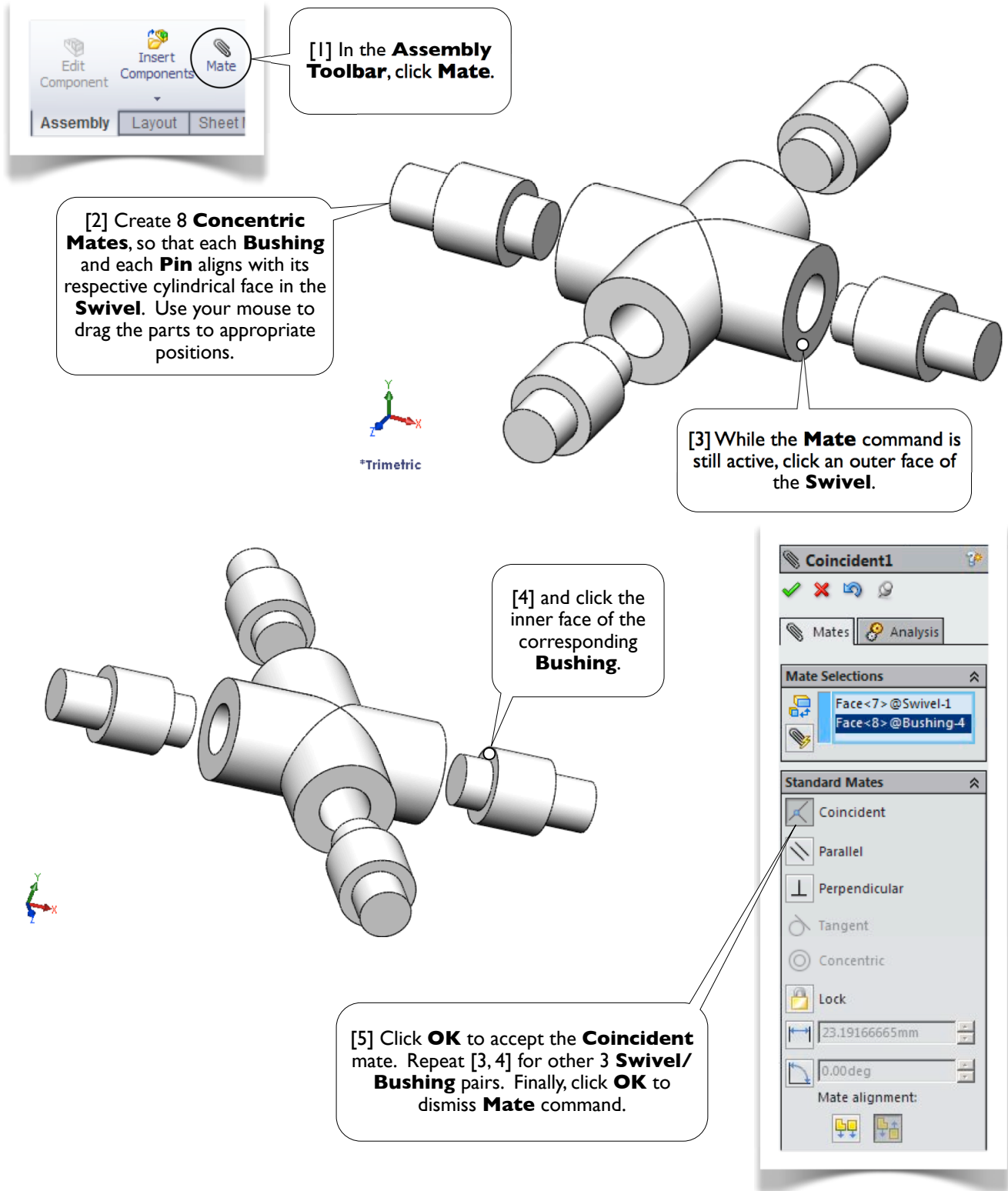
[7] In the **Property Box**, select **Swivel**.

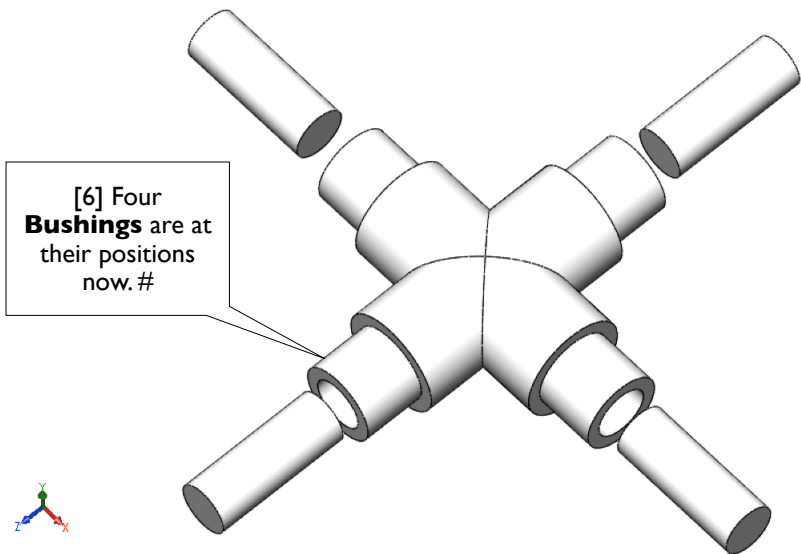


3.2-7 Insert **Bushings** and **Pins**



3.2-8 Assemble **Bushings** and **Pins**





3.2-9 Assemble **Yokes**

Insert Component

Message

Select a part or assembly to insert and then place the component in the graphics area. Use the push pin to insert multiple copies of the same or different components.

Hit OK button to insert a component at the origin.

Part/Assembly to Insert

Open documents:

- Bushing
- Pin
- Swivel
- Yoke**

Browse...

Thumbnail Preview

Options

- ☒ Start command when creating new assembly
- ☒ Graphics preview
- ☐ Make virtual
- ☐ Envelope
- ☒ Show Rotate context toolbar

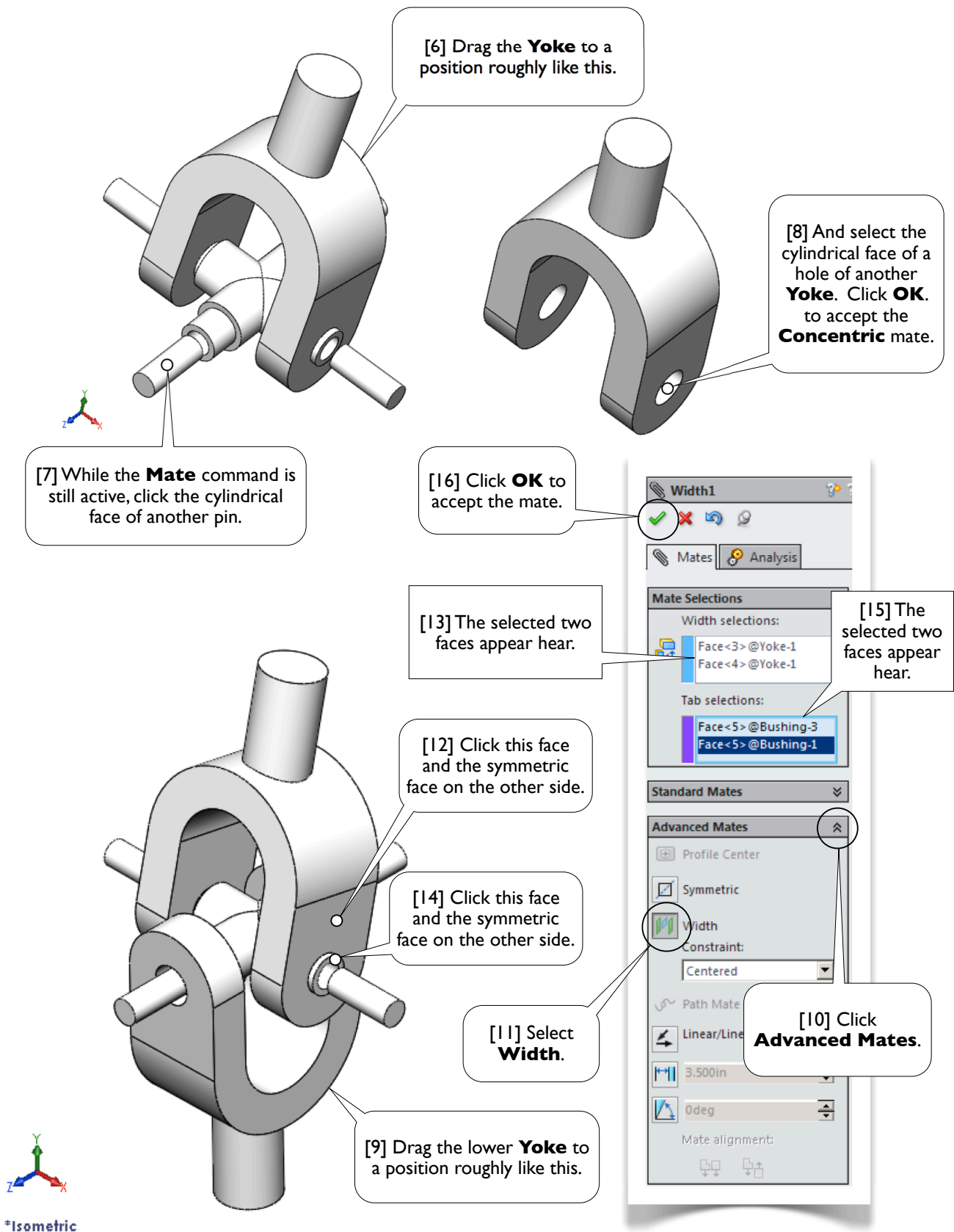
[1] Click **Insert Components**.

[3] Click **Mate**.

[2] Select **Yoke** and click anywhere in the **Graphics Area** to park the part. Repeat steps [1, 2] one more times.

[4] Select the cylindrical face of a pin.

[5] And select the cylindrical face of a hole of a **Yoke**. Click **OK** to accept the **Concentric** mate.

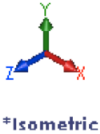


[17] Now, the middle plane of **Yoke's** two selected faces is coincident with the middle plane of **Bushings' two** selected faces.

[18] Repeat foregoing procedure (steps [11-16], last page) for the lower **Yoke**.





[21] And click this face.

[20] Click this face.



[22] Click **OK**.

Coincident5



Mates Analysis

Mate Selections

Face<7> @Pin-4
Face<4> @Yoke-2

Standard Mates

Coincident

Parallel

Perpendicular

Tangent


Concentric

Lock

54.84657392mm

0.00deg

Mate alignment:

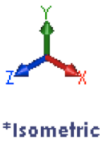


[19] Click **Standard Mates**.

[24] Repeat the foregoing procedure (steps [20-22]) for 3 other pins. Finally, click **OK** to dismiss **Mate** command.

[25] Note that the **Swivel** is fixed (3.2-6[8], page 101). The **Yokes** can be move relative to the **Swivel**. Now, we want to release the **Swivel** and fix the upper **Yoke** instead. #

[23] Now, a **Pin's** outer face aligns with a **Yoke's** outer face.



3.2-10 Fix Upper Yoke

[1] An **(f)** before the **Swivel** indicates that the **Swivel** is fixed. Right-click the **Swivel** and select **Float** from the **Context Menu**. The **(f)** sign turns to **(-)** sign, indicating that it is not fixed any more. Using your mouse, you can move every part of the assembly. Let's fix the upper **Yoke**. To do that, you could simply right-click **Yoke<1>** and select **Fix** from the **Context Menu**. Another way is to create three **Coincident Mates** [2-4].

[2] Click **Mate**.

[3] Click **Front** plane of the assembly.

[4] Click **Front** plane of the upper **Yoke (Yoke<1>)**. And click **OK**. Repeat [3, 4] for **Top** plane and **Right** plane. Click **OK** to dismiss **Mate** command.

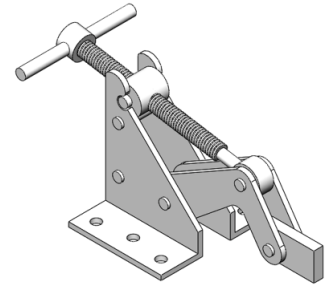
[5] Now, the upper **Yoke** is fixed in the space.

[7] Save the assembly with the file name **Joint**. Exit **SOLIDWORKS**. #

[6] Use your mouse to move the lower **Yoke**.

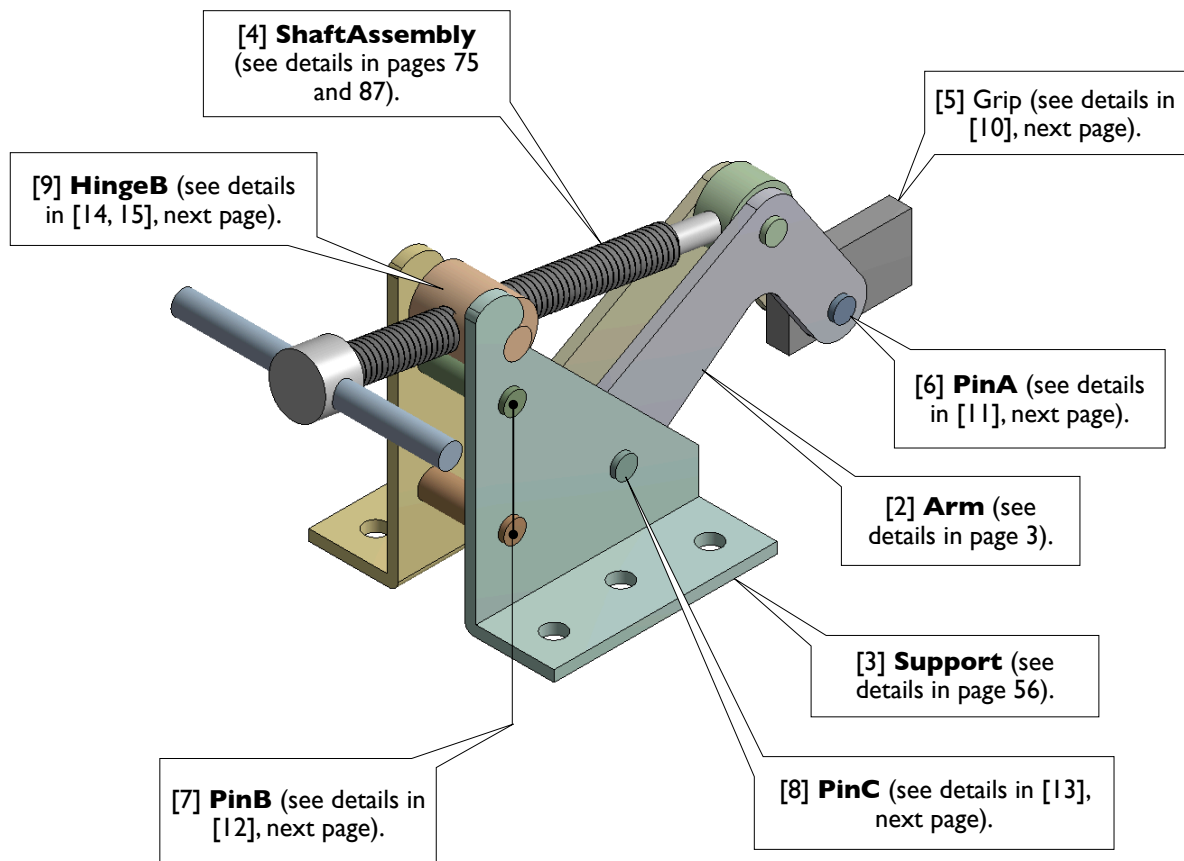
Section 3.3

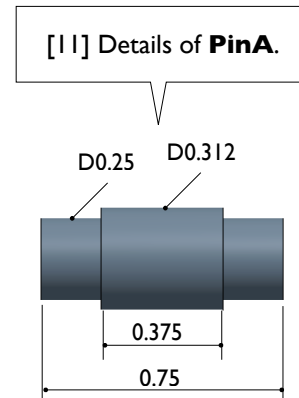
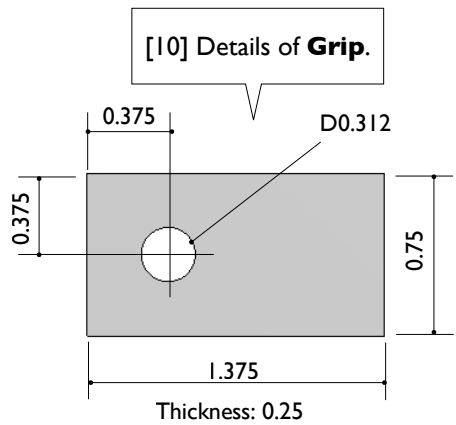
Clamp



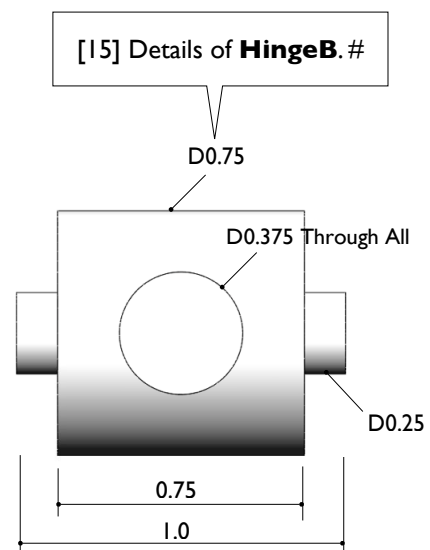
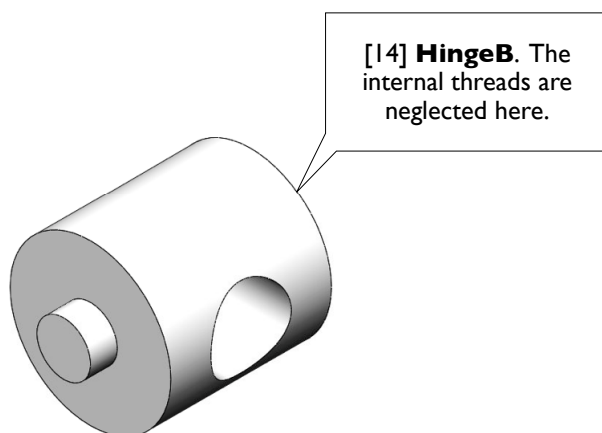
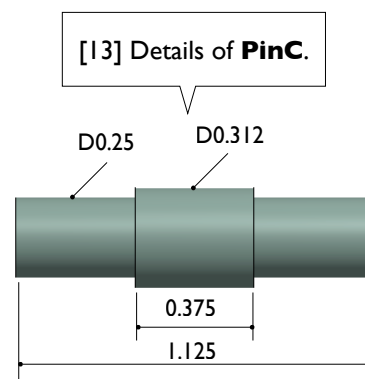
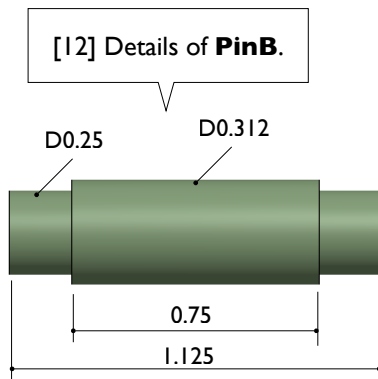
3.3-1 Introduction

[1] In this section, we'll create a **clamping mechanism** mentioned in Sections 1.1, 2.4, 2.7, and 3.1. The assembly consists of 8 kinds of components [2-9], of which the **Arm** [2] was created in Section 1.1, the **Support** [3] was created in Section 2.4, and the **ShaftAssembly** [4] was created in Sections 2.7 and 3.1. Details of other components are shown in [10-15].





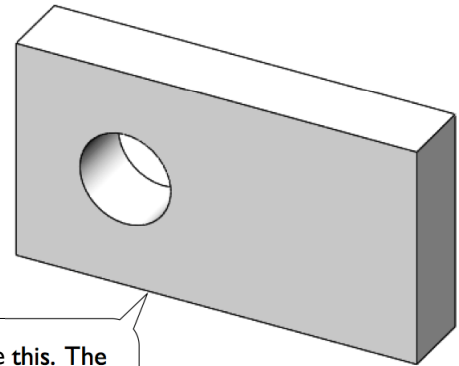
Unit: in.



3.3-2 Create **Grip**



[1] Launch **SOLIDWORKS**. Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

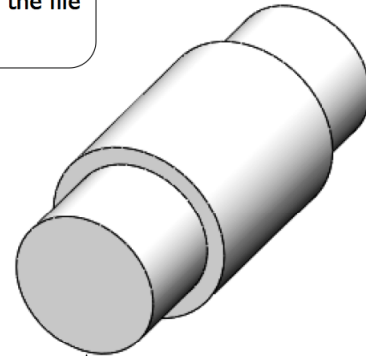


[2] Create a 3D model like this. The details are shown in 3.3-1[10] (last page). Use any coordinate system as your convenience. Save the part with the file name **Grip.#**

3.3-3 Create **PinA**



[1] Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

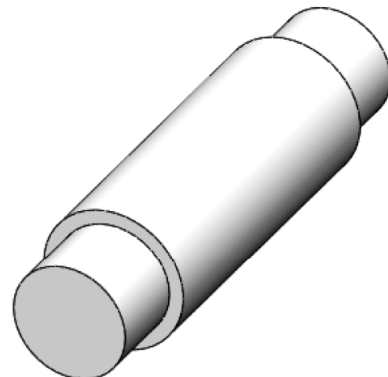


[2] Create a 3D model like this. The details are shown in 3.3-1[11] (last page). Use any coordinate system as your convenience. Save the part with the file name **PinA.#**

3.3-4 Create **PinB**



[1] Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

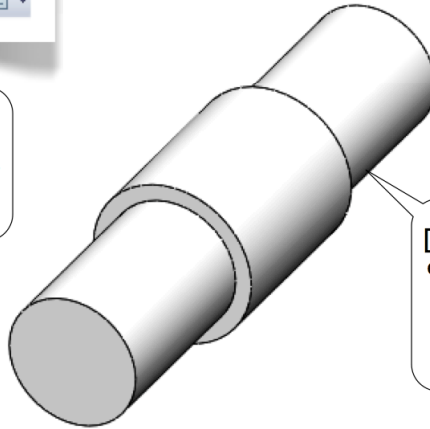


[2] Create a 3D model like this. The details are shown in 3.3-1[12] (last page). Use any coordinate system as your convenience. Save the part with the file name **PinB.#**

3.3-5 Create **PinC**



[1] Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

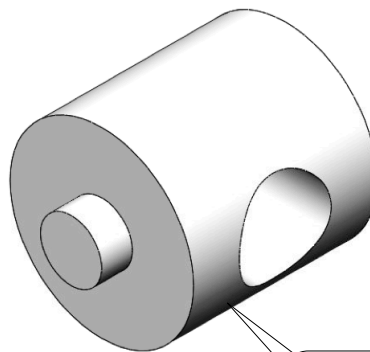


[2] Create a 3D model like this. The details are shown in 3.3-1[13] (page 108). Use any coordinate system as your convenience. Save the part with the file name **PinC.#**

3.3-6 Create **HingeB**

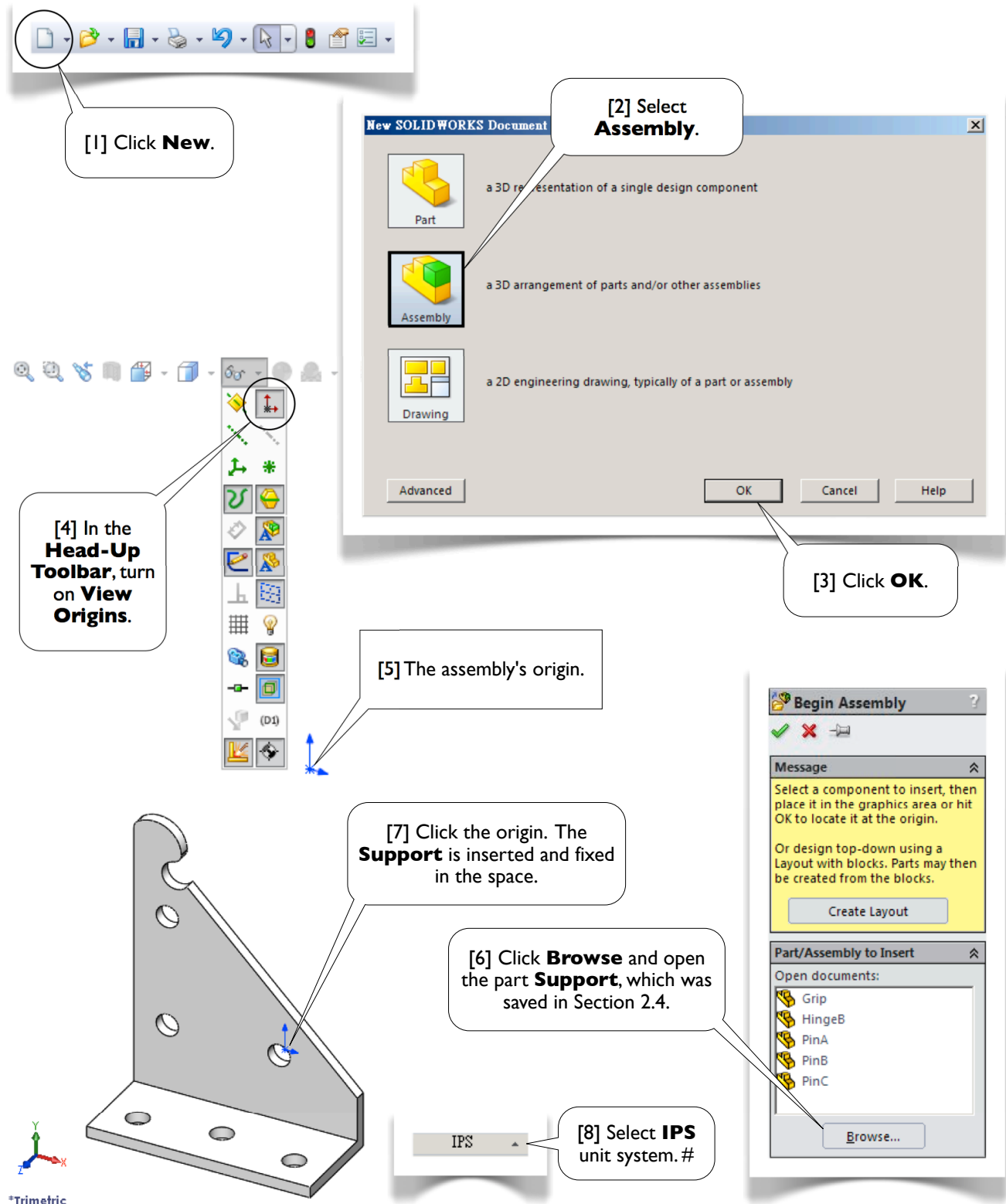


[1] Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.



[2] Create a 3D model like this. The details are shown in 3.3-1[14, 15] (page 108). Use any coordinate system as your convenience. Save the part with the file name **HingeB.#**

3.3-7 Create a New Assembly and Insert a **Support**



3.3-8 Mirror the **Support**

[1] In **Assembly Toolbar**, pull-down **Linear Component Pattern** and select **Mirror Components**.

[2] For **Mirror plane**, select the **Front** plane of the assembly (see [3]).

[3] The **Front** plane of the assembly.

[4] For **Components to Mirror**, select **Support** from the **Part Tree** (see [5]).

[5] The **Support**.

[6] Click **Next**.

[7] Click **Create opposite hand version**.

[8] Click **Next**.

[9] It says that the geometry of the mirrored **Support** is different from the original one. By default, a new file with the name **MirrorSupport** (in the same folder as **Support**) will be created.

[10] Click **OK**. Click **Yes** for any warning messages.

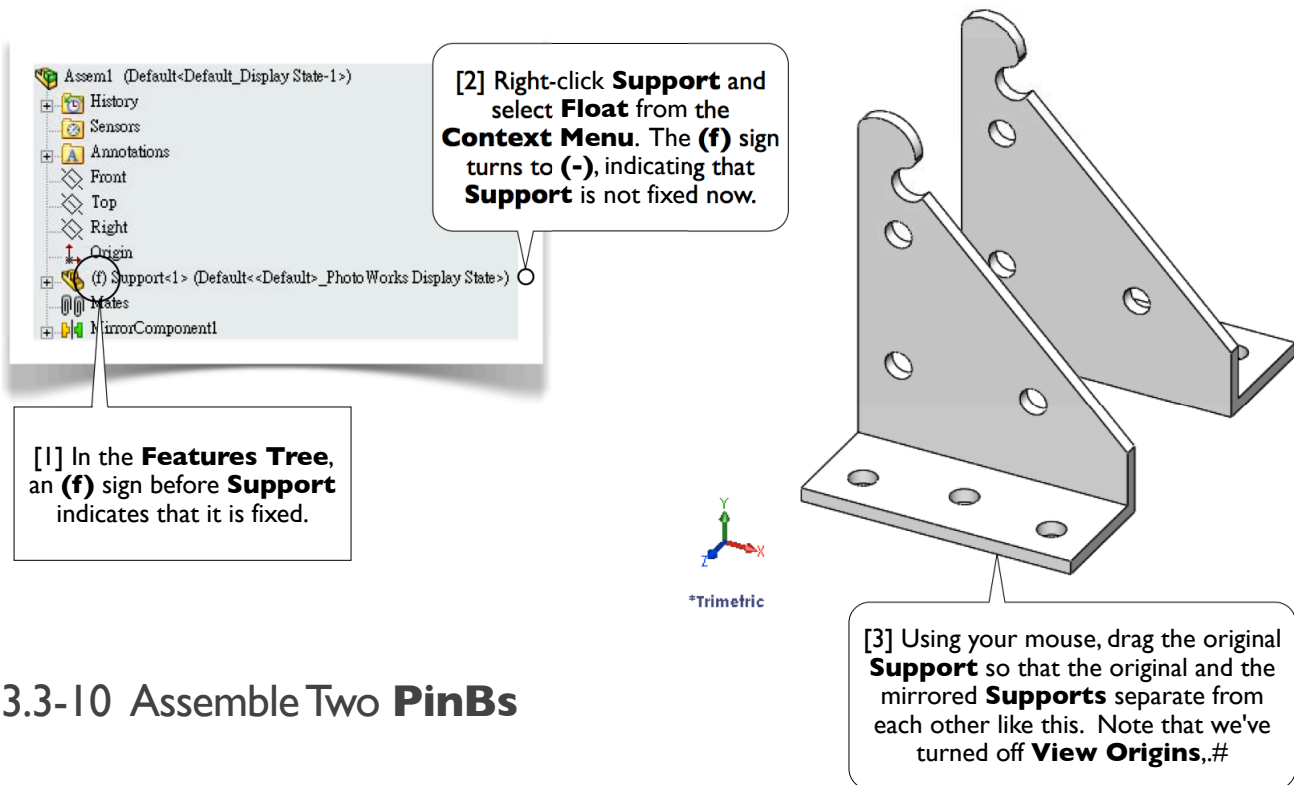
[11] Save the assembly with the name **Clamp.#**

Mirror Components dialog box details:

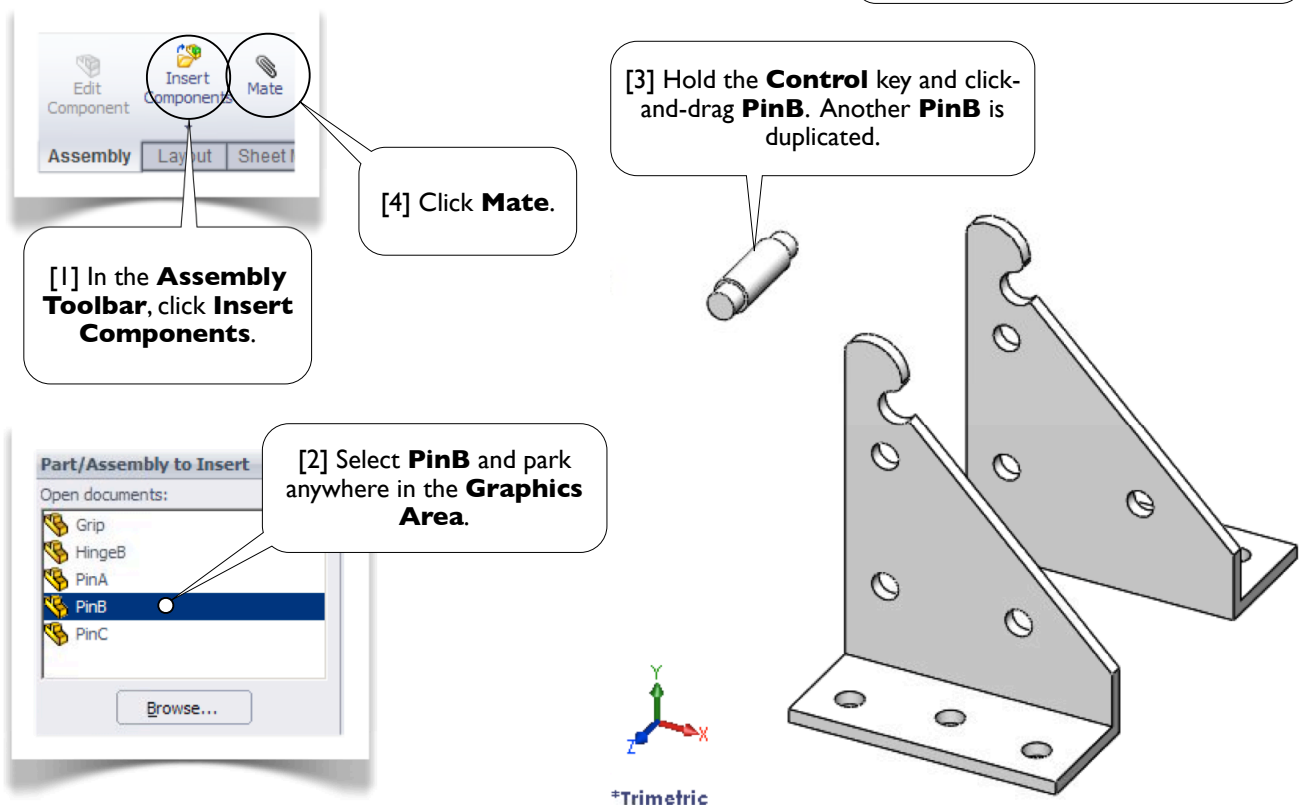
- Step 1: Selections**: Select face/plane to mirror about and the components to be mirrored. Mirror plane: Front. Components to Mirror: Support-1.
- Step 2: Set Orientation**: Verify the orientation of the components to be mirrored and adjust accordingly using the buttons below. Orient Components: Support-1.
- Step 3: Opposite Hand**: The mirrored components selected in step 2 need new geometries. This may be new files or new configurations in existing files. Specify the new configuration or file name using the options below. Opposite Hand Versions: Support-1.
- Reorient components**: Bounding box center, Center of mass, Create opposite hand version (selected), Isolate selected component.
- Create new derived configuration in existing files** (unselected).
- Create new files** (selected). Add Prefix: Mirror. File name: MirrorSupport.SLDPR. Place files in one folder (unselected).

*Isometric

3.3-9 Unfix the **Supports**



3.3-10 Assemble Two **PinBs**

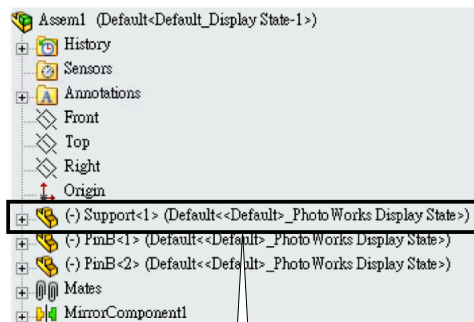


[5] Apply a **Concentric Mate** and two **Coincident Mates** to assemble this **PinB** (see [6]).

[7] Apply a **Concentric Mate** and a **Coincident Mate** to assemble this **PinB** (see [8]).

[6] After the assembling, the spacing between two **Supports** is 0.75 in.

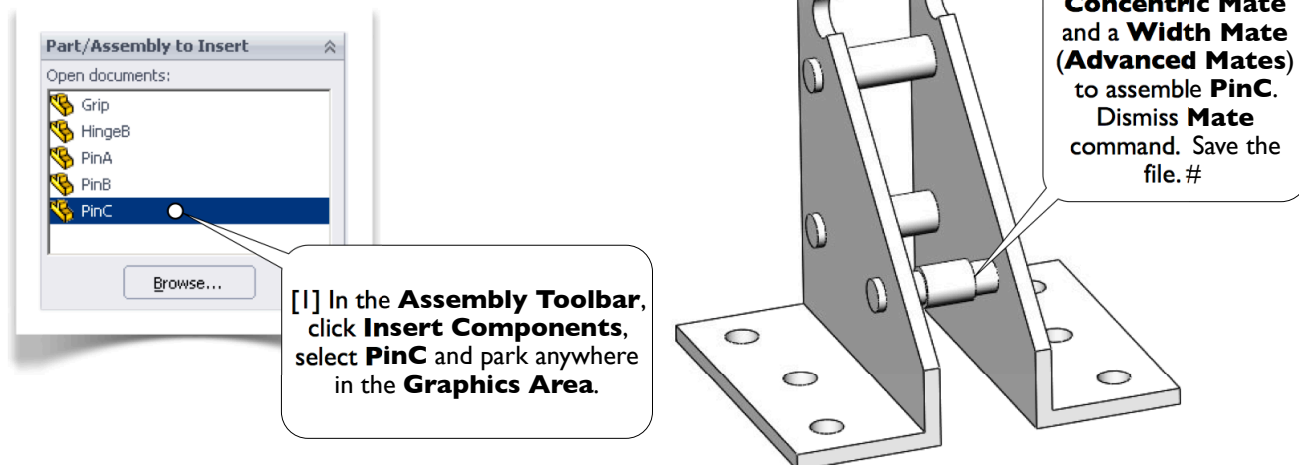
[8] Both **PinBs** are assembled. Click **OK** to dismiss **Mate** command. Save the file.



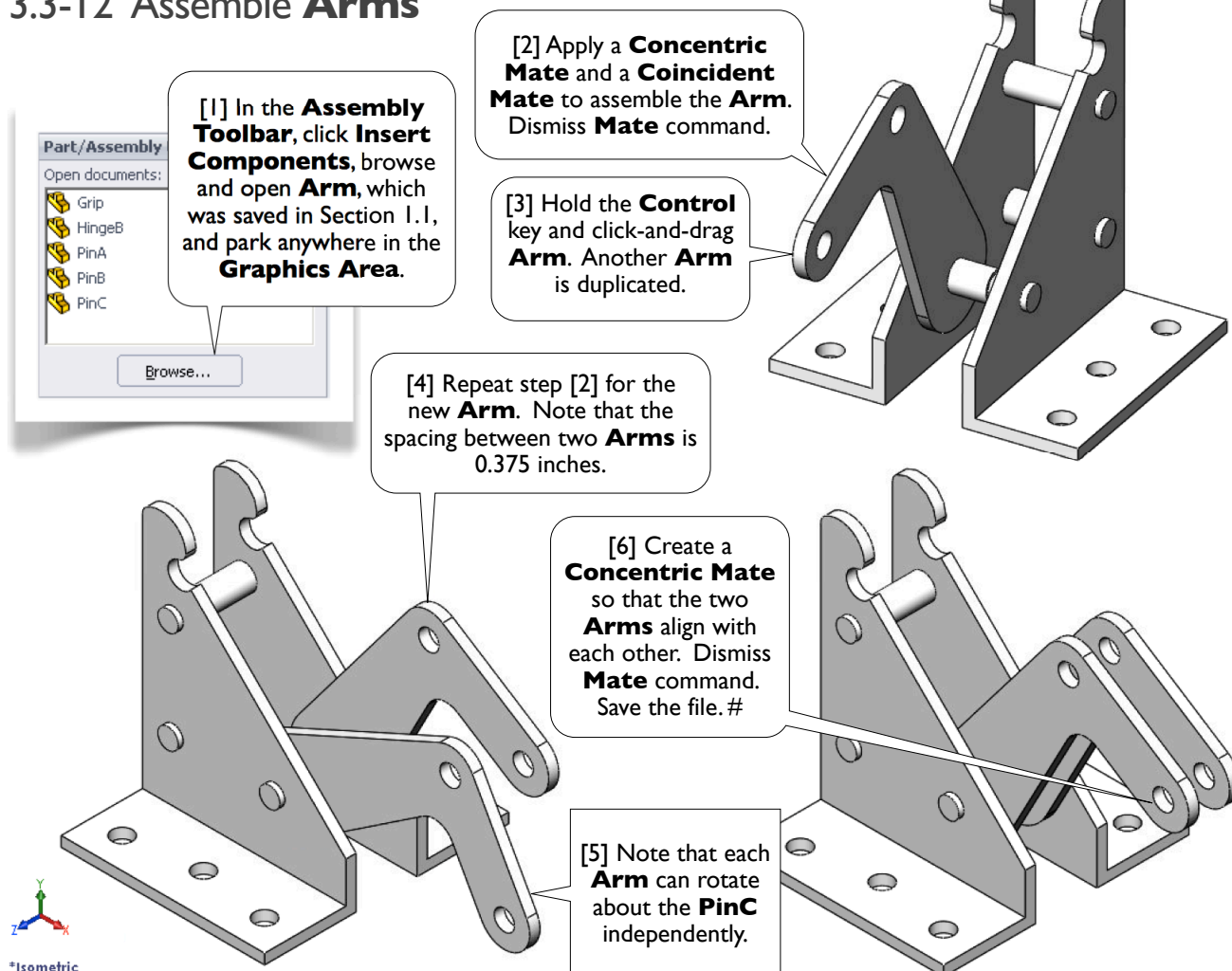
[9] Right-click **Support<1>** and select **Fix** from the **Context Menu**. The **(-)** sign turns to **(f)**, indicating that the **Support** is fixed in the space. If an error messages window appears, see [10].

[10] An error messages window may appear when executing step [9], saying that the assembly is over defined. The assembly is actually not over defined. To fix this problem, close the messages window, right-click **Support<1>** and select **Float**, and execute step [9] again. #

3.3-11 Assemble **PinC**

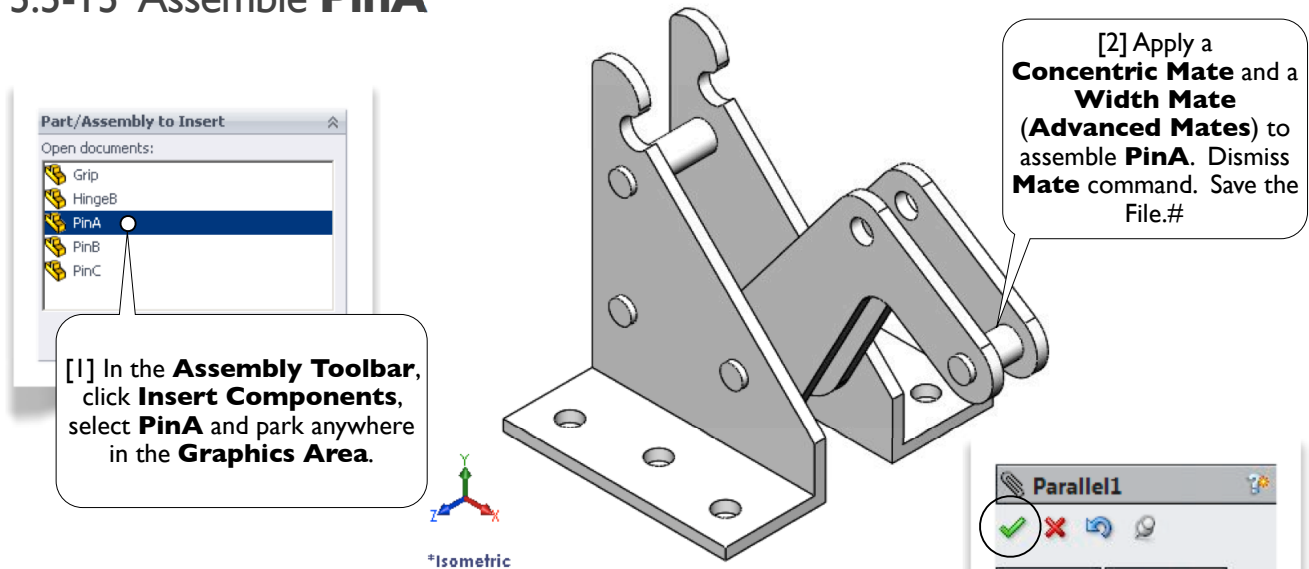


3.3-12 Assemble **Arms**

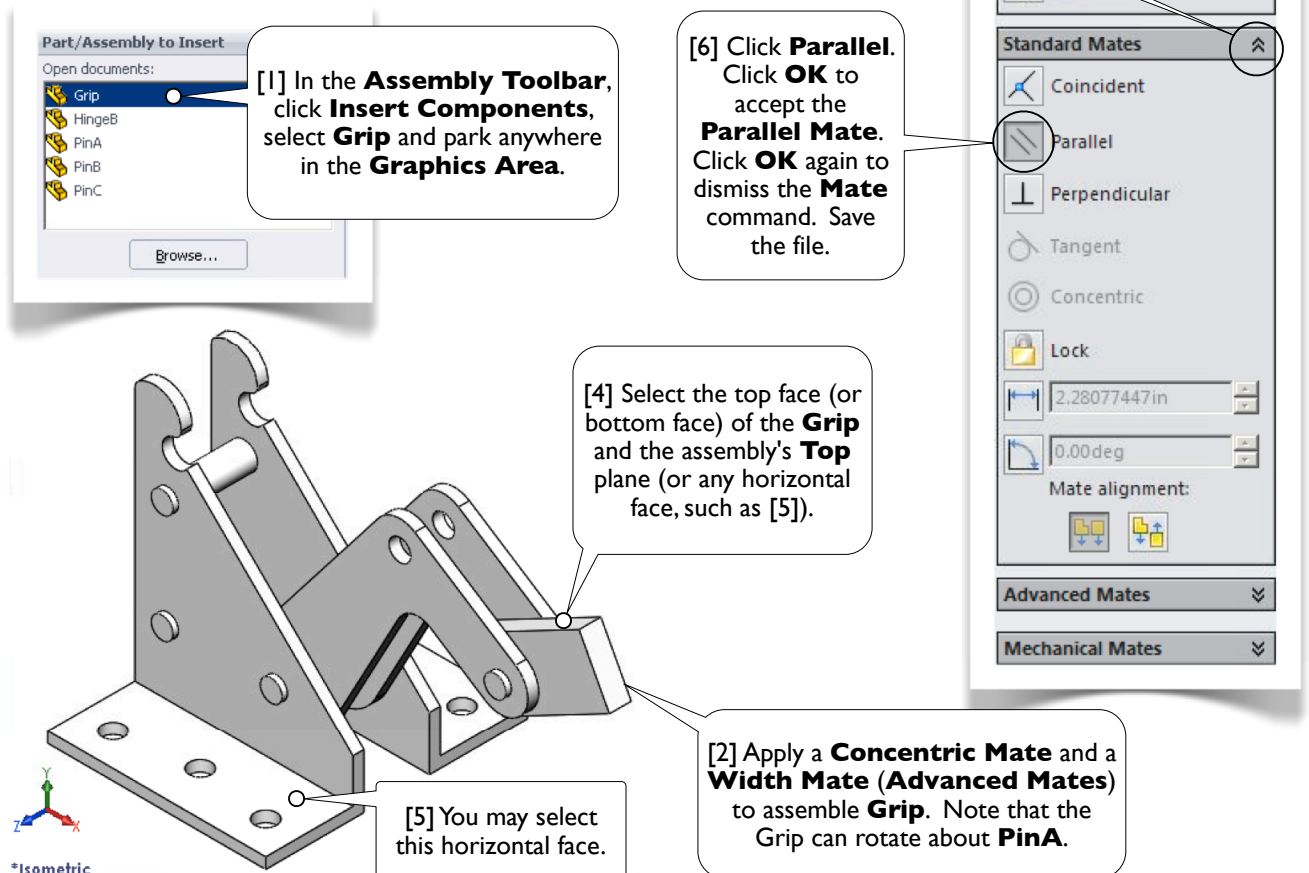


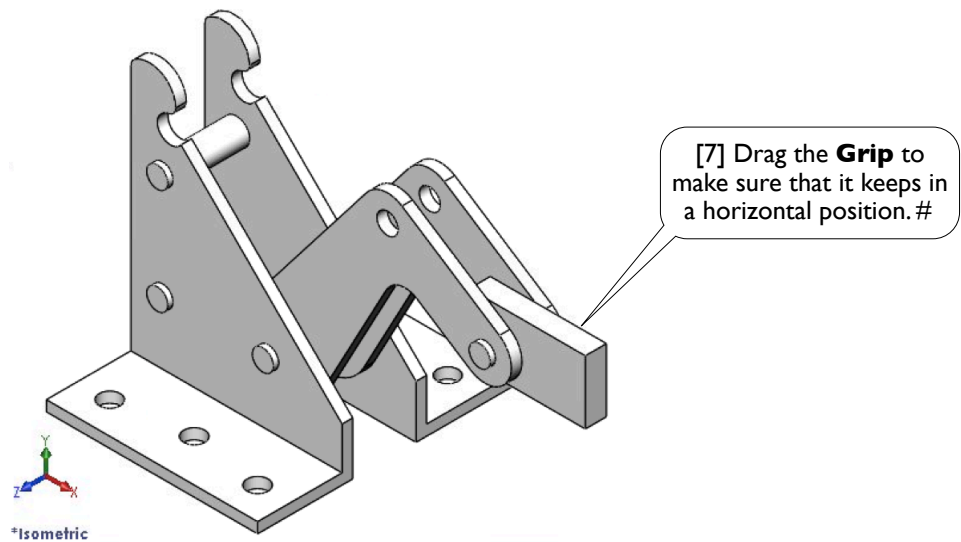
*Isometric

3.3-13 Assemble **PinA**

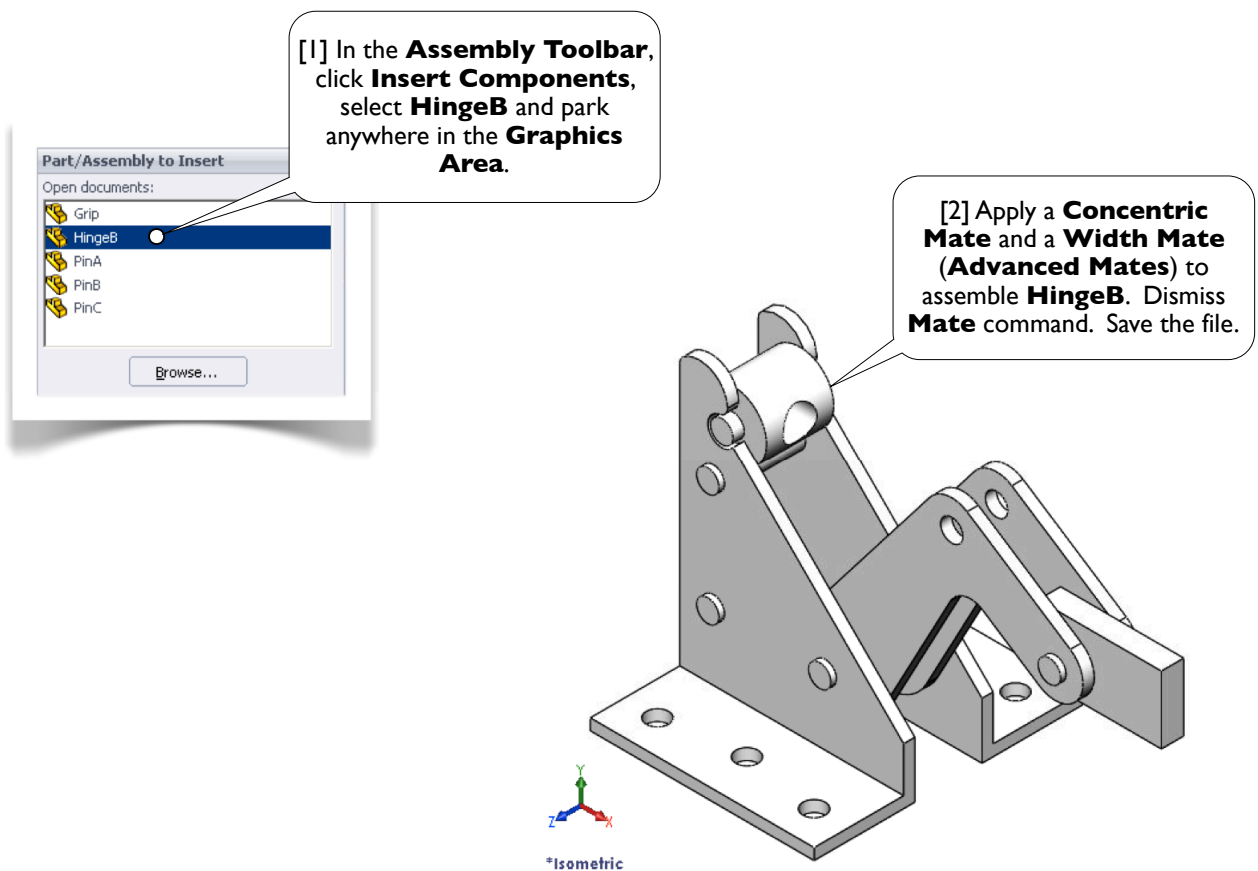


3.3-14 Assemble **Grip**

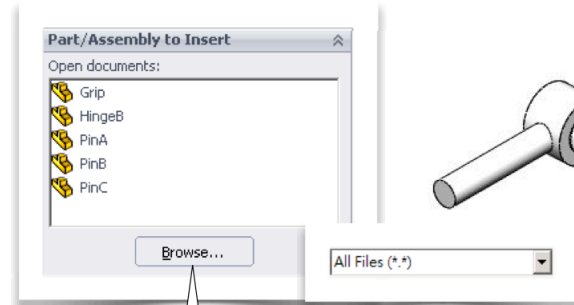




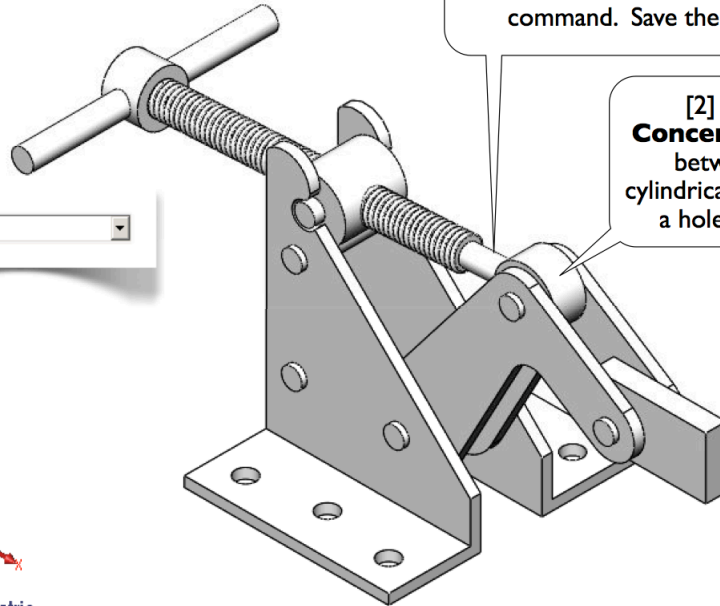
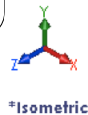
3.3-15 Assemble **HingeB**



3.3-16 Assemble **ShaftAssembly**



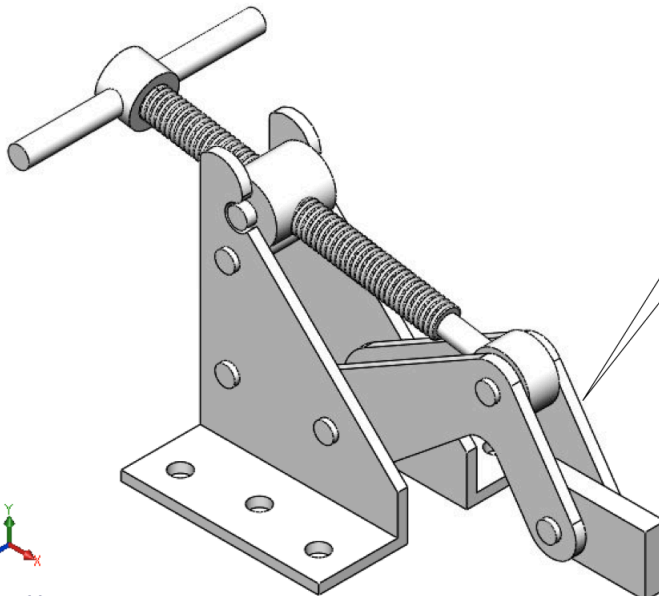
[1] In the **Assembly Toolbar**, click **Insert Components**, browse and open **ShaftAssembly**, which was saved in Section 3.1, and park anywhere in the **Graphics Area**.



[3] And apply a **Concentric Mate** between the hole of **HingeB** and a cylindrical surface of **Shaft**. Dismiss **Mate** command. Save the file.

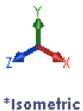
[2] Apply a **Concentric Mate** between this cylindrical surface and a hole in **Arm**.

3.3-17 Test the Clamping Mechanism



[1] Drag any component to see how the mechanism works. Note that the **Grip** keeps horizontal.

[2] Exit **SOLIDWORKS**.#



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