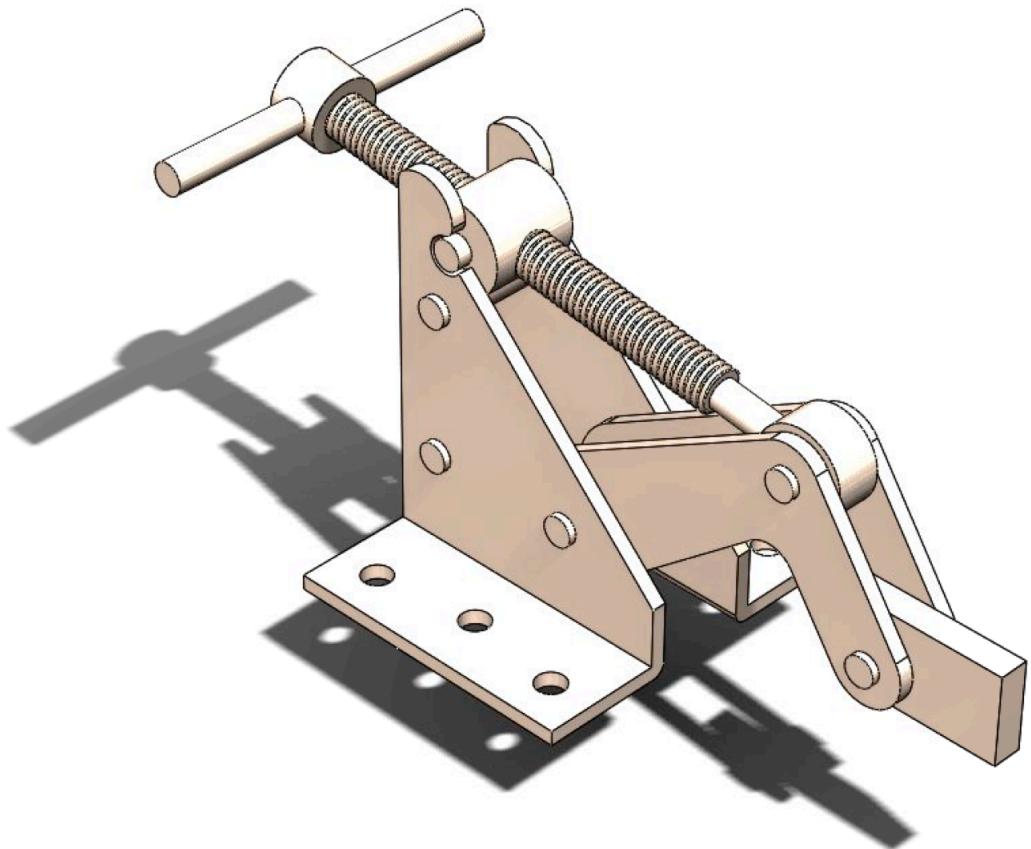


Part and Assembly Modeling

with SOLIDWORKS 2015

Huei-Huang Lee



Contents

Preface 1

Chapter 1 Sketching 2

- 1.1 Arm 3
- 1.2 Ratchet Wheel 17
- 1.3 Ratchet Stop 23
- 1.4 Cover Plate 28

Chapter 2 Part Modeling 36

- 2.1 Crank 37
- 2.2 Geneva Gear Index 43
- 2.3 Yoke 50
- 2.4 Support 56
- 2.5 Wheel 62
- 2.6 Transition Pipe 66
- 2.7 Threaded Shaft 75
- 2.8 Lifting Fork 80

Chapter 3 Assembly Modeling 86

- 3.1 Shaft Assembly 87
- 3.2 Universal Joint 97
- 3.3 Clamp 107

Index 119

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.

Huei-Huang Lee

Associate Professor

Department of Engineering Science

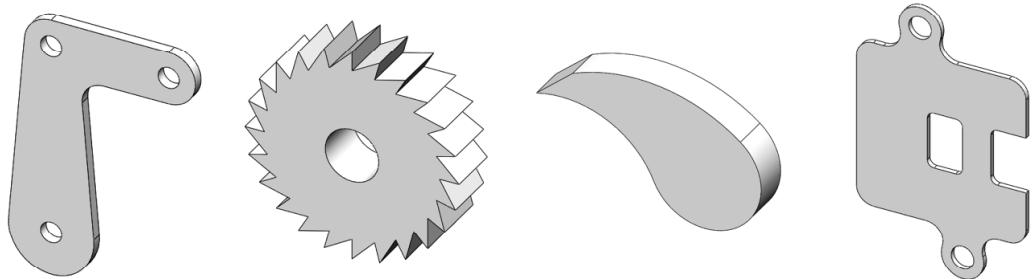
National Cheng Kung University, Tainan, Taiwan

e-mail: hhlee@mail.ncku.edu.tw

webpage: myweb.ncku.edu.tw/~hhlee

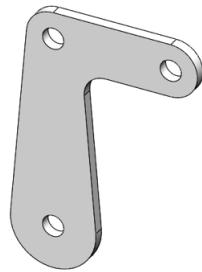
Chapter I

Sketching

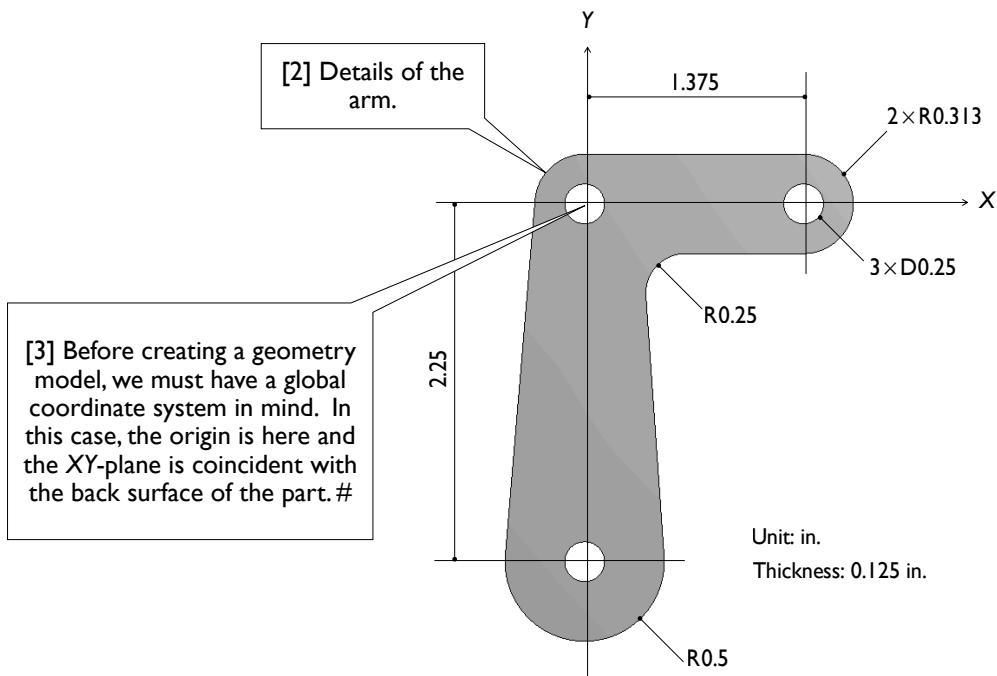
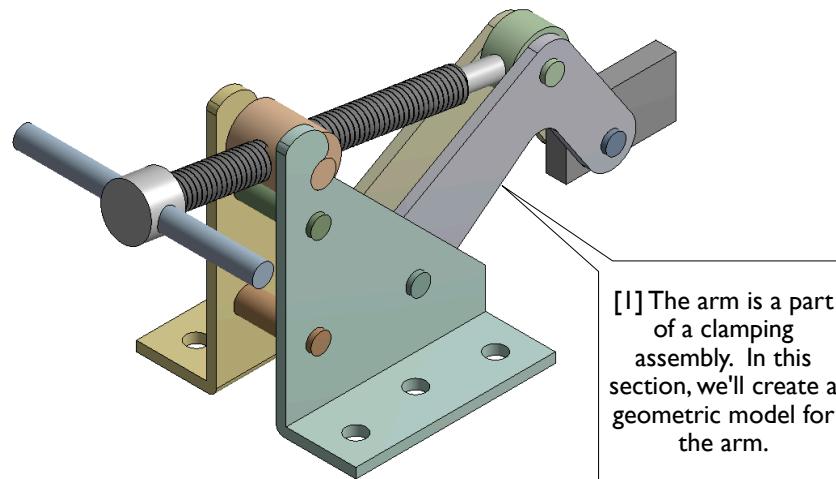


Section I.I

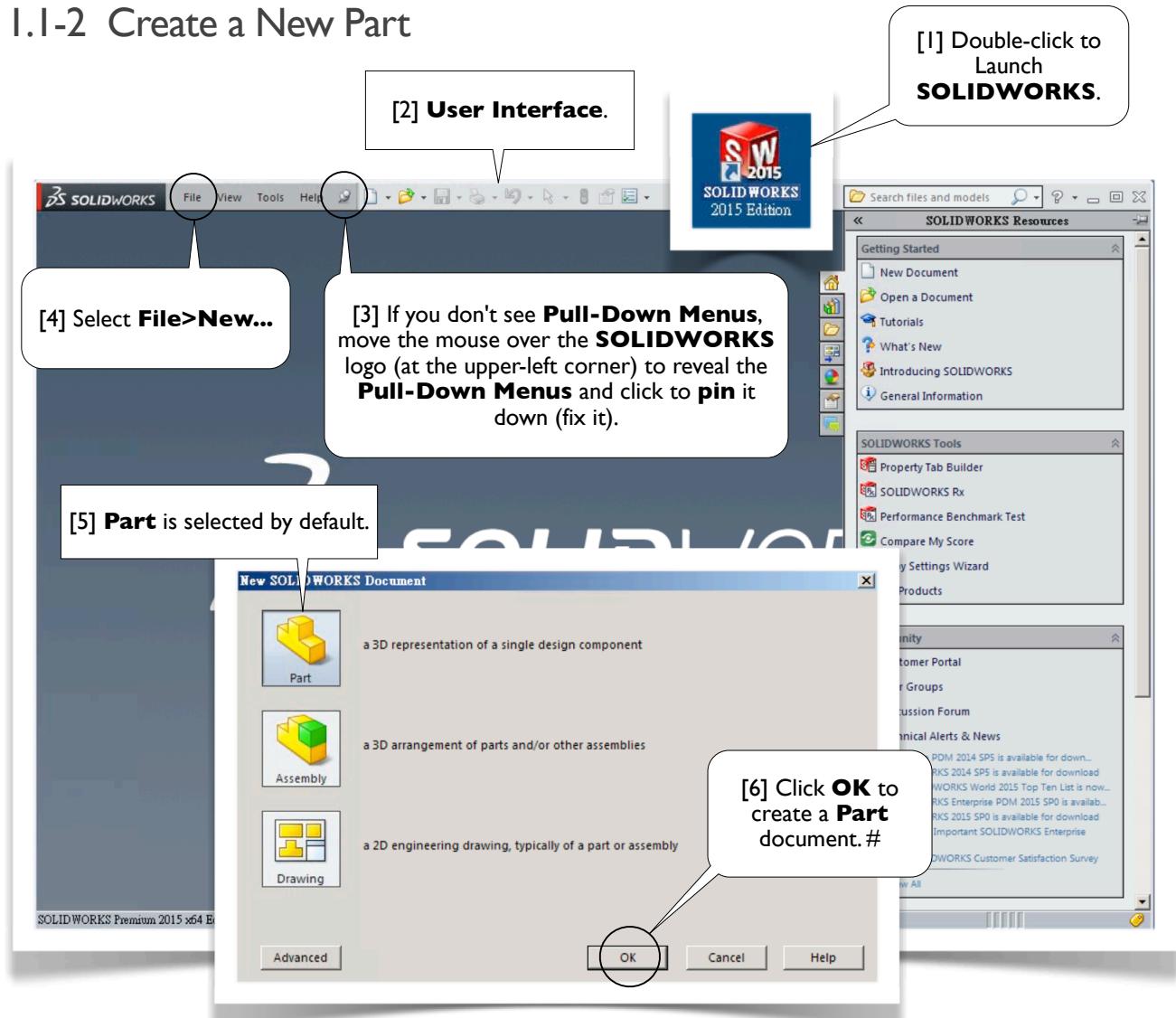
Arm



I.I-1 About the Arm



I.I-2 Create a New Part



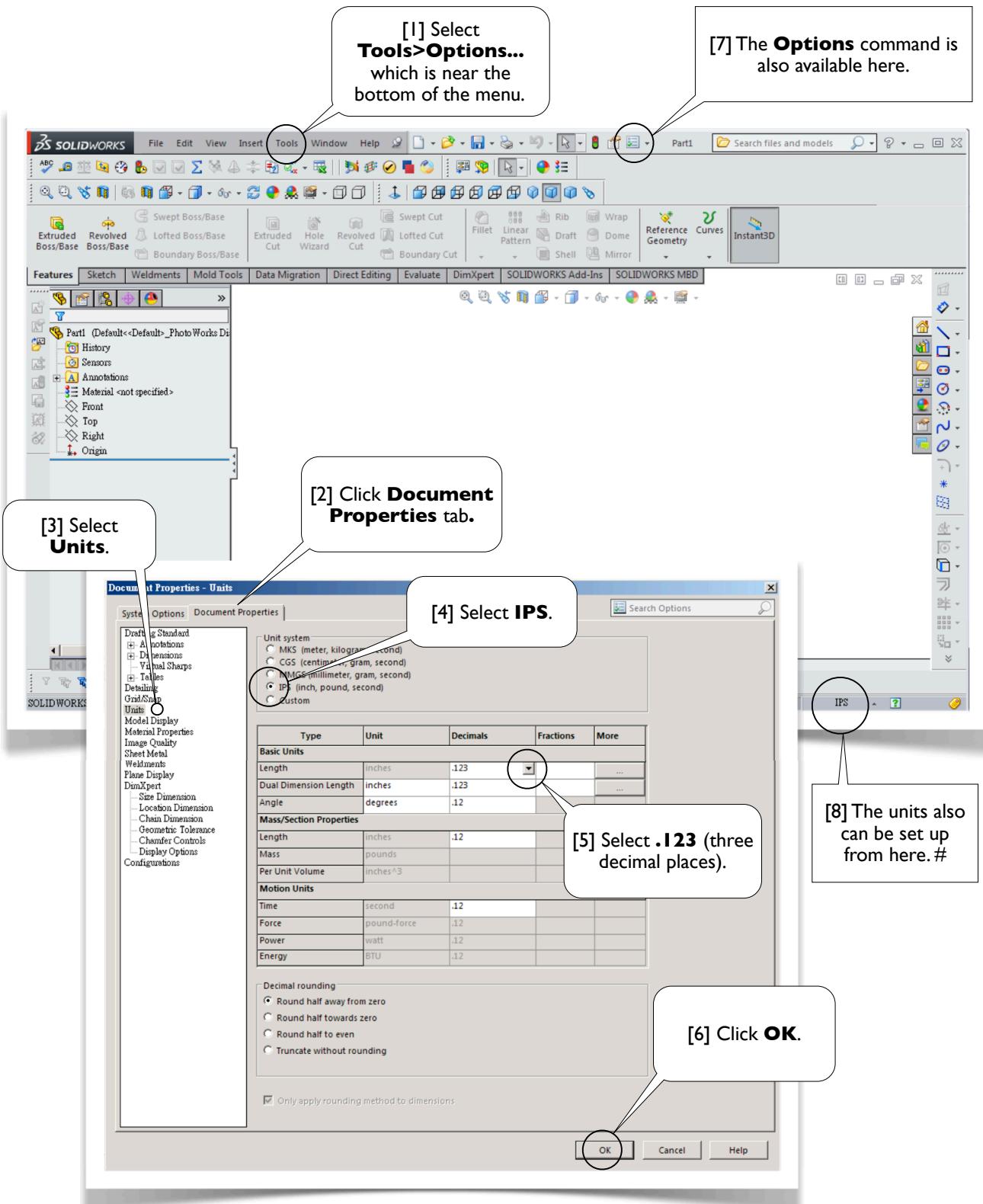
About the Textboxes

1. Within each subsection (e.g., I.I-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., I.I-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.

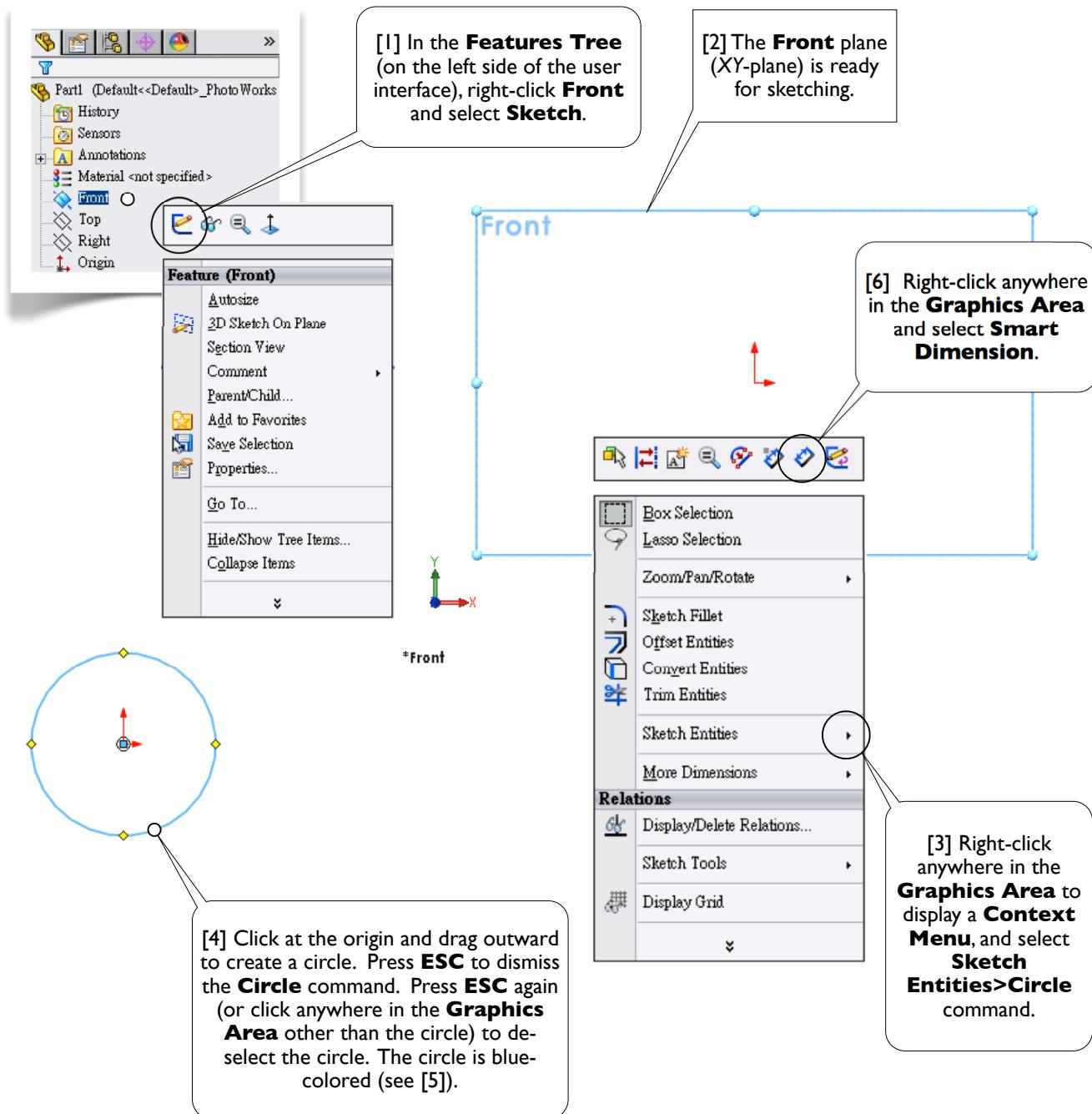
SOLIDWORKS Terms

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

I.I-3 Set Up Units



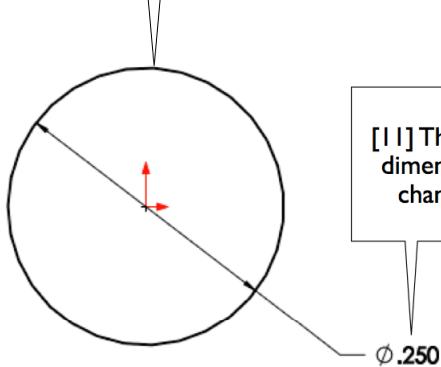
I.I-4 Draw a Circle



[5] Color Codes of Sketch Entities

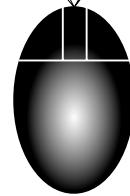
A sketch entity is blue-colored (either light-blue or dark-blue) when it is not yet well-defined [4]. A well-defined entity (i.e., fixed in the space) becomes black (e.g., [7], next page). When over-defined, an entity becomes red.

[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**.

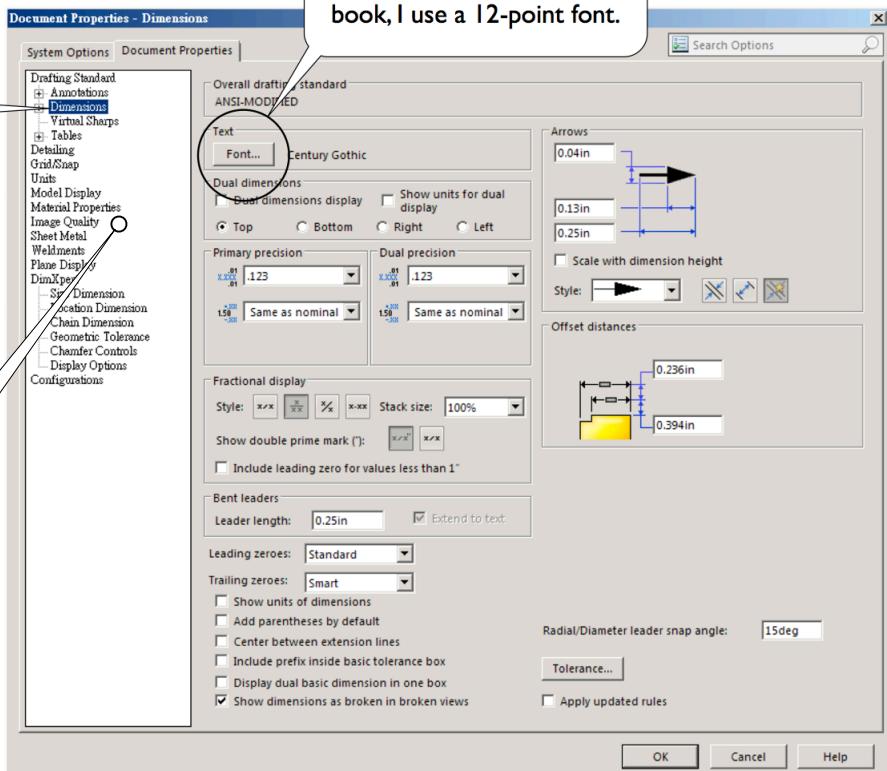


[8] Scrolling the **Mouse Wheel** allows you to zoom in/out the sketch.

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



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



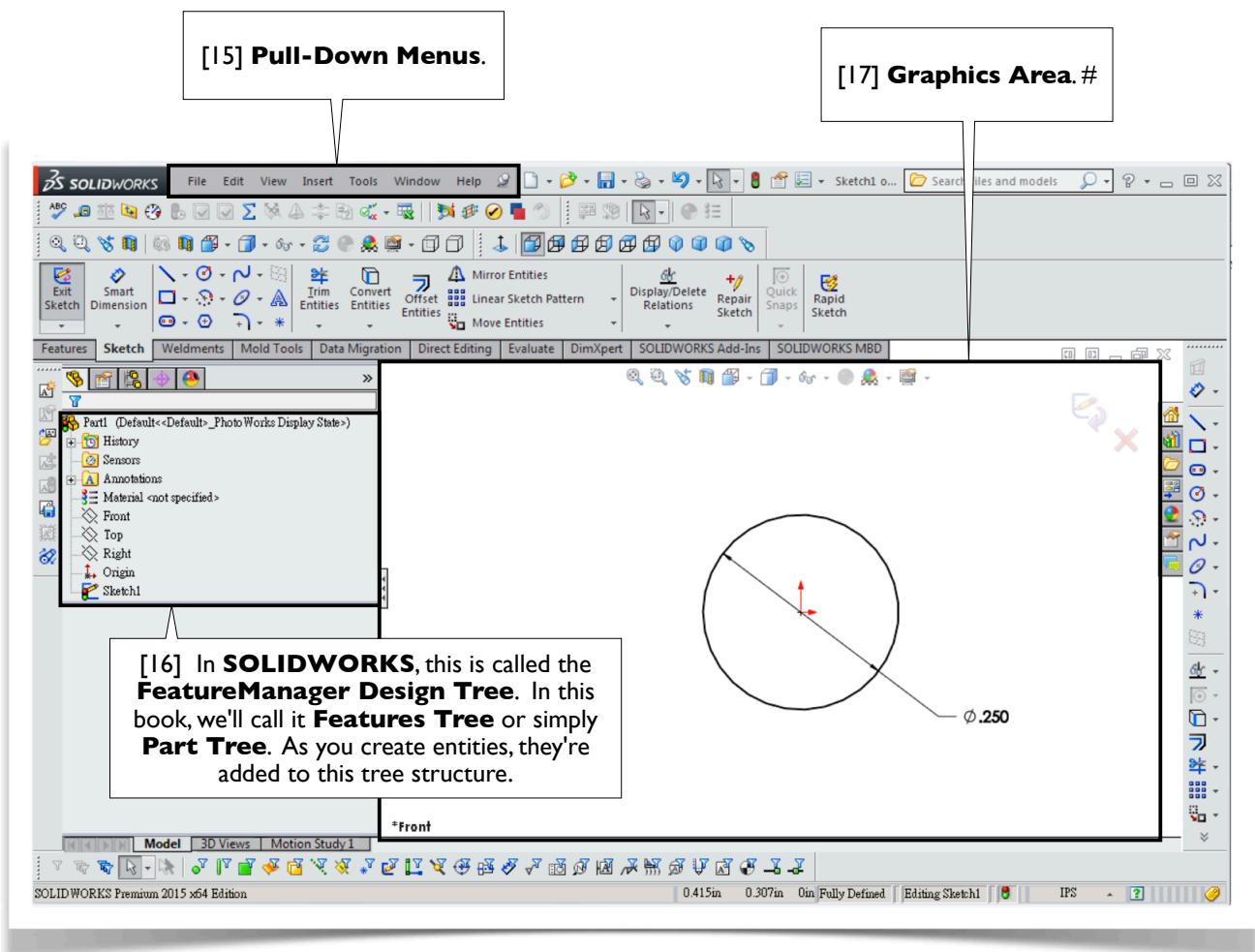
[14] **Image Quality** can be used to improve the smoothness of the model.#

SOLIDWORKS Commands

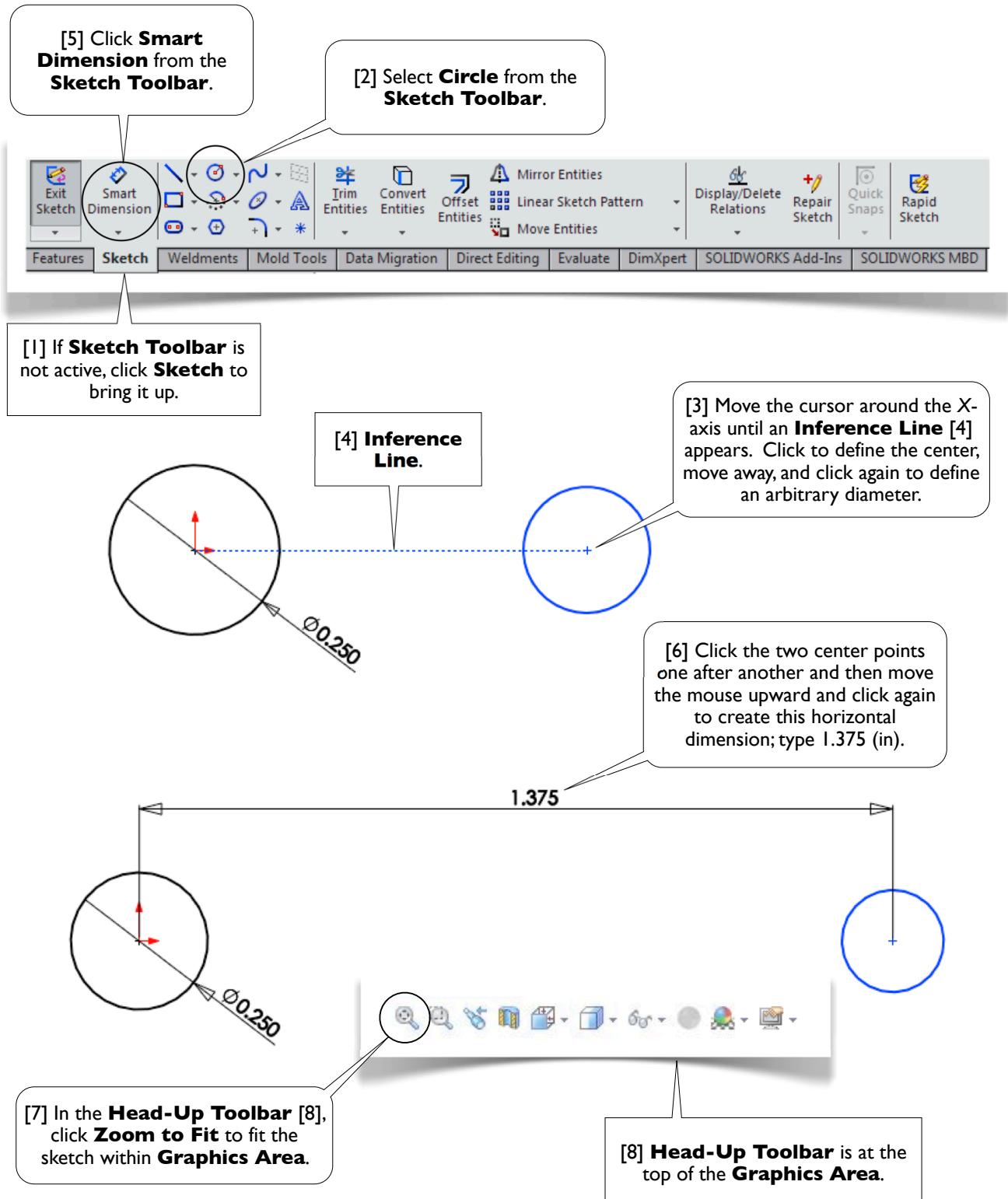
There are literally hundreds of **SOLIDWORKS** commands (tools). All commands can be found in the **Pull-Down Menus** [15]. Nevertheless, the most intuitive way to issue a command is through a context-sensitive menu, or simply called **Context Menu** [1, 3, 6] (page 6). To issue a command with a **Context 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**.

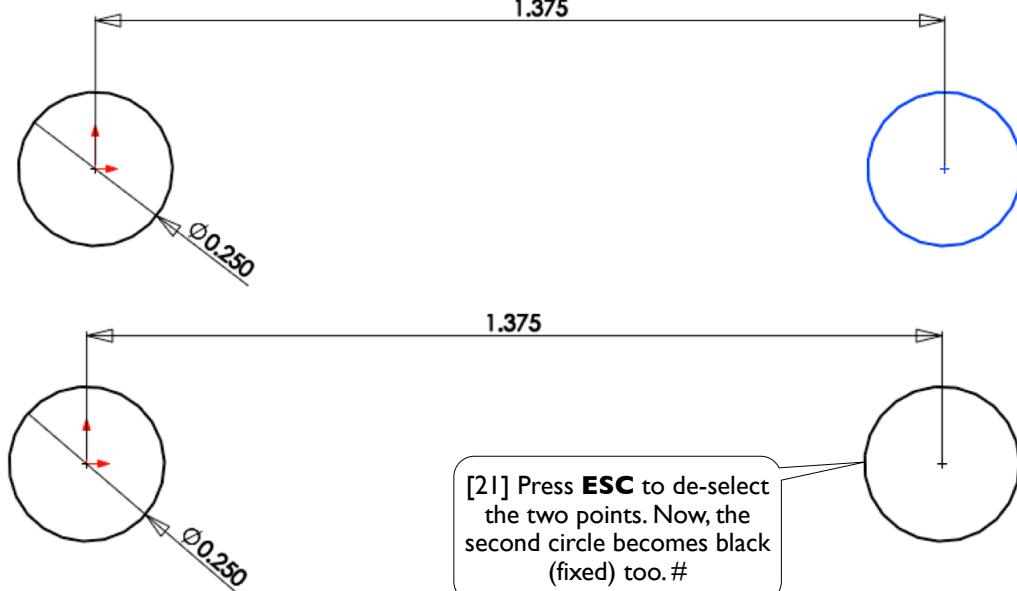
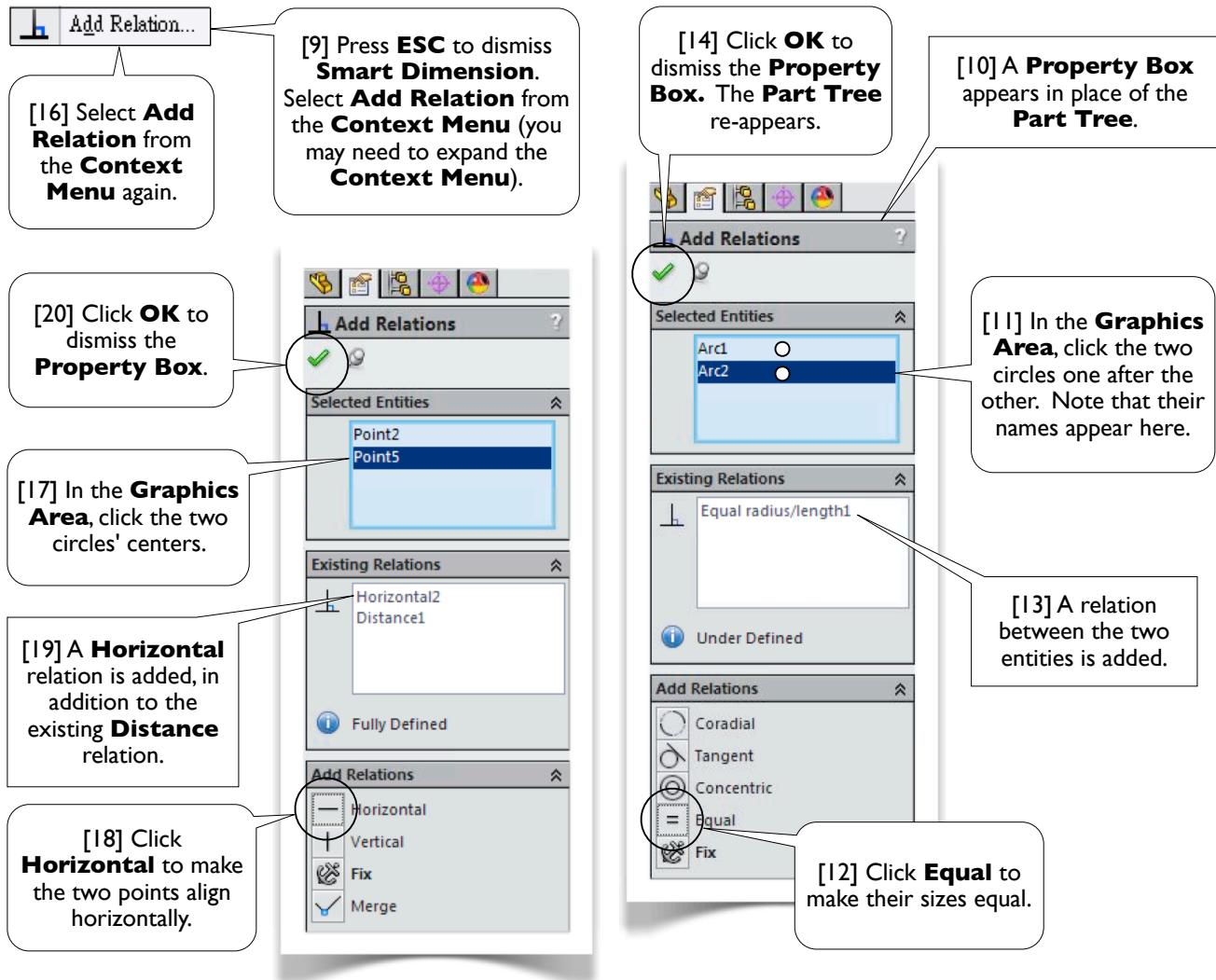
After you accumulate some experiences, you may find that a more convenient way to issue a command is simply clicking a command on a **Toolbar** (e.g., [10], last page). In this book, we roughly follow these rules to issue a command:

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.

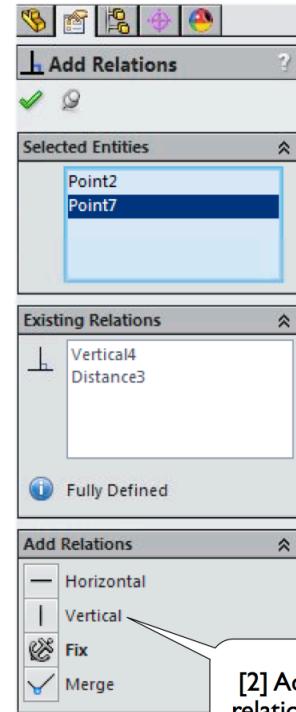
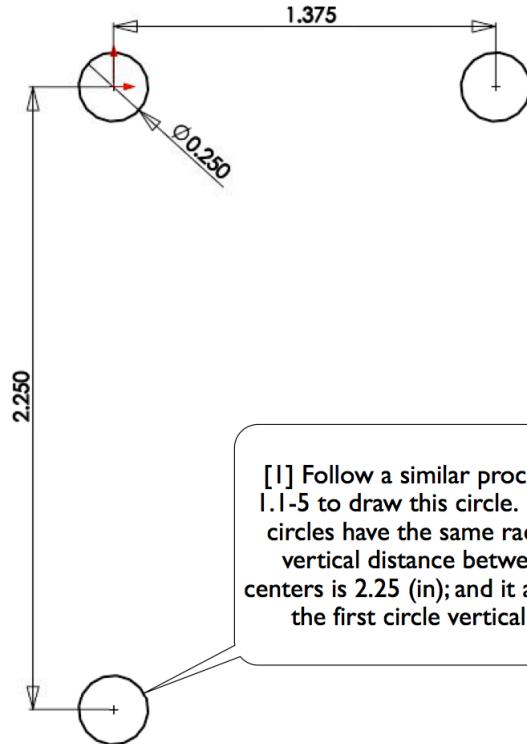


1.1-5 Draw Another Circle

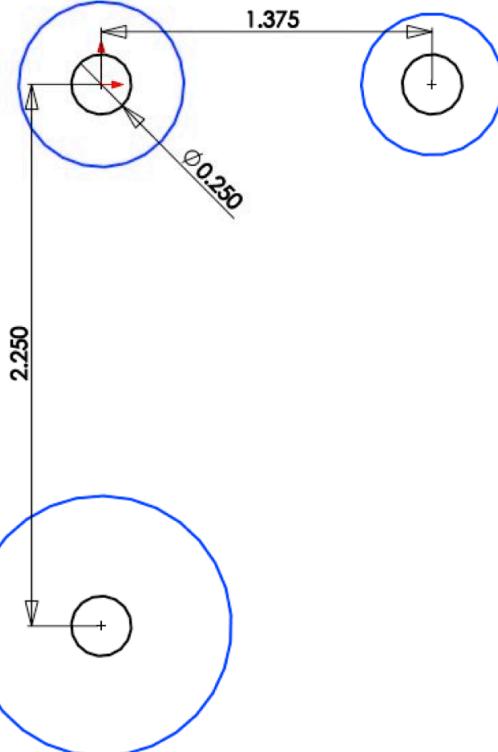


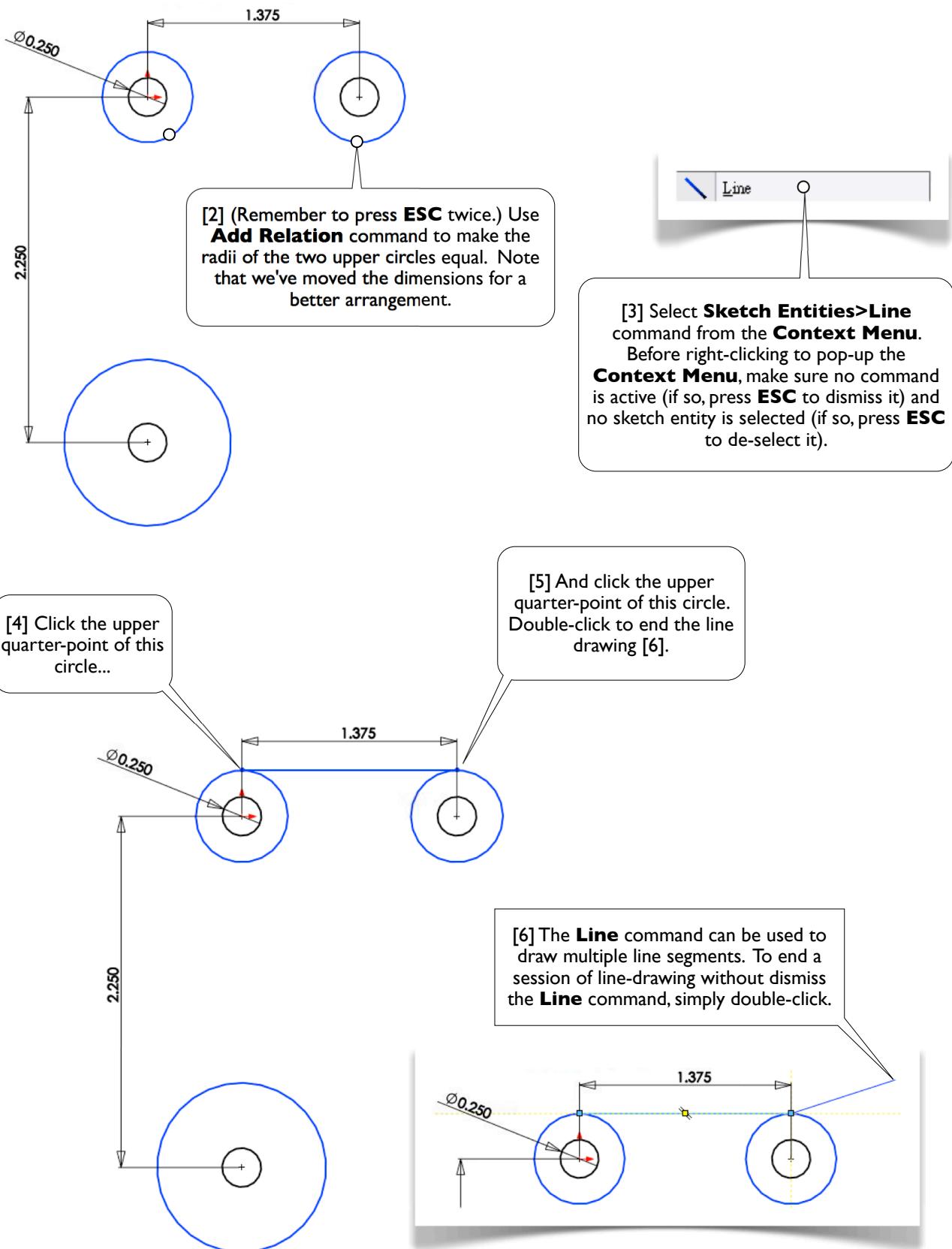


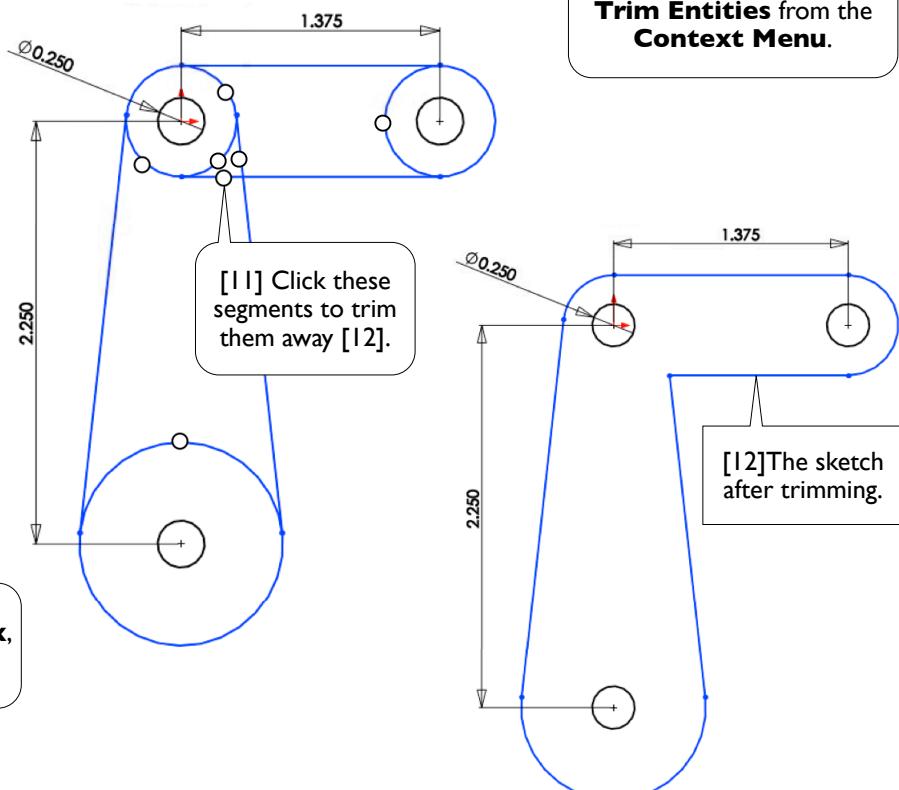
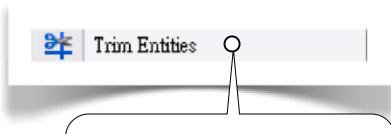
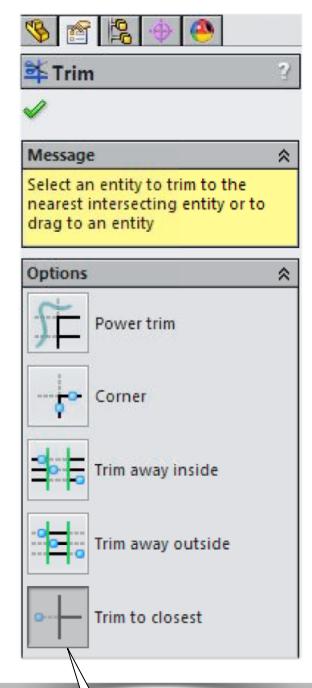
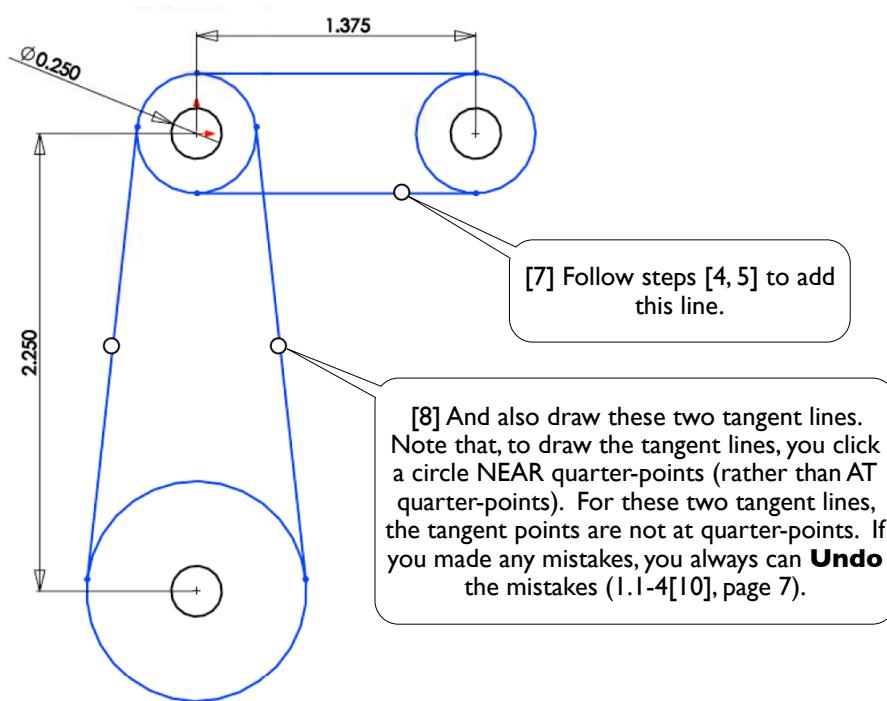
I.I-6 Draw the Third Circle

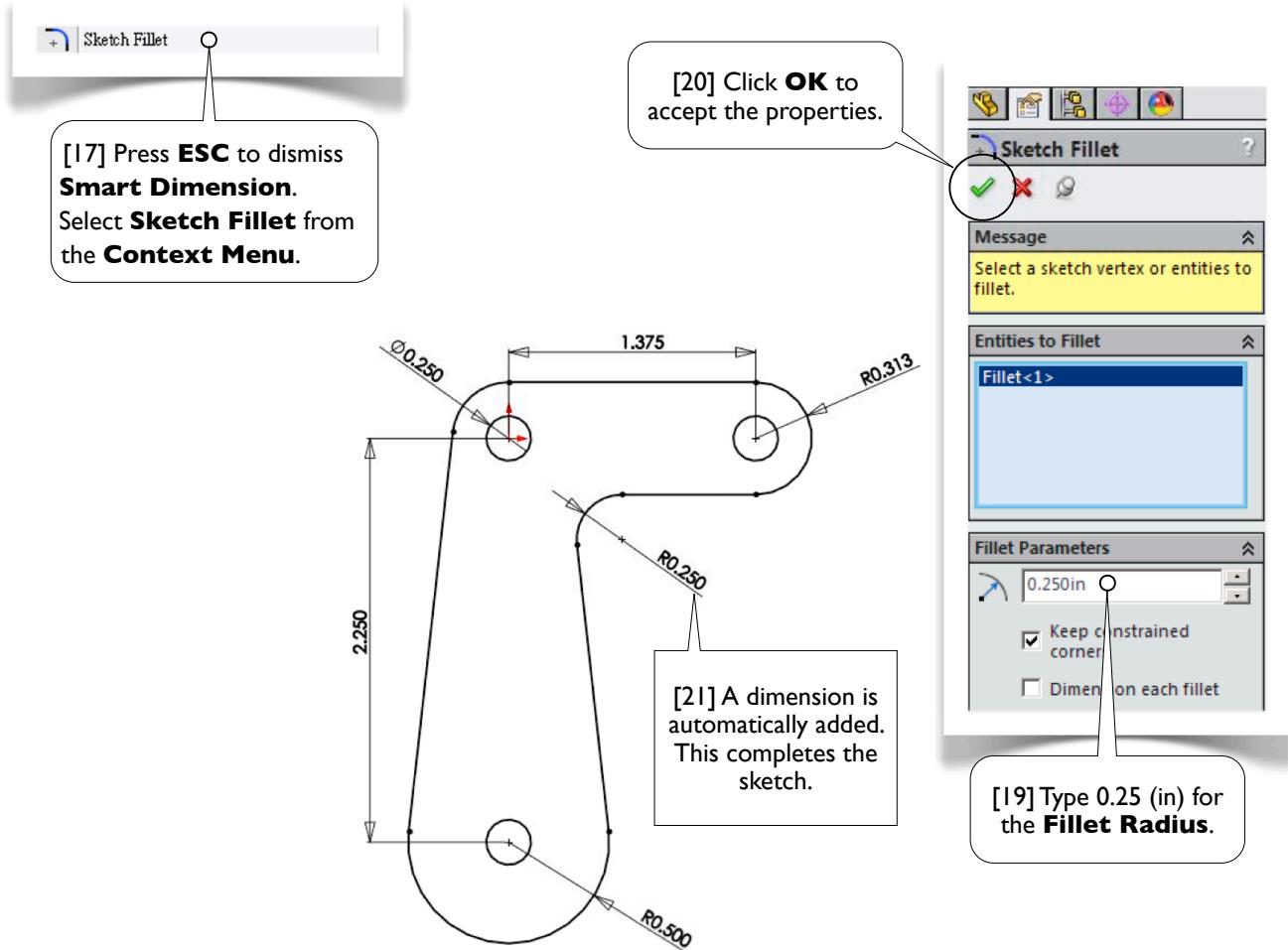
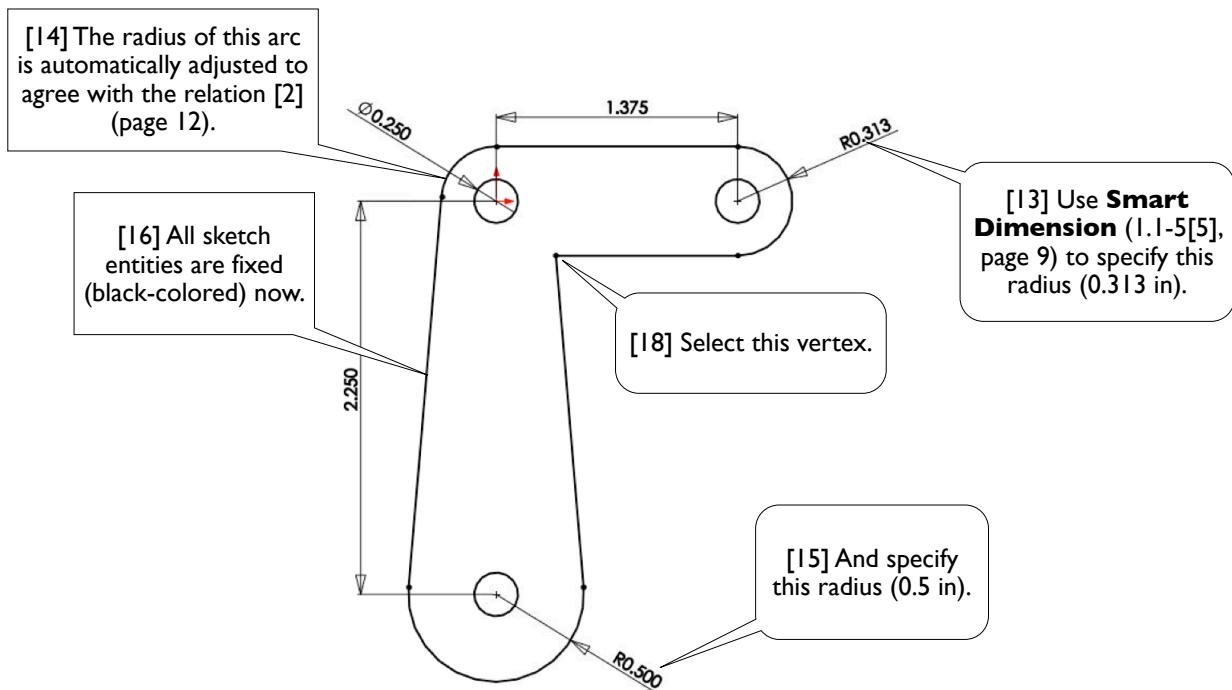


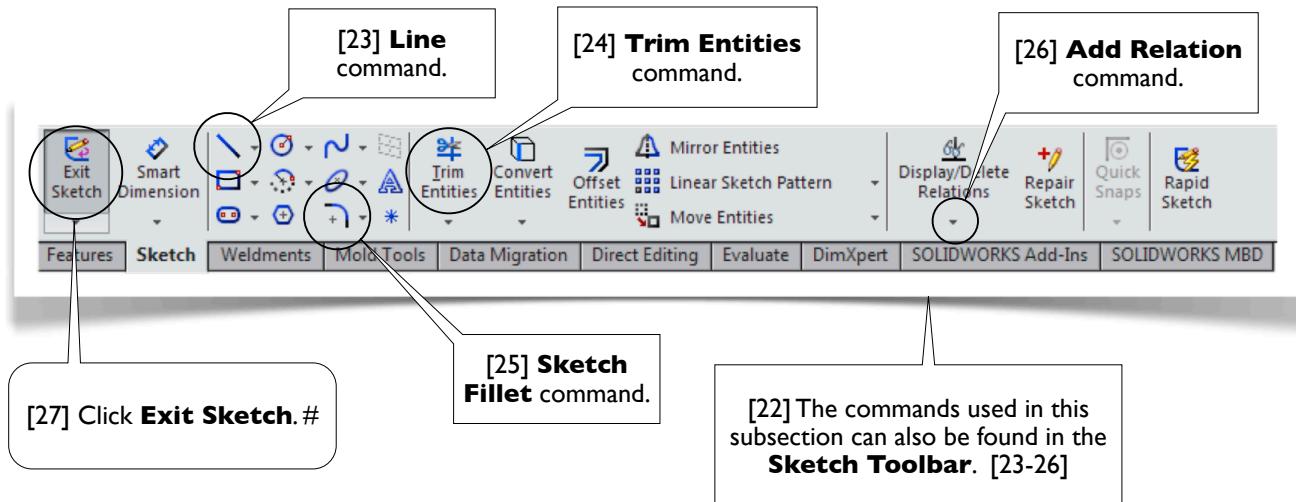
I.I-7 Complete the Sketch



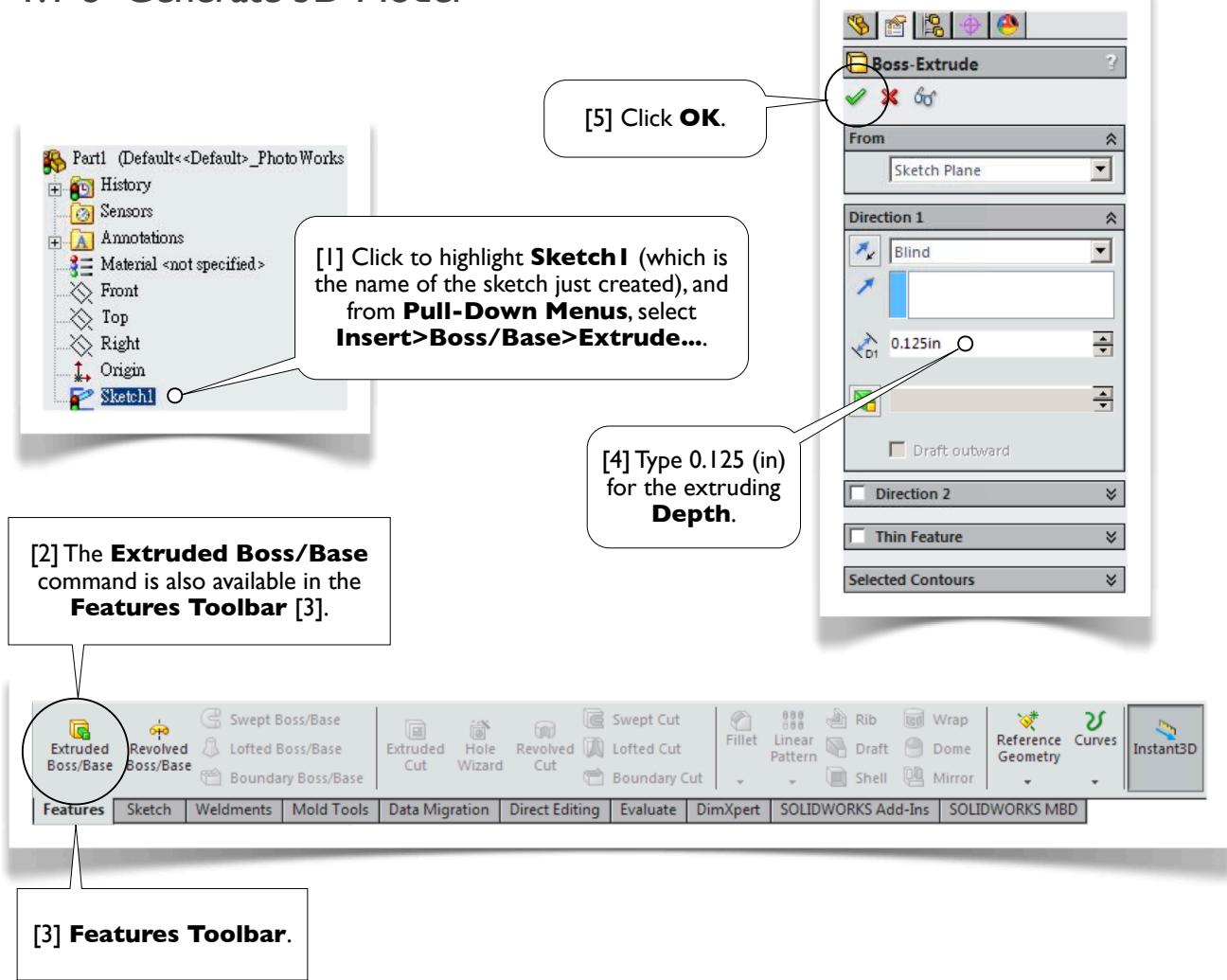


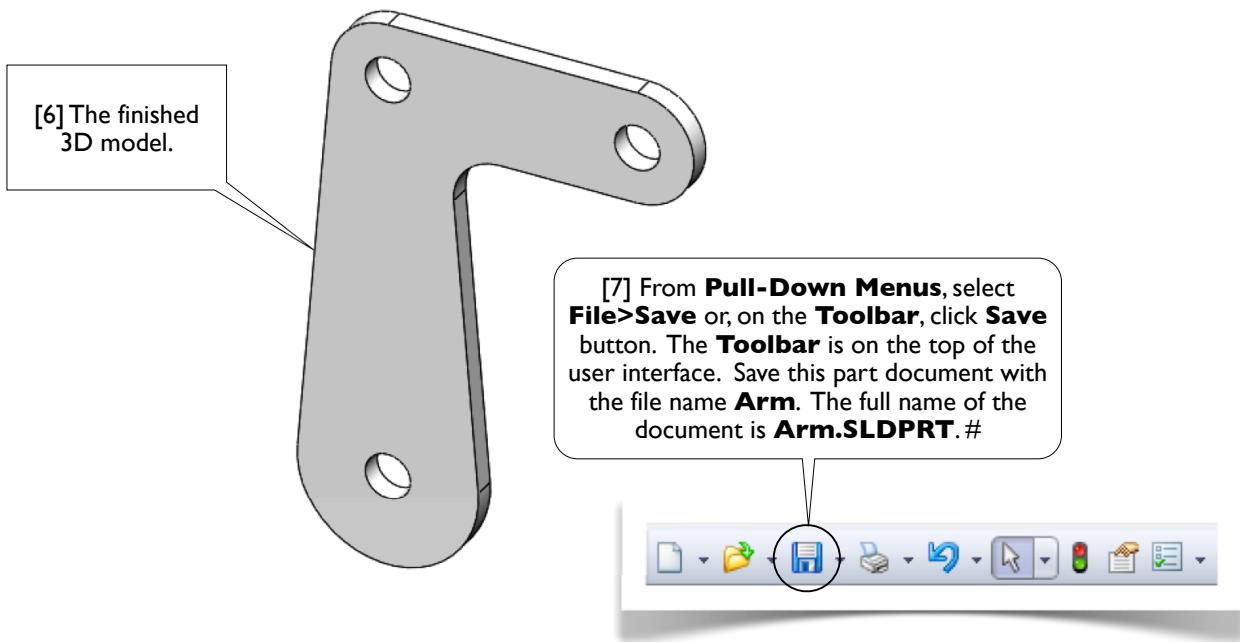




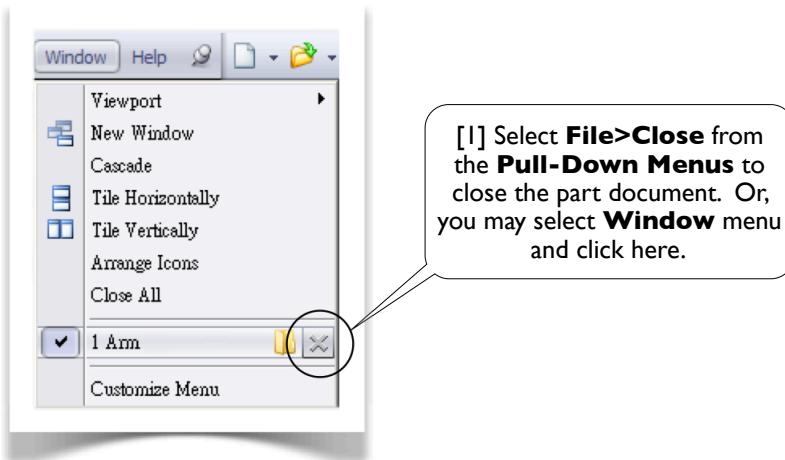


I.I-8 Generate 3D Model





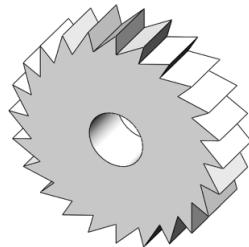
I.I-9 Wrap Up



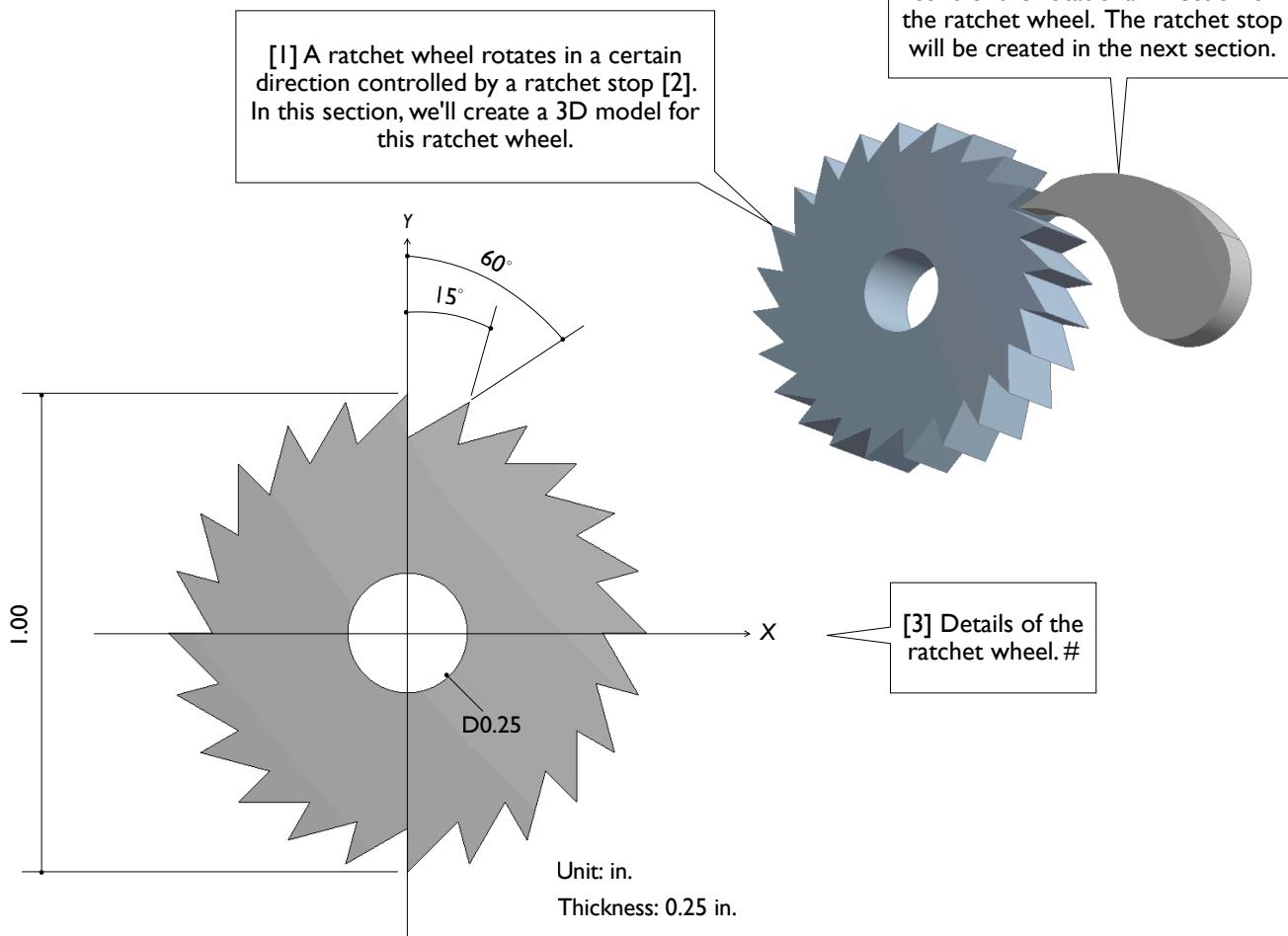
[2] Select **File>Exit** from **Pull-Down Menus** to quit **SOLIDWORKS**.#

Section 1.2

Ratchet Wheel



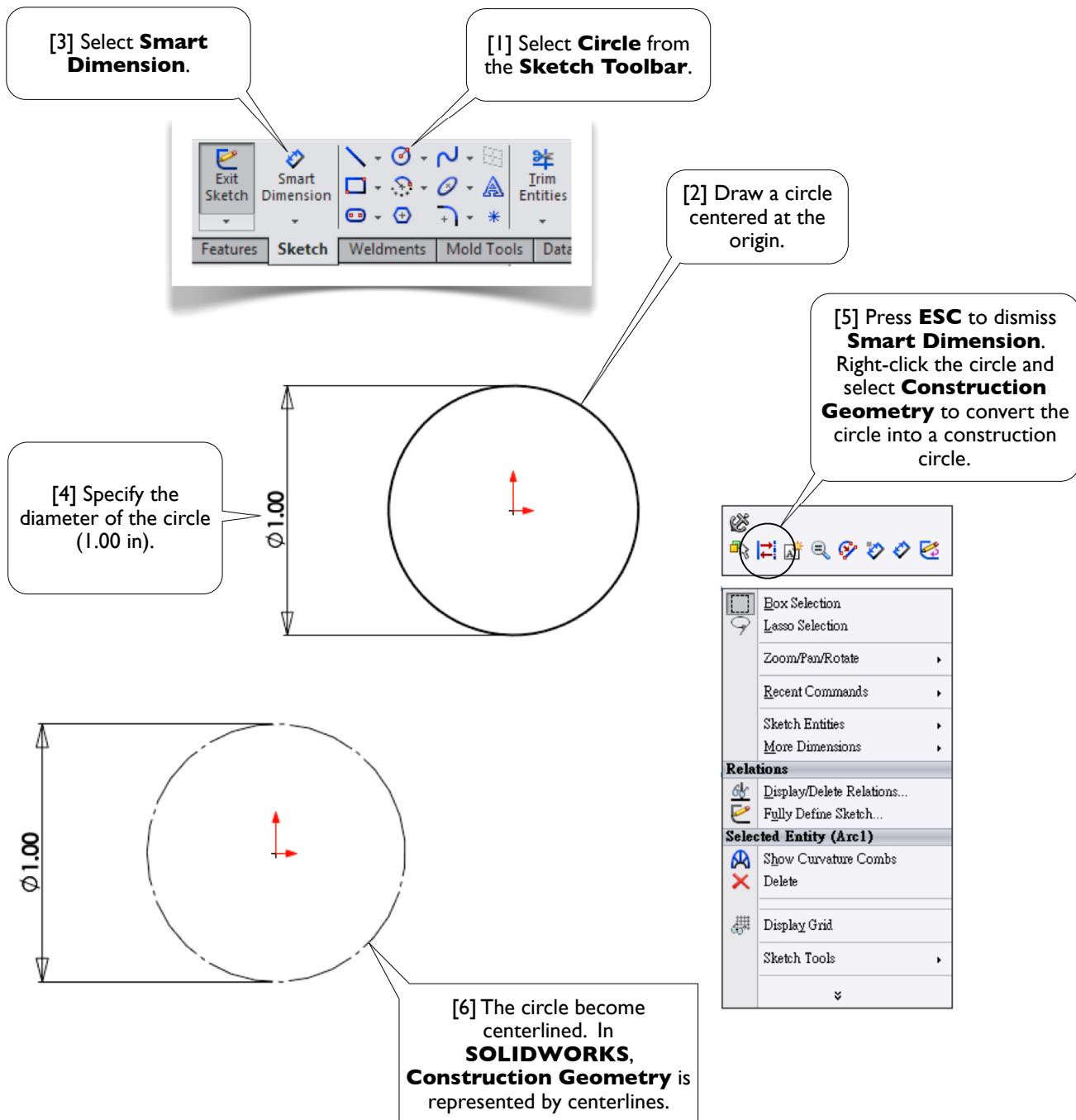
1.2-1 About the Ratchet Wheel



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).#

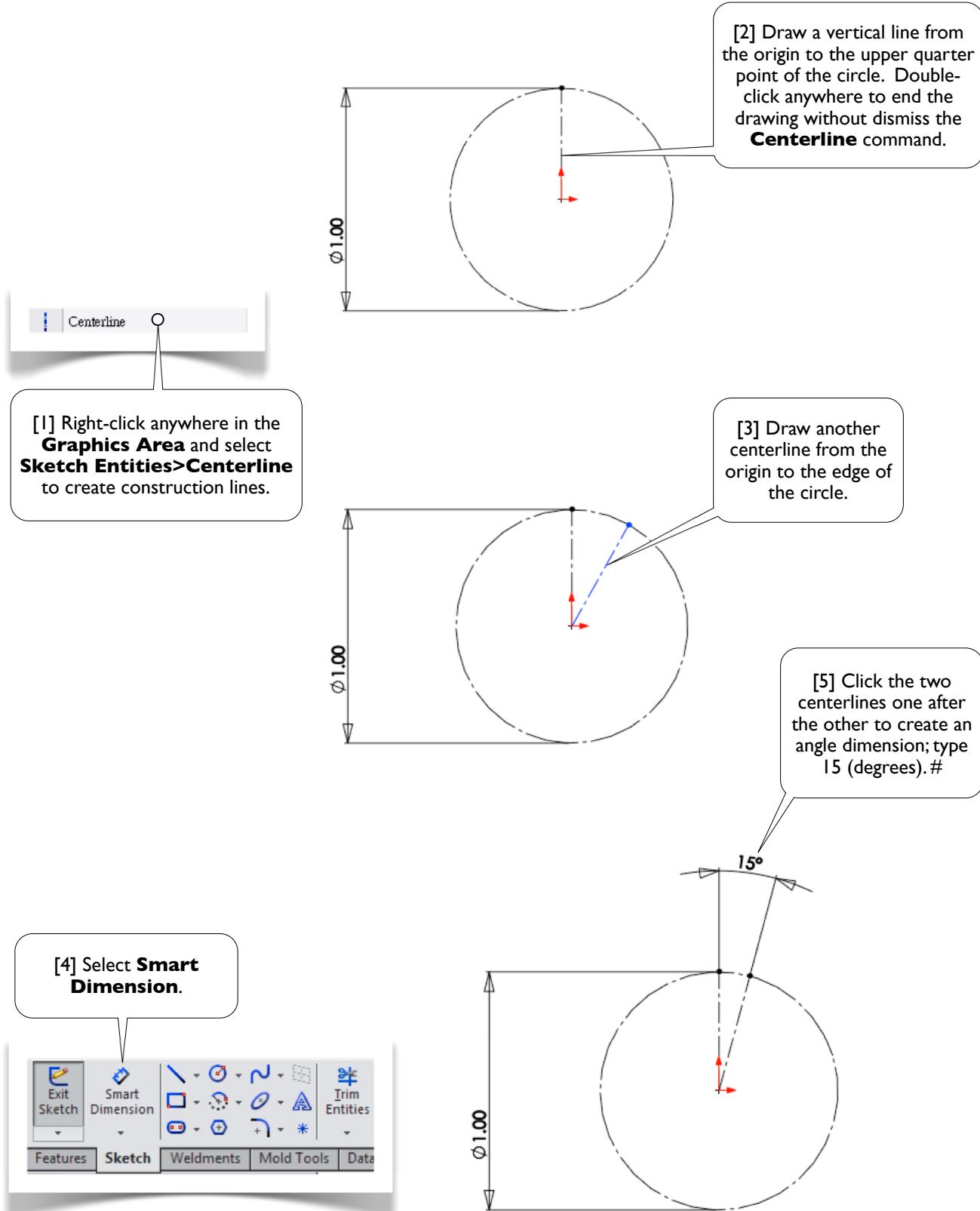
1.2-3 Draw a Construction Circle



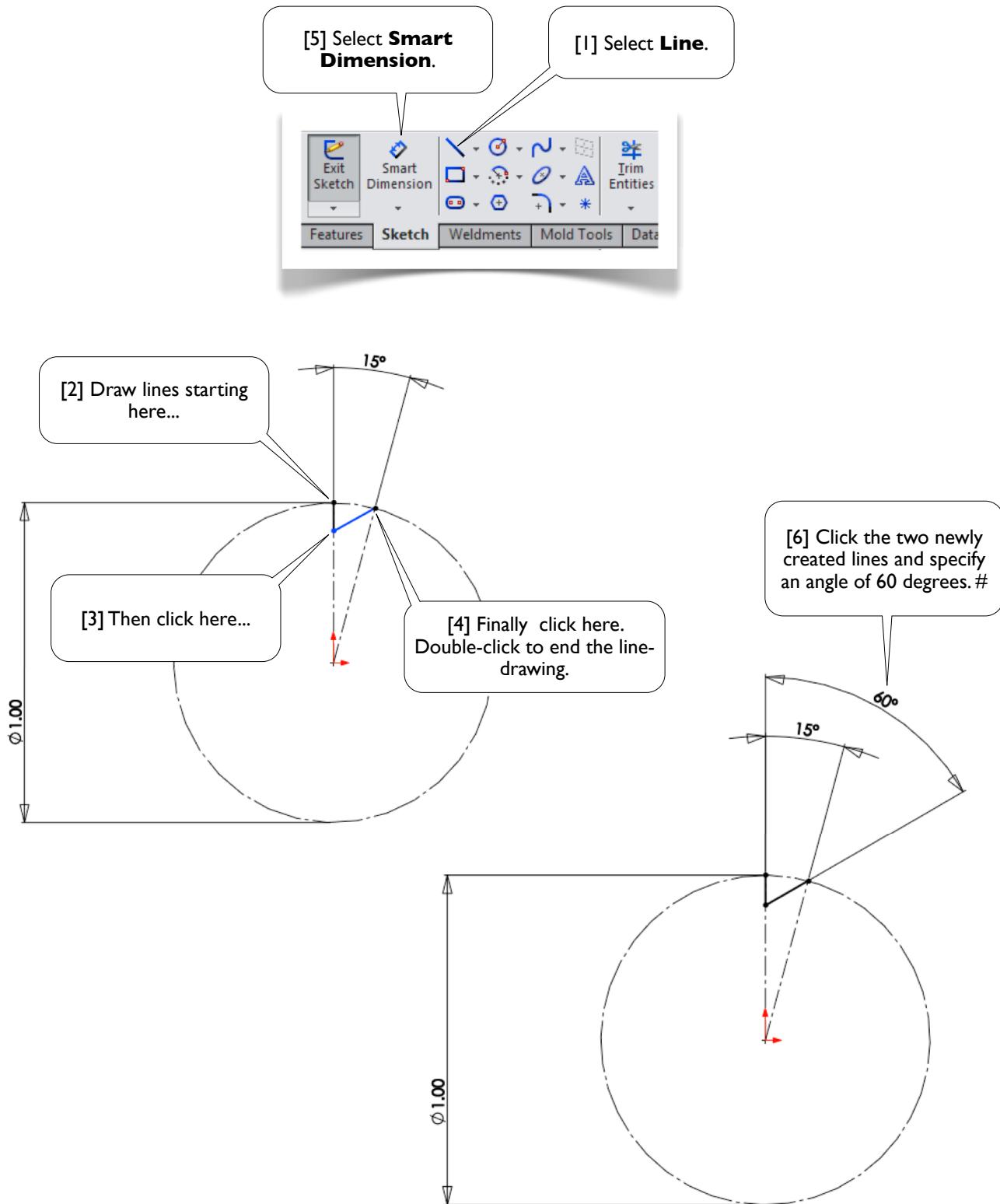
[7] Construction Geometry

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

[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**.

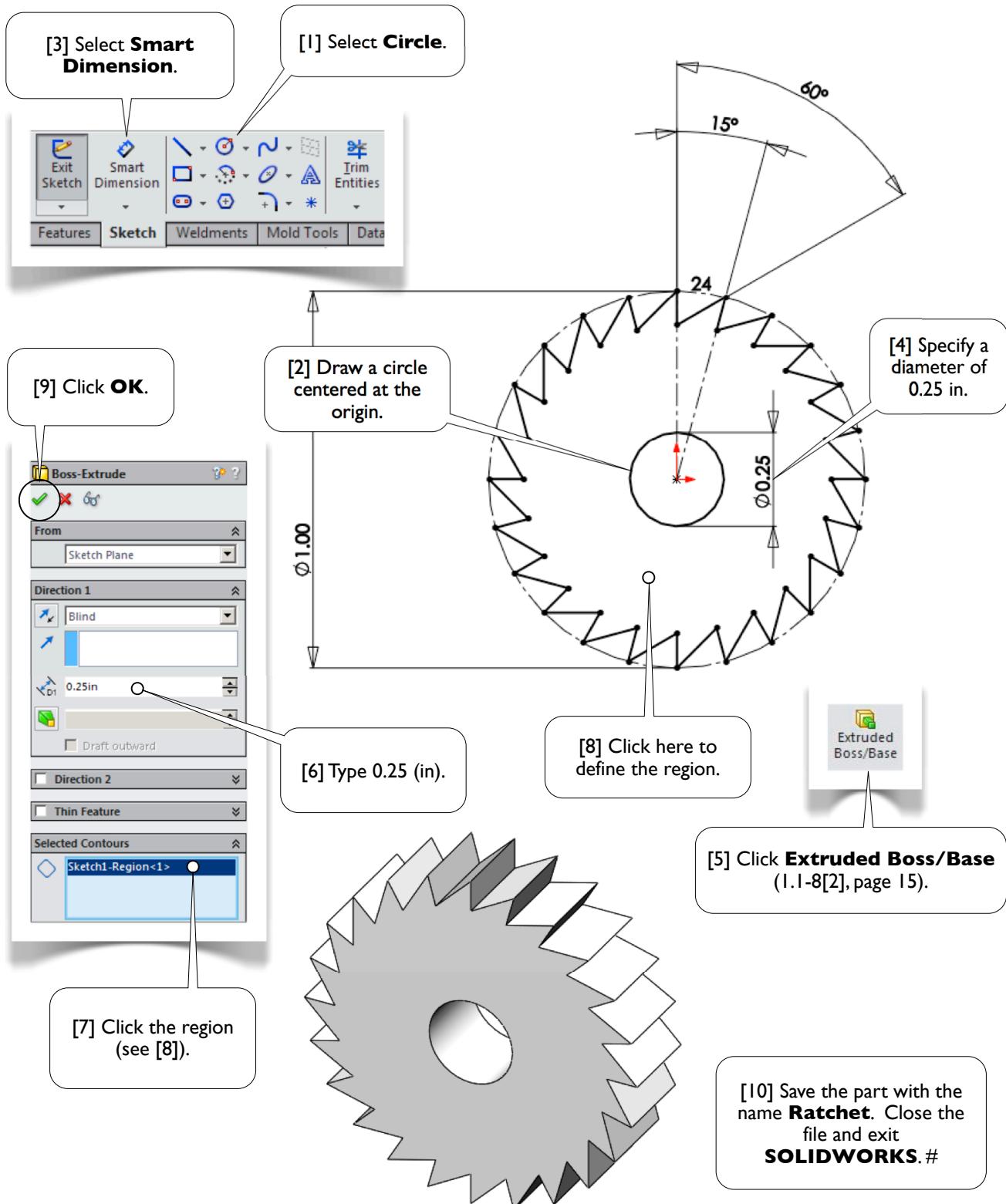
[3] Right-click this box and select **Clear Selections** from the **Context Menu** and then select the two line segments created in 1.2-5 (last page) for **Entities to Pattern**.

[4] Click **OK**.

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

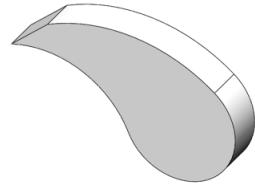
The image shows the SOLIDWORKS interface with the 'Circular Pattern' dialog box open. The 'Parameters' tab is selected, showing 'Point-1' as the center point, '0in' for both X and Y offsets, '360deg' for the angle, '24' for the number of instances, and '0.41in' for the dimension radius. The 'Entities to Pattern' tab shows 'Line5' and 'Line6' selected. A callout points to the 'Number of Instances' field with the instruction [2]. Another callout points to the 'Entities to Pattern' list with the instruction [3]. A callout points to the 'OK' button with the instruction [4]. A callout points to the 'Linear Sketch Pattern' icon in the toolbar with the instruction [5]. To the right, the sketch of a ratchet wheel tooth profile is shown, consisting of a circular arc with a radius of 1.00, a vertical line segment of height 0.41in, and a curved profile with 24 segments. Dimension lines indicate the 1.00 radius and the 0.41in height.

1.2-7 Finished the Sketch and Generate 3D Model

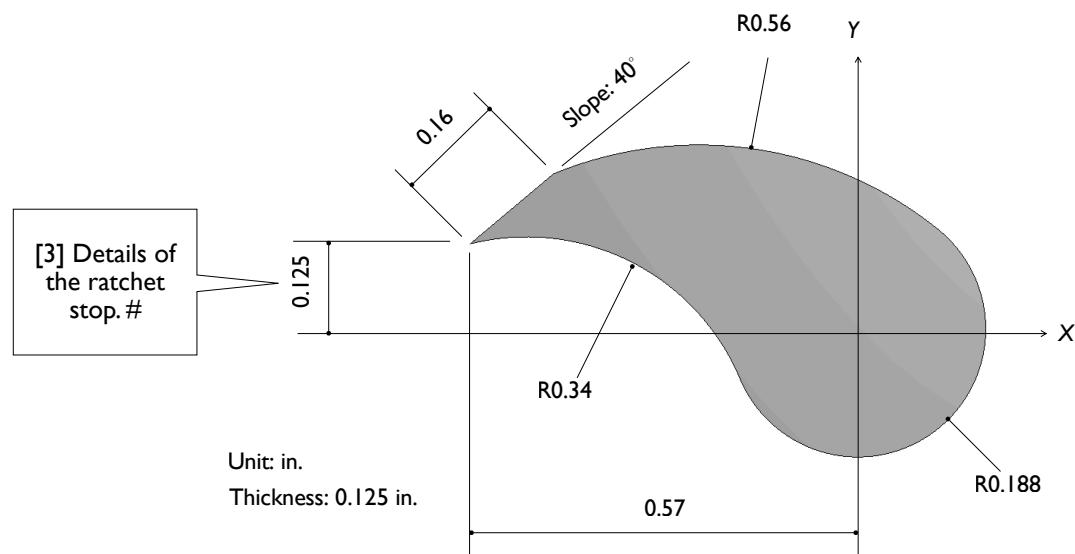
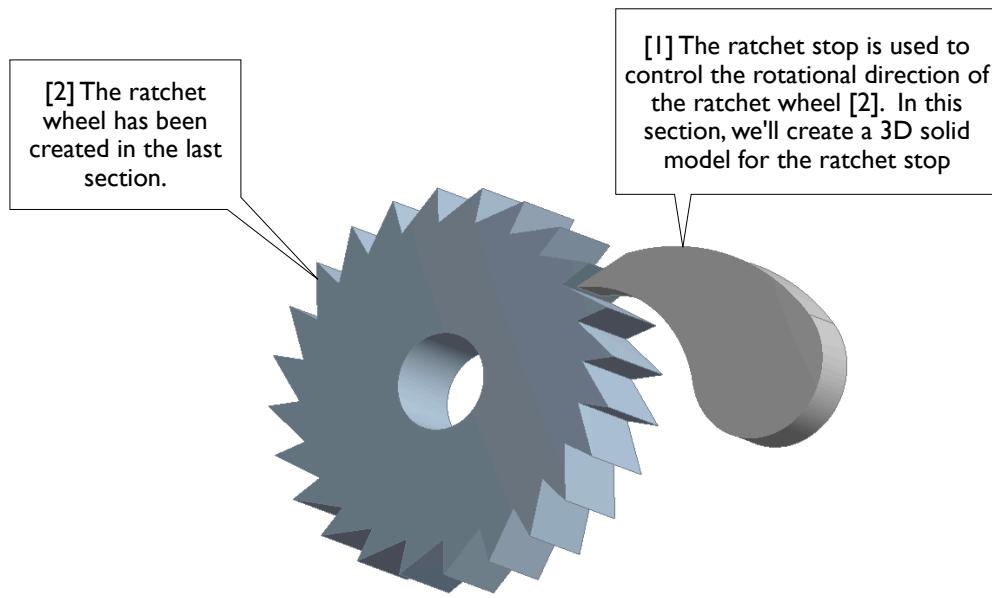


Section 1.3

Ratchet Stop



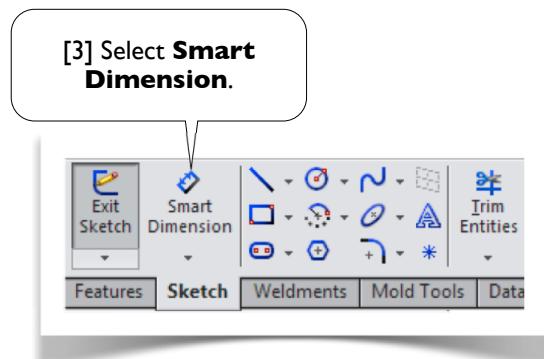
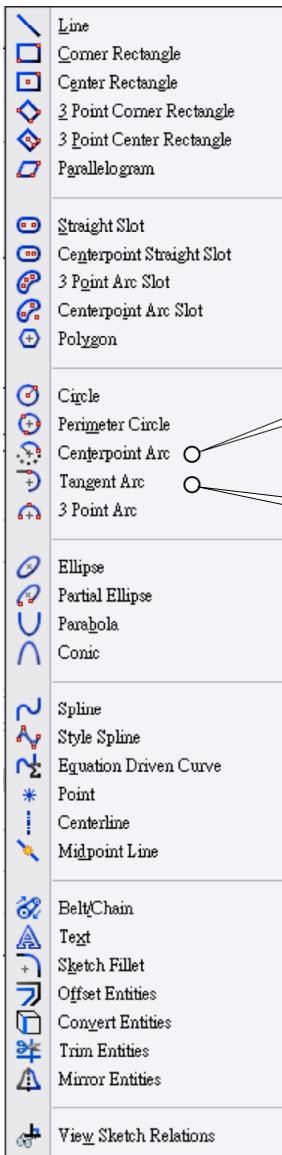
1.3-1 About the Ratchet Stop



1.3-2 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). #

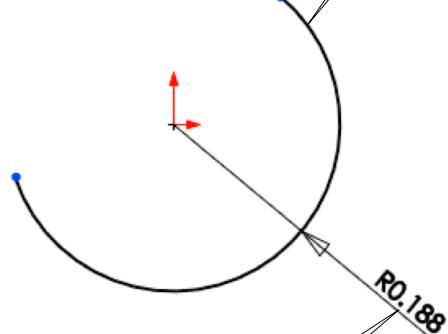
1.3-3 Draw the Sketch



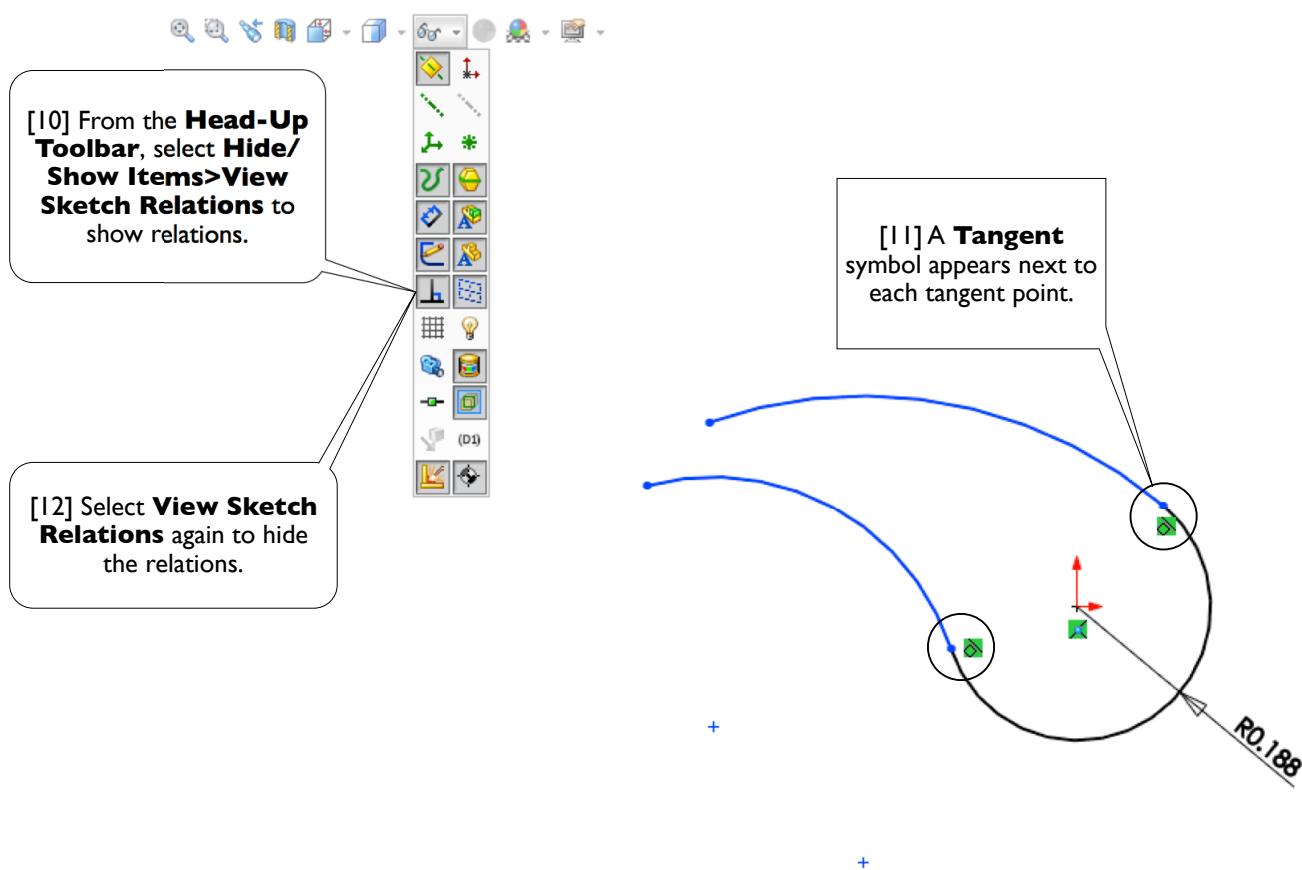
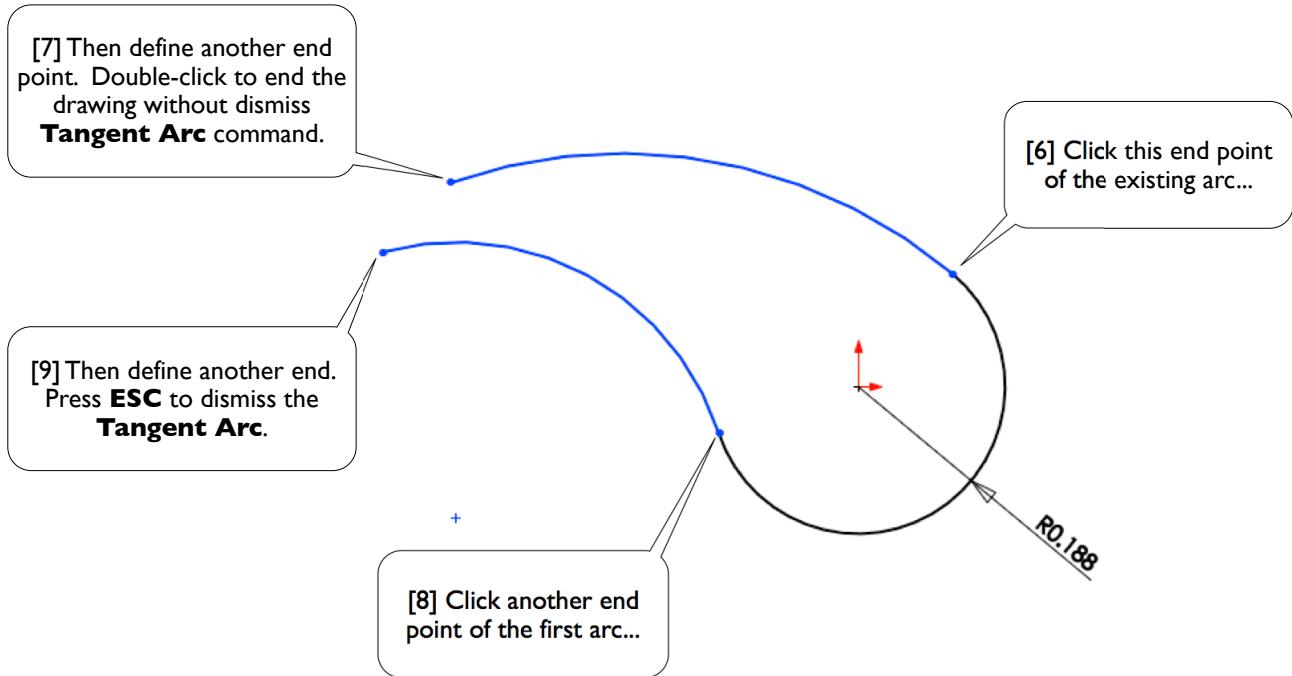
[1] Right-click the **Graphics Area** and select **Sketch Entities>Centerpoint Arc**.

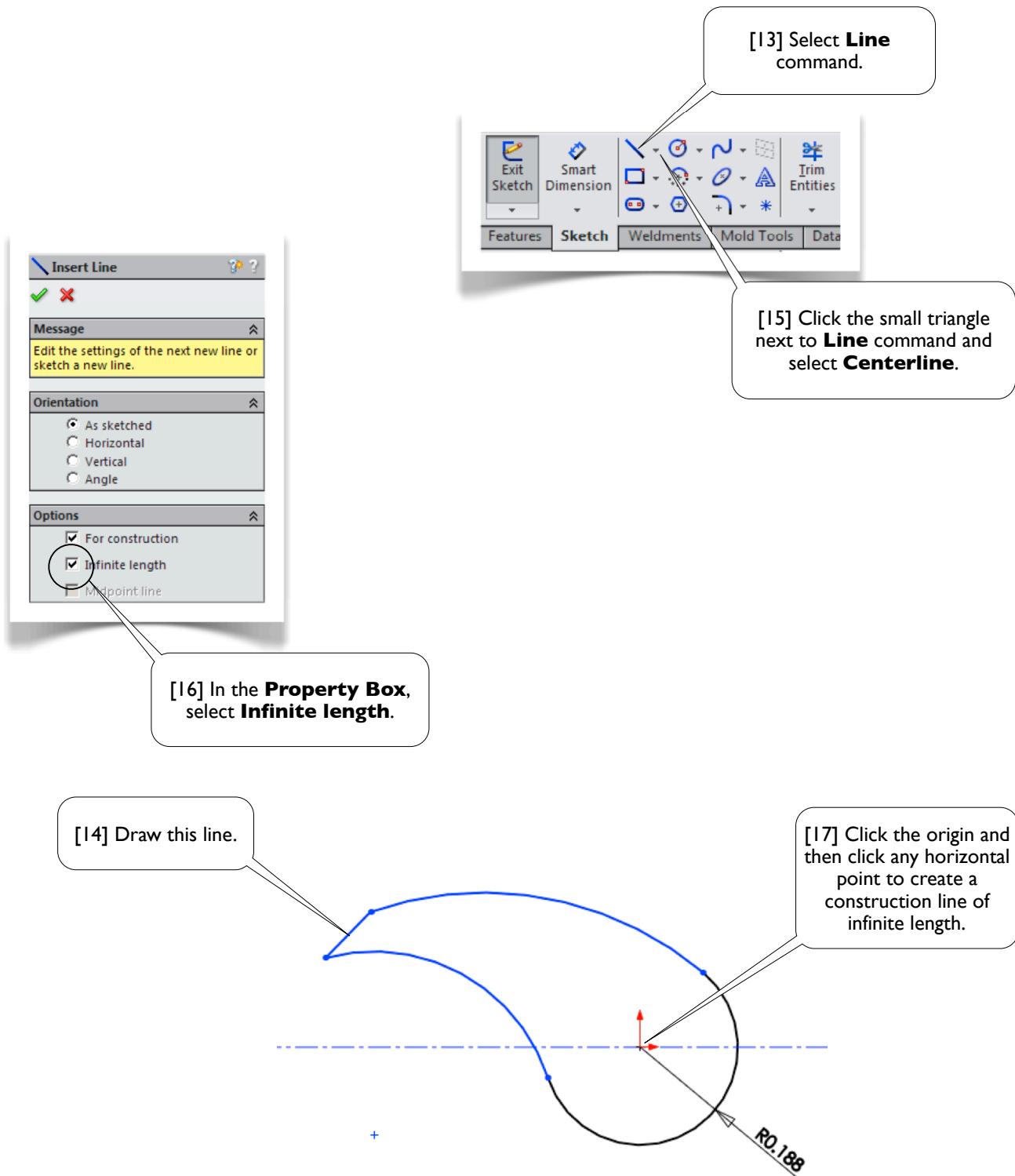
[5] Press **ESC** to dismiss **Smart Dimension**. Select **Sketch Entities>Tangent Arc** from the **Context Menu**.

[2] Create an arc like this. Click the origin first, then starting point, and finally the ending point.

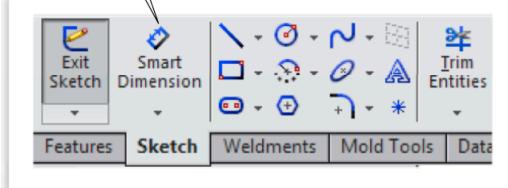


[4] Specify a radius of 0.188 in.

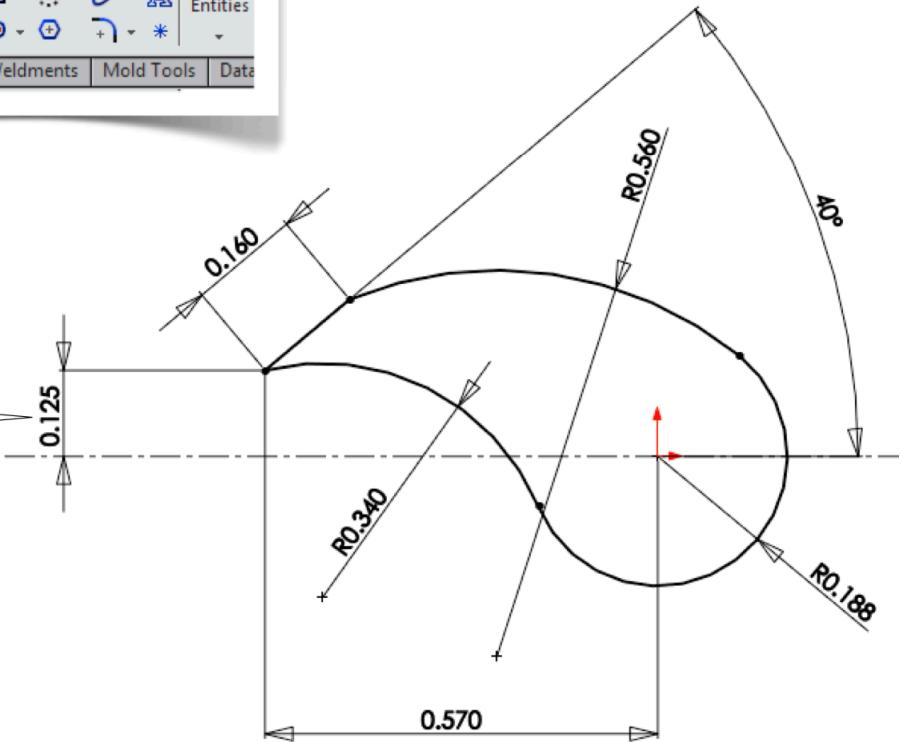




[18] Select **Smart Dimension**.



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



1.3-4 Generate 3D Model

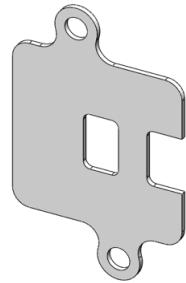
[1] **Extrude** the sketch (1.1-8[2], page 15) 0.125 inches to create this 3D model.



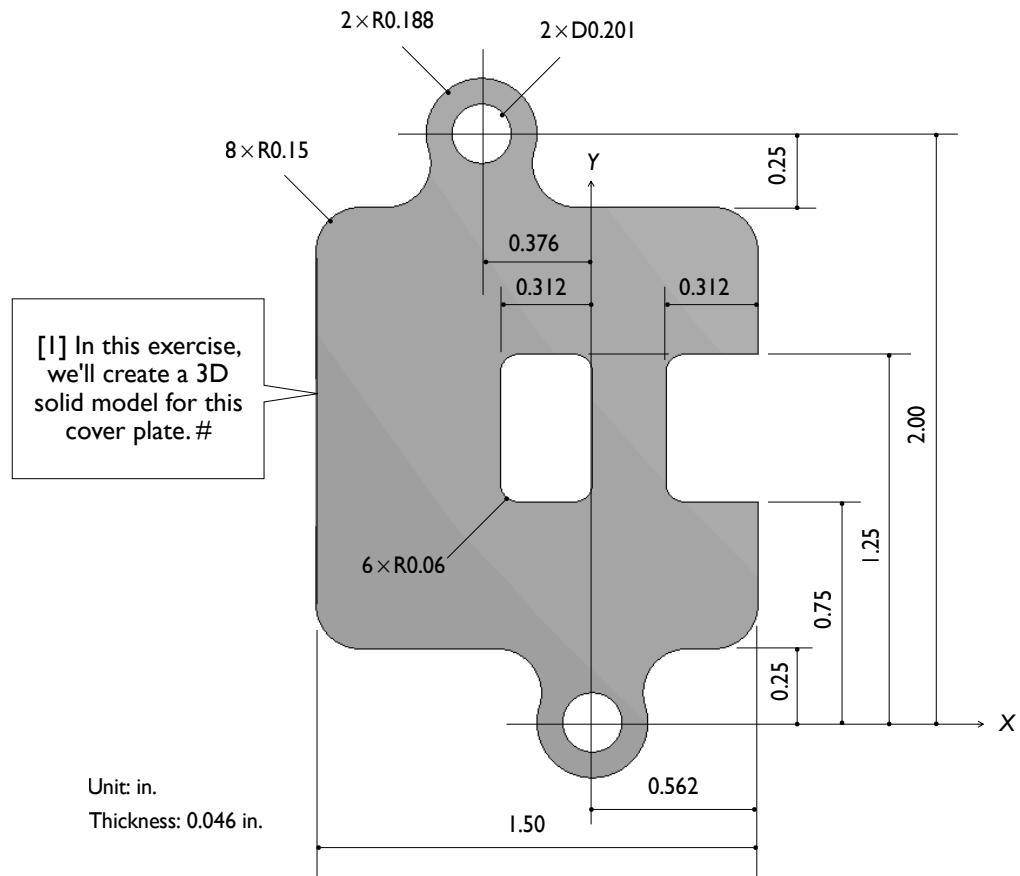
[2] Save the part with the name **Stop**. Close the file and exit **SOLIDWORKS**.#

Section 1.4

Cover Plate



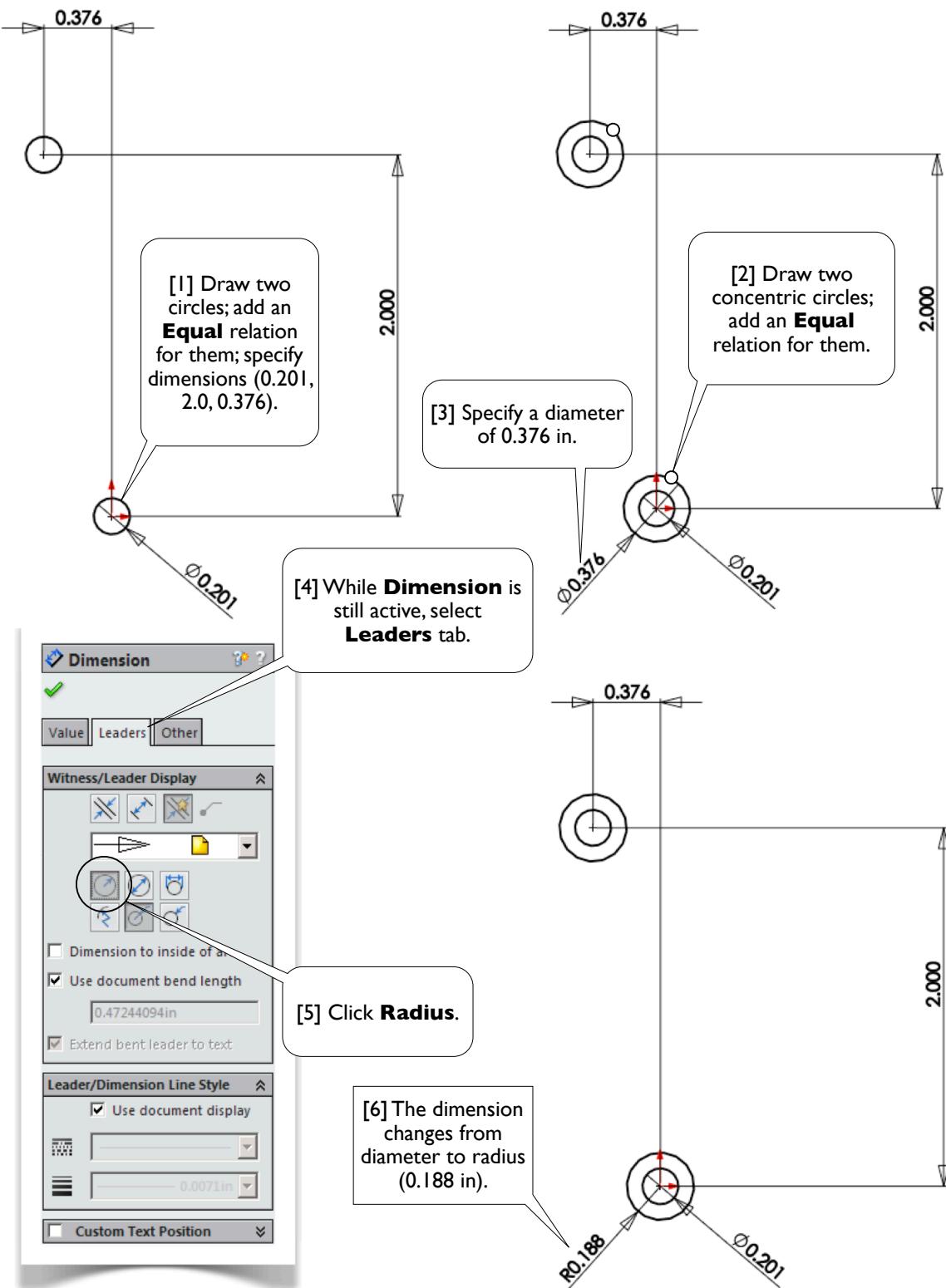
1.4-1 About the Cover Plate

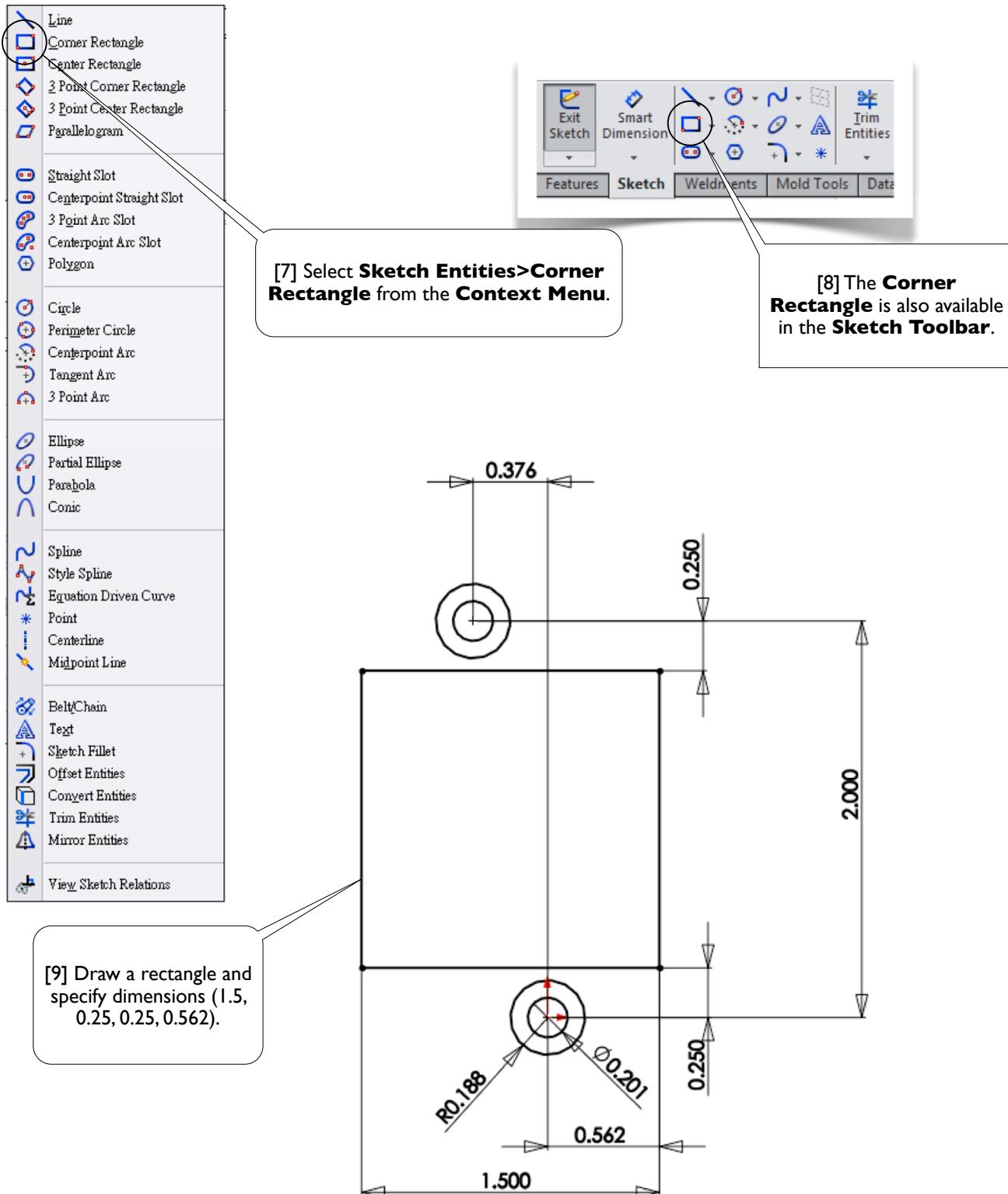


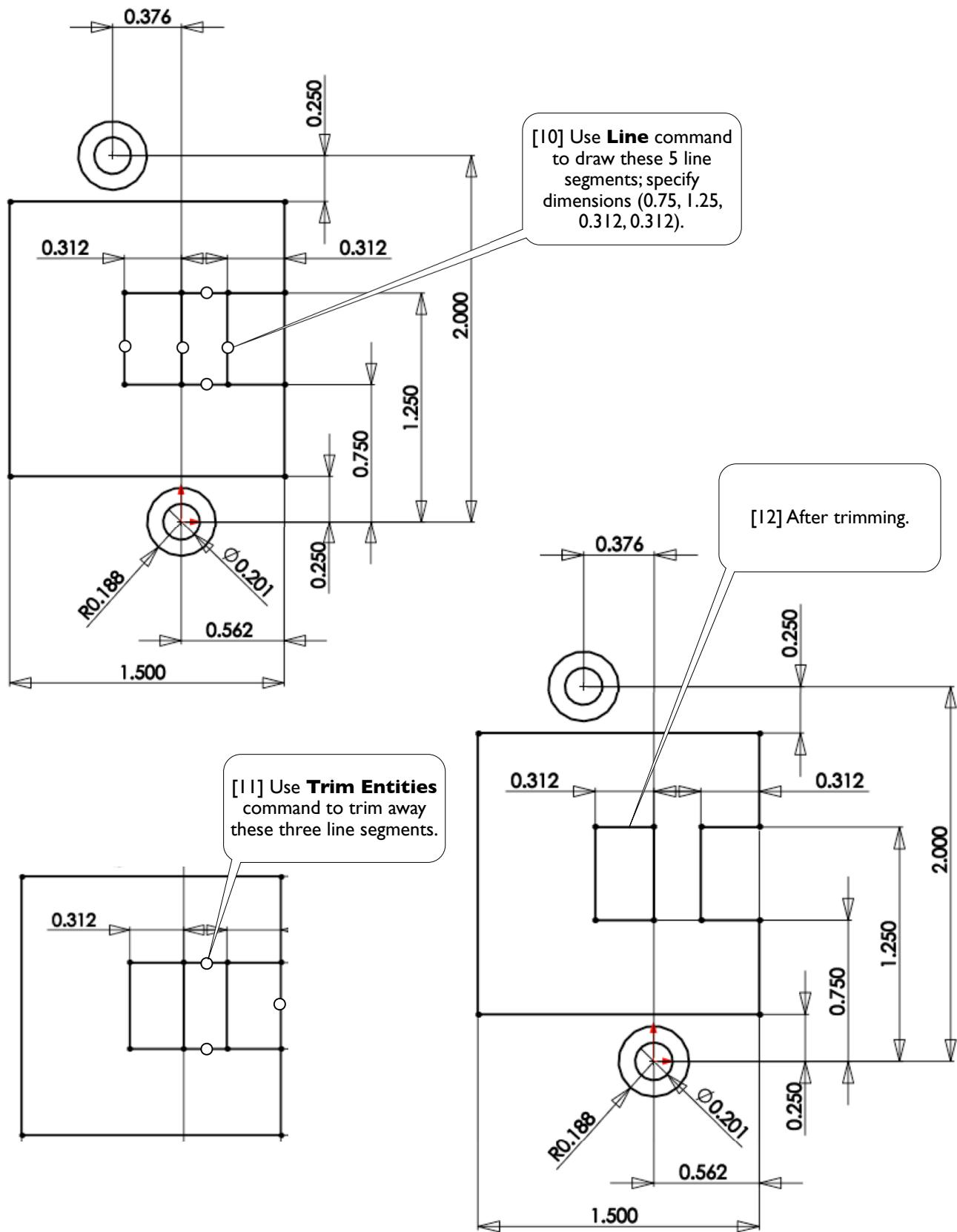
1.4-2 Start Up

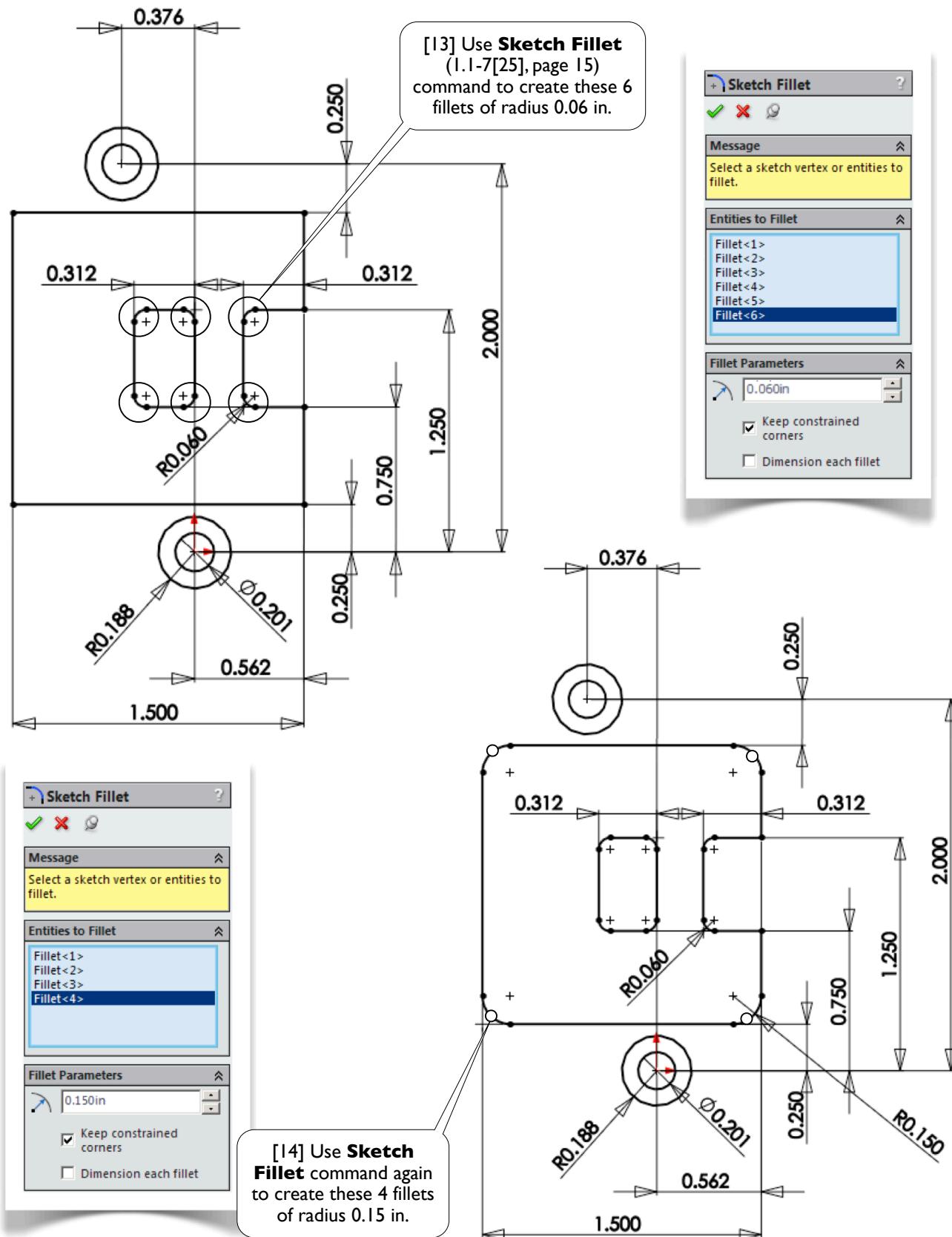
[!] Launch **SOLIDWORKS** and create a new part (1.1-2, page 4). Set up **IPS** unit system with 3 decimal places for the length unit (1.1-3, page 5). Create a sketch on **Front** plane (1.1-4[1, 2], page 6). #

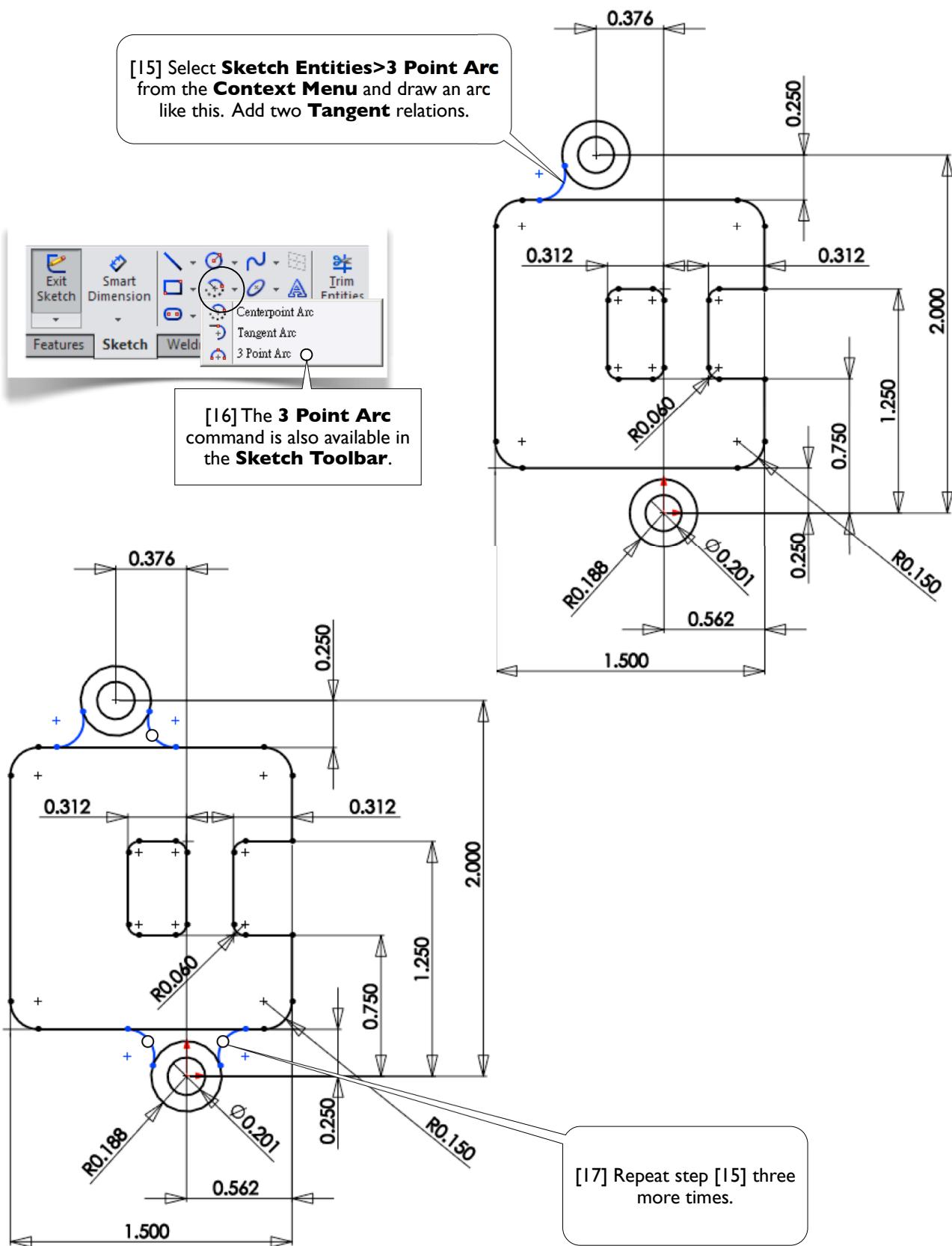
1.4-3 Draw the Sketch

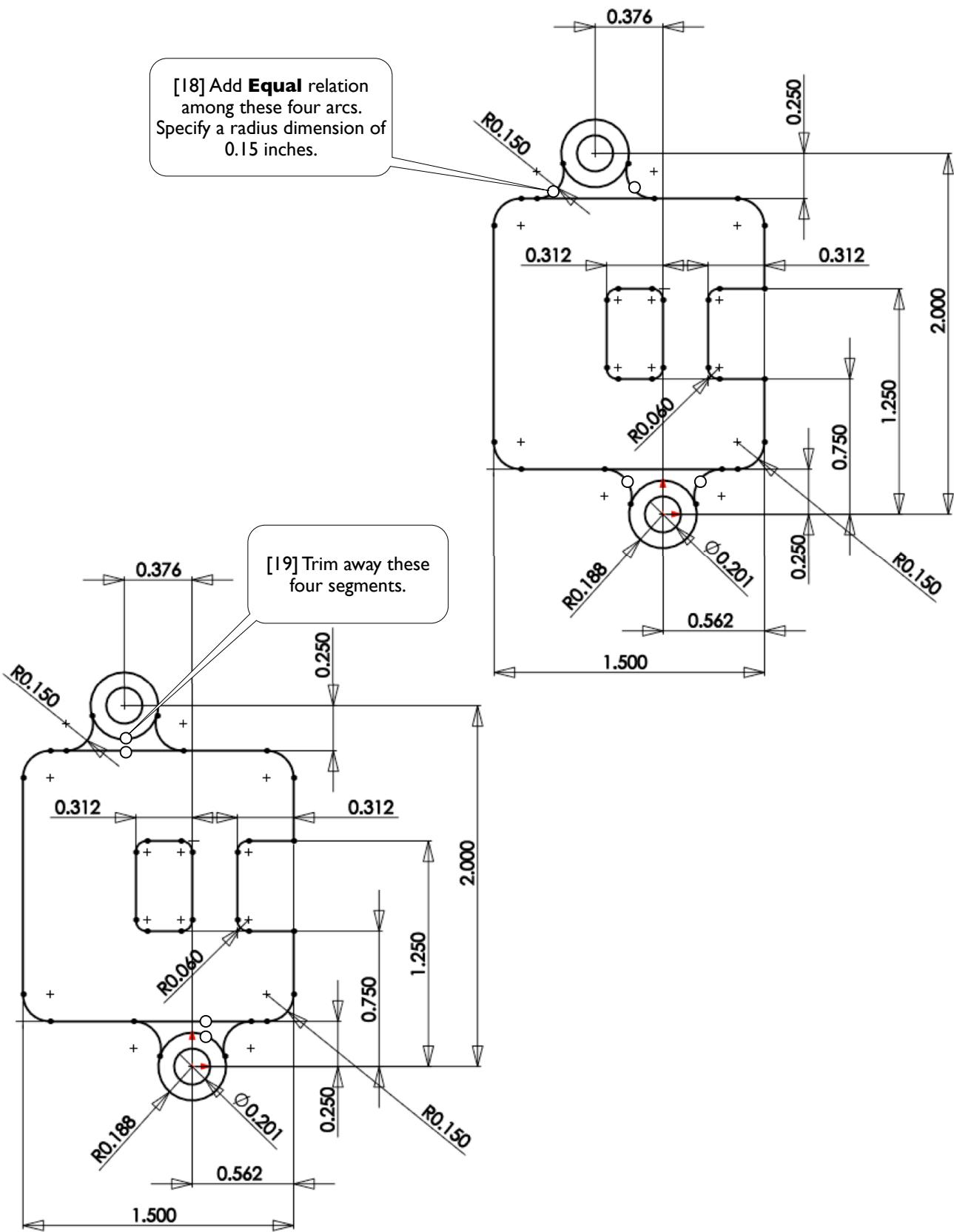


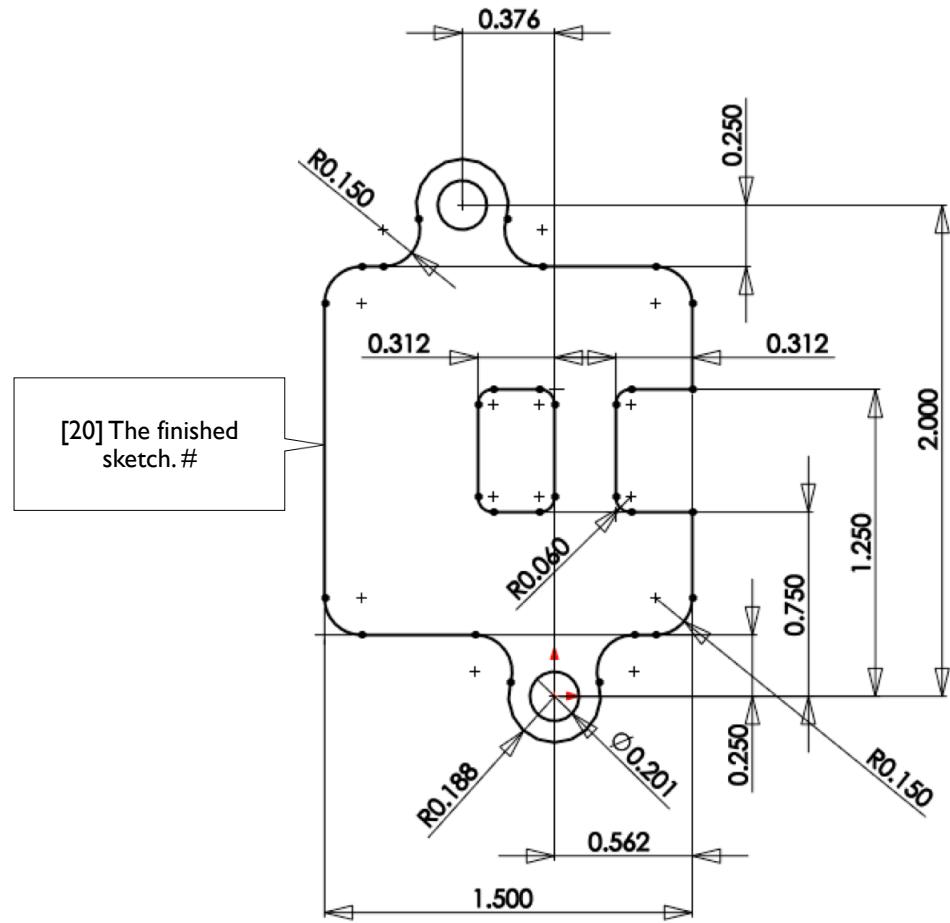




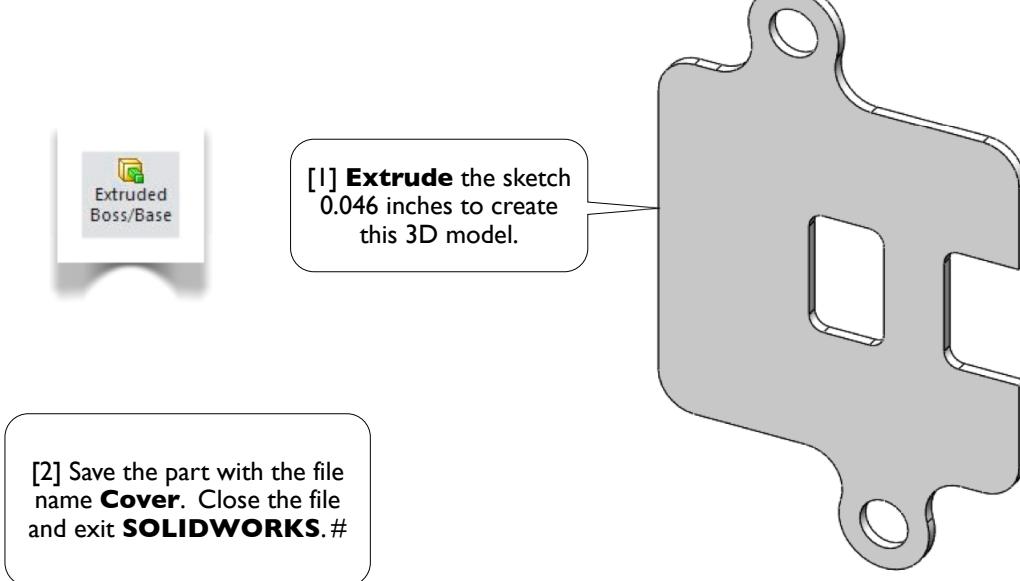






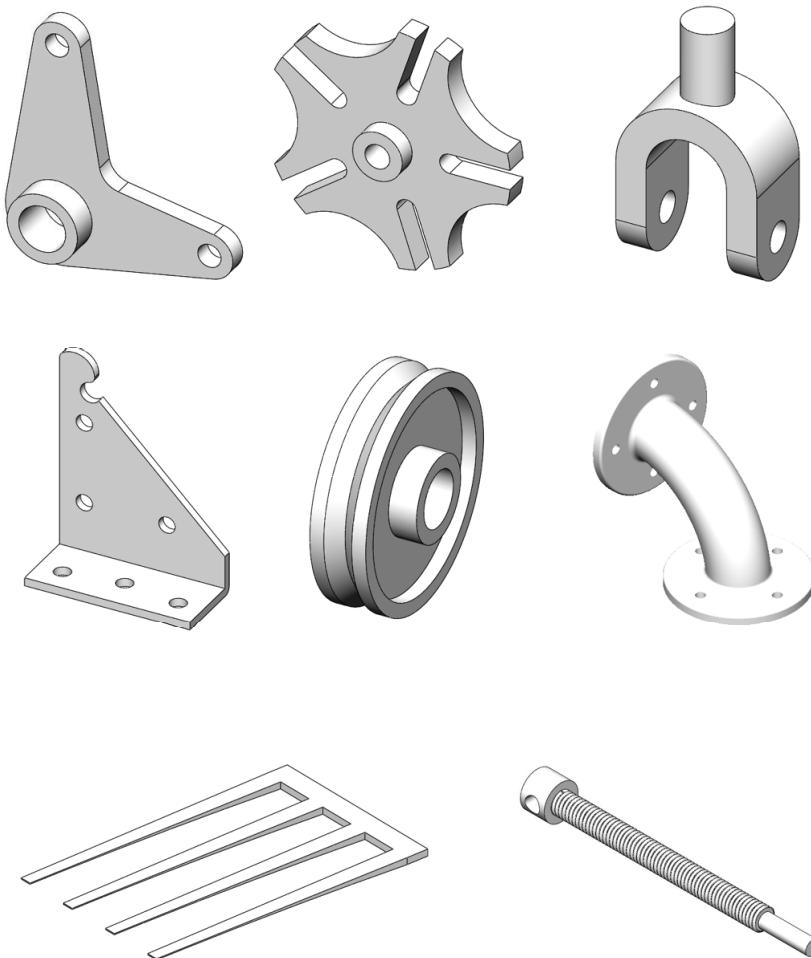


1.4-4 Generate 3D Model



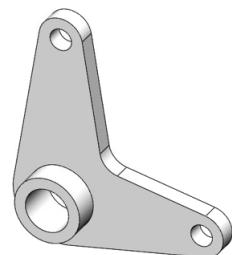
Chapter 2

Part Modeling



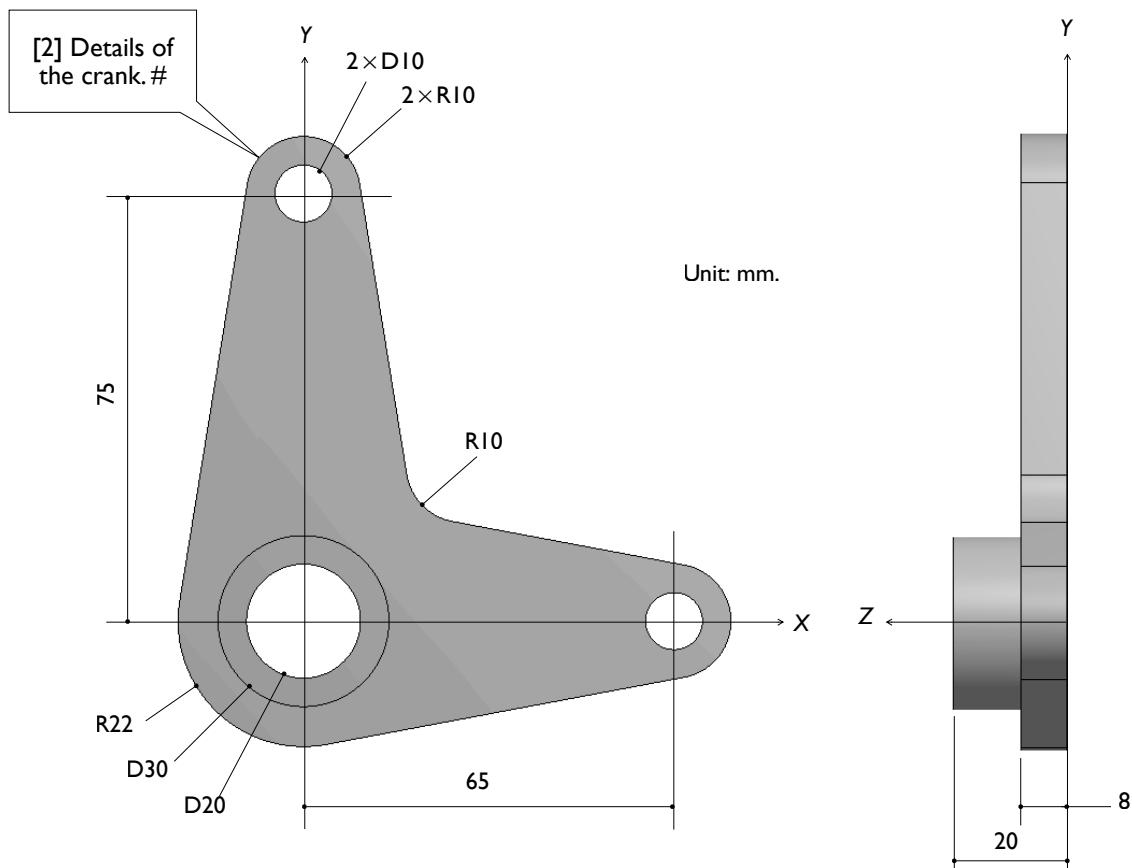
Section 2.I

Crank



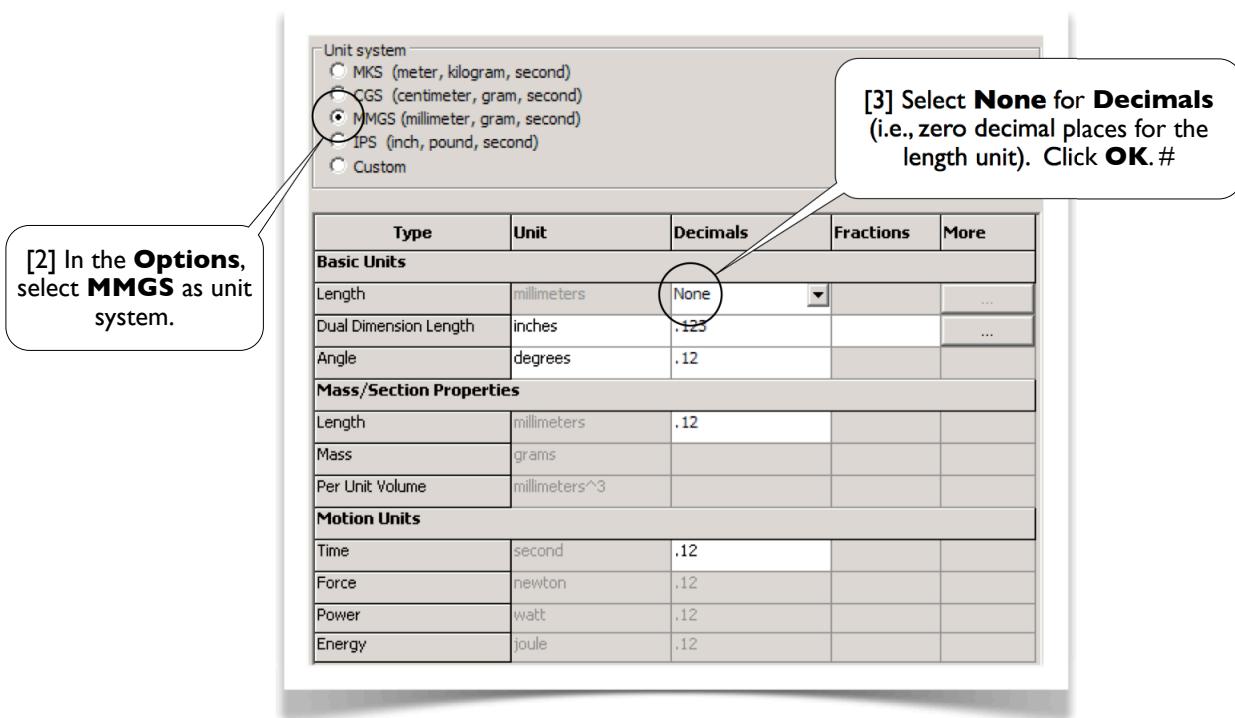
2.I-1 About the Crank

[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.



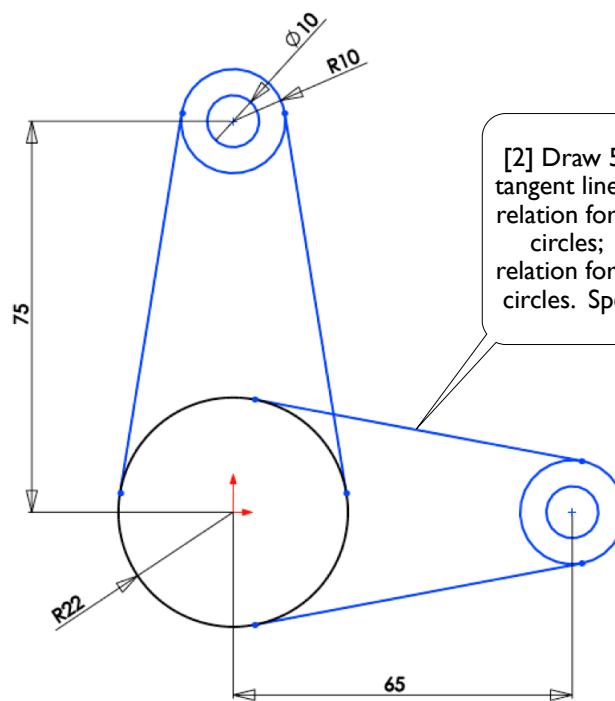
2.1-2 Start Up

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

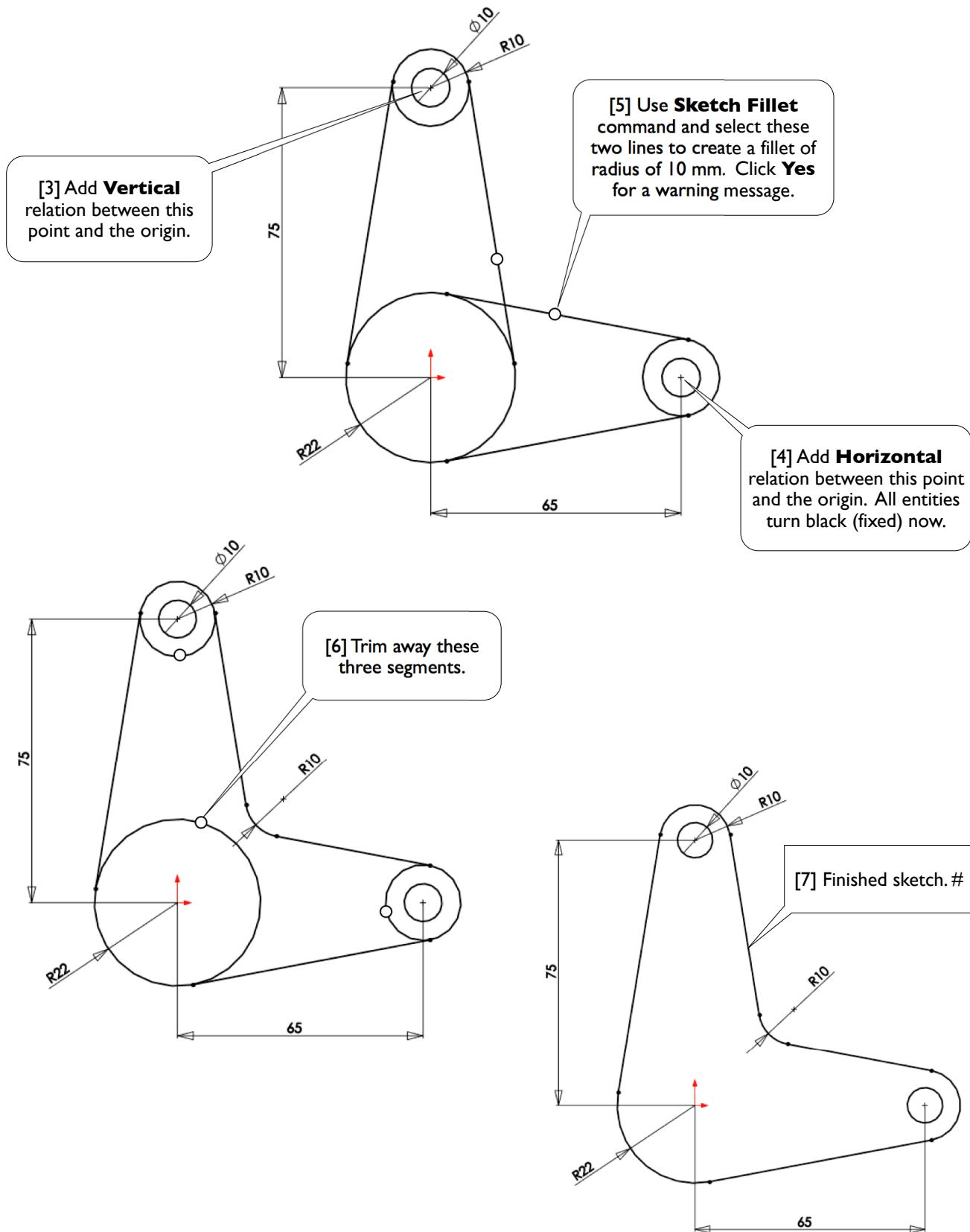


2.1-3 Draw a Sketch for the Base Body

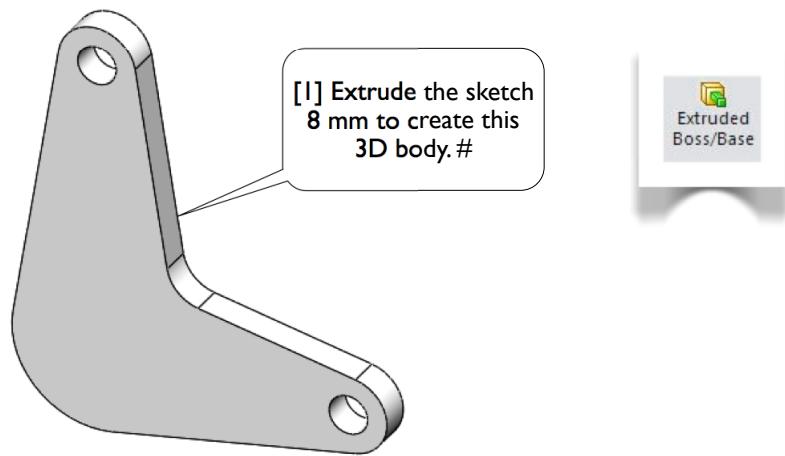
[1] Create a sketch on **Front** plane.



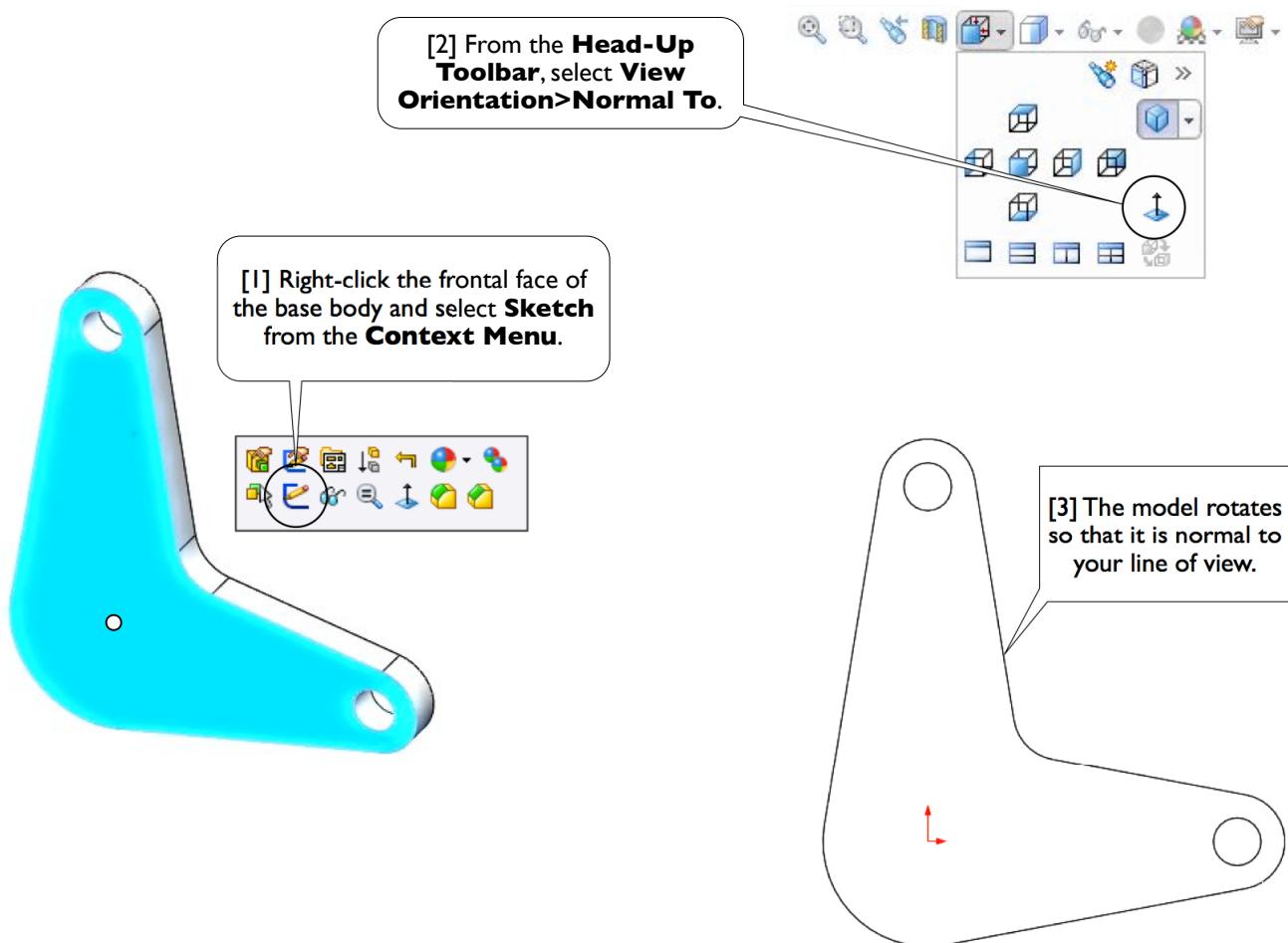
[2] Draw 5 circles and four tangent lines. Add an **Equal** relation for the two smallest circles; add an **Equal** relation for the two medium circles. Specify dimensions.

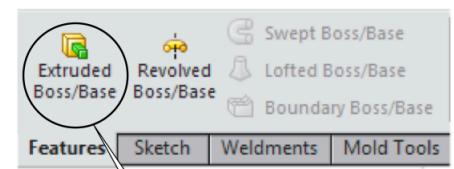
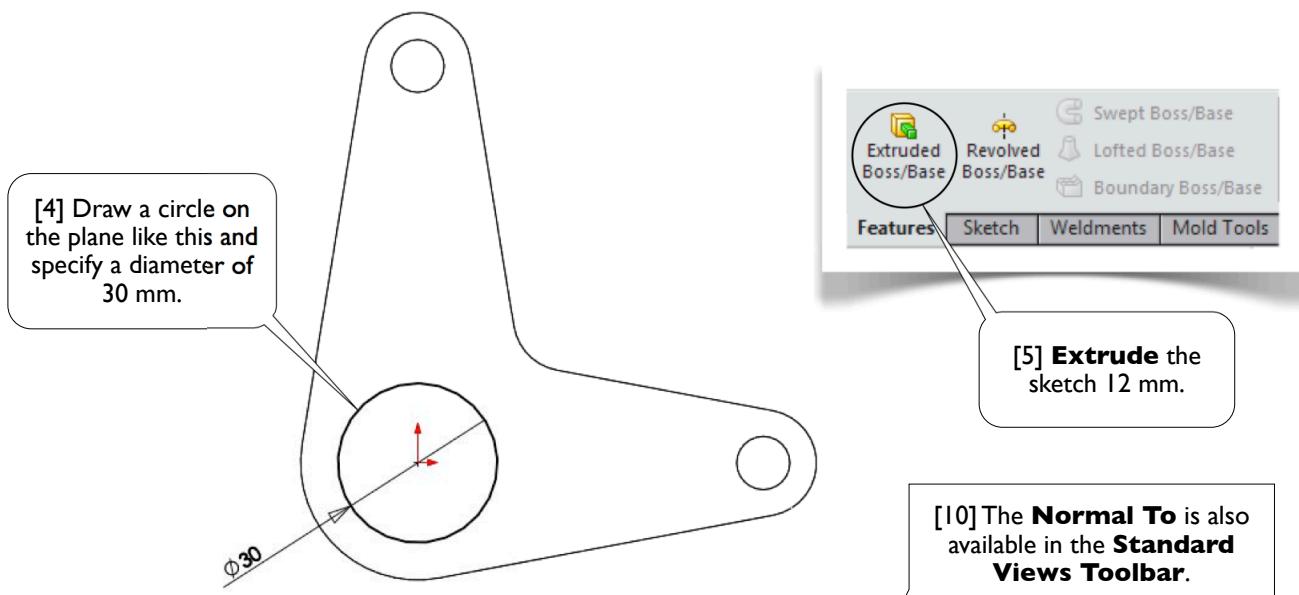


2.1-4 Extrude the Sketch to Create the Base Body



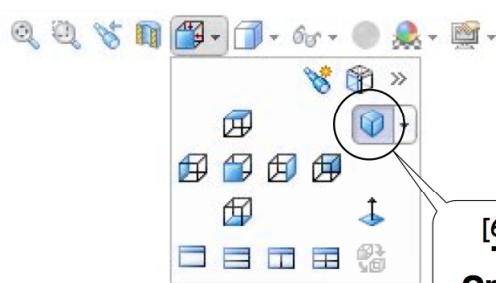
2.1-5 Add Features to the Base Body





[5] Extrude the sketch 12 mm.

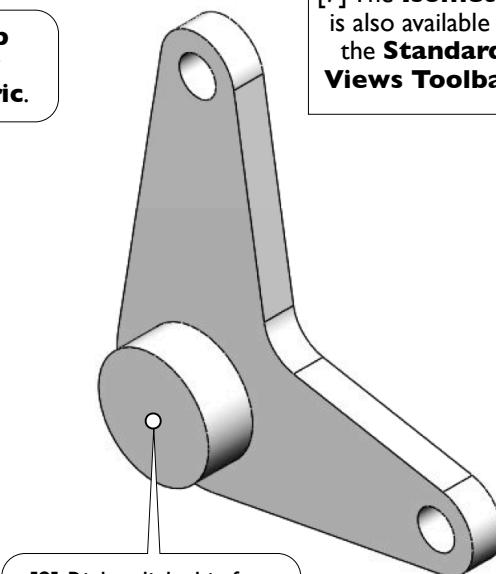
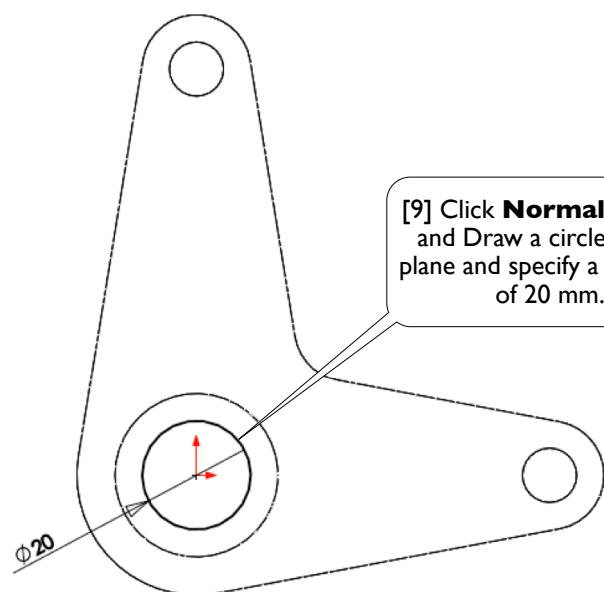
[10] The **Normal To** is also available in the **Standard Views Toolbar**.

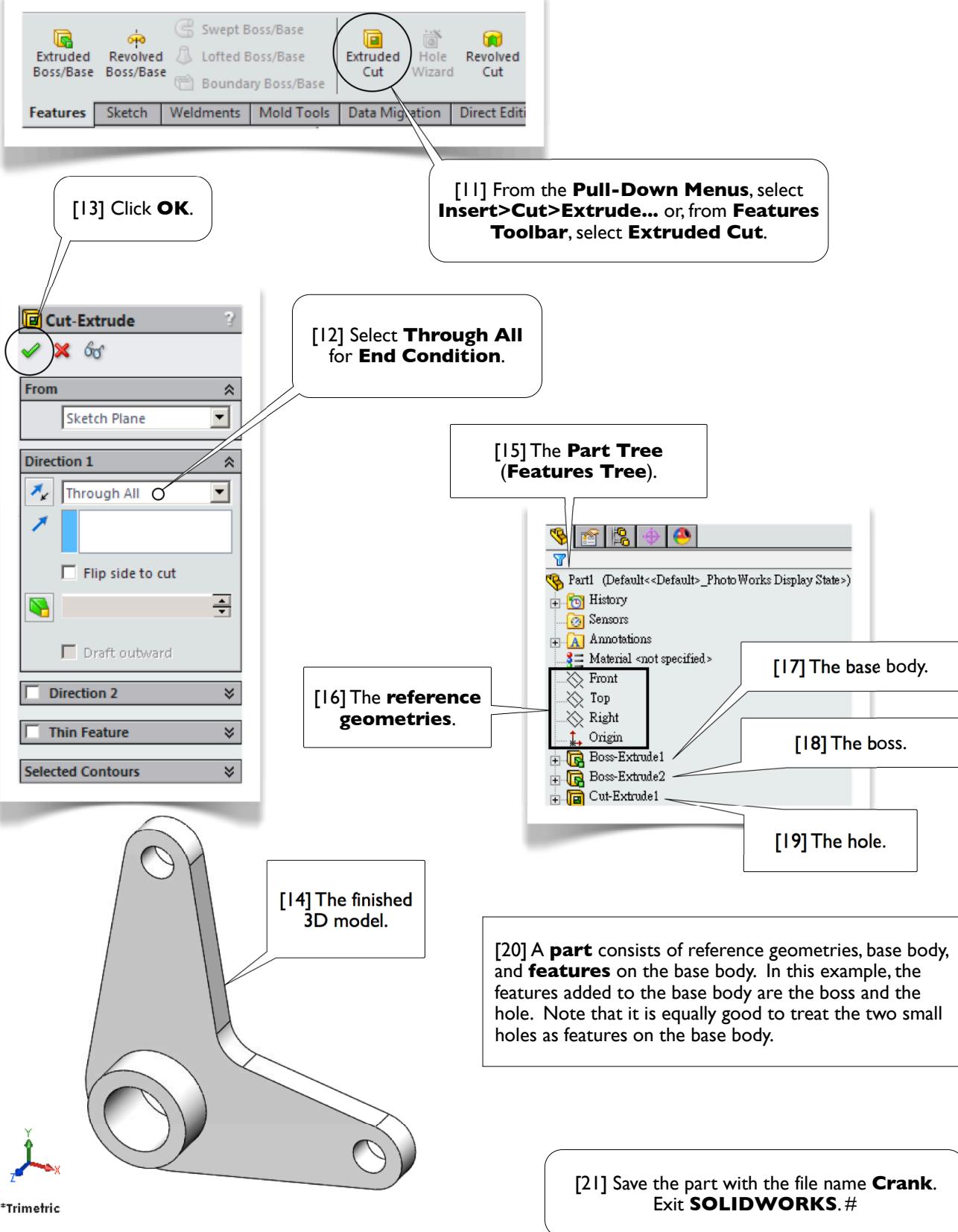


[6] From the **Head-Up Toolbar**, select **View Orientation>Isometric**.



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





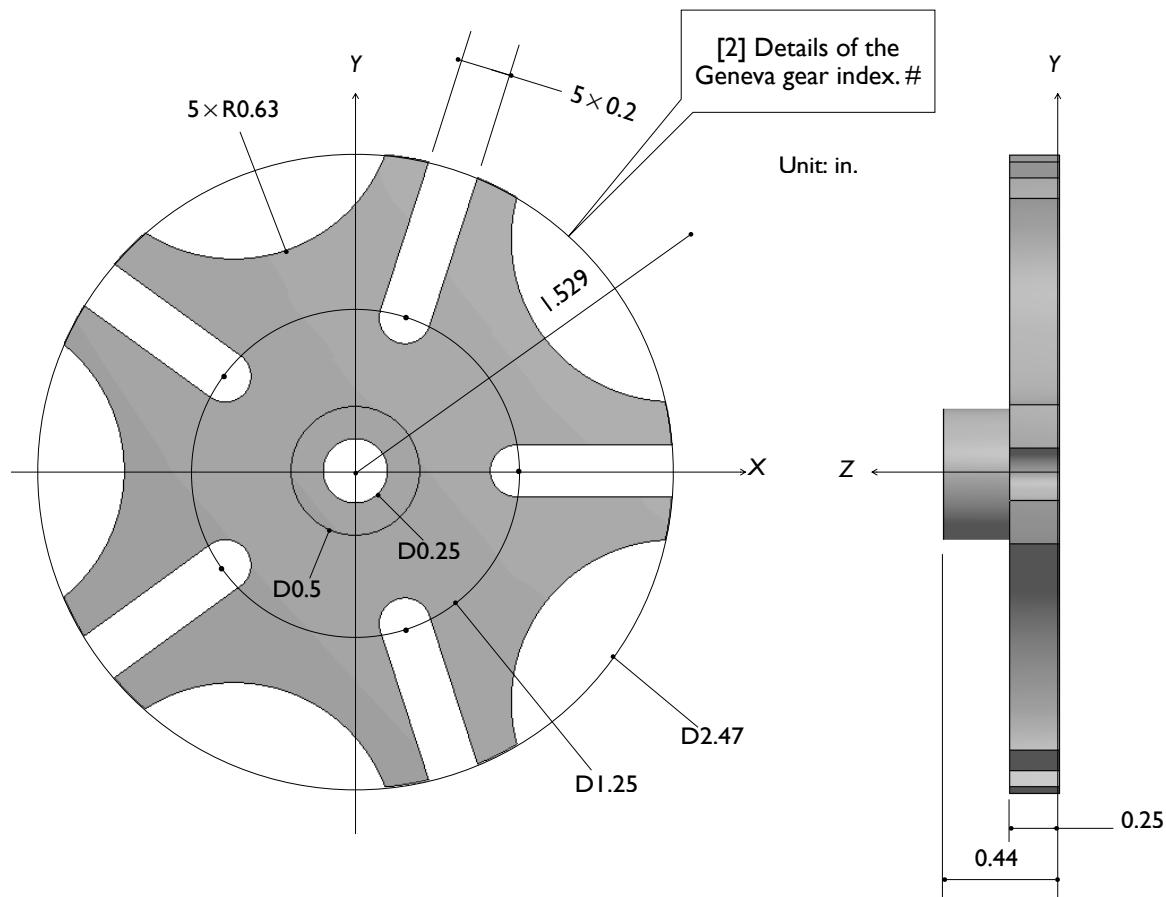
Section 2.2

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].

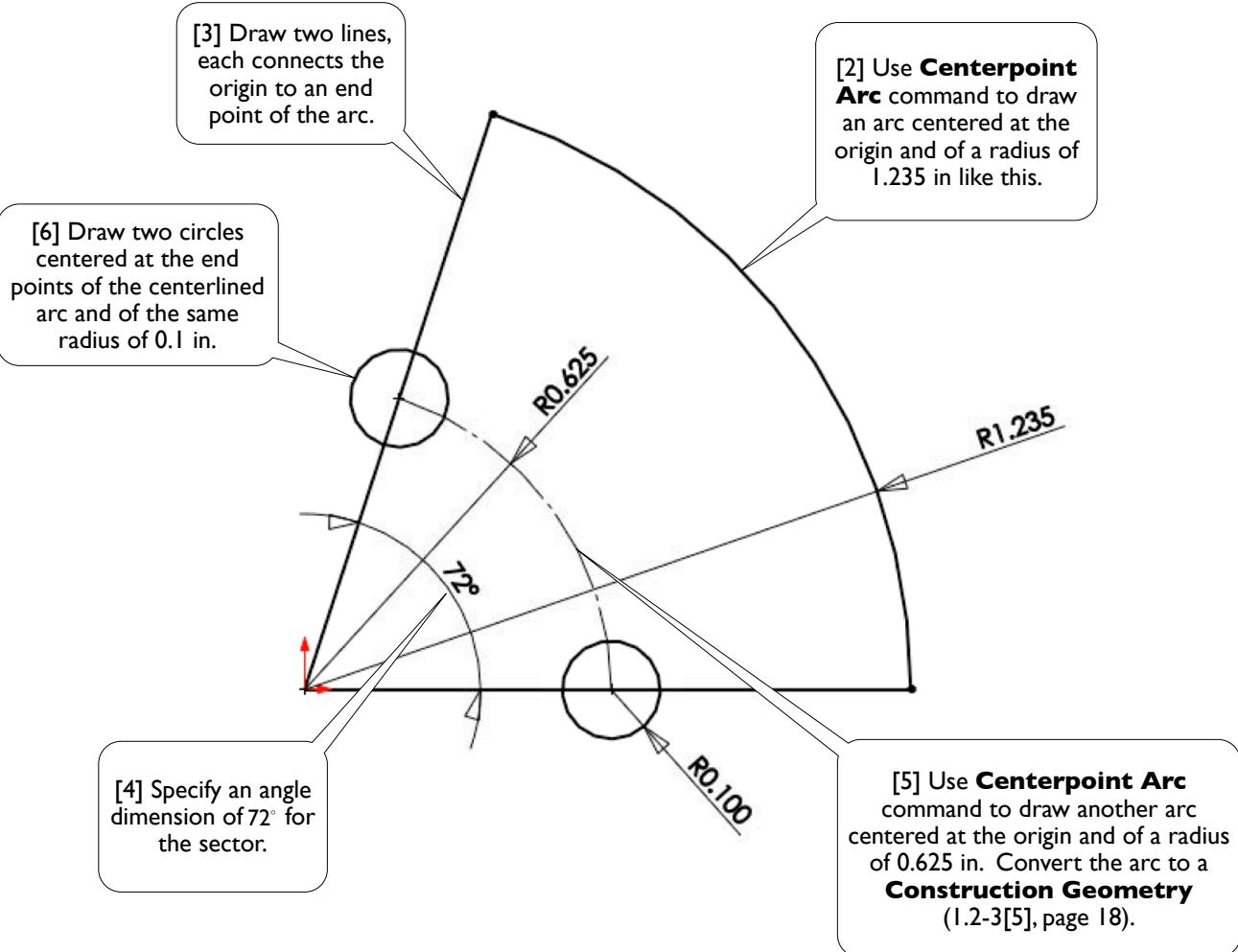


2.2-2 Start Up

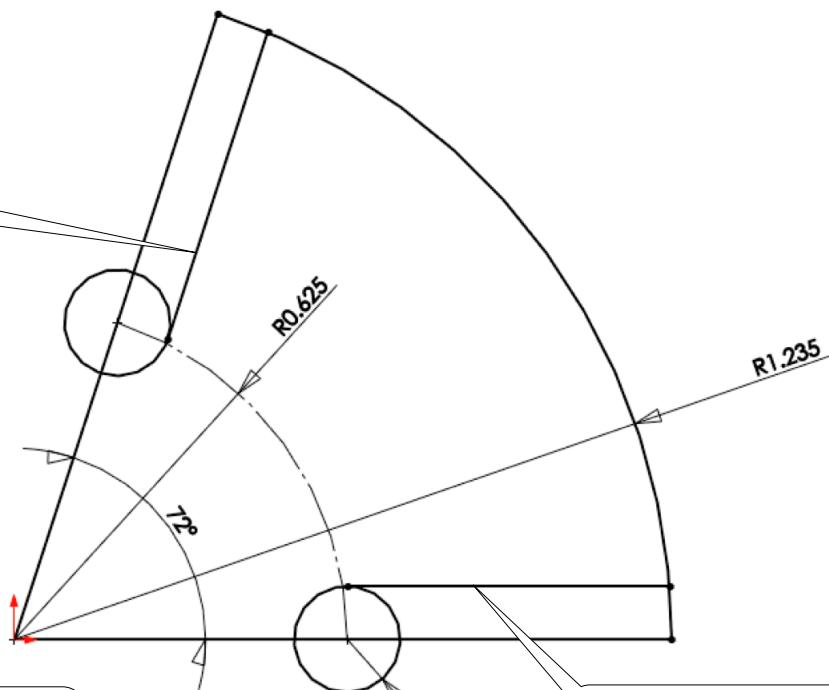
[!] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.#

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

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

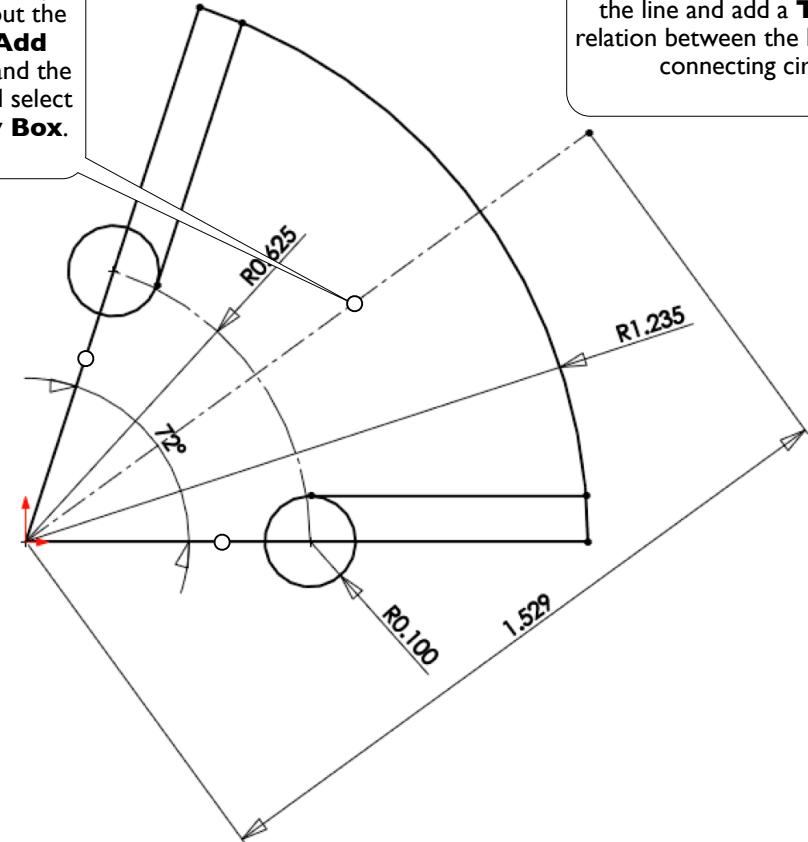


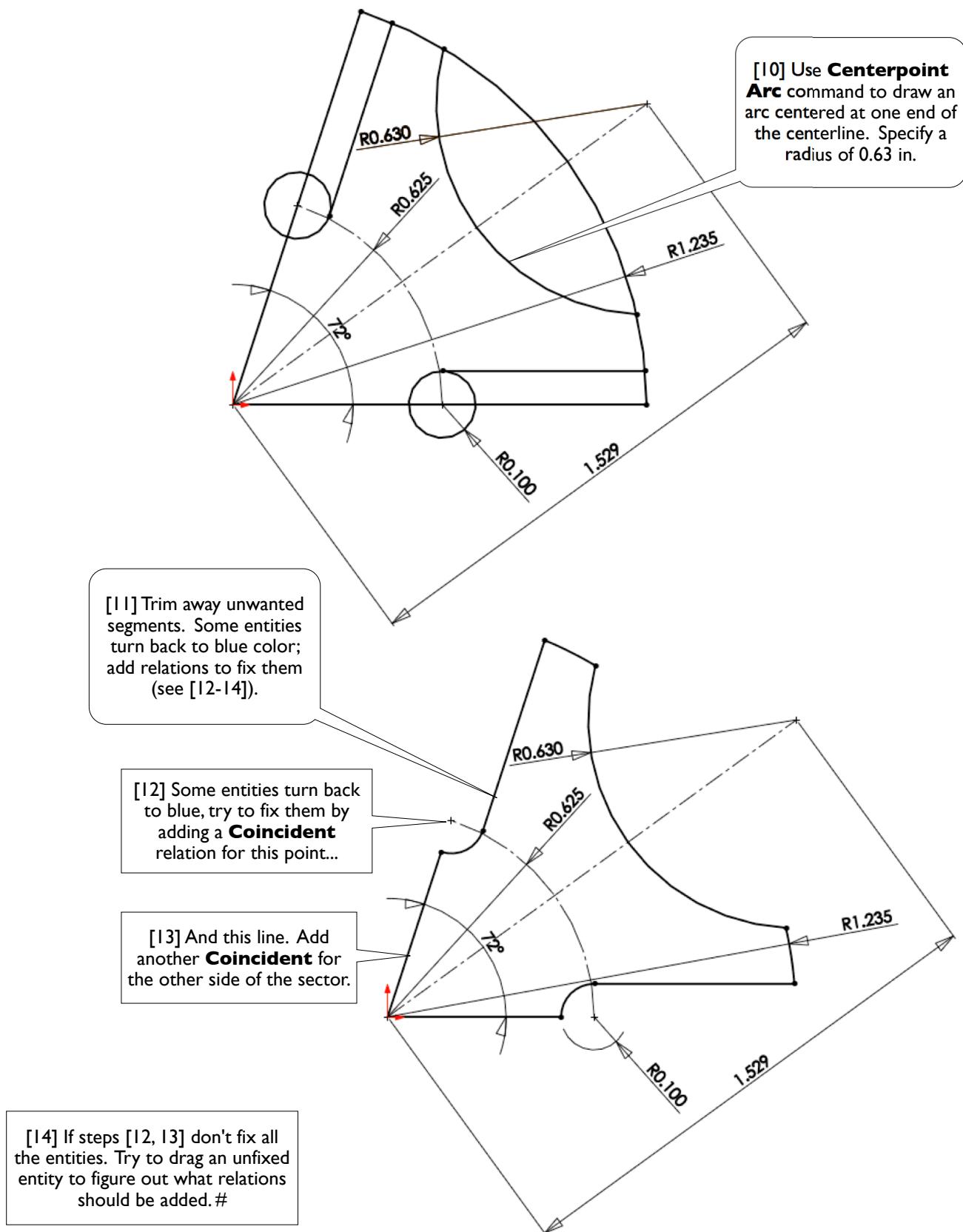
[8] Draw a line connecting the upper circle to the outer arc. Add a **Parallel** relation between the line and the line next to it. Add a **Tangent** relation between the line and the connecting circle.



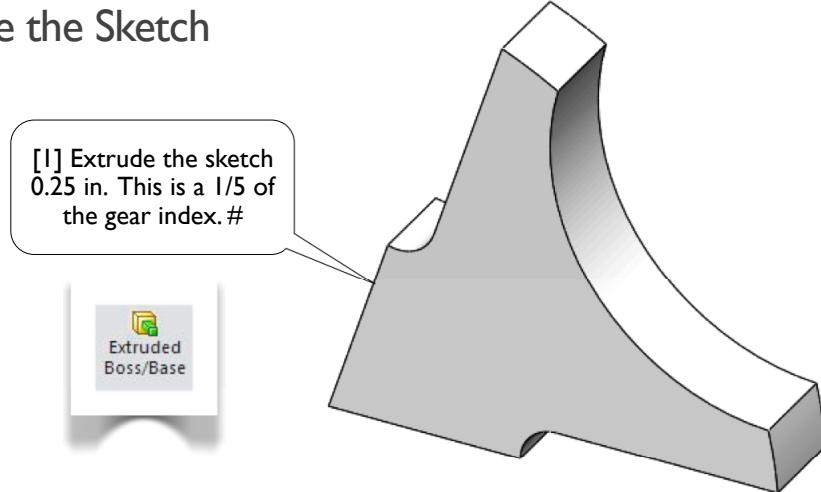
[9] Use **Centerline** command to draw a centerline starting from the origin. Specify the length (1.529 in). Make the sector symmetric about the centerline. To do this, select **Add Relation**, click the centerline and the two edge lines of the sector, and select **Symmetric** in the **Property Box**.

[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.

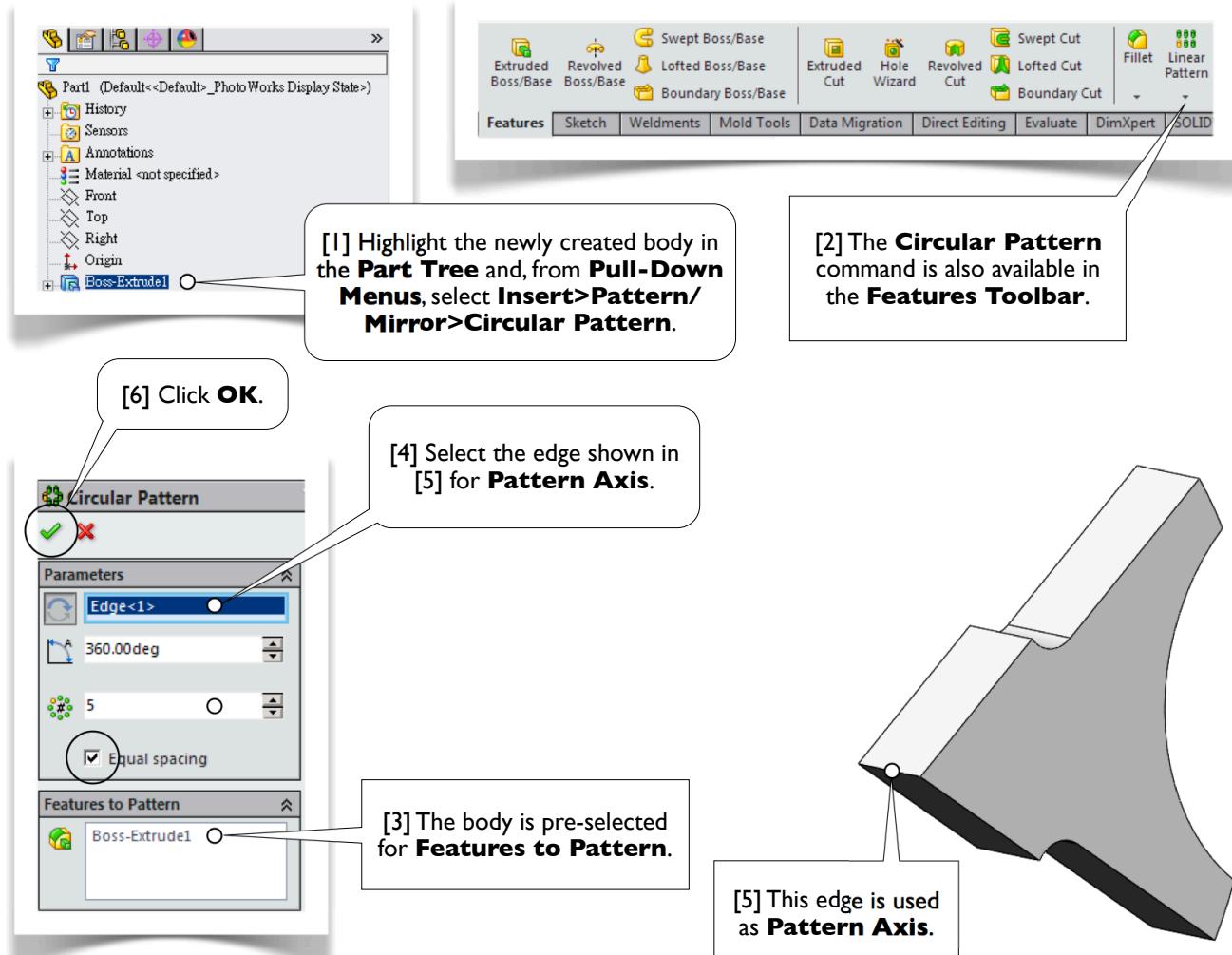


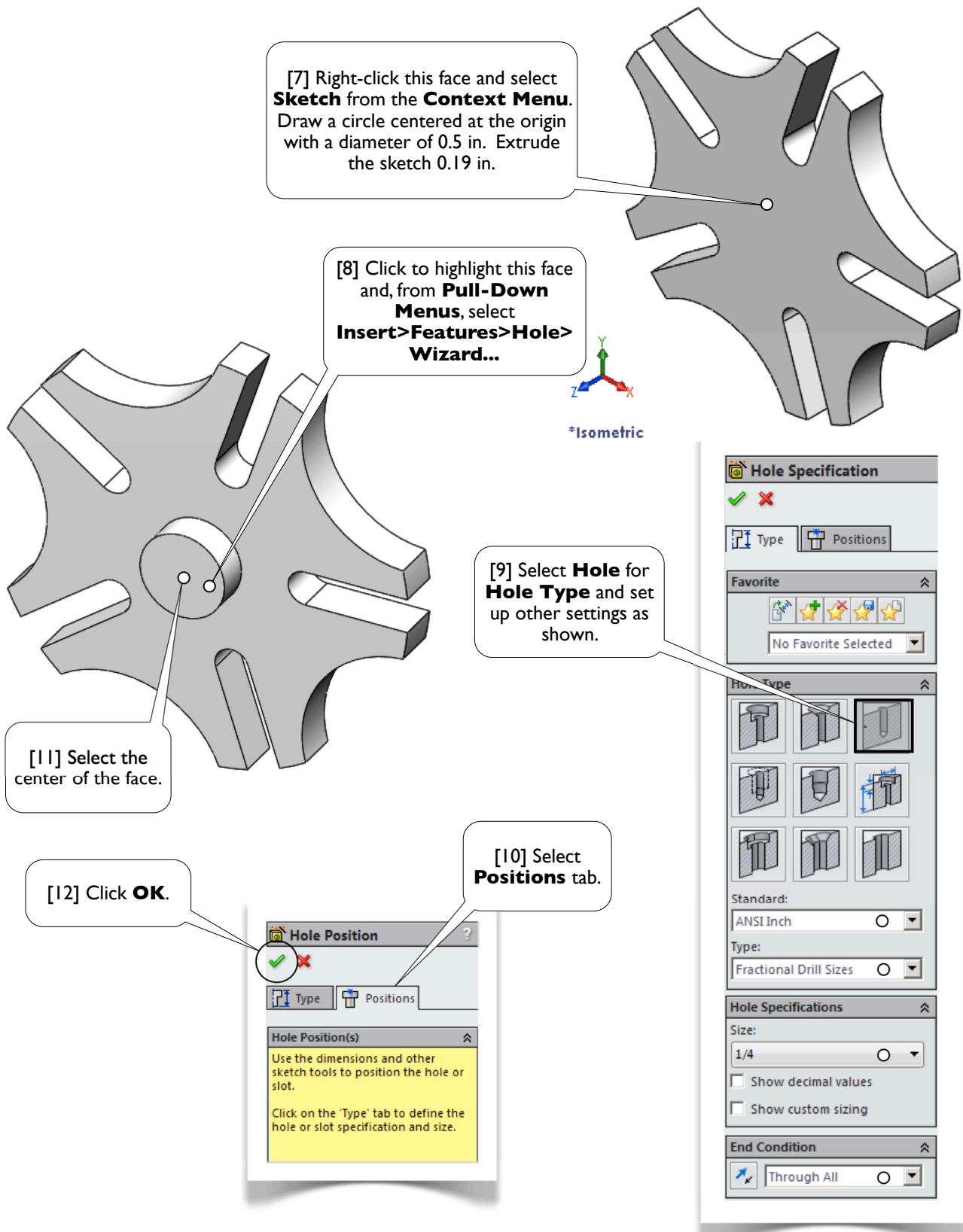


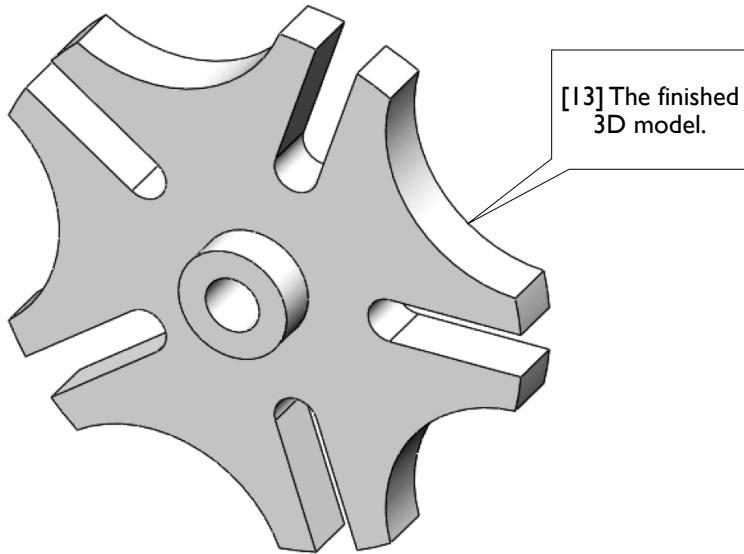
2.2-4 Extrude the Sketch



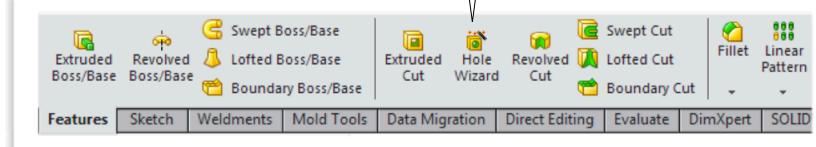
2.2-5 Complete the Full Model







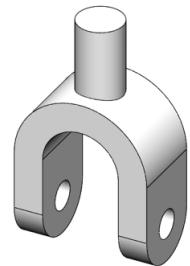
[14] The **Hole Wizard** command is also available in **Features Toolbar**.



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

Section 2.3

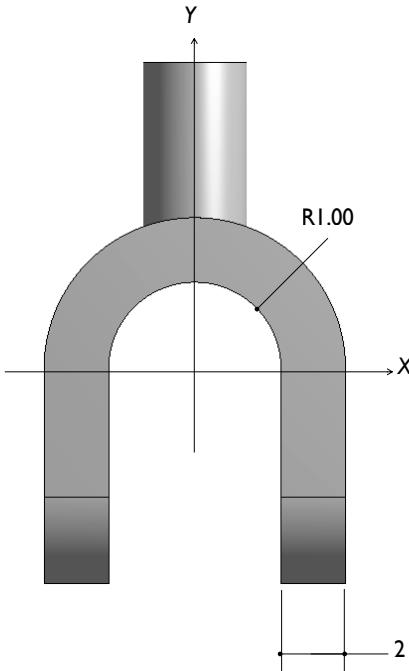
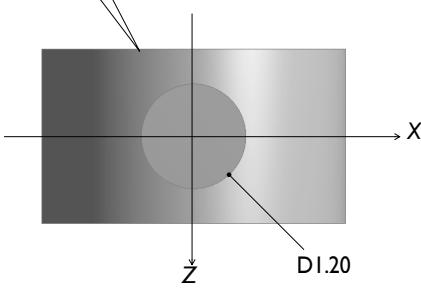
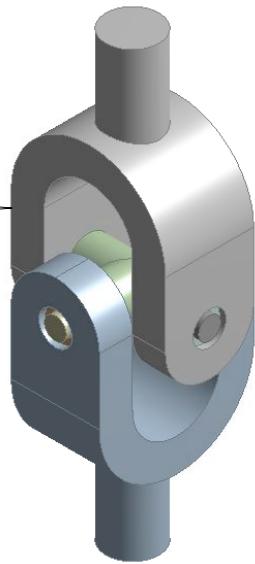
Yoke



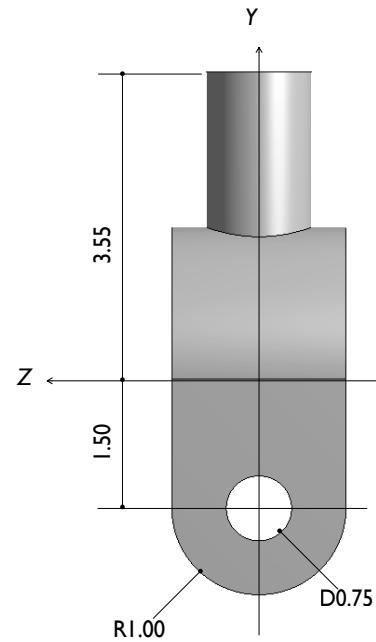
2.3-1 About the Yoke

[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.

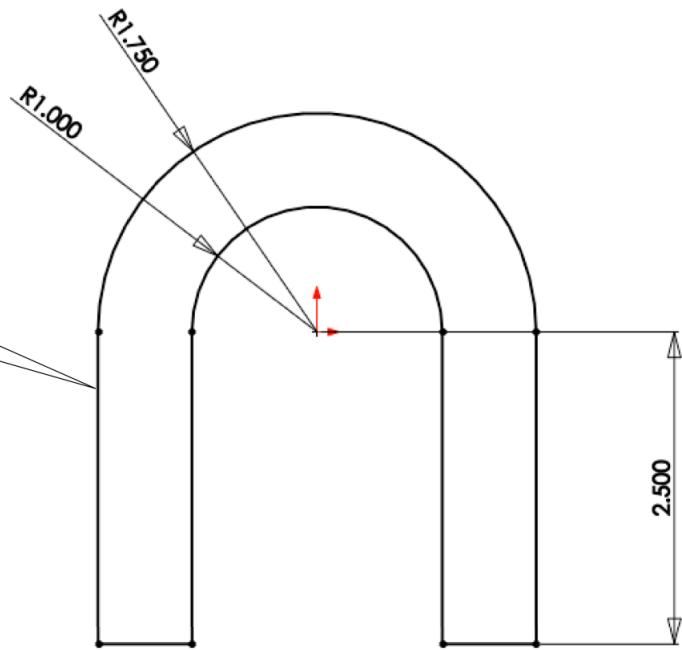


2.3-2 Start Up

[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

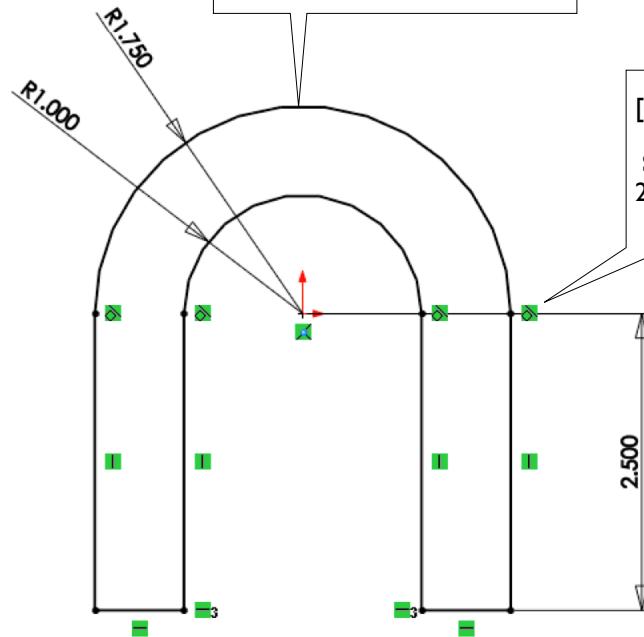
[1] Create a sketch on **Front** plane.

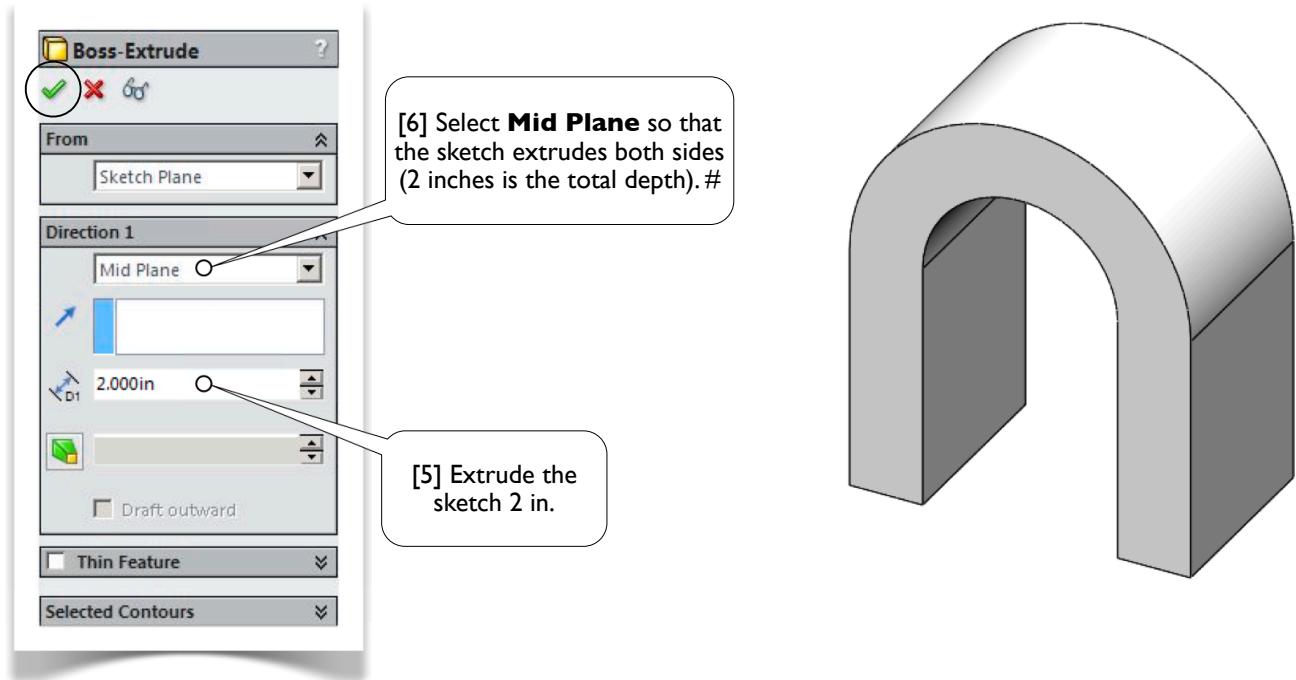


[2] Draw a sketch like this. If there are any blue entities (not well-defined), see [3, 4].

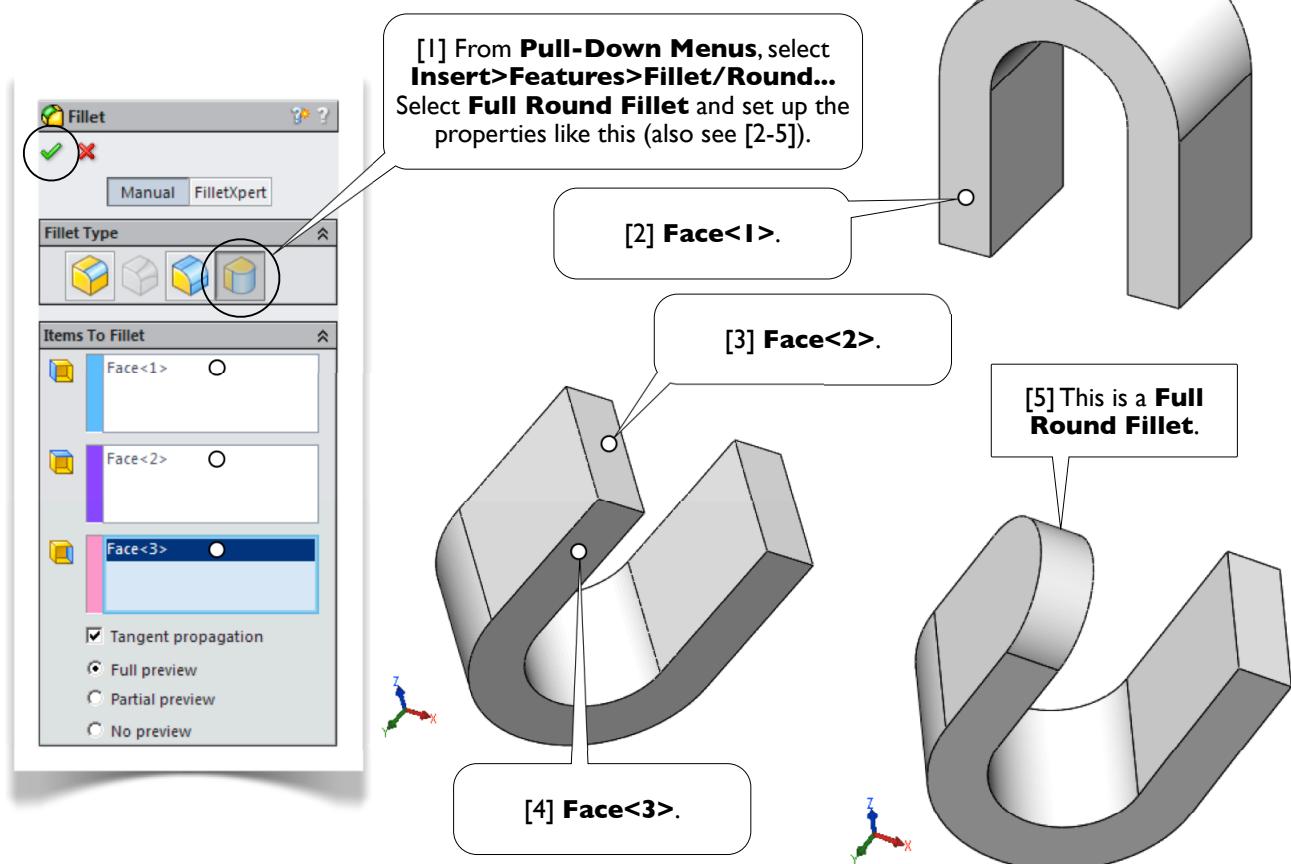
[4] Another way is to drag an unfixed entity to figure out what relations should be added.

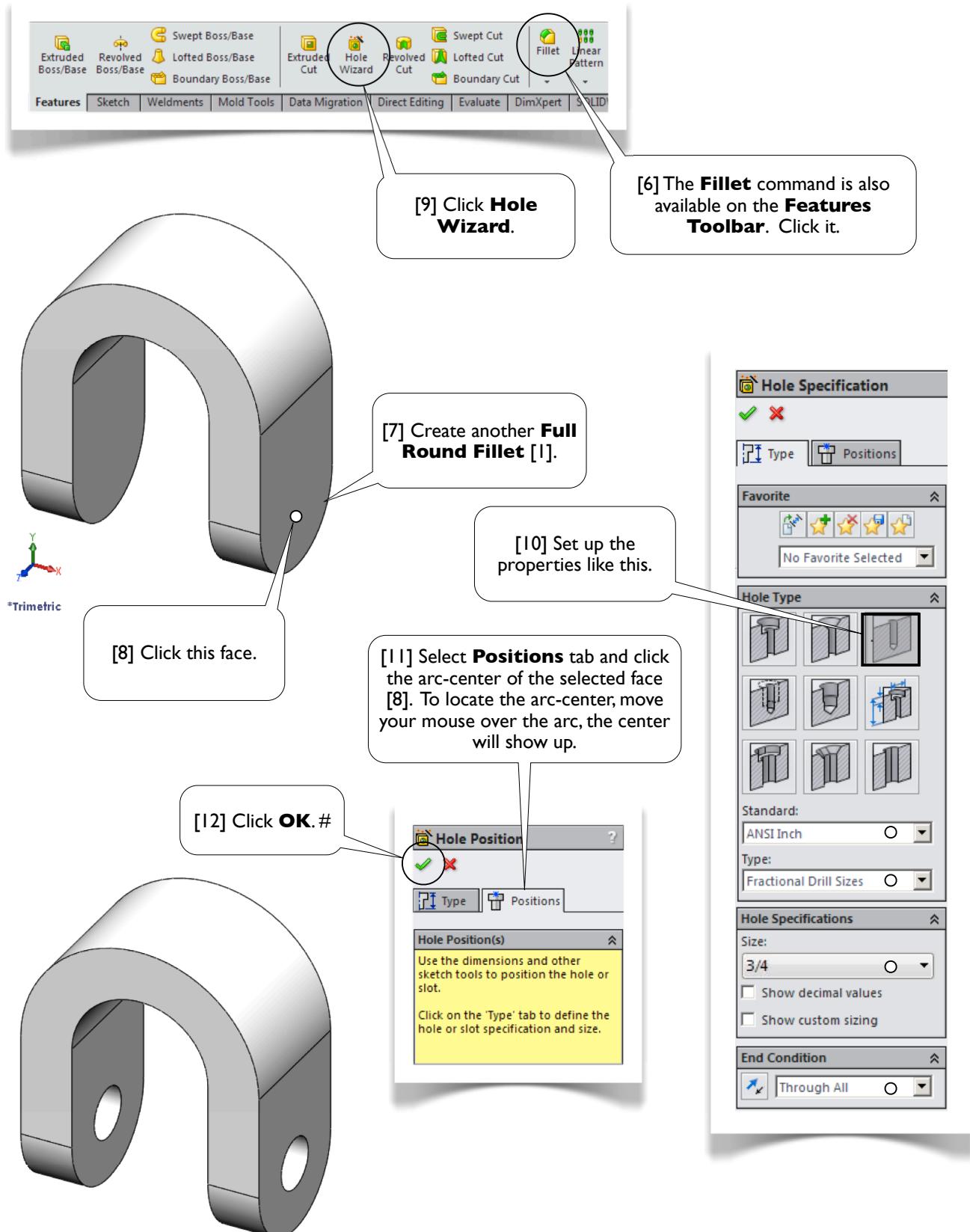
[3] If there are any blue entities, select **Hide/Show Items>View Sketch Relations** (I.3-3[10], page 25) to view all relations. Add relations to fix the entities.



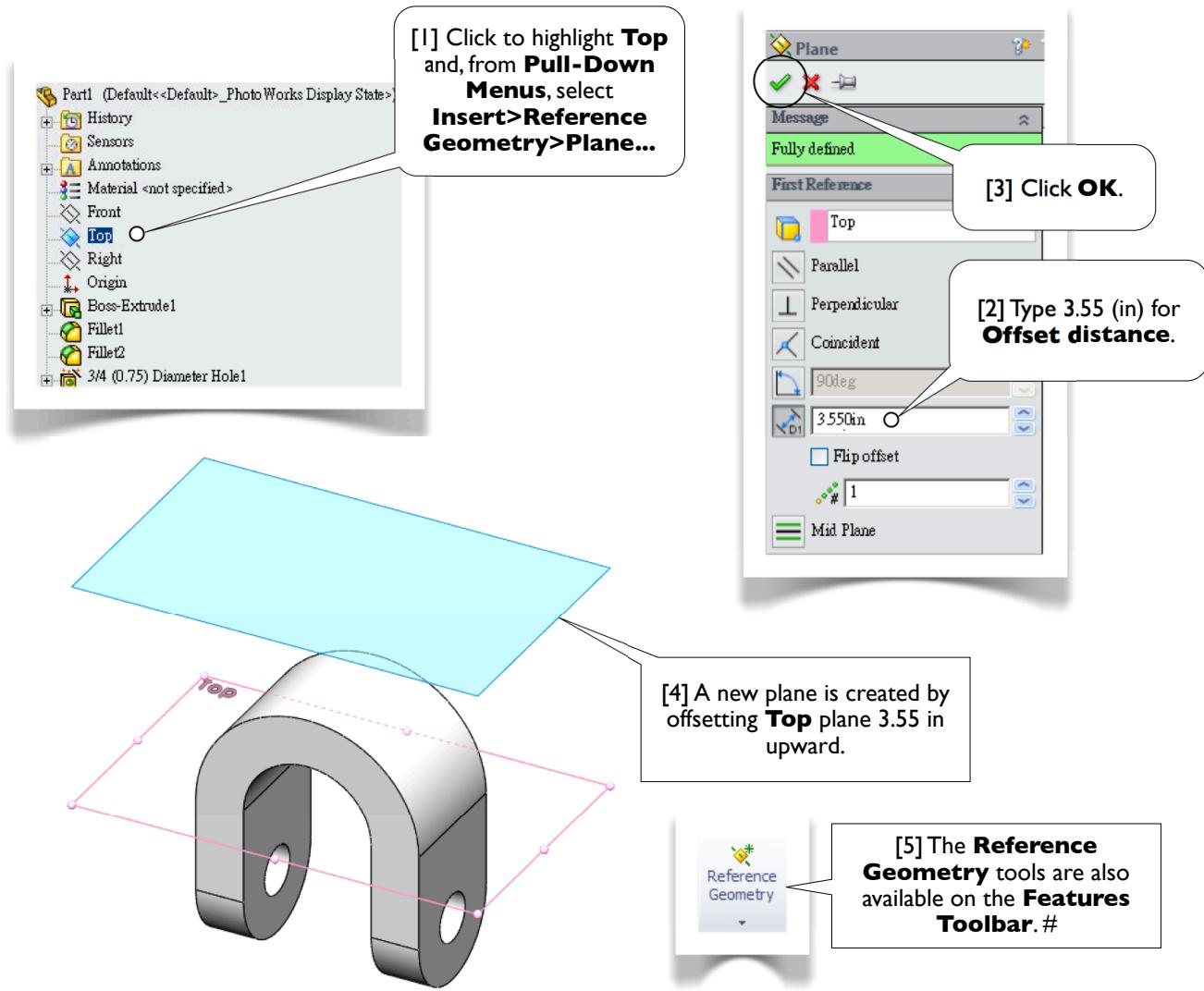


2.3-4 Create Rounds and Holes

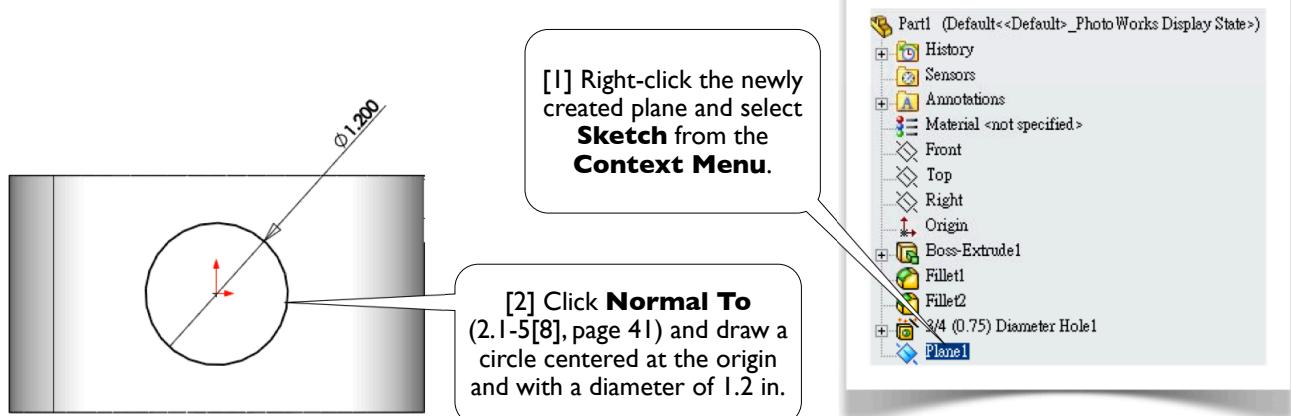


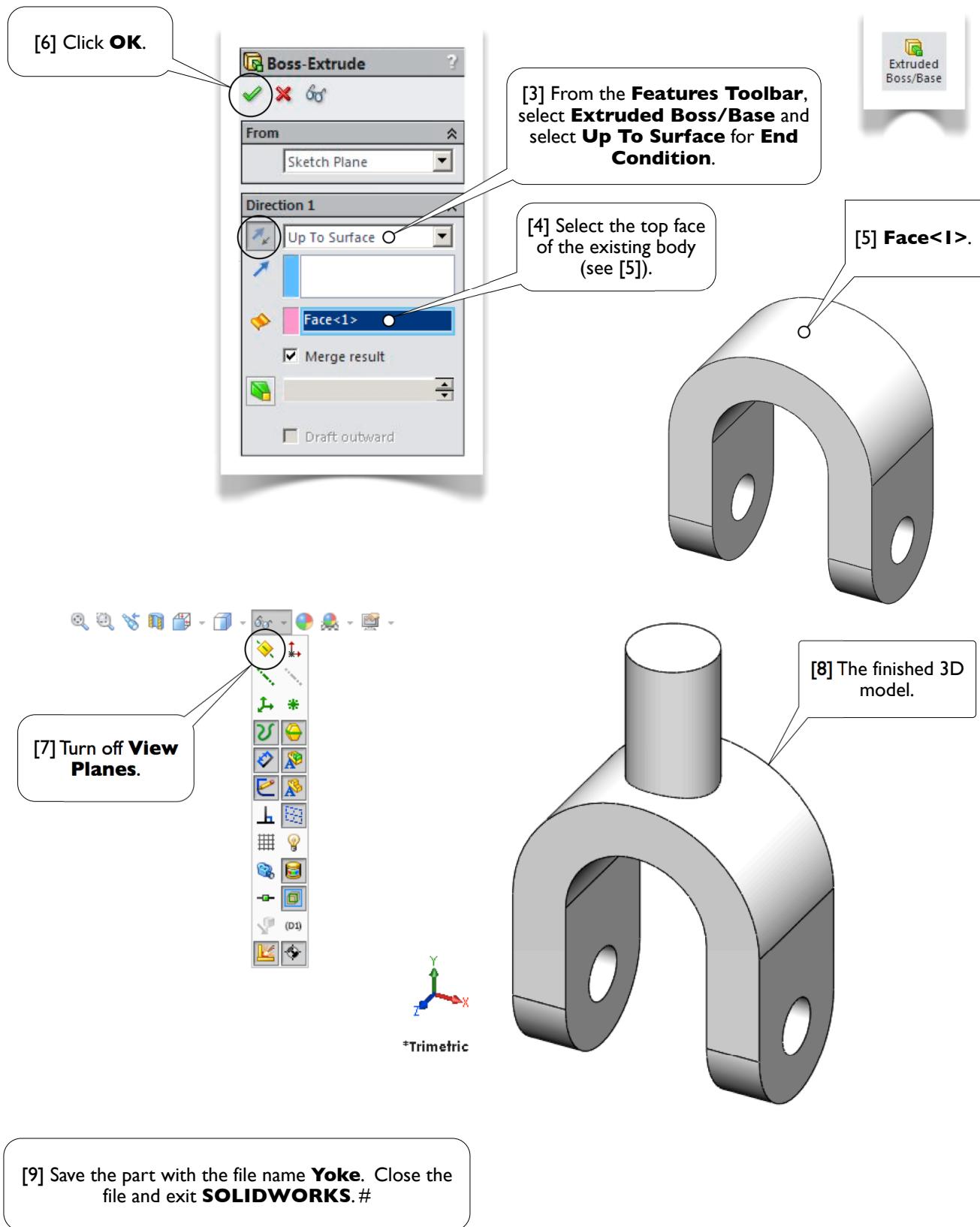


2.3-5 Create a Plane



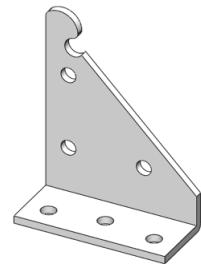
2.3-6 Create the Shaft



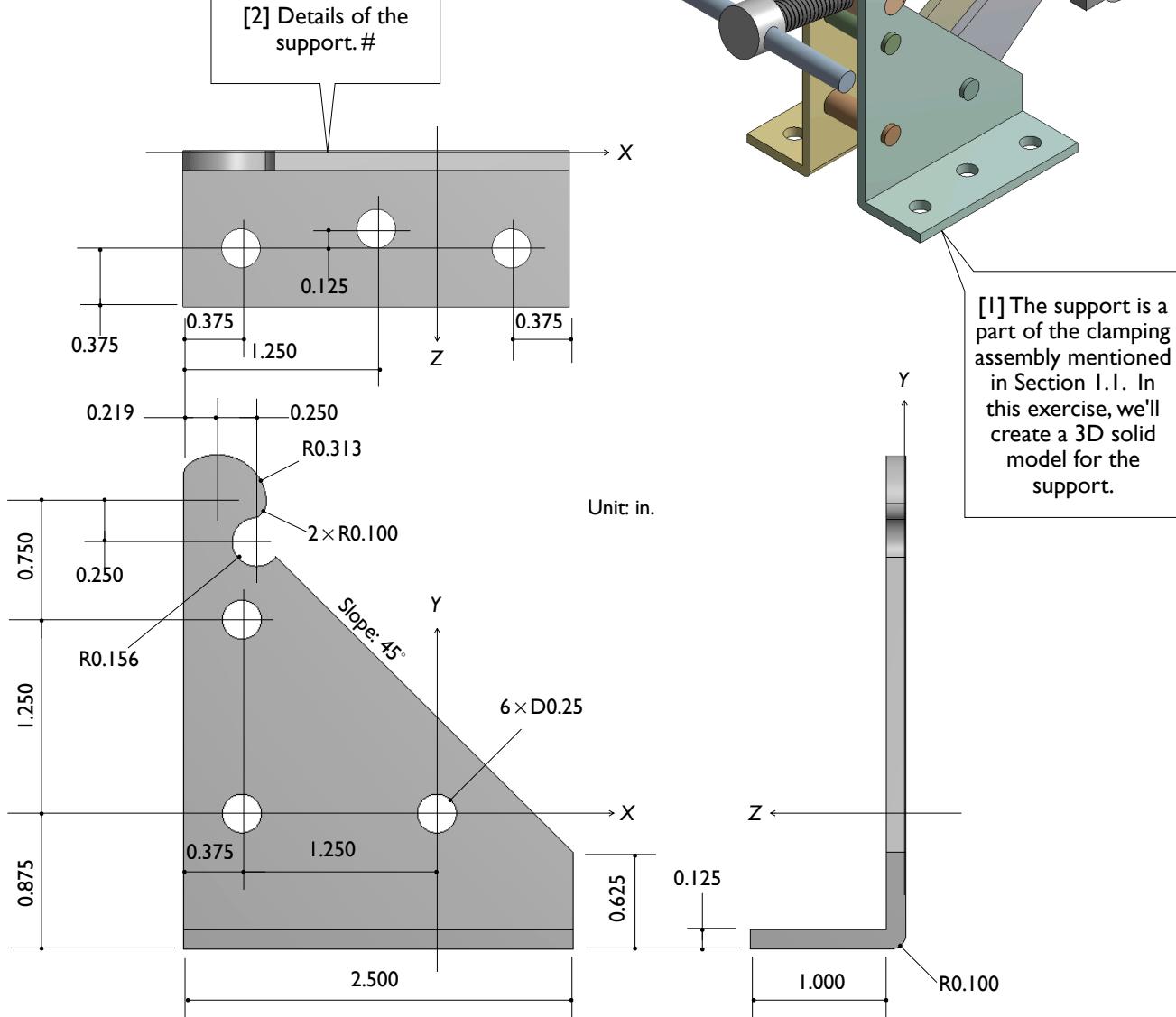


Section 2.4

Support



2.4-1 About the Support

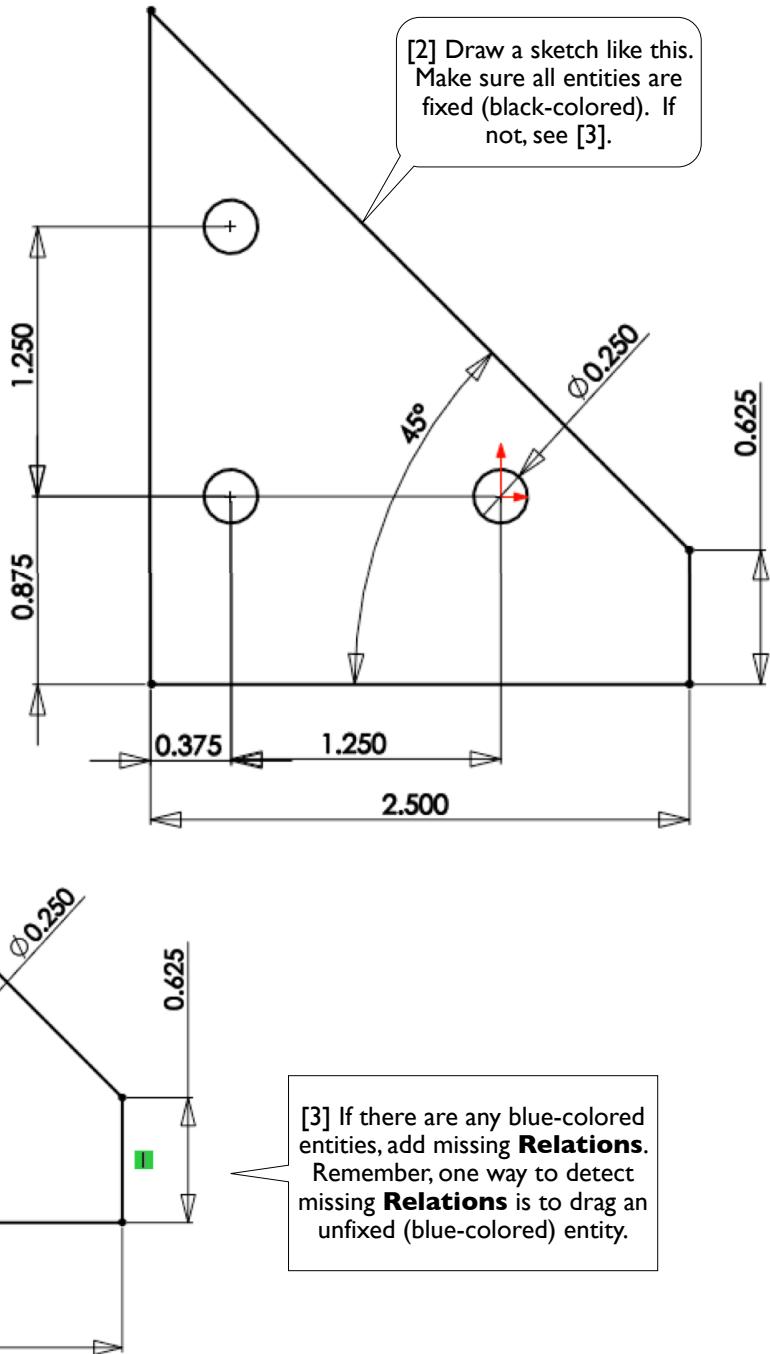


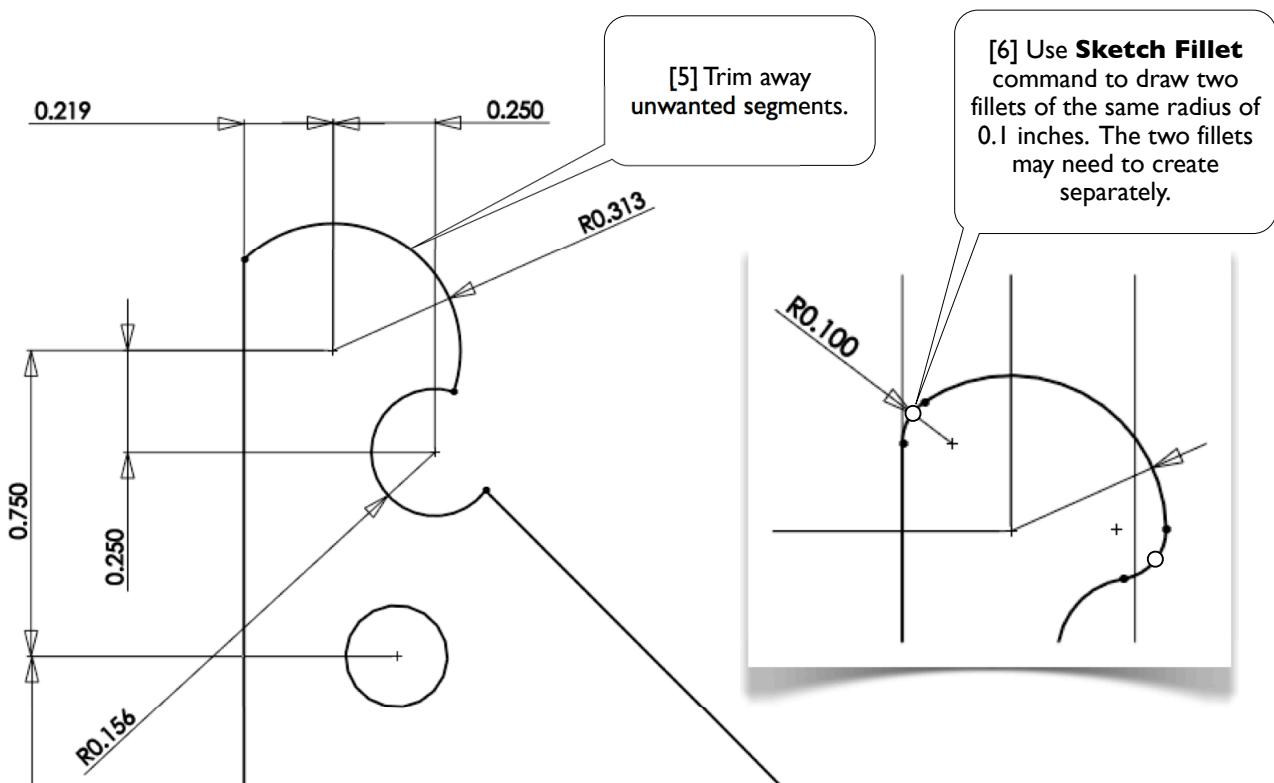
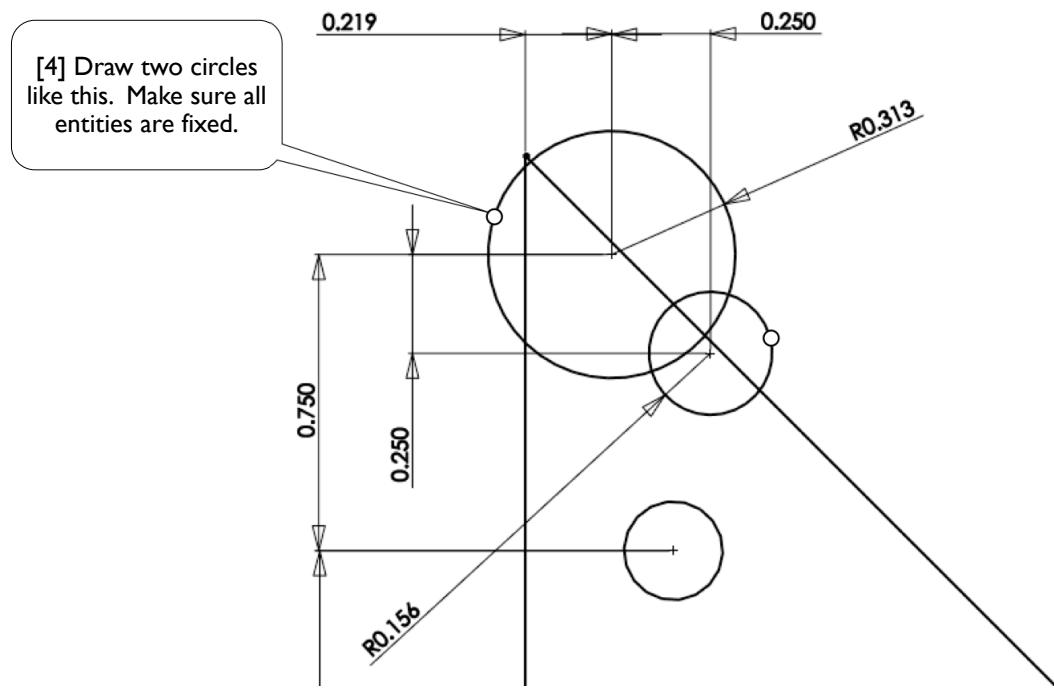
2.4-2 Start Up

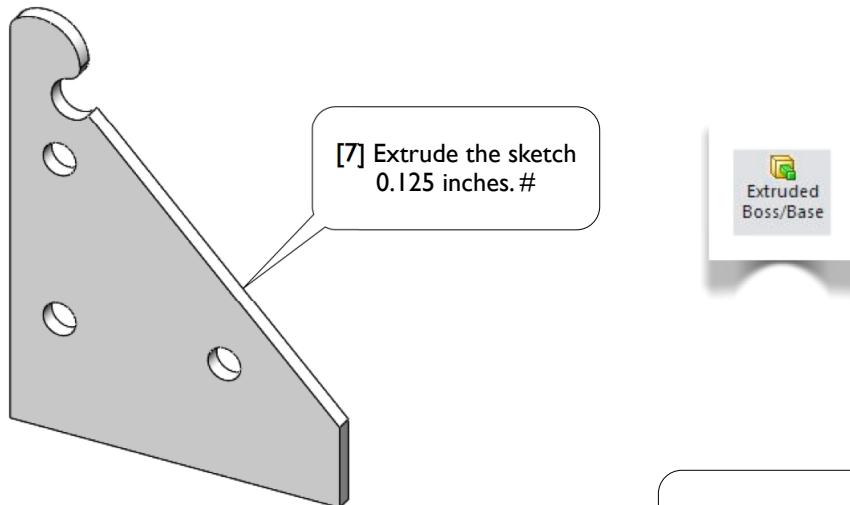
[!] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system and with 3 decimal places for the length unit. #

2.4-3 Create Vertical Plate

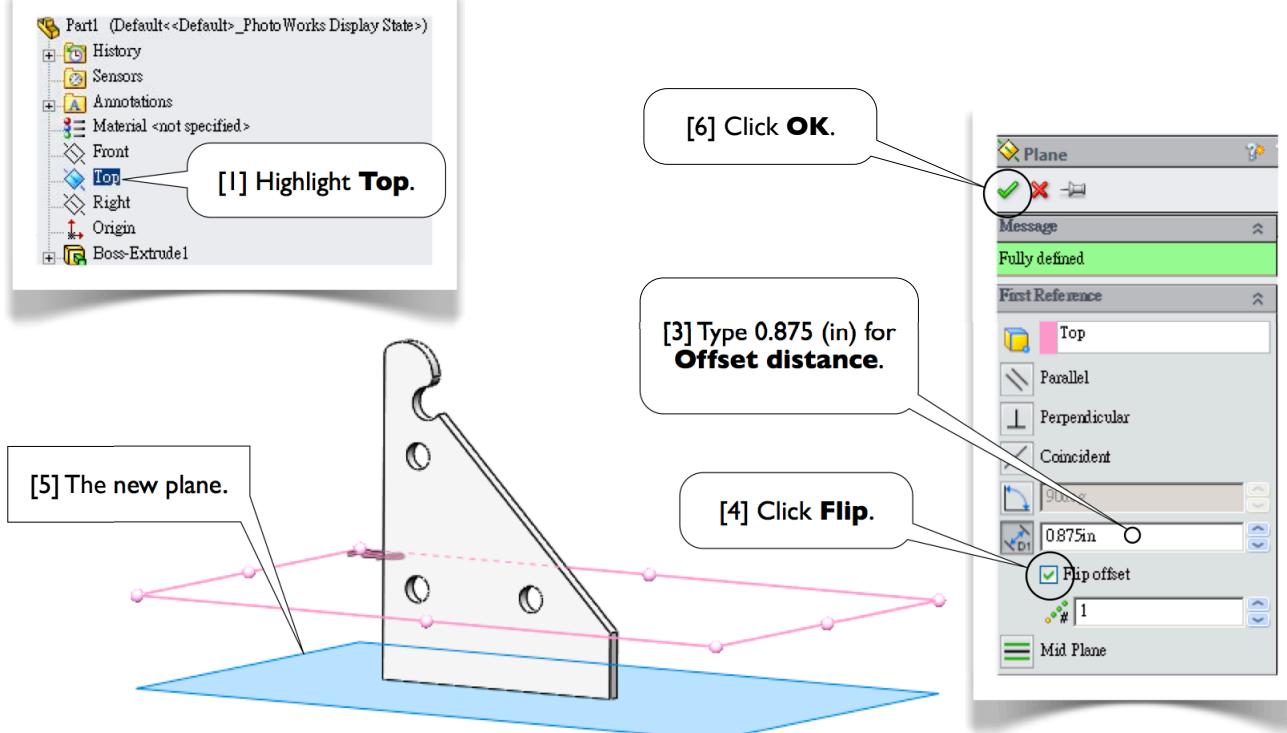
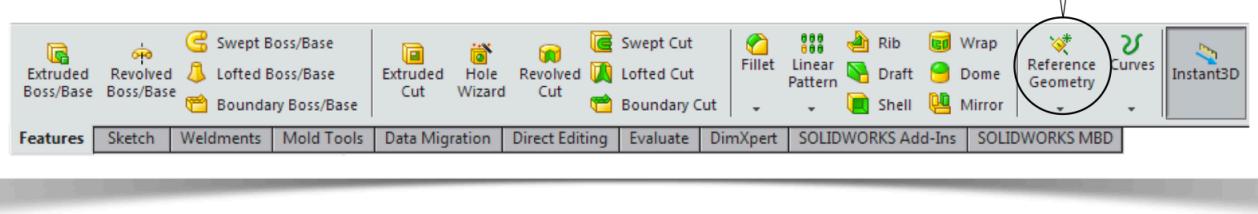
[!] Create a sketch on **Front** plane.

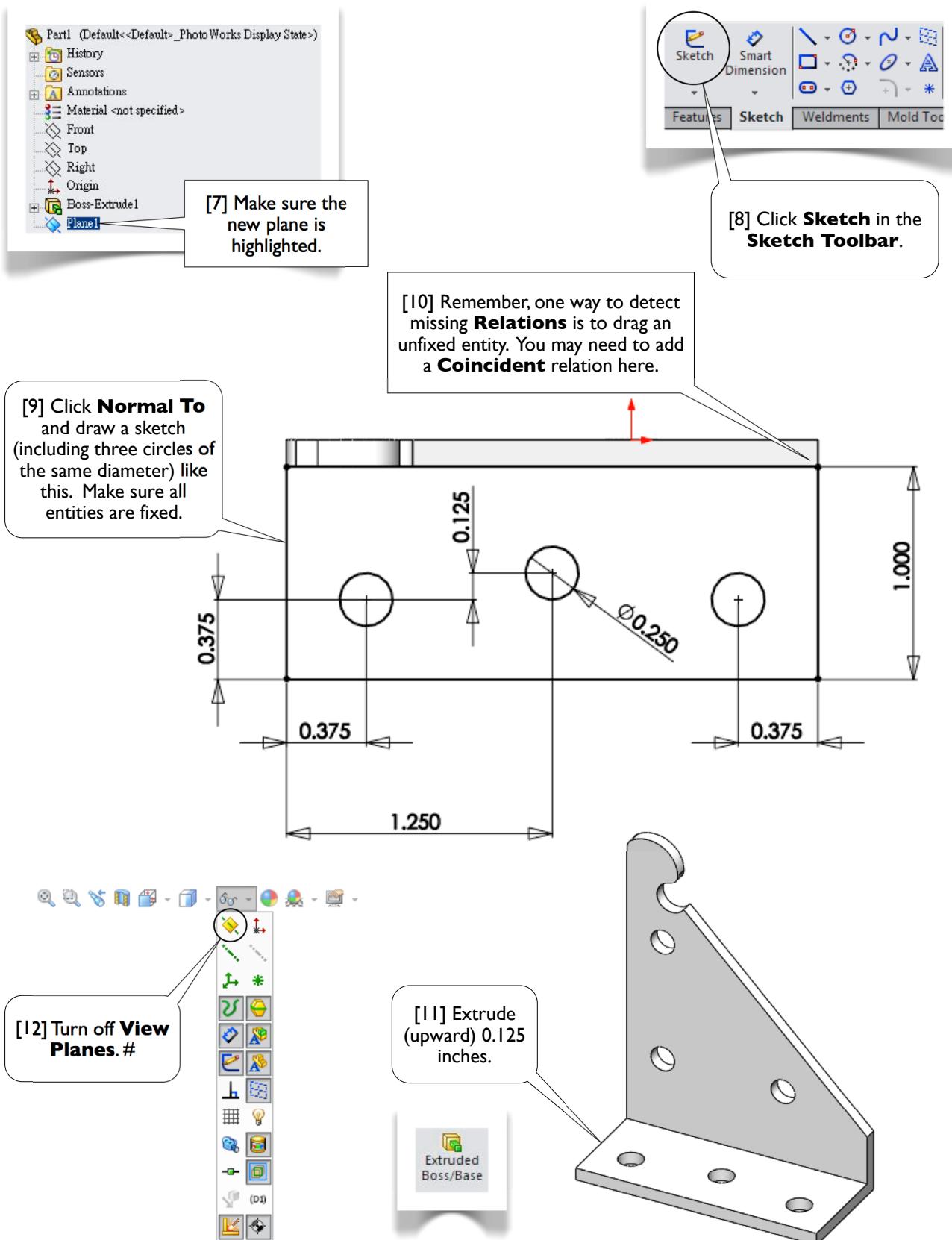




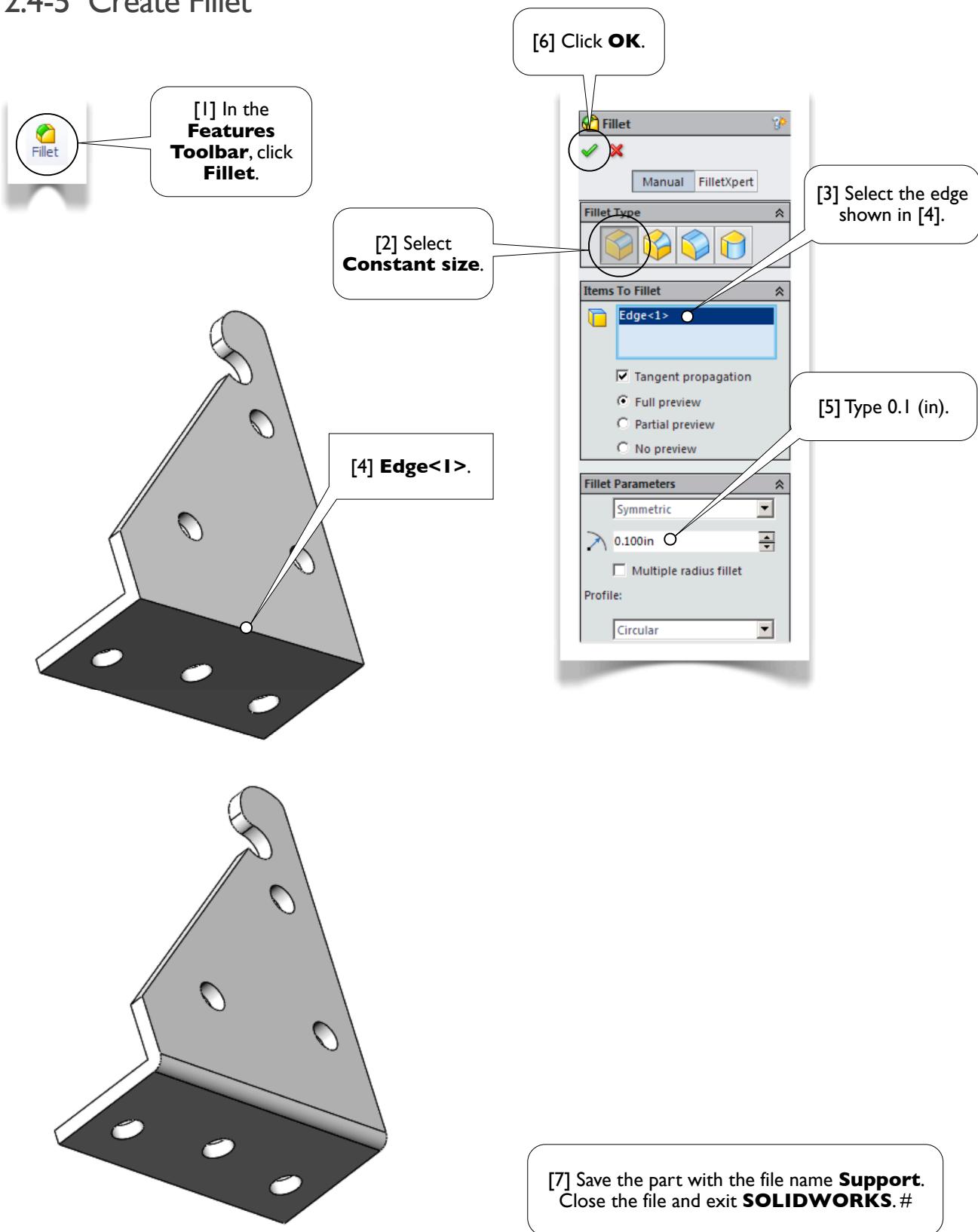


2.4-4 Create Horizontal Plate





2.4-5 Create Fillet



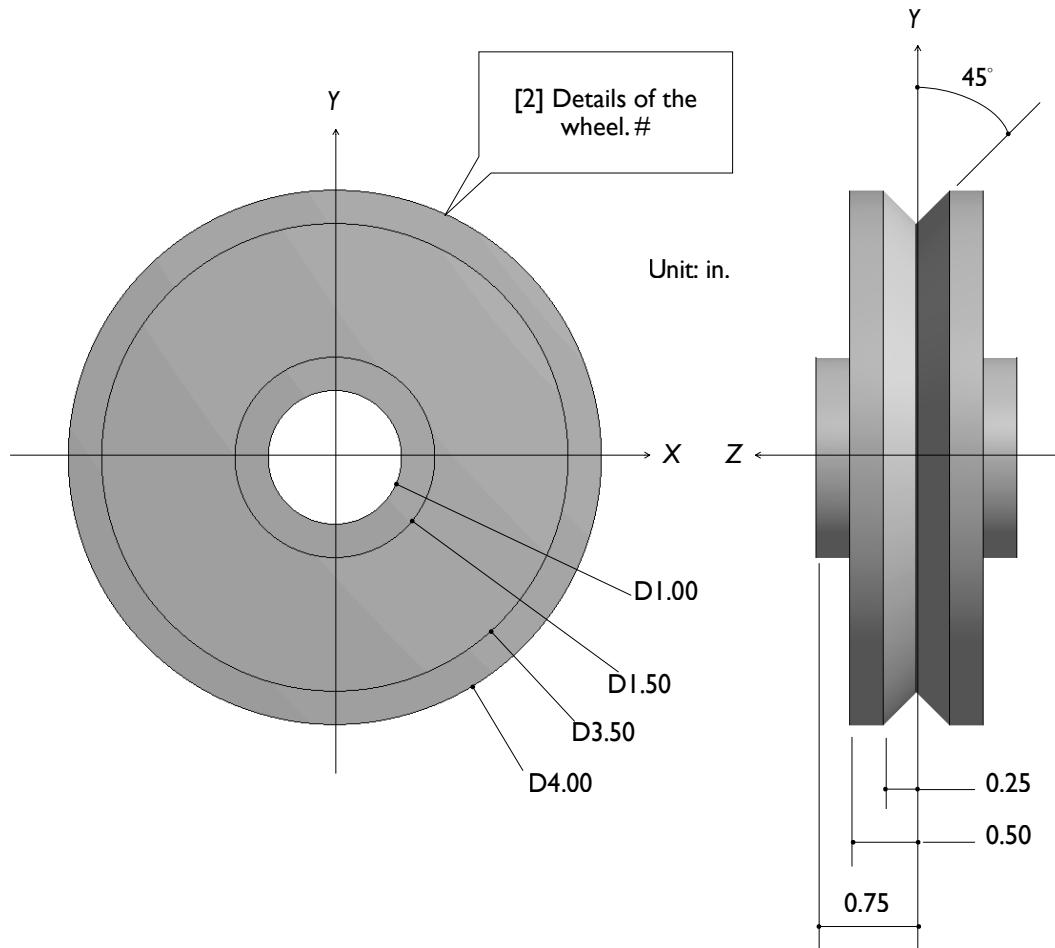
Section 2.5

Wheel



2.5-1 About the Wheel

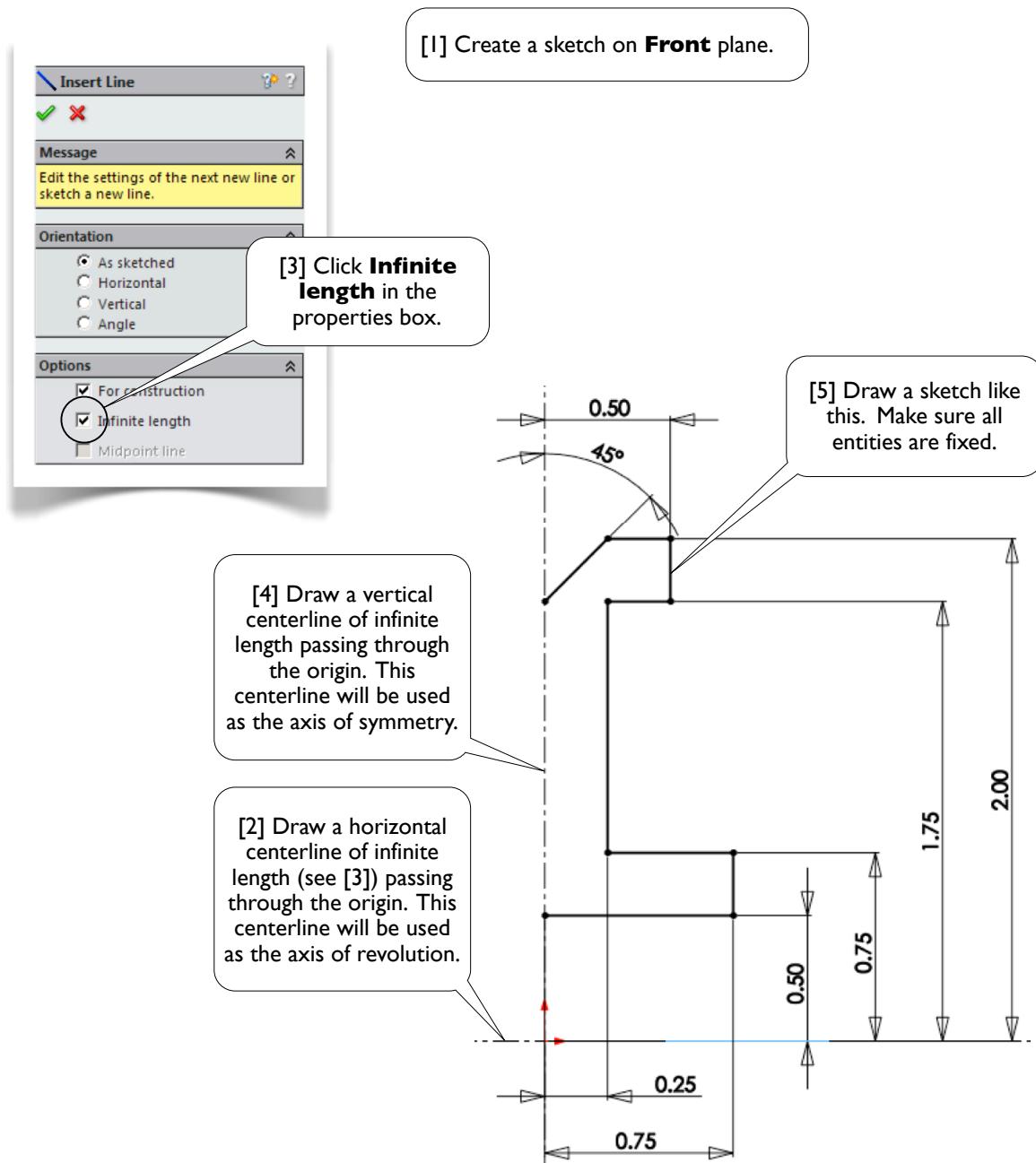
[1] So far, we exclusively used **Extrude** command to create 3D solids. In this section, we introduce another command to create 3D solids: **Revolve**, which takes a sketch as the profile and revolves about an axis to create a 3D solid body. We'll create a 3D solid model for a wheel [2]. The wheel is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

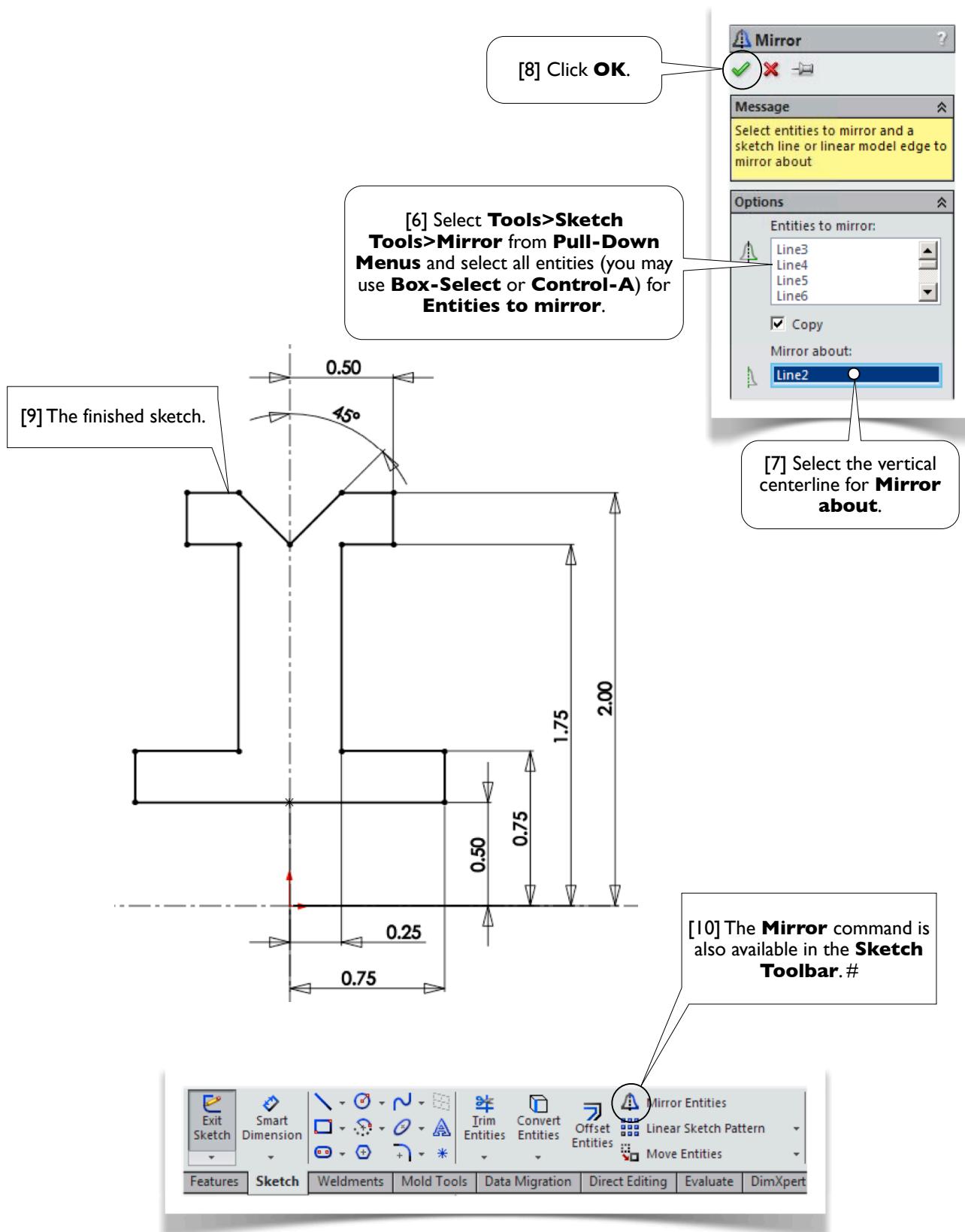


2.5-2 Start Up

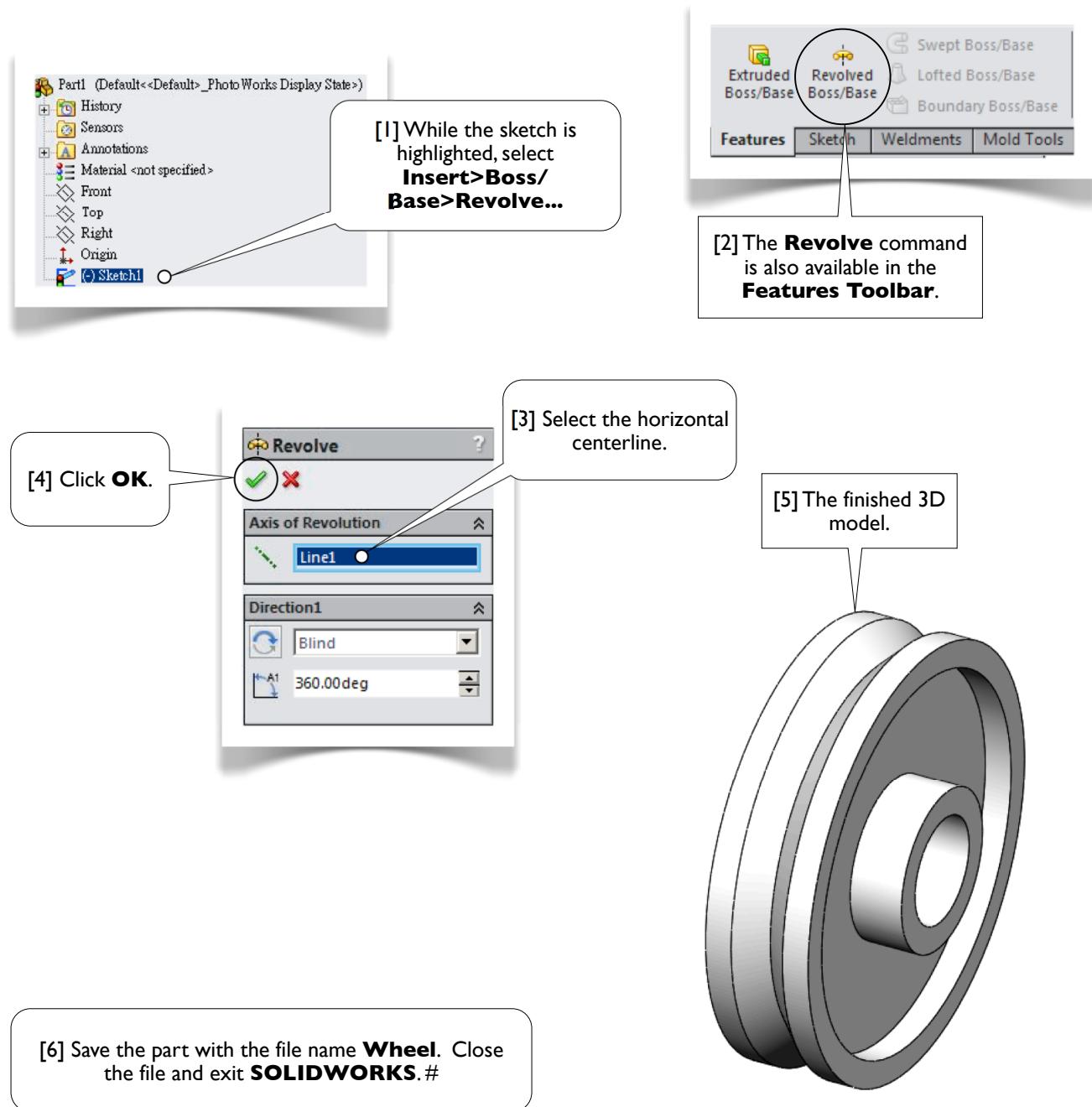
[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 2 decimal places for the length unit.#

2.5-3 Create a Sketch for the Profile





2.5-4 Revolve the Sketch



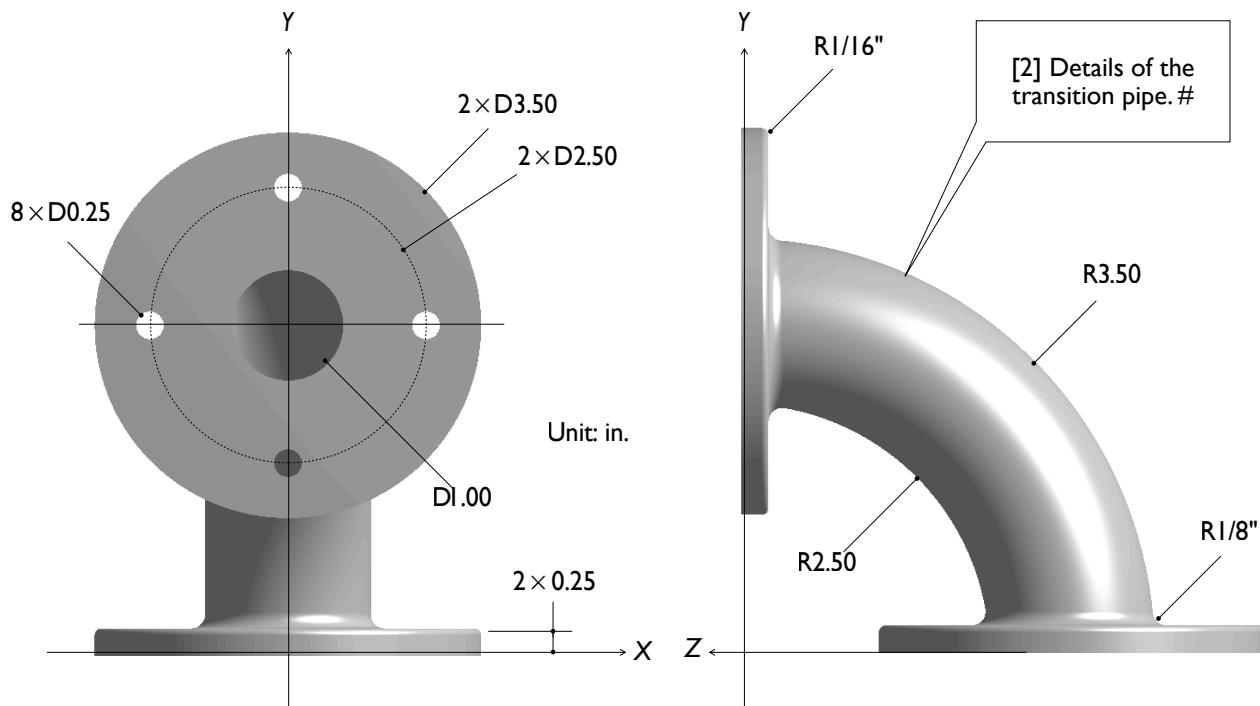
Section 2.6

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.



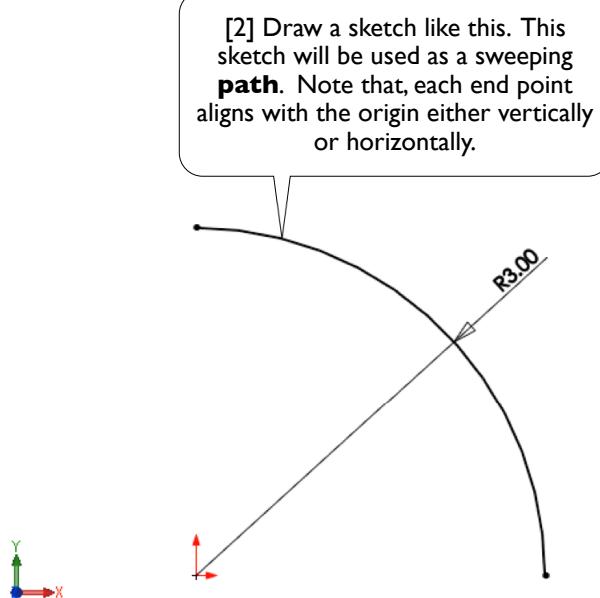
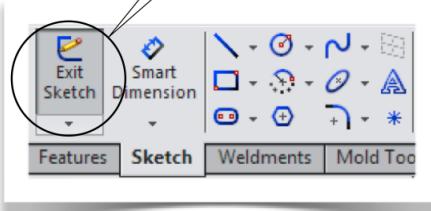
2.6-2 Start Up

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

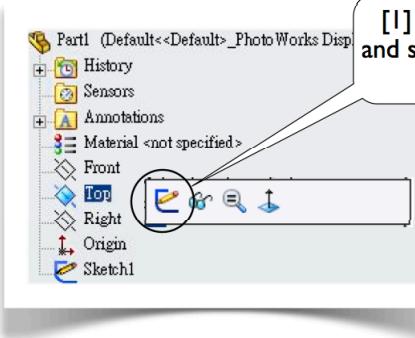
2.6-3 Create a Sketch for the Path

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

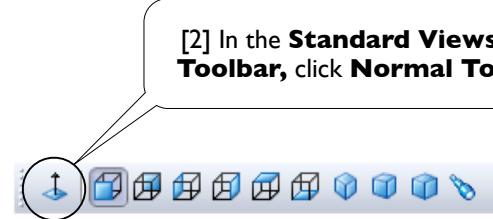
[3] Click **Exit Sketch** in the **Sketch Toolbar**.#



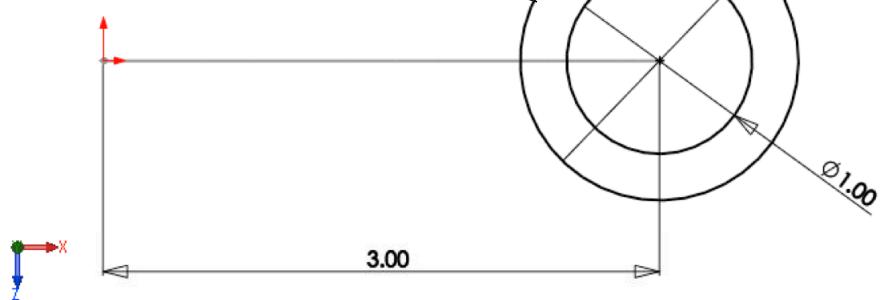
2.6-4 Create a Sketch for the Profile

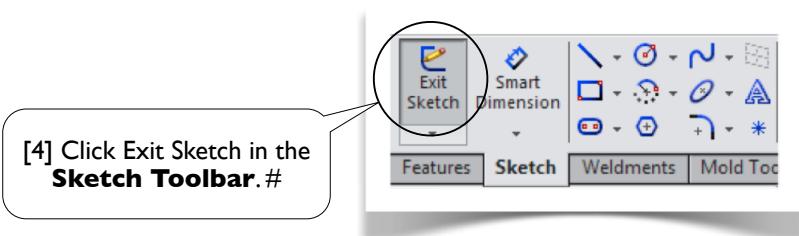


[1] Right-click **Top** plane and select **Sketch** to create a second sketch.



[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].





2.6-5 Create the Curved Pipe

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

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

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

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

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

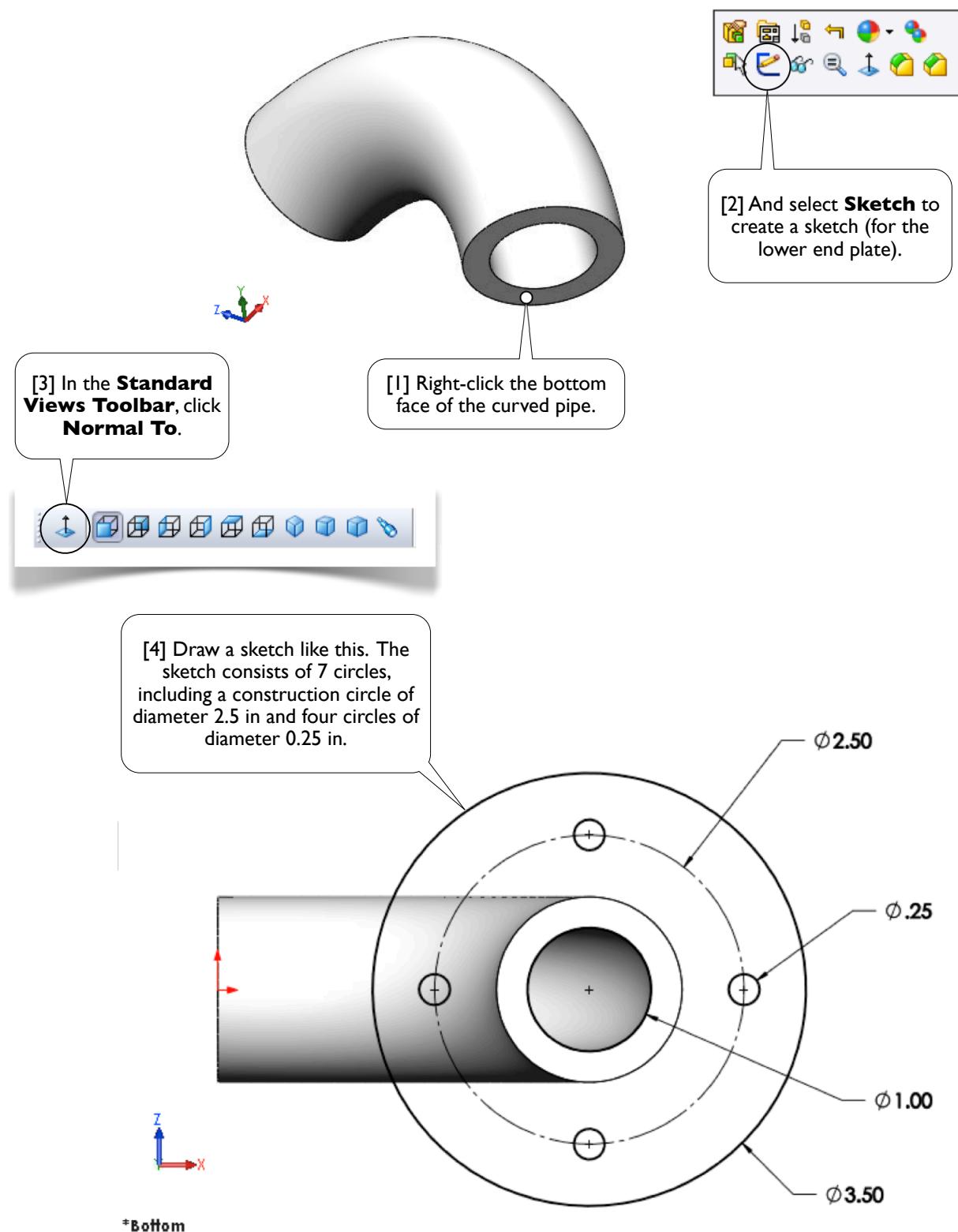
[6] Select the **Path** sketch (**Sketch1**).

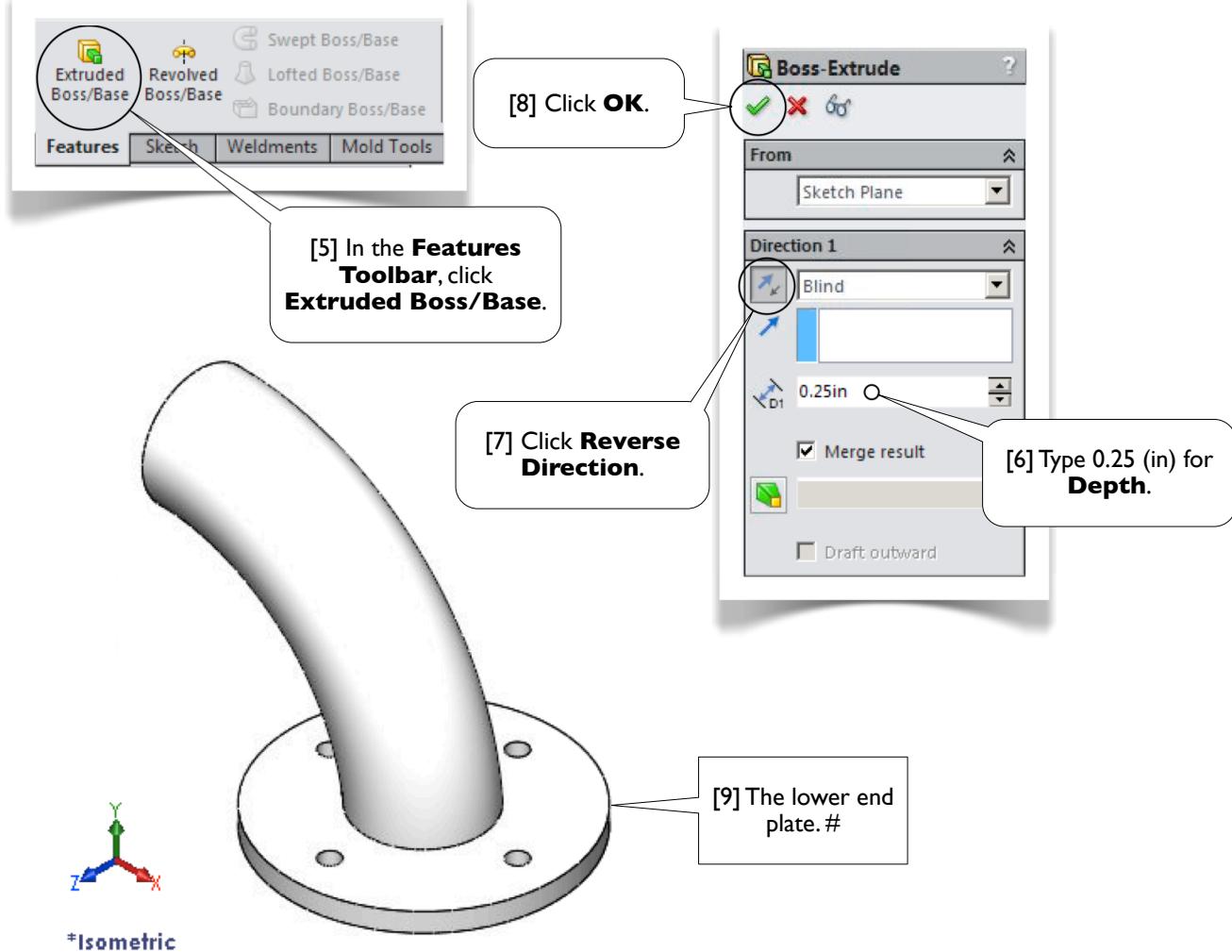
[7] Click **OK**.

[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. #

*Isometric

2.6-6 Create the Lower End Plate

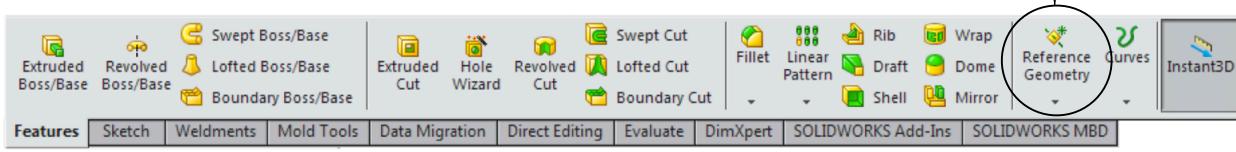


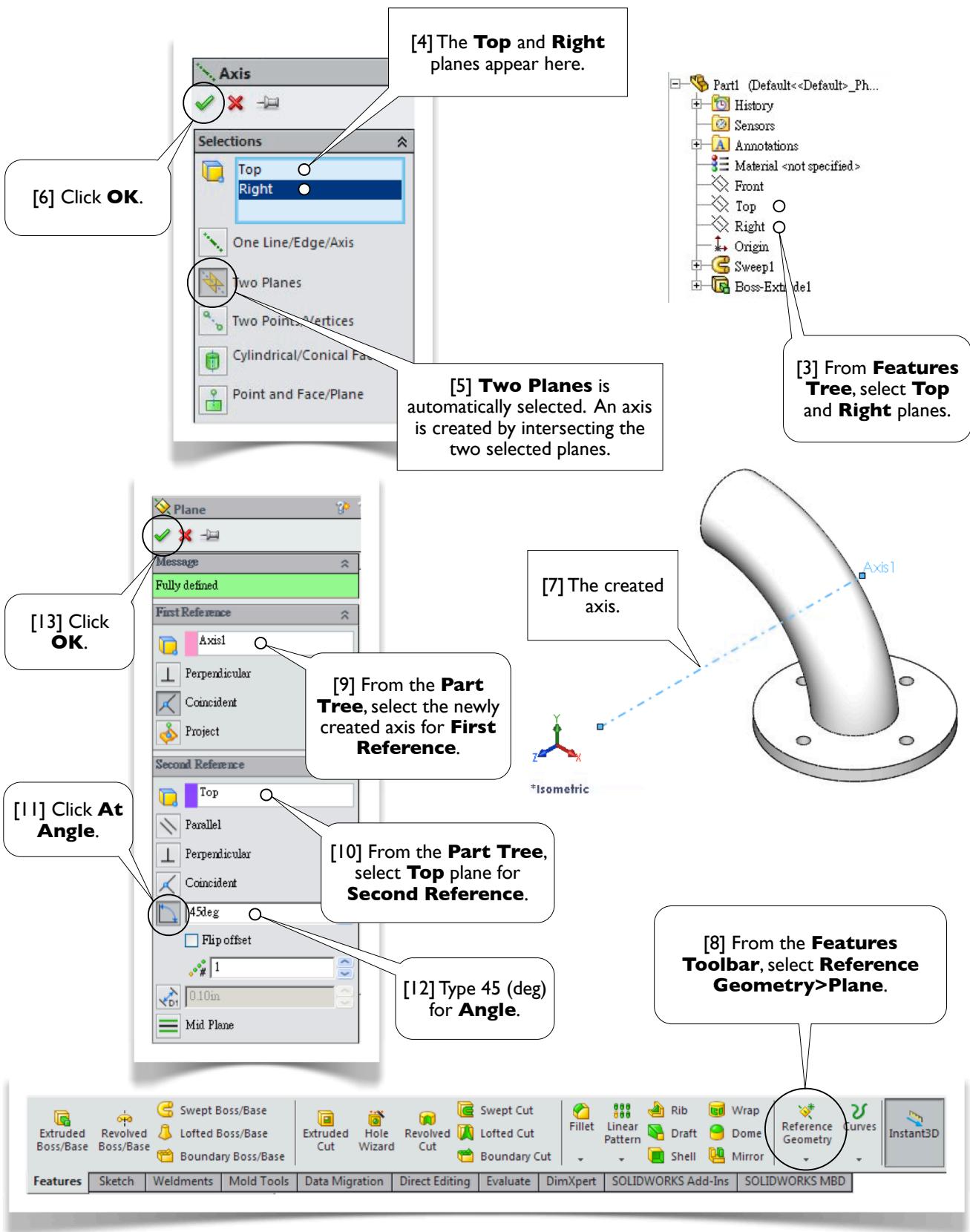


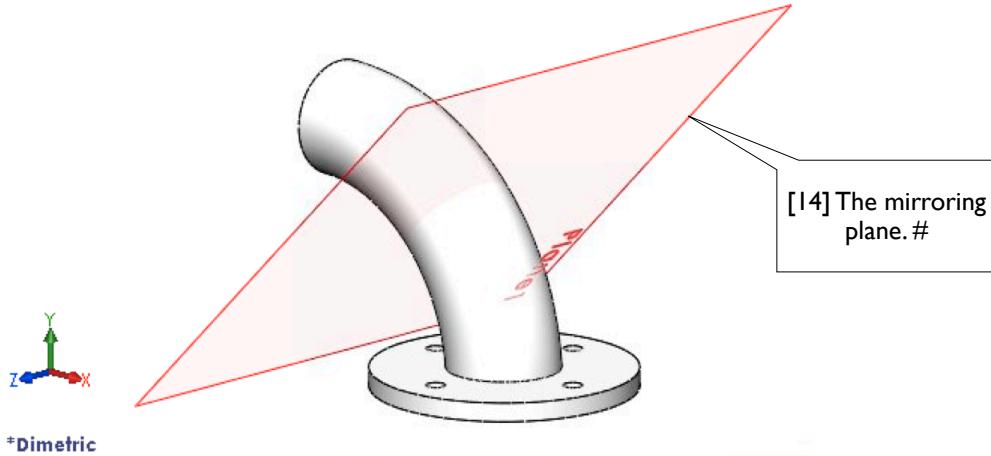
2.6-7 Create a Mirroring Plane

[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**.







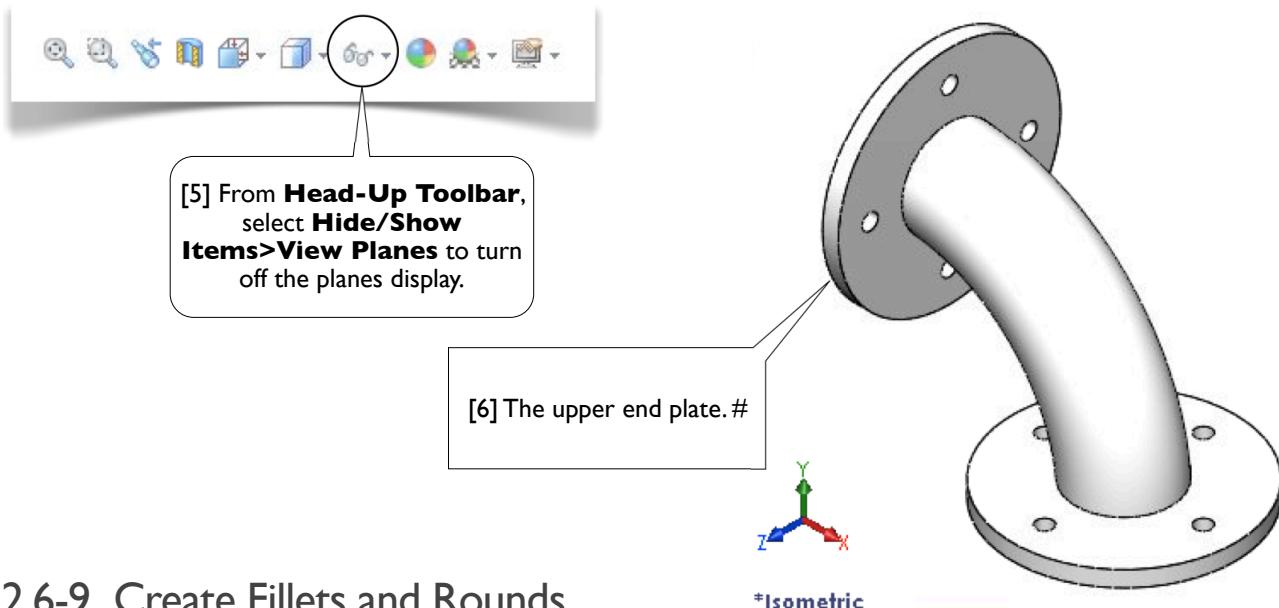
2.6-8 Create the Upper End Plate

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

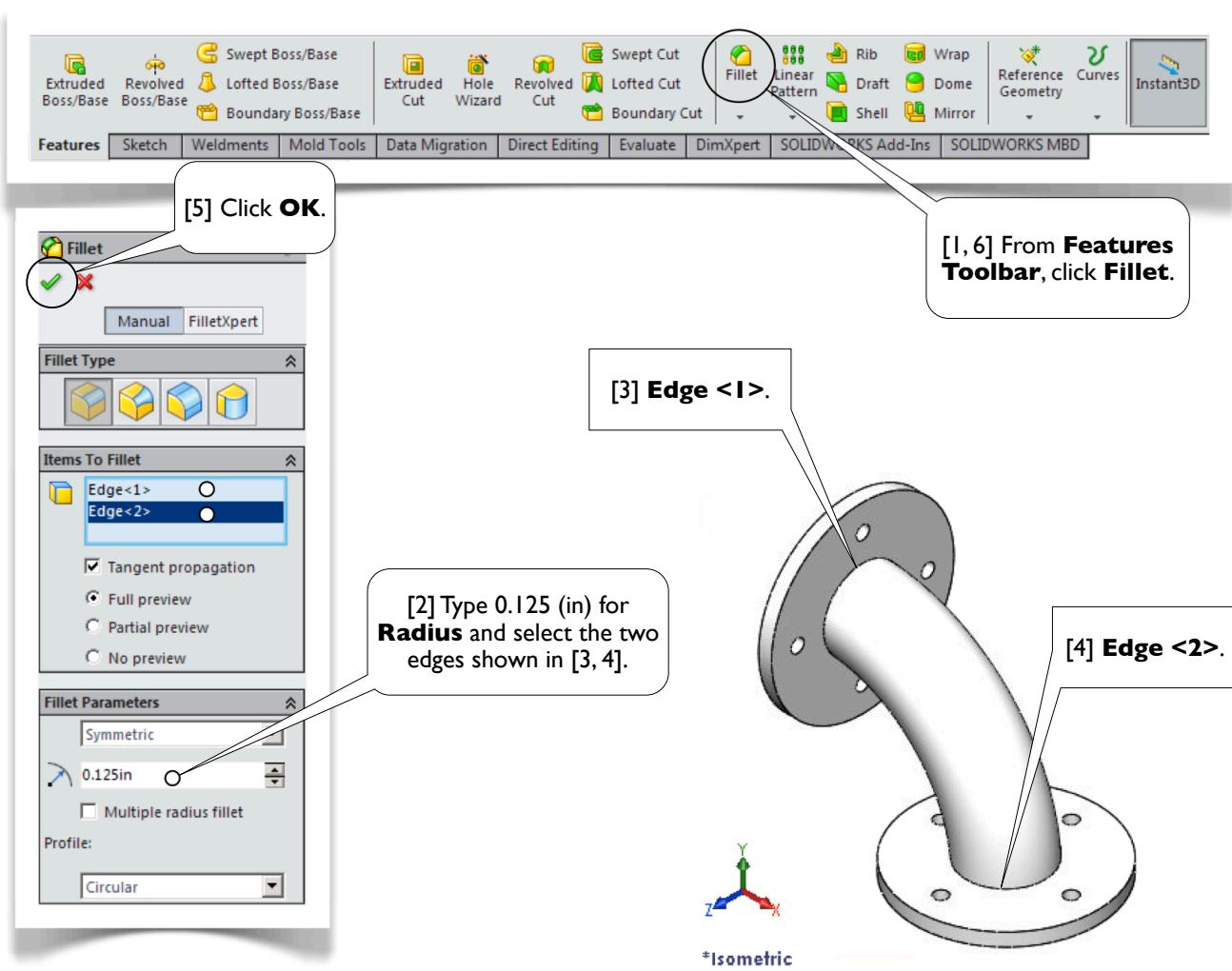
[2] In **Features Toolbar**, click **Mirror**.

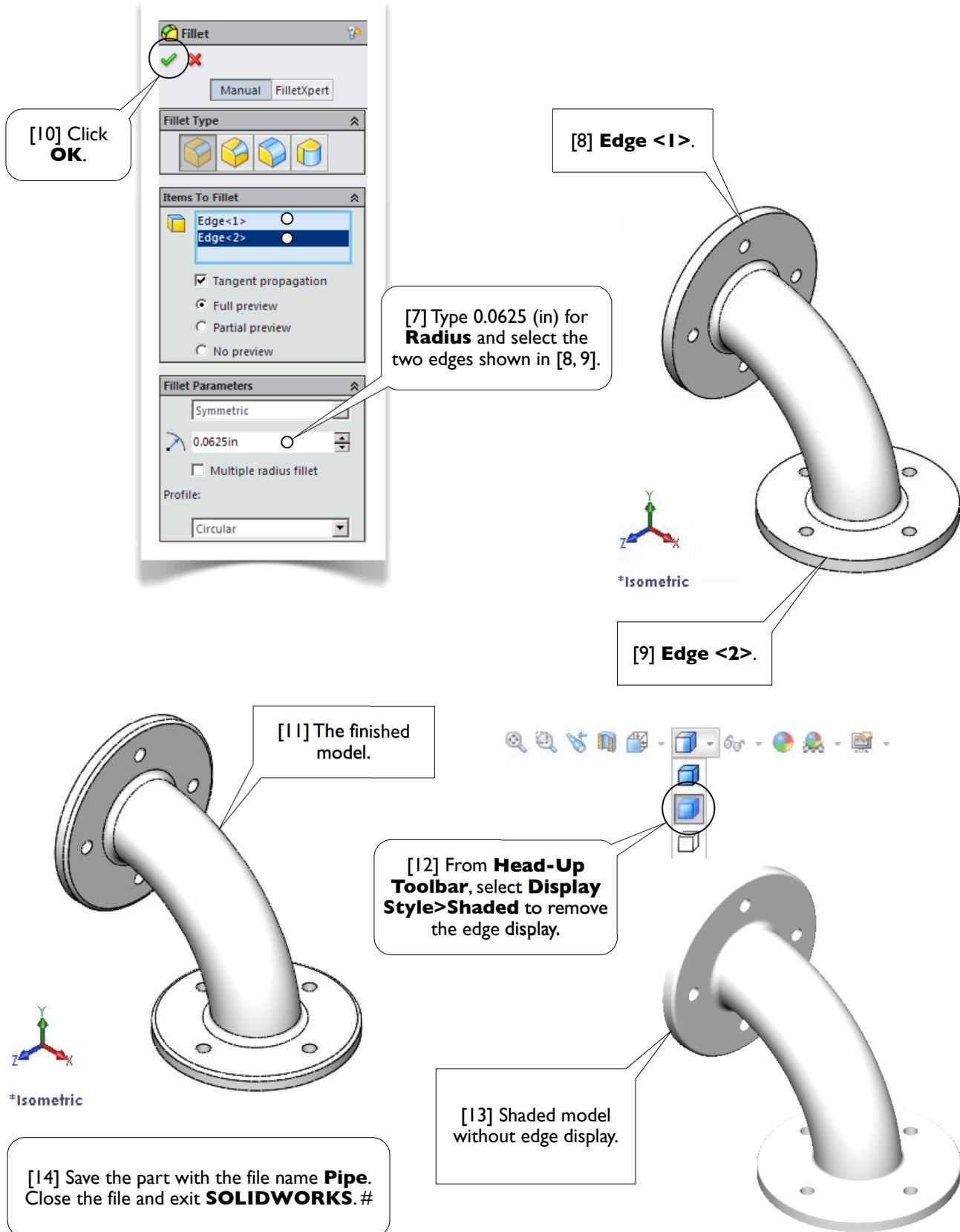
[3] From the **Part Tree** (or from the **Graphics Area**) select the lower end plate (**Boss-Extrude1**).

[4] Click **OK**.



2.6-9 Create Fillets and Rounds





Section 2.7

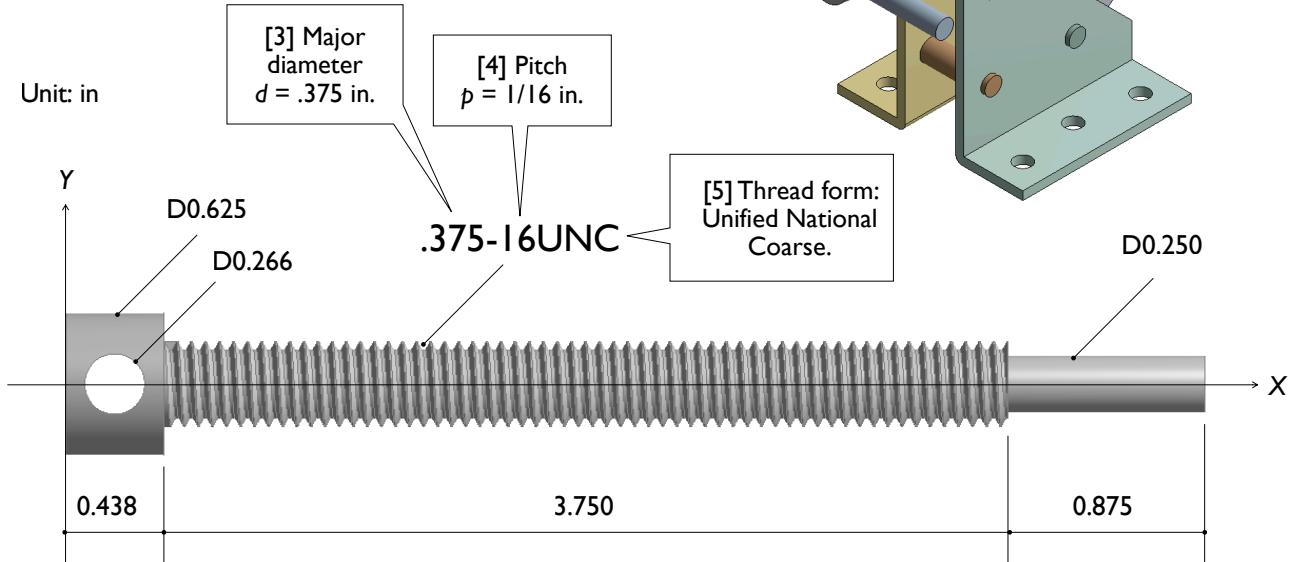
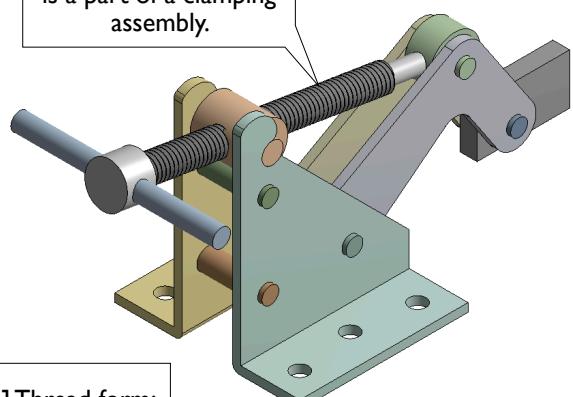
Threaded Shaft



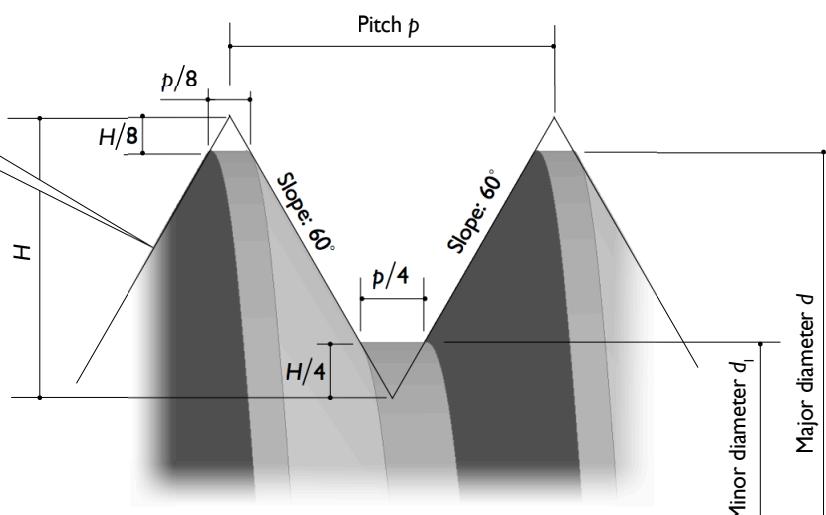
2.7-1 About the Threaded Shaft

[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.



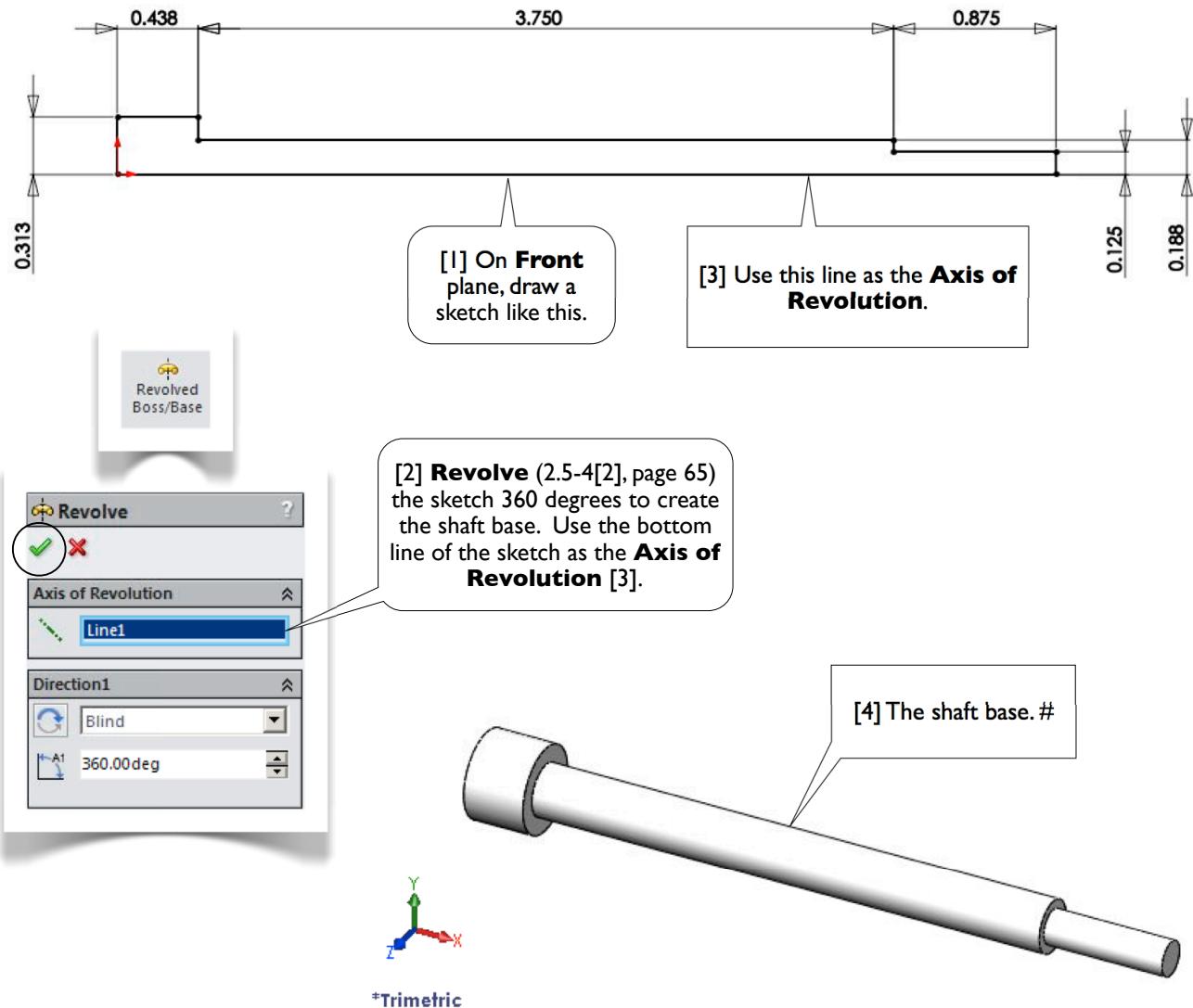
$$\begin{aligned}
 d &= 0.375 \text{ in} \\
 p &= 0.0625 \text{ in} \\
 H &= (\sqrt{3}/2)p = 0.0541266 \text{ in} \\
 d_i &= d - \frac{5H}{8} \times 2 = 0.307342 \text{ in} \\
 \frac{p}{4} &= 0.015625 \text{ in} \\
 \frac{p}{8} &= 0.0078125 \text{ in}
 \end{aligned}$$



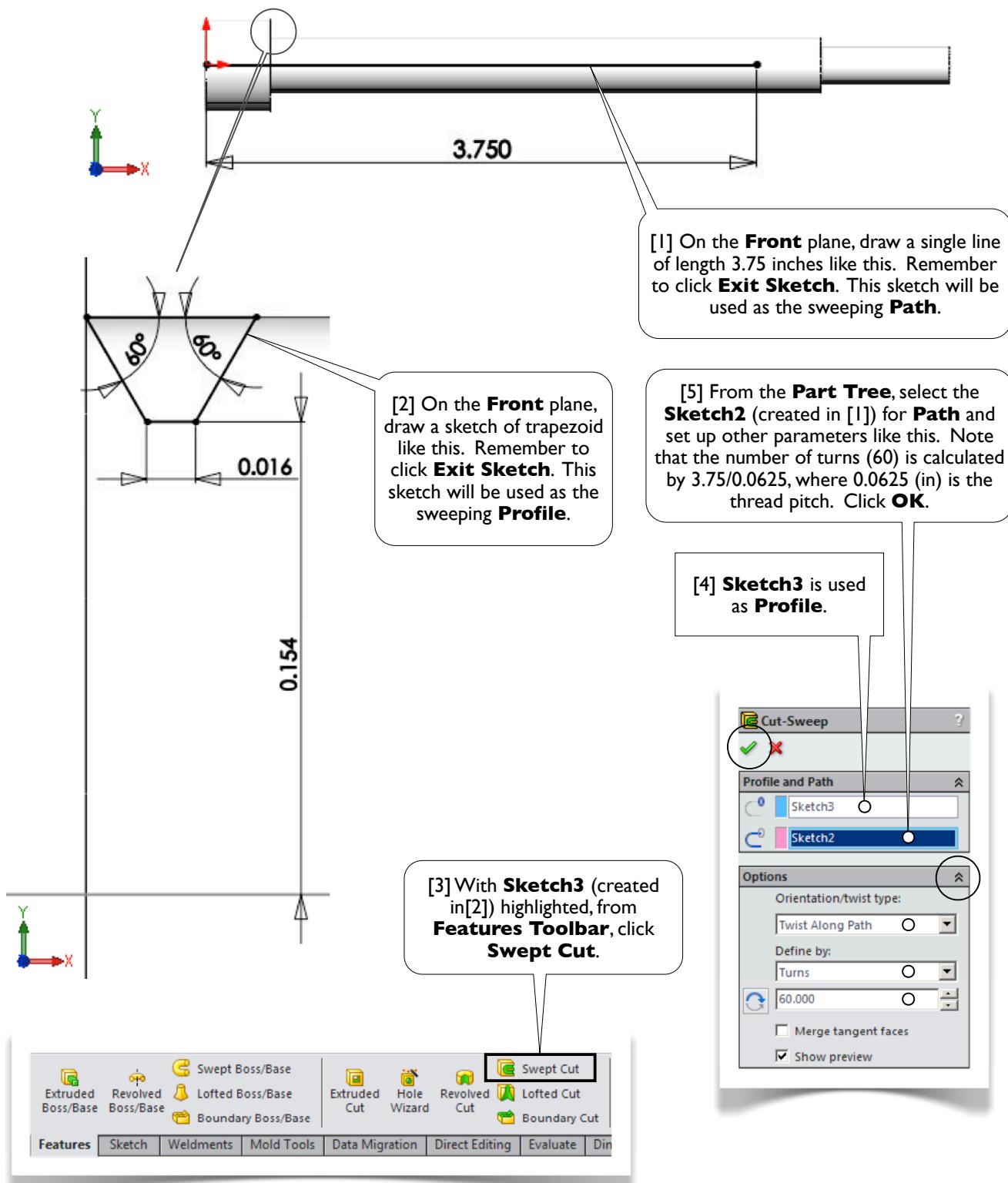
2.7-2 Start Up

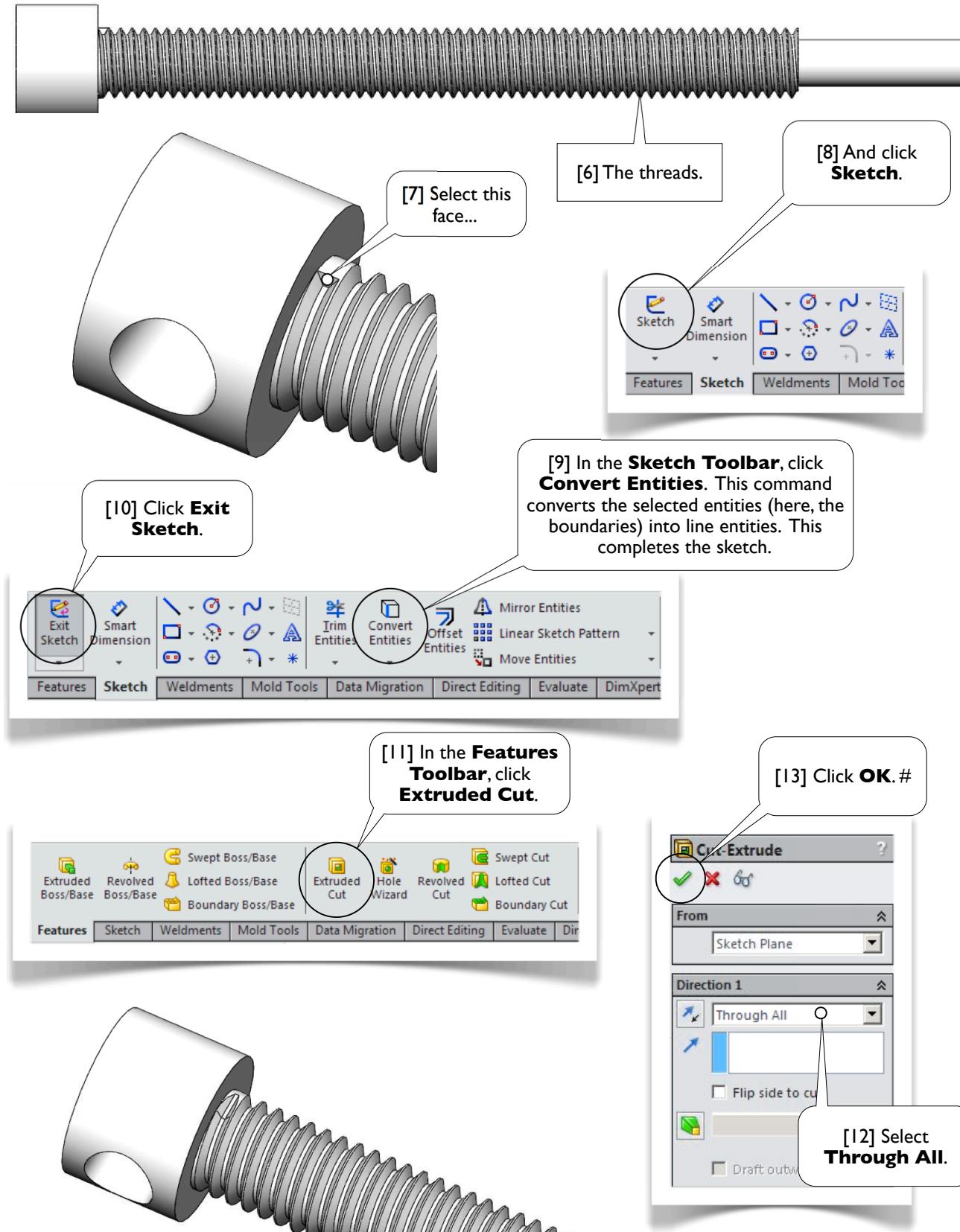
[1] Launch **SOLIDWORKS** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit. #

2.7-3 Create a Shaft Base

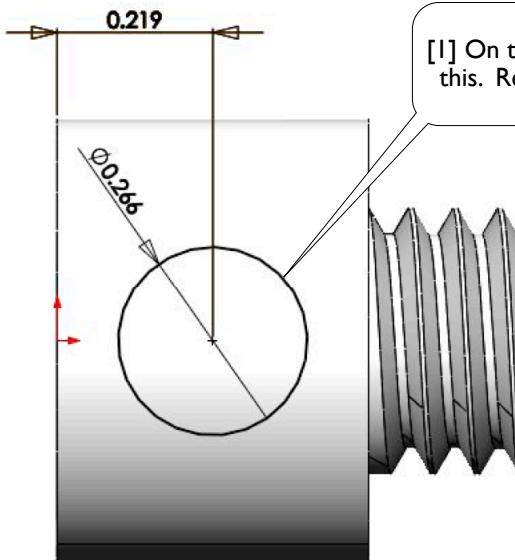


2.7-4 Create Threads

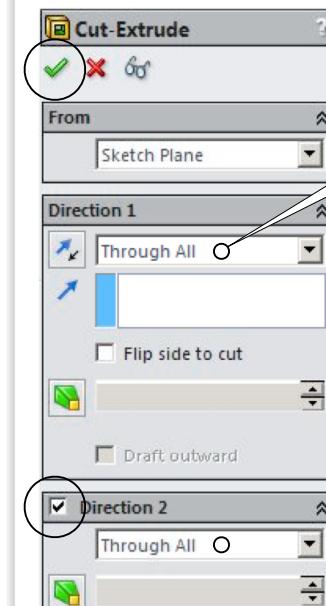




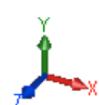
2.7-5 Create a Hole



[1] On the **Front** plane, draw a circle like this. Remember to click **Exit Sketch**.



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

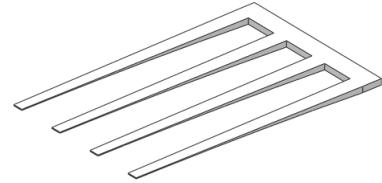


[3] The finished model.

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

Section 2.8

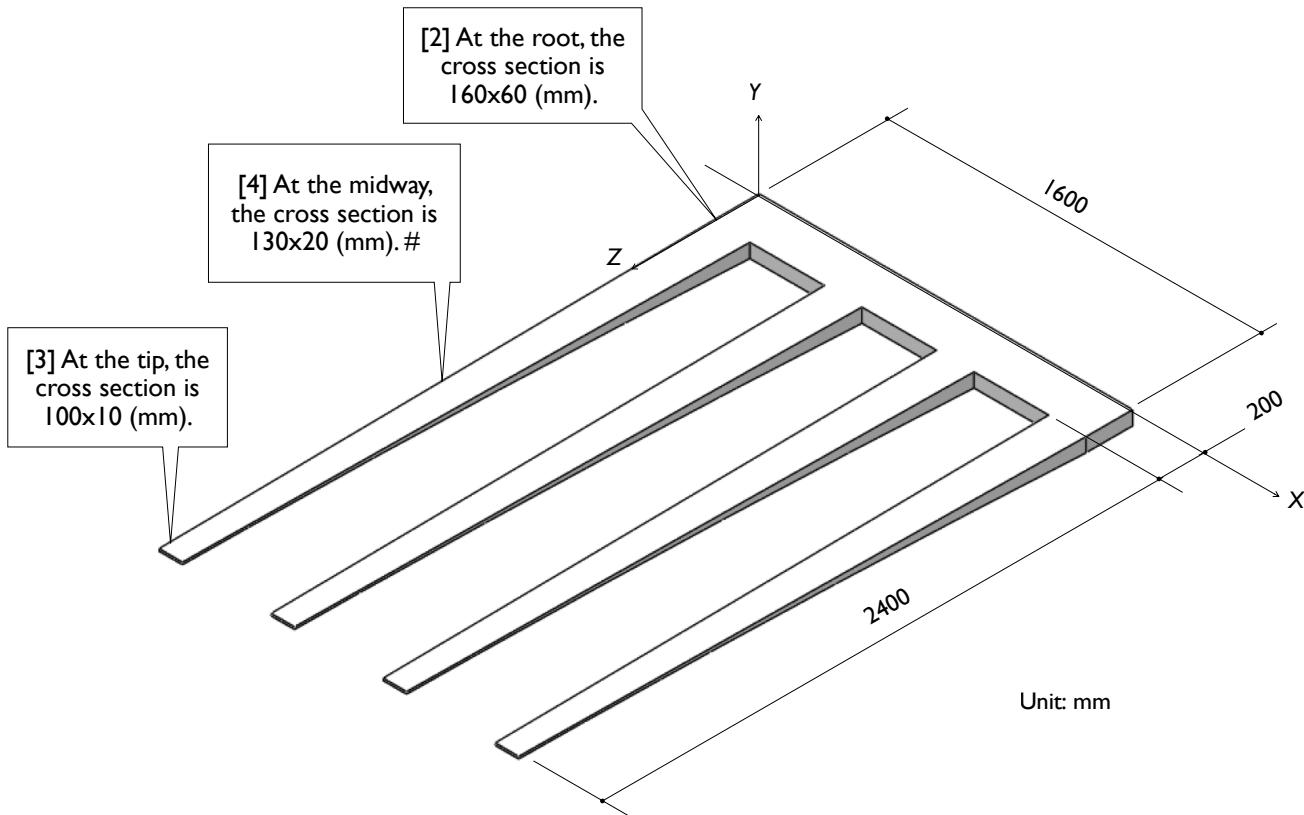
Lifting Fork



2.8-1 About the Lifting Fork

[!] 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.

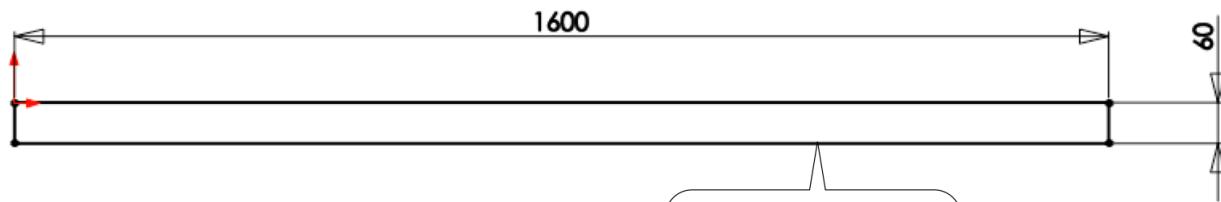
The cross sections of the prongs (fingers) are not uniform along the length [2, 3, 4]. The **Extrude** command or **Sweep** command can not be used to created the prongs. This exercise introduces a new command to create 3D solids: **Loft**, which takes a series of profiles and creates a 3D solid that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



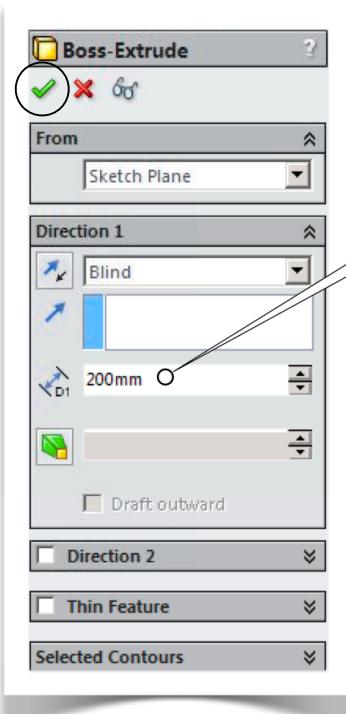
2.8-2 Start Up

[!] 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

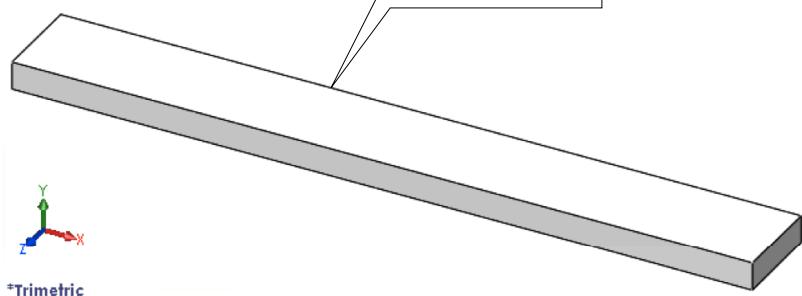


[!] On **Front** plane, draw a sketch like this.

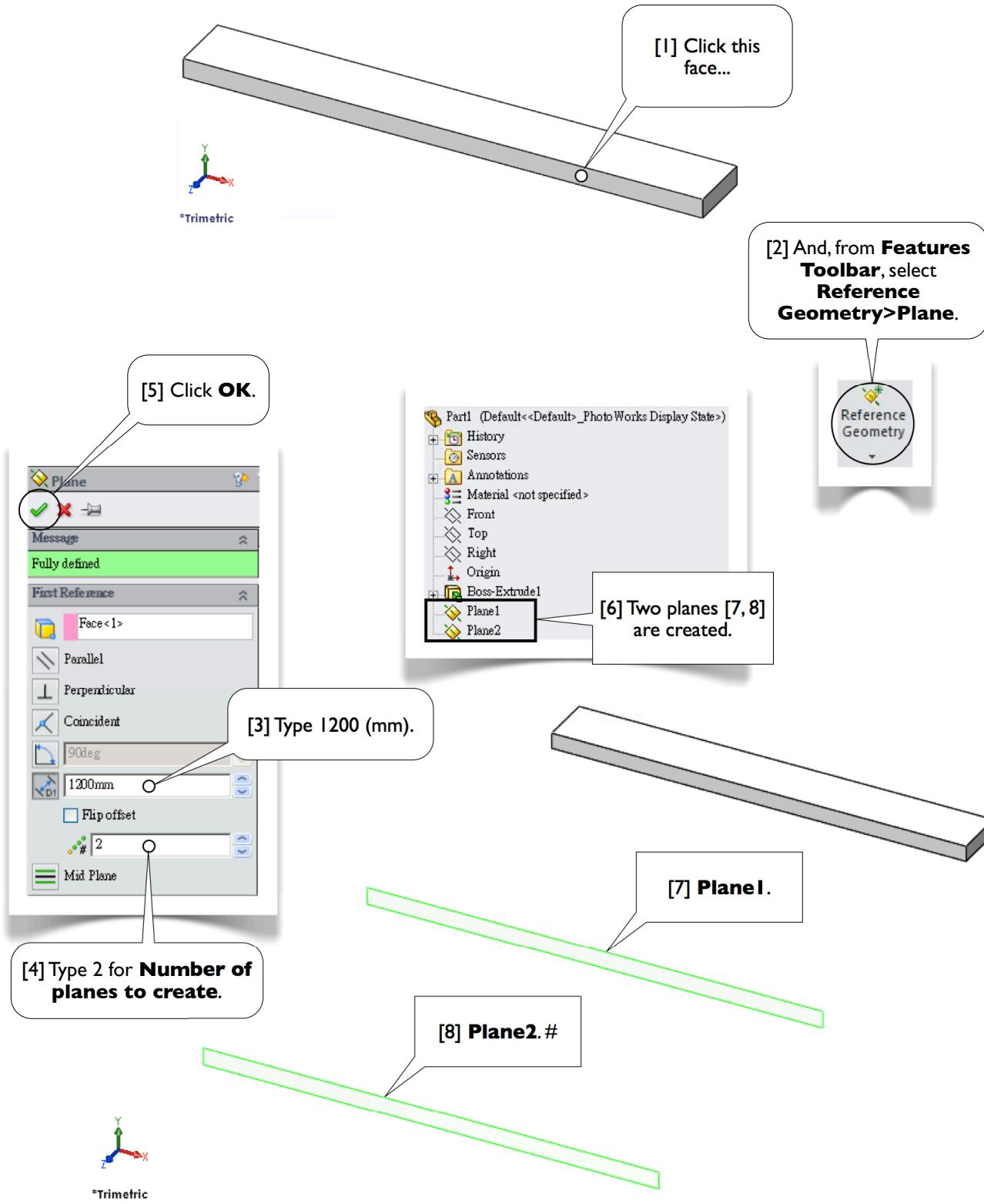


[2] **Extrude** the sketch 200 mm to create the transversal beam.

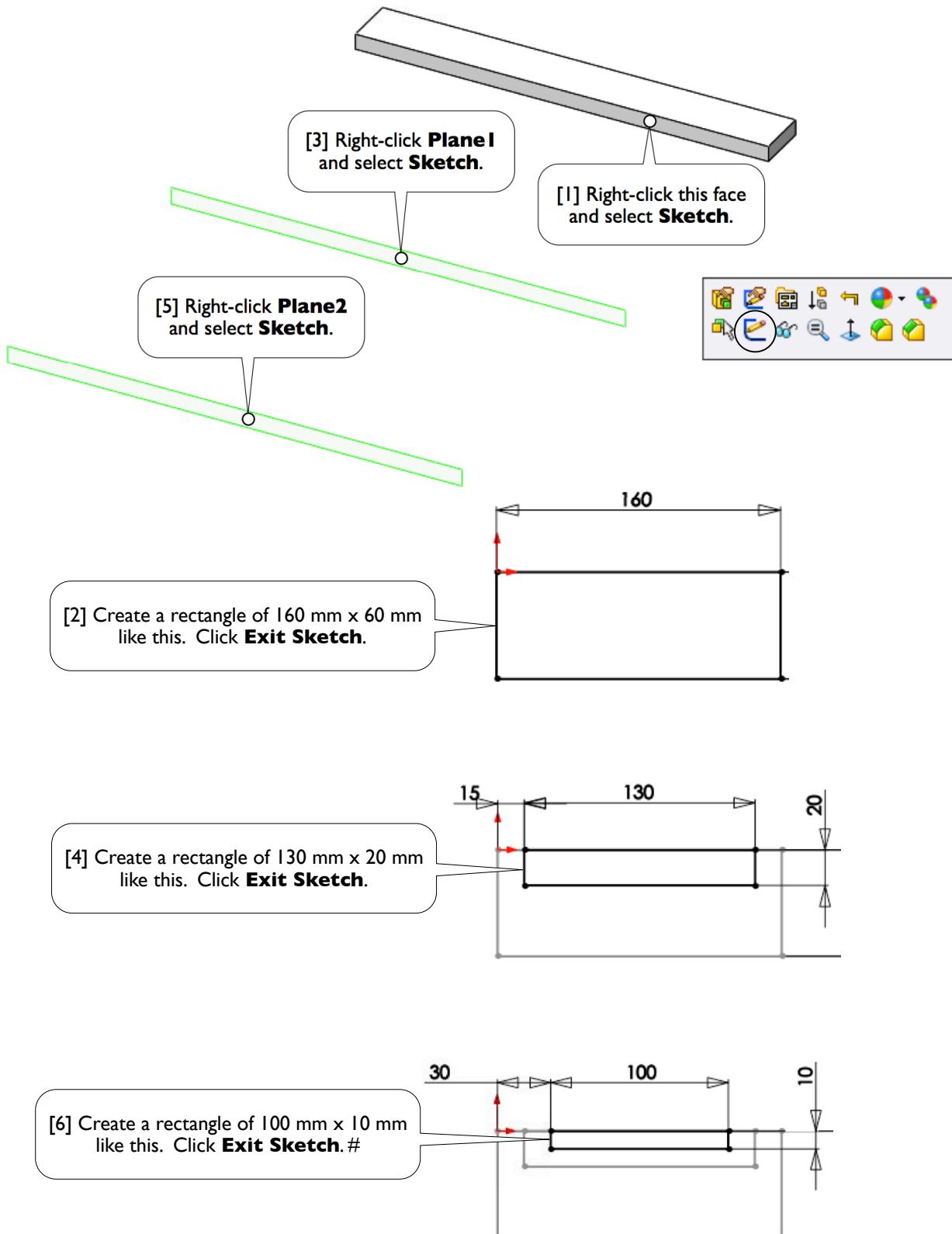
[3] The transversal beam.#



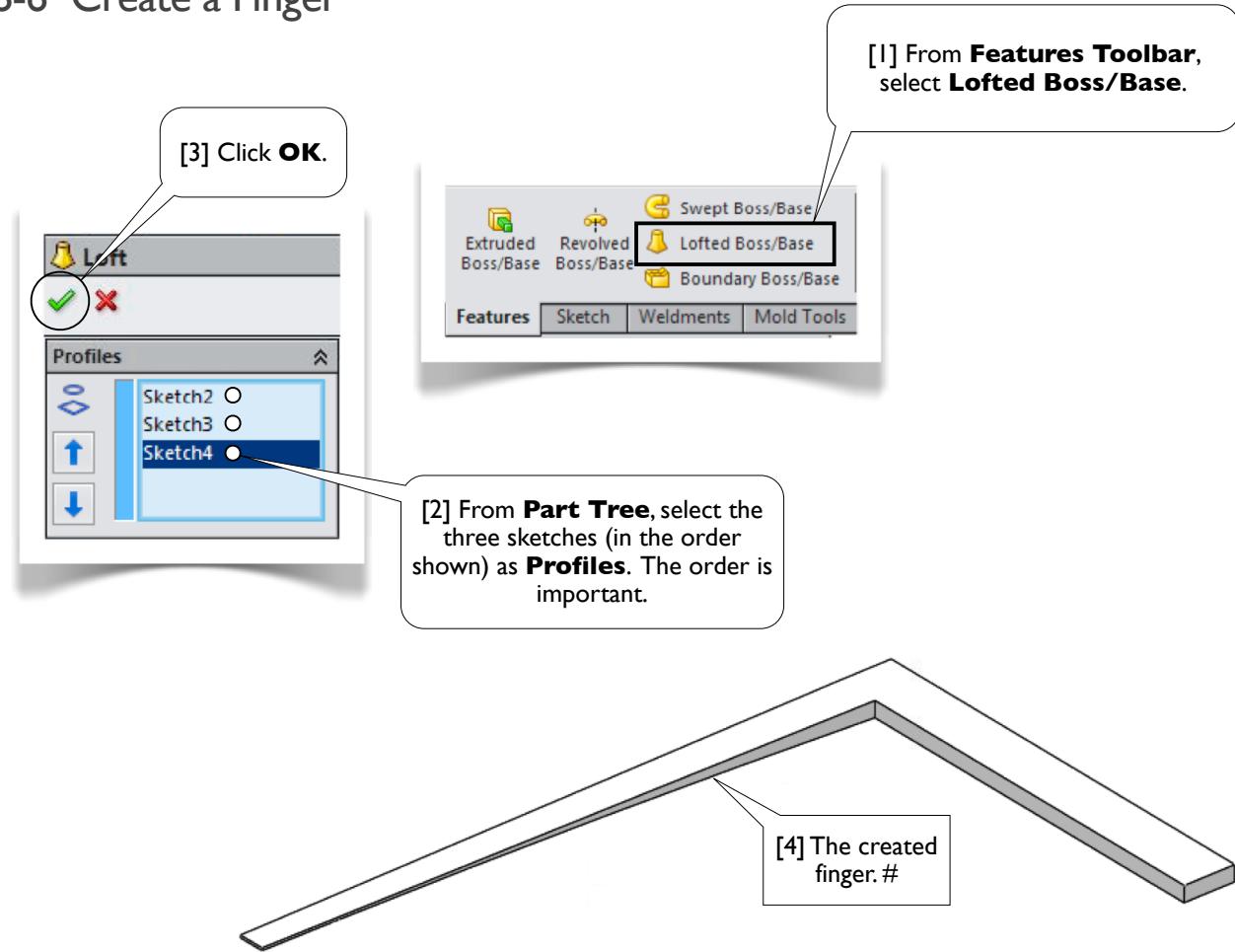
2.8-4 Create Two Planes



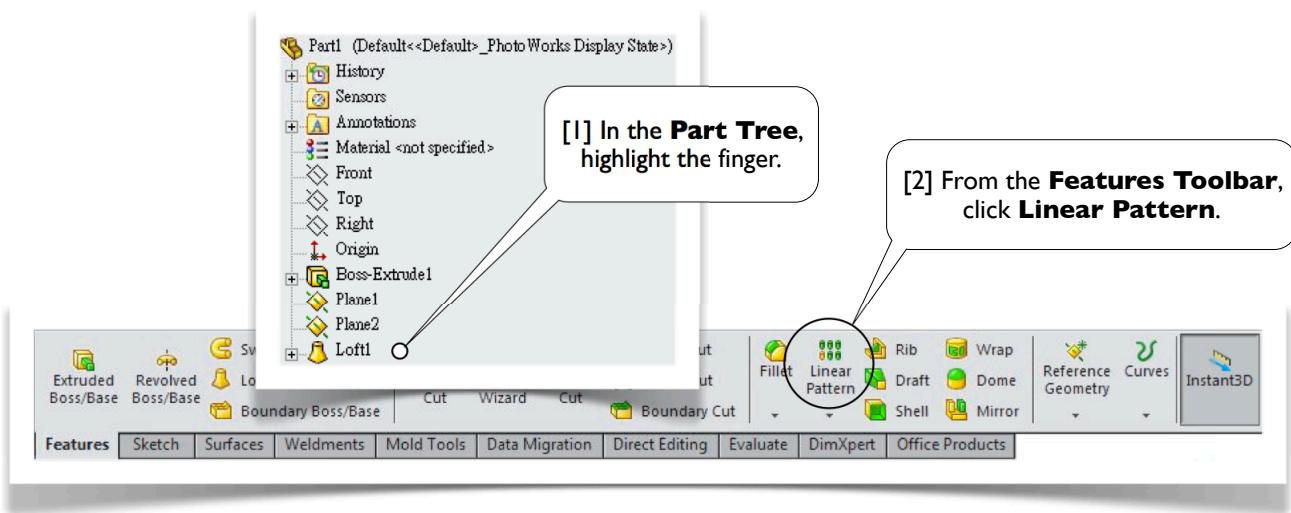
2.8-5 Sketch Three Profiles

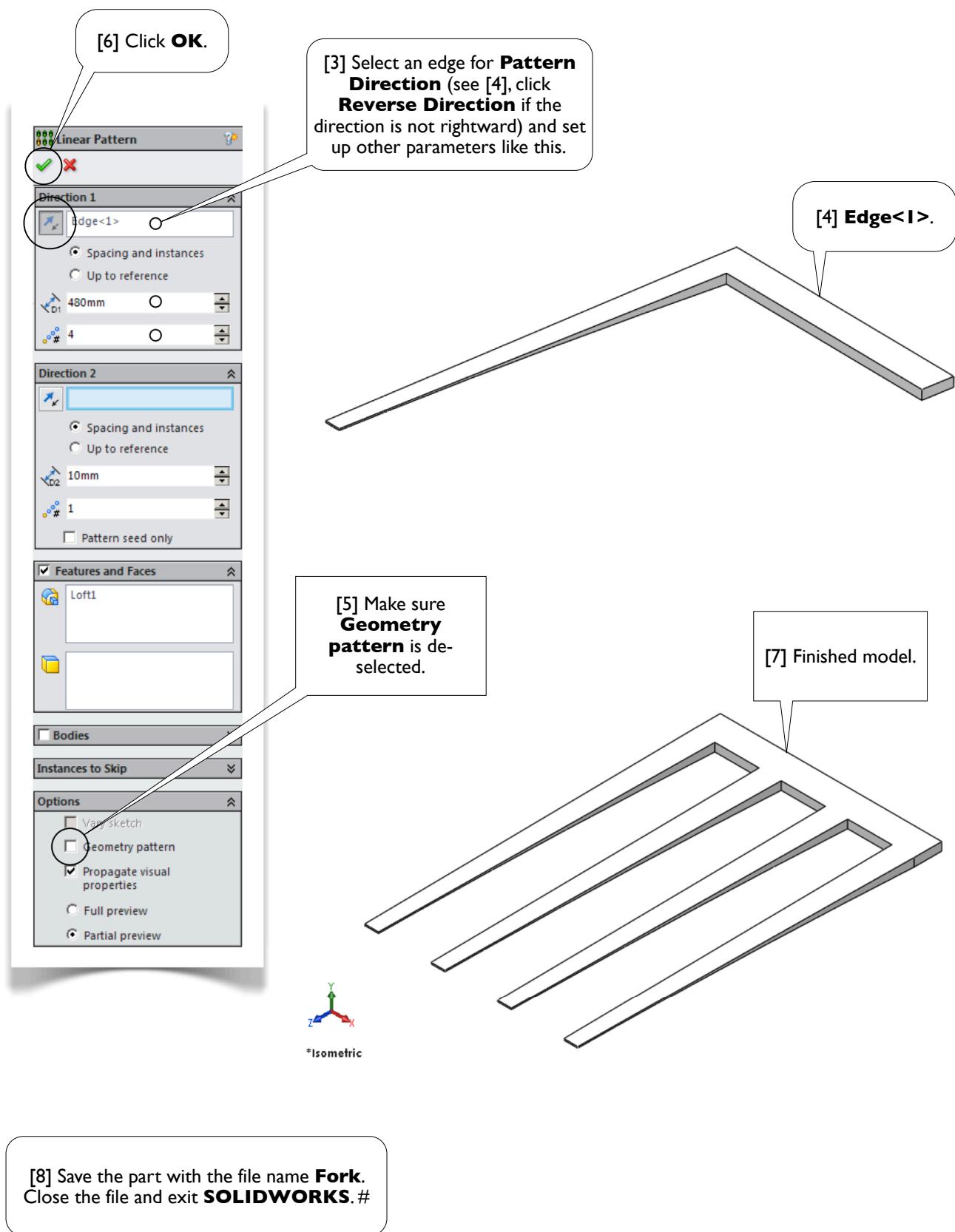


2.8-6 Create a Finger



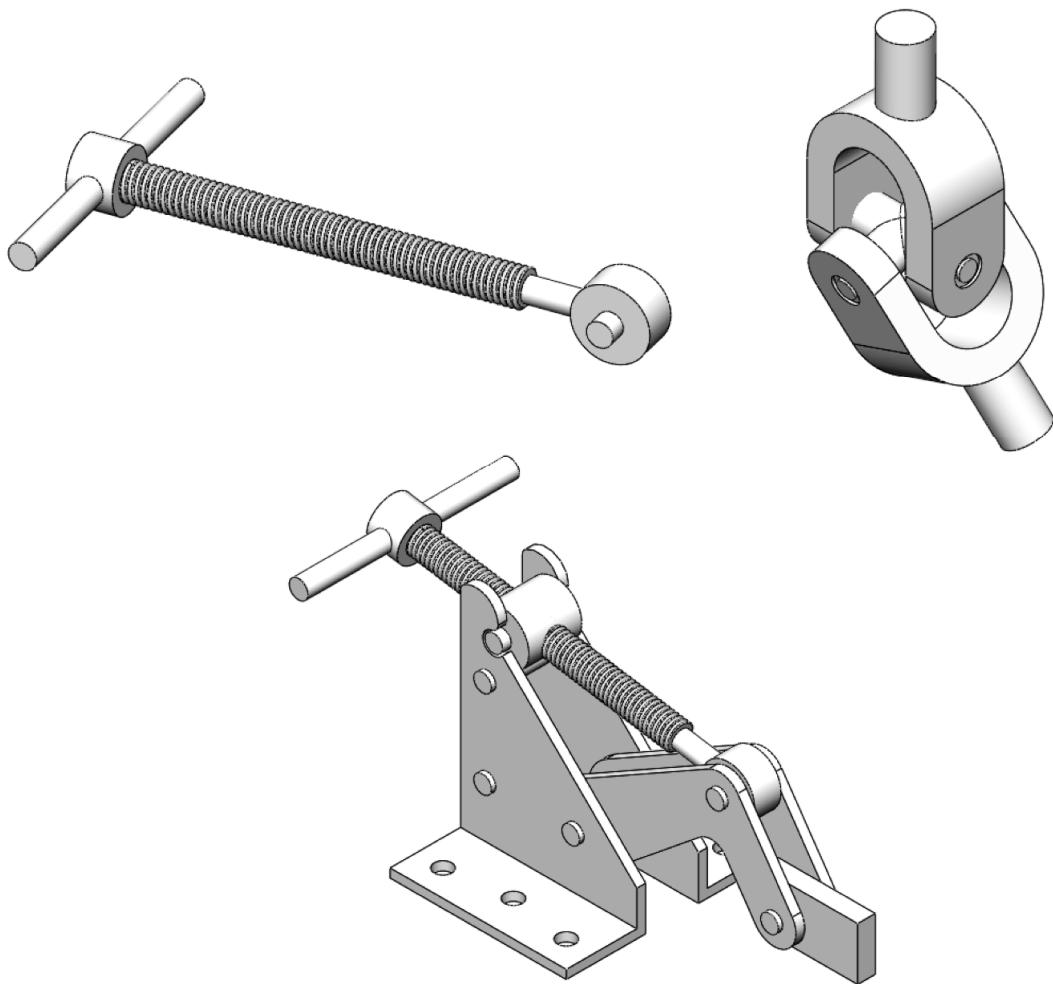
2.8-7 Create the Other Fingers





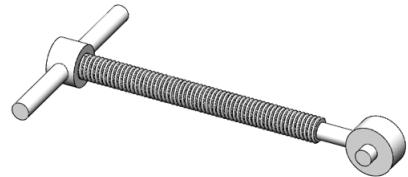
Chapter 3

Assembly Modeling



Section 3.I

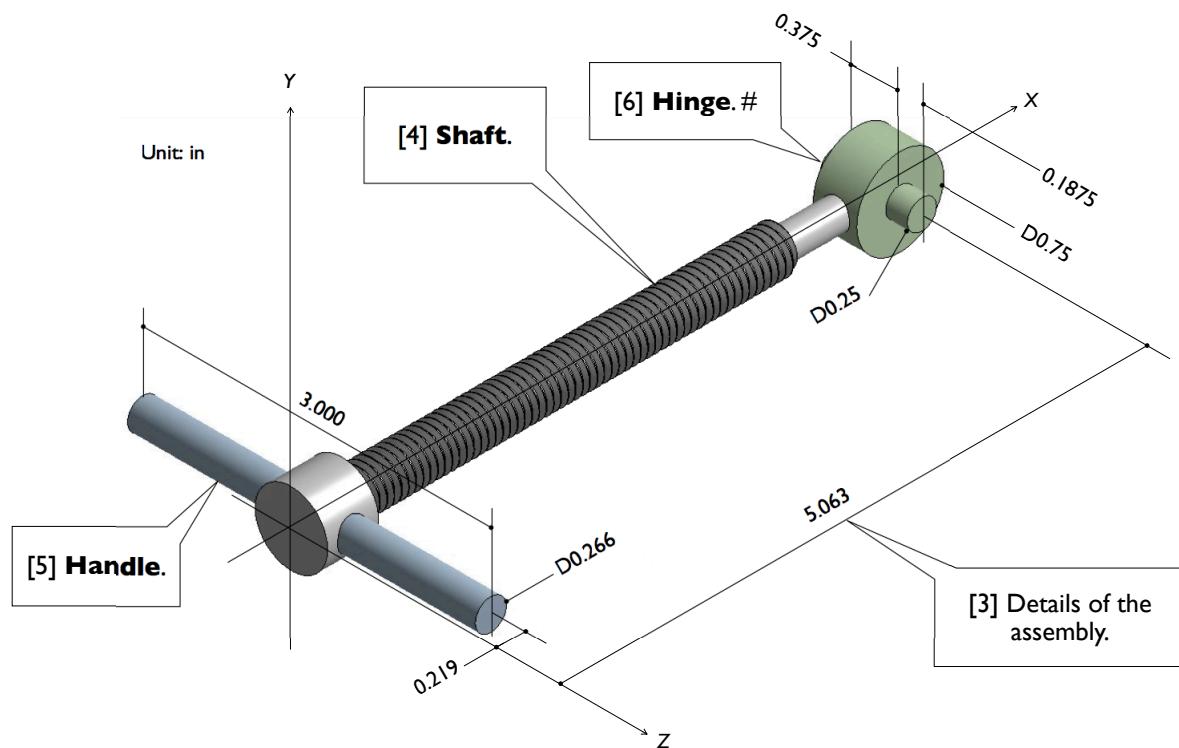
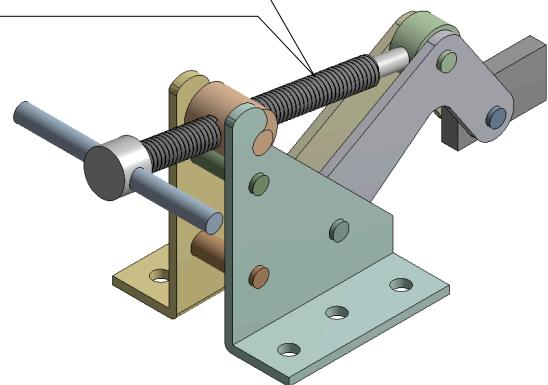
Shaft Assembly



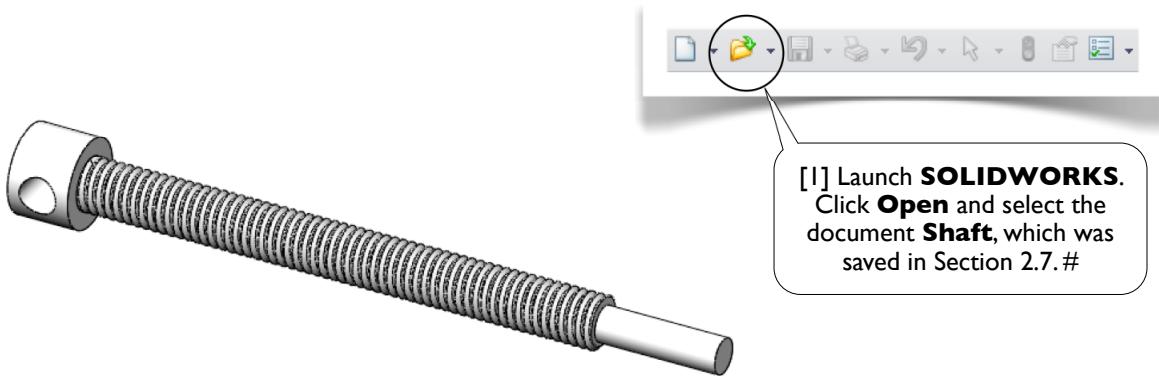
3.I-1 Introduction

[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**.

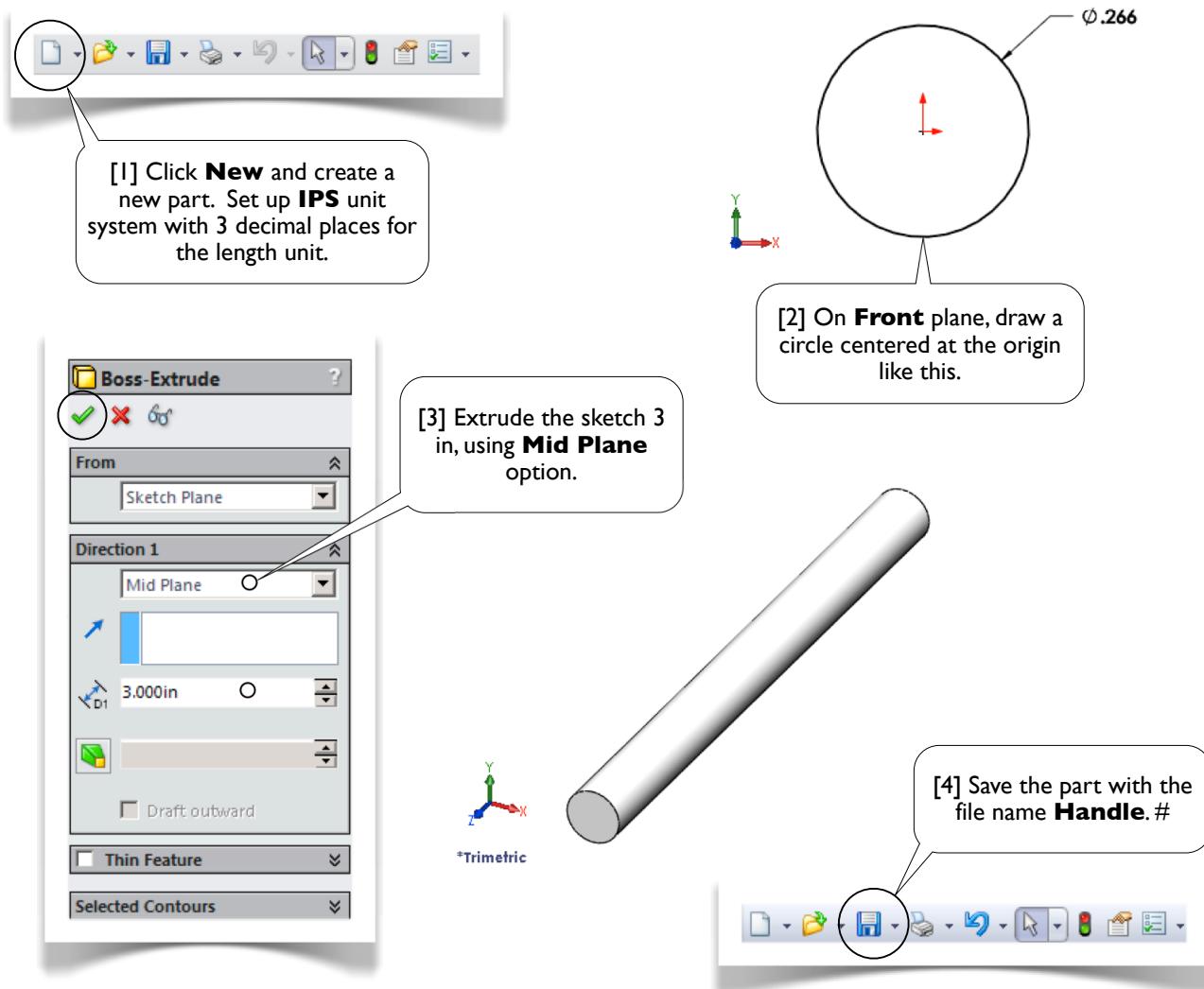
[2] The shaft assembly is a sub-assembly of the clamping mechanism.



3.I-2 Open Shaft



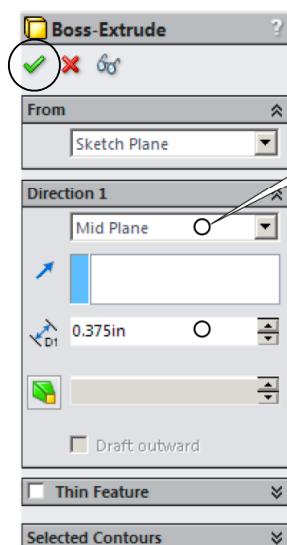
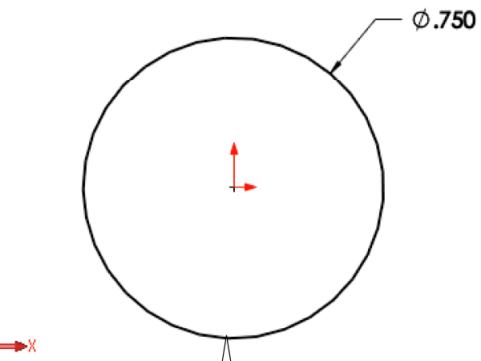
3.I-3 Create Handle



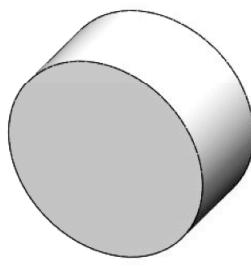
3.I-4 Create Hinge



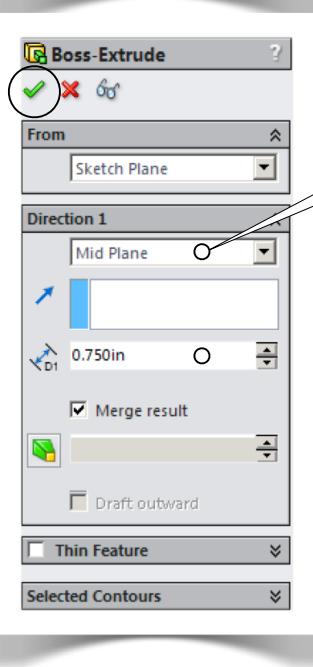
[1] Click **New** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.



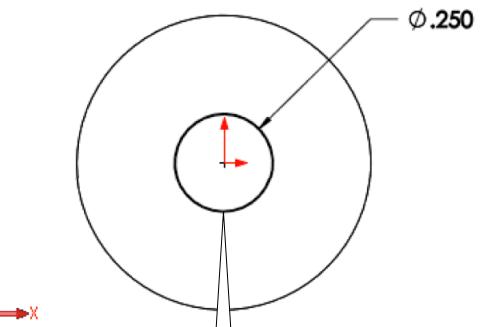
[3] Extrude the sketch 0.375 inches, using **Mid Plane** option.



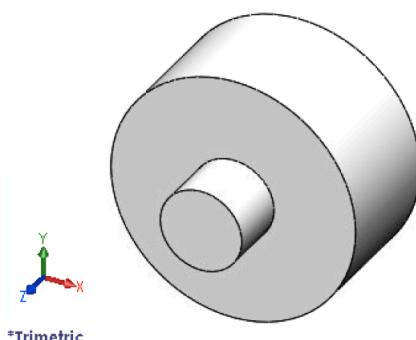
[2] On **Front** plane, draw a circle centered at the origin like this.

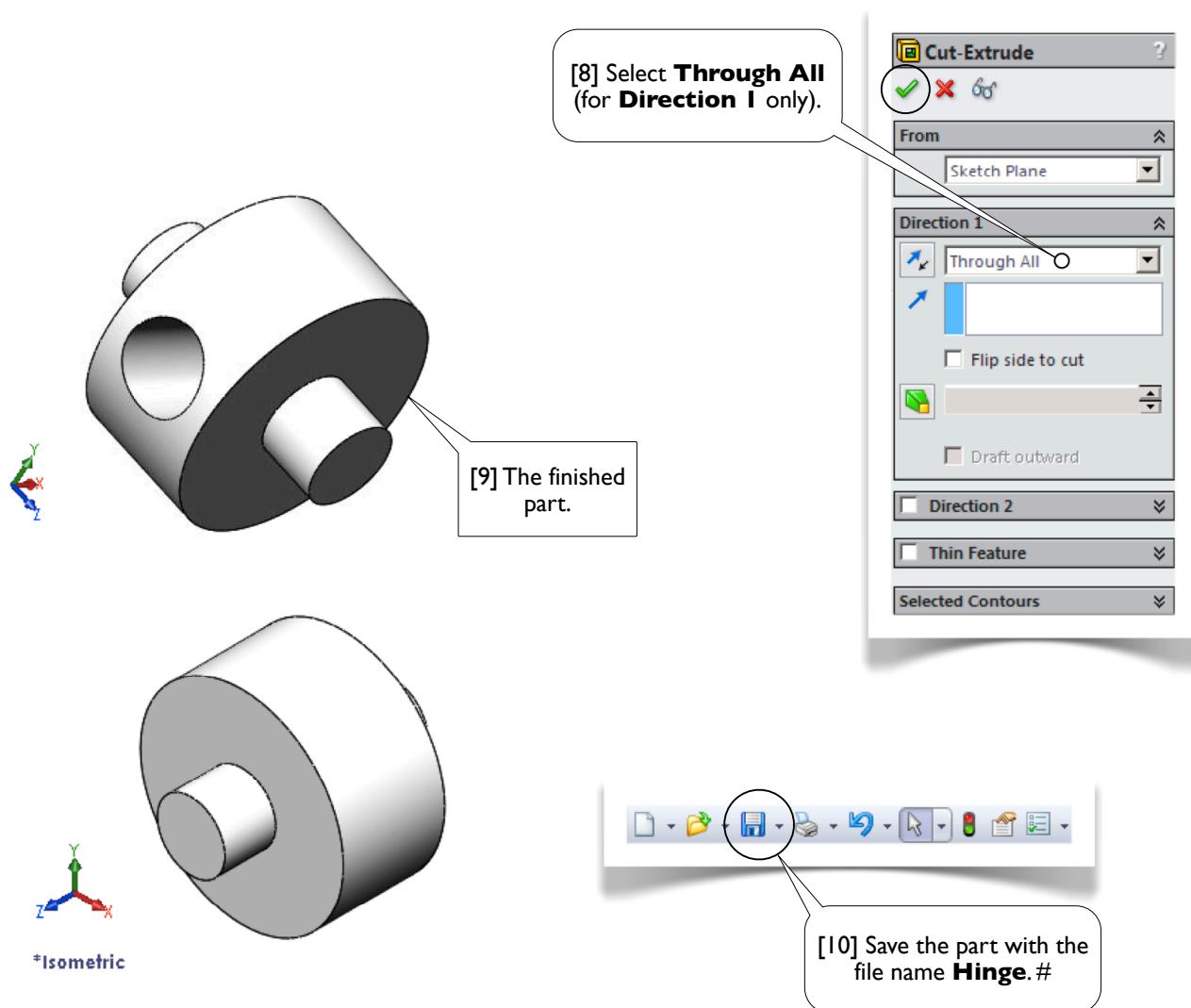
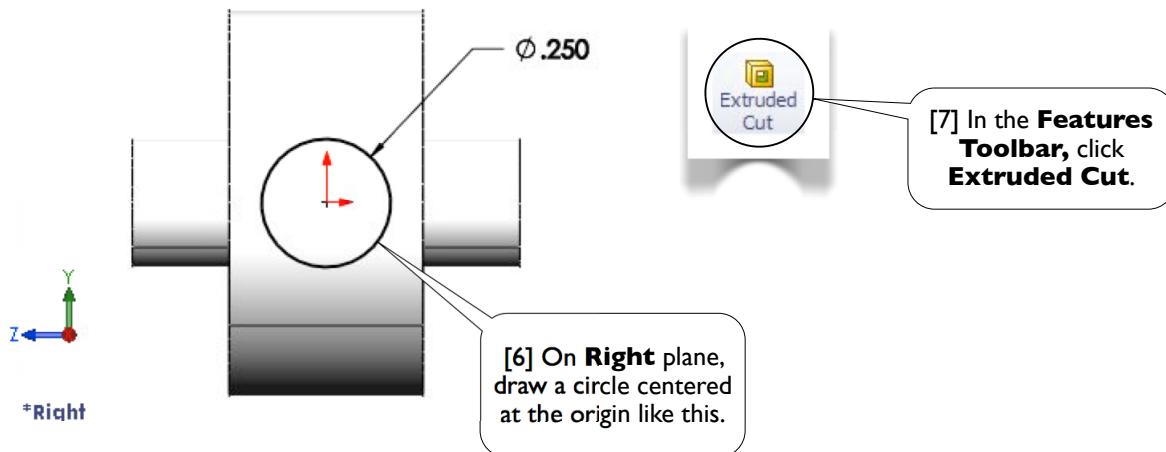


[5] Extrude the sketch 0.75 inches, using **Mid Plane** option.

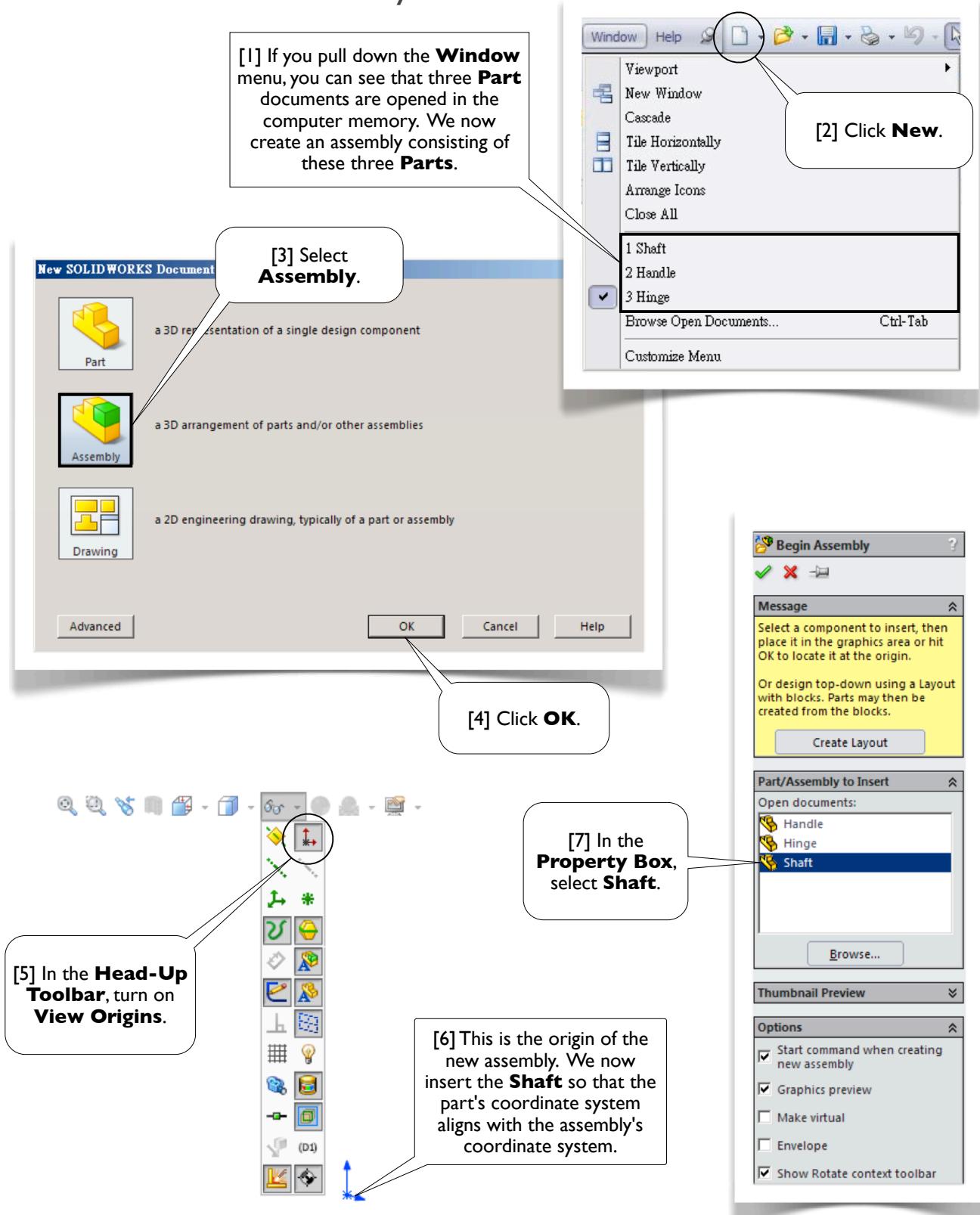


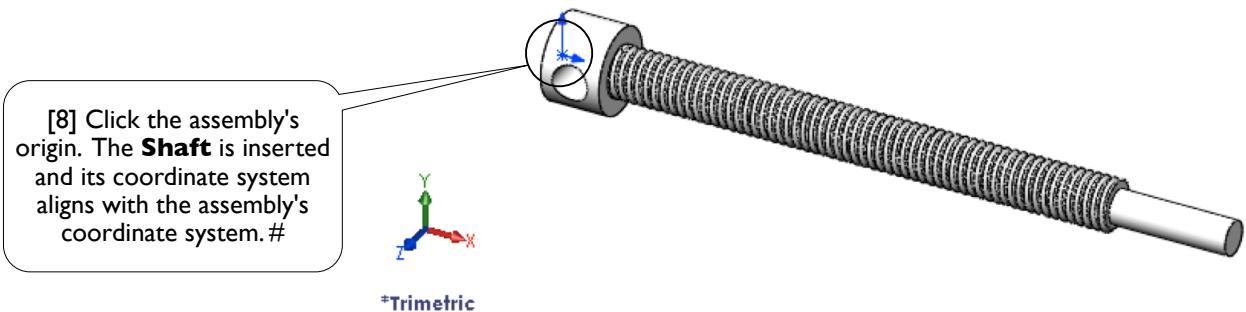
[4] On **Front** plane, draw a circle centered at the origin like this.



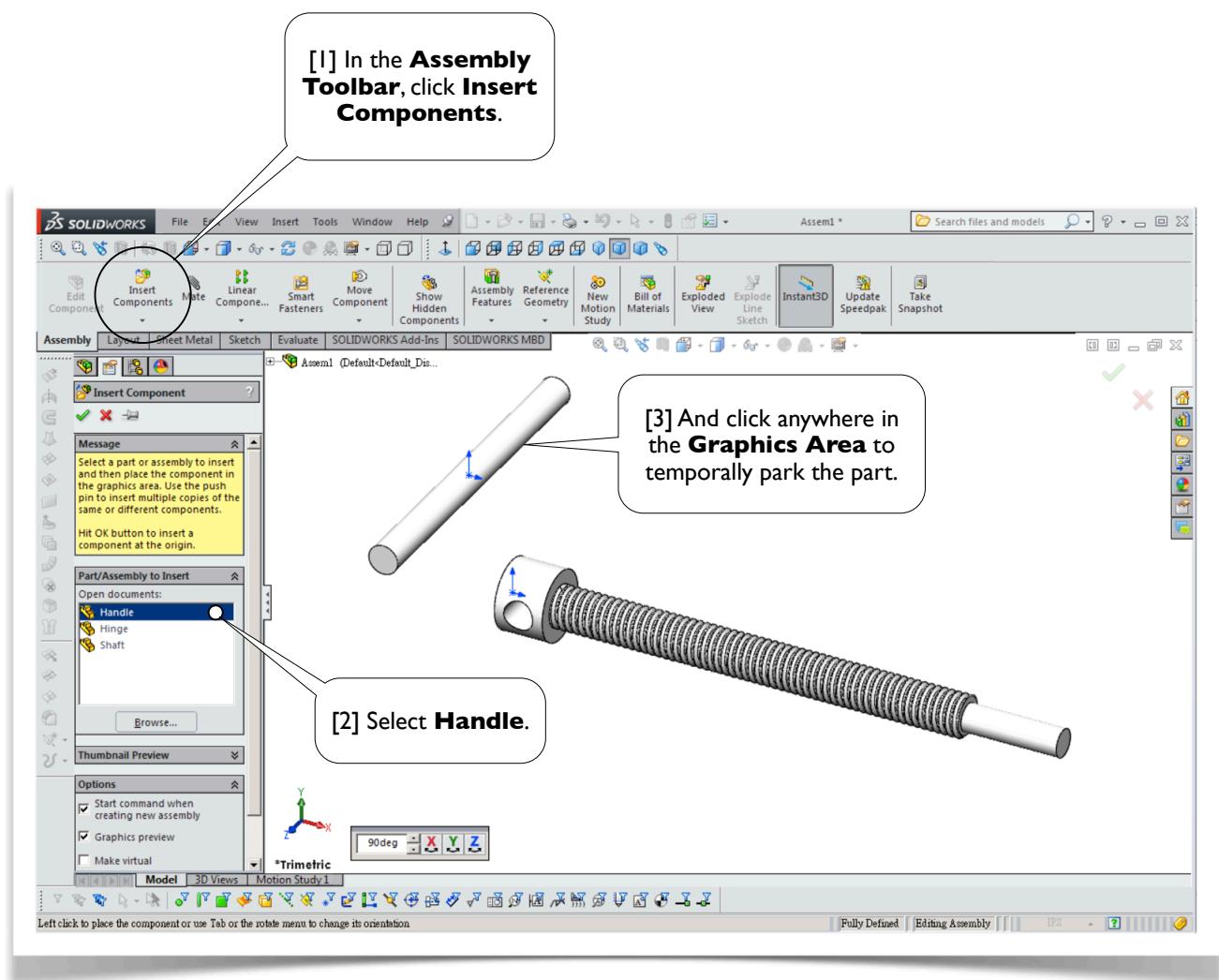


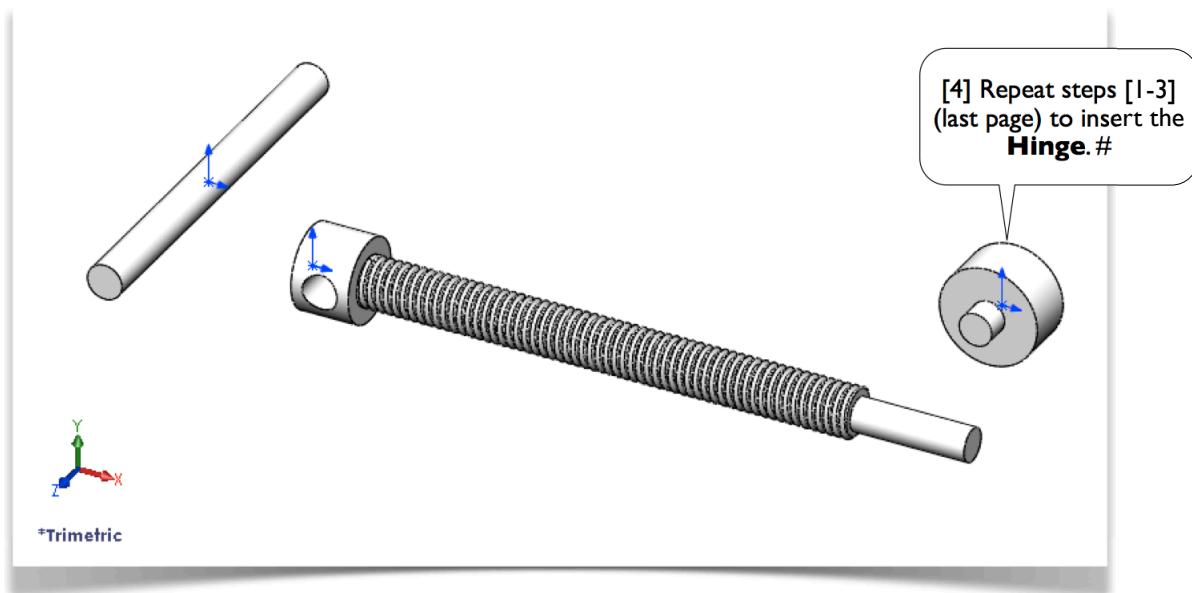
3.1-5 Create a New Assembly



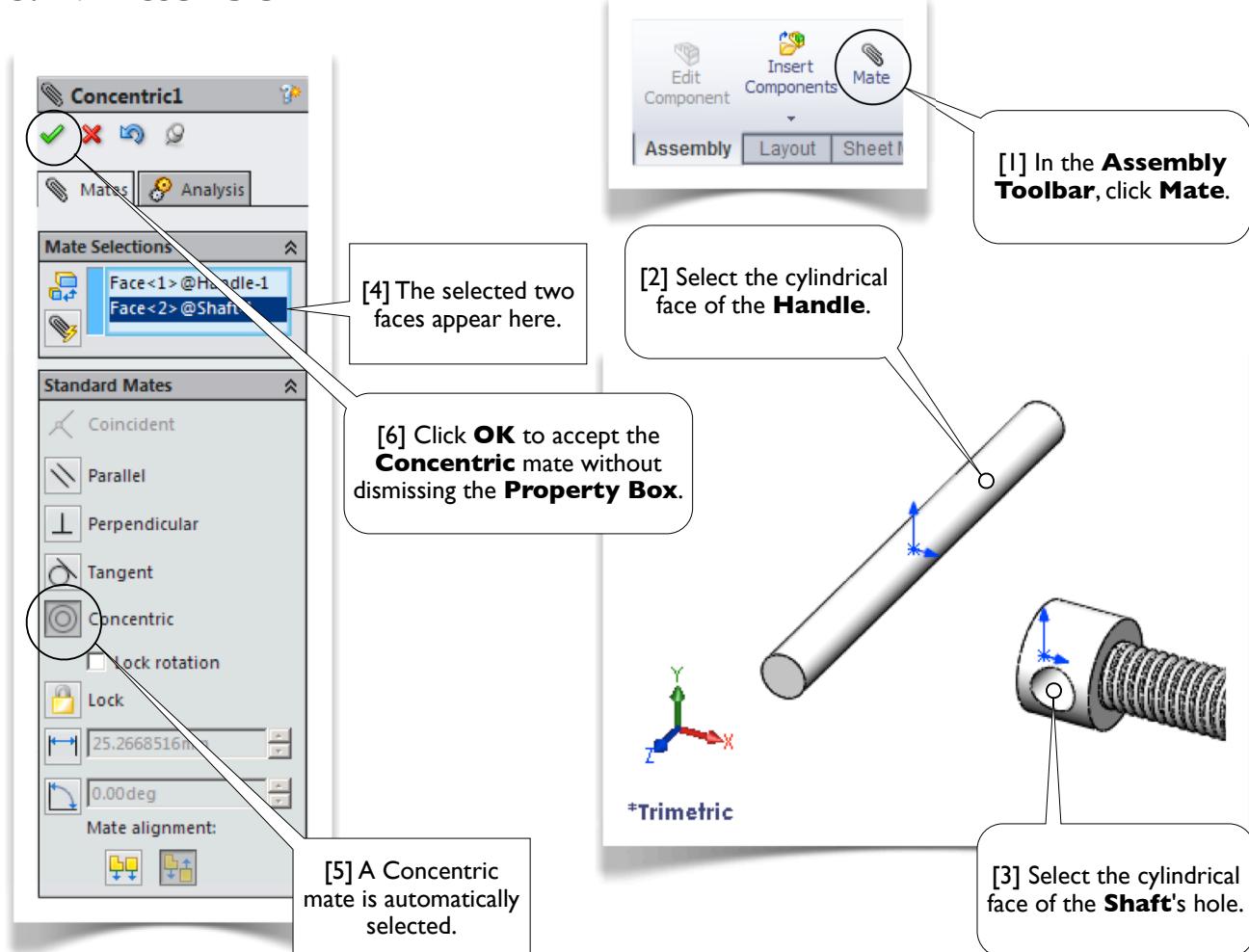


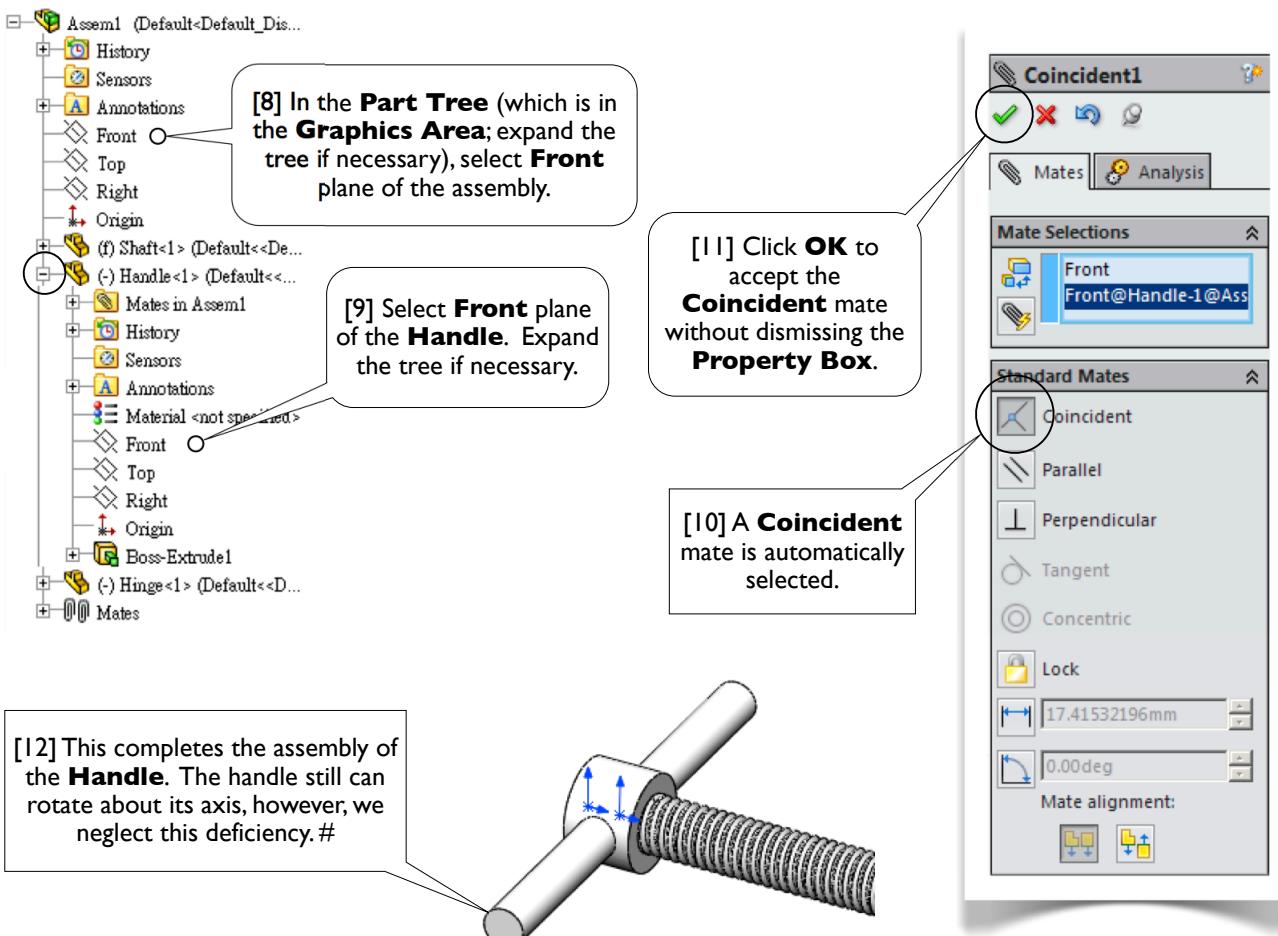
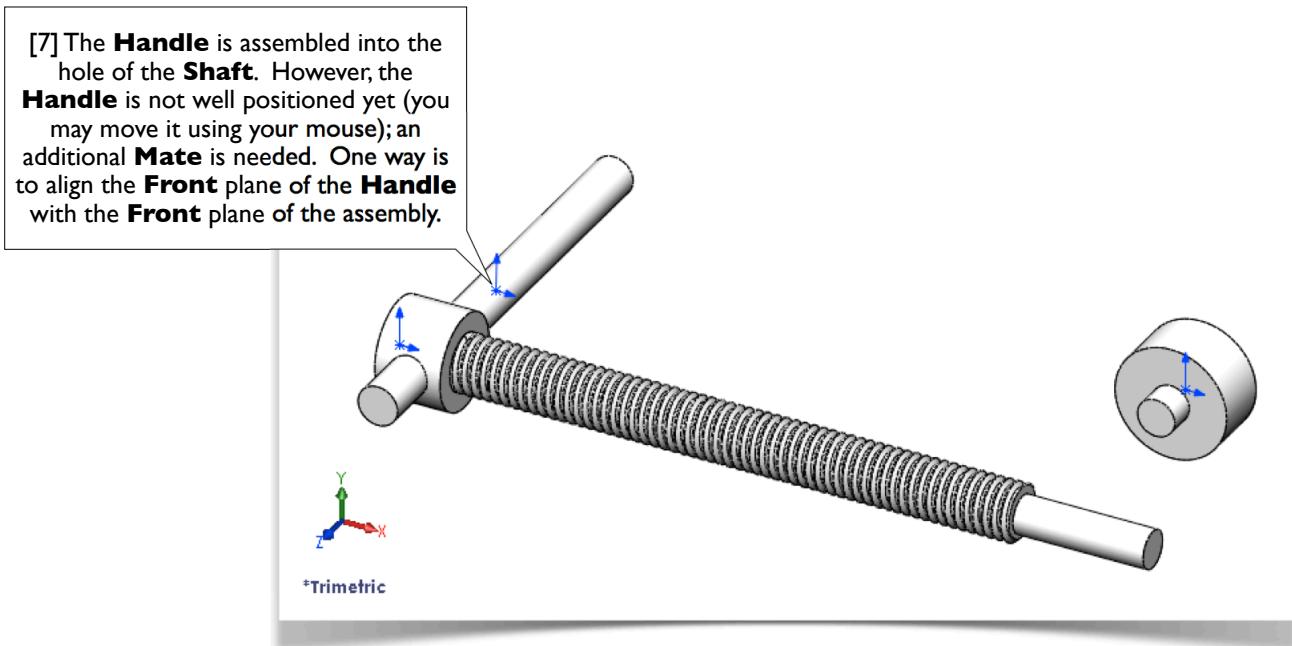
3.I-6 Insert the Other Components



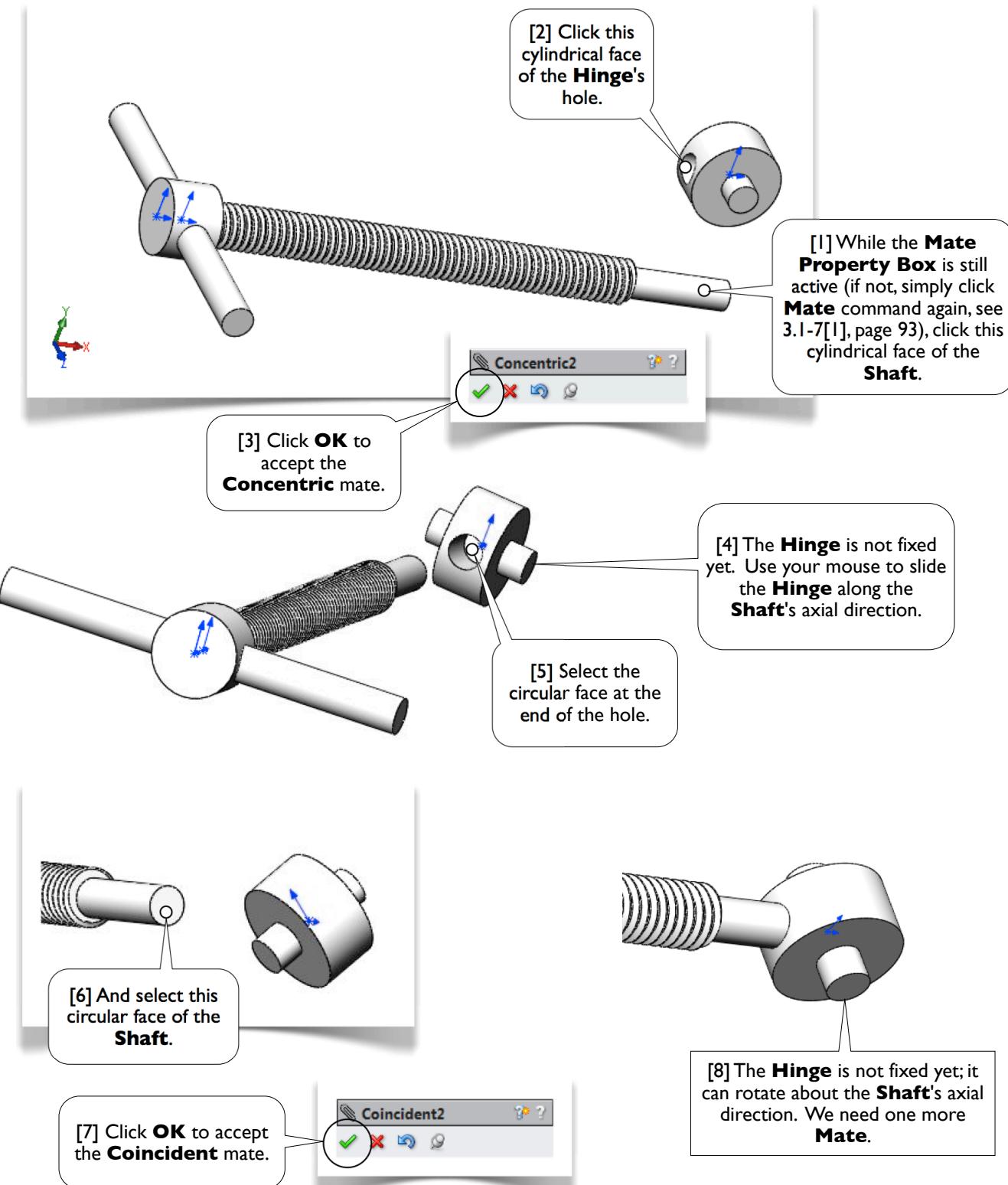


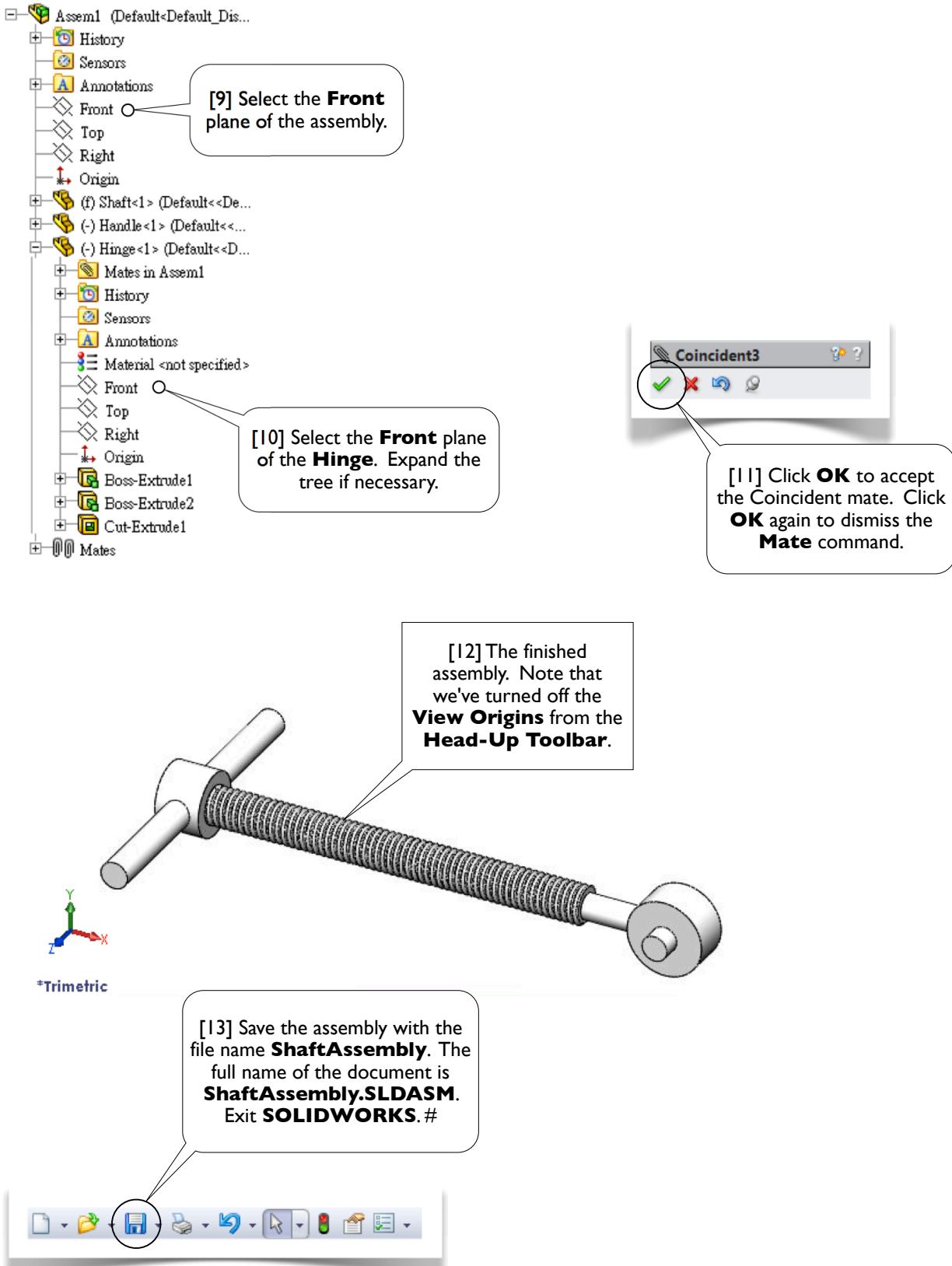
3.I-7 Assemble Handle





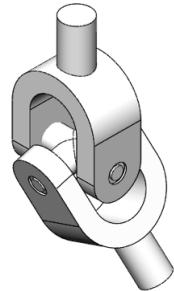
3.I-8 Assemble Hinge





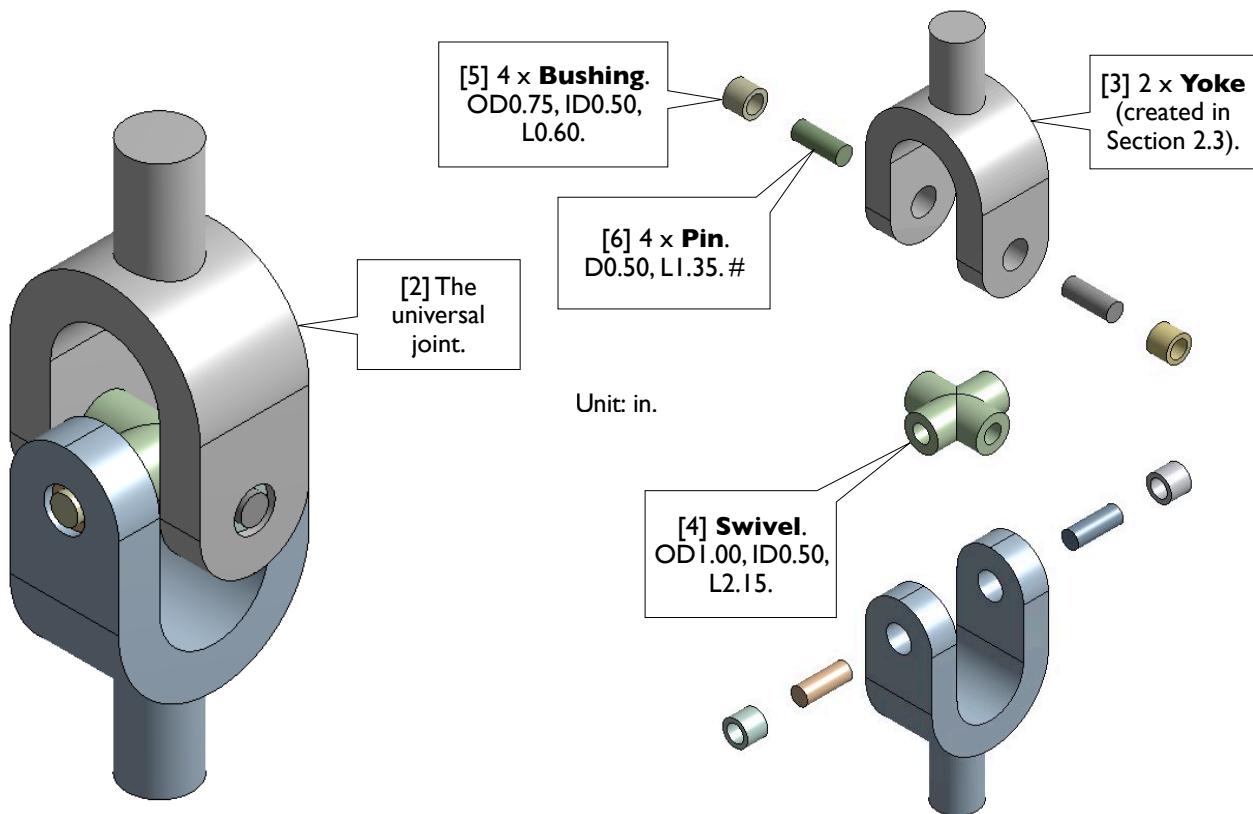
Section 3.2

Universal Joint

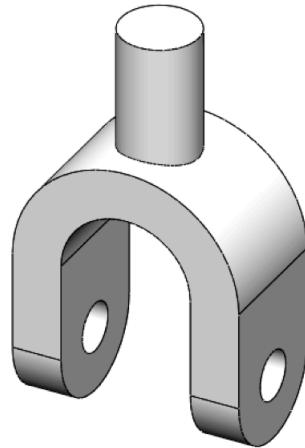
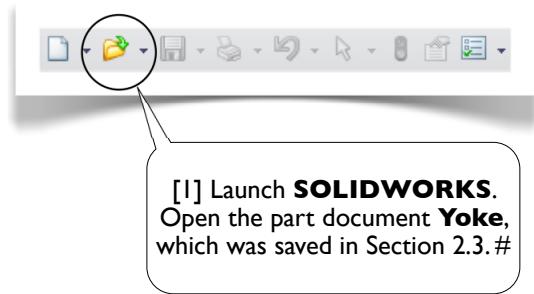


3.2-1 Introduction

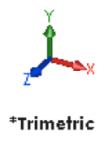
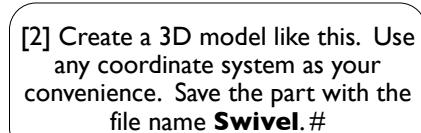
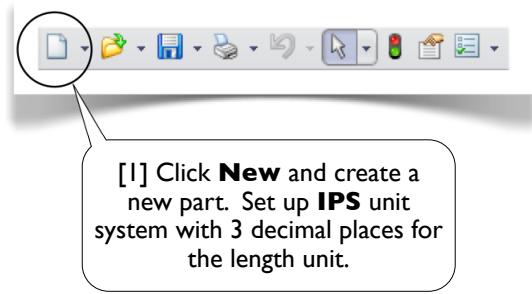
[1] In this exercise, we'll create a universal joint [2]. The assembly consists of four kinds of parts [3-6], of which the **Yoke** [3] was created in Section 2.3.



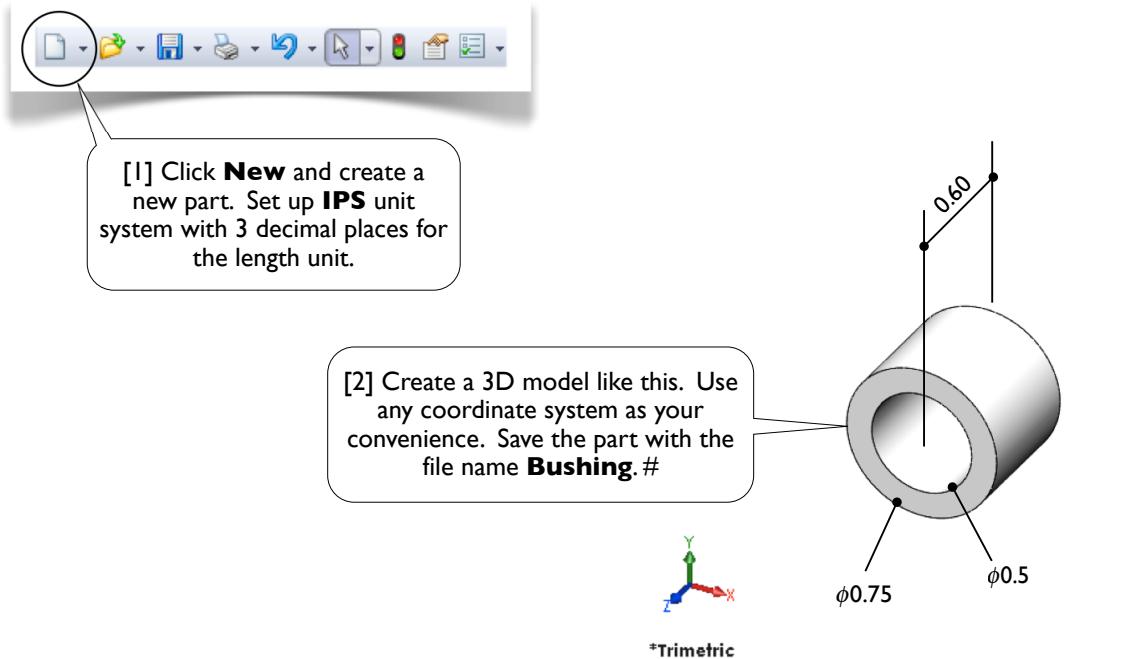
3.2-2 Open Yoke



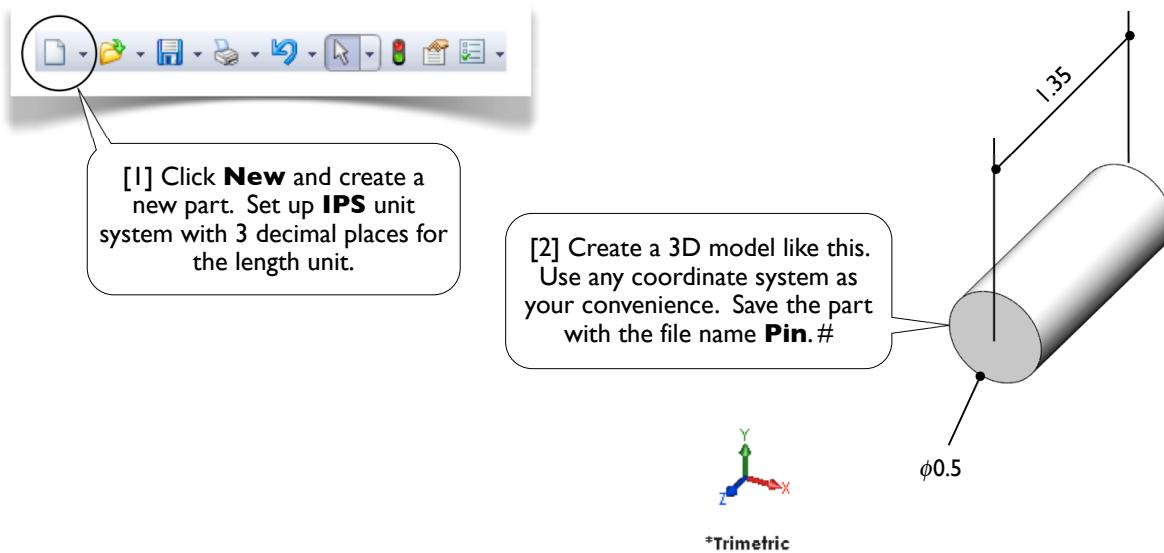
3.2-3 Create **Swivel**



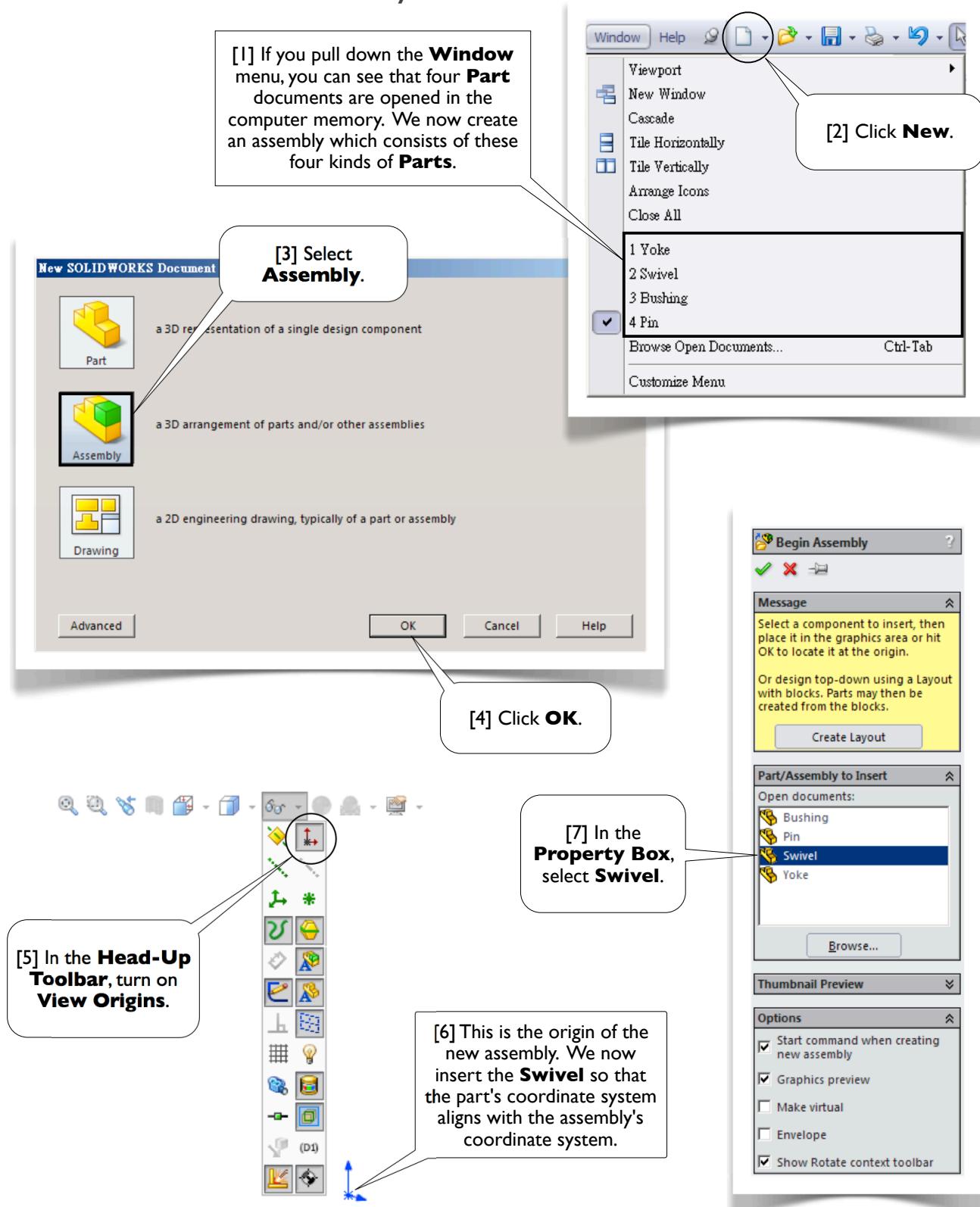
3.2-4 Create Bushing

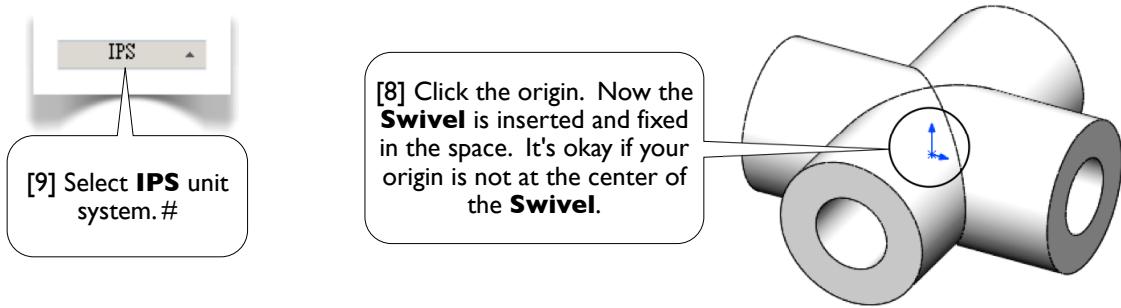


3.2-5 Create Pin

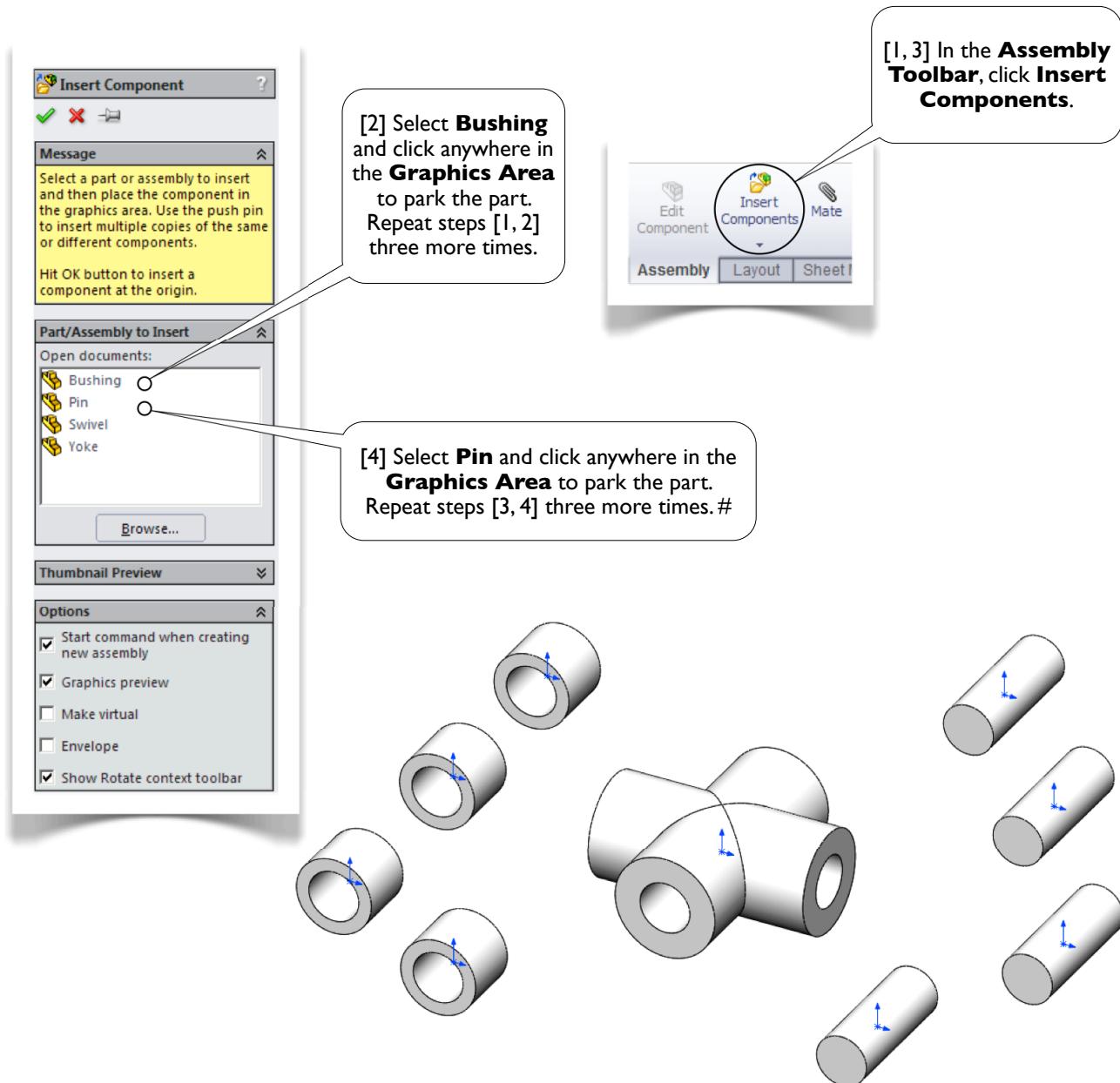


3.2-6 Create a New Assembly

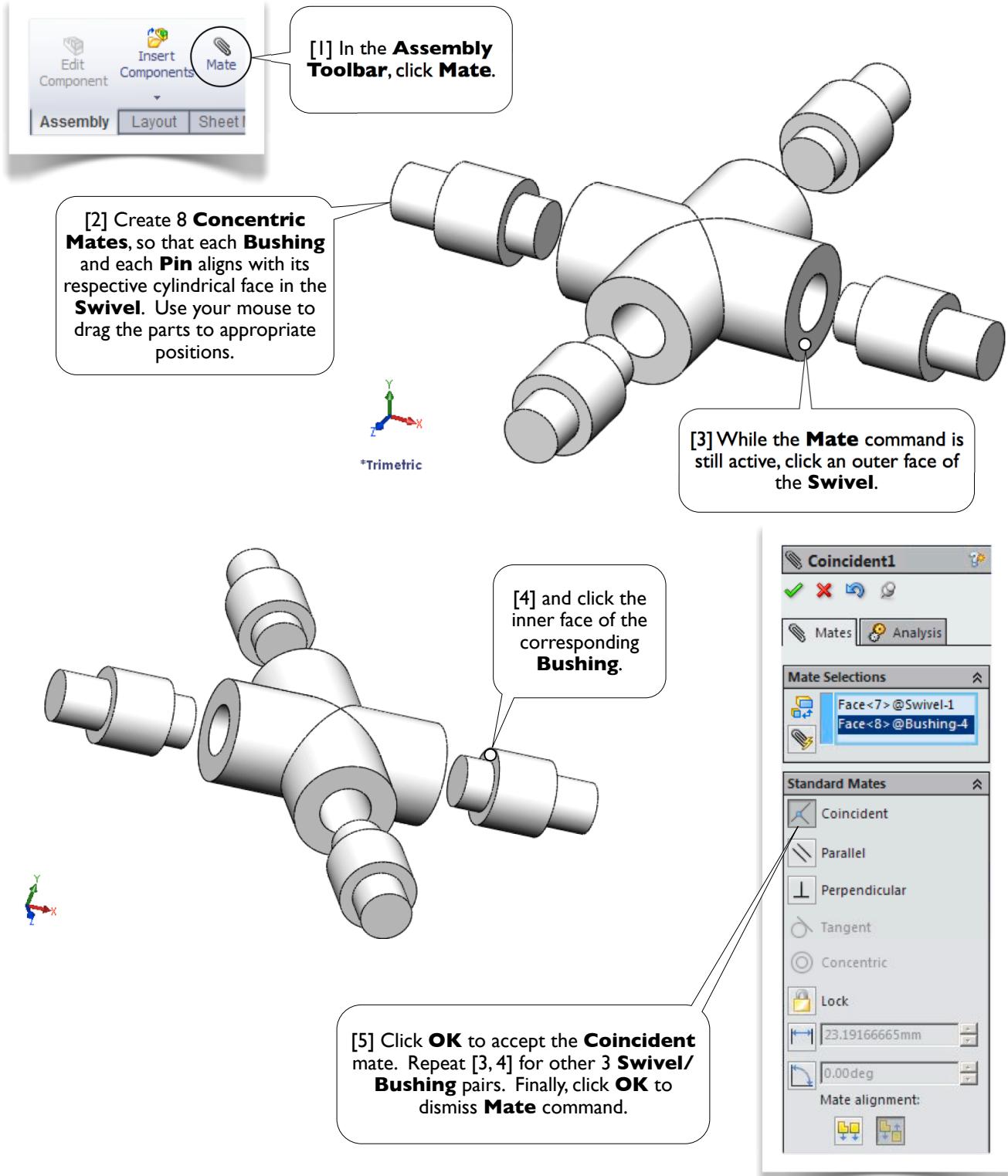


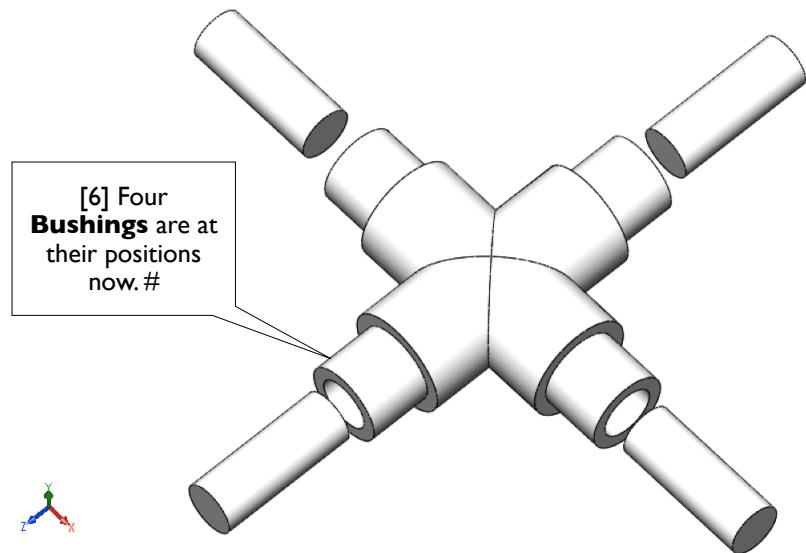


3.2-7 Insert Bushings and Pins

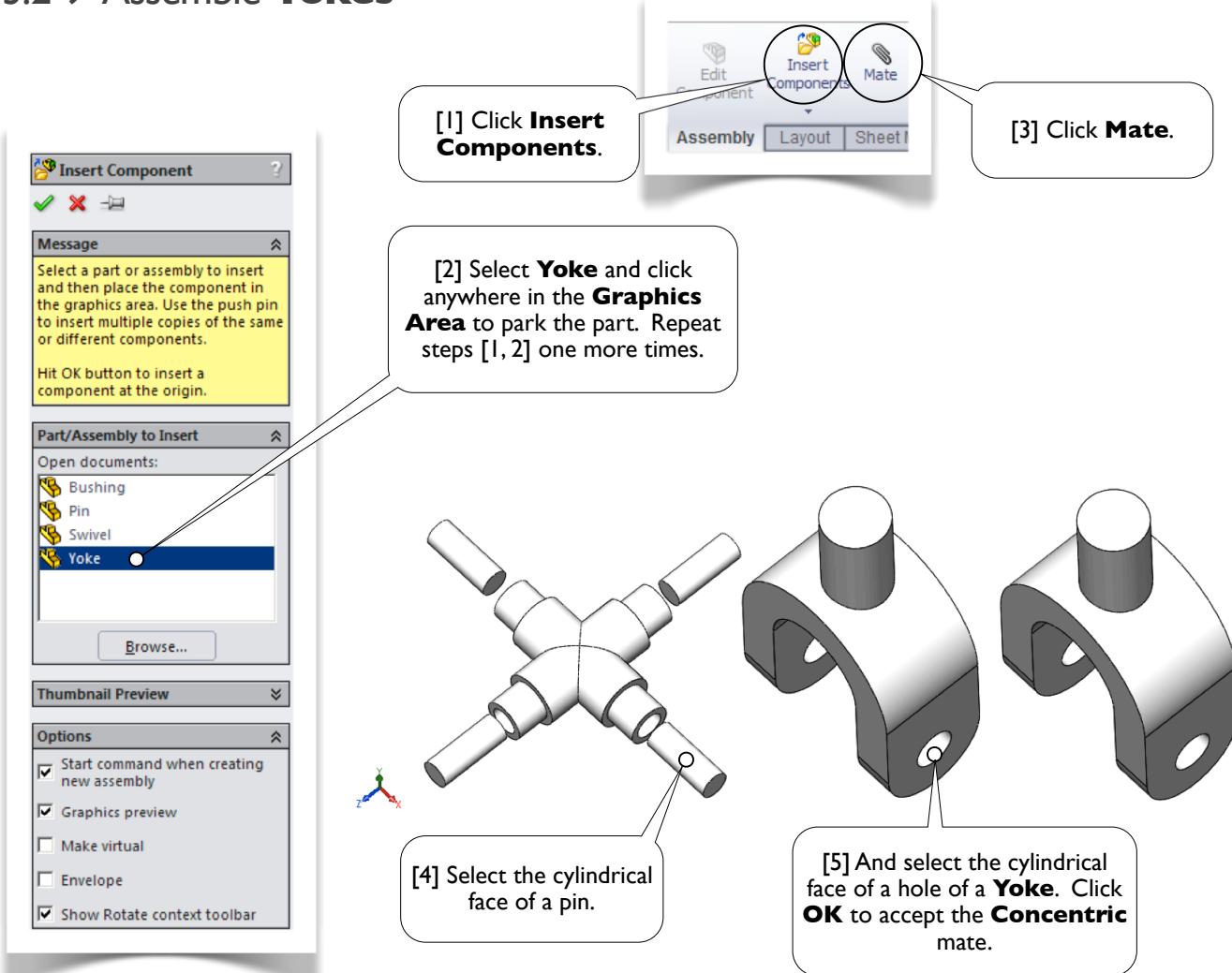


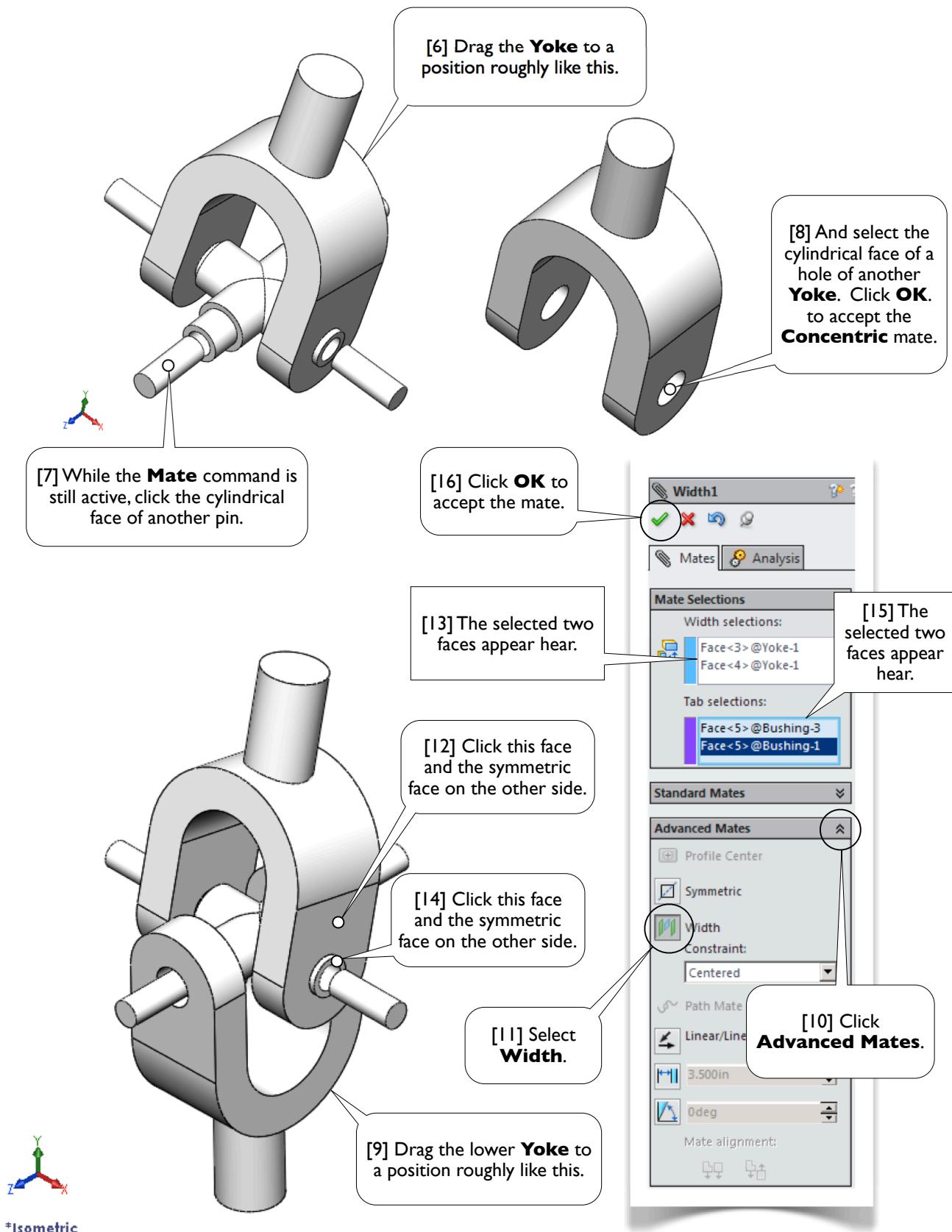
3.2-8 Assemble Bushings and Pins

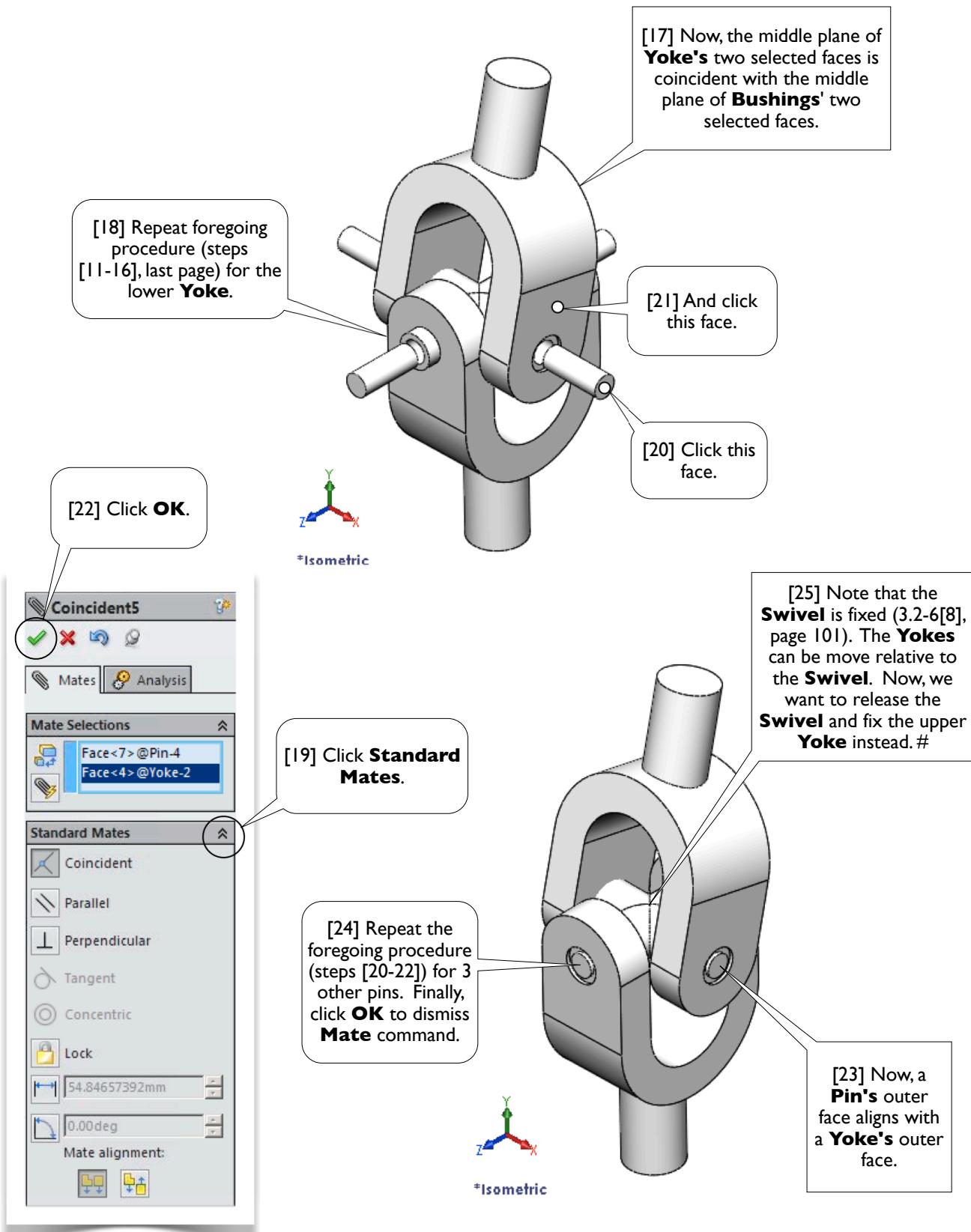




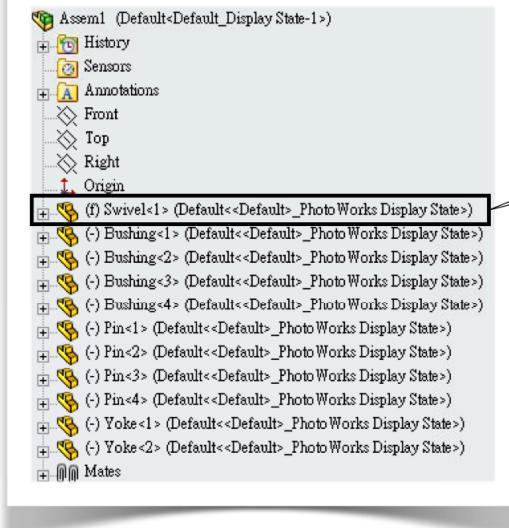
3.2-9 Assemble Yokes



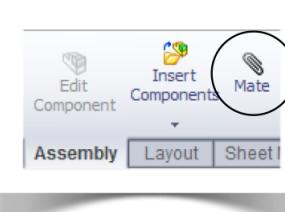




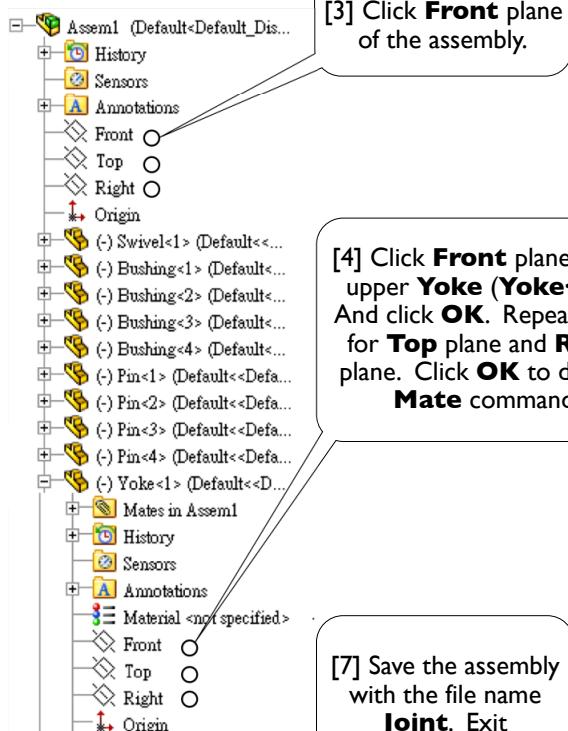
3.2-10 Fix Upper Yoke



[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].

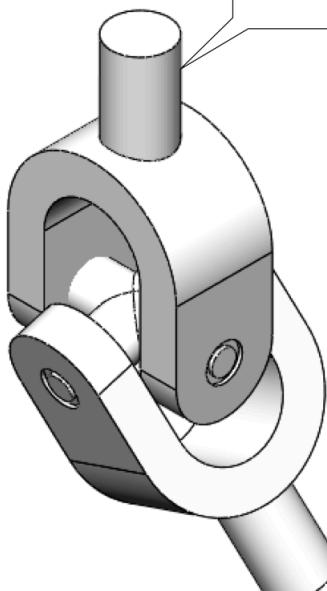


[2] Click **Mate**.



[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.

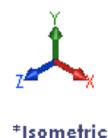


[5] Now, the upper **Yoke** is fixed in the space.

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

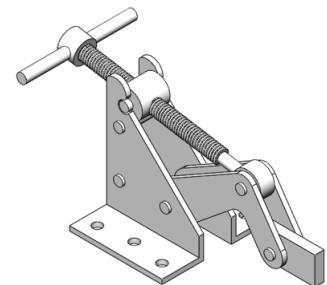


[6] Use your mouse to move the lower **Yoke**.



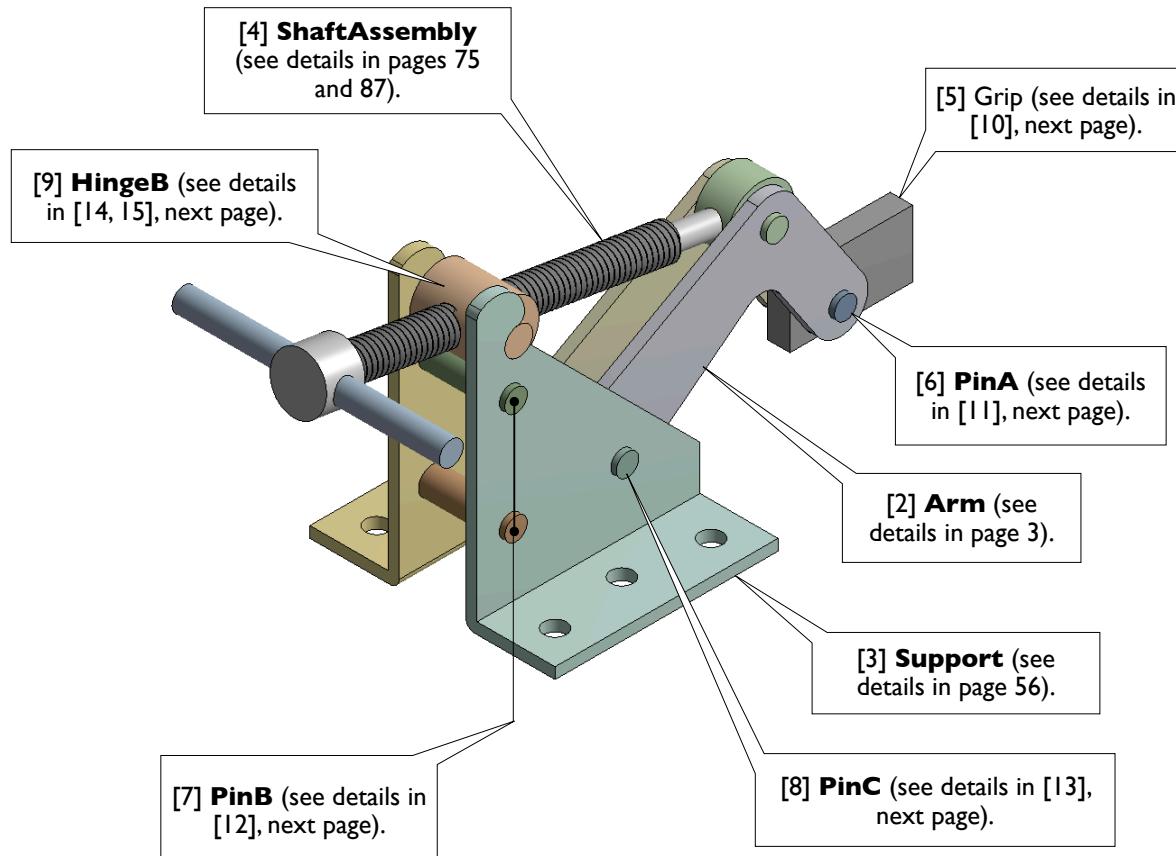
Section 3.3

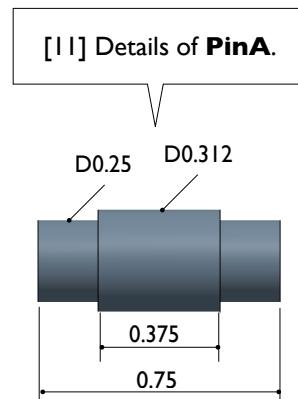
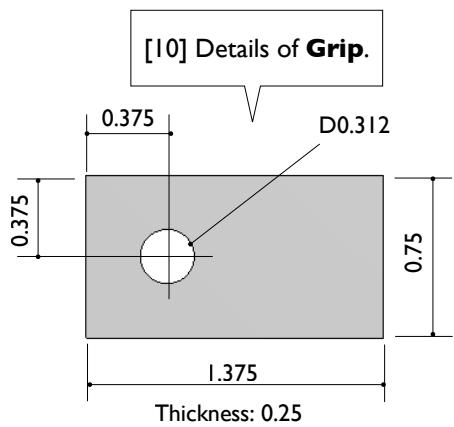
Clamp



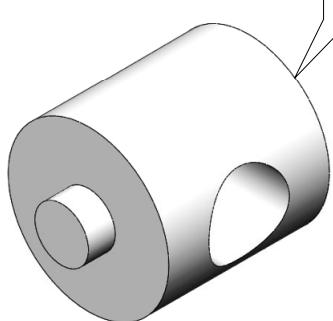
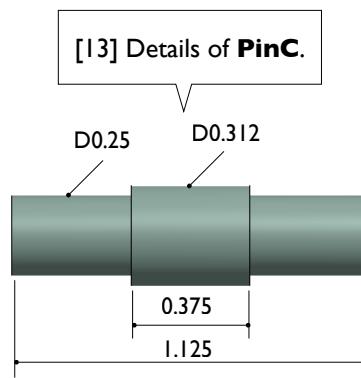
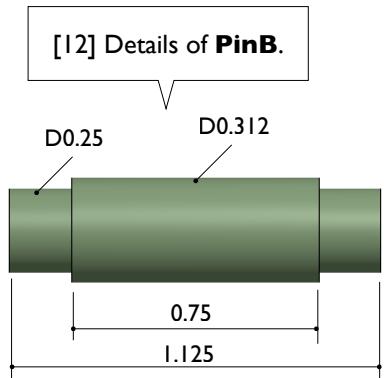
3.3-1 Introduction

[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].

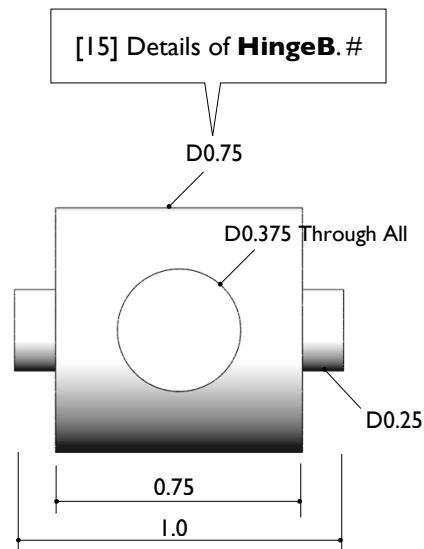




Unit: in.



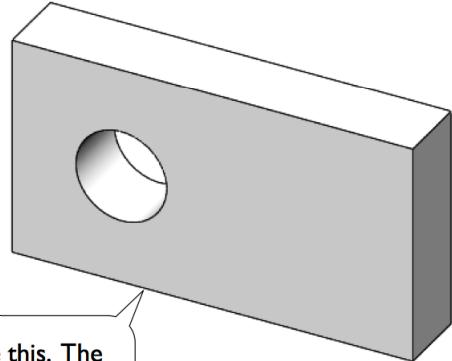
[14] **HingeB**. The internal threads are neglected here.



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.

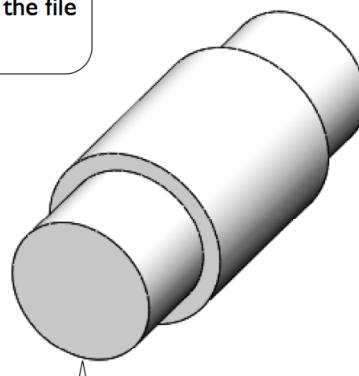


[2] Create a 3D model like this. The details are shown in 3.3-1[10] (last page). Use any coordinate system as your convenience. Save the part with the file name **Grip.#**

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.

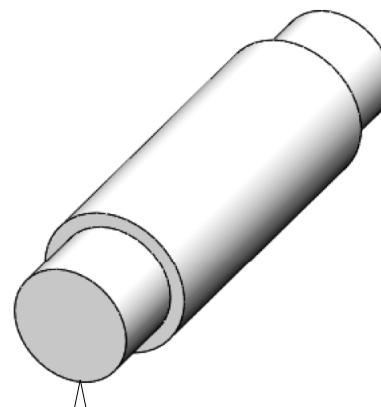


[2] Create a 3D model like this. The details are shown in 3.3-1[11] (last page). Use any coordinate system as your convenience. Save the part with the file name **PinA.#**

3.3-4 Create PinB

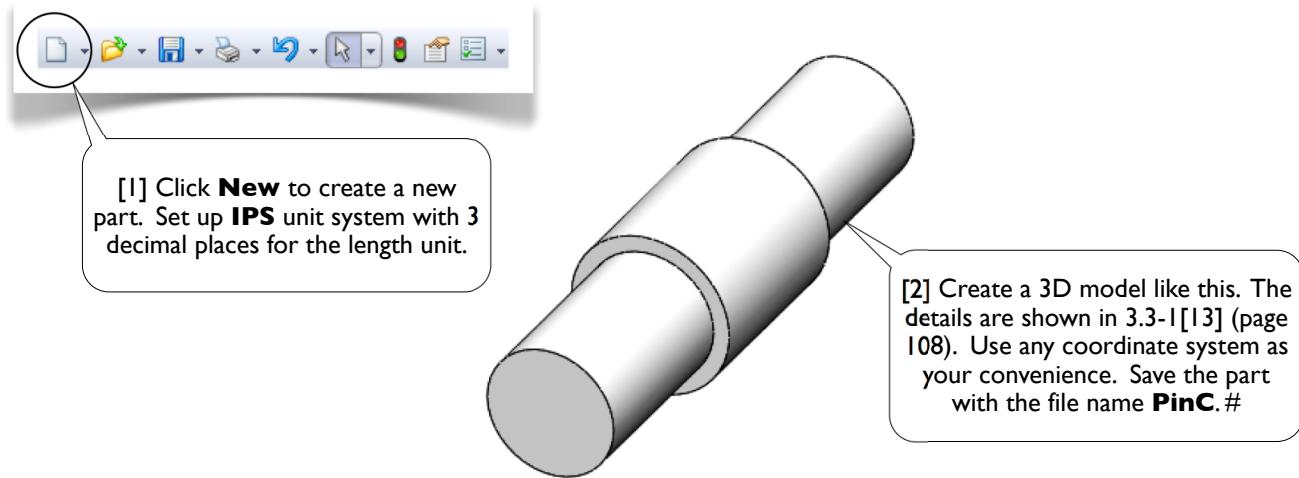


[1] Click **New** to create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.

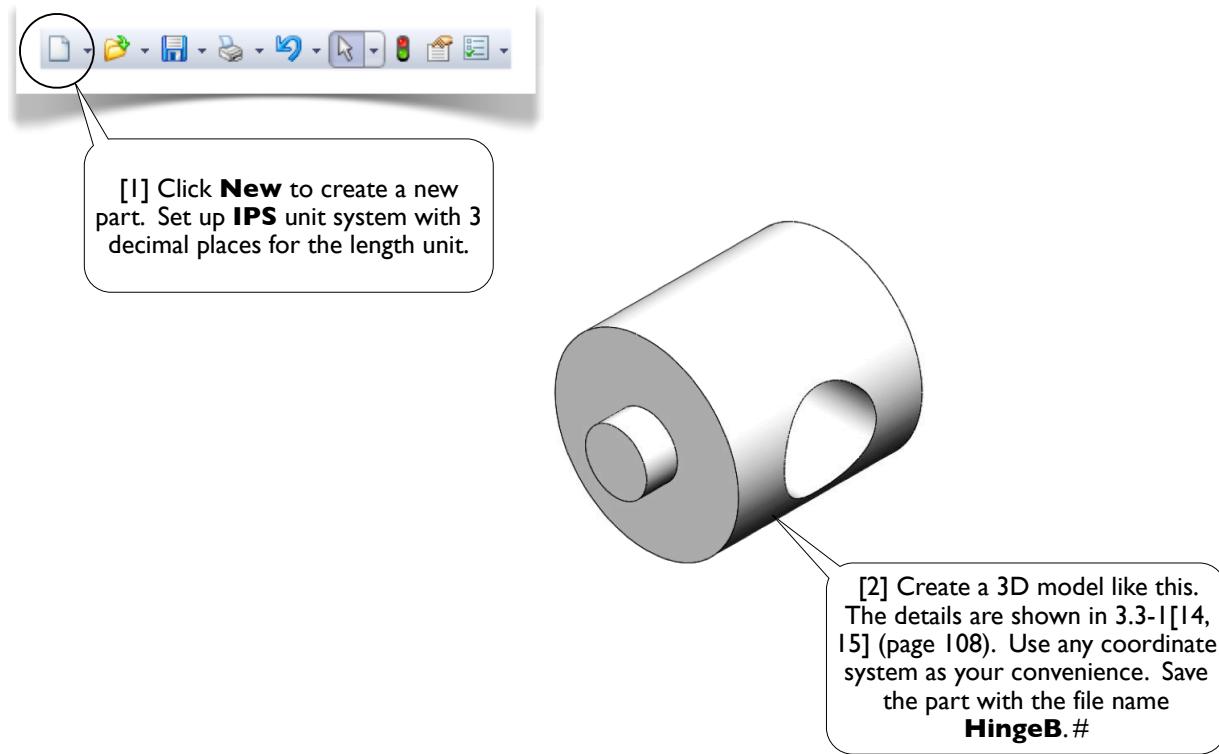


[2] Create a 3D model like this. The details are shown in 3.3-1[12] (last page). Use any coordinate system as your convenience. Save the part with the file name **PinB.#**

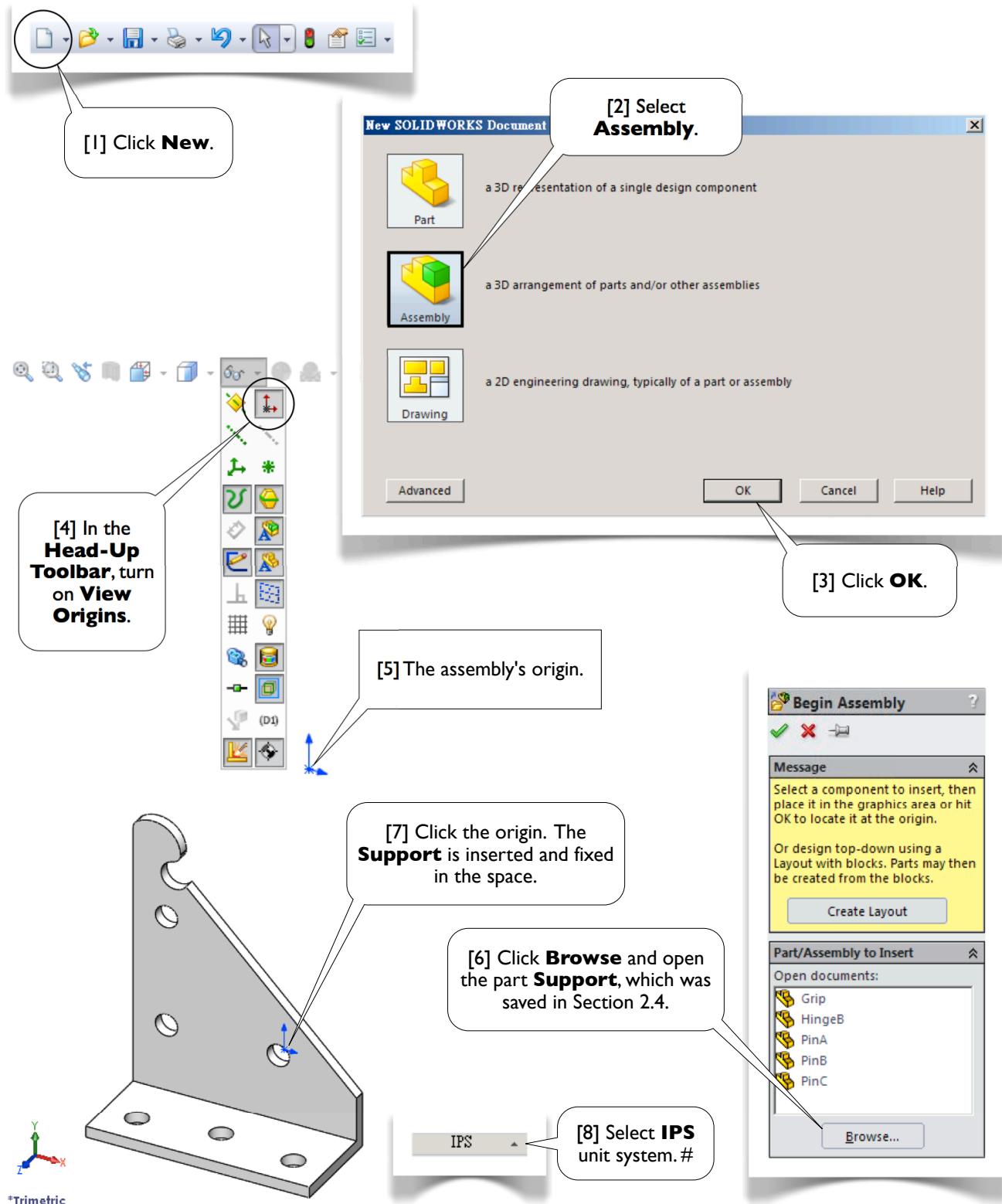
3.3-5 Create **PinC**



3.3-6 Create **HingeB**

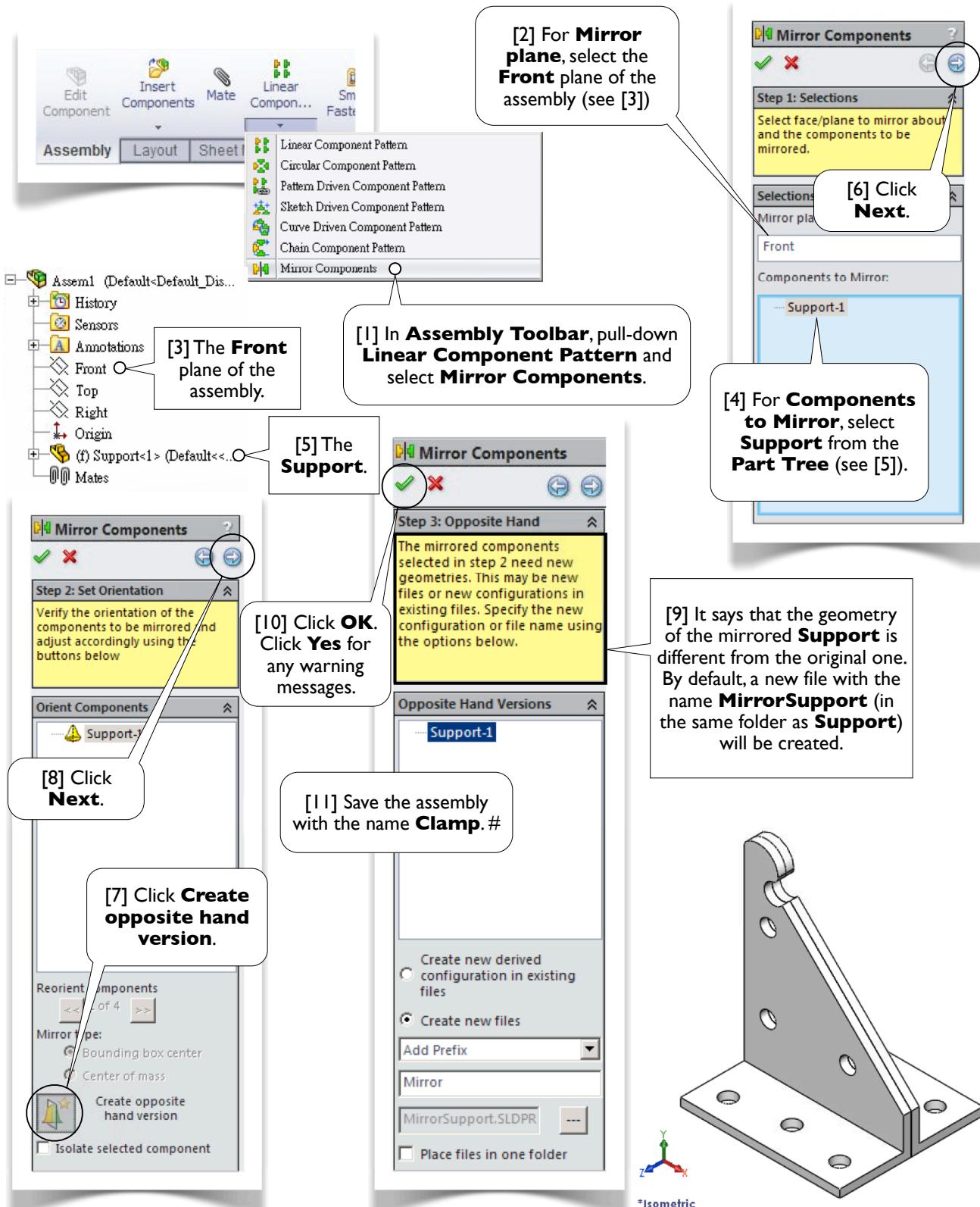


3.3-7 Create a New Assembly and Insert a **Support**

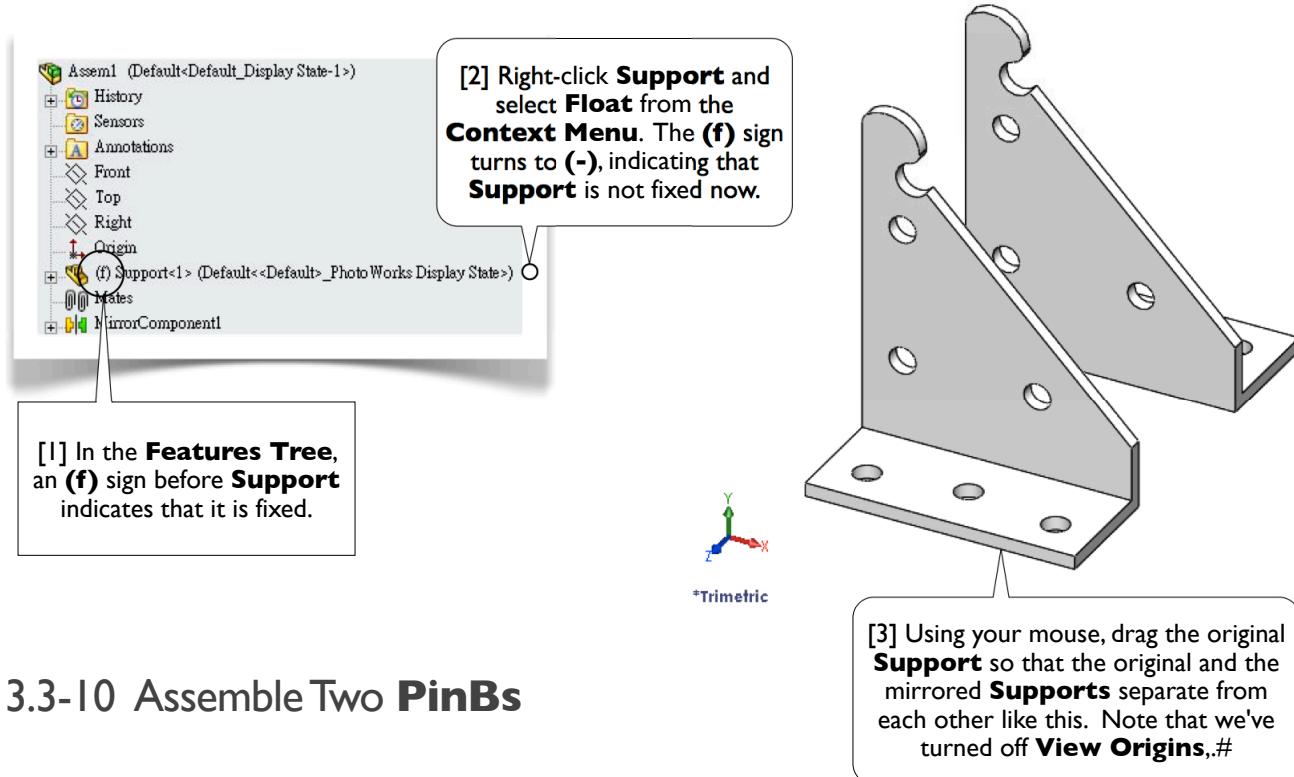


*Trimetric

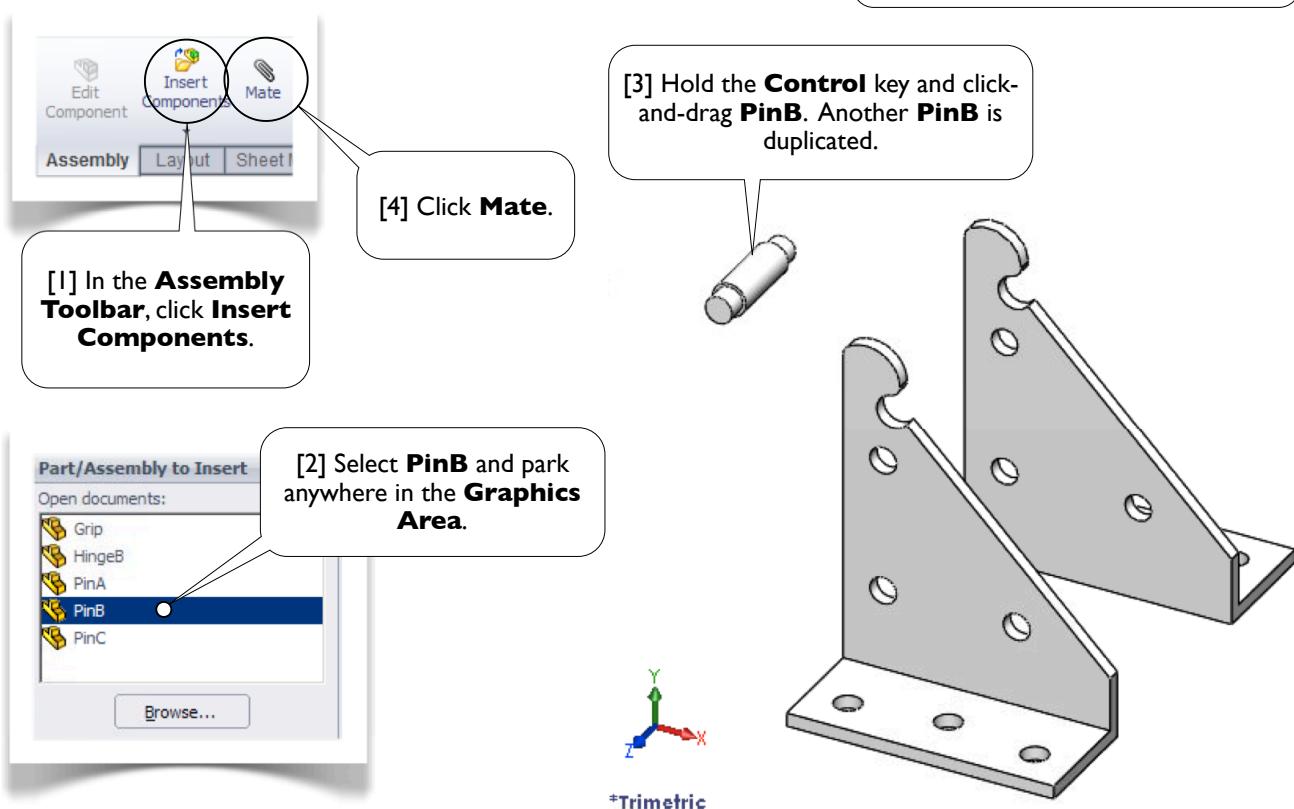
3.3-8 Mirror the Support

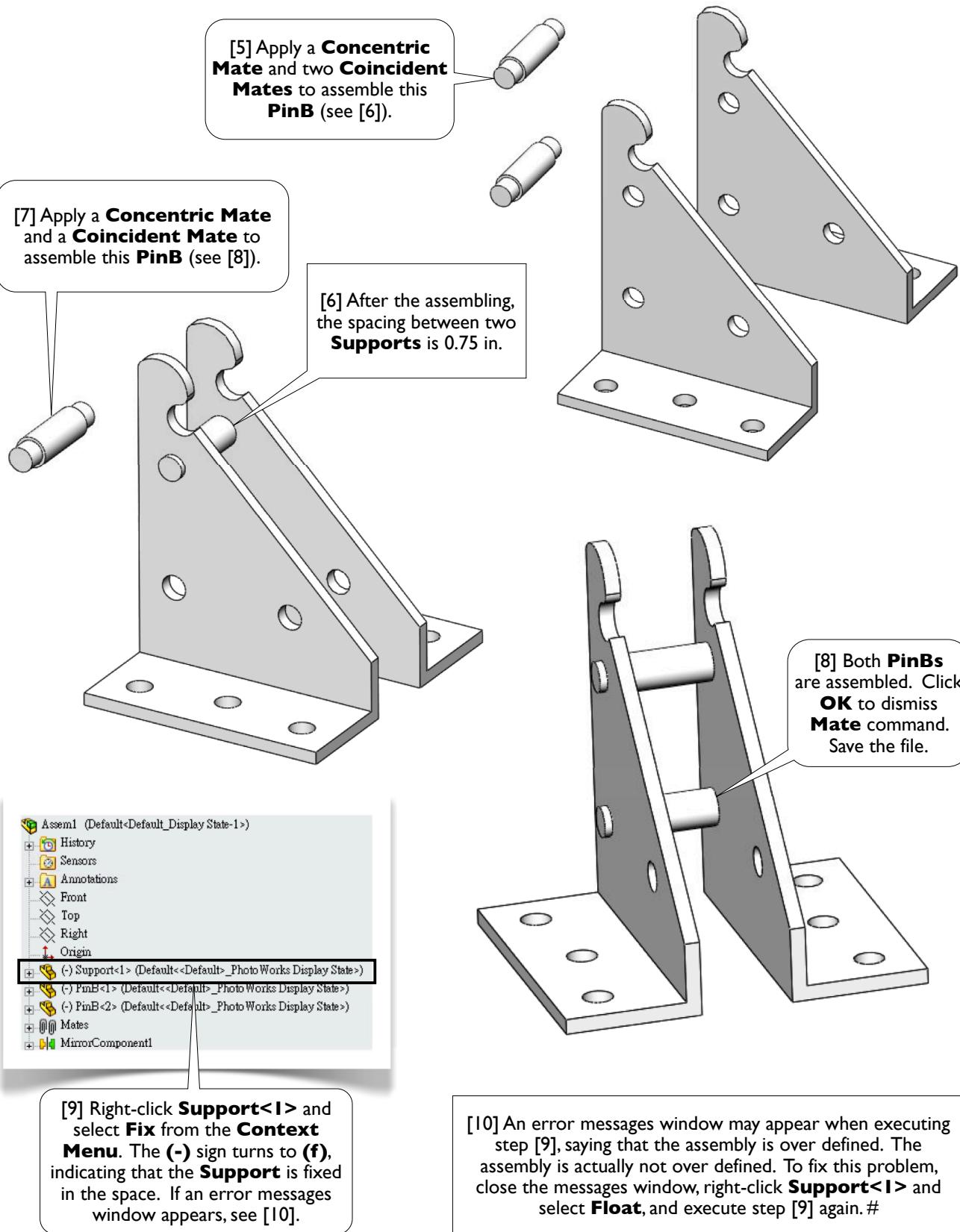


3.3-9 Unfix the Supports

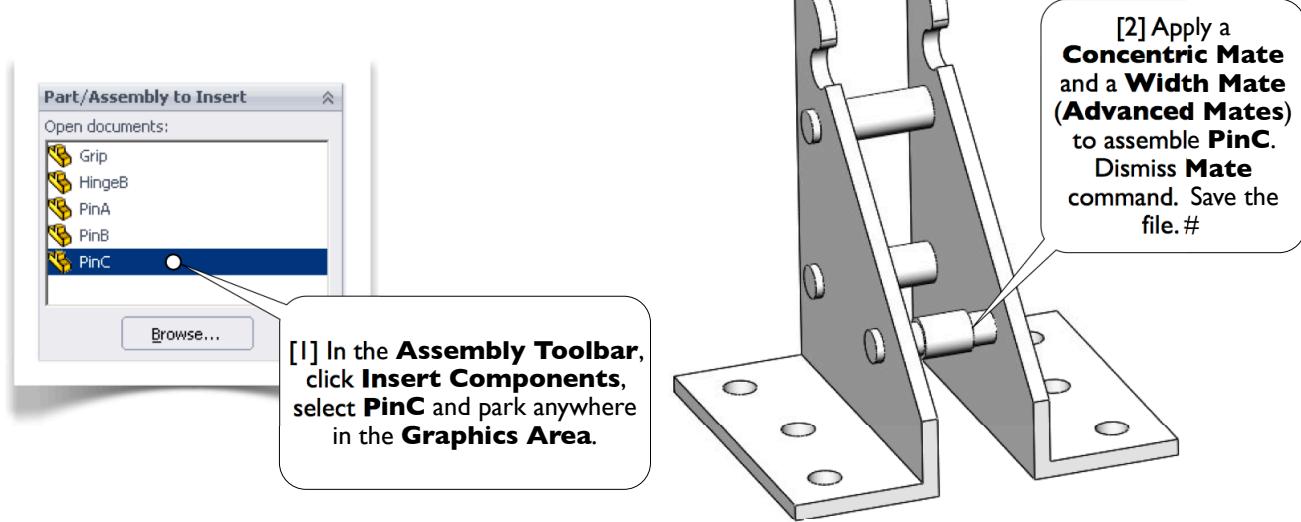


3.3-10 Assemble Two PinBs

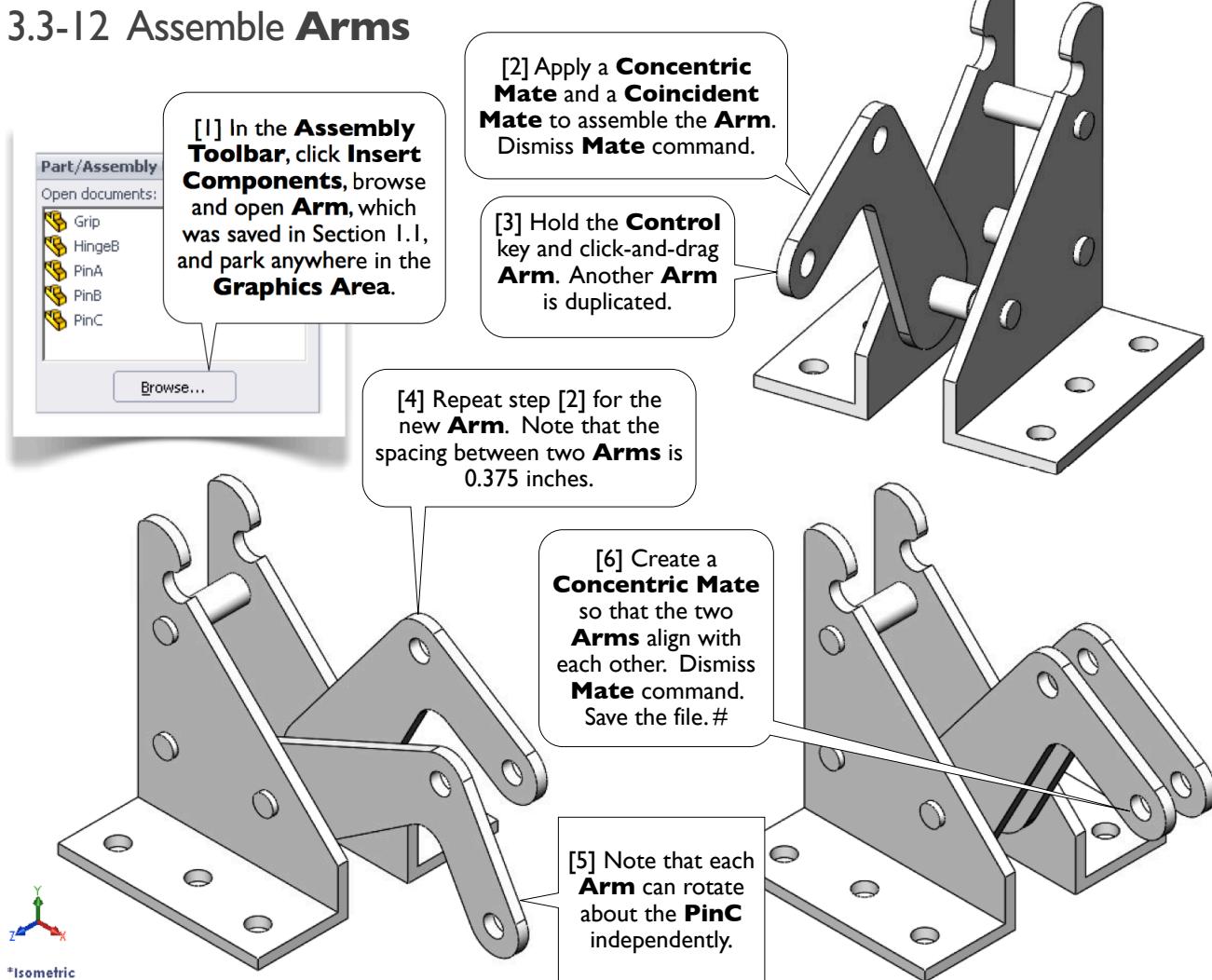




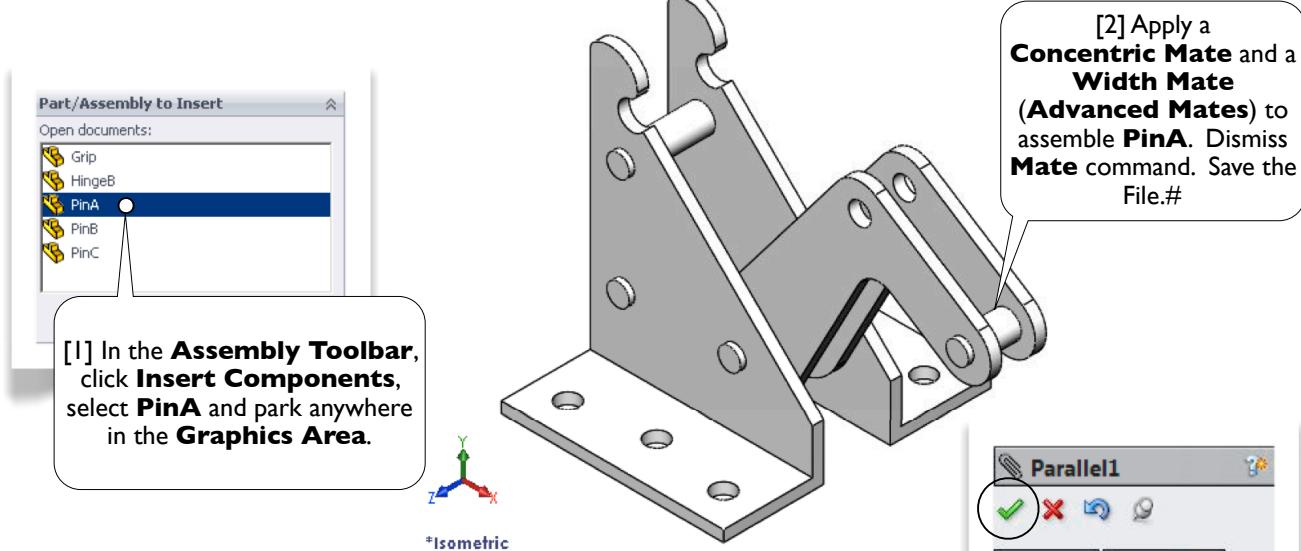
3.3-11 Assemble PinC



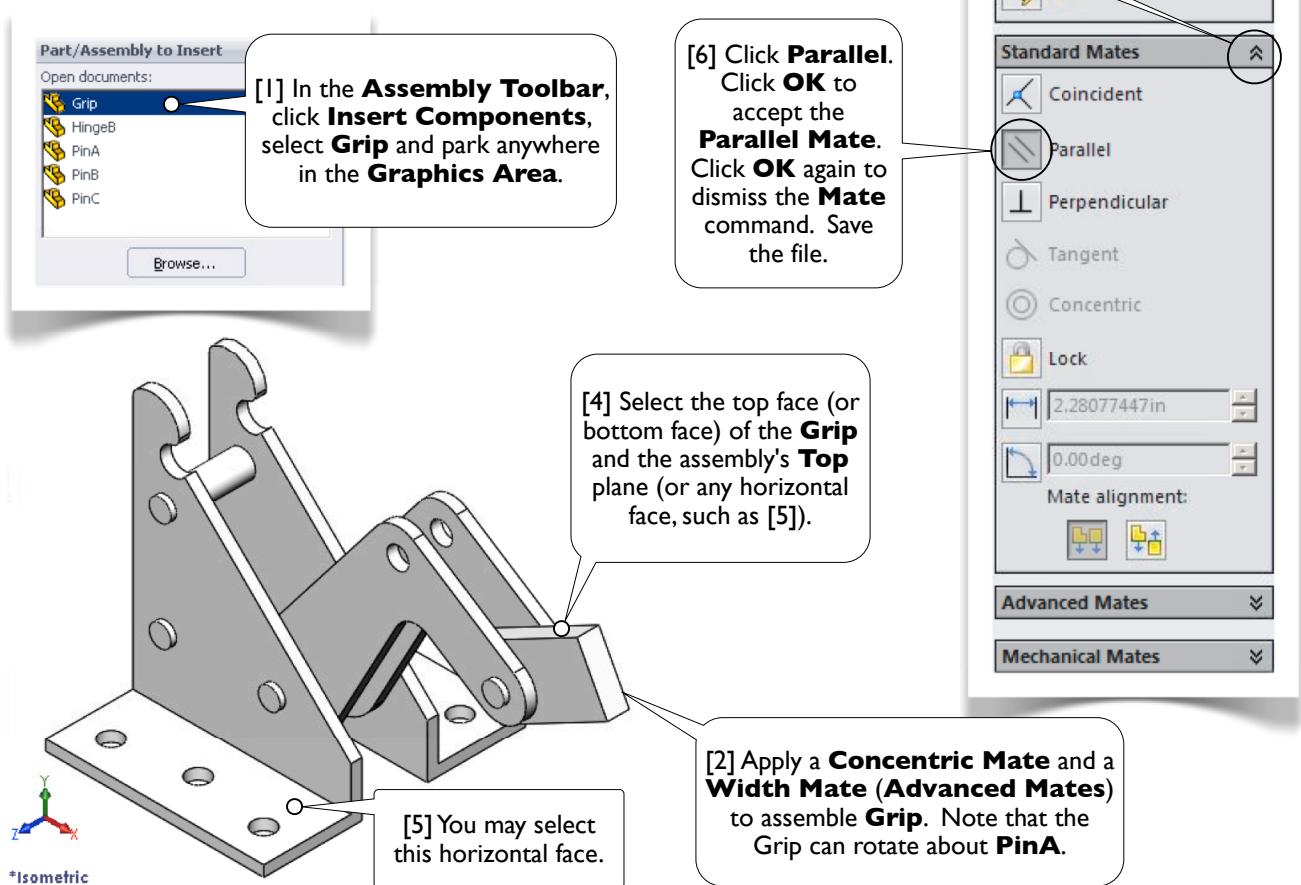
3.3-12 Assemble Arms

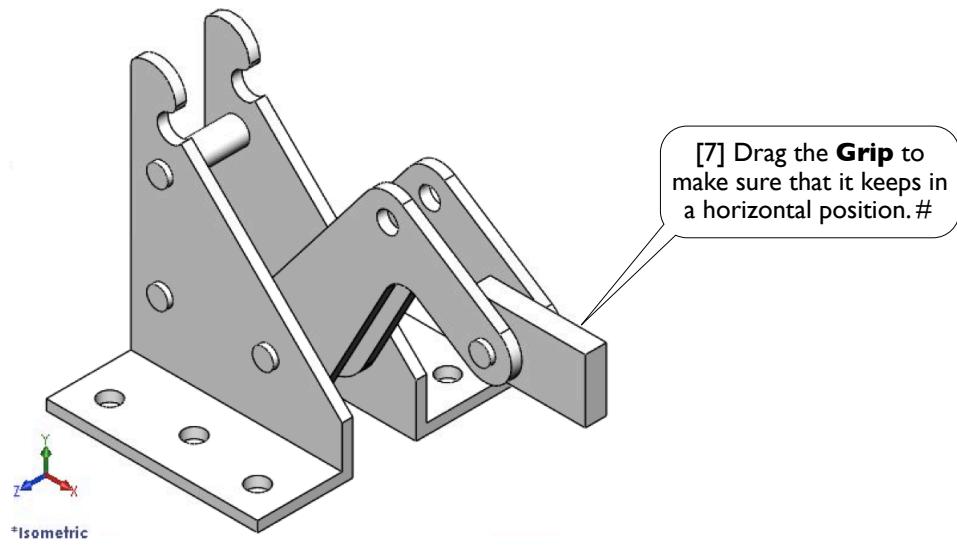


3.3-13 Assemble PinA

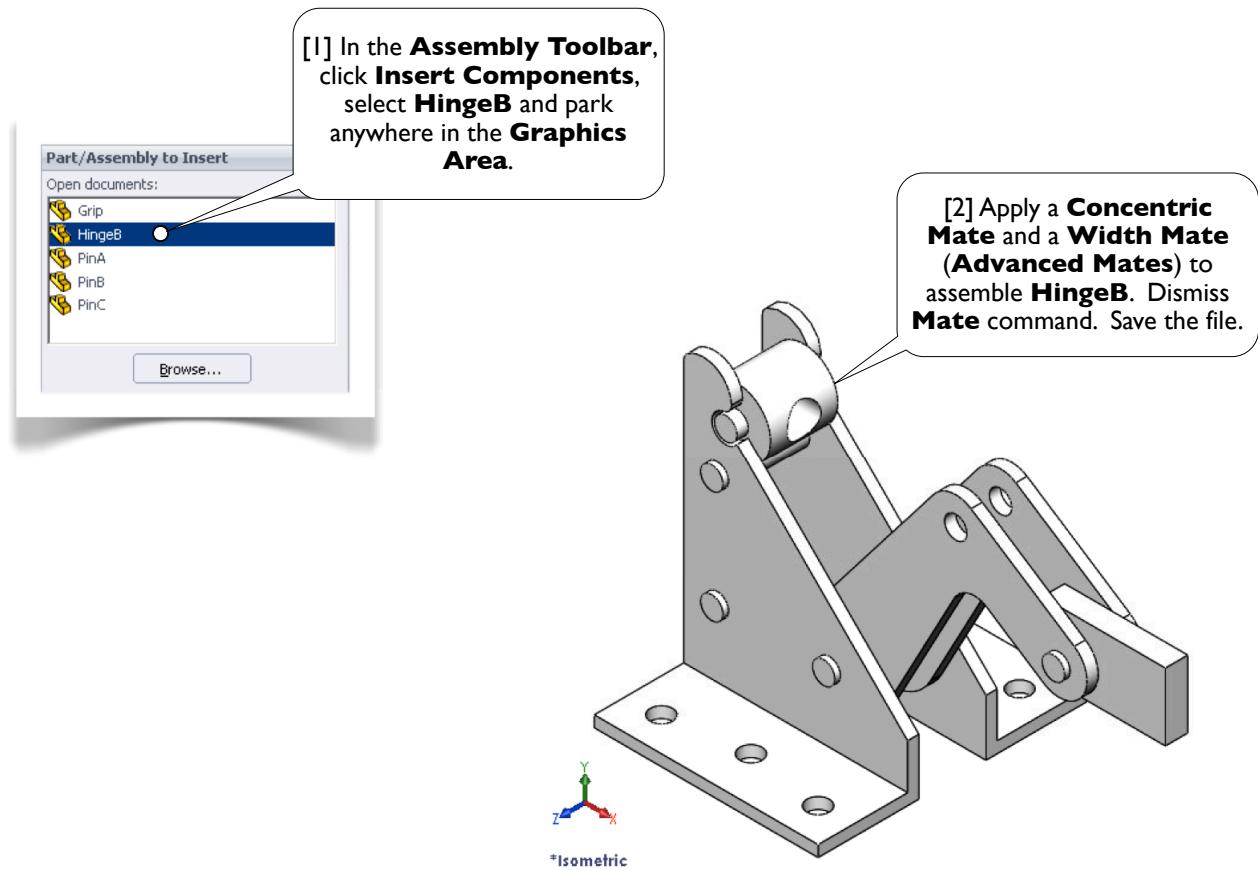


3.3-14 Assemble Grip

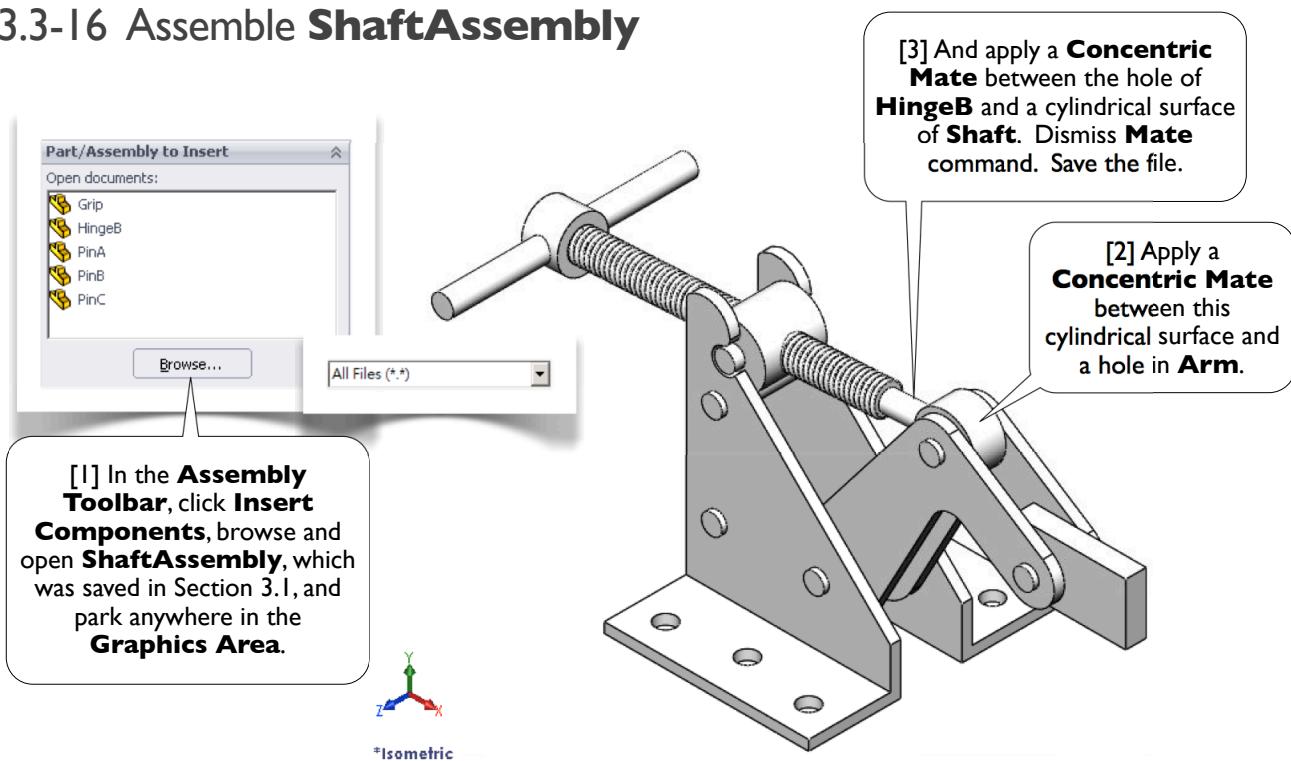




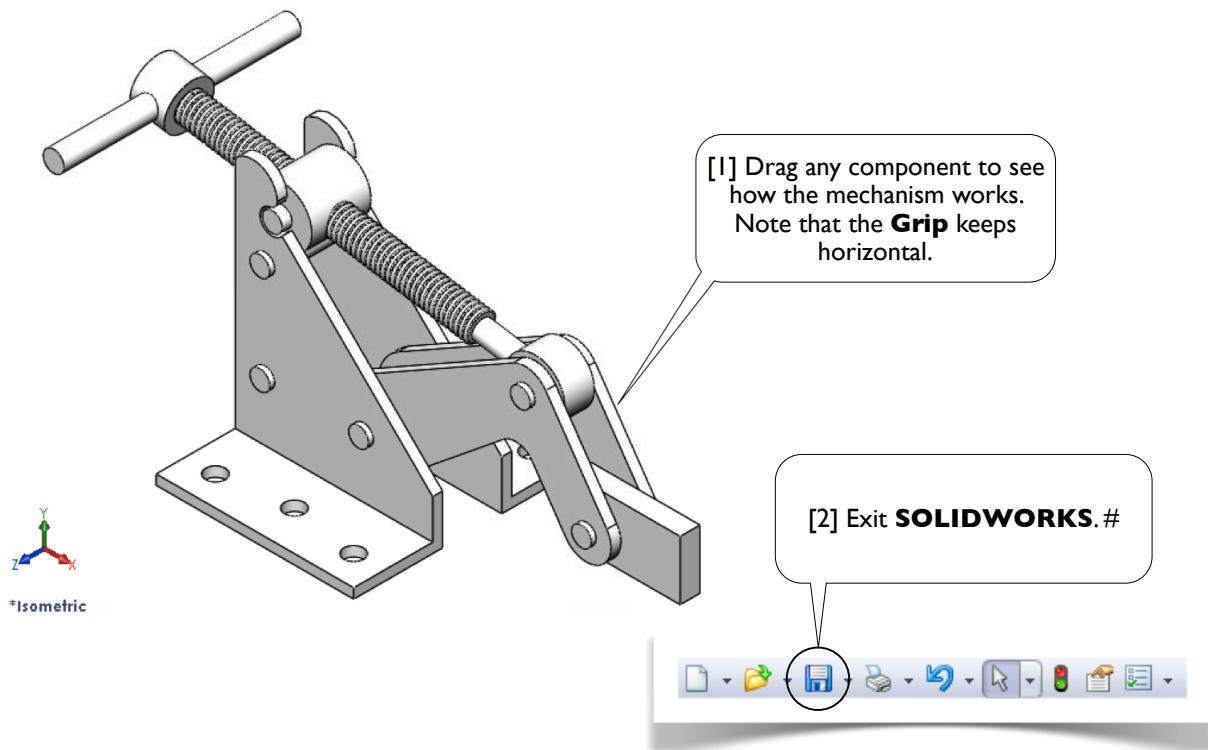
3.3-15 Assemble HingeB



3.3-16 Assemble ShaftAssembly



3.3-17 Test the Clamping Mechanism



Index

- Add Relation, **10**, **15**, **45**
- Advanced Mates, **104**, **114**
- Angle dimension, **19**, **44**
- Arm, **3**, **16**, **107**, **115**
- Arm.SLDPR_T, **16**
- Assembly, **91**, **100**
- Assembly Modeling, **86**
- Assembly Toolbar, **92**, **93**
- At Angle, **71**
- Axis of Revolution, **76**
- Axissymmetric, **62**
- Base Body, **38**, **42**
- Boss, **42**
- Boundaries, **78**
- Box-Select, **64**
- Browse, **111**
- Bushing, **97**, **99**
- Centerline, **19**, **45**
- Centerpoint Arc, **24**, **44**
- Circle, **6**, **11**
- Circular Pattern, **47**
- Circular Sketch Pattern, **21**
- Clamp, **107**, **112**
- Clamping assembly, **3**, **56**, **75**
- Clamping mechanism, **87**, **107**, **118**
- Clear Selections, **21**
- Coincident, **94**, **102**
- Color Codes, **6**
- Components to Mirror, **112**
- Concentric, **93**, **95**
- Constant size, **61**
- Construction Geometry, **18**, **44**
- Context Menu, **6**, **8**
- Control, **113**, **115**
- Control-Middle-Button, **7**
- Convert Entities, **78**
- Coordinate system, **87**, **91**
- Crank, **37**
- Create opposite hand version, **112**
- Depth, **70**
- Direction **1**, **90**
- Display Style>Shaded, **74**
- Distance, **10**
- Document Properties, **5**, **7**
- End Condition, **42**, **55**
- Entities to mirror, **64**
- Entities to Pattern, **21**
- Equal, **10**
- ESC, **6**, **10**, **12**
- Exit Sketch, **15**, **67**, **77**
- Extrude, **22**, **27**, **41**
- Extruded Boss/Base, **15**
- Extruded Cut, **42**, **78**
- Extruding Depth, **15**
- FeatureManager Design Tree, **8**
- Features, **40**
- Features to Pattern, **47**
- Features Toolbar, **15**
- Features Tree, **6**, **8**, **42**
- File>Close, **16**
- File>Exit, **16**
- File>New, **4**
- File>Save, **16**
- Fillet, **53**, **73**
- Fillet radius, **14**
- Finger, **84**
- First Reference, **71**
- Fix, **114**
- Fixed, **106**
- Flip, **59**
- Float, **106**, **113**
- Font, **7**
- Font size, **7**
- Fork, **85**

Front, 6
 Full Round Fillet, **52**, 53
 Geneva, 49
 Geneva Gear Index, 43
 Geometry pattern, 85
 Global coordinate system, 3
 Graphics Area, 6, 8
 Grip, 107, 108, **109**, 116
 Handle, **87**, 88
 Head-Up Toolbar, **9**, 25
 Hide/Show Items>View Planes, 73
 Hide/Show Items>View Sketch Relations, **25**, 51
 Hinge, **87**, 89
 HingeB, 107, 108, **110**, 117
 Hole, 42, **48**
 Hole Type, 48
 Hole Wizard, **49**, 53
 Horizontal, **10**, 39, 45
 Horizontal dimension, 9
 Image Quality, 7
 Inference Line, 9
 Infinite length, 18, **26**
 Insert Components, **92**, 101
 Insert>Boss/Base>Extrude, 15
 Insert>Boss/Base>Revolve, 65
 Insert>Boss/Base>Sweep, 68
 Insert>Cut>Extrude, 42
 Insert>Features>Fillet/Round, 52
 Insert>Features>Hole>Wizard, 48
 Insert>Pattern/Mirror>Circular Pattern, 47
 Insert>Reference Geometry>Plane, 54
 IPS, **5**, 17
 Isometric, 41
 Joint, 106
 LCD, 80
 Line, **12**, 15
 Linear Component Pattern, 112
 Linear Pattern, 84
 Linear Sketch Pattern, 21
 Liquid crystal display, 80
 Loft, 80
 Lofted Boss/Base, 84
 Major diameter, 75
 Mate, **93**, 102
 Mid Plane, 52
 Mirror, **64**, 72, 112
 Mirror about, 64
 Mirror Components, 112
 Mirror plane, 112
 Mirroring plane, 72
 MirrorSupport, 112
 MMGS, 38
 Mouse functions, 7
 Mouse Wheel, 7
 New, 88
 Next, 112
 Normal To, **41**, 67, 69
 Number of Instances, **21**, 47
 Number of planes to create, 82
 Offset distance, **54**, 59
 Options, 5
 Over-defined, 6
 Pan, 7
 Parallel, 45
 Parallel mate, 116
 Park the part, 92
 Part, **4**, 42
 Part documents, 91
 Part Modeling, 36
 Part Tree, 8
 Path, 66
 Pattern Axis, 47
 Pattern Direction, 85
 Pierce, 67
 Pin, **97**, **99**
 Pin down, 4
 PinA, 107, 108, **109**, 116
 PinB, 107, 108, **109**, 113
 PinC, 107, 108, **110**, 115
 Pipe, 73
 Pitch, 75
 Positions, **48**, 53
 Profile, **62**, 66, 84
 Property Box, 10
 Pull-Down Menus, 4
 Radius, **14**, 73
 Ratchet, 22
 Ratchet stop, 17, **23**
 Ratchet Wheel, **17**, 23
 Reference geometries, **42**, 54
 Reference Geometry>Axis, 70
 Reference Geometry>Plane, **59**, 71, 82
 Reverse Direction, **55**, 70, 85
 Revolve, 62, **65**, 76
 Right, 71
 Round-cornered box, 4

Save, 16
 Second Reference, 71
Shaft, 79, 87, 88
 Shaft Assembly, 87
ShaftAssembly, 96, 107, 118
 ShaftAssembly.SLDASM, 96
 Sharp-cornered box, 4
 Sketch, 6
 Sketch Fillet, 14, 15, 39
 Sketch Toolbar, 9, 15
 Sketching, 2
 Smart Dimension, 6, 9
 SolidWorks, 4
 SolidWorks Terms, 4
Standard Mates, 105, 116
 Standard Views Toolbar, 41, 67, 69
 Stop, 27
 Sub-assembly, 87
 Support, 56, 61, 107, 111, 113
Sweep, 66, 68
 Sweeping Path, 77
 Sweeping Profile, 77
 Swept Cut, 77
 Swivel, 97, 98
 Symmetric, 45
Tangent, 25, 45
 Tangent Arc, 24
 Tangent line, 13
 Textboxes, 4
 Thread form, 75
Through All, 42, 78
 Toolbar, 8, 16
 Tools>Options, 5
 Tools>Sketch Tools>Circular Pattern, 21
 Tools>Sketch Tools>Mirror, 64
 Top, 54
 Transition Pipe, 66
 Trim Entities, 13, 15
 Trim to closest, 13
 Two Planes, 71
 Undo, 7, 13
 Unfixed entity, 46, 51
 Unified National Coarse, 75
 Units, 5
 Universal Joint, 50, 97
 Up To Surface, 55
 User interface, 4
 Vertical, 11, 39
 View Orientation>Isometric, 41
 View Orientation>Normal To, 40
 View Origins, 91, 96, 100
 View Planes, 55, 60
 Well-defined, 6
 Wheel, 62, 65
 Width Mate, 104, 115, 116
 Window, 16, 91
 Yoke, 50, 55, 97, 98, 103, 106
 Zoom in/out, 7
 Zoom to Fit, 9