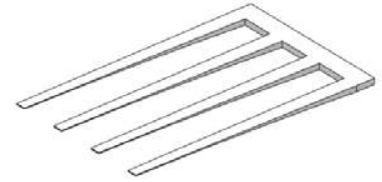


# Section 2.8

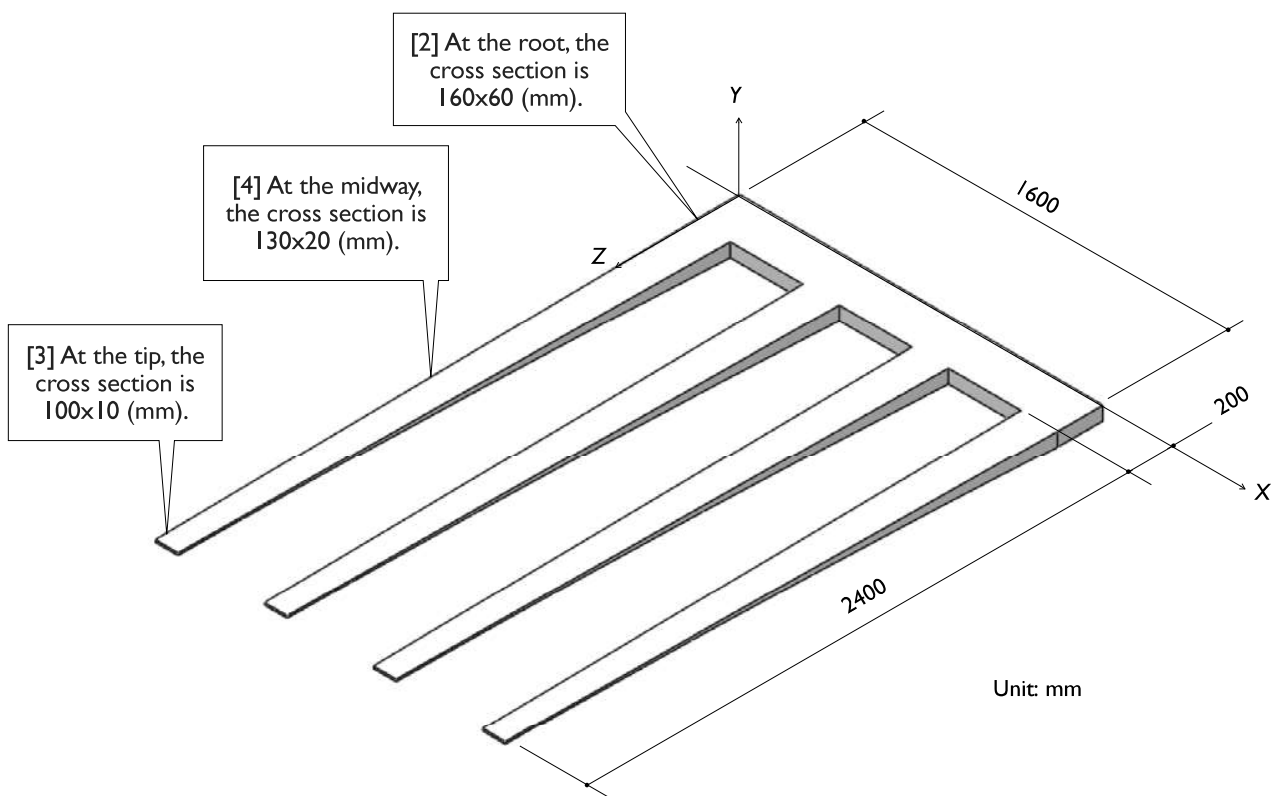
## Lifting Fork



### 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.

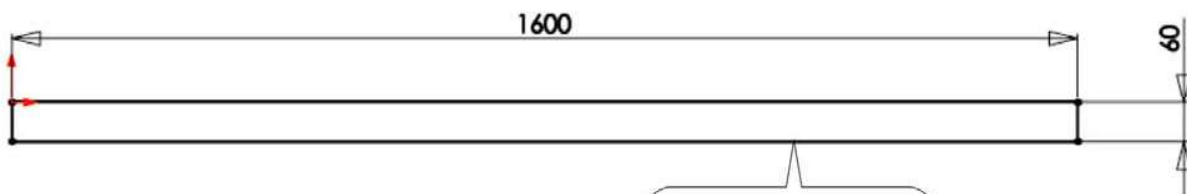
The cross sections of the prongs (fingers) are not uniform along the length [2, 3, 4]. The **Extrude** command or **Sweep** command can not be used to create the prongs. This exercise introduces a new command to create 3D solids: **Loft**, which takes a series of profiles and creates a 3D solid that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



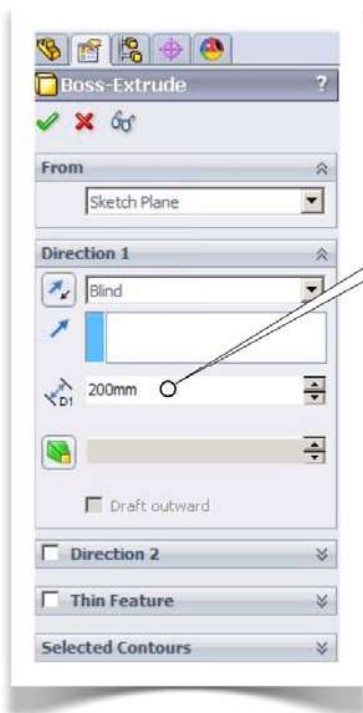
## 2.8-2 Start Up

[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

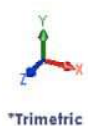
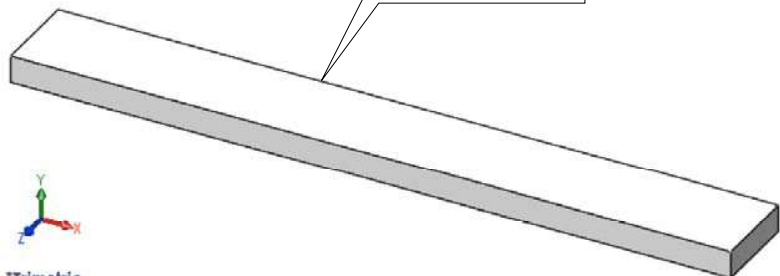


[1] On the **Front** plane, draw a sketch like this.

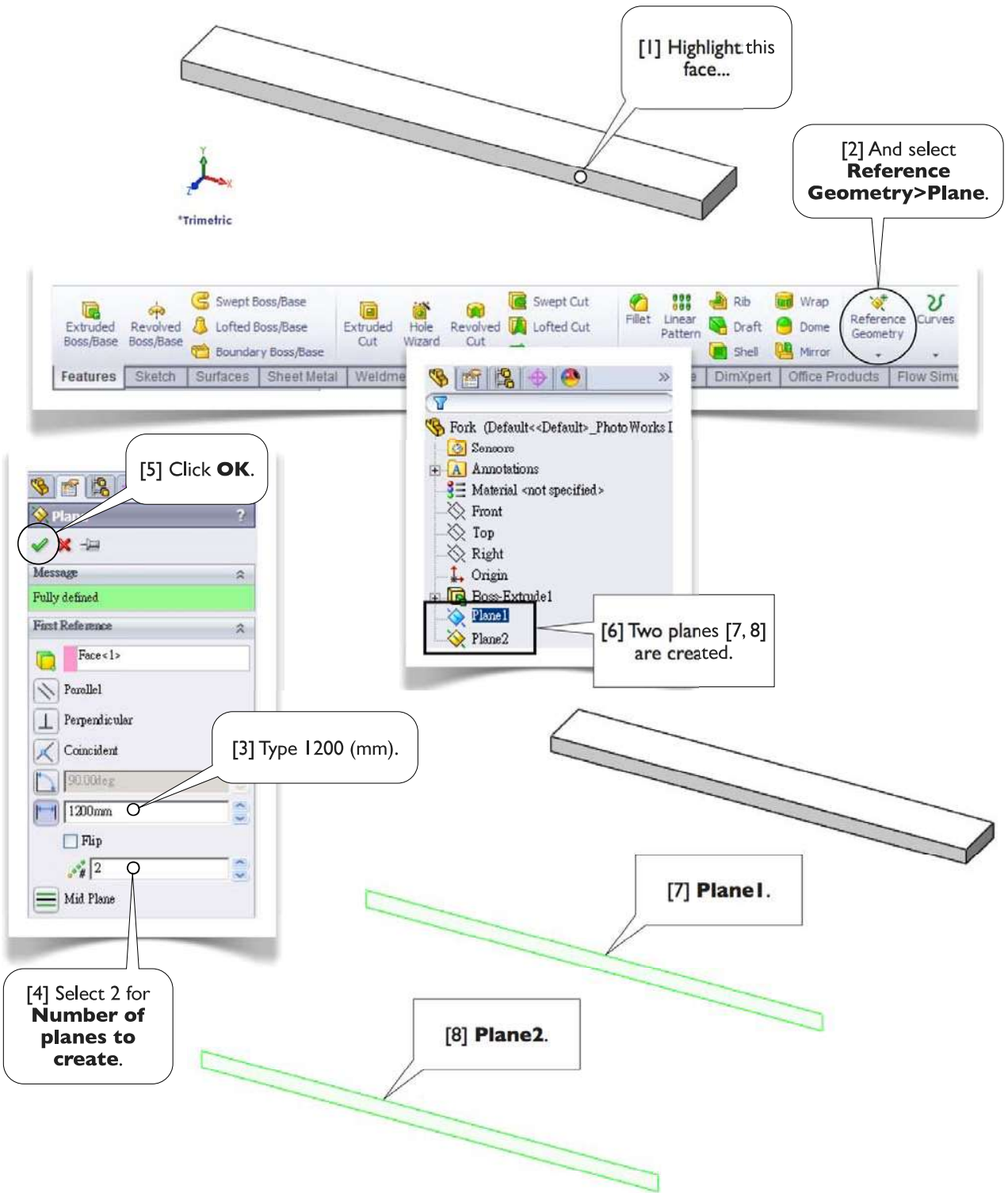


[2] **Extrude** the sketch 200 mm to create the transversal beam.

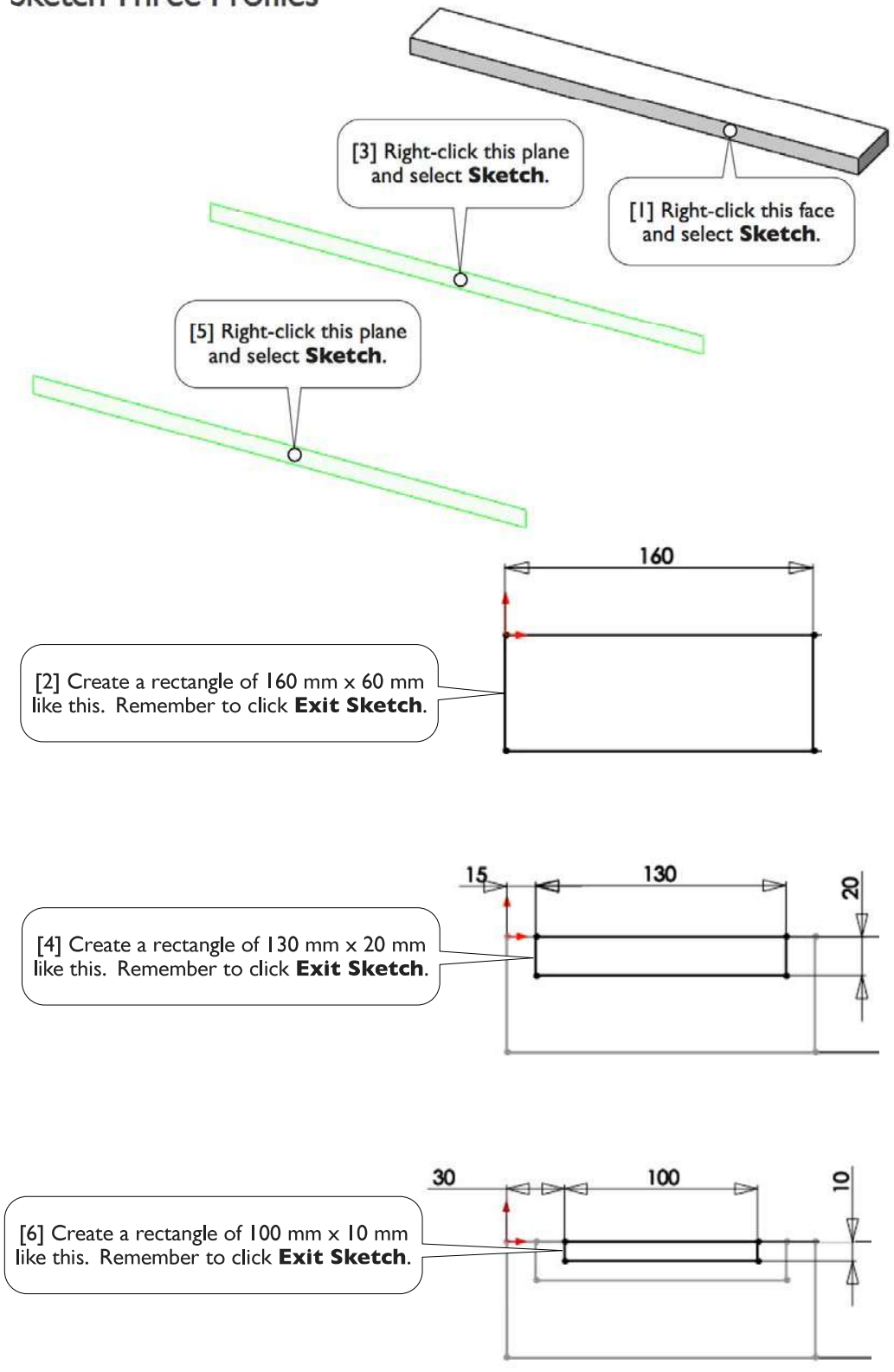
[3] The transversal beam.



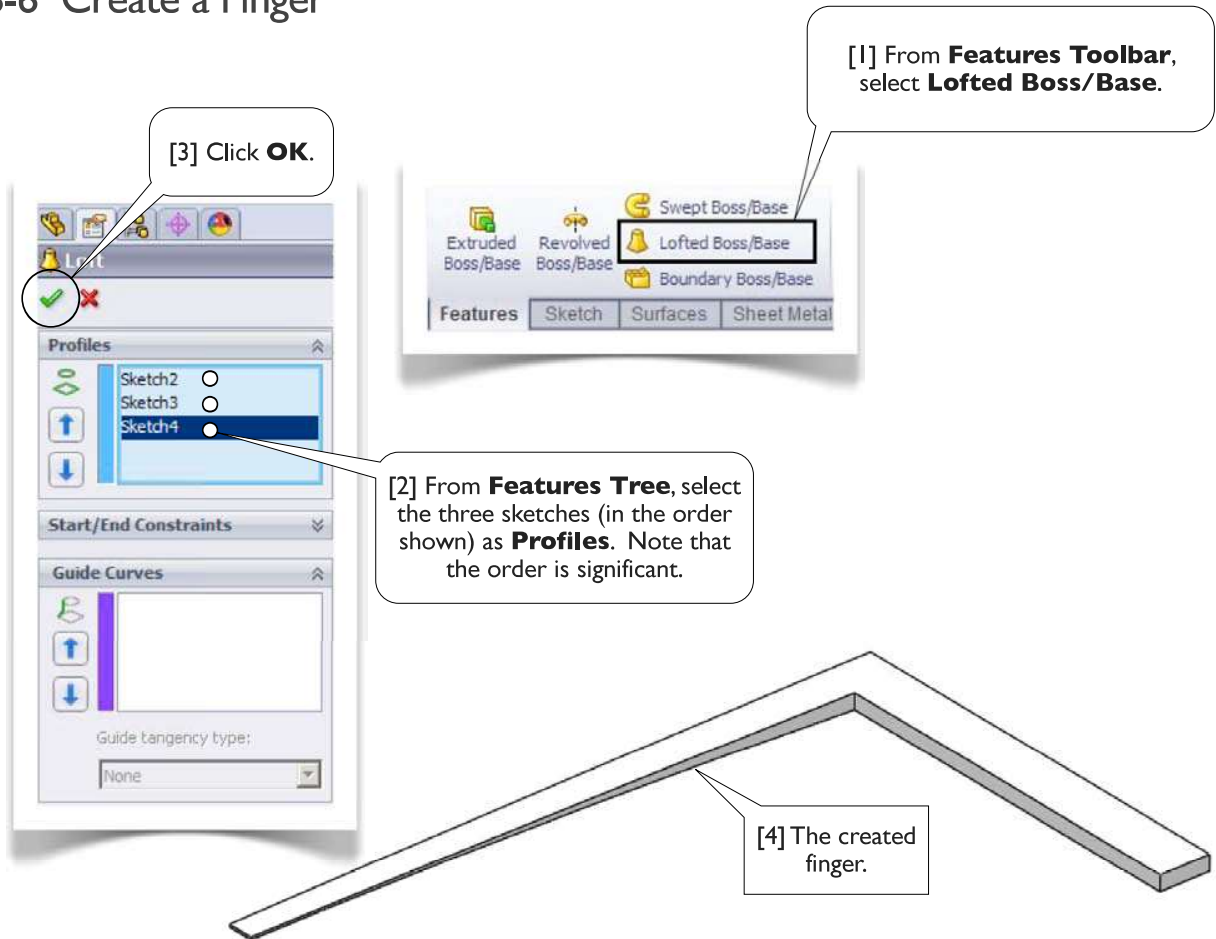
### 2.8-4 Create Two Planes



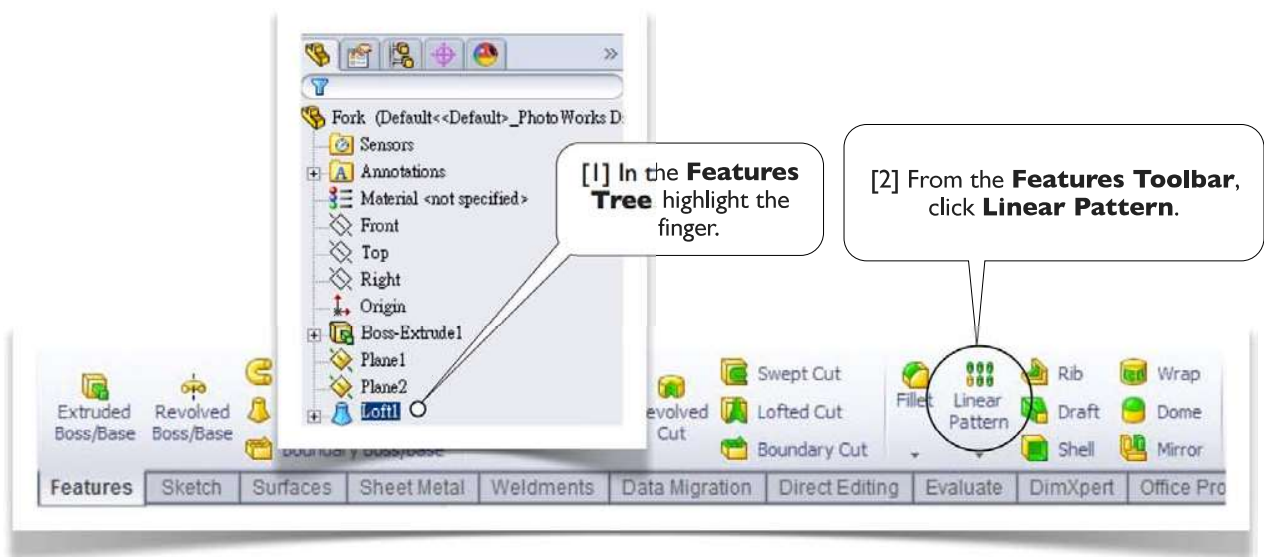
## 2.8-5 Sketch Three Profiles

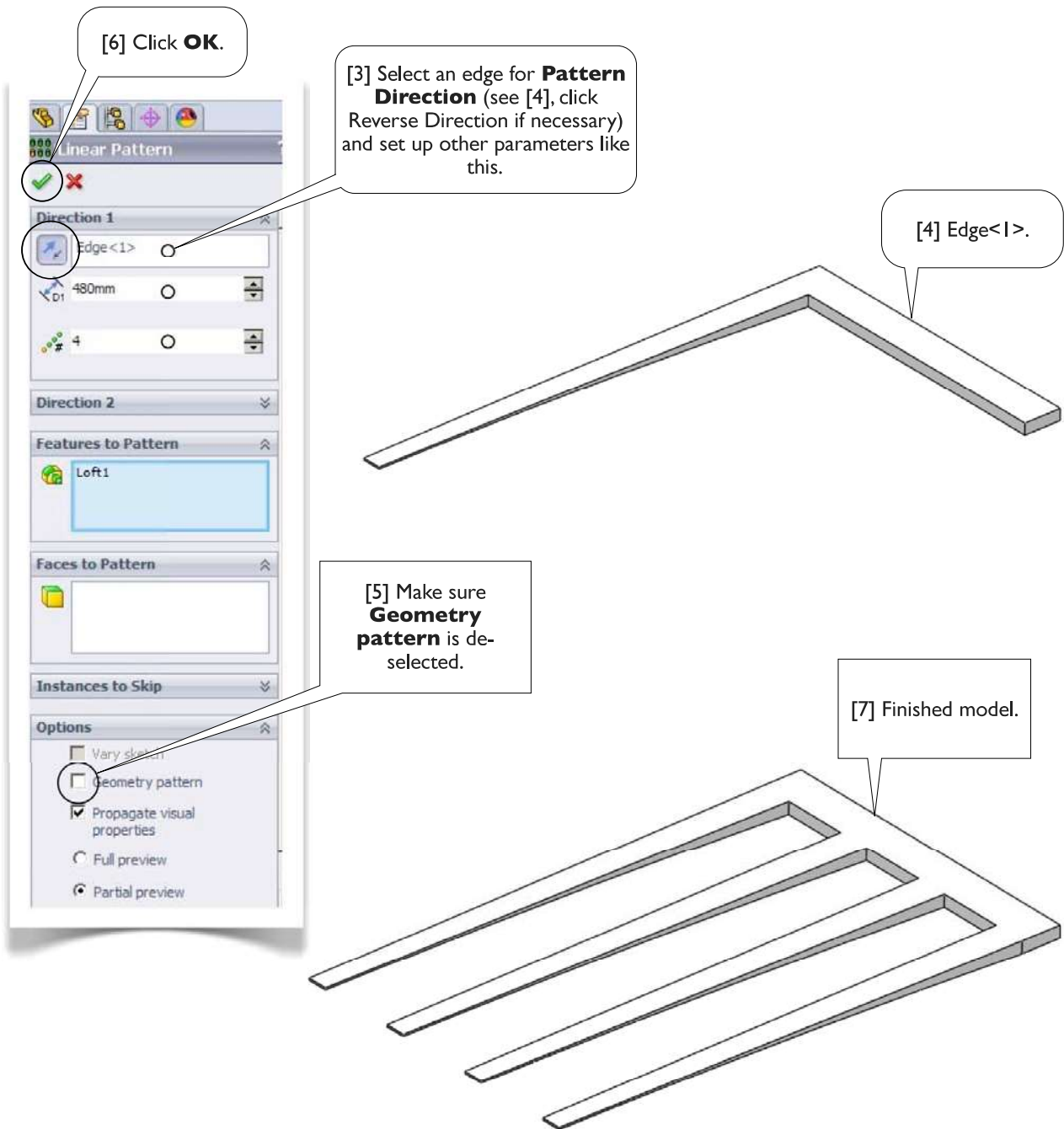


### 2.8-6 Create a Finger



### 2.8-7 Create the Other Fingers



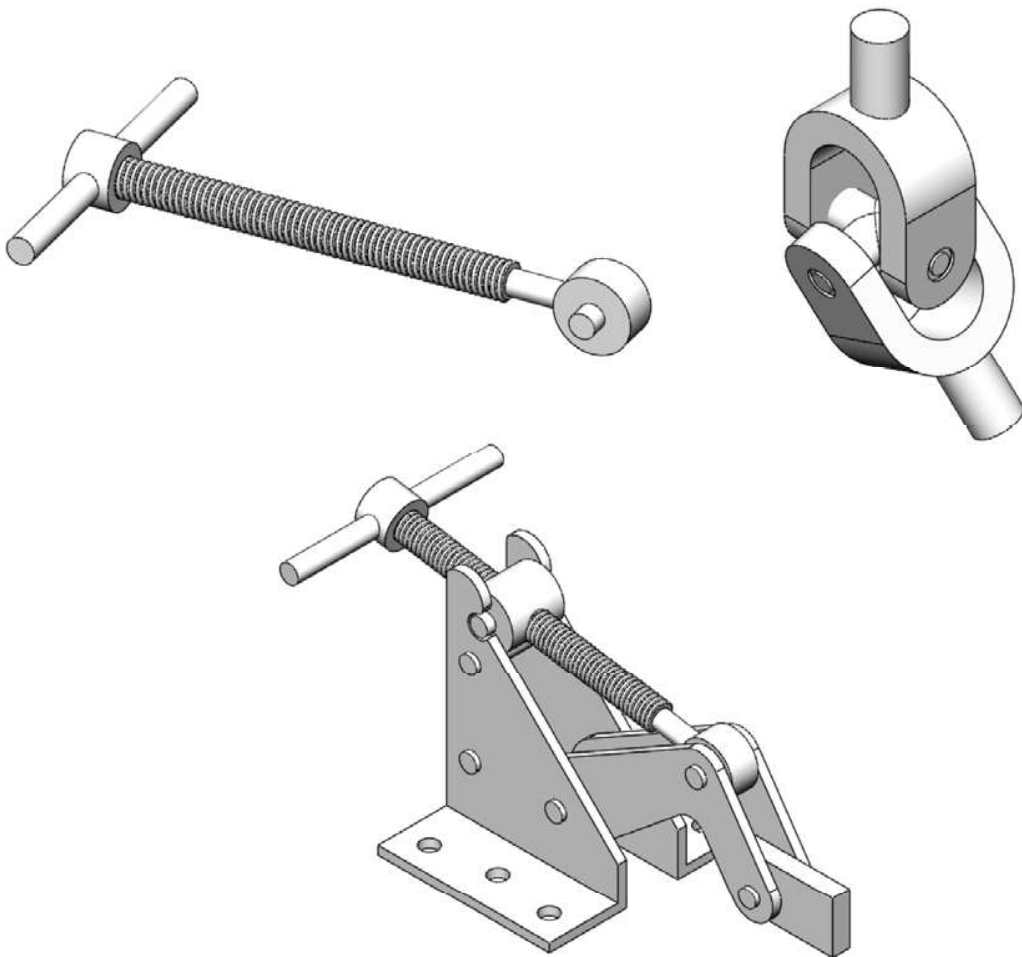


[8] Save the part with the file name **Fork**. Close the file and exit **SolidWorks**.

# Chapter 3

## Assembly Modeling

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# Section 3.1

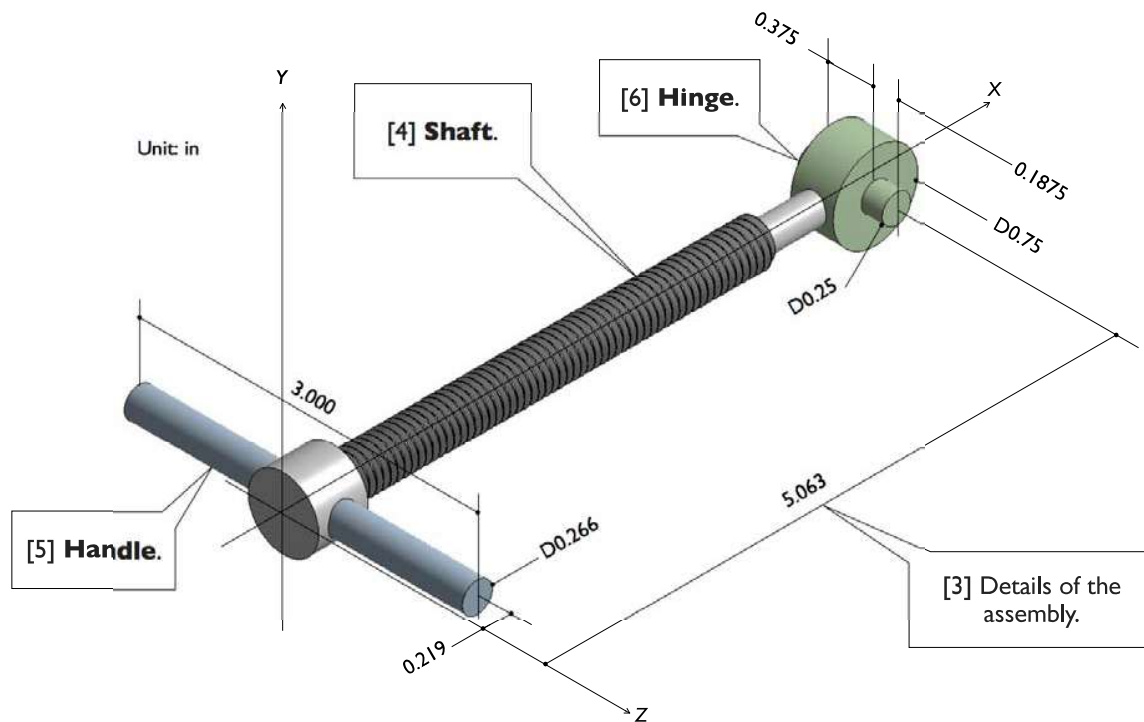
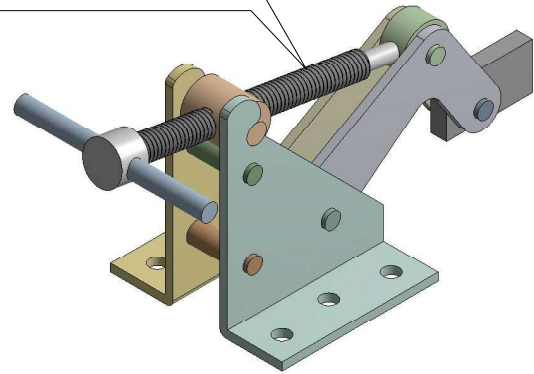
## Shaft Assembly



### 3.1-1 Introduction

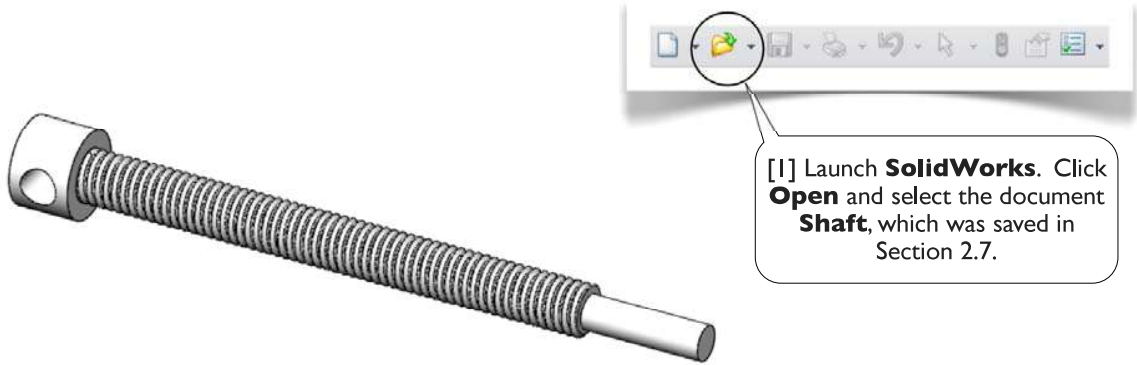
[1] In this exercise, we'll create a shaft assembly [2, 3]. The assembly consists of three parts: the **Shaft** [4] created in Section 2.7, a **Handle** [5], and a **Hinge** [6]. We use a coordinate system for the assembly which is coincident with that of the part **Shaft**.

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

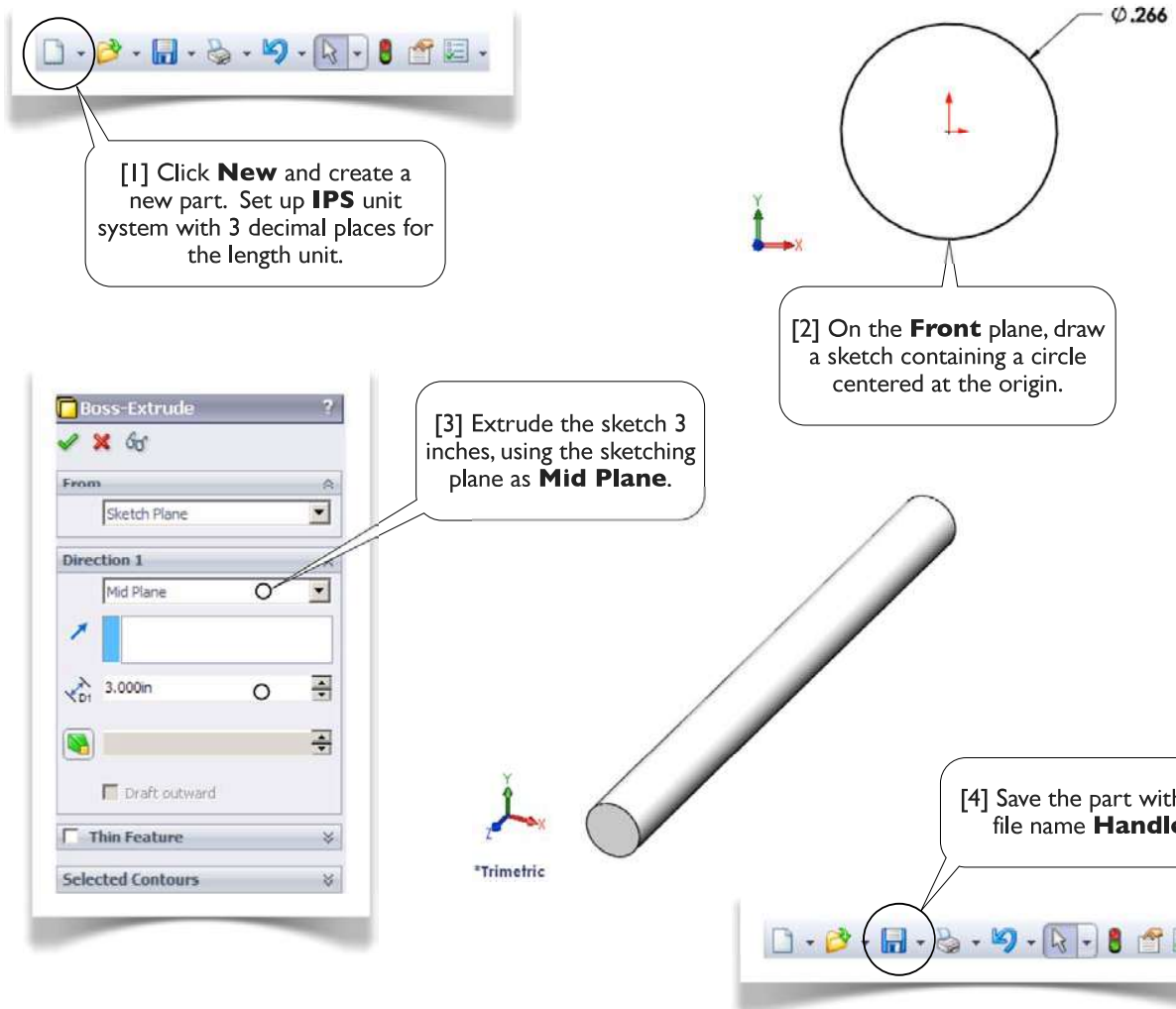




### 3.1-2 Open **Shaft**



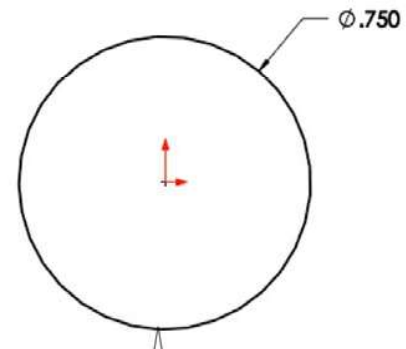
### 3.1-3 Create **Handle**



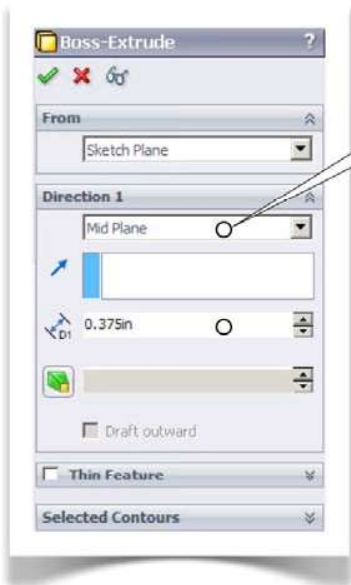
### 3.1-4 Create Hinge



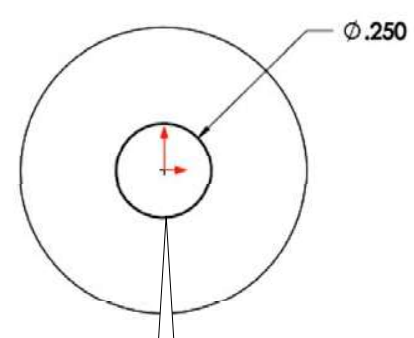
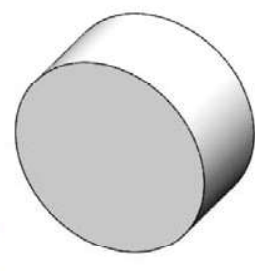
[1] Click **New** and create a new part. Set up **IPS** unit system with 3 decimal places for the length unit.



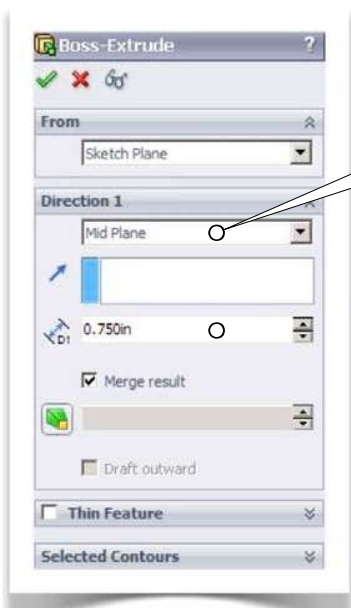
[2] On the **Front** plane, draw a sketch containing a circle centered at the origin.



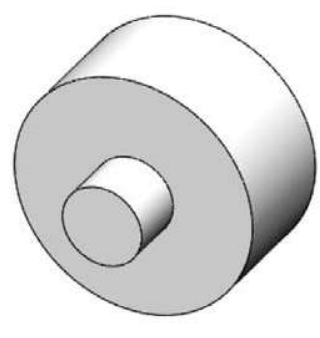
[3] Extrude the sketch 0.375 inches, using the sketching plane as **Mid Plane**.



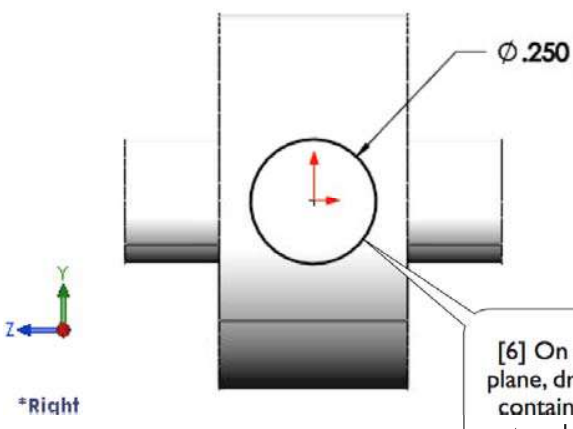
[4] On the **Front** plane, draw a sketch containing a circle of diameter 0.25 inches like this.



[5] Extrude the sketch 0.75 inches, using the sketching plane as **Mid Plane**.



\*Trimetric



[6] On the **Right** plane, draw a sketch containing a circle centered at the origin.



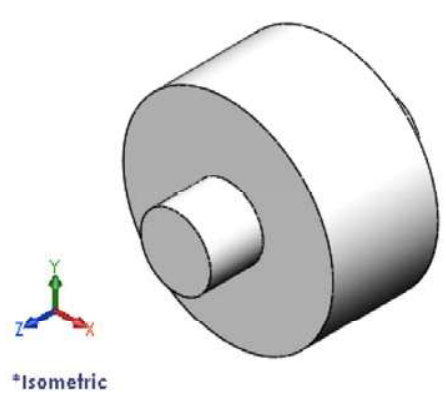
[7] In the **Features Toolbar**, click **Extruded Cut**.



[9] The finished part.



[8] Select **Through All** (for **Direction 1** only).



\*Isometric



[10] Save the part with the file name **Hinge**.

### 3.1-5 Create a New Assembly

[1] If you pull down the **Window** menu, you would 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**.

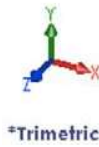
[4] Click **OK**.

[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.

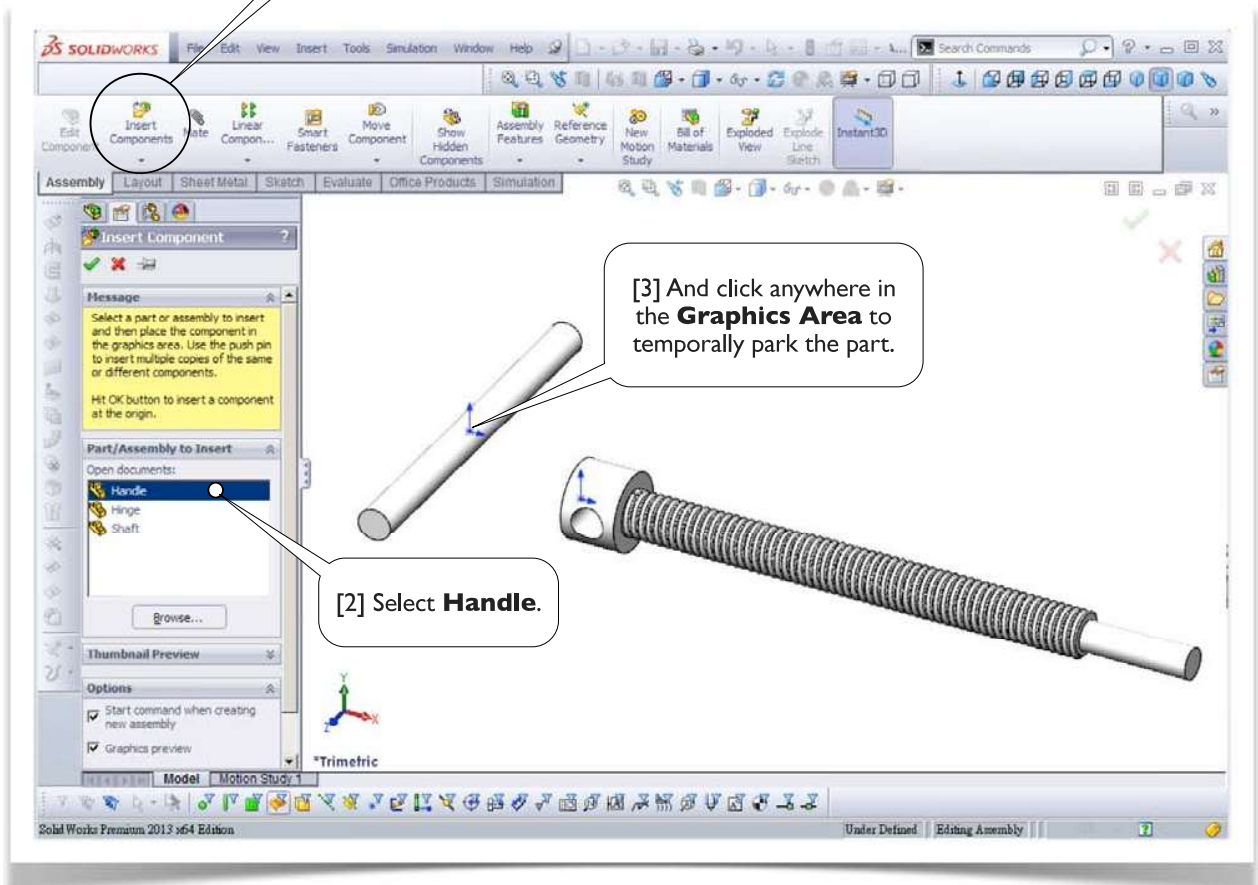
[7] In the **Property Box**, select **Shaft**.

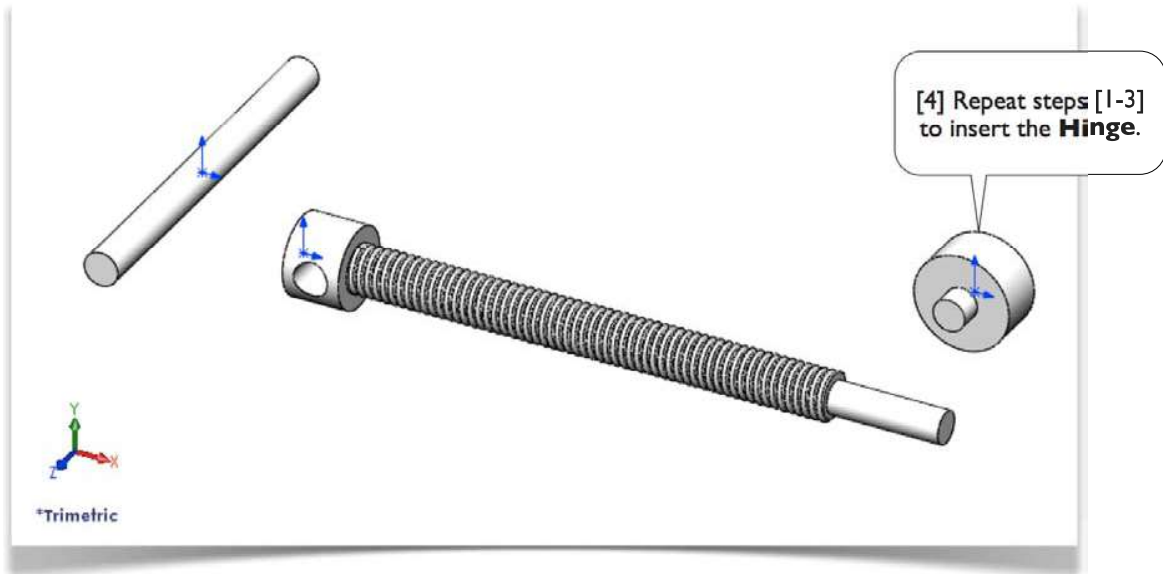
[8] Click the origin. Now the **Shaft** is inserted and its coordinate system aligns with the assembly's coordinate system.



### 3.1-6 Insert the Other Components

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





### 3.1-7 Assemble **Handle**

[1] In the **Assembly Toolbar**, click **Mate**.

[2] Select the cylindrical face of the **Handle**.

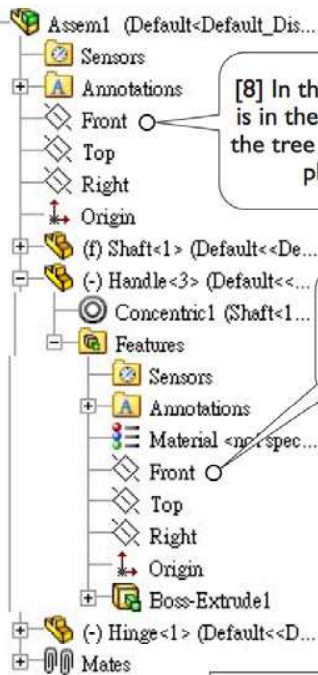
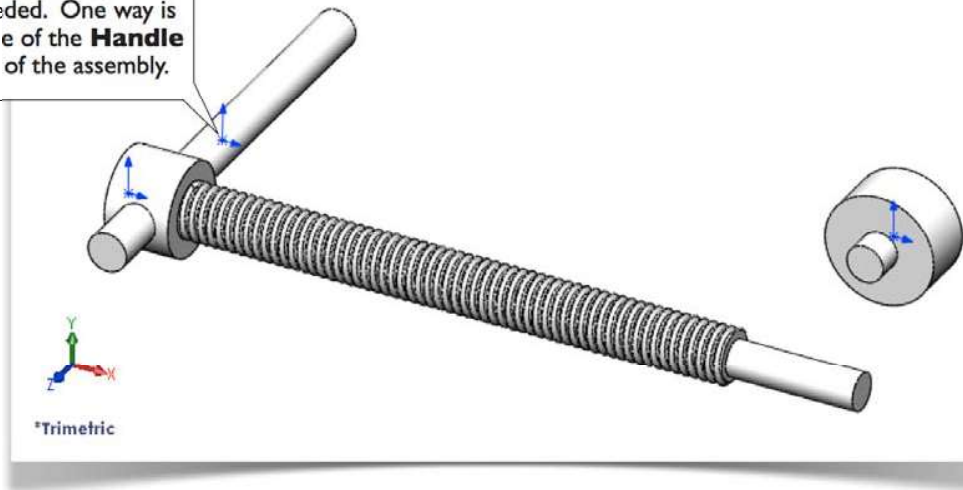
[3] Select the cylindrical face of the **Shaft's** hole.

[4] The selected two faces appear here.

[5] A **Concentric** mate is automatically selected.

[6] Click **Add/Finish Mate**. Note that the **Property Box** is not dismissed.

[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.



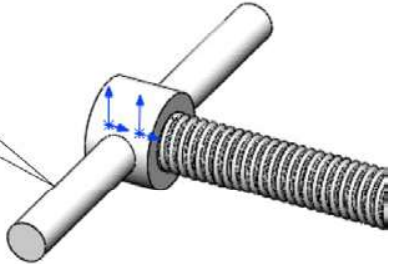
[8] In the **Features Tree** (which is in the **Graphics Area**); expand the tree if necessary), select **Front** plane of the assembly.

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

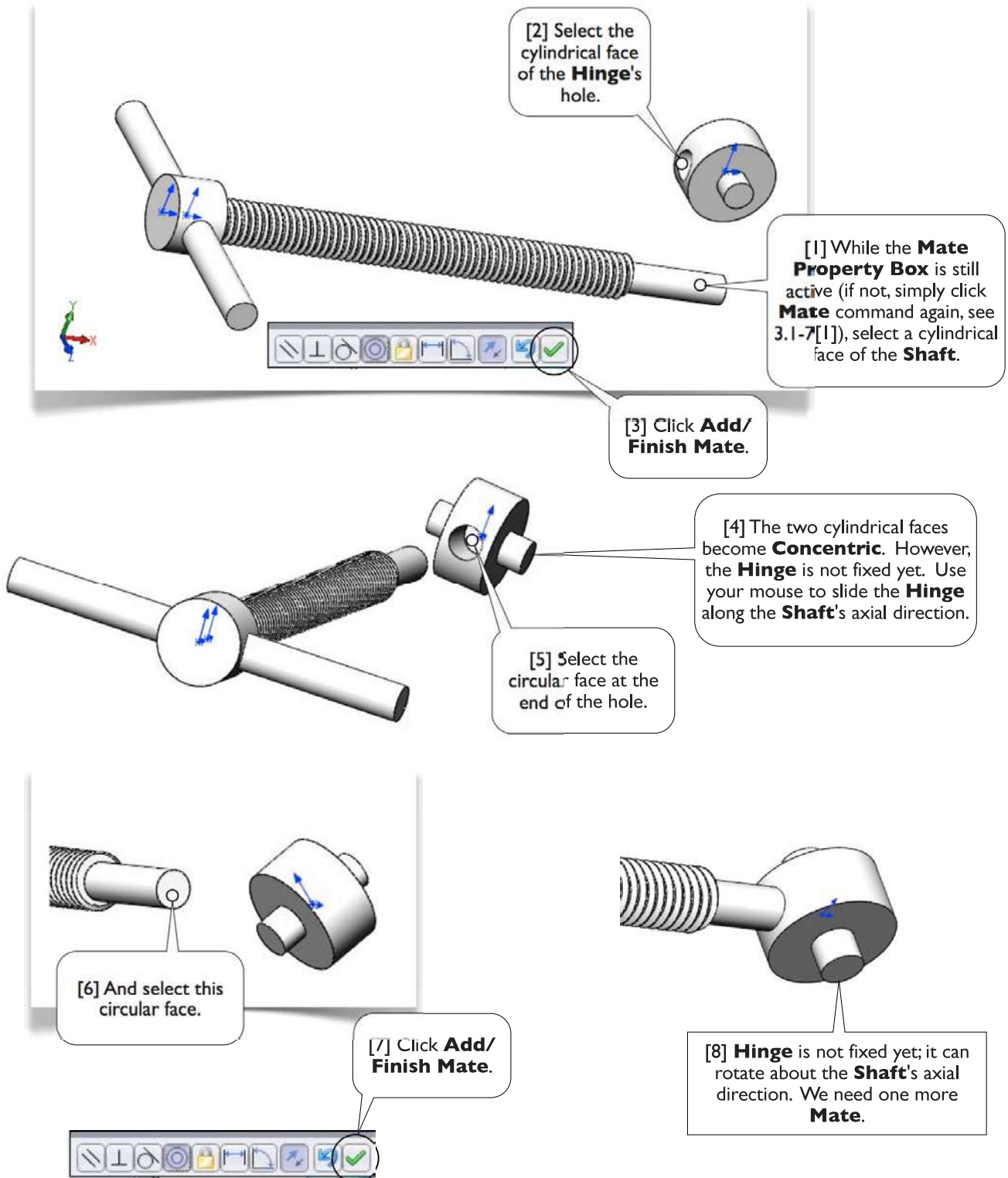
[10] A **Coincident** mate is automatically selected.

[11] Click **Add/Finish Mate**.

[12] This completes the assembly of the **Handle**.



### 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 **Add/Finish Mate**.

[12] Click **OK** to dismiss the **Mate** command.

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

[14] Save the assembly with the file name **ShaftAssembly**. The full name of the document is **ShaftAssembly.SLDASM**. Exit **SolidWorks**.