

Lecture #10

ALIGNING PARTS IN AN ASSEMBLY

This week

You will learn how to align parts in an assembly.

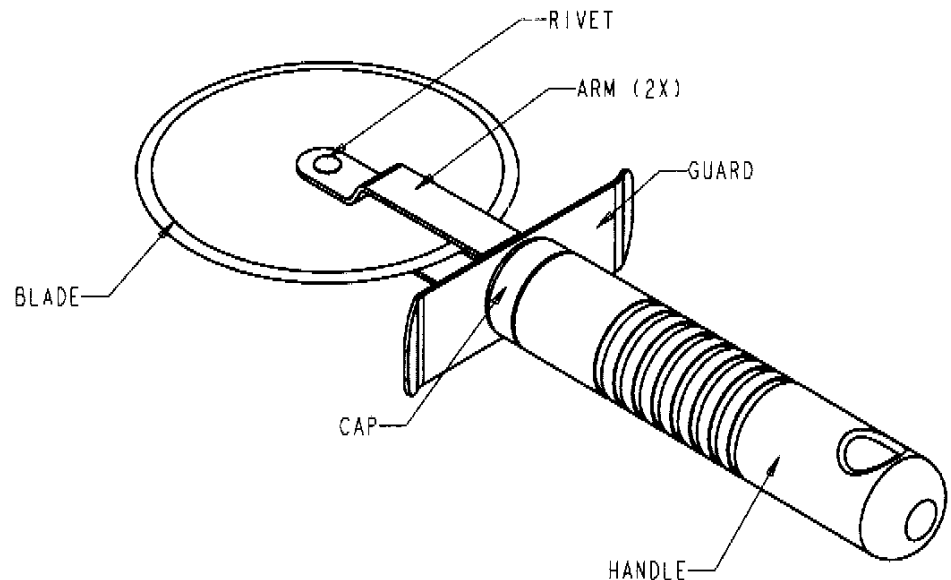
The steps to follow are:

- Introduction to assembly
- Features of assembly
- Creating assembly
- Assembly strategy
- Assembly constraints
- Aligning parts
- **Assignment # 9**

Introduction to Assembly

Assembly drawings show how the components of a design fit together dimensions and other details are usually omitted in assembly drawings to enhance clarity.

Several styles of assembly drawings are commonly used.



Introduction to Assembly

When components are added to an assembly, they are placed in their functional orientation and location.

An assembly is a document that contains two or more parts. An assembly inserted into another assembly is called a sub-assembly. A part or sub-assembly inserted into an assembly is called a component.

Features of Assembly

Assembled or exploded view

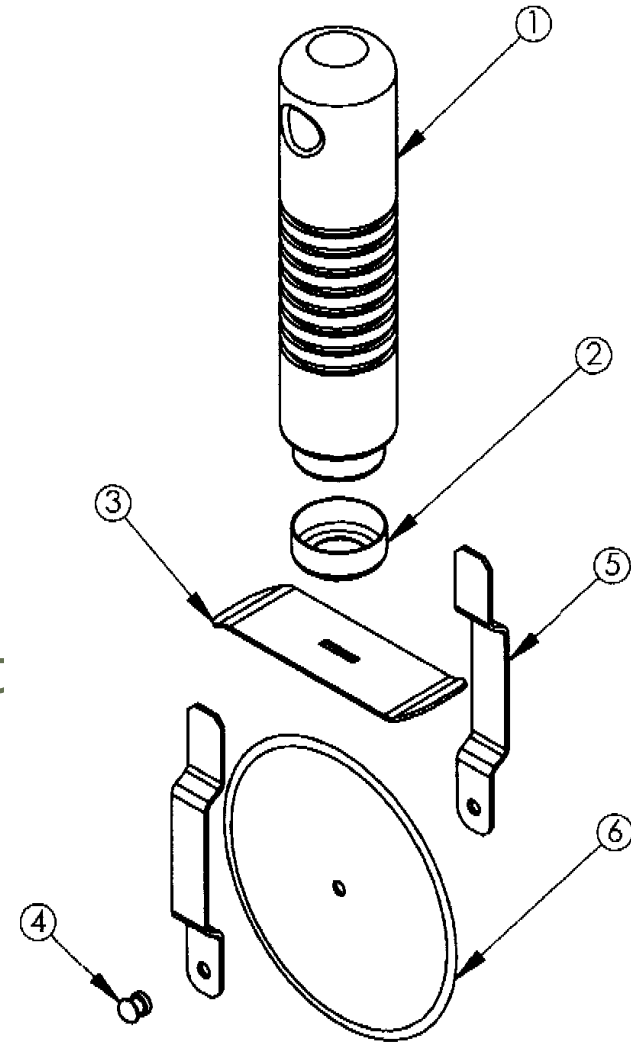
Include sections

Omitting the dimensions

Hidden-lines not necessary

Leader lines with (ballooned or not)

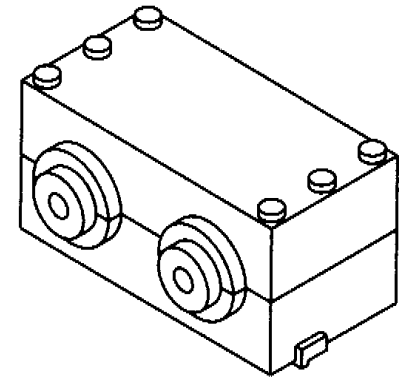
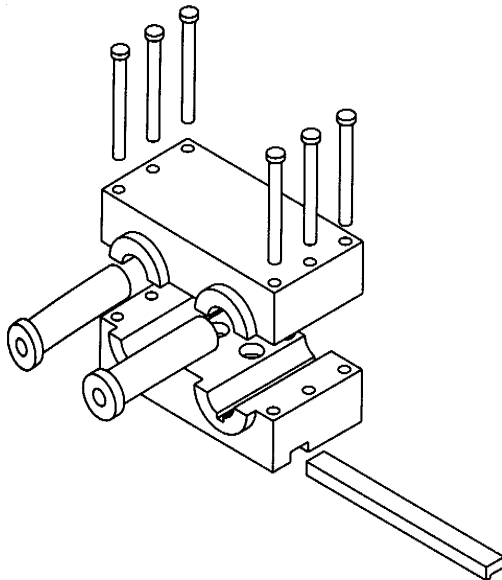
Part list (or bill of materials)



Assembly Strategy

Once again, the most important thing that you can do to successfully model an assembly is plan ahead.

Before you sit down to create an assembly, you should plan the system hierarchy on paper.



Assembly Strategy

Create all required parts in the Modeler.

Switch to the Assembly and establish your system hierarchy.

Orient the instances in the system relative to one another, applying assembly constraints as desired

Create the desired configurations for the assembly by orienting the instances.

Perform the necessary interference and clearance checks for the system.

Obtain a bill of materials for the assembly if that is desired.

Assembly Constraints

In assembly modeling, you can apply constraints between two 3-D parts so that the parts will maintain dimensional or geometric relationships with respect to one another within the assembly.

One of the more useful assembly constraints that can be applied is to define different lines on two different parts to be coincident with one another.

When a component is placed into an assembly using the Assemble option, it can be fully constrained to existing components and features.

Mates

Establishing the correct component relationship in an assembly requires forethought on component interaction.

Mates are geometric relationships that align and fit components in an assembly. Mates remove degrees of freedom from a component.

Mates reflect the physical behavior of a component in an assembly.

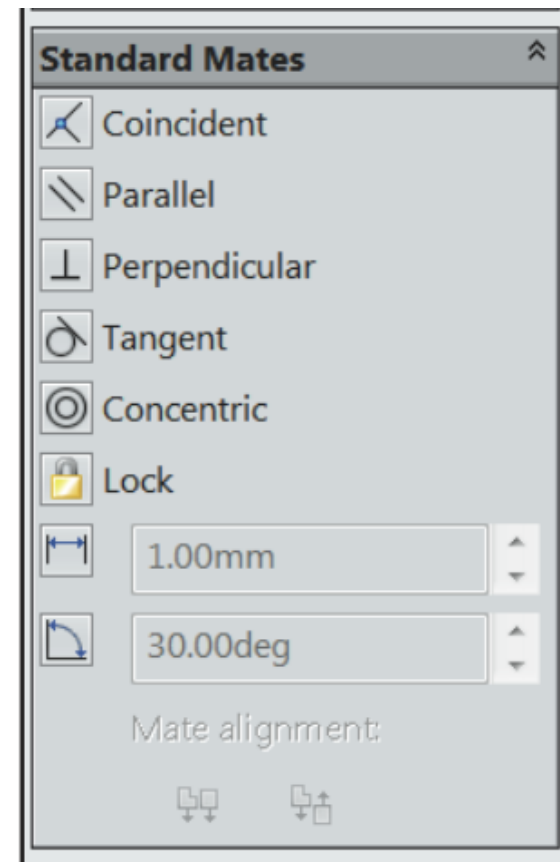
Standard, Advanced and Mechanical mate types.

Standard mates

Angle, Coincident, Concentric, Distance, Lock, Parallel, Perpendicular and Tangent mates are under **Standard Mates**.

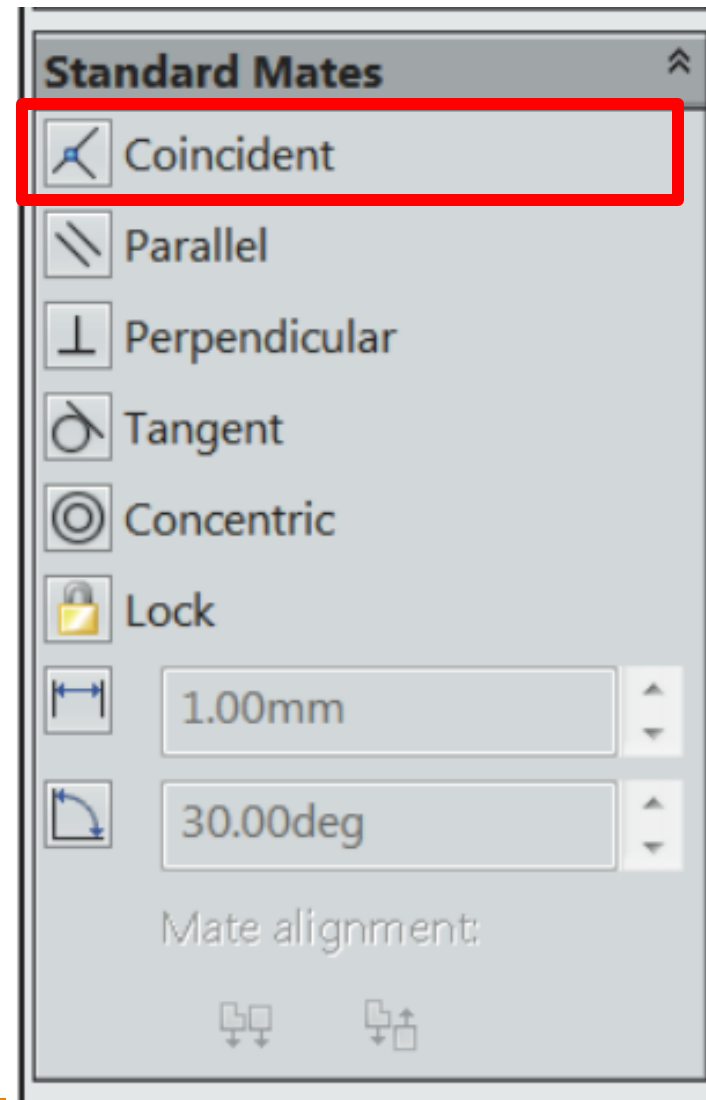
All the mate types are always shown in the **PropertyManager**, but only the mates that are applicable to the current selections are available.

Works with axis; planes; corners; vertex and surfaces.



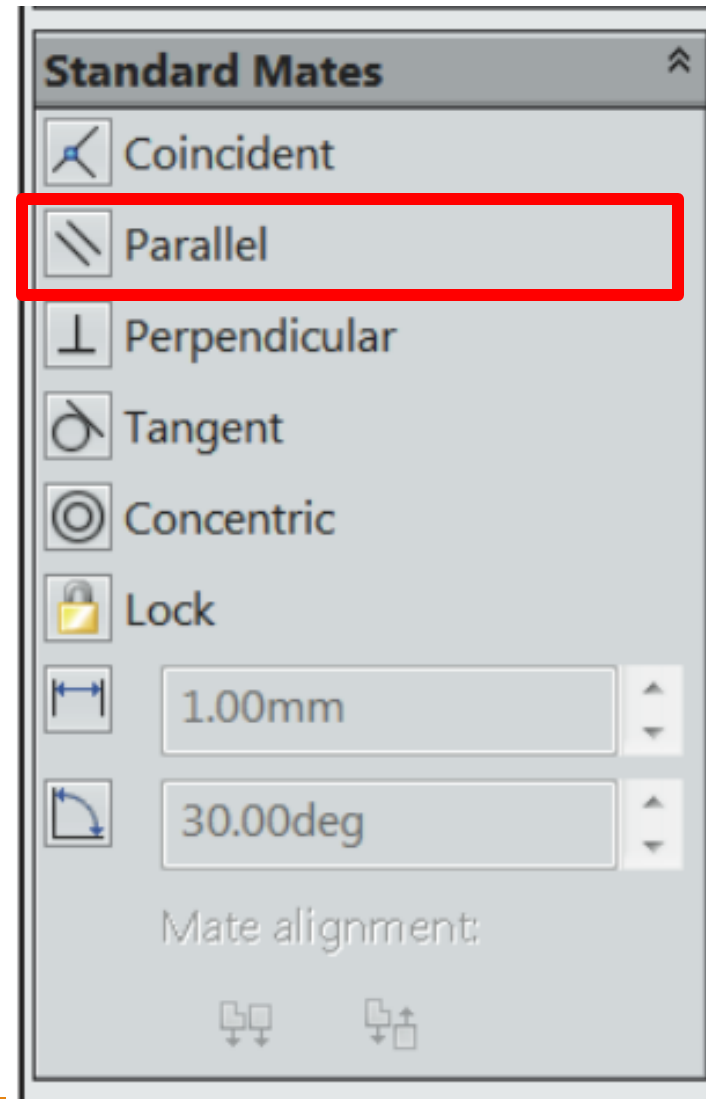
Standard mate - Coincident

Locates the selected faces, edges, or planes so they use the same infinite line. A coincident mate positions two vertices for contact.



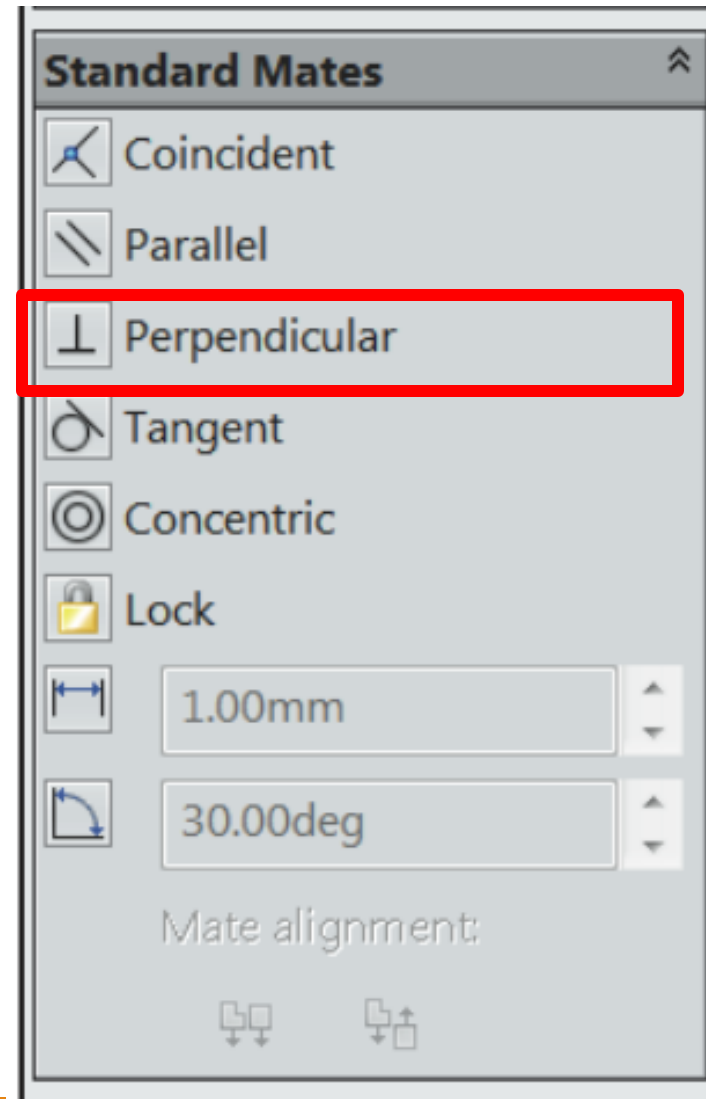
Standard mate - Parallel

Locates the selected items to lie in the same direction and to remain a constant distance apart.



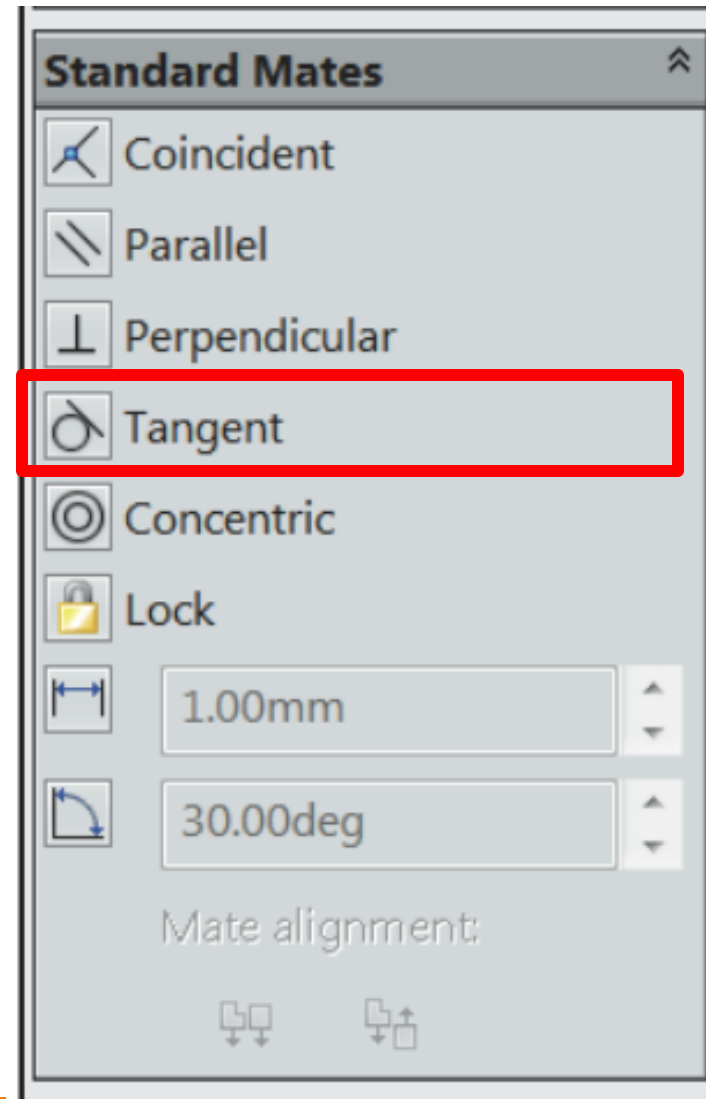
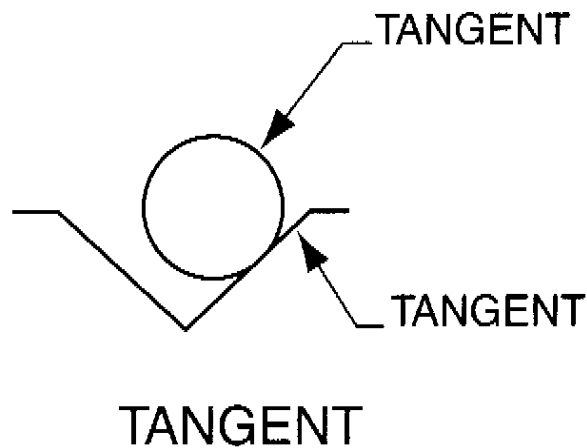
Standard mate - Perpendicular

Locates the selected items at a 90 degrees angle to each other.



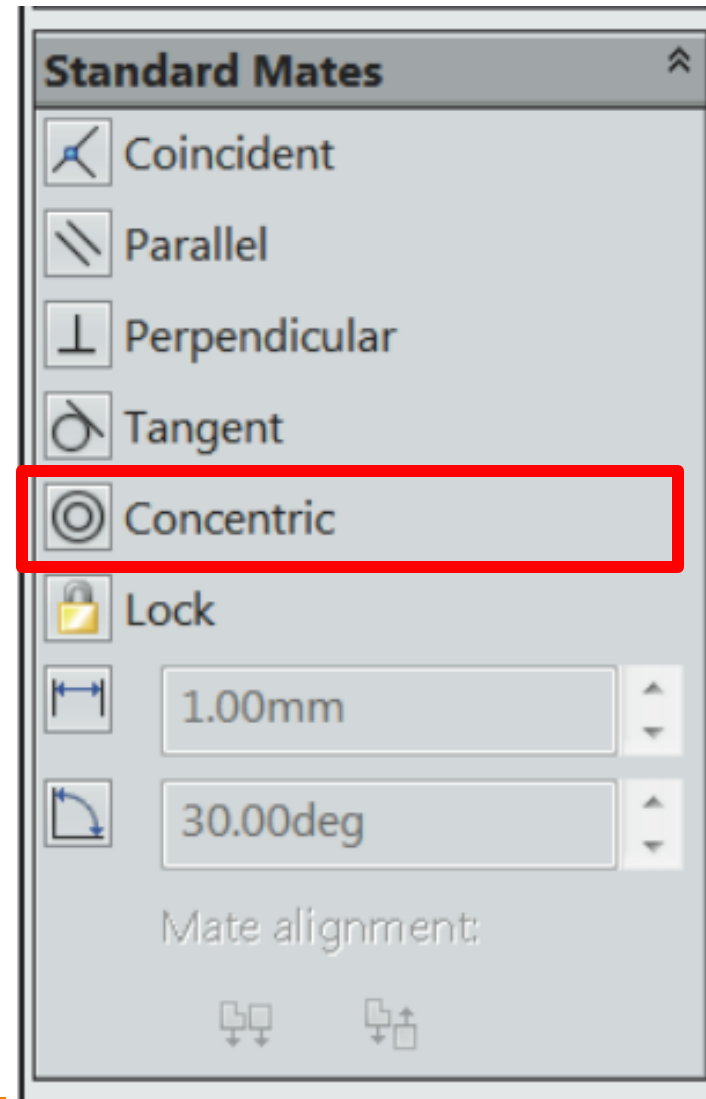
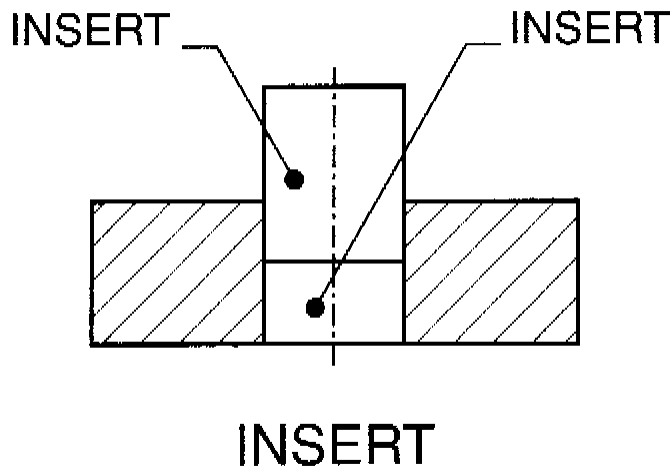
Standard mate - Tangent

Locates the selected items in a tangent mate. At least one selected item must be either a conical, cylindrical, spherical face.



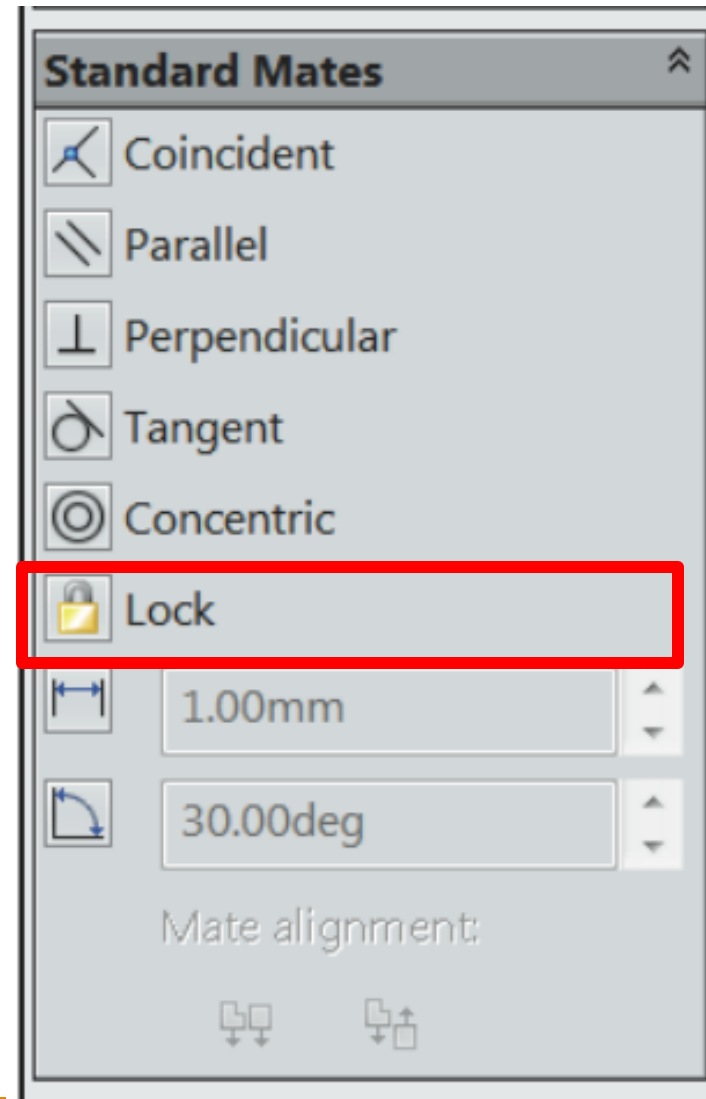
Standard mate - Concentric

Locates the selected items so they can share the same center point.



Standard mate - Lock

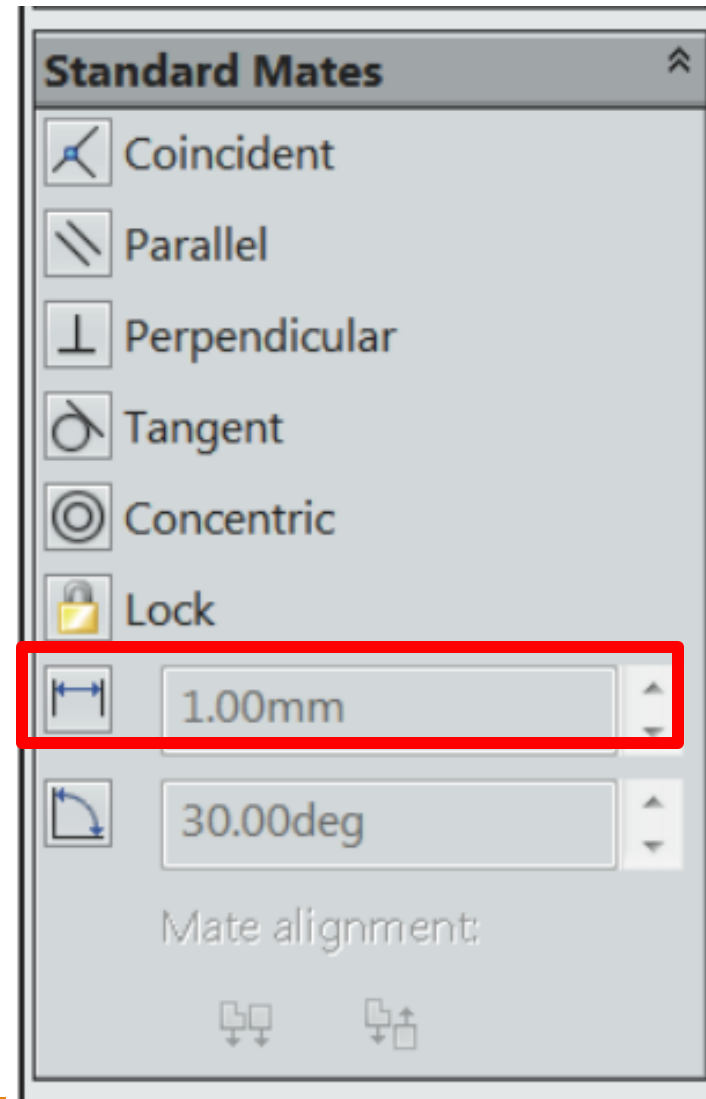
Maintains the position and orientation between two components.



Standard mate - Distance

Locates the selected items with a specified distance between them.

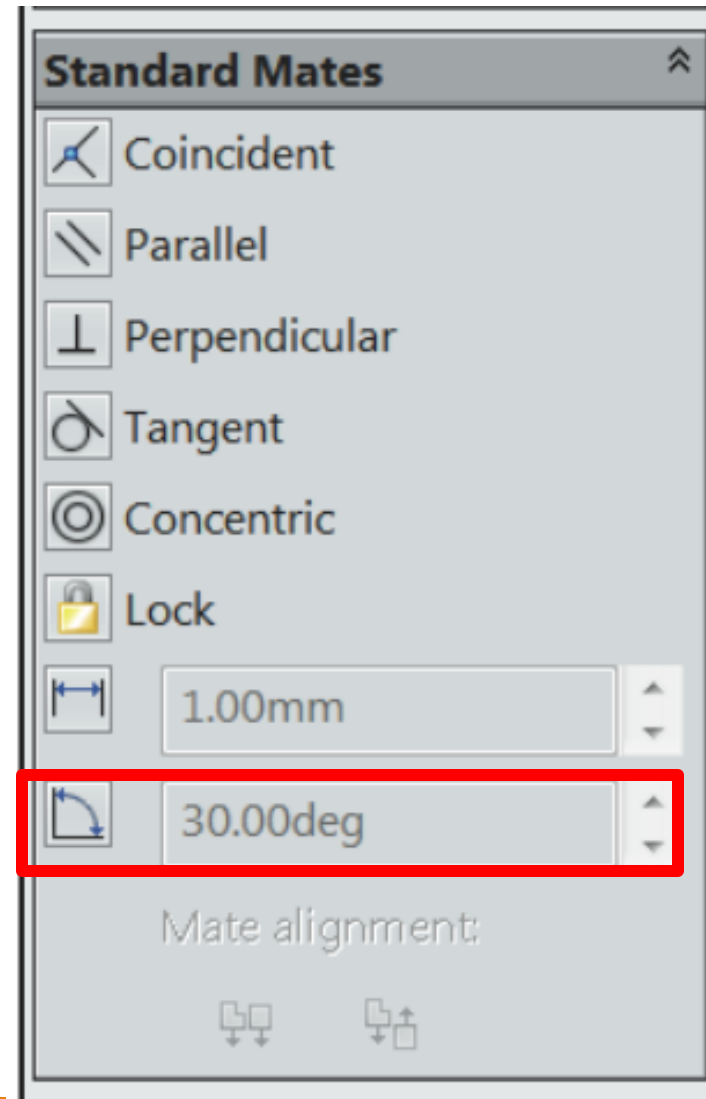
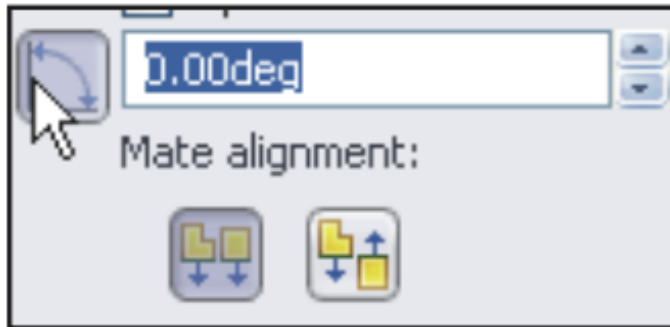
Use the drop-down arrow box or enter the distance value directly.



Standard mate - Distance

Locates the selected items at the specified angle to each other.

Use the drop-down arrow box or enter the angle value directly.

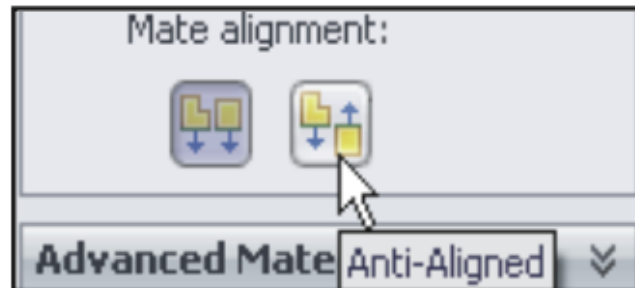


Mate alignment

There are two Mate Alignment options :

The **Aligned** option positions the components so that the normal vectors from the selected faces point in the same direction.

The **Anti-Aligned** option positions the components so that the normal vectors from the selected faces point in the opposite direction.

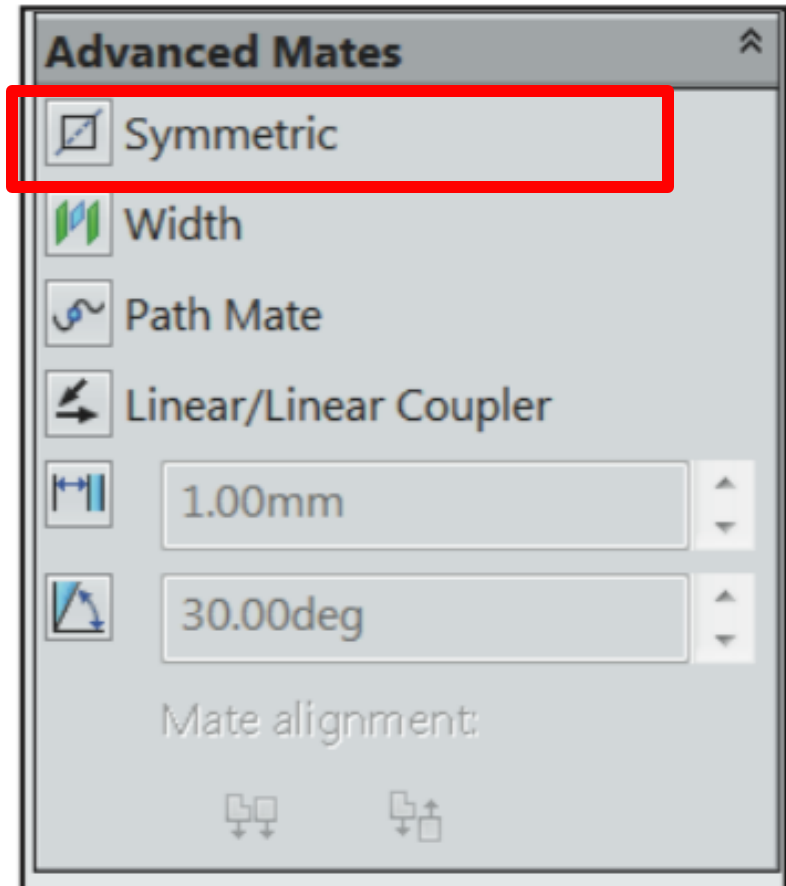


Advanced mates

Symmetric mate :

Positions two selected entities to be symmetric about a plane or planar face.

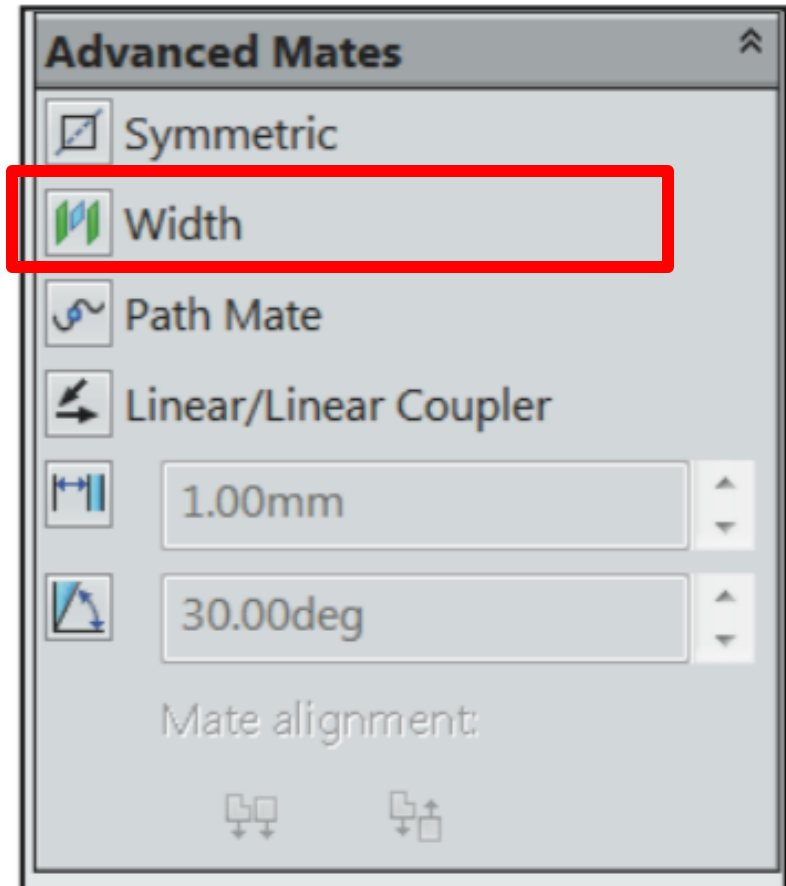
A Symmetric Mate does not create a Mirrored Component.



Advanced mates

Width mate :

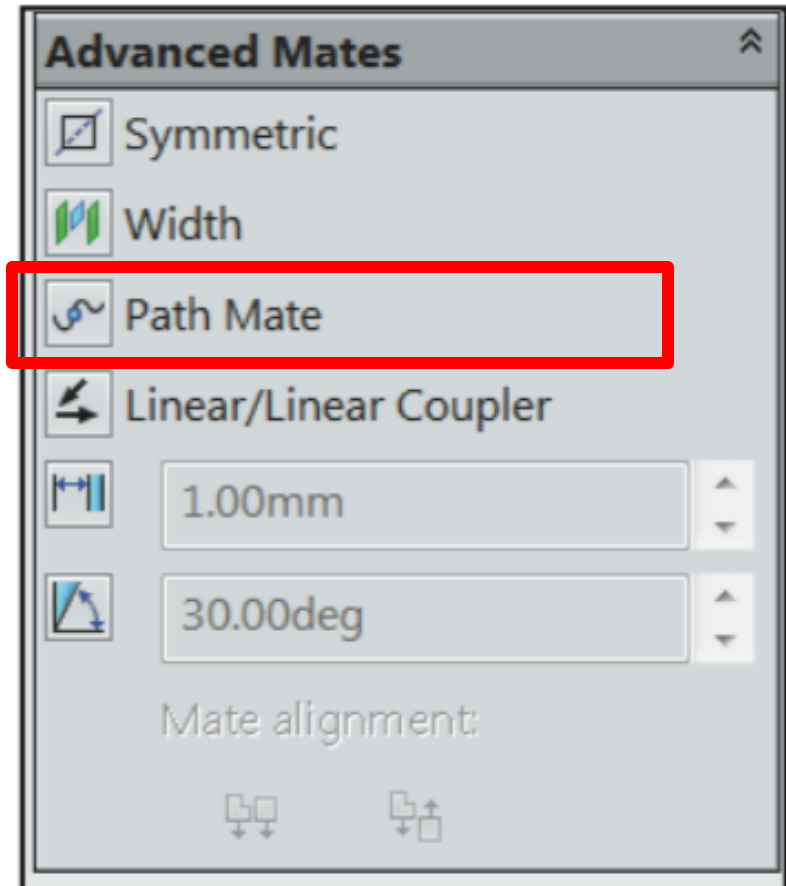
Centers a tab within the width of a groove



Advanced mates

Path mate :

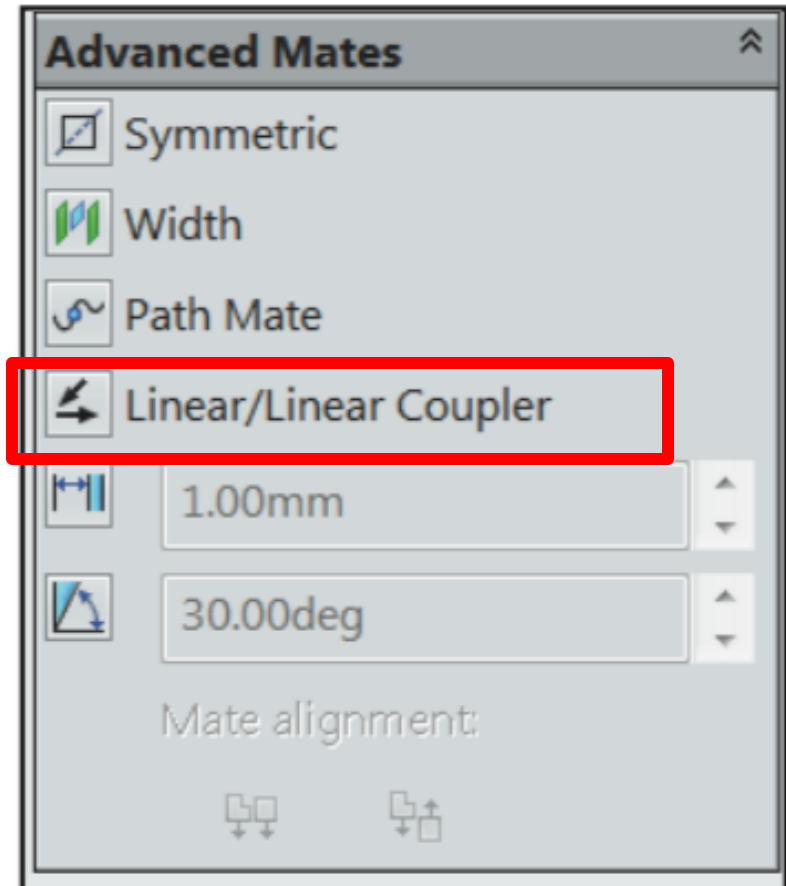
Constraint a selected point on a component to a path.



Advanced mates

Linear/Linear Coupler mate :

Establishes a relationship between the translation of one component and the translation of another component.

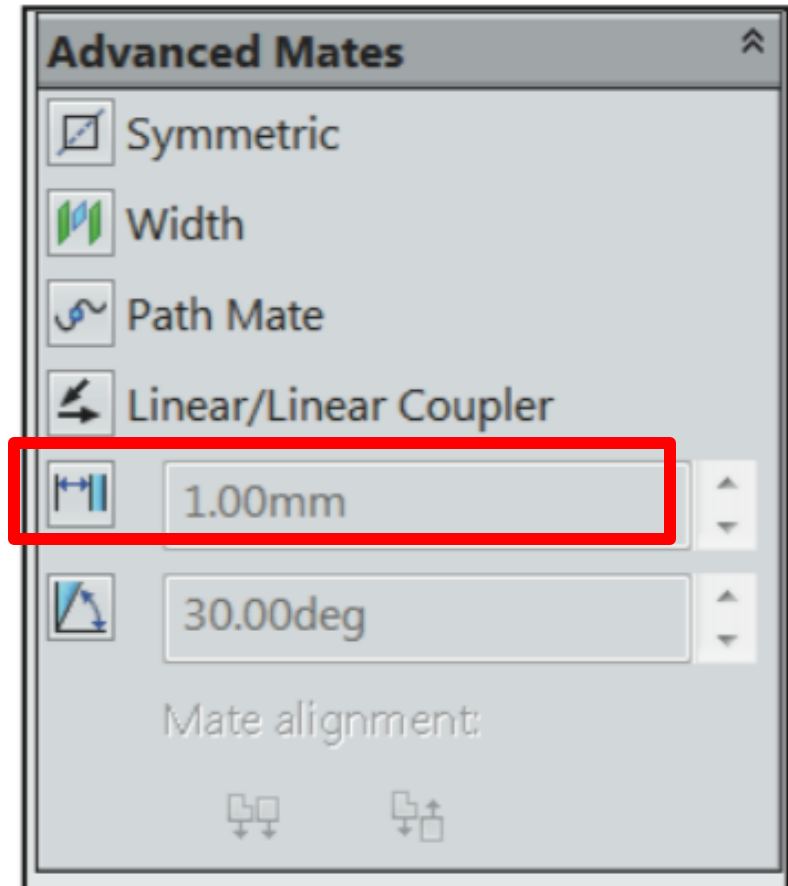


Advanced mates

Distance (limit) mate :

Locates the selected items with a specified distance between them

Use the drop-down arrow box or enter the distance value directly.

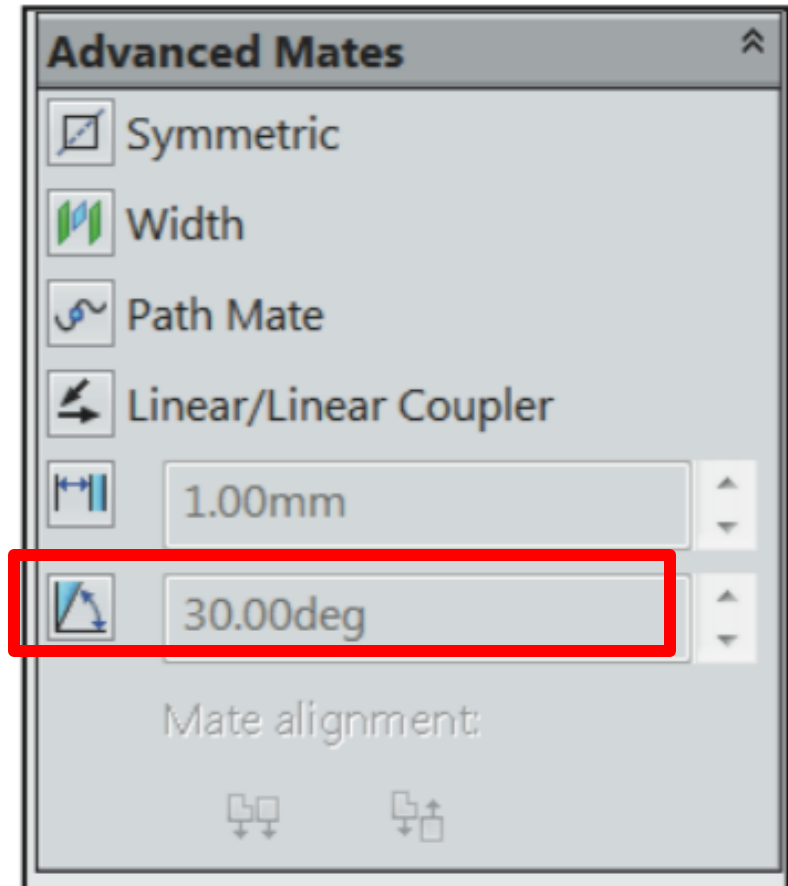


Advanced mates

Angle mate :

Locates the selected items with a specified angle to each other.

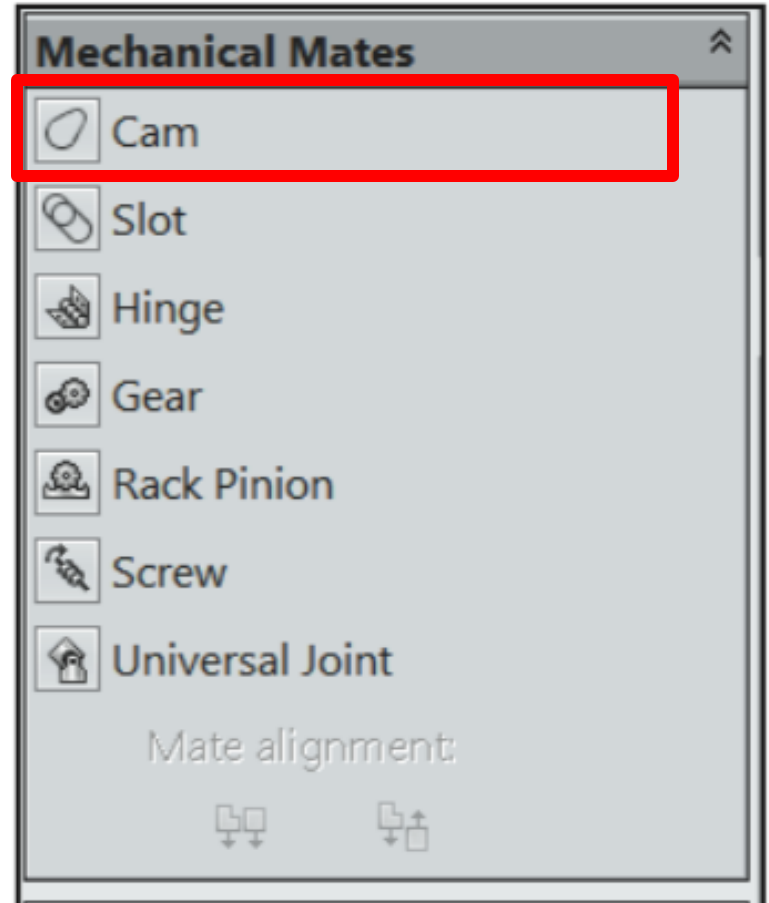
Use the drop-down arrow box or enter the angle value directly.



Mechanical mates

Cam mate :

Forces a plane, cylinder or point to be tangent or coincident to a series of tangent extruded faces.

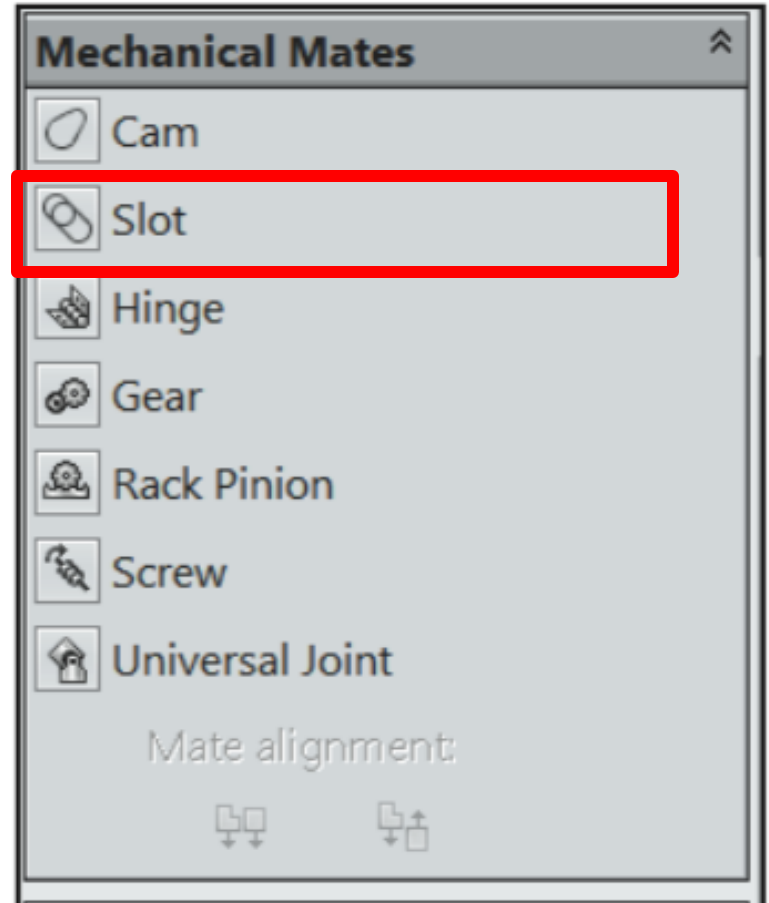


Mechanical mates

Slot mate :

Mate bolts to straight or arced slots and you can mate slots to slots.

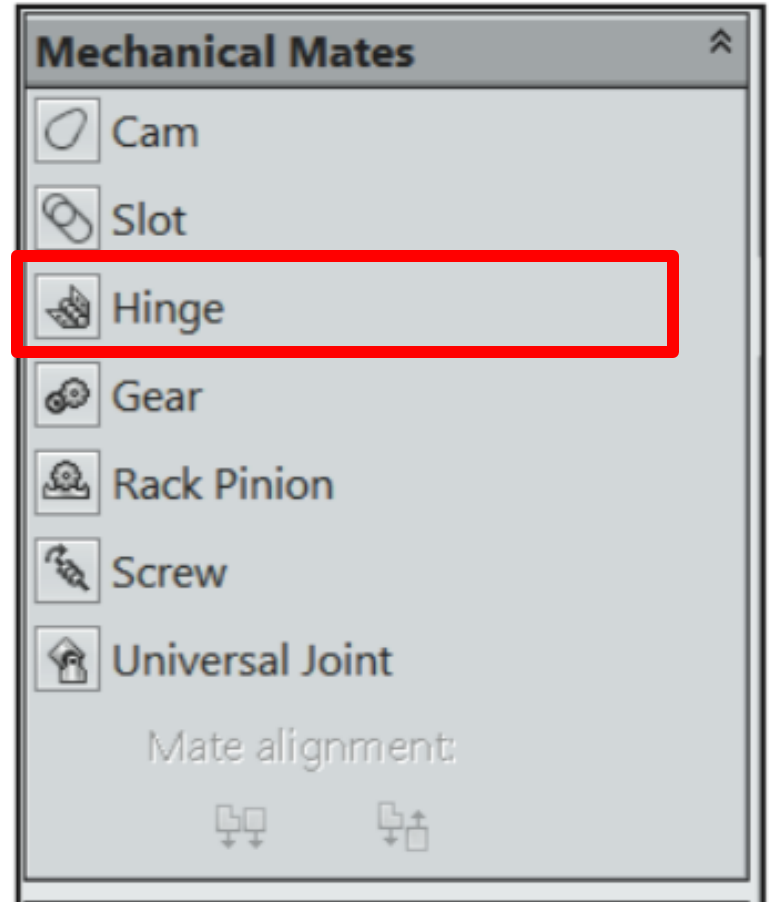
Selected an axis, cylindrical face, or a slot to create slot mates.



Mechanical mates

Hinge mate :

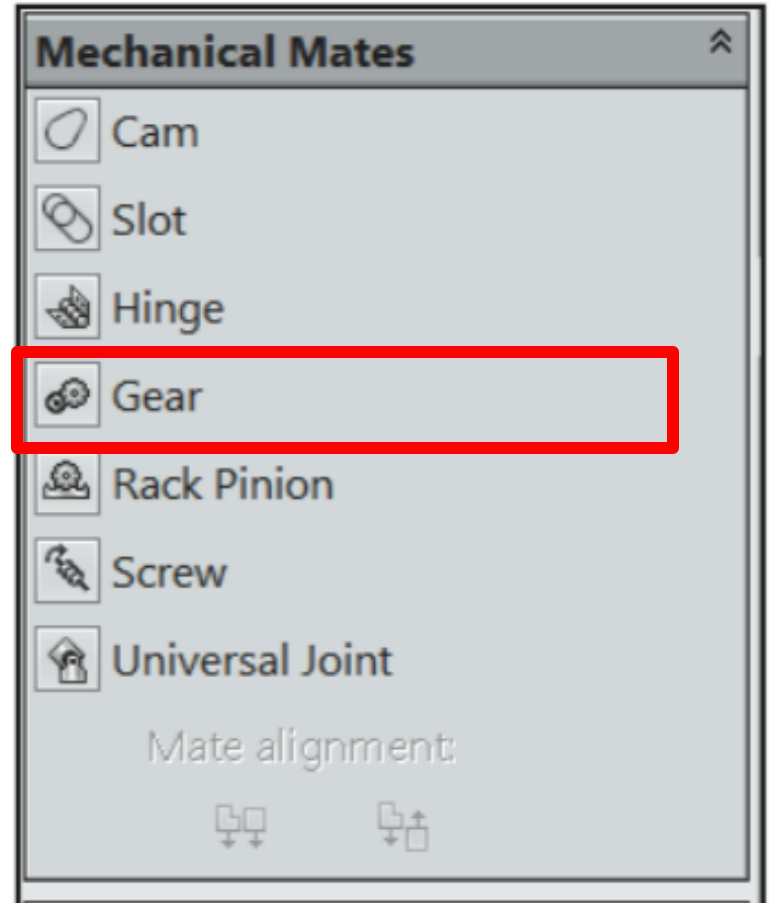
Limits the movement between two components to one rotational degree of freedom.



Mechanical mates

Gear mate :

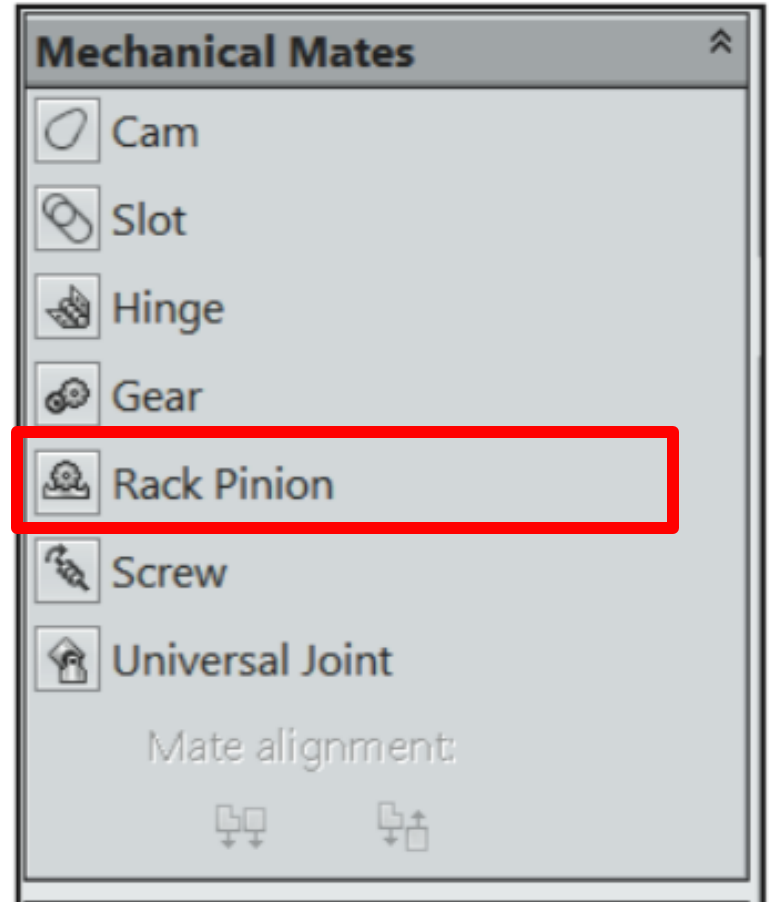
Forces two components to rotate relative to one another around selected axes.



Mechanical mates

Rack Pinion mate :

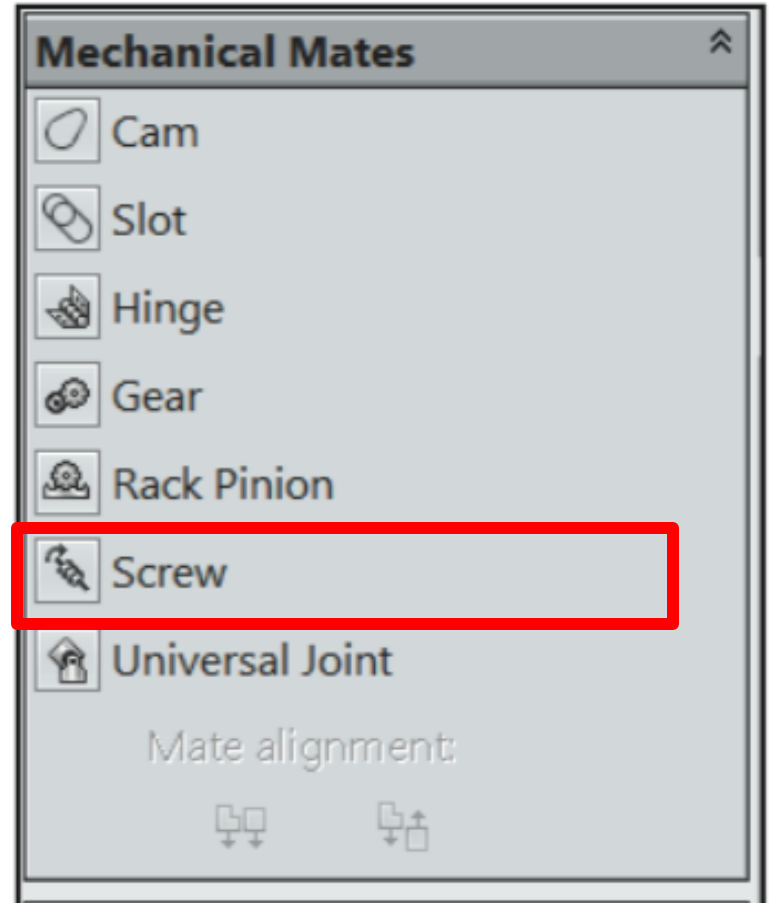
Provides the ability to have linear translation of a part, rack causes circular rotation in another around selected axes.



Mechanical mates

Screw mate :

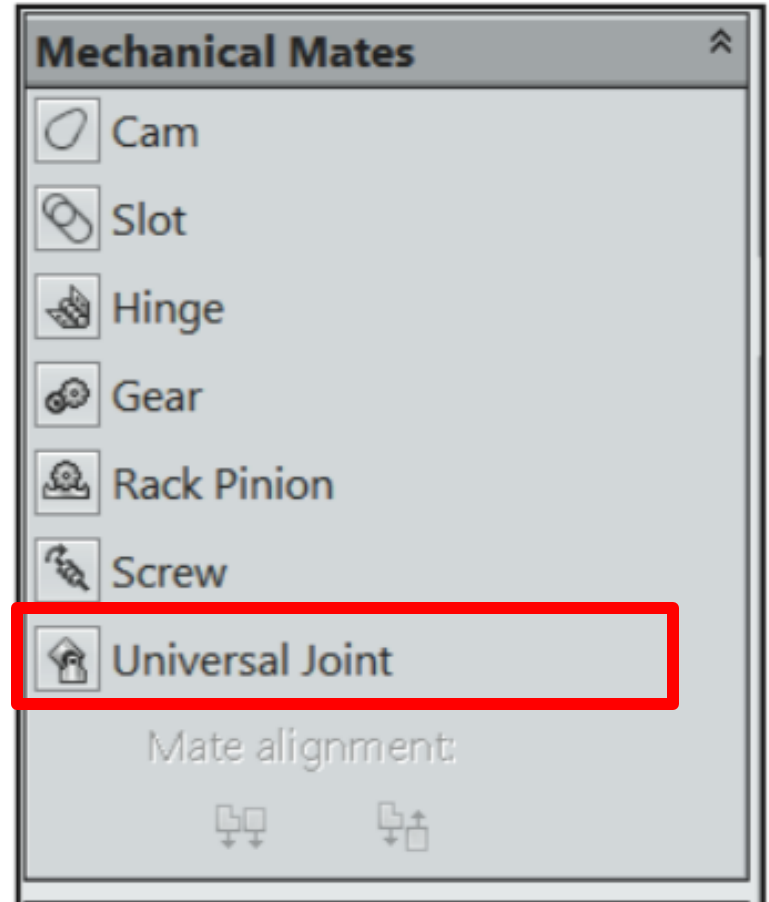
Constrains two components to be concentric and adds a pitch relationship between the rotation of one component and the translation of the other.



Mechanical mates

Universal Joint mate :

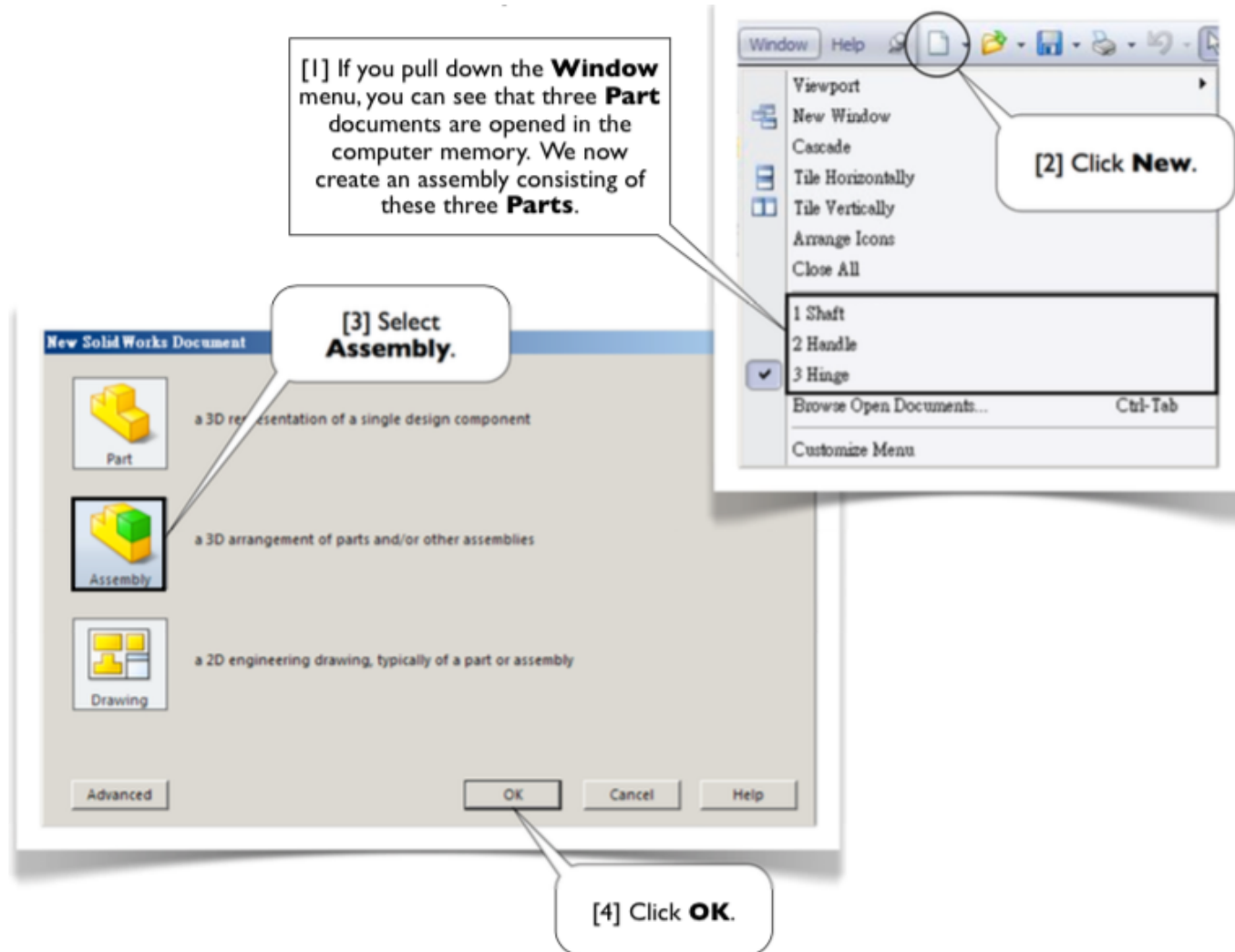
The rotation of one component (**the output shaft**) about its axis is driven by the rotation of another component (**the input shaft**) about its axis.



Aligning Parts

SOLIDWORKS 2014

Create a New Assembly



Create a New Assembly

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

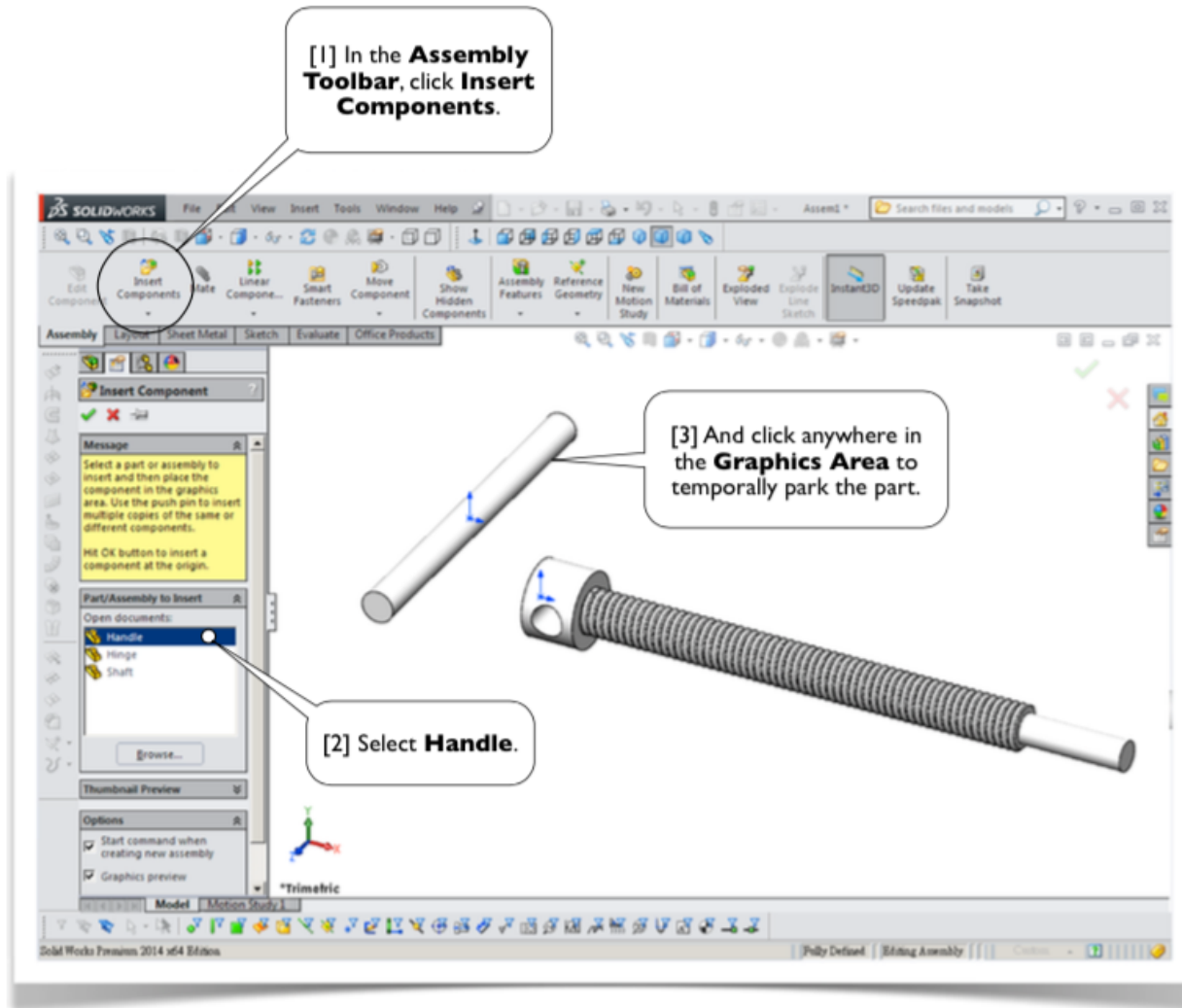
[6] This is the origin of the new assembly. We now insert the **Shaft** so that the part's coordinate system aligns with the assembly's coordinate system.

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

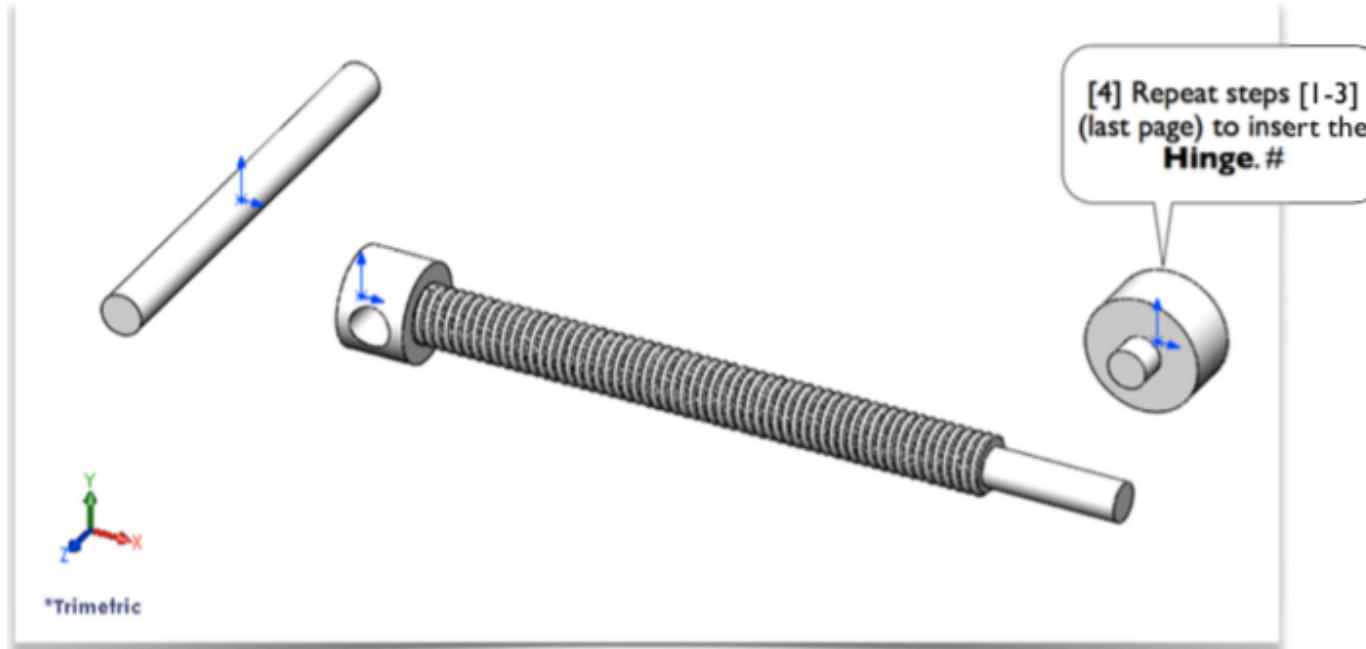
[8] Click the assembly's origin. The **Shaft** is inserted and its coordinate system aligns with the assembly's coordinate system. #

*Trimetric

Insert the other components



Insert the other components



Assemble Handle

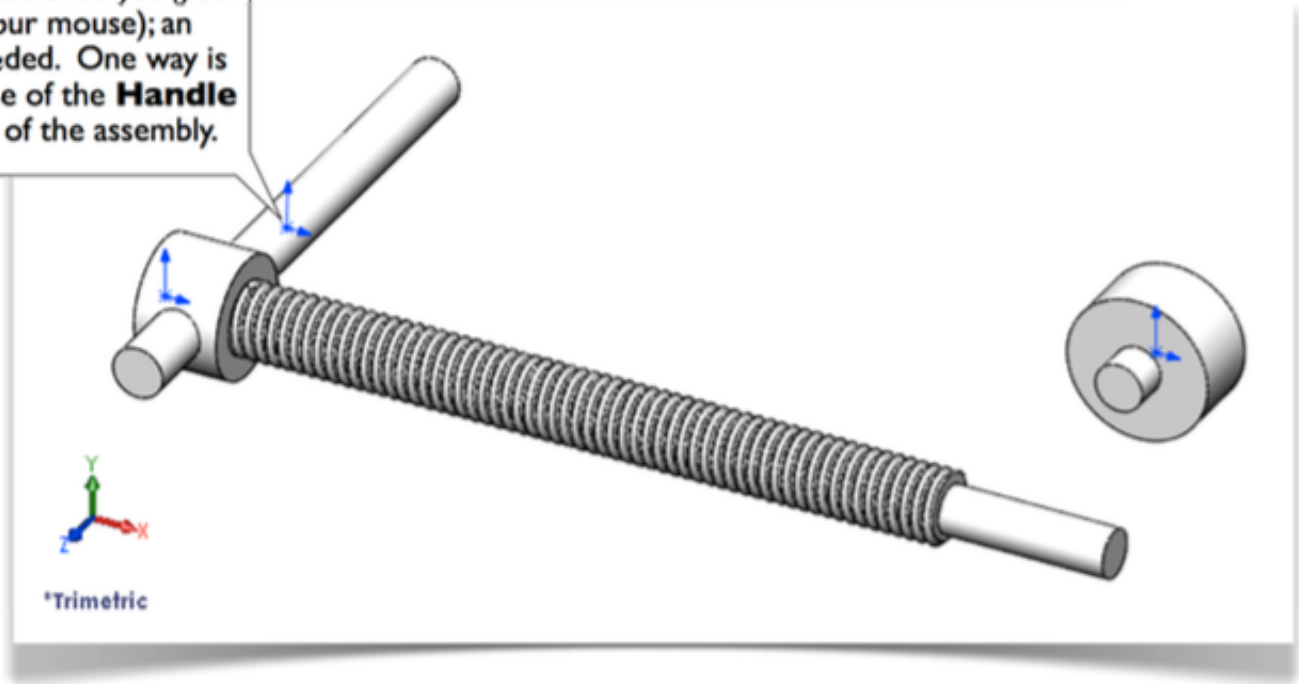
The image shows a CAD software interface with several callouts explaining the steps to create a concentric mate between a handle and a shaft.

- [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 **OK** to accept the **Concentric** mate without dismissing the **Property Box**.

The interface includes a **Concentric1** property box with a green checkmark, a **Mate Selections** list showing "Face<1>@Handle-1" and "Face<2>@Shaft", and a **Standard Mates** list where "Concentric" is selected. The **Assembly** toolbar shows the **Mate** button circled. A 3D model shows a handle and a shaft with blue arrows indicating the selection of their cylindrical faces. A coordinate system labeled ***Trimetric** is also visible.

Assemble Handle

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



Assemble Handle

[8] In the **Part 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.

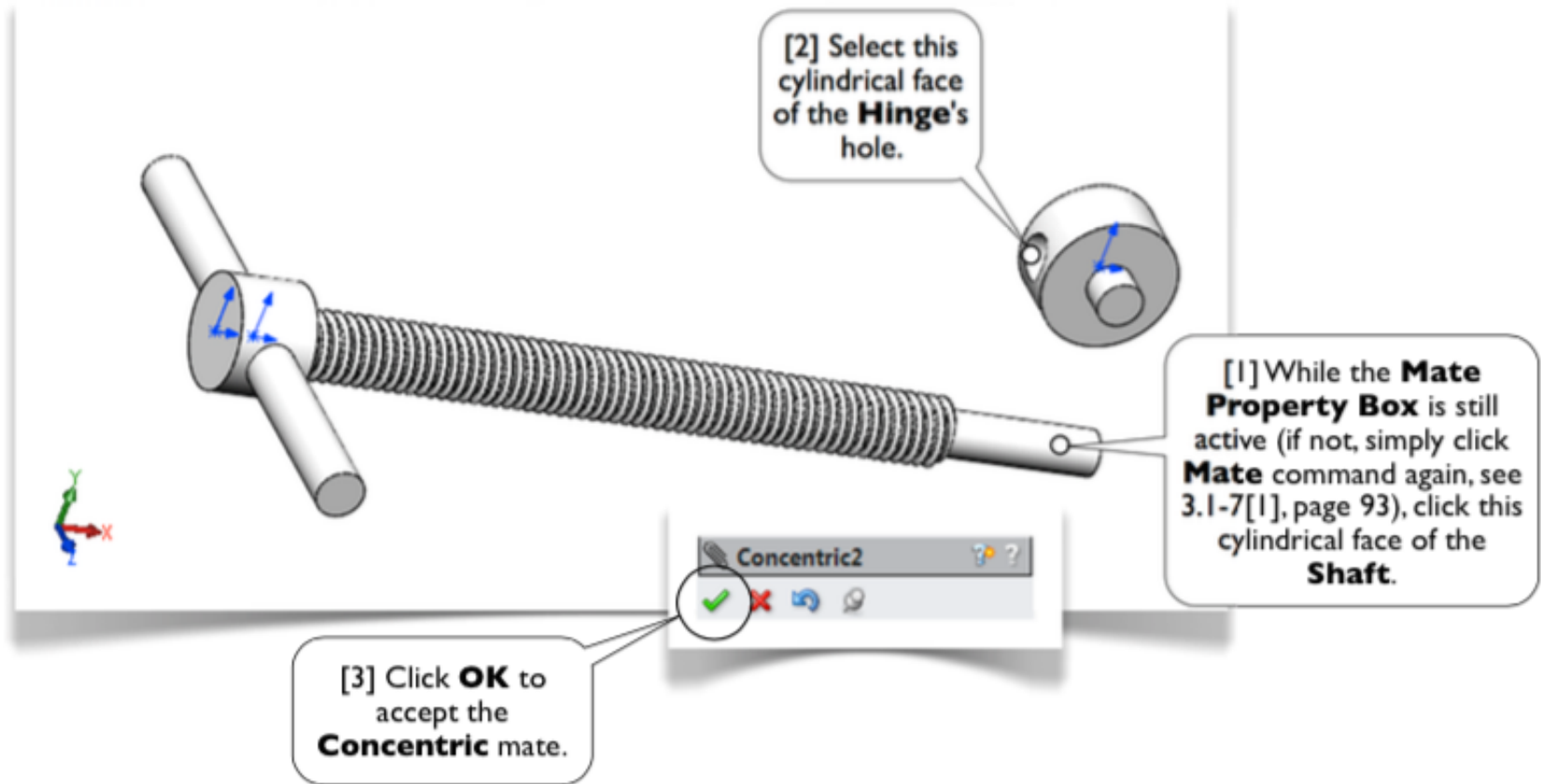
[10] A **Coincident** mate is automatically selected.

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

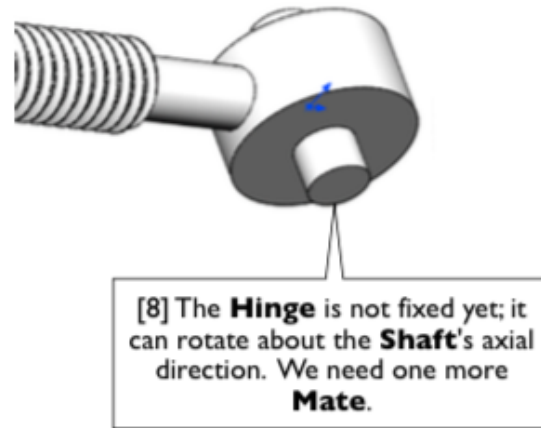
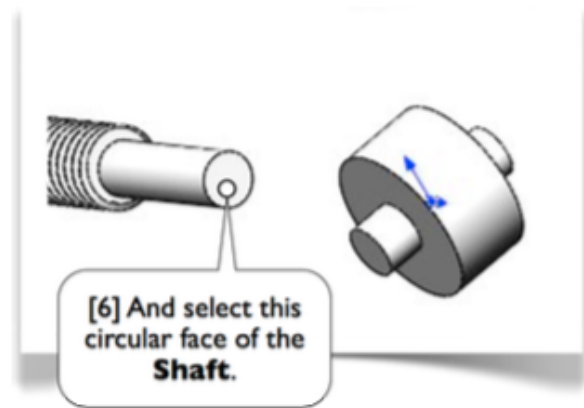
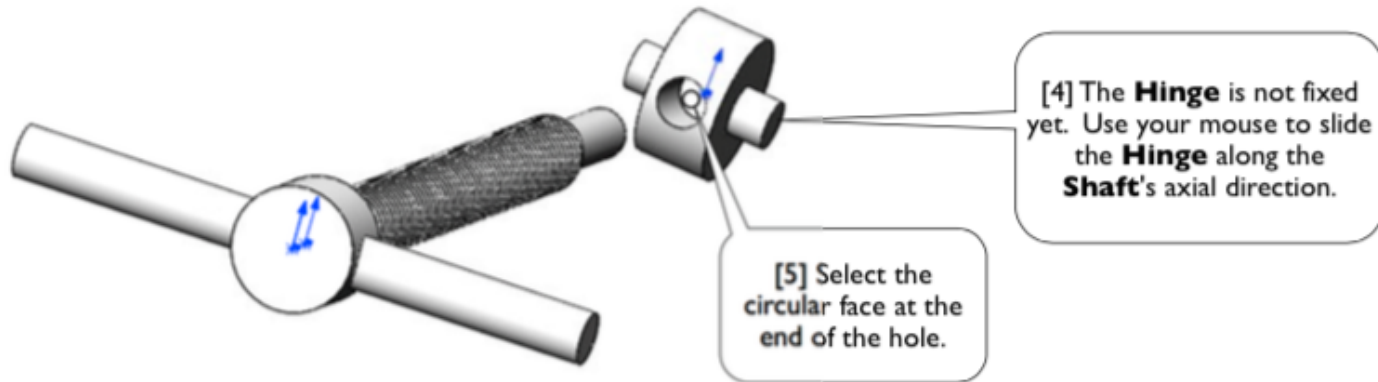
[12] This completes the assembly of the **Handle**. The handle still can rotate about its axis, however, we neglect this deficiency. #

The image shows a CAD software interface. On the left is the Part Tree, which is expanded to show the 'Front' plane of the 'Handle' component. In the center is the Property Box, which shows a 'Coincident' mate selected between the 'Front' plane of the 'Handle' and the 'Front' plane of the assembly. On the right is a 3D model of the assembly, showing a handle with a threaded shaft inserted through it. The handle is shown in a perspective view, and the shaft is shown in a top-down view. The handle has a cylindrical body with a threaded section. The shaft is a long, thin cylinder with a threaded section at one end. The handle is shown in a perspective view, and the shaft is shown in a top-down view. The handle is shown in a perspective view, and the shaft is shown in a top-down view. The handle is shown in a perspective view, and the shaft is shown in a top-down view.

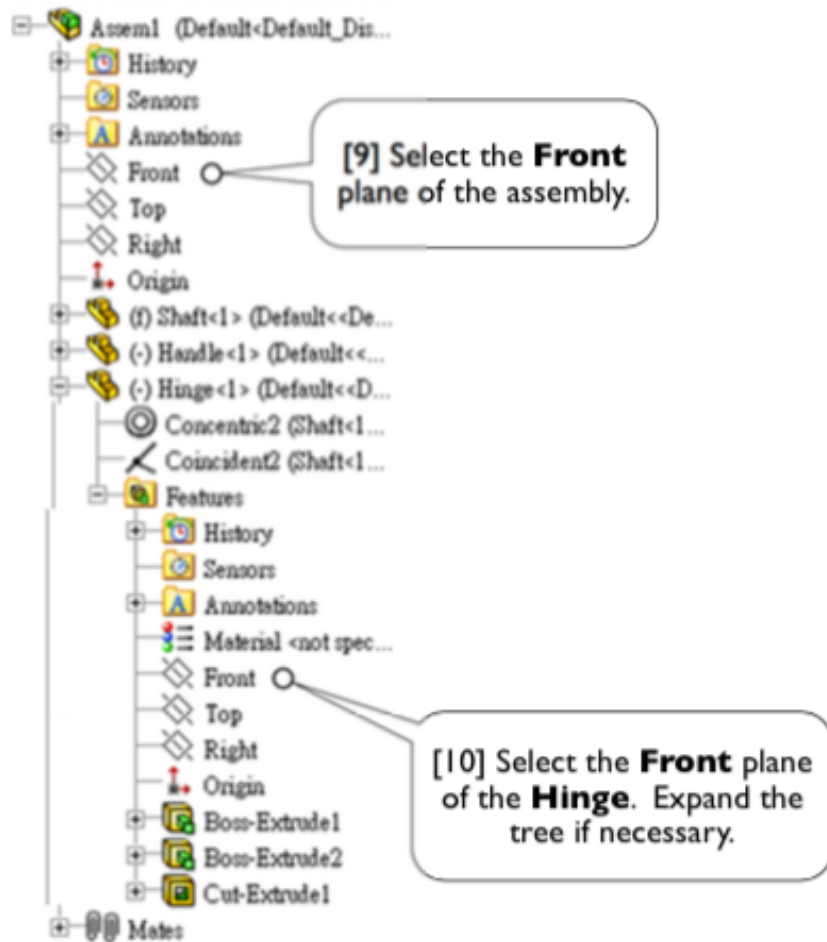
Assemble Hinge



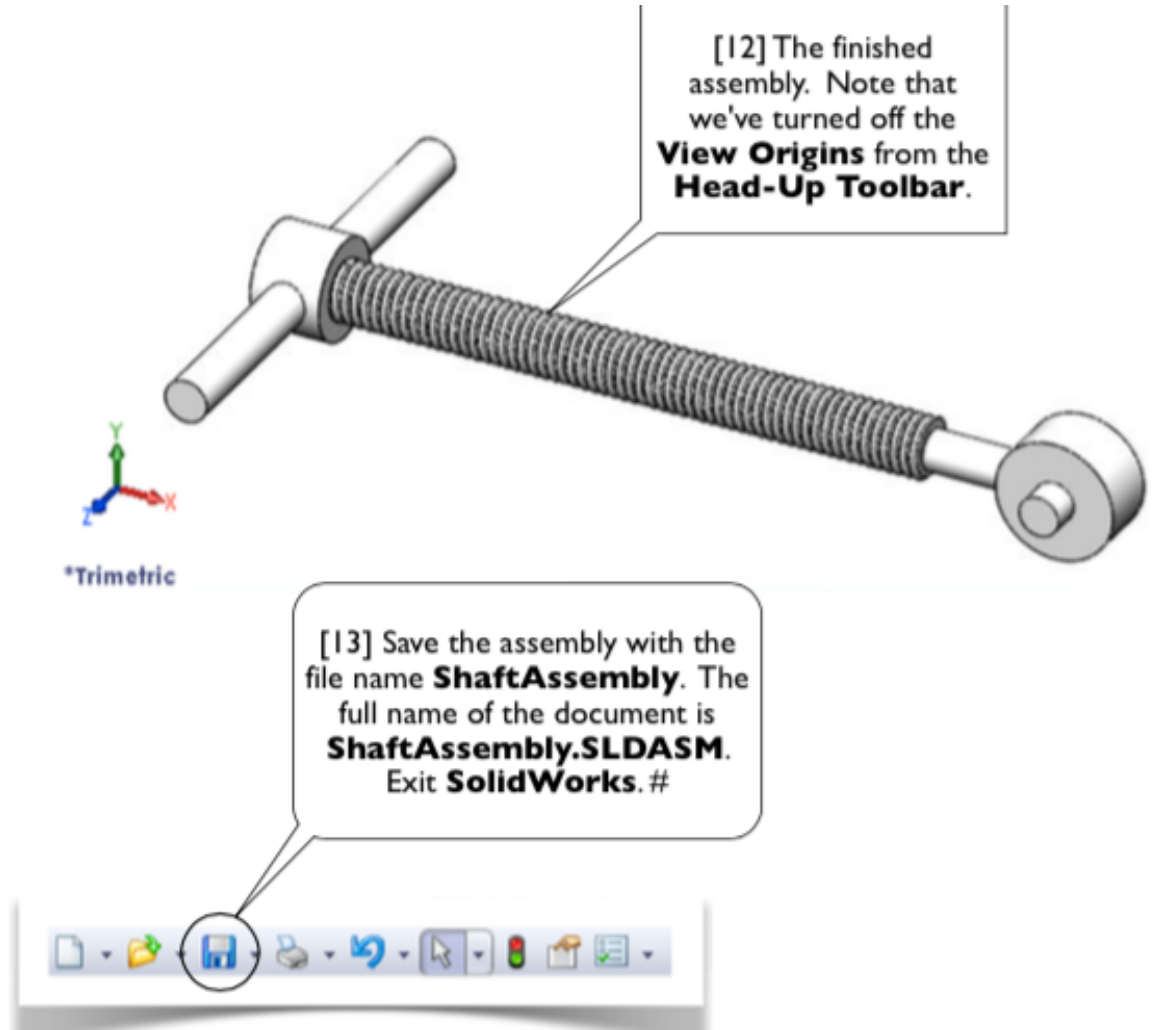
Assemble Hinge



Assemble Hinge











Assemble Hinge



For more [details](#) ...

SolidWorks

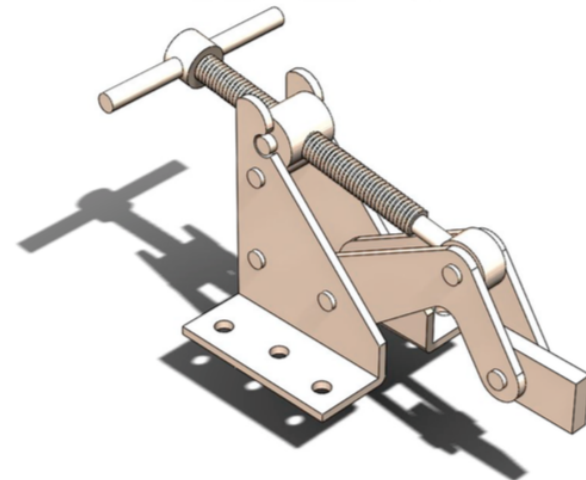
-  SolidWorks 2001 Version Tutorial
PDF, 873,86 KB
-  SolidWorks 2003 Version Tutorial
PDF, 4,40 MB
-  SolidWorks 2004 Version Tutorial
PDF, 3,87 MB
-  SolidWorks 2005 Version Tutorial
PDF, 182,60 KB
-  Solidworks 2014 Version Tutorial
PDF, 7,42 MB
-  Solidworks 2014 Part
PDF, 6,42 MB
-  Solidworks 2014 Drawing
PDF, 7,66 MB
-  Solidworks 2014 Assembly
PDF, 49,46 MB



Part and Assembly Modeling

with SolidWorks 2014

Huei-Huang Lee



The following week

You will learn assembly modeling and assembling parts.

The steps to follow are:

- Assembly drawing
- Assembly section
- Bill of materials
- Exploded assembly
- Animation
- **Assignment # 10**



The following week ...

week
13

DON'T FORGET

Homework #4

Due to : 13th course week

Complete the assembly by adding all the components of your design project and draw the following views

- use your judgment for the scale and selection of views
- make a two-view assembly drawing including a sectional view
- include on the drawing a bill of materials and identify each part on the assembly



<http://transport.itu.edu.tr/dersler/lisansdersleri/res112e/homeworks>

Assignment #9

Page

Figure

You will build the assembly model

Submit the assignment on time

Upload file into NINOVA

