CHƯƠNG 3: BẢN VẼ LẮP
ASSEMBLY MODELING

Instructor SolidWork 2013:

Nguyễn Tấn Ý

The Assembly- Bản vẽ lắp

Lession create a new assembly. (Bài học tạo một Assembly mới)

■ **Insert components into an assembly.**

- Chèn chi tiết vào môi trường assembly

■ **Add mating relationshipsbetween components.**

- Thiết lập các mối liên kế giữa các chi tiết với nhau

■ **FeatureManager design**

- Tính năng quản lý thiết kế

■ **Insert sub-assemblies.**

- Chèn các bản vẽ lắp con

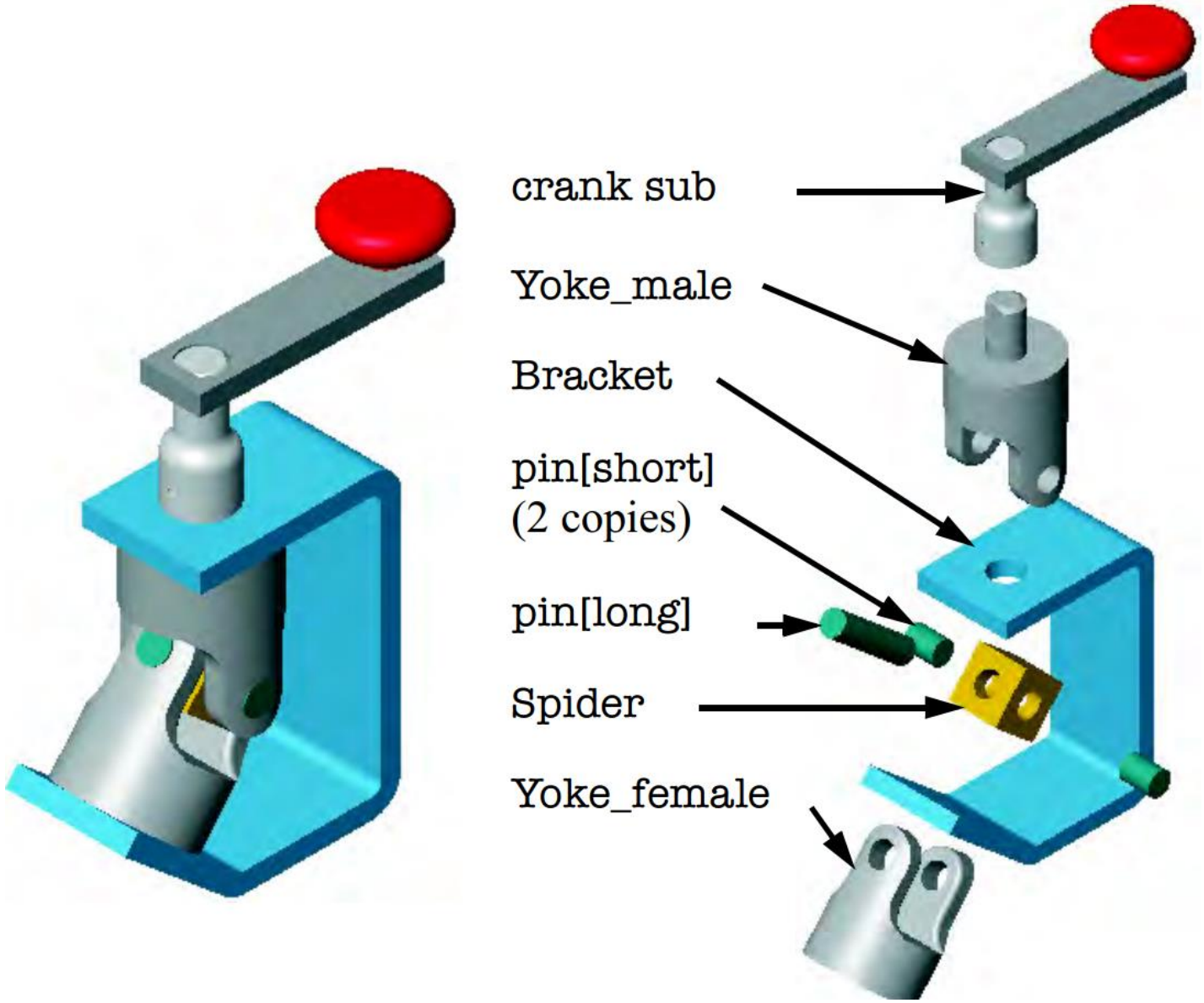
■ **Use part configurations in an assembly.**

-Sử dụng chi chức năng cấu hình chi tiết trong bản vẽ lắp

Tham khảo: ***SolidWork Essentials 2011***

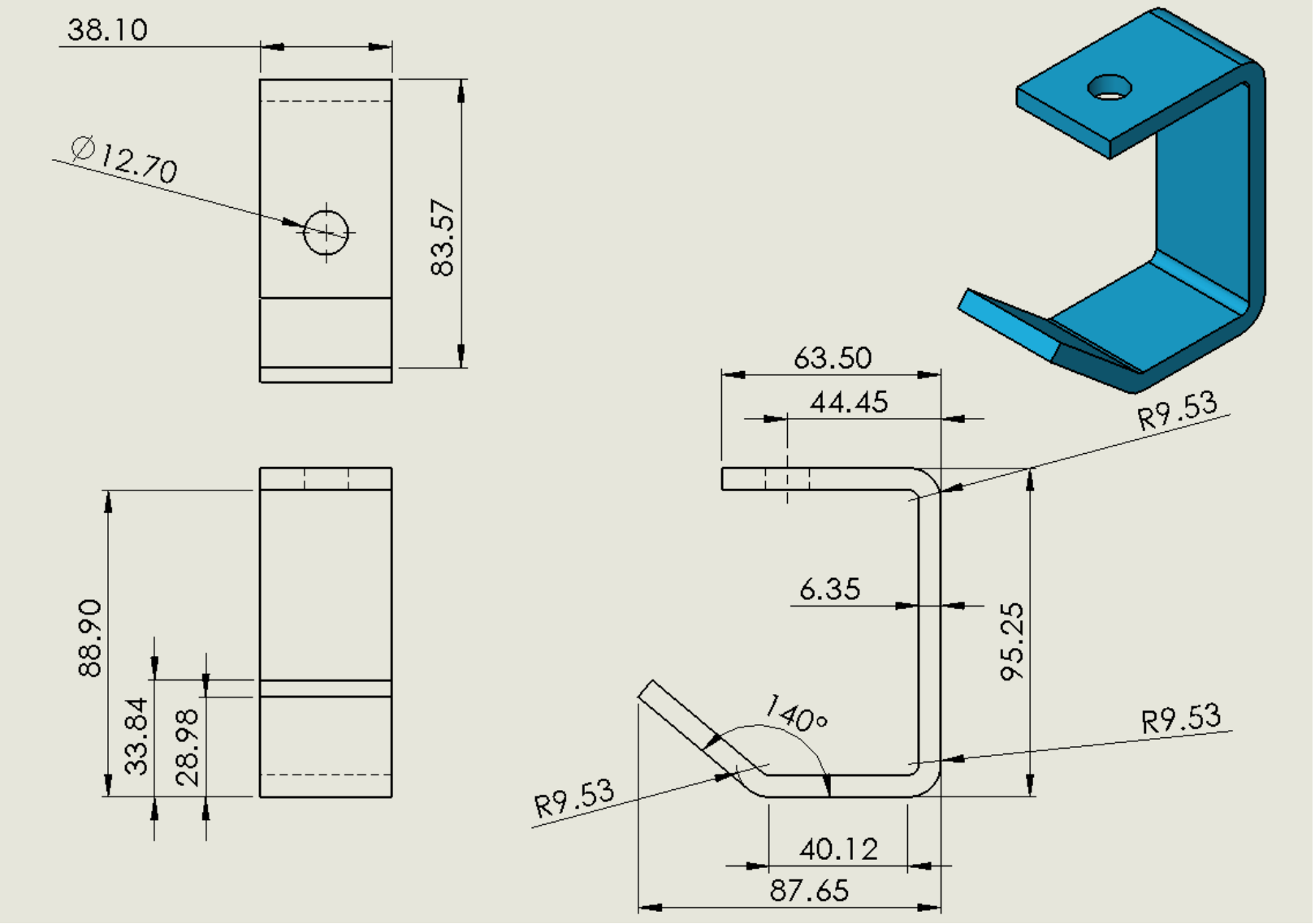
Lesson 12:Bottom-Up Assembly Modeling (Trang 403)

Exercise:
Universal Joint – Trục Các- đăng

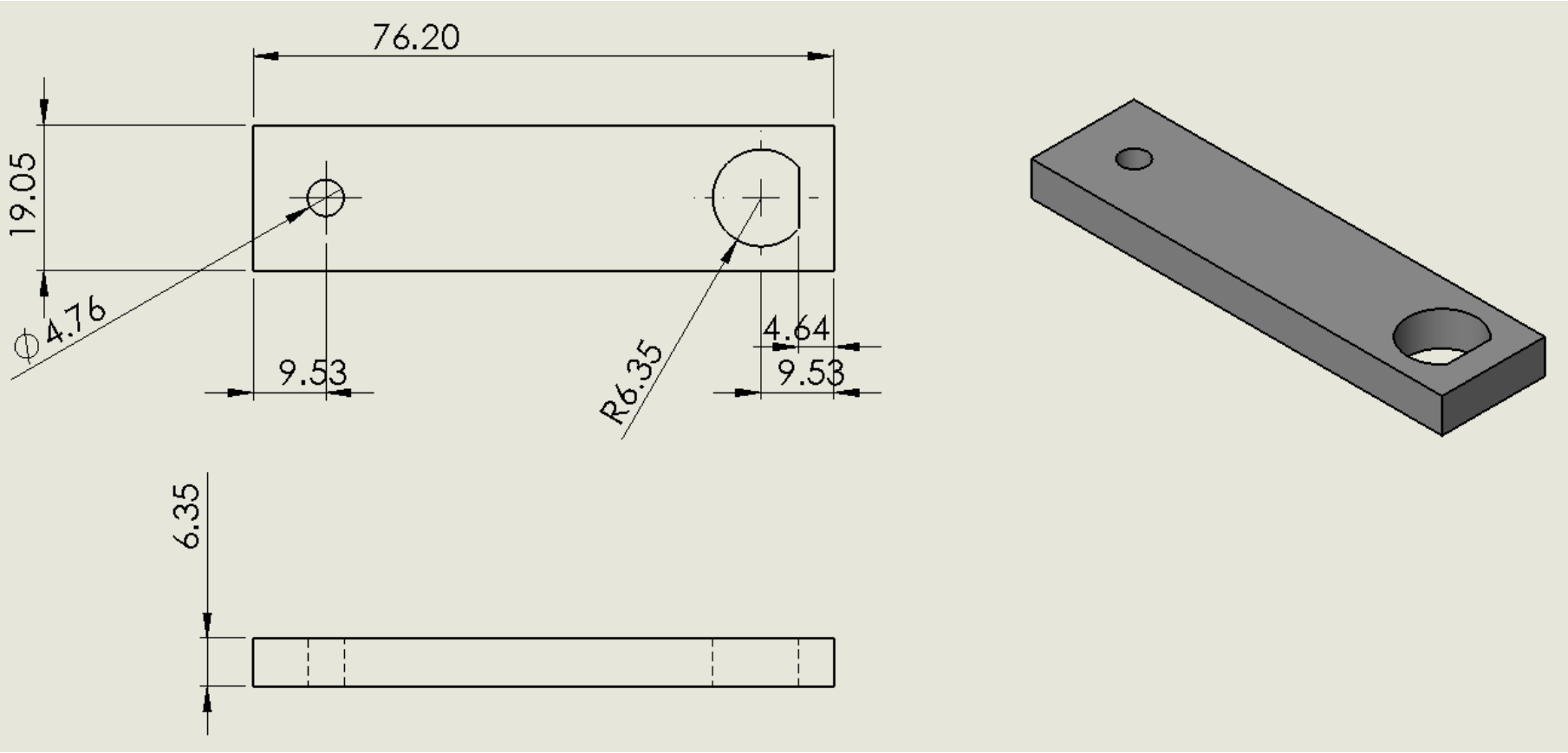


Tiến hành vẽ các chi tiết sau

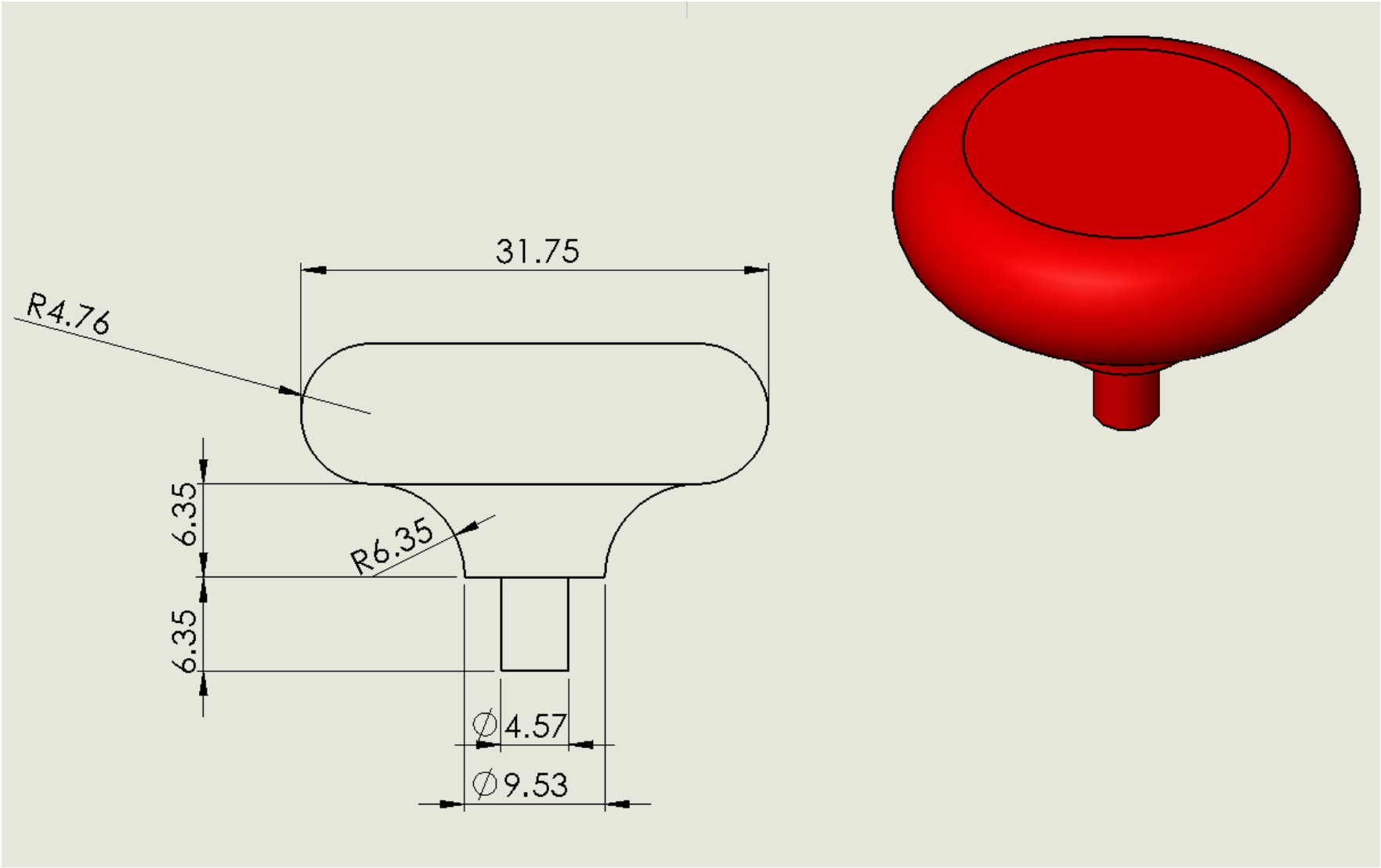
Bracket



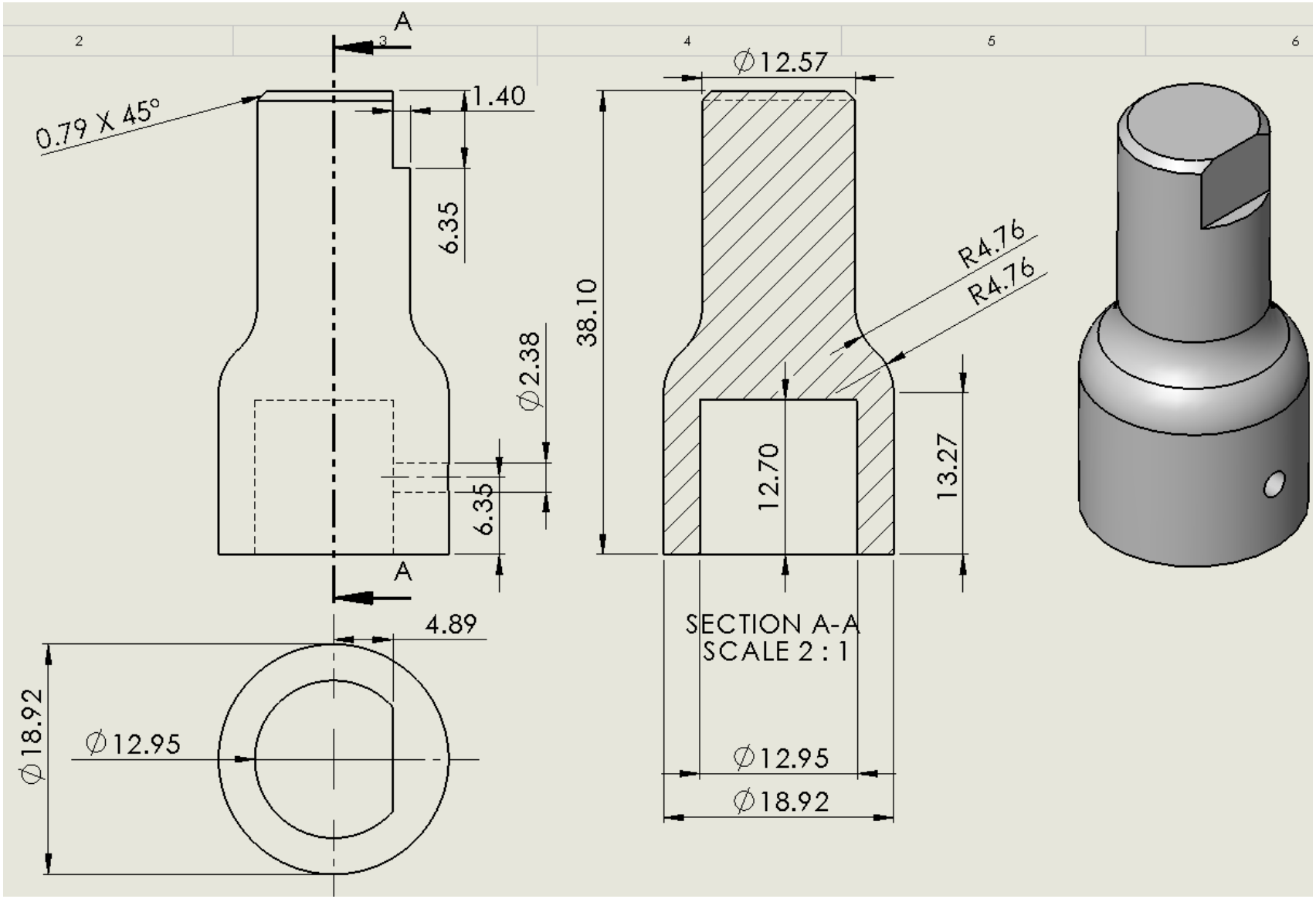
Crank-arm



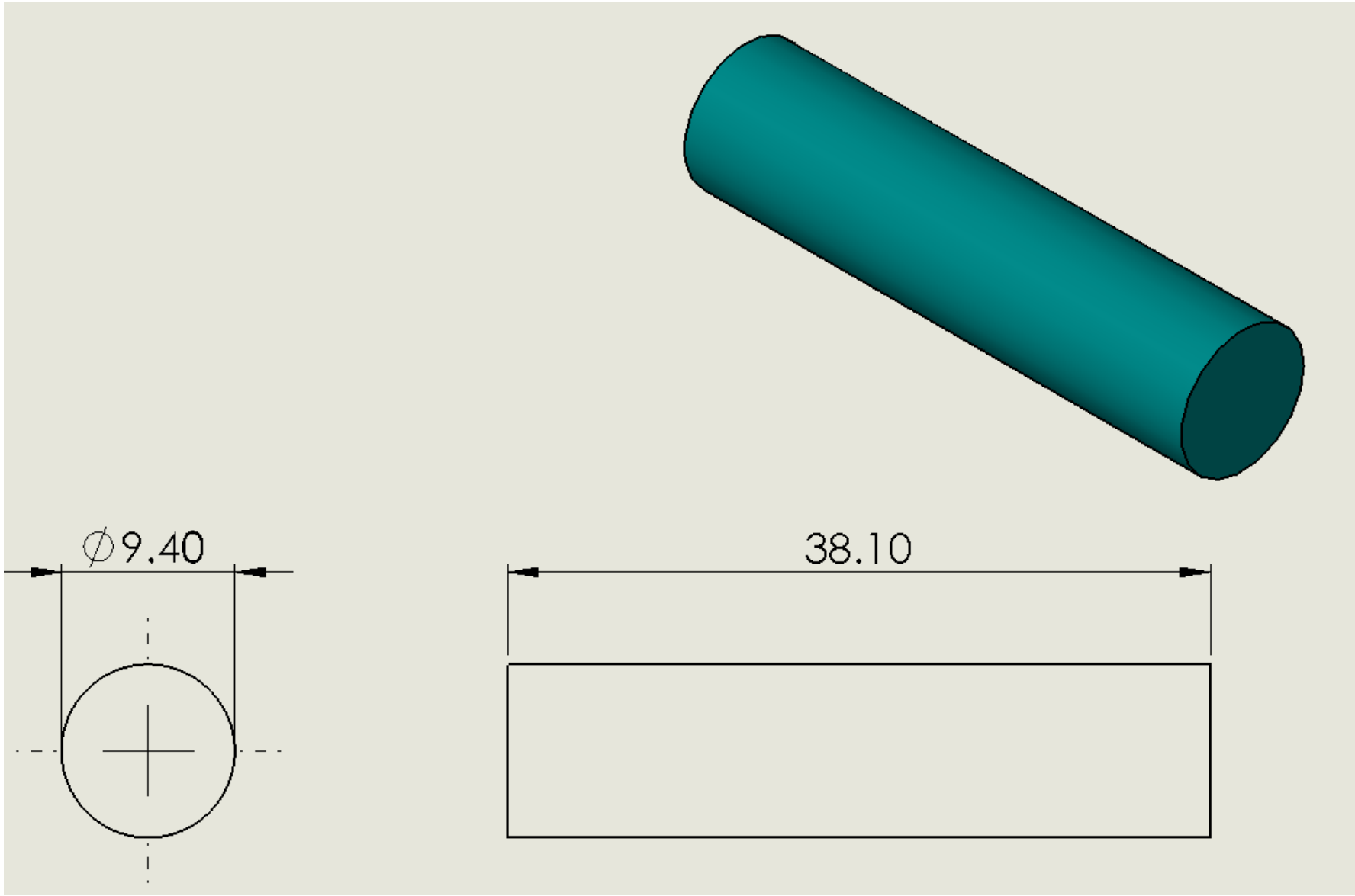
Crank-knob



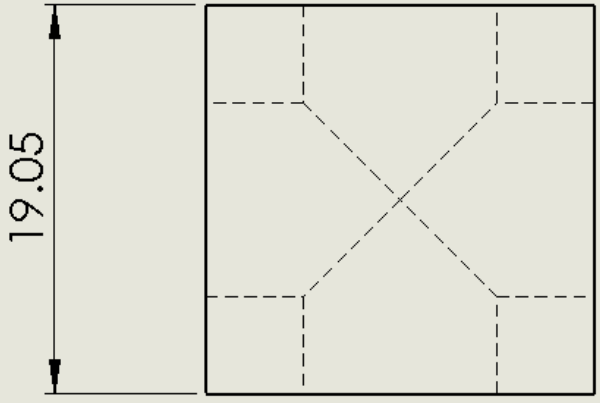
Crank-shaft

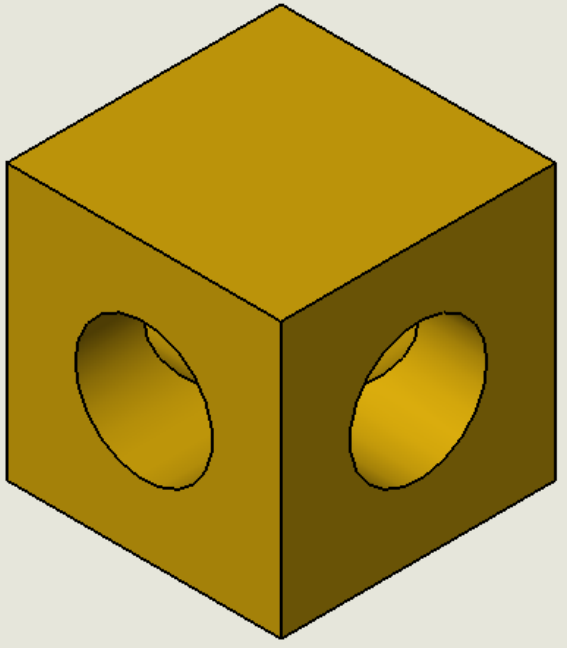

A
0.79 X 45°
1.40
6.35
Ø2.38
6.35
A
4.89
Ø18.92
Ø12.95
Ø12.57
38.10
R4.76
R4.76
12.70
13.27
SECTION A-A
SCALE 2 : 1
Ø12.95
Ø18.92

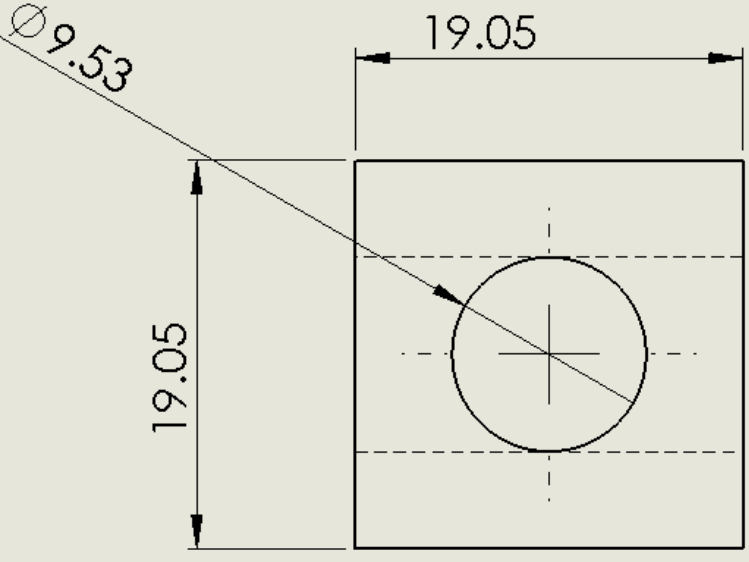
Pin



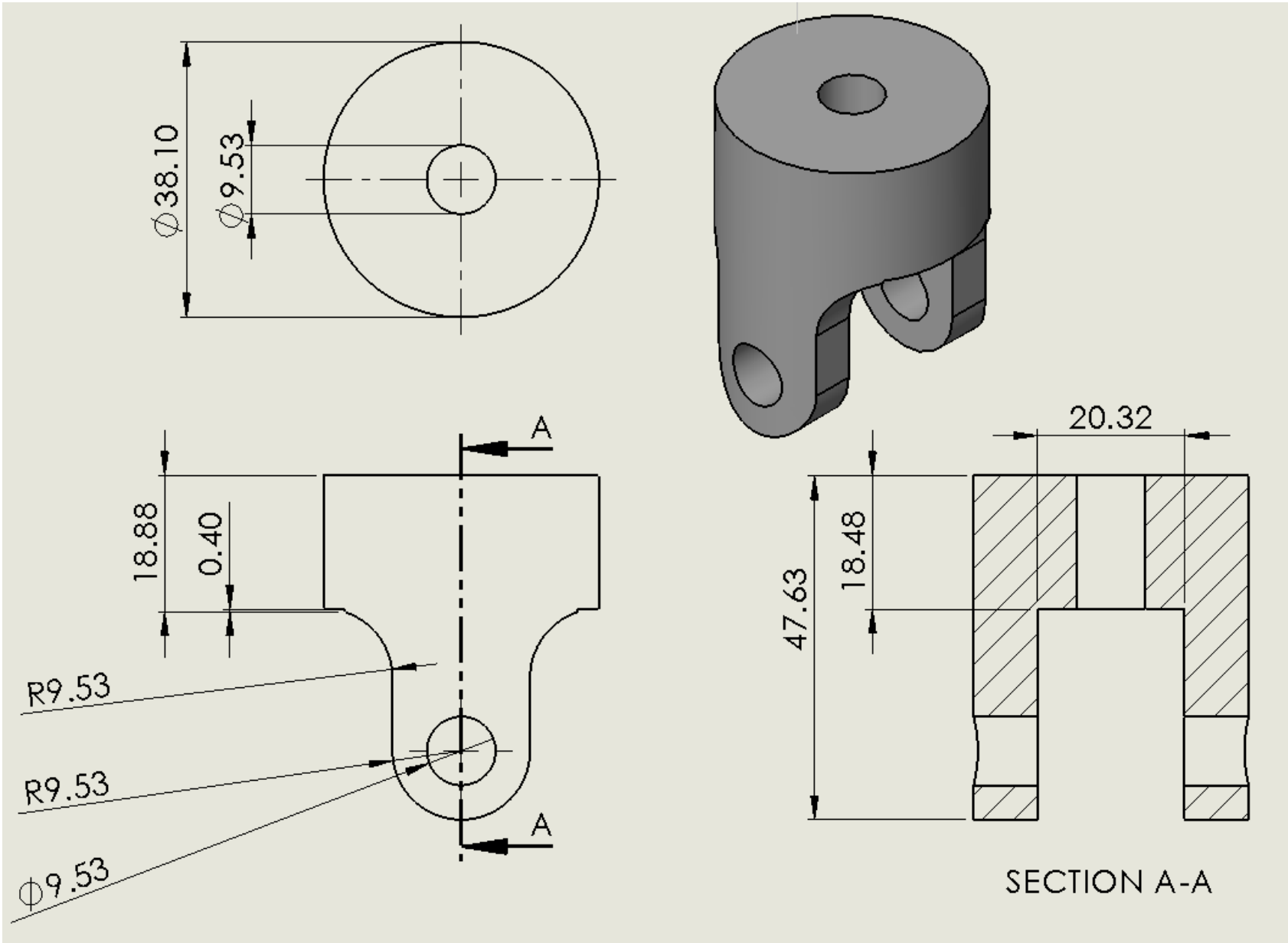
Spider



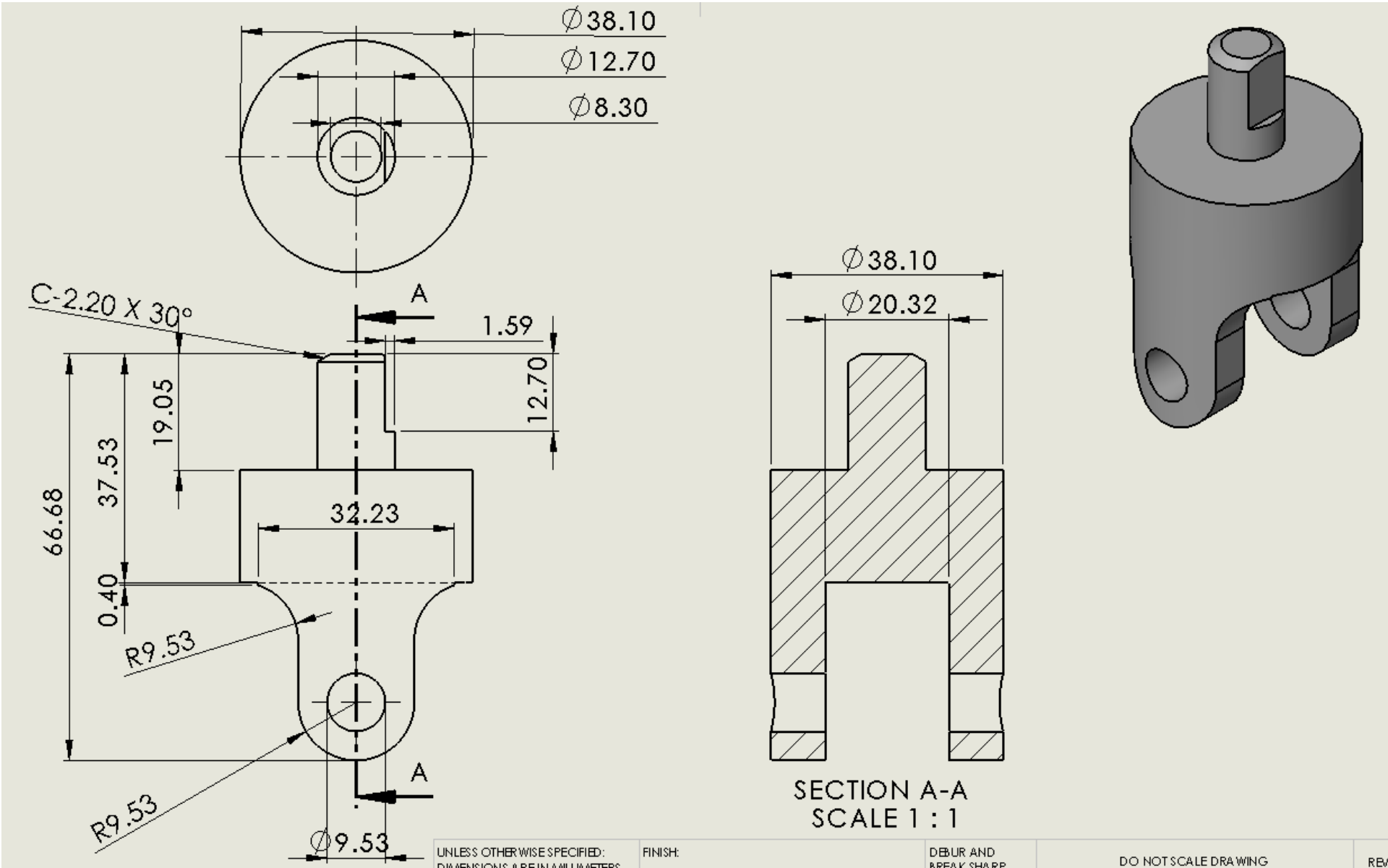




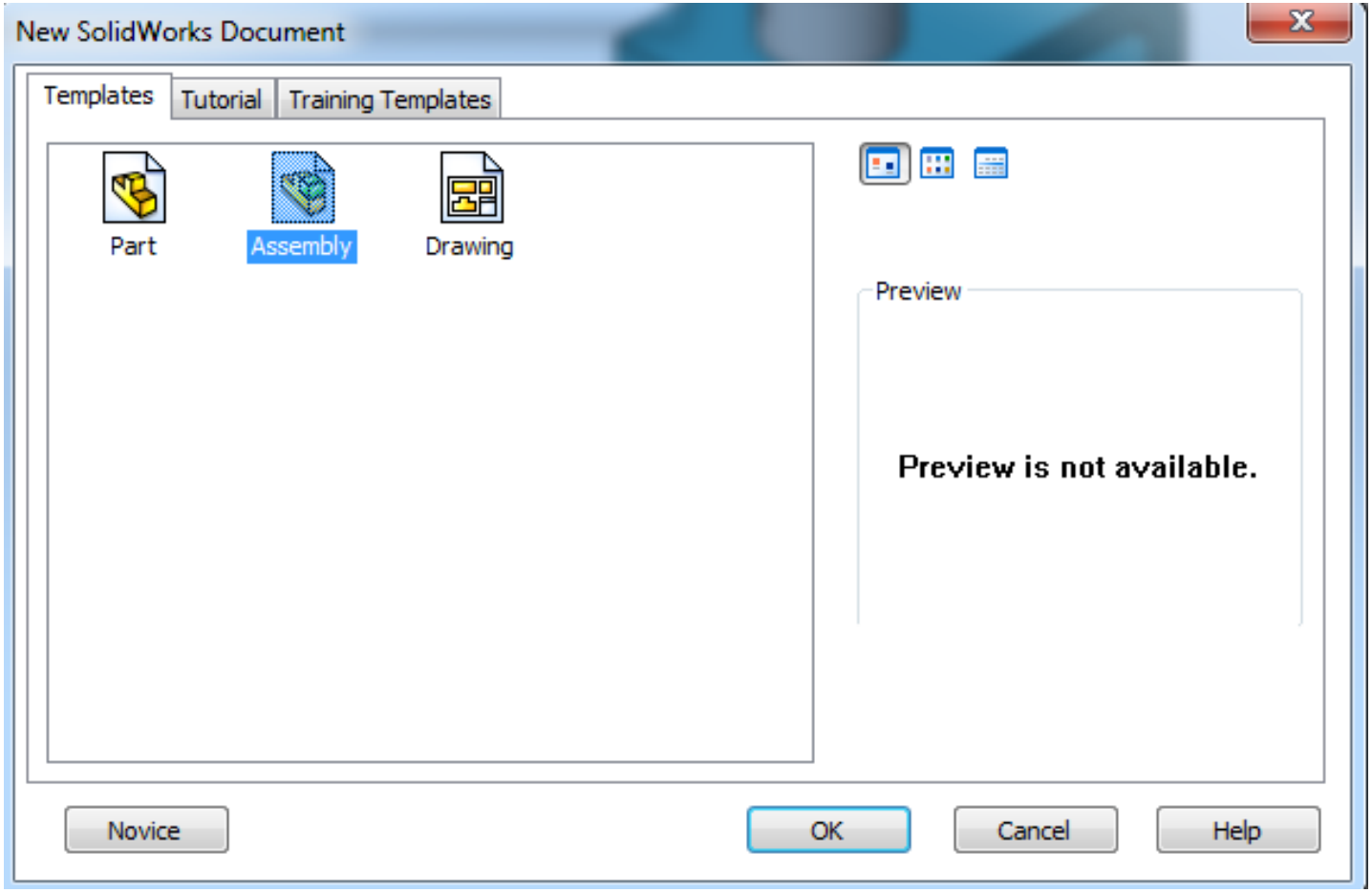
Yoke_female



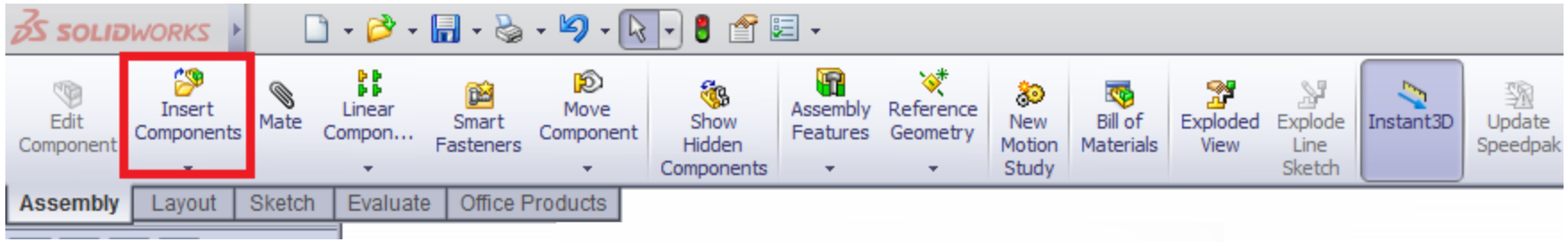
Yoke_male

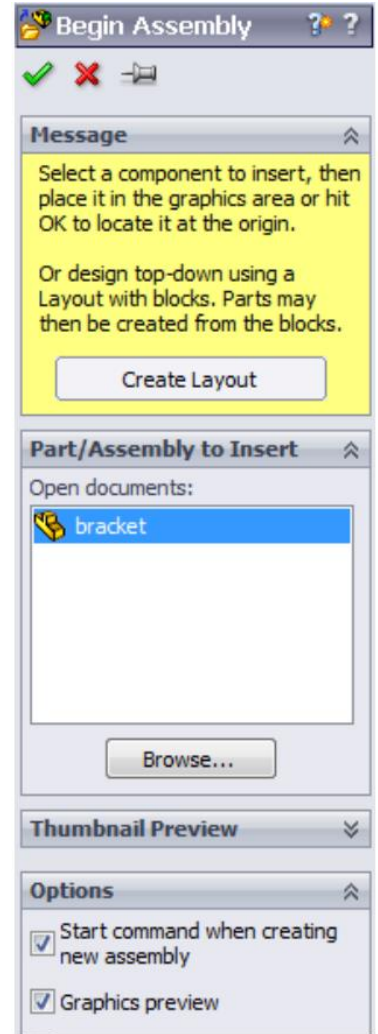

∅38.10
∅12.70
∅8.30
C-2.20 X 30°
A
1.59
12.70
19.05
37.53
66.68
32.23
0.40
R9.53
R9.53
A
∅9.53
∅38.10
∅20.32
SECTION A-A
SCALE 1 : 1
UNLESS OTHERWISE SPECIFIED:
FINISH:
DEBUR AND
DO NOT SCALE DRAWING

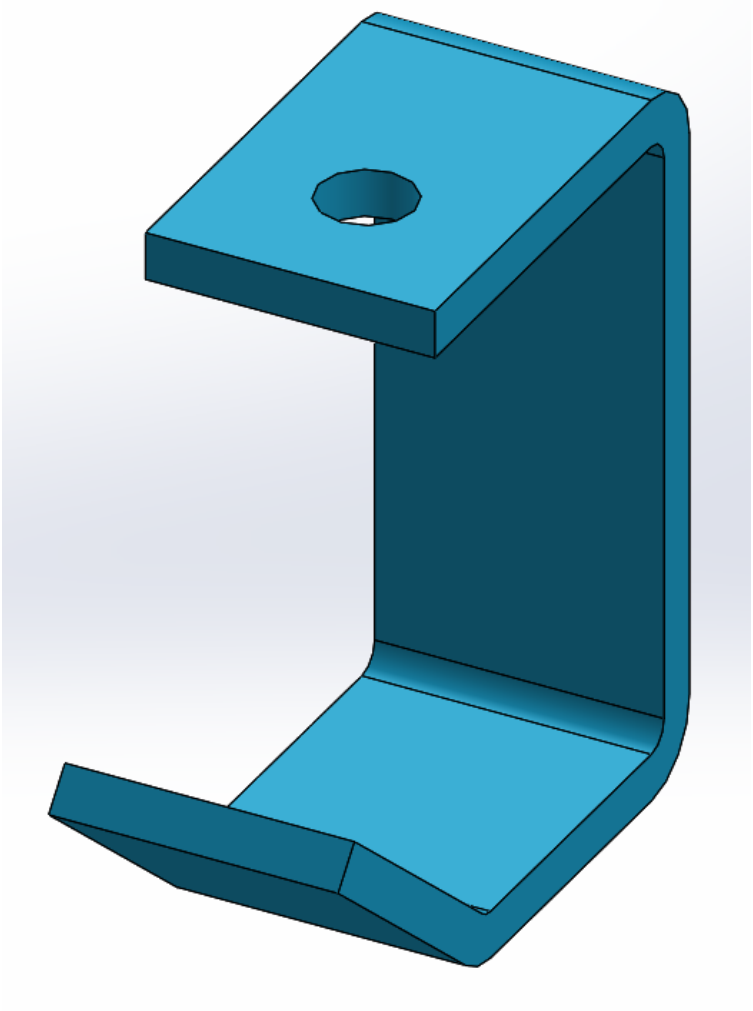
Creating a New Assembly



Chọn lệnh Insert Components để chèn chi tiết vào

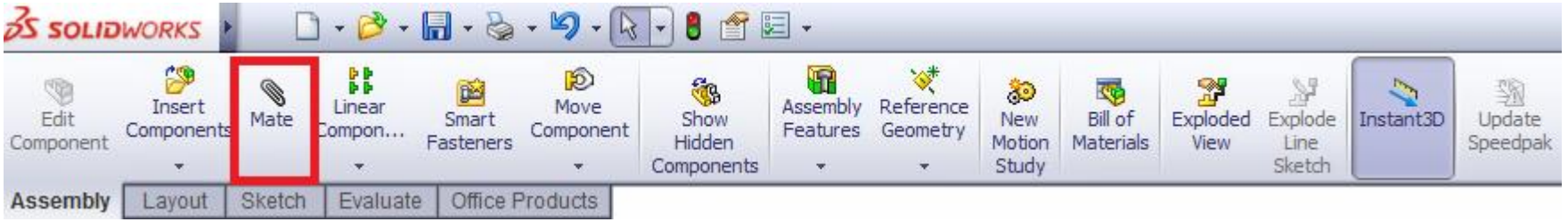




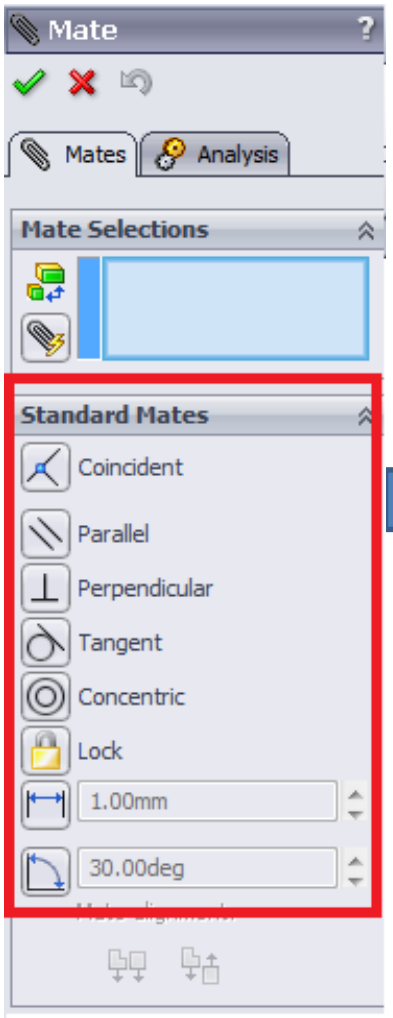


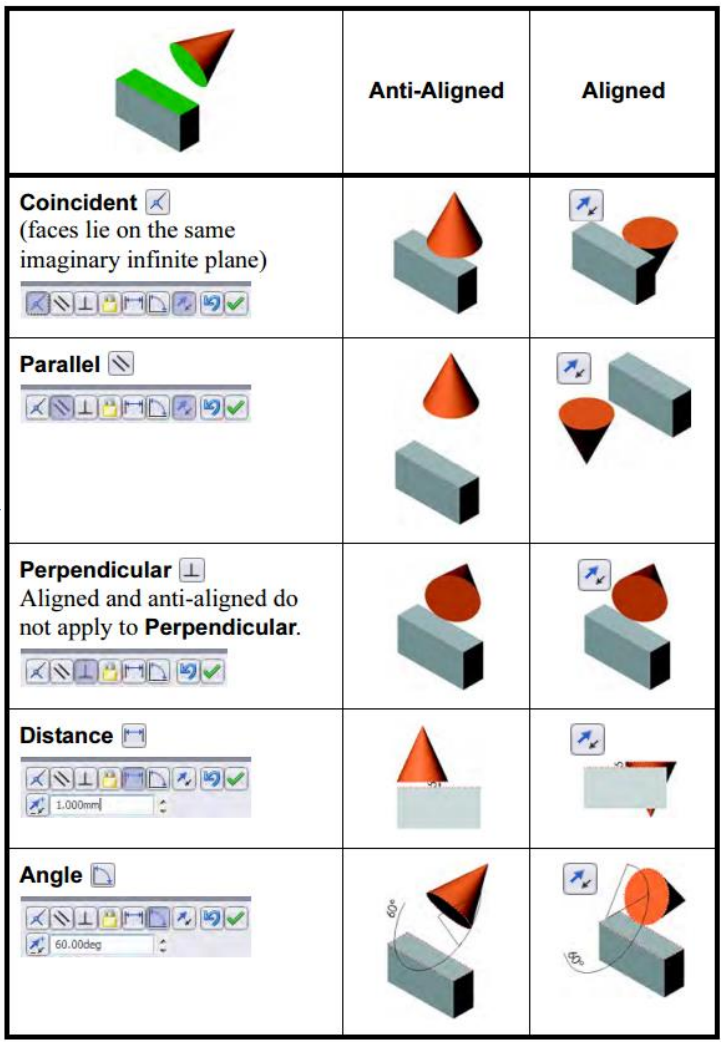
Lưu ý chi tiết đầu tiên sẽ được cố định (FIX)

Chọn lệnh Mate để tạo liên kết giữa các chi tiết

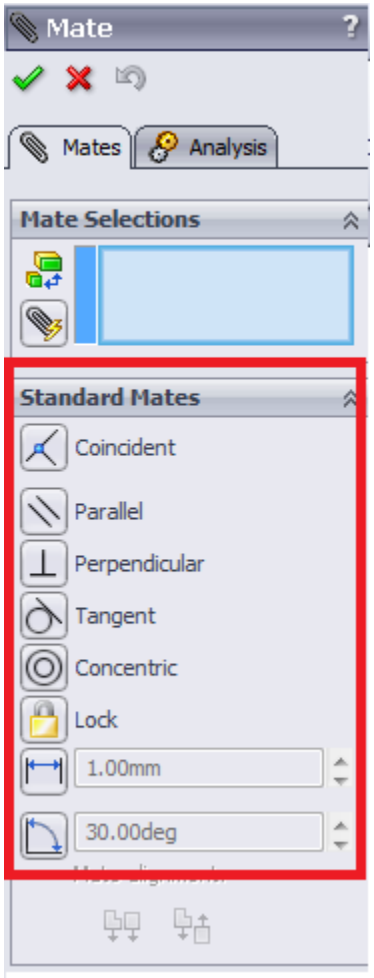


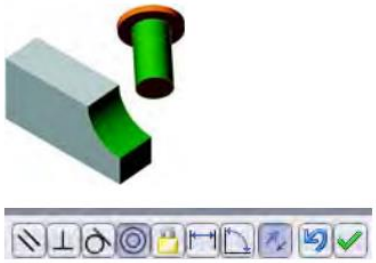
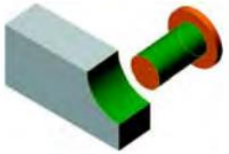
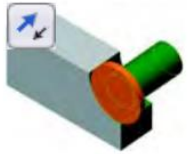
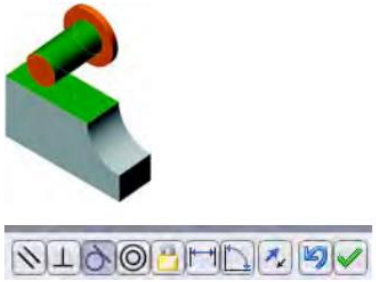
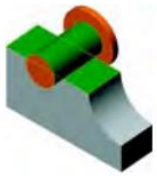
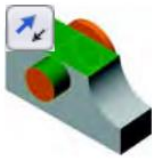
Chức năng của Standard Mates



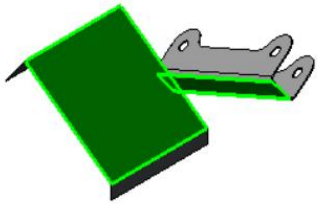
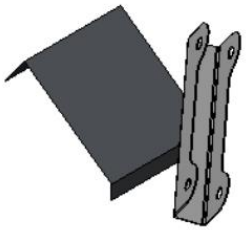
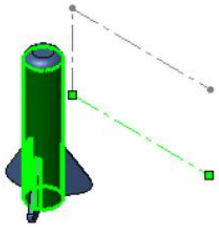
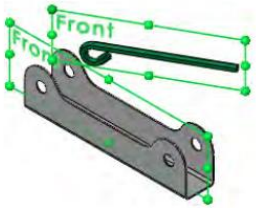
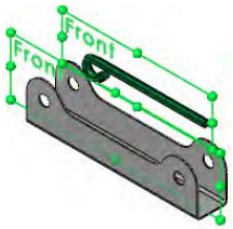
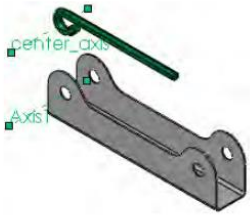
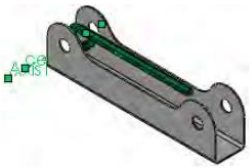
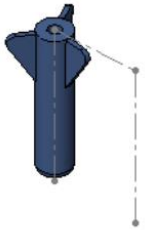
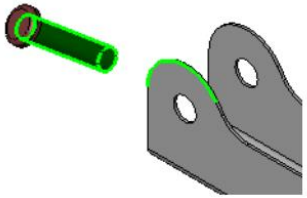
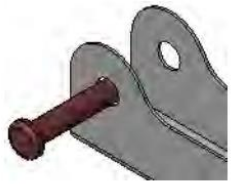


Chức năng của Standard Mates

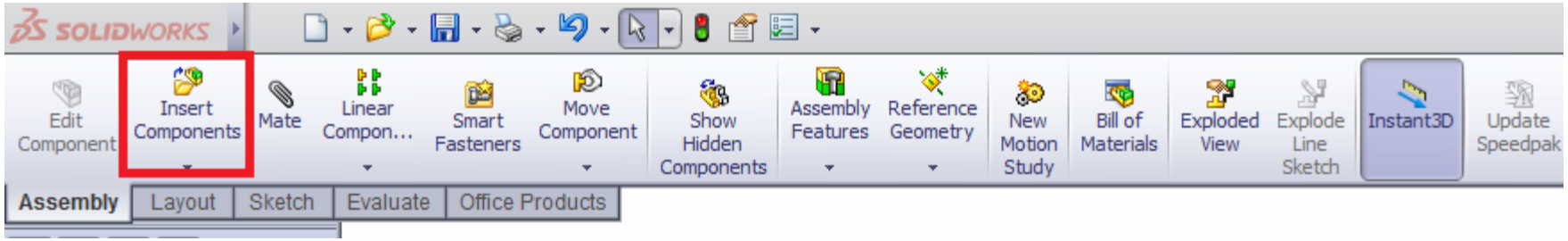


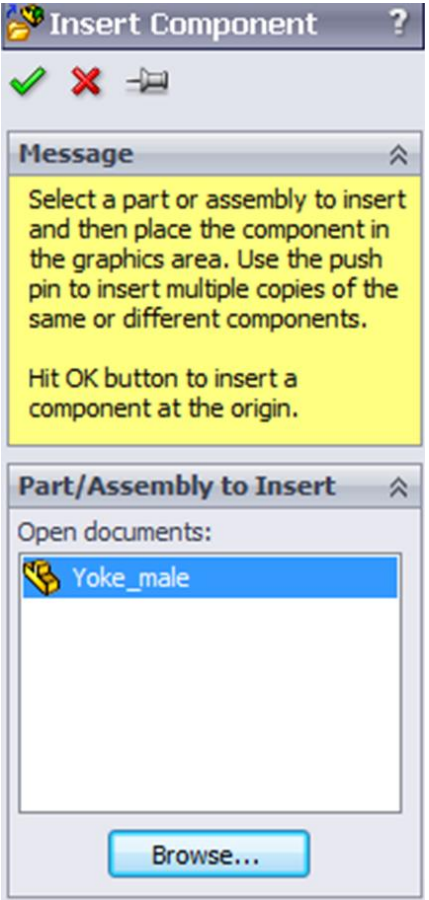
	Anti-Aligned	Aligned
Concentric 		
Tangent 		
Lock  Select anywhere on component.	Components that are locked will move together. No alignment options.	

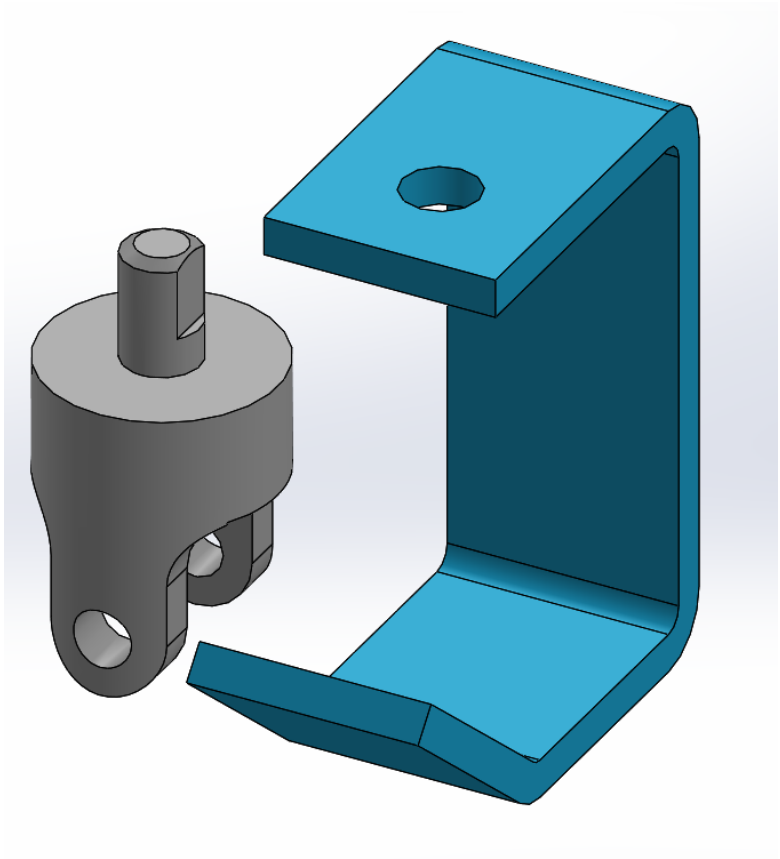
Các dạng liên kết

Topology/ Geometry	Selections	Mate
Faces or Surface		
Line or Linear Edge		
Plane		
Axis or Temporary Axis		
Point, Vertex, Origin or Coordinate System		
Arc or Circular Edge		

Chọn lệnh Insert Components/chọn file *"Yoke_male"*

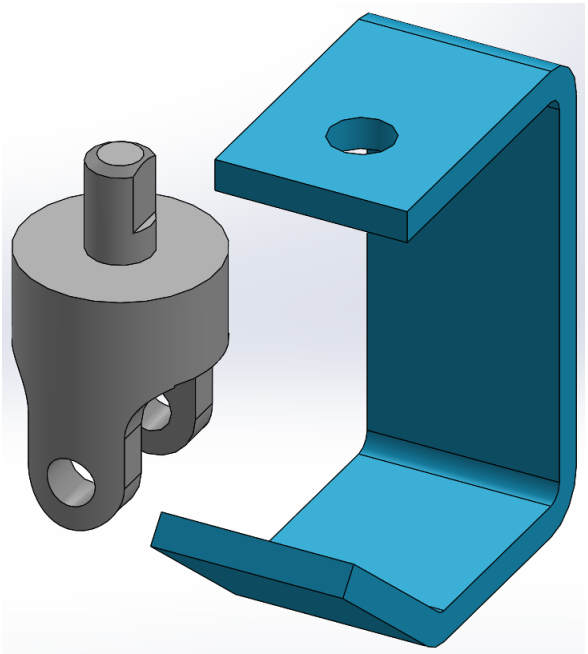






Các bước thực hiện liên kết

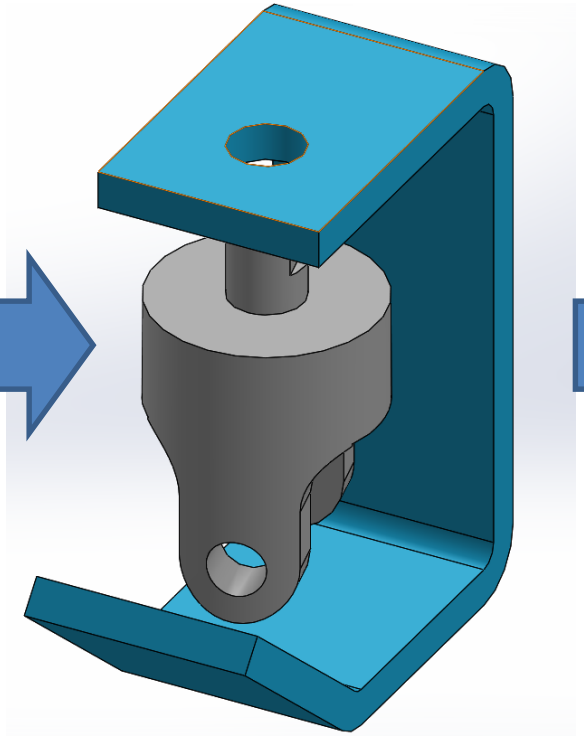
Chuẩn bị



Bước 1

Concentric

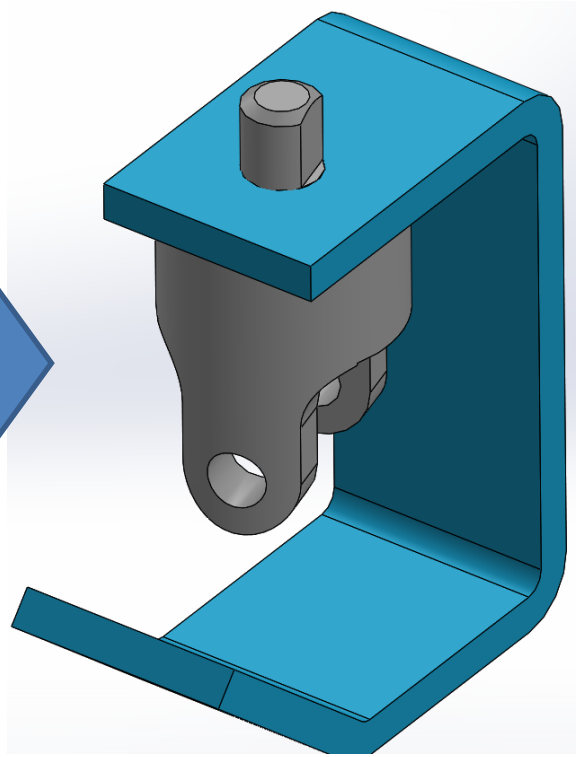
Liên kết đồng tâm



Bước 2

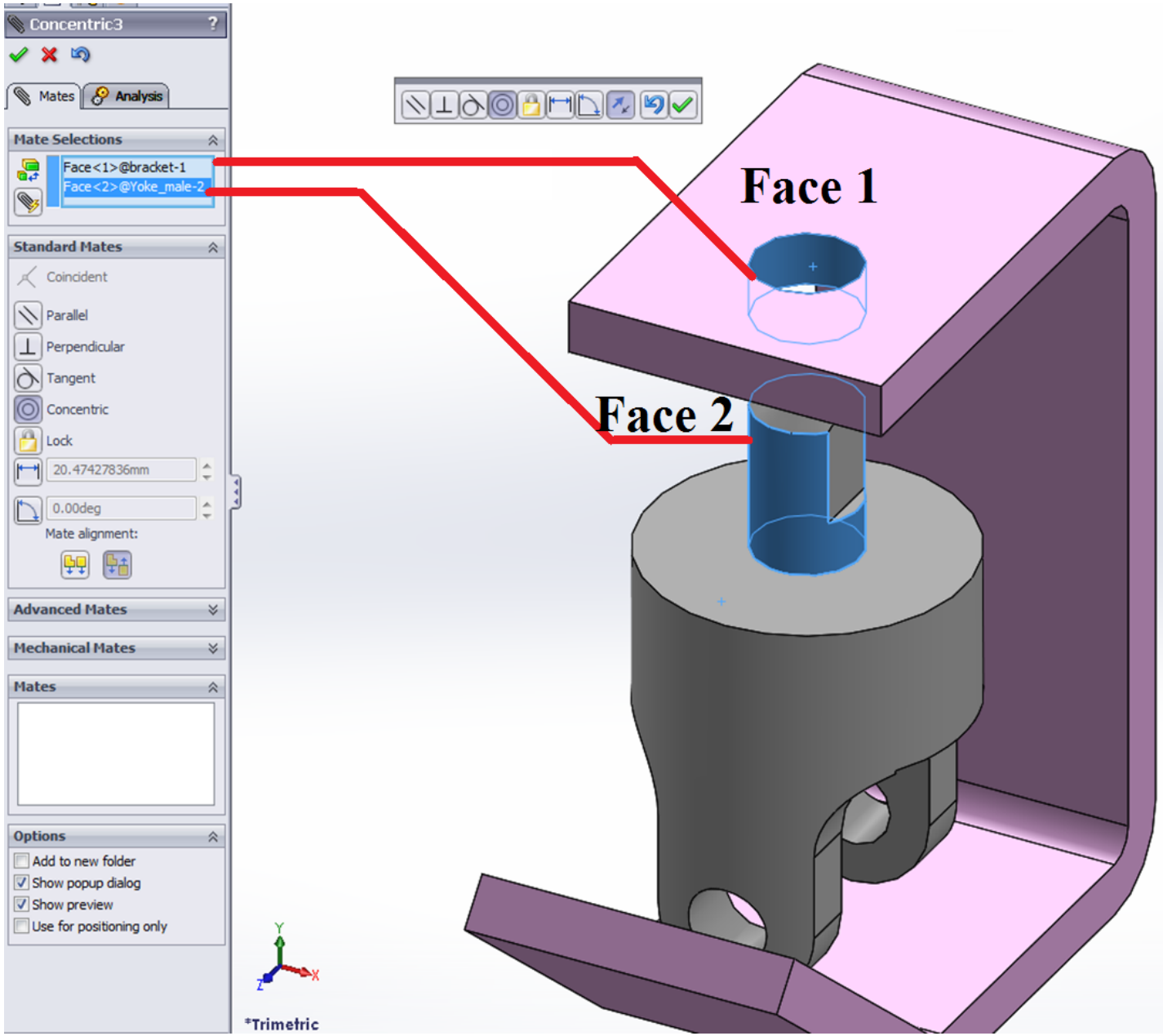
Coincident

Liên kết tiếp xúc



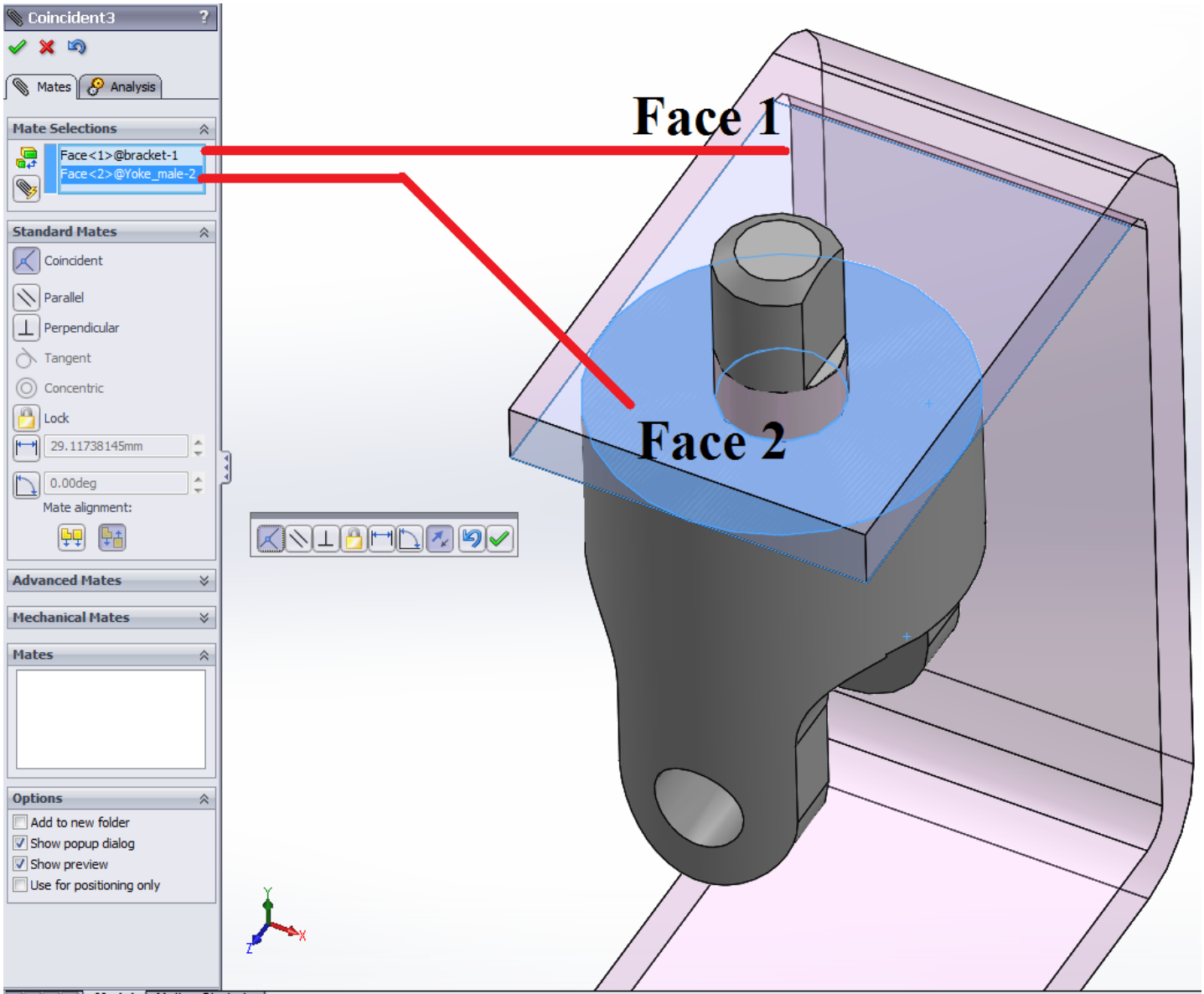
Bước 1: Chọn lệnh Mate/ Concentric *(đồng tâm)*

Sau đó ta chọn
Face 1: Bracket
Face 2: Yoke_male_2
Kết quả:
Face 1 và Face 2 đồng tâm với nhau.



Bước 2: Chọn lệnh Mate/ Coincident *(Tiếp xúc)*

Sau đó ta chọn
Face 1: Bracket
Face 2: Yoke_male_2
Kết quả:
Face 1 và Face 2 tiếp xúc với nhau

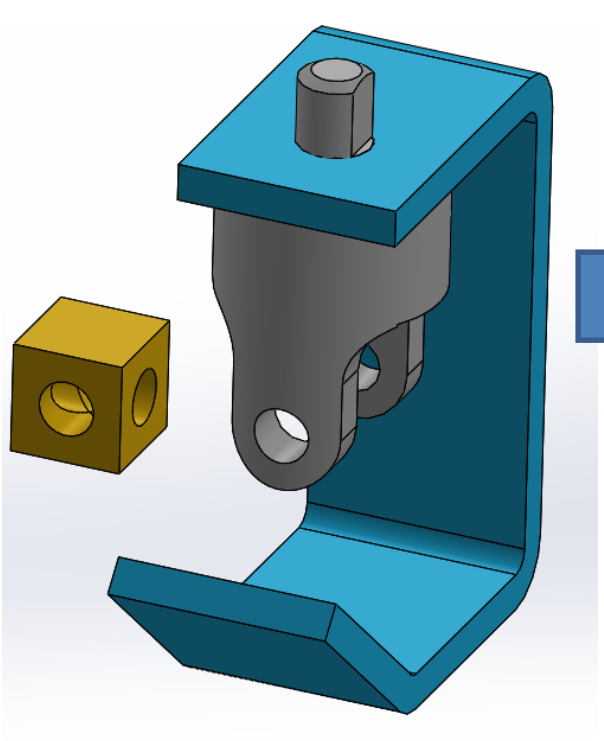


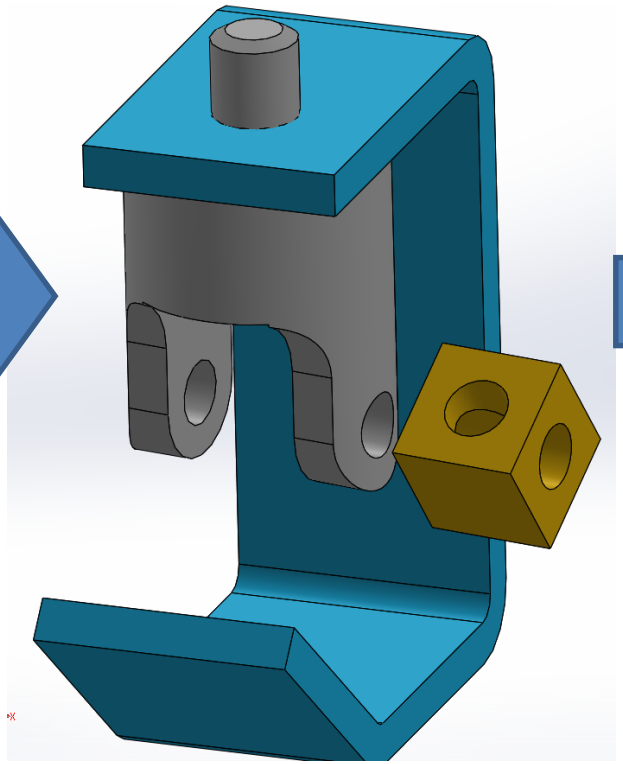
Các bước thực hiện liên kết

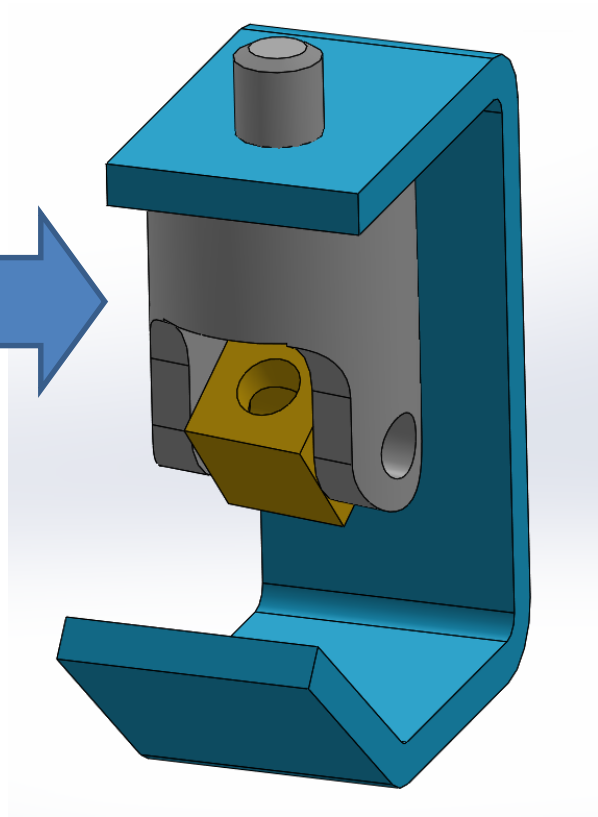
Chuẩn bị

Bước 3
Concentric
Liên kết đồng tâm

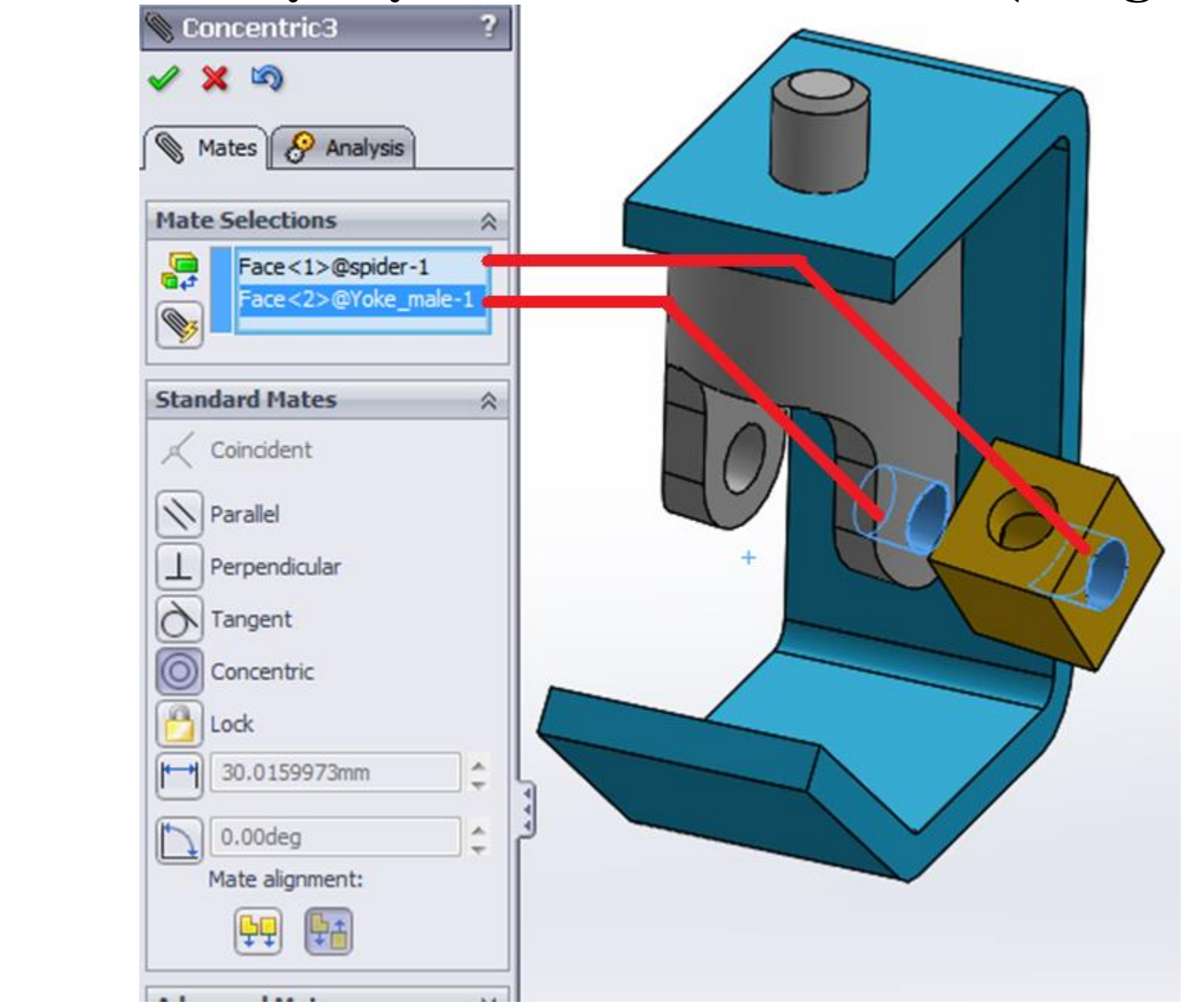
Bước4
Width
Liên kết bề rộng



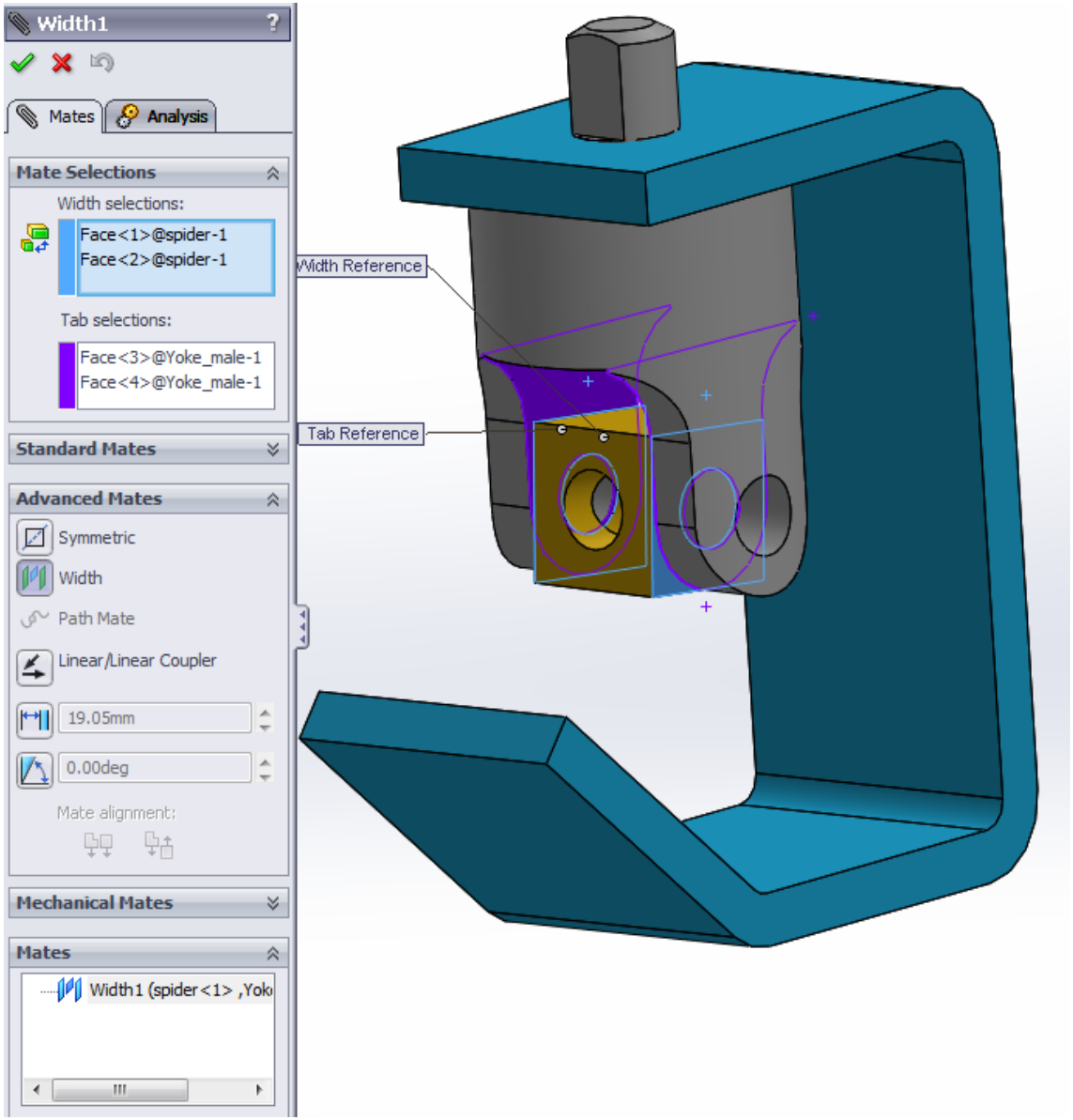




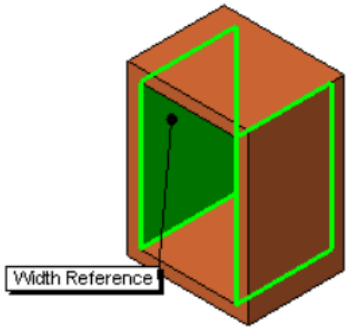
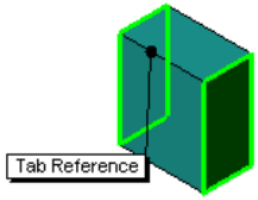
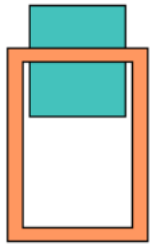
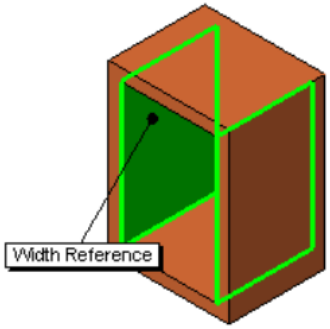
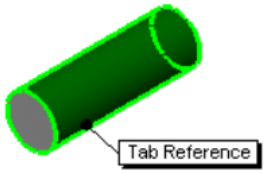
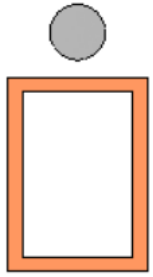
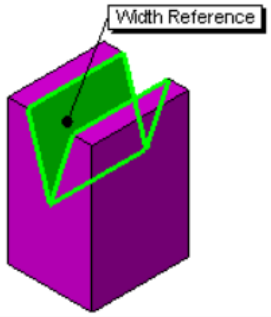
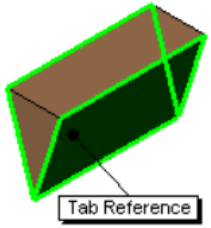
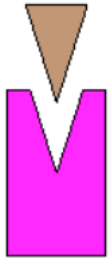
Bước 3: Chọn lệnh Mate/ Concentric *(đồng tâm)*



Bước 4: Chọn lệnh Mate/ Avdvanced Mates/ Width *(Liên kết bề rộng)*



Chức năng của lệnh Mate Width (Liên kết bề rộng)

Width selections	Tab selection(s)	Result
		 (Front view)
	 (single selection)	 (Front view)
		 (Front view)

Advanced Techniques Assembly

Lession 1: Top-Down Assembly Modeling

Lession 2: Assembly Features, Smart Fasteners

Lession 3: Assembly Mate Techniques

Lession 4: Using Configurations with Assemblies

Lession 5: Display States and Appearances

Lession 6: Assembly Editing

Lession 7: Layout-based Assembly Design

Lession 1: Top-Down Assembly Modeling

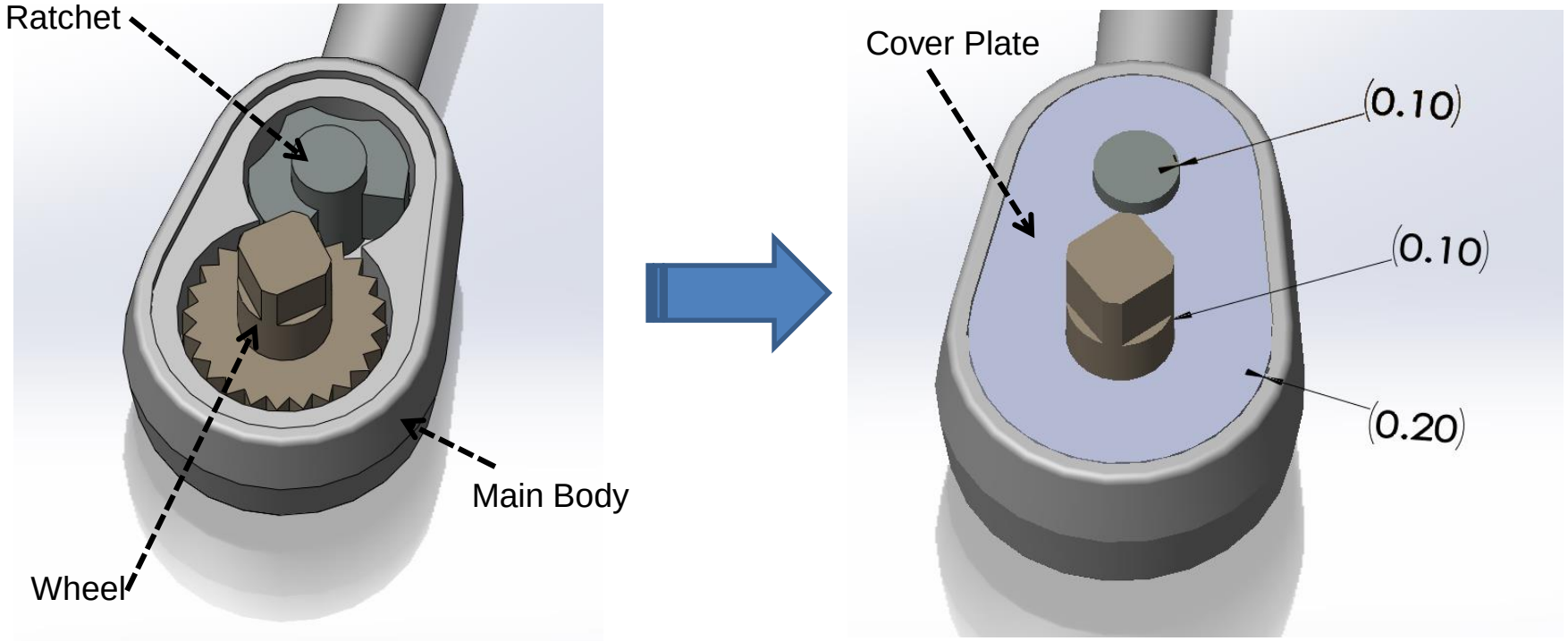
Exercise 1: Top-Down Assembly Modeling

Design Cover Plate :

Cover Plate to Main Body **=0.20mm**
Cover Plate to Ratchet =**0.10mm**
Cover Plate to Wheel **=0.10mm**

1. Open an assembly file

Open *TOP DOWN ASSY* from *Lesson01\Exercises\Top Down Assy* folder.



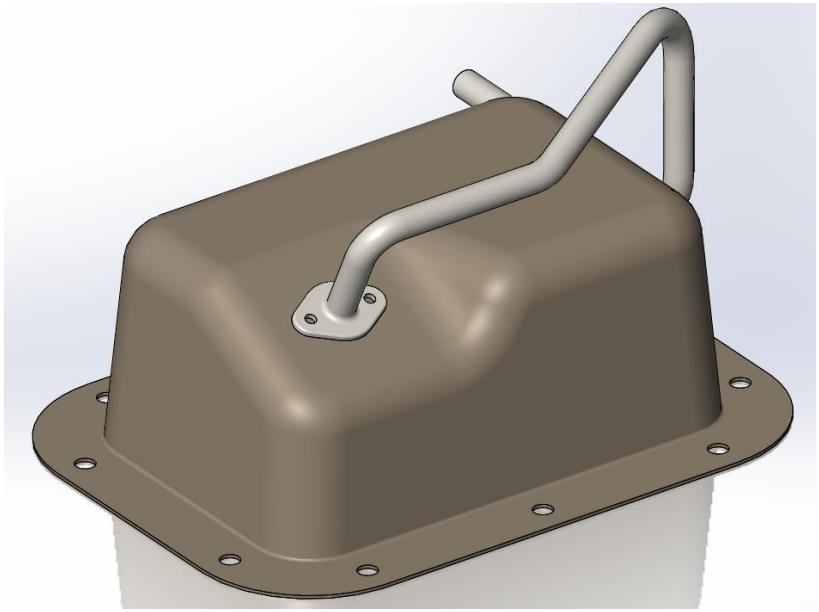
Exercise 1.1: In-context Features

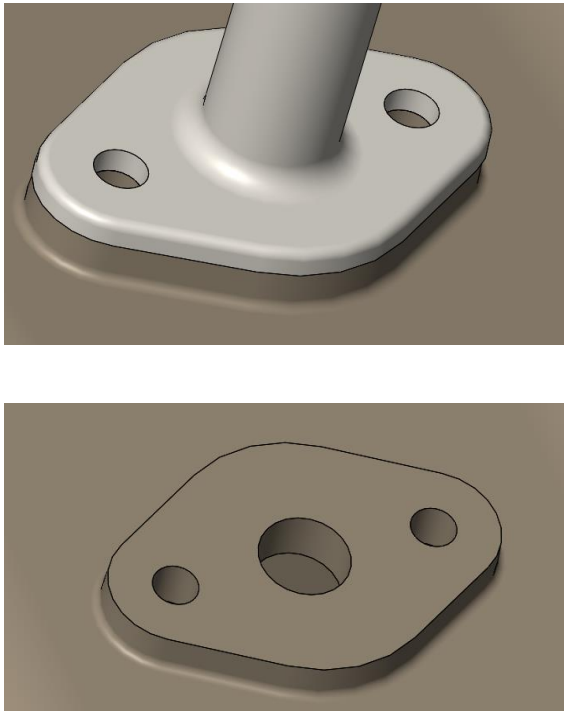
Edit part **Oil Pan**:

1.The flange on the **Pipe** creates the extruded shape of the corresponding flange on Oil Pan. Use 3^0 of draft.
2. Filet radius is **2 mm**
3. The holes for the bolts and pipe pass through the flange and the wall thickness on the **Oil Pan.**

1. Open an assembly file

Open *Oil Pan Assy* from *Lesson01\Exercises\InContextFeatures*folder.

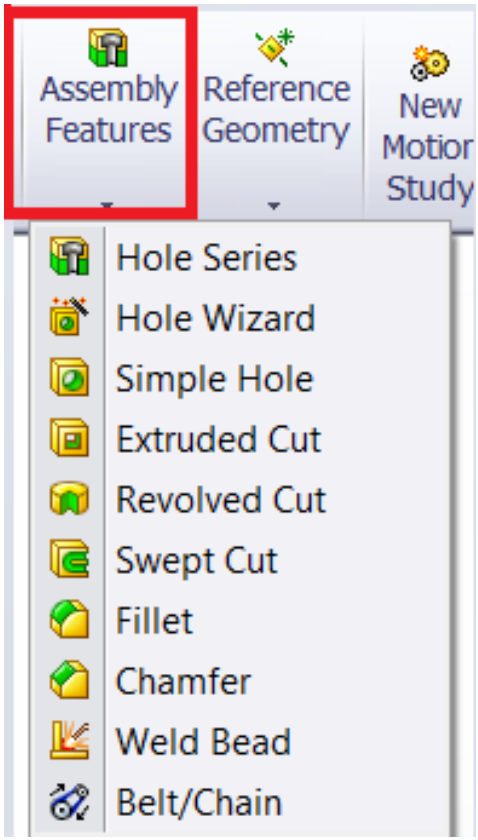




Lession 2: Assembly Features and Smart Fasteners

Stages in the Process

- Creating a new Hole Series assembly feature
- Adding holes using an existing Hole Series feature
- Adding hardware into the holes



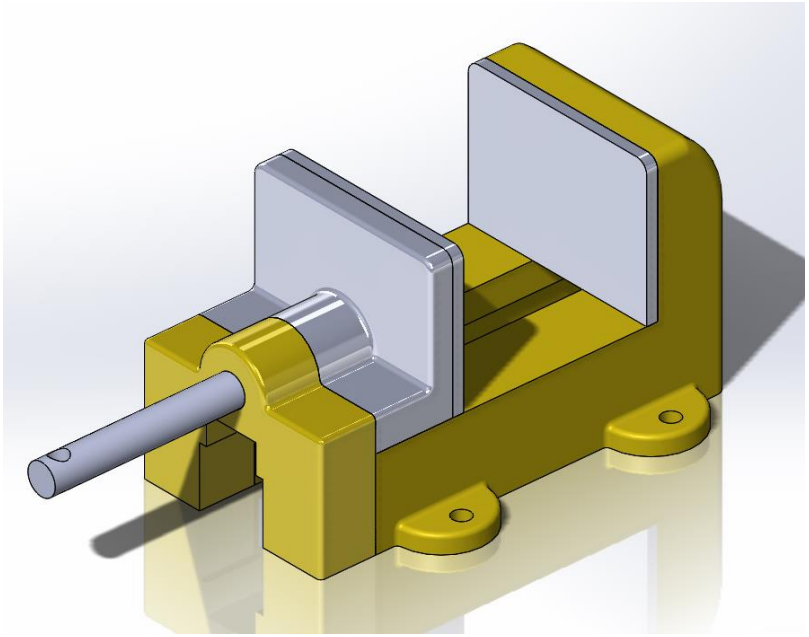
CommandManager: **Assembly Features**
Menu: **Insert, Assembly Feature**

Features:
Hole Series
Hole Wizard
Simple Hole
Extruded Cut
Swept Cut
Fillet
Chamfer
Weld Bead
Belt/Chain

■ Creating a new Hole Series assembly feature

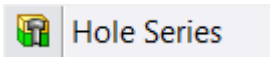
1. Open an assembly file

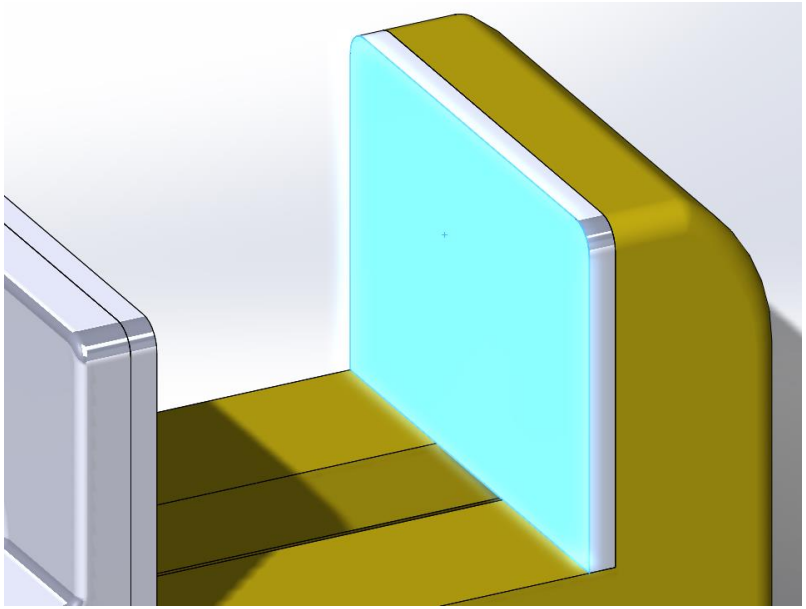
Open *Machine_Vise_&* from *Lession02\Case Study\Built Parts* folder.



2. Hole series

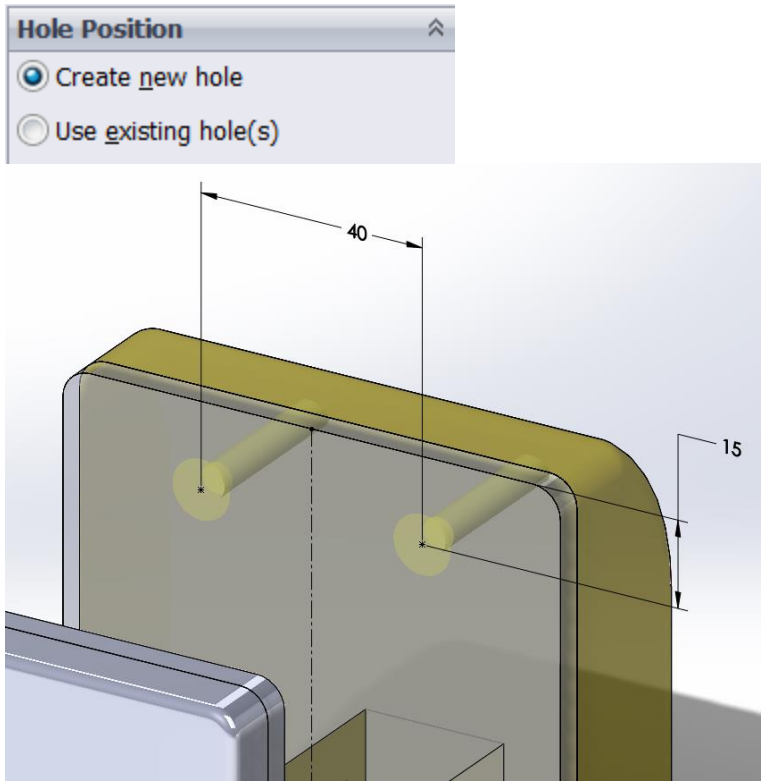
Select the face of *Jaw_plate* and click Hole Series



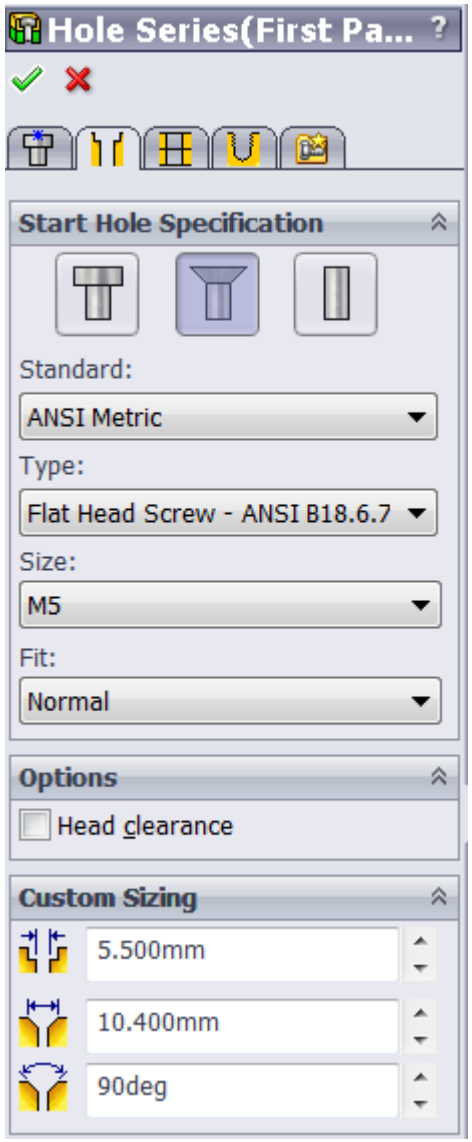


■ Creating a new Hole Series assembly feature

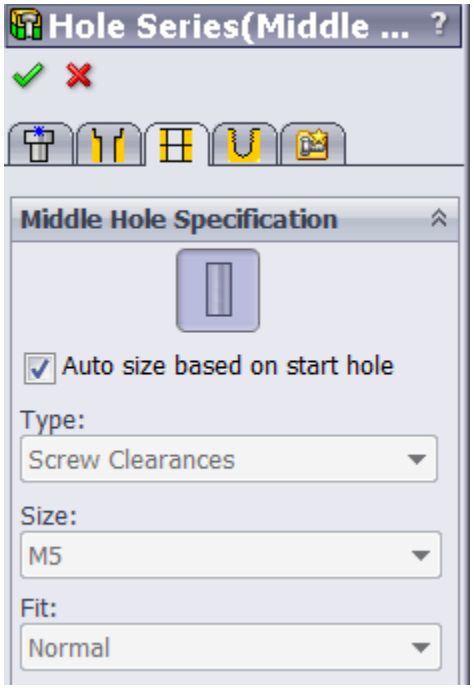
3. Hole position



4. First Part

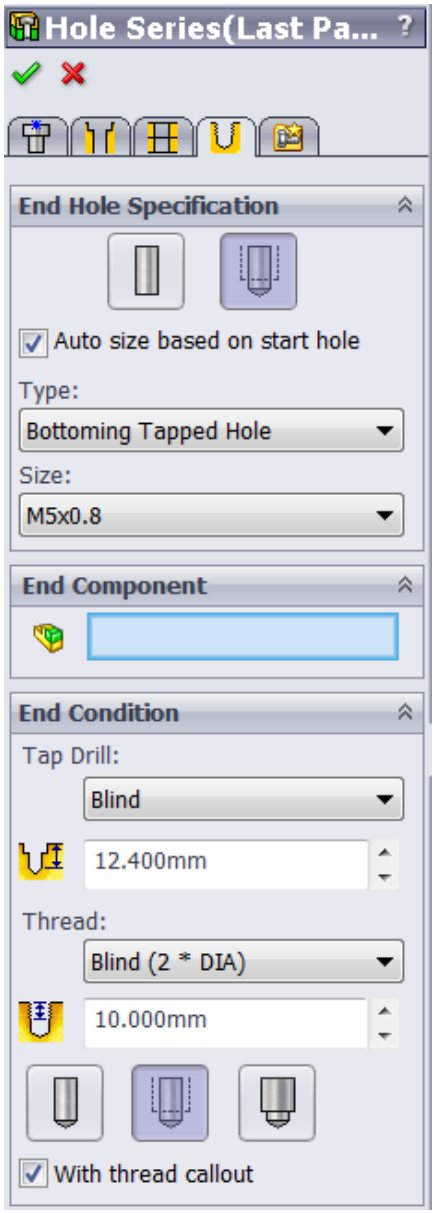


5. Middle Part

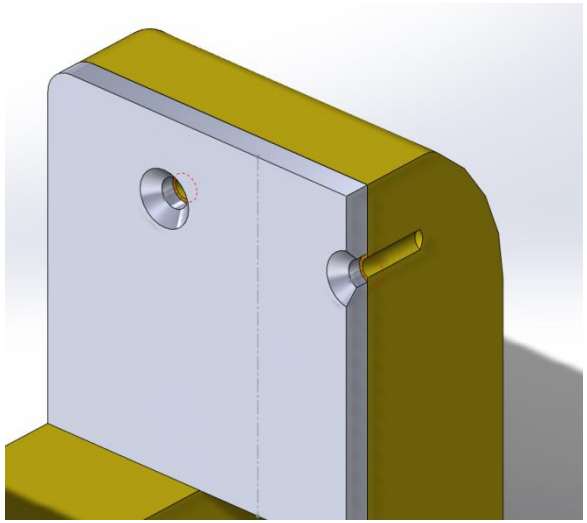


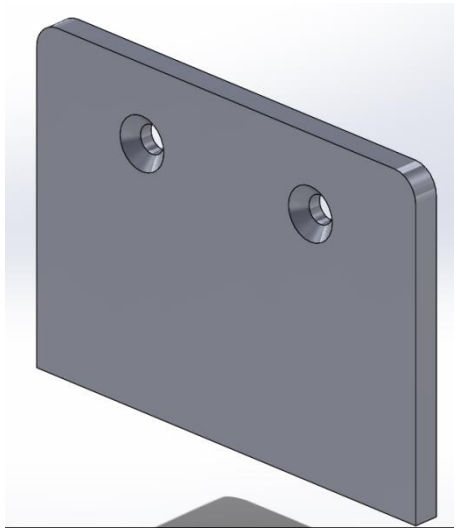
■ Creating a new Hole Series assembly feature

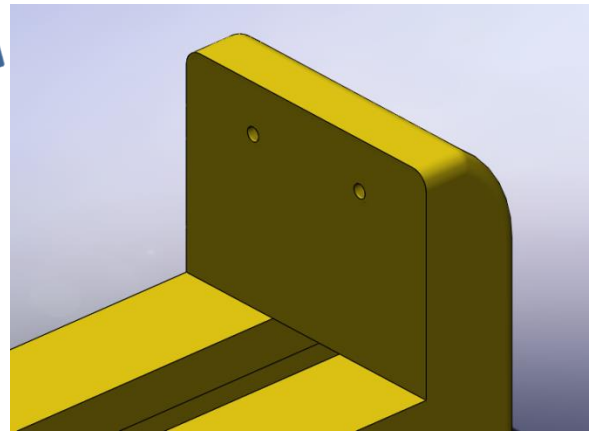
6. Last Part



7. Section view





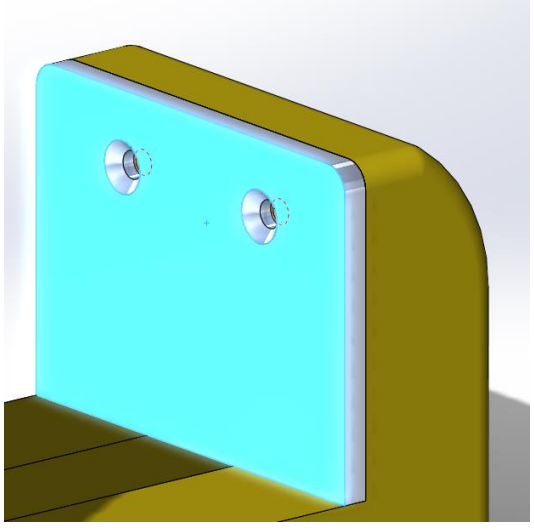


■ Adding hardware into the holes using Smart Fasteners

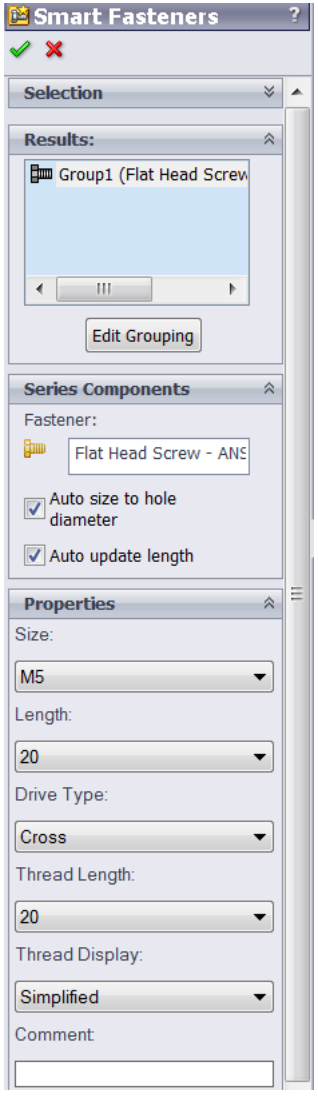
CommandManager: **Assembly > Smart Fasteners**
Menu: **Insert, Smart Fasteners**

1. Click Smart Fastener

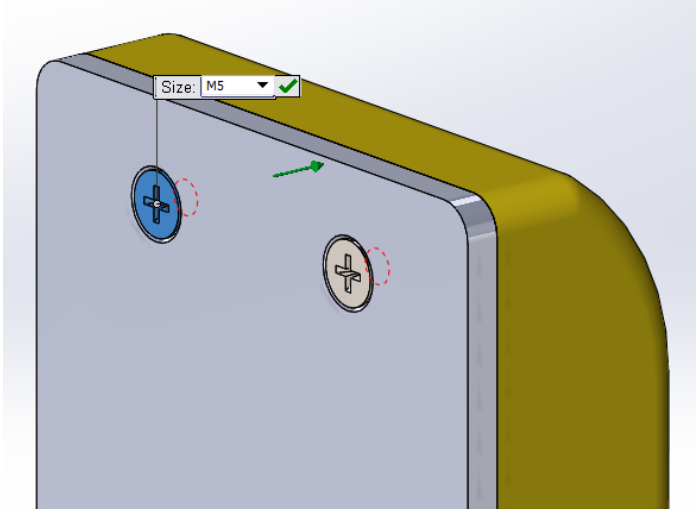
2. Select the planar face



3. Setting



4. Results



Changes to Existing Fasteners

- Right click the **Hole Series** feature **> Edit Feature**
- Right click the **Smart Fastener** feature **> Edit Smart Fastener**
- Right click the fastener **> Edit Toolbox component.**

Exercise 2: The Hole Wizard and Smart Fasteners

Using the following skills:

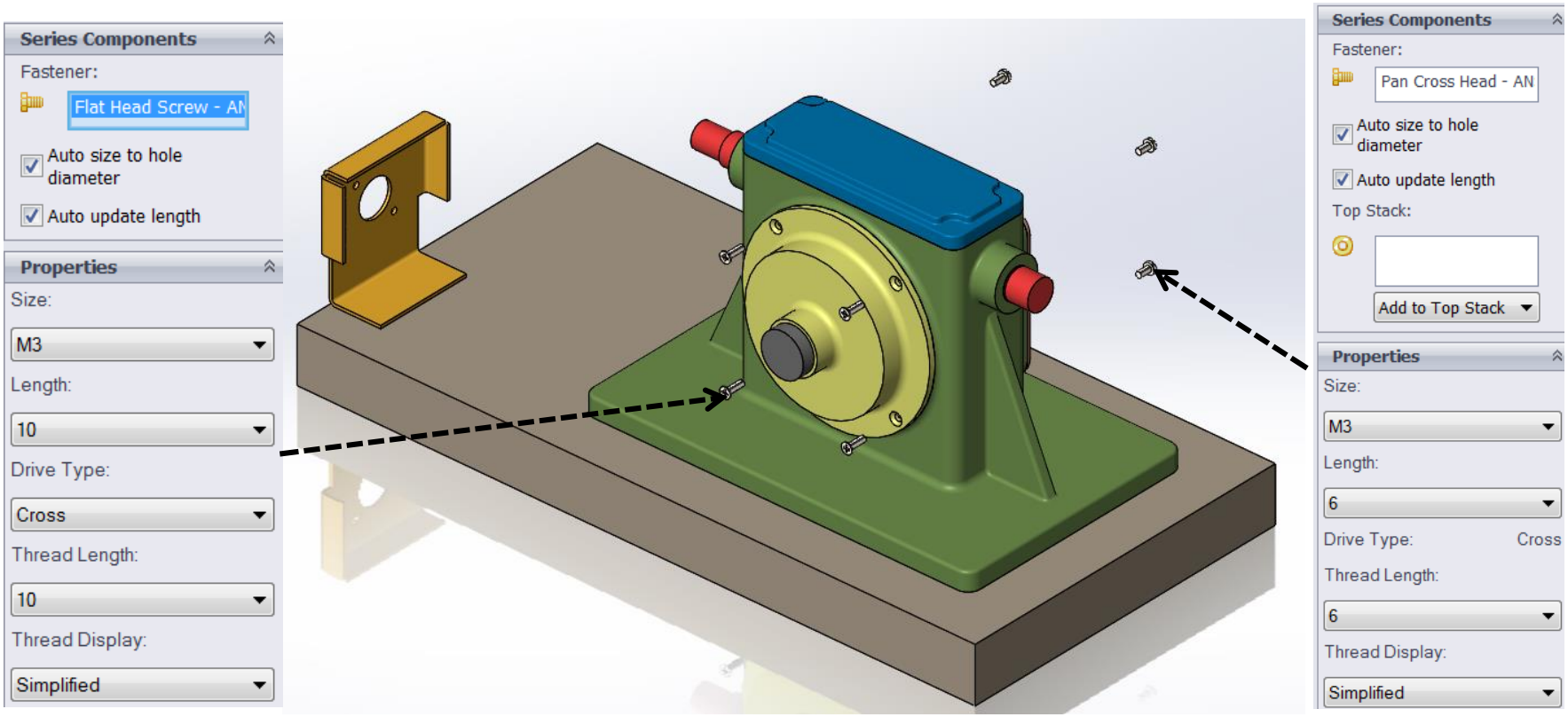
- *Assembly Features*
- *Hole Series*
- *Smart Fasteners*

1. Open an assembly file

Open *TBassy* from *Lession02\Exercises\SmFastenerLab* folder.

2. Smart Fastener

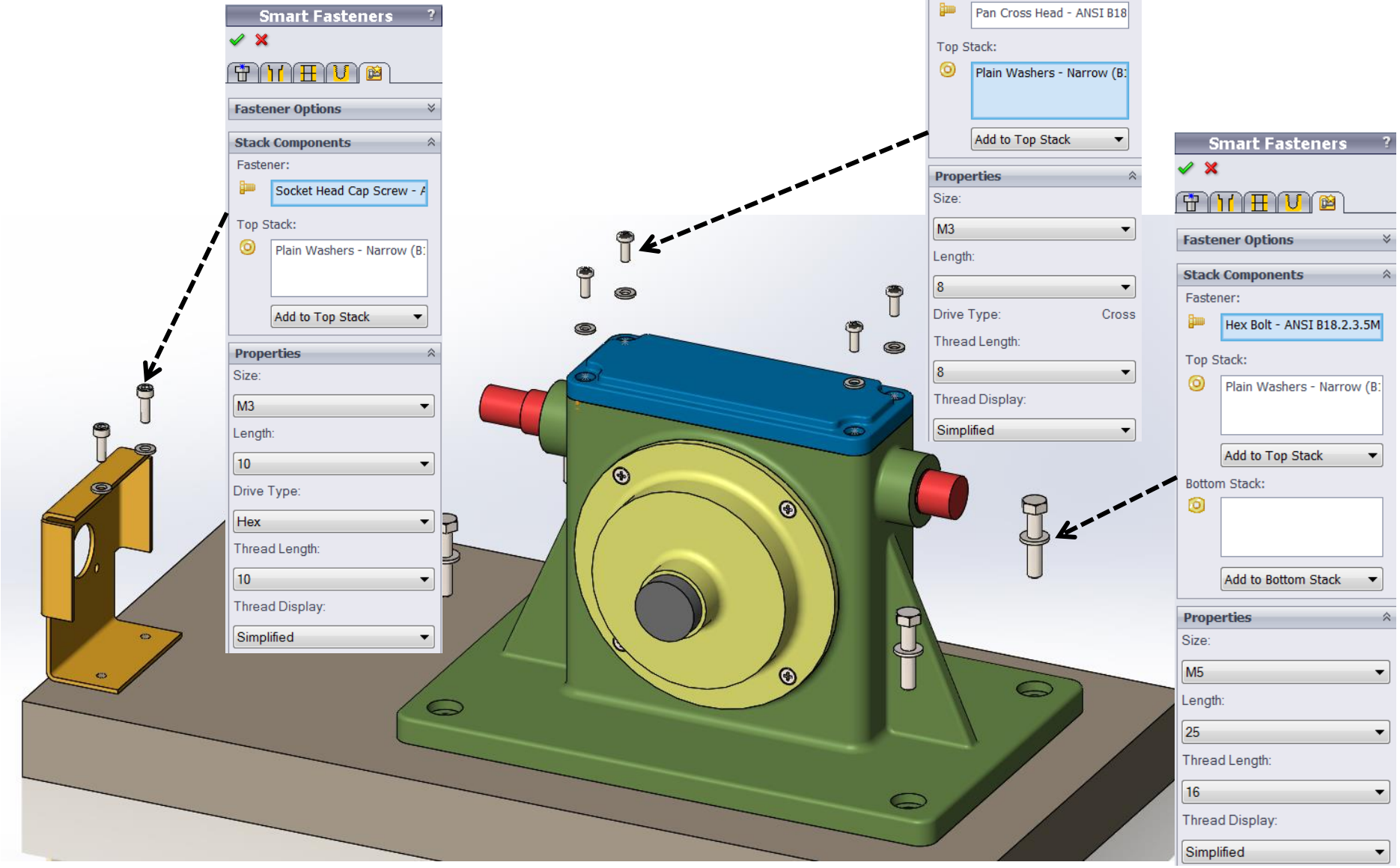
Use **Smart Fastener** to add hardware to the existing holes.



Exercise 2: The Hole Wizard and Smart Fasteners

3. Hole Series

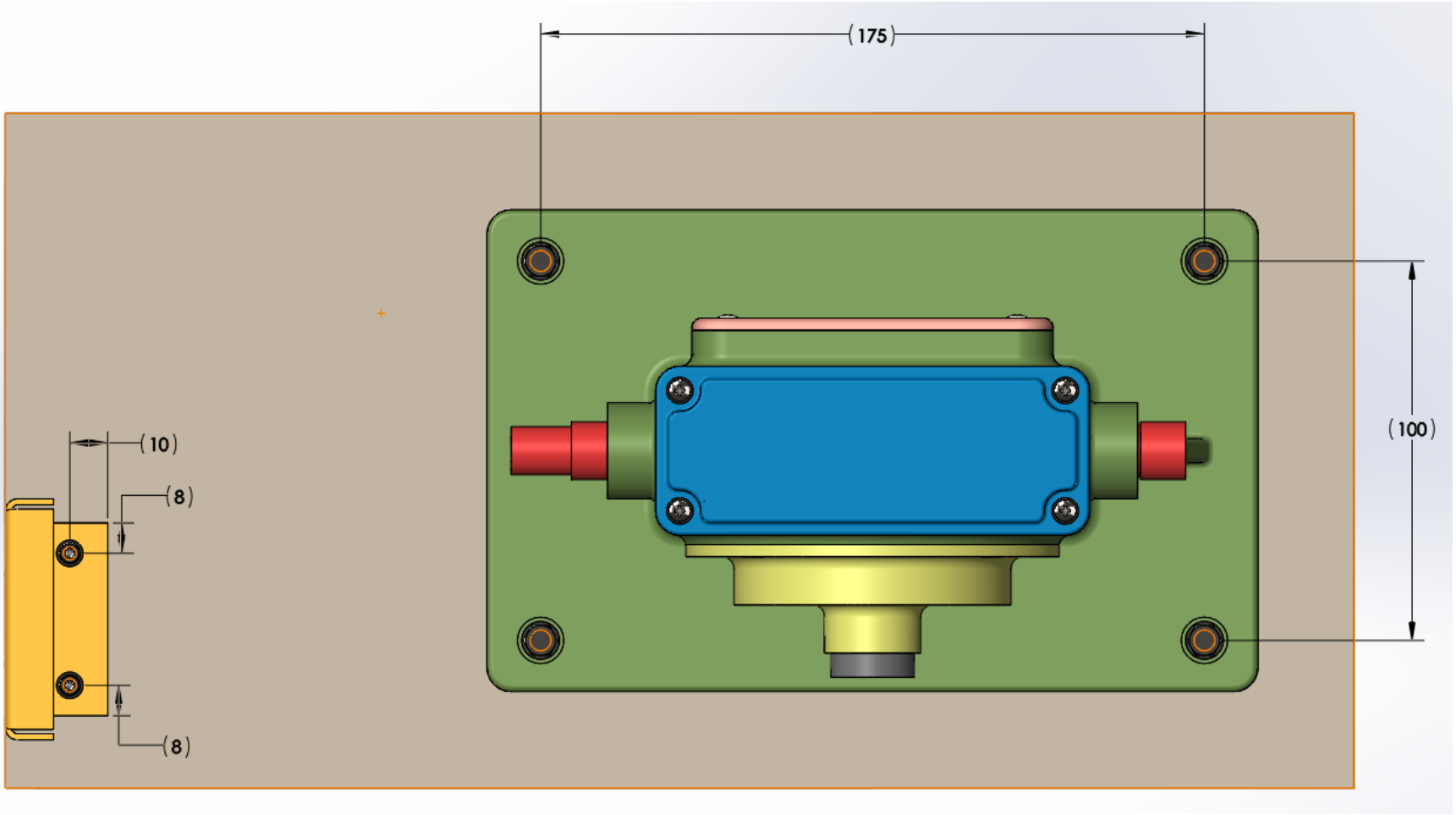
Use **Hole Series** and **Smart Fastener** to add holes and add hardware



Exercise 2: The Hole Wizard and Smart Fasteners

4. Locations

Use these location for Hole Series holes

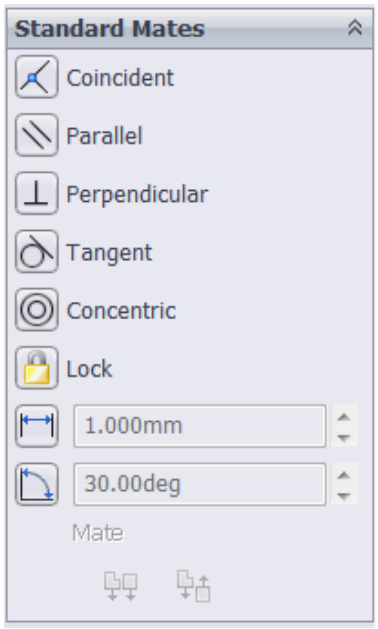


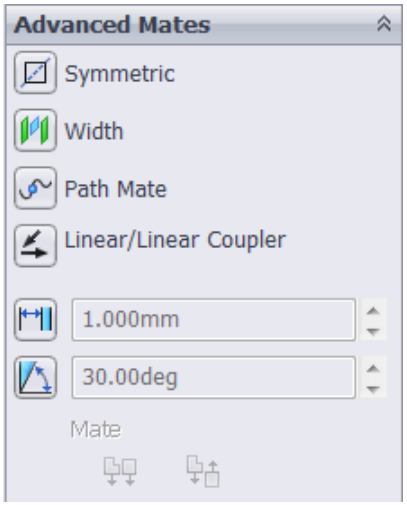
Lession 3: Assembly Mate Techniques

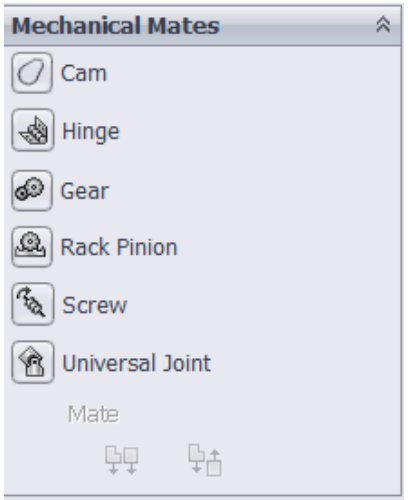
Stages in the Process

- Advanced Mates and Mechanical Mates
- Rack Pinion Mate
- Multiple Mate Mode
- Mate and Collision Detection
- Exercise 3: Planetary Gear Mates

- Standard Mates
- Advanced Mates
- Mechanical Mates



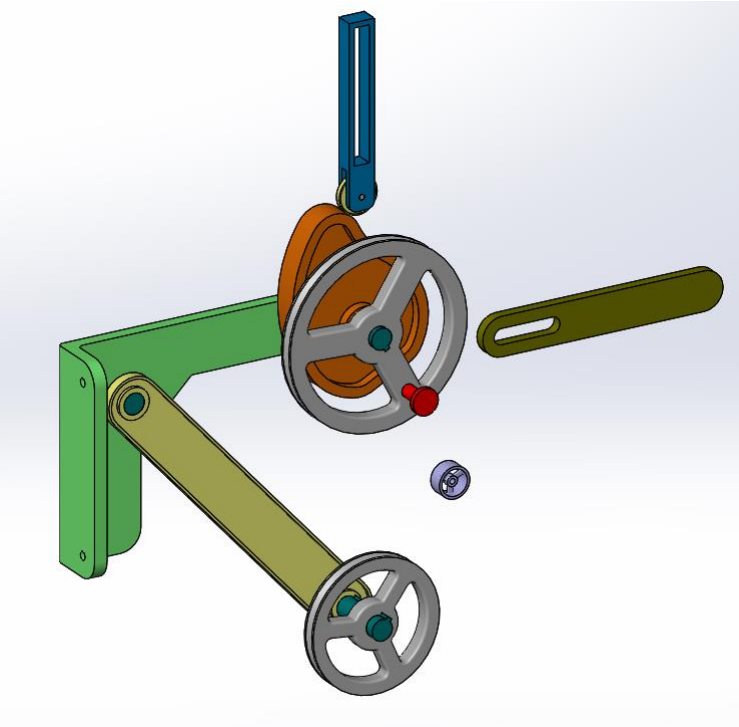




Lession 3: Advanced Mates and Mechanical Mates

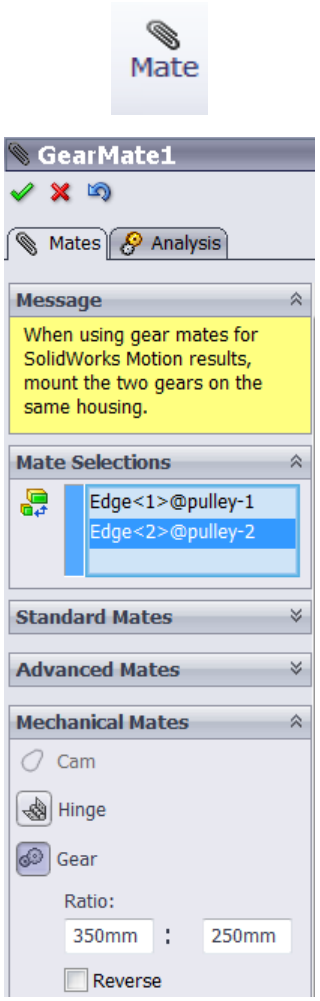
1. Open an assembly file

Open *AdvMates* from *Lession03\Case Study\AdvMates* folder.

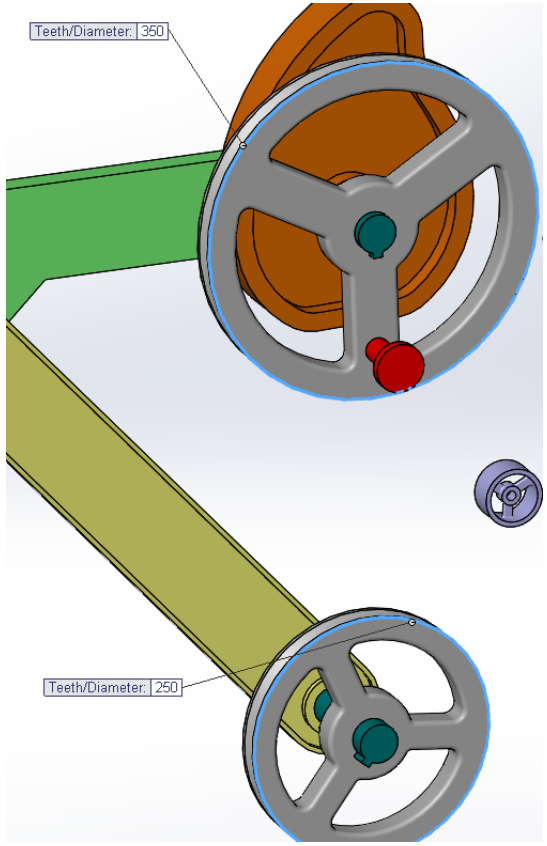


2. Insert mate

Click **Mate** and expand **Mechanical Mates**



3. Define a gear ratio mate between the pulleys.

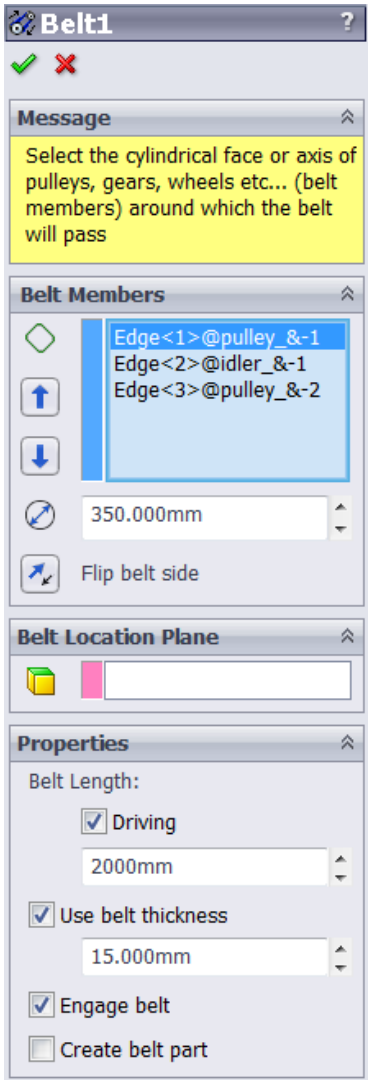


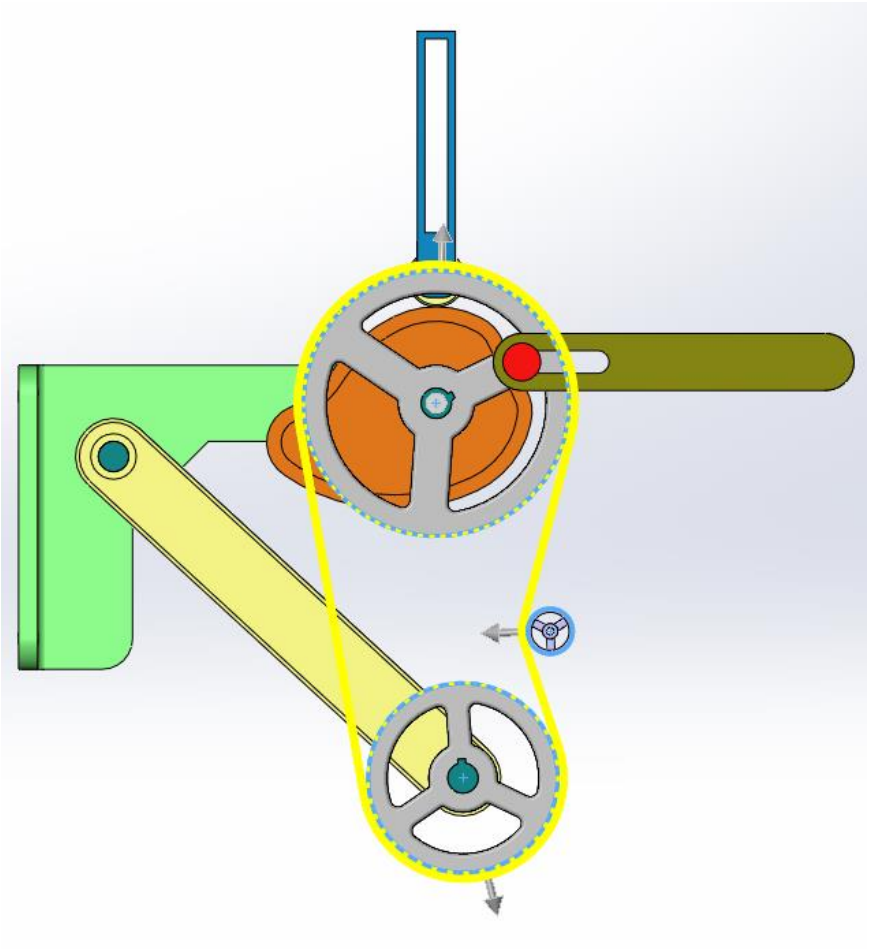
Lession 3: Advanced Mates and Mechanical Mates

The Belt/Chain Assembly Feature

CommandManager: **Assembly > Assembly Feature> Belt/Chain**
Menu: **Insert, Assembly Feature, Belt/Chain** 

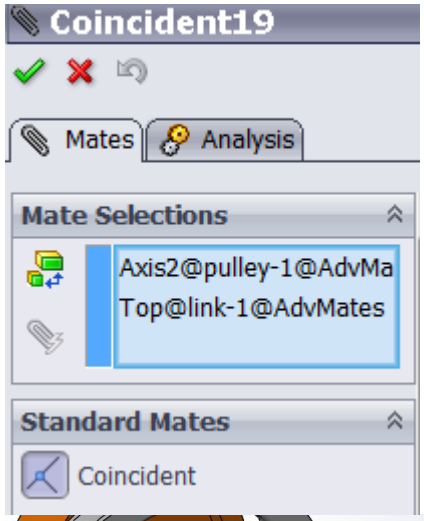
4. Add Belt/Chain feature

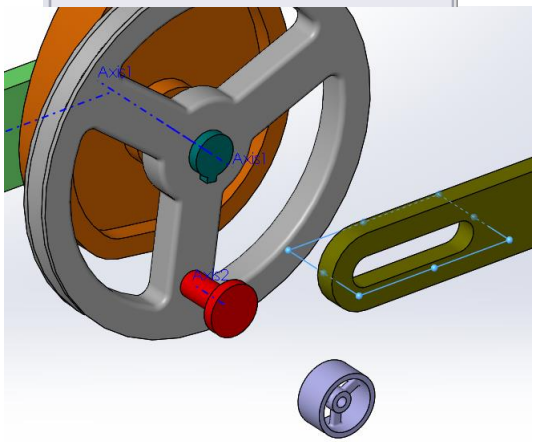




Lession 3: Advanced Mates and Mechanical Mates

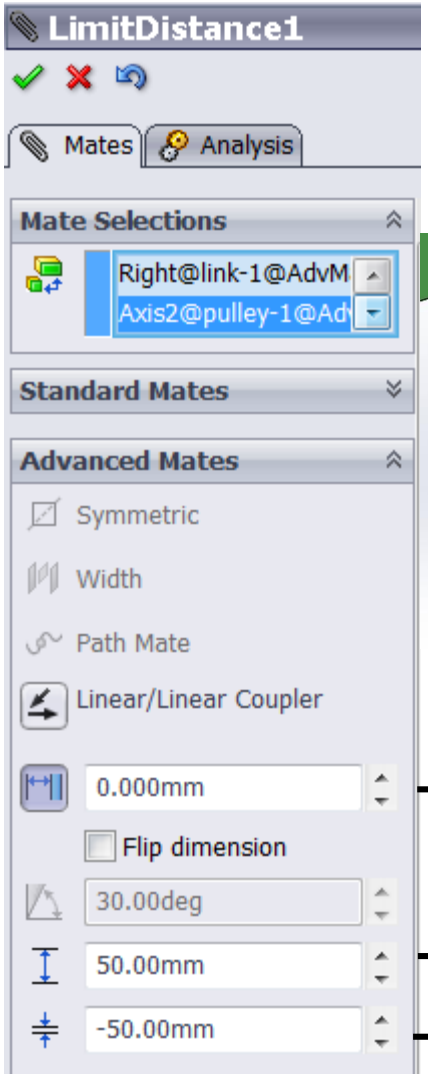
4. Add Concident Mates to the pin-in-slot relationship

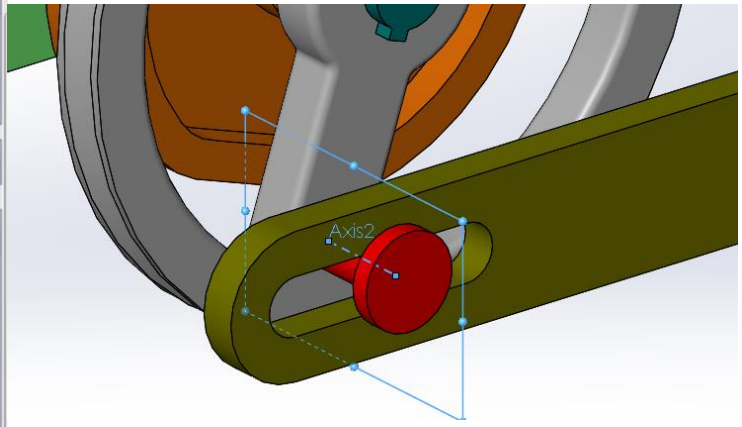




5. Limit Mate

Click on the
Advanced Mates > Distance





--→ Base dimension to **0mm**

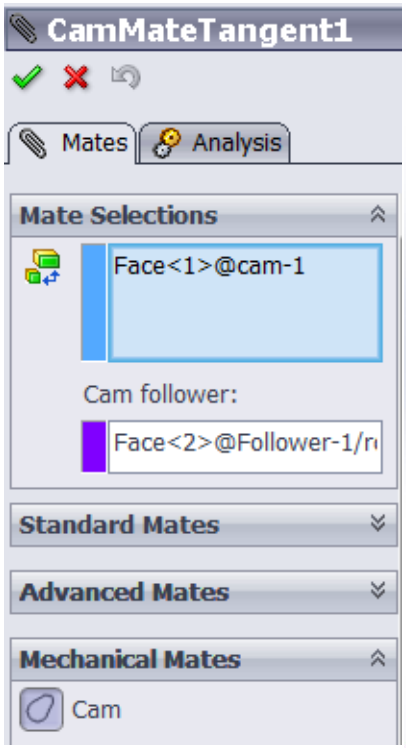
--→ Maximum Value to **50mm**

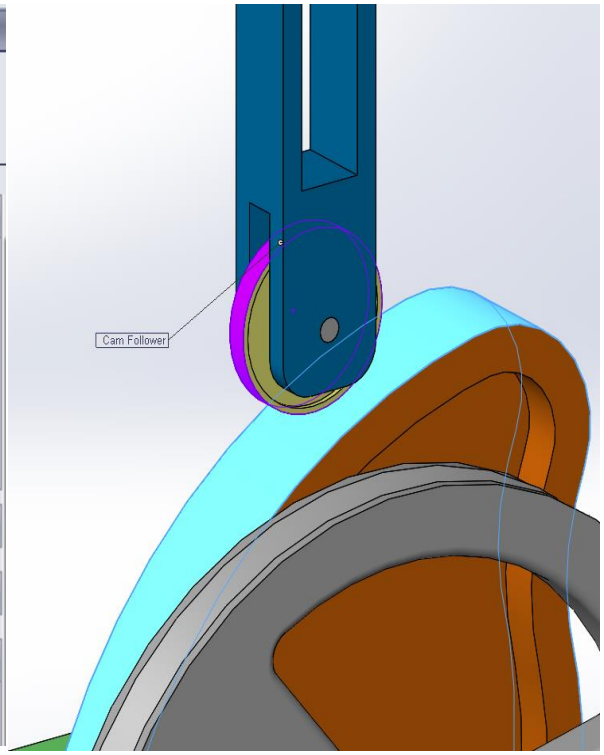
--→ Minimum Value to **-50mm**

Lession 3: Advanced Mates and Mechanical Mates

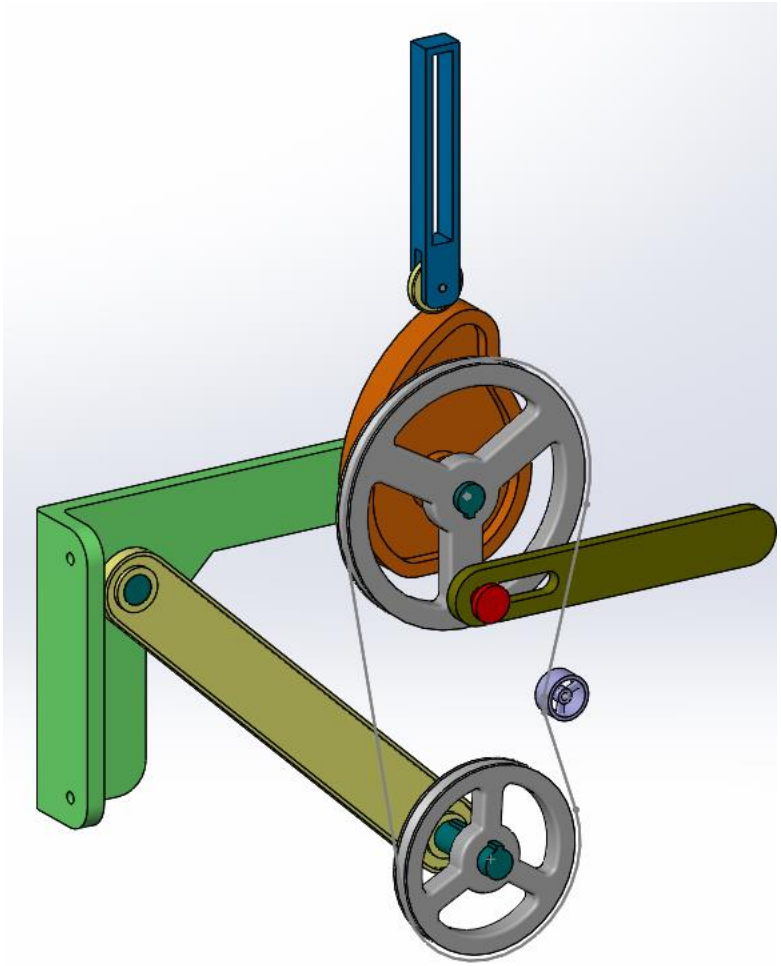
5. Add Cam Mate

Click on the **Mechanical Mates > Cam**





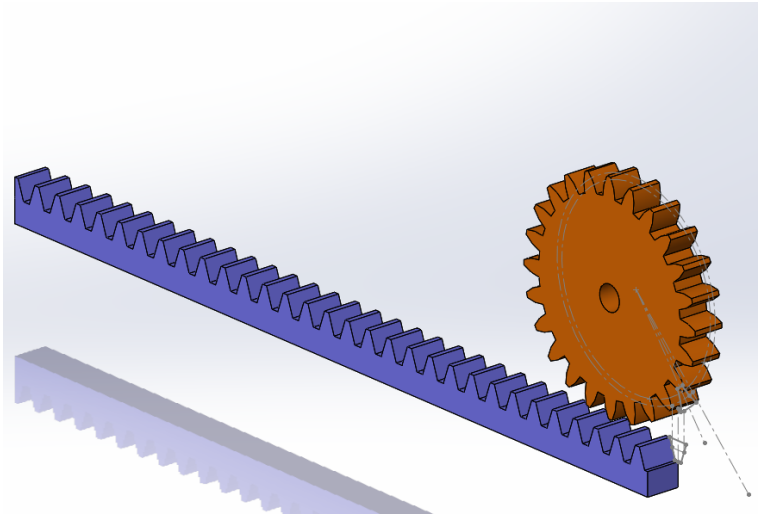
6. Finish



Lession 3: Rack Pinion Mate

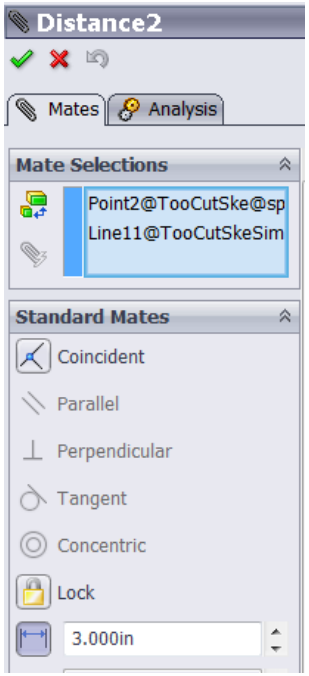
1. Open an assembly file

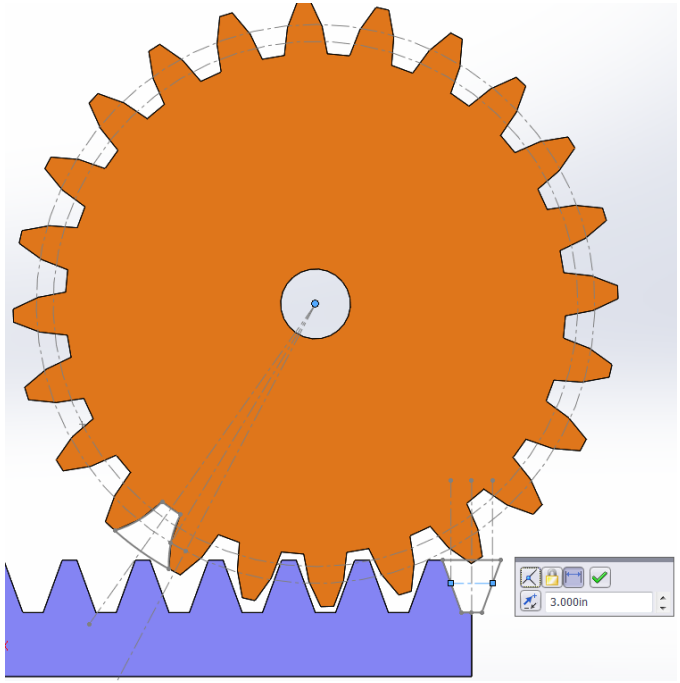
Open *Rack&Pinion* from *Lession03\Case Study\RackPinionMate* folder.



2. Tangency

Tangent relationship between **spur gear** and **rack**. Using **distance mate**.

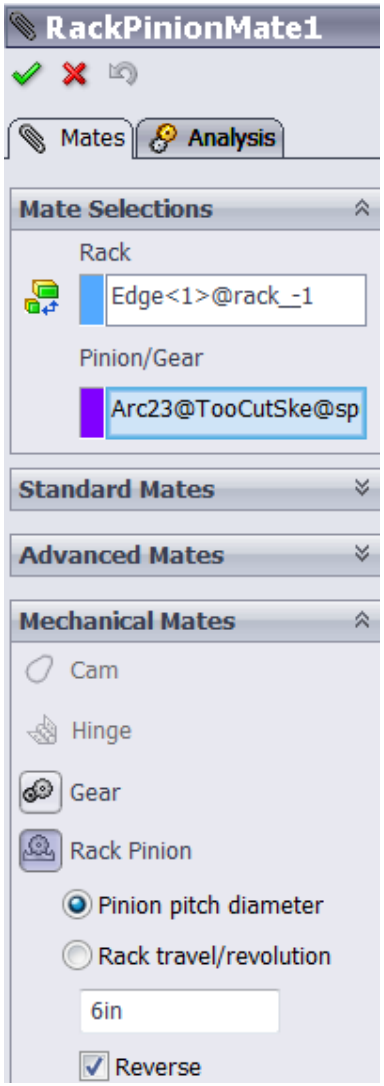




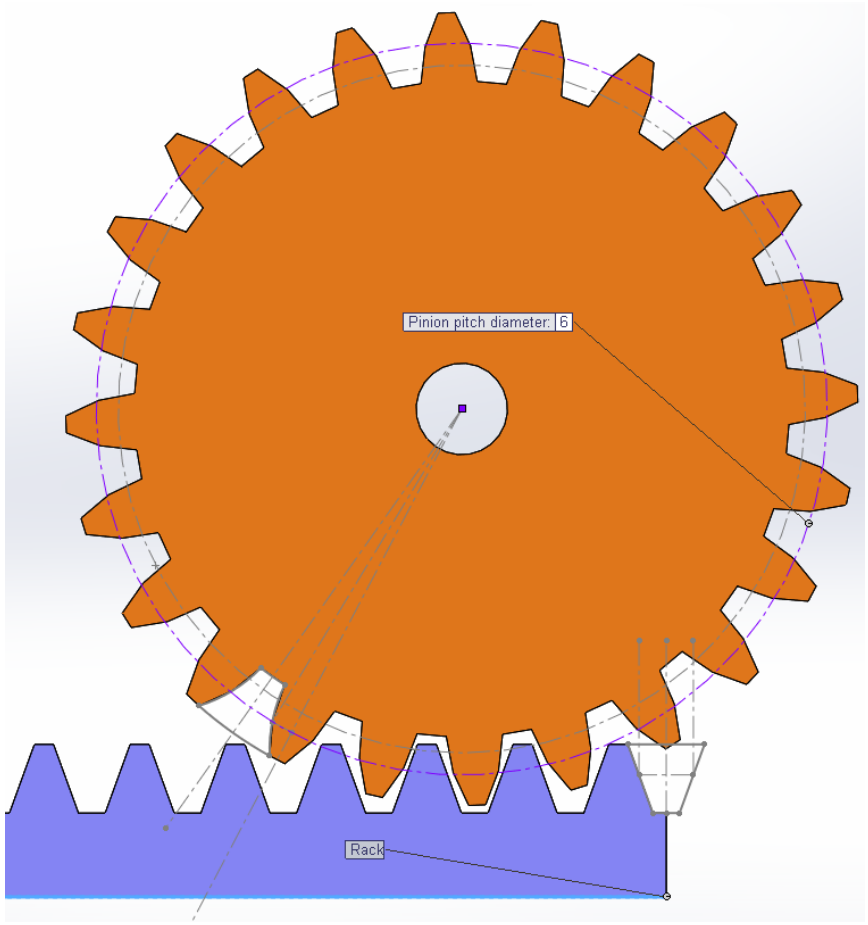
Lession 3: Rack Pinion Mate

3. Rack Pinion Mate

Click **Mate** and expand **Mechanical Mates > Rack Pinion**



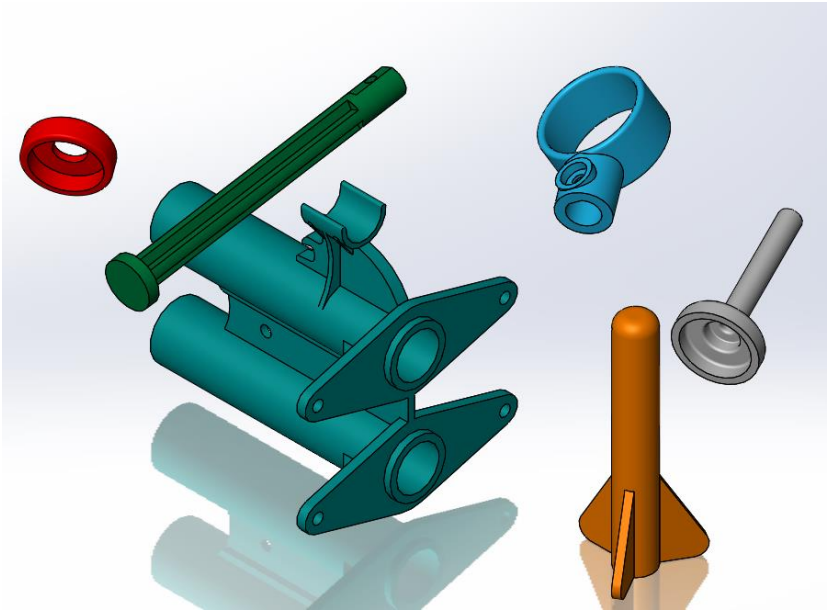
6. Finish



Lession 3: Multiple Mate Mode

1. Open an assembly file

Open *Multiple_Mates* from *Lession03\Case Study\ Multiple Mates* folder.

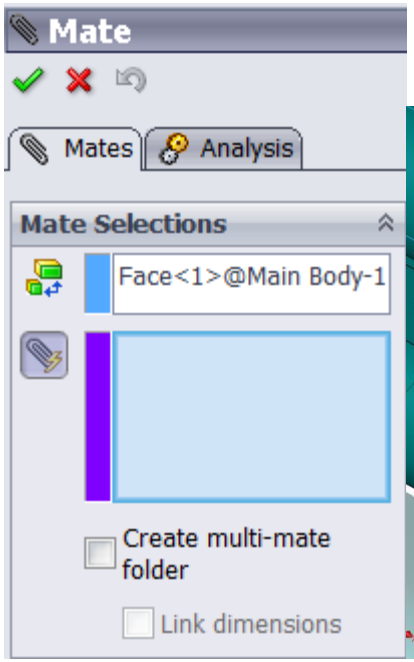


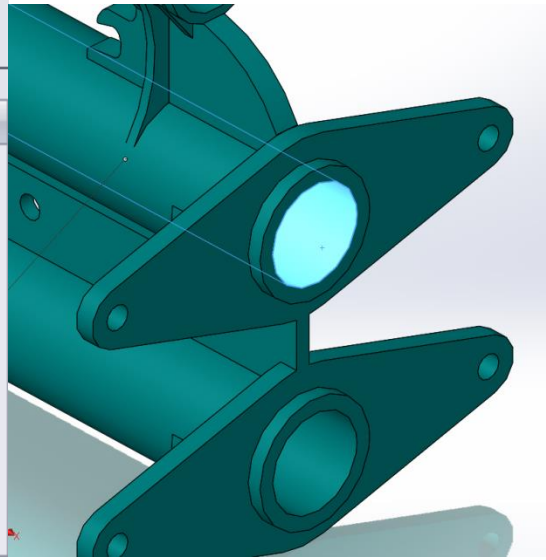
2. Select common face

Click **Mate**

Click **Multiple mate mode**

Select Main Face

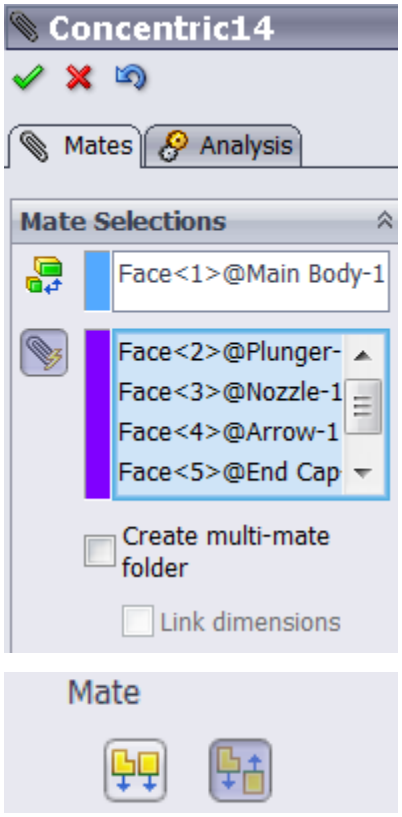


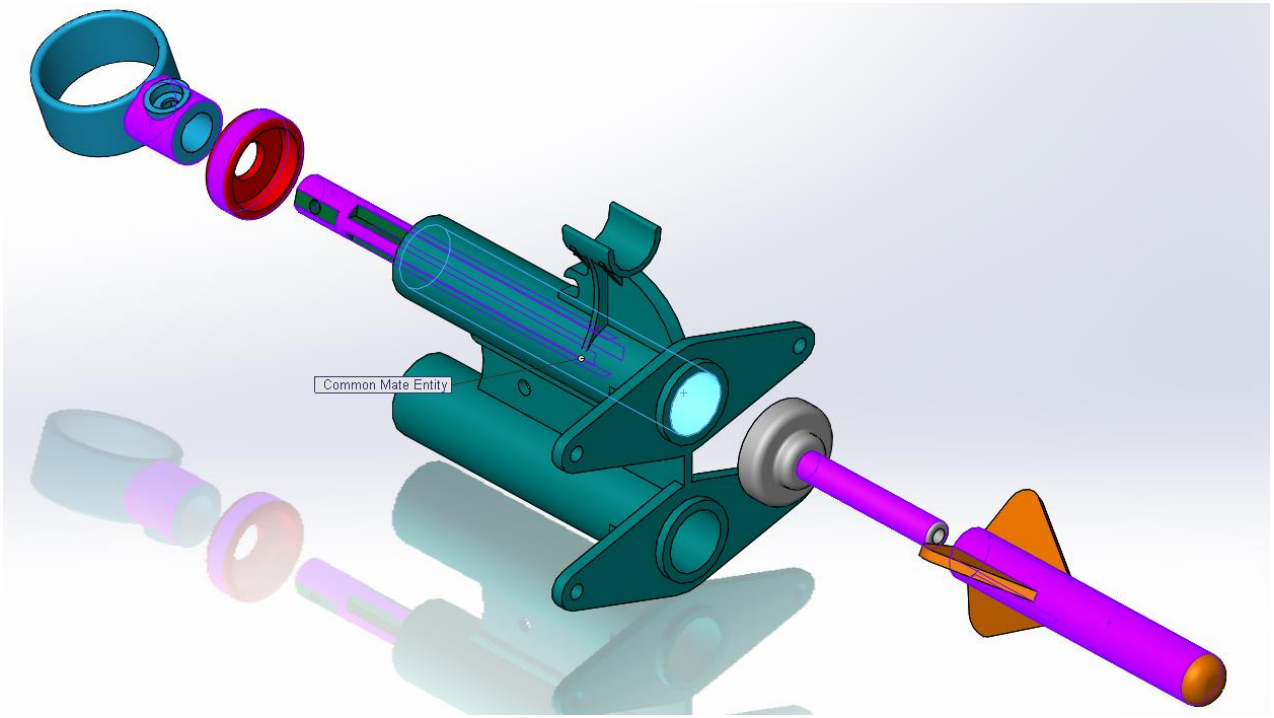


Lession 3: Multiple Mate Mode

3. Multiple mate selection

Select multiple face and using Alignment

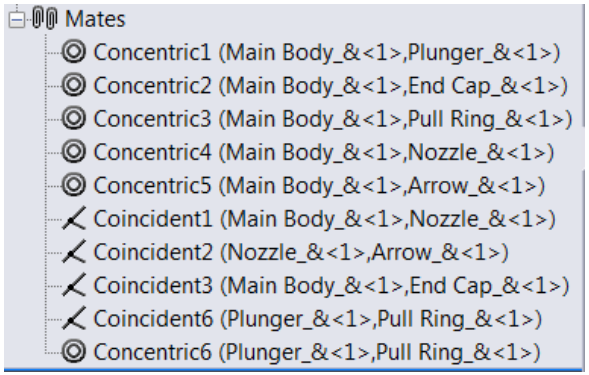


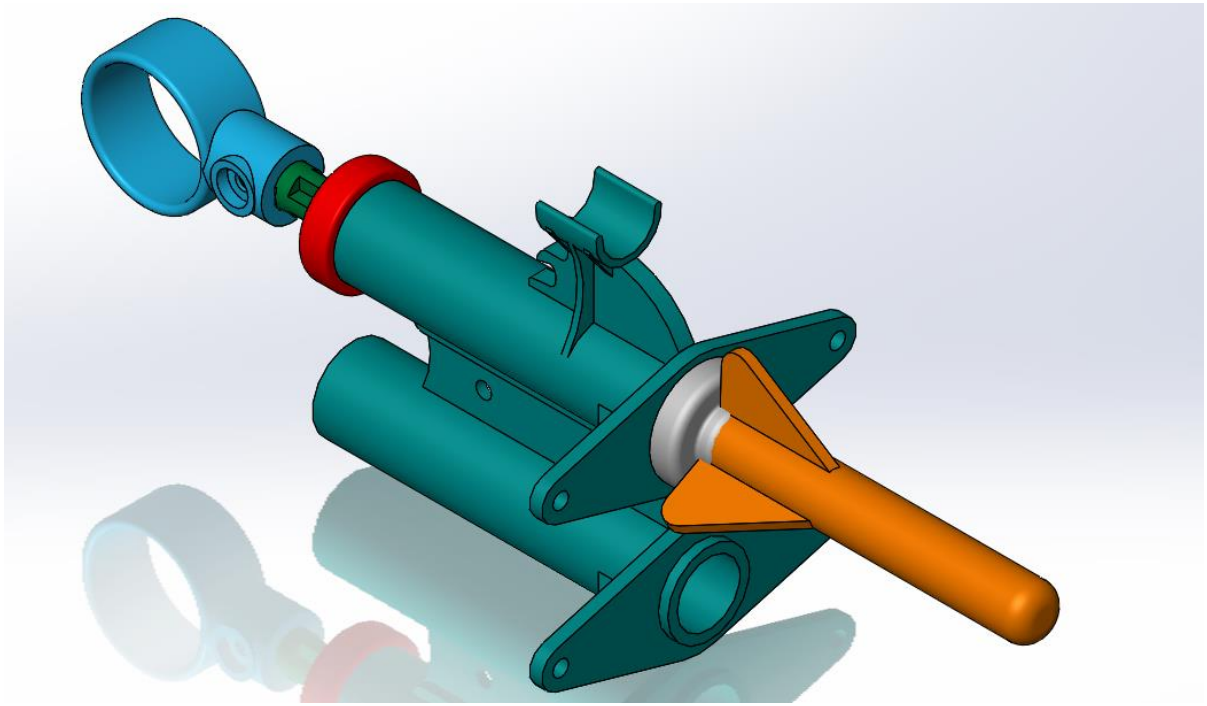


Lession 3: Multiple Mate Mode

4. Completed

Complete the mating by moving components and adding mates

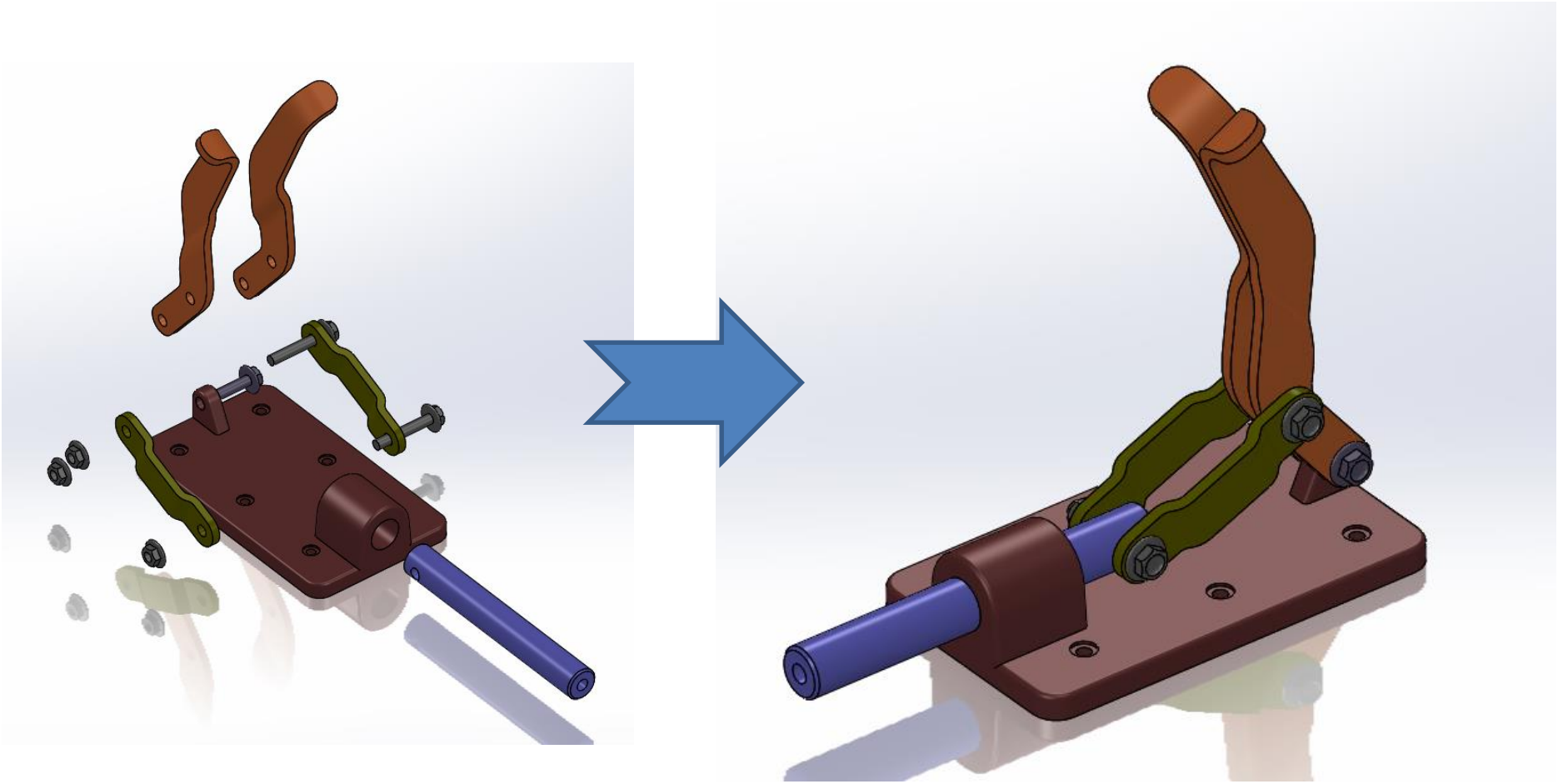




Lession 3: Mate and Animation

1. Create an assembly file

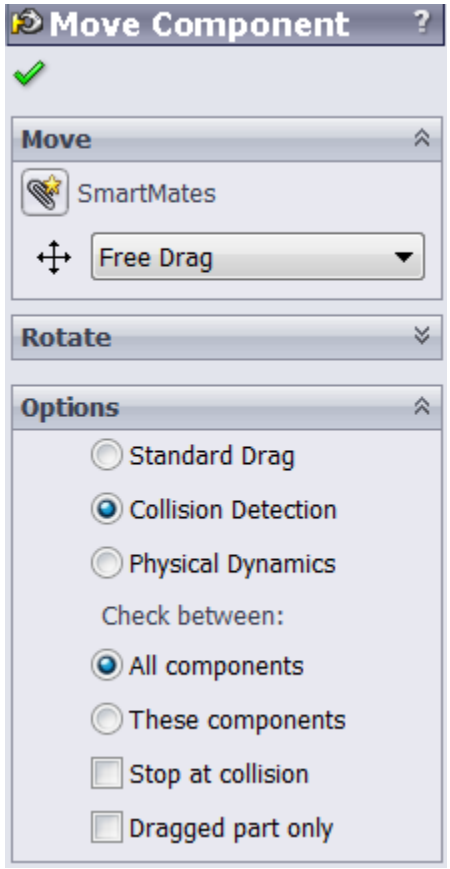
Create *Assembly_IN* from *Lession03\Exercise\ MateRef* folder

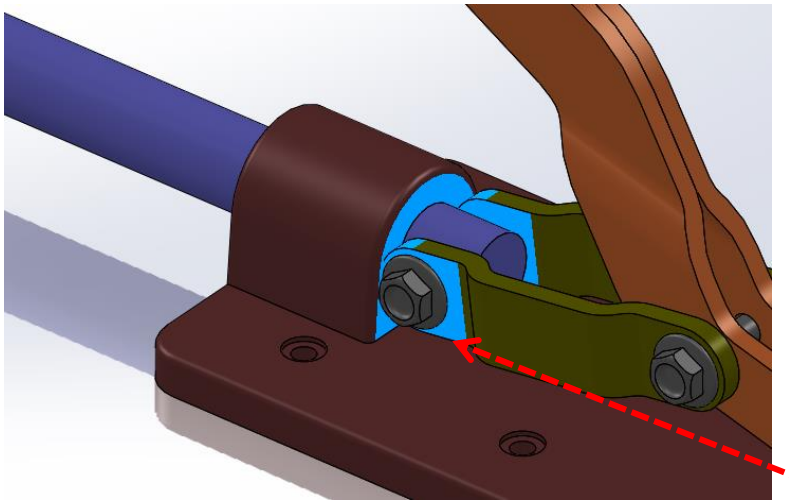


Lession 3: Mate and Animation

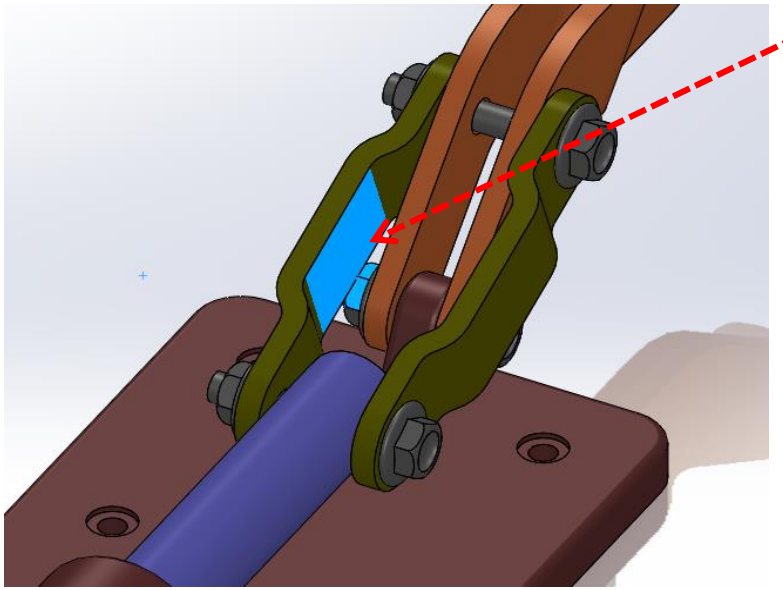
2. Dynamic collision detection

Click **Move Component** and turn on **Collision Detection.**





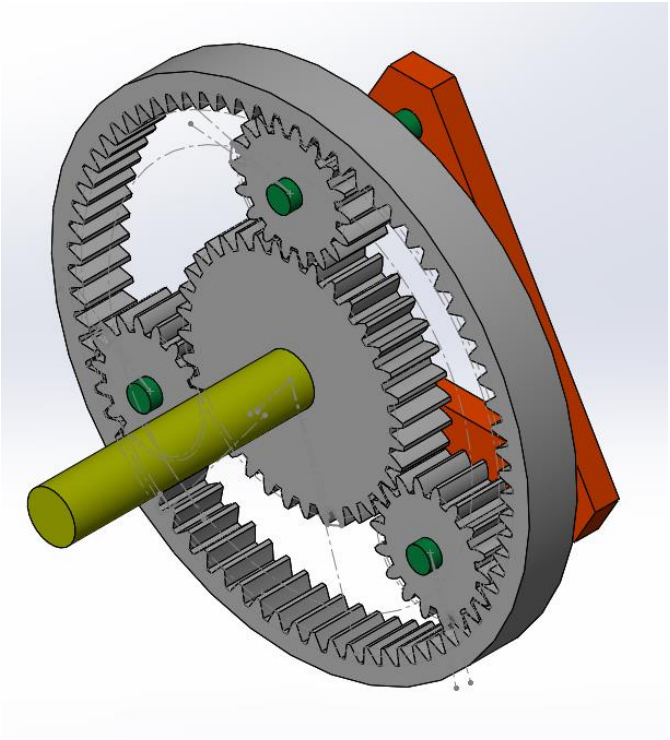
Collision Detection

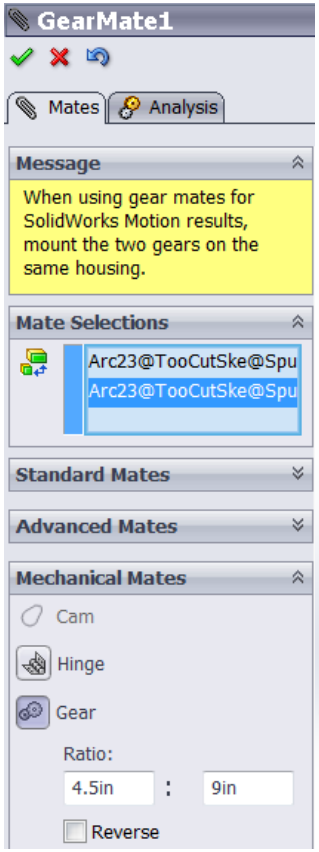


Exercise 3: Planetary Gear Mates

1. Open an assembly file

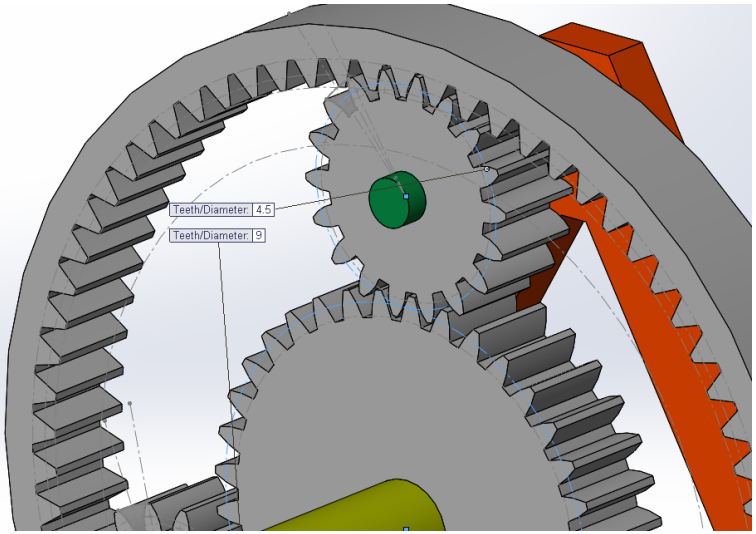
Open *Gear* from *Lession03\Excercises\Gear* folder.





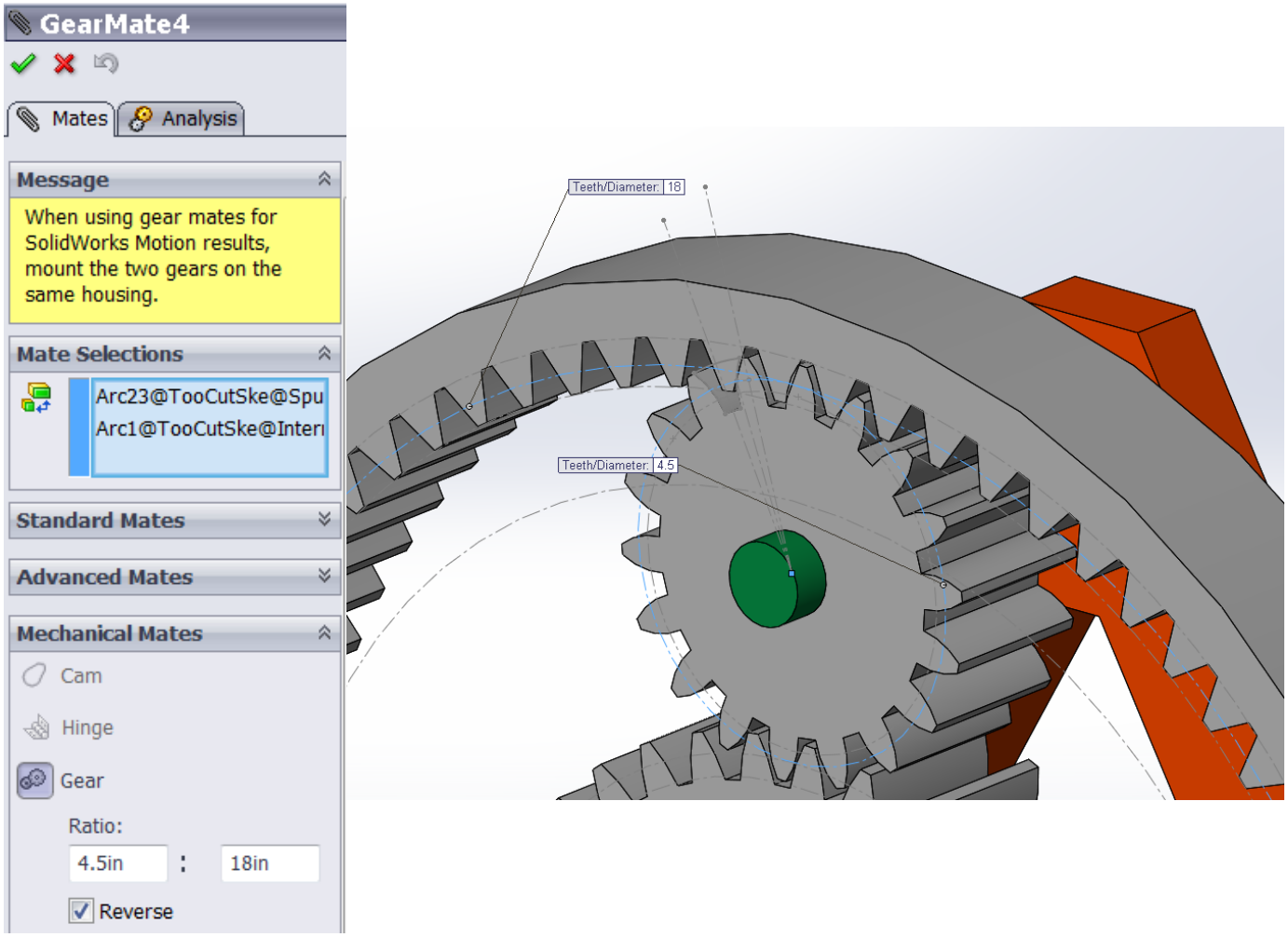
2. Mate the planetary Spur gears to the central Spur gear

Ratio is **2 : 1**



Exercise 3: Planetary Gear Mates

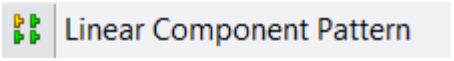
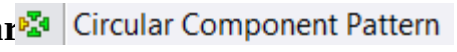
3. Mate **Internal Spur Gear** to one of small planetary gears
Ratio is **4 : 1**



Lession 4: Using Configurations with Assemblies

- Pattern components
- Create a configuration of an assembly.
- Use configure component to automate the creation of configurations
- Create a custom Property Manager for a part

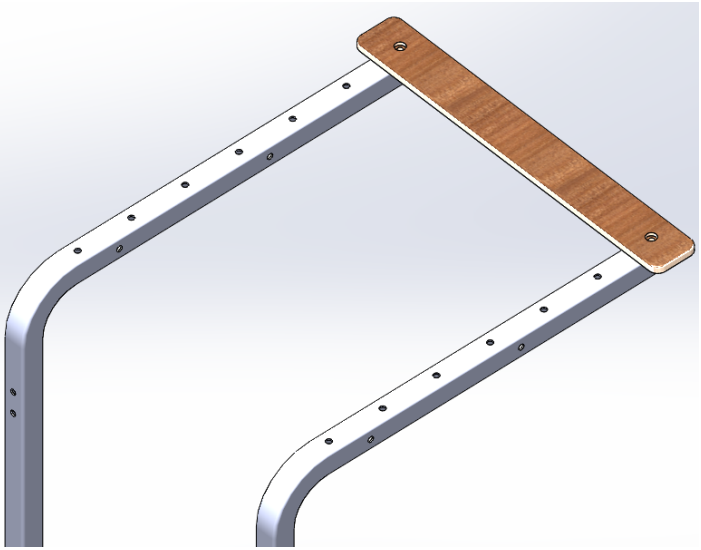
Components Pattern

Componenent Pattern	Based on Part-level Feature or Hole
Linear 	None
Circular 	None
Feature Driven 	Sketch Driven Table Driven Curve Driven Fill Hole Series Hole Wizard

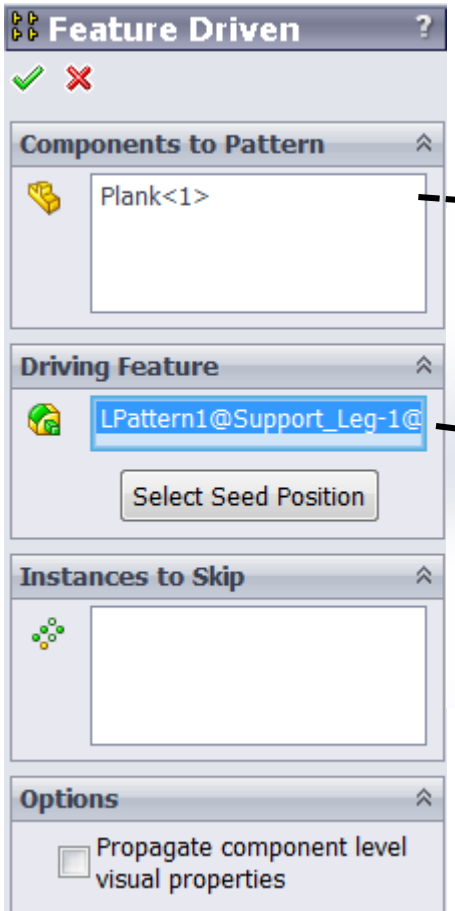
Lession 4: Components Pattern

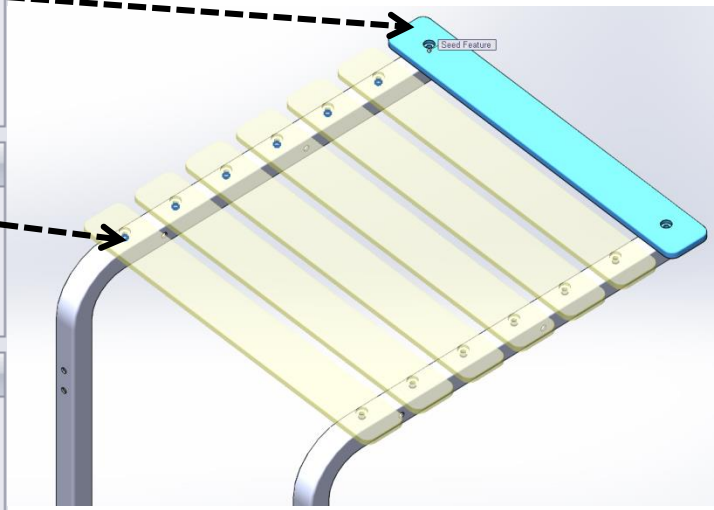
CommandManager: **Assembly > Linear Component Pattern> Feature Driven Component Pattern**
Menu: **Insert, Component Pattern, Feature Driven**

1. Open an assembly file
Open *Support_Frame* from *Lession04\Case Study* folder.



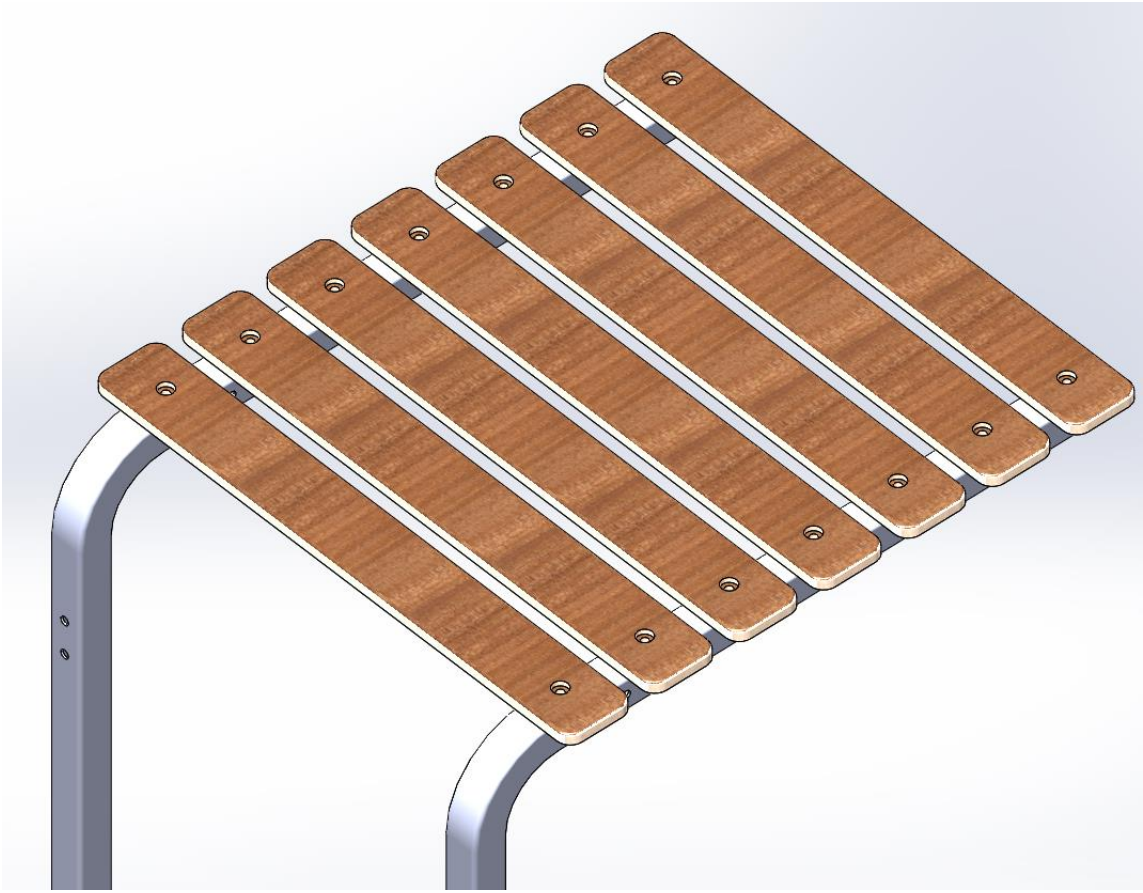
2. Feature Driven Component Pattern





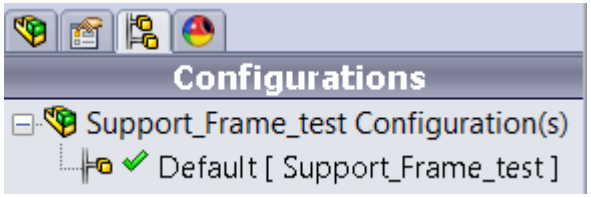
Lession 4: Components Pattern

3. Component

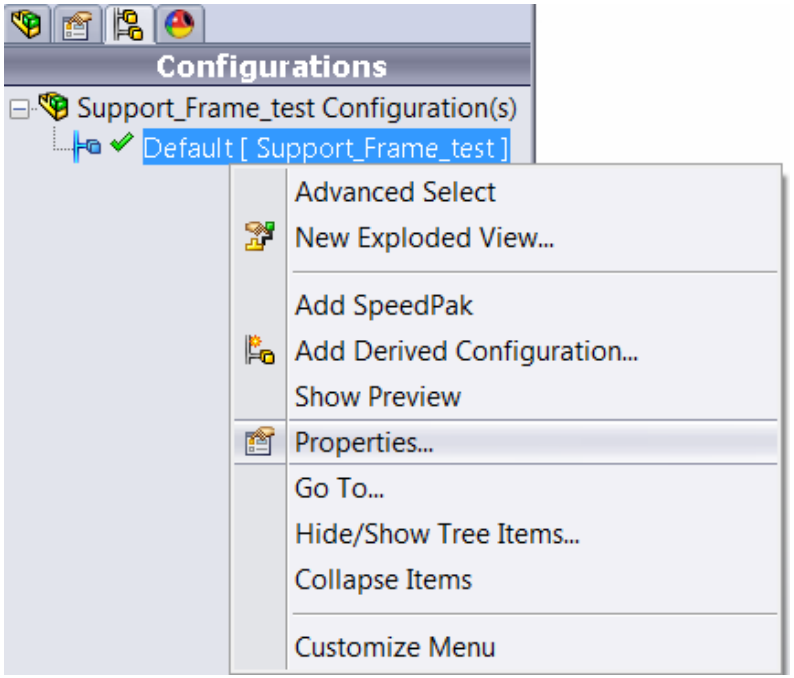


Lession 4: Creating Configurations Manually

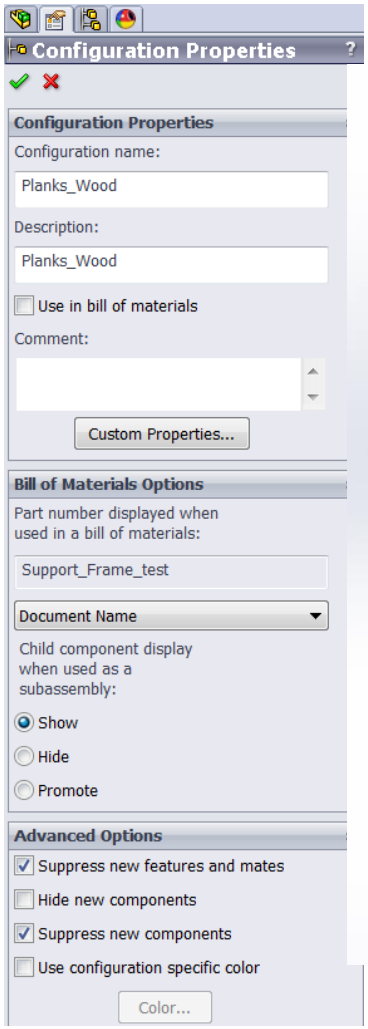
CommandManager: Right-click the top-level icon and click **Add Configuration**
Shortcut Menu: Right-click the component and click **Add Configuration**

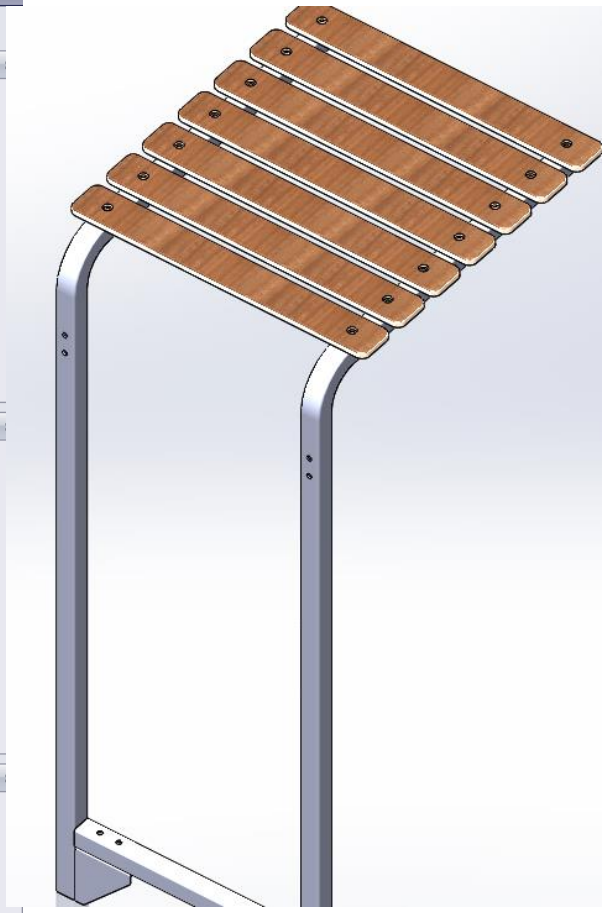


4. Configuration properties



5. Rename configuration





Lession 5: Display States and Appearances

Display States set the visibility, color, texture,display mode and transparency of components at the assembly level.

- **Create new display states**
- **Change appearances of parts and components**
- **Change appearances of parts and components**
- **Change scene**
- **Edit the material**

Display States vs Configurations

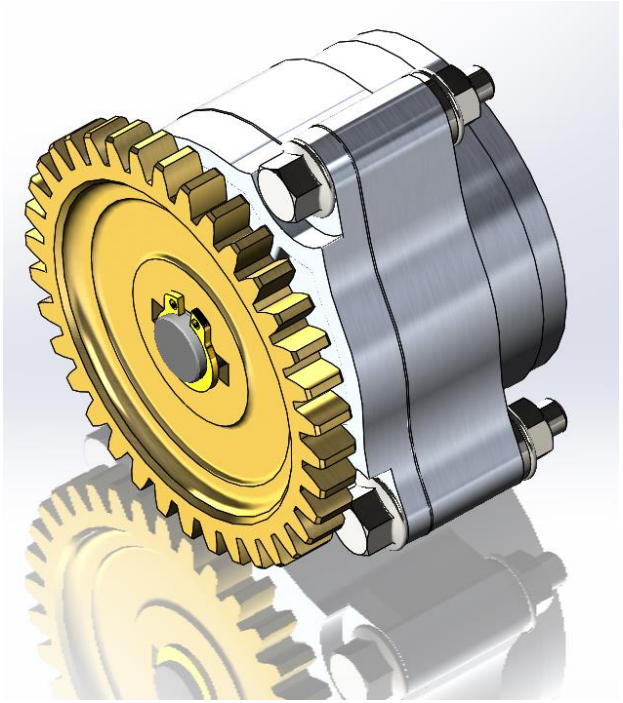
Configurations	Display States
■ Suppress/ Resolve component ■ Part configurations ■ Part meterial properties ■ Component positioning	■ Hide/Show components ■ Appearances (textures and color) ■ Display mode (HLR, Shaded) ■ Transparency

Exercise 5: Display States, Appearances and Materials

Complete this assembly by creating new display states and adding appearances and material

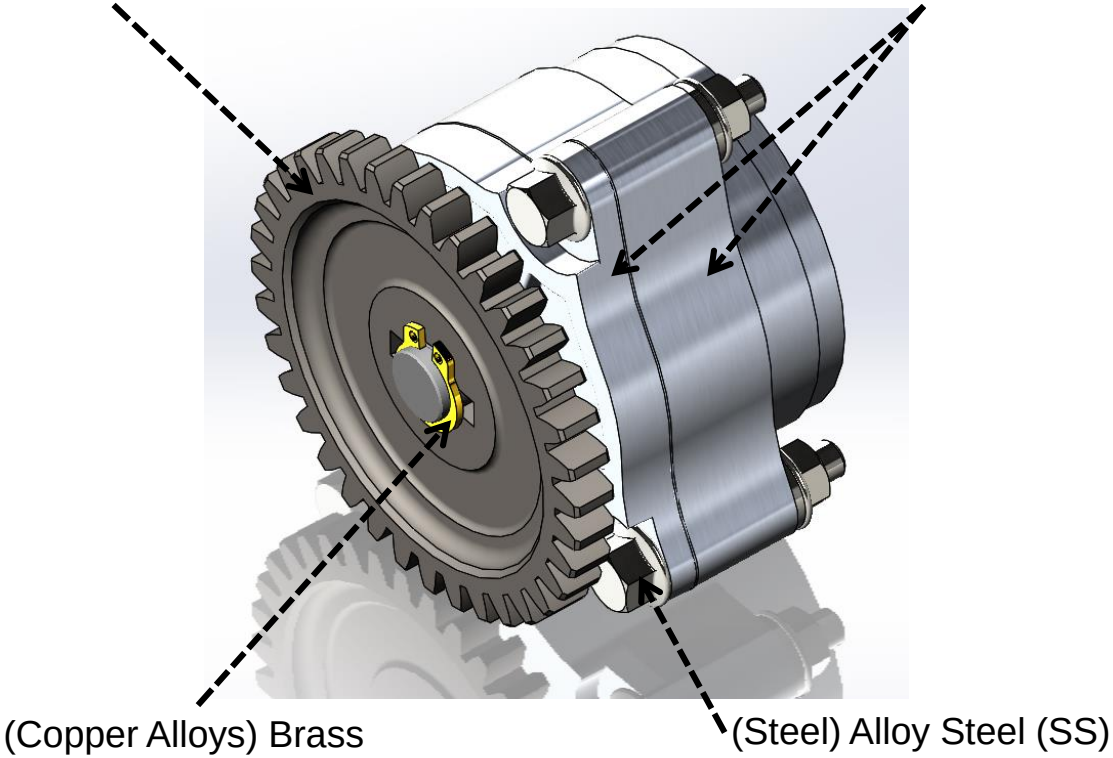
1. Open an assembly file

Open *Display States 2* from *Lession05\Exercises\Display State 2* folder.



2. Material

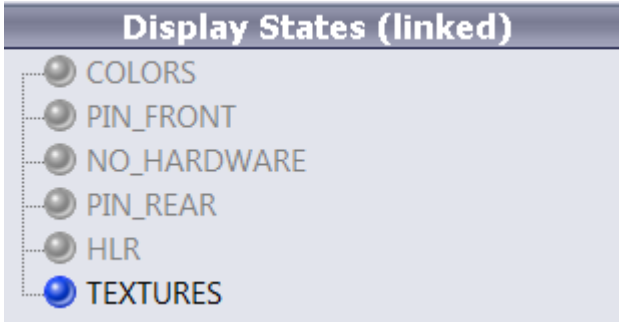
Add the following material to these component

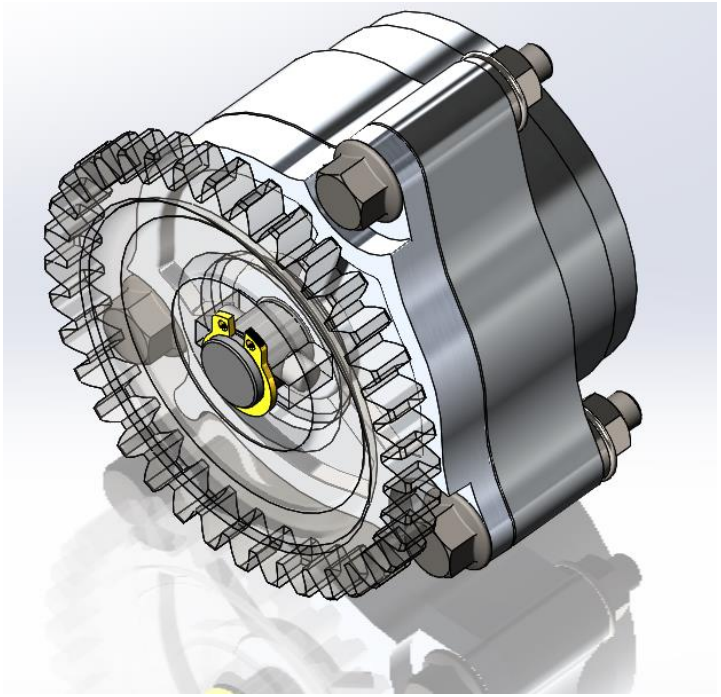
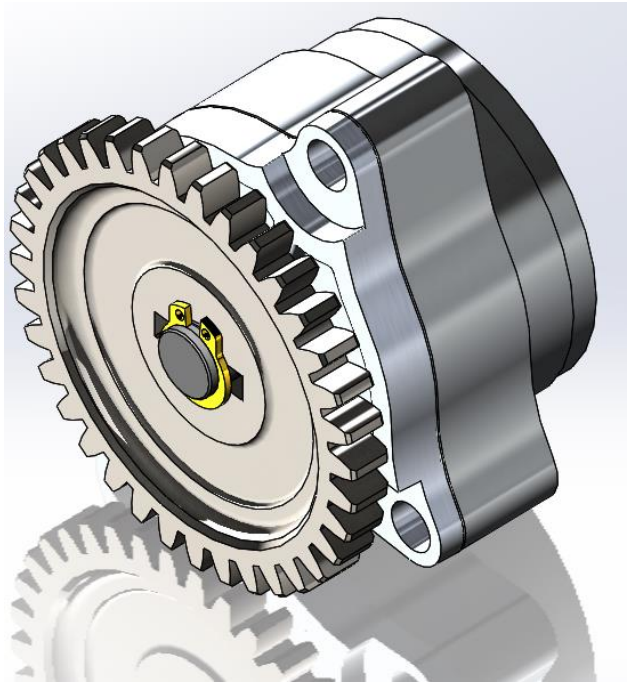


Exercise 5: Display States, Appearances and Materials

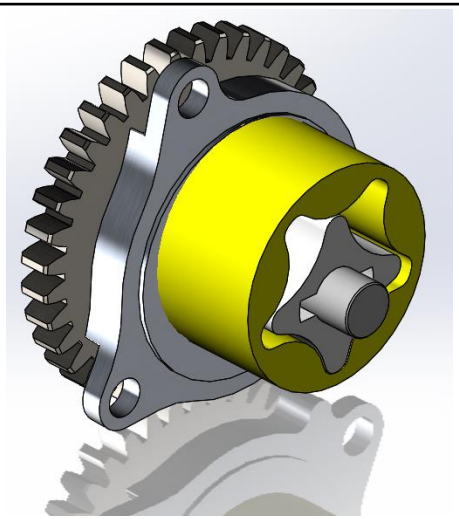
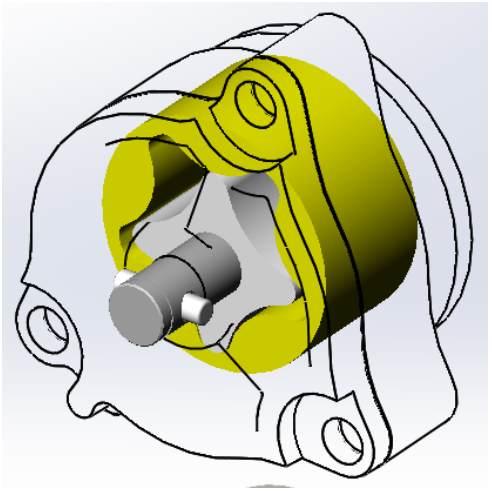
3. Display States

Add the following display states using these names and changes



PIN_FRONT	NO_HARDWARE
	

Exercise 5: Display States, Appearances and Materials

PIN_REAR	HLR
	
COLORS	**TEXTURES**