SOLIDWORKS ASSEMBLY DESIGN COMMANDS CHEAT SHEET

Essential Assembly Design Commands Every SolidWorks User Should Know

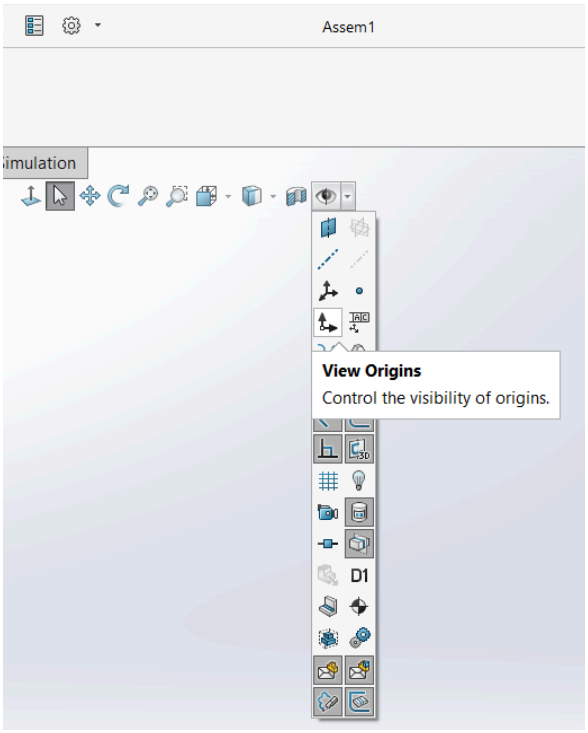
1. Insert Components

What it does: Adds parts or subassemblies to an assembly.

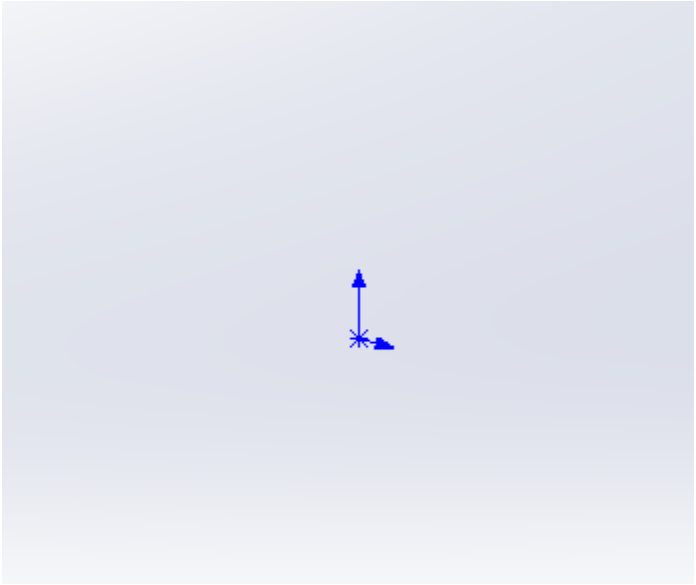
Quick Tip: The first component is commonly used as the foundation for positioning the rest of the assembly.

How it works:

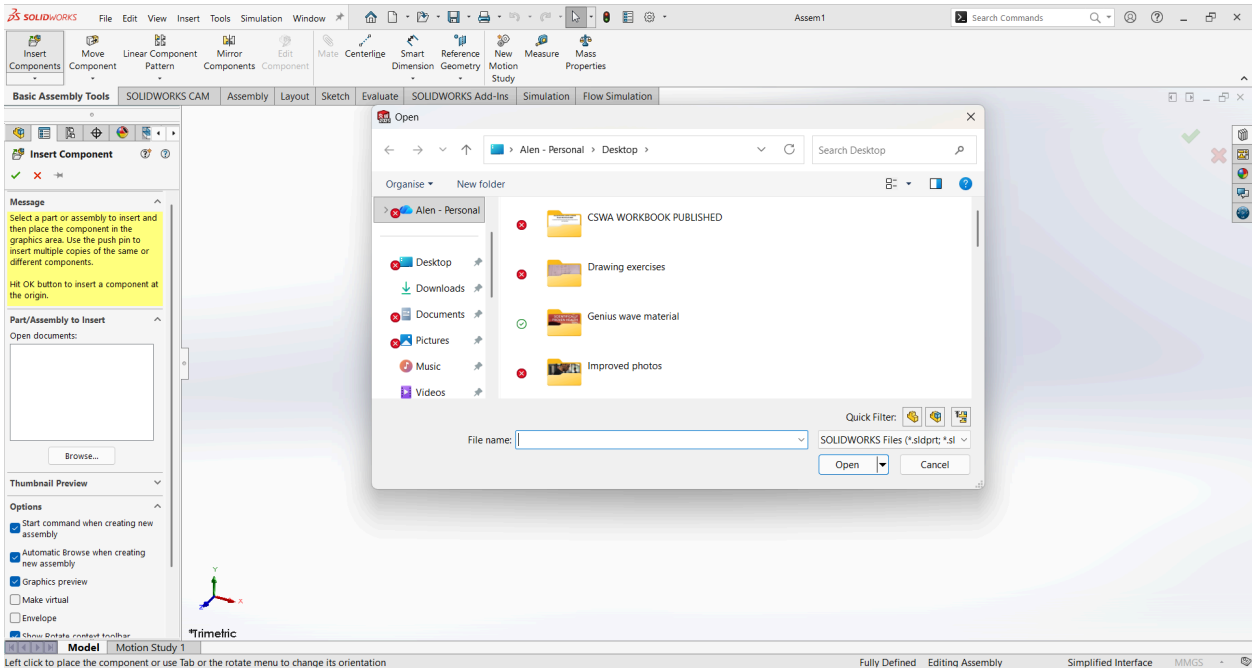
Now we will insert components into the assembly. Before we do that, let's make the assembly origin visible. Go to the **Heads-up View Toolbar**, click the small arrow next to the eye icon, then select **View Origins**.



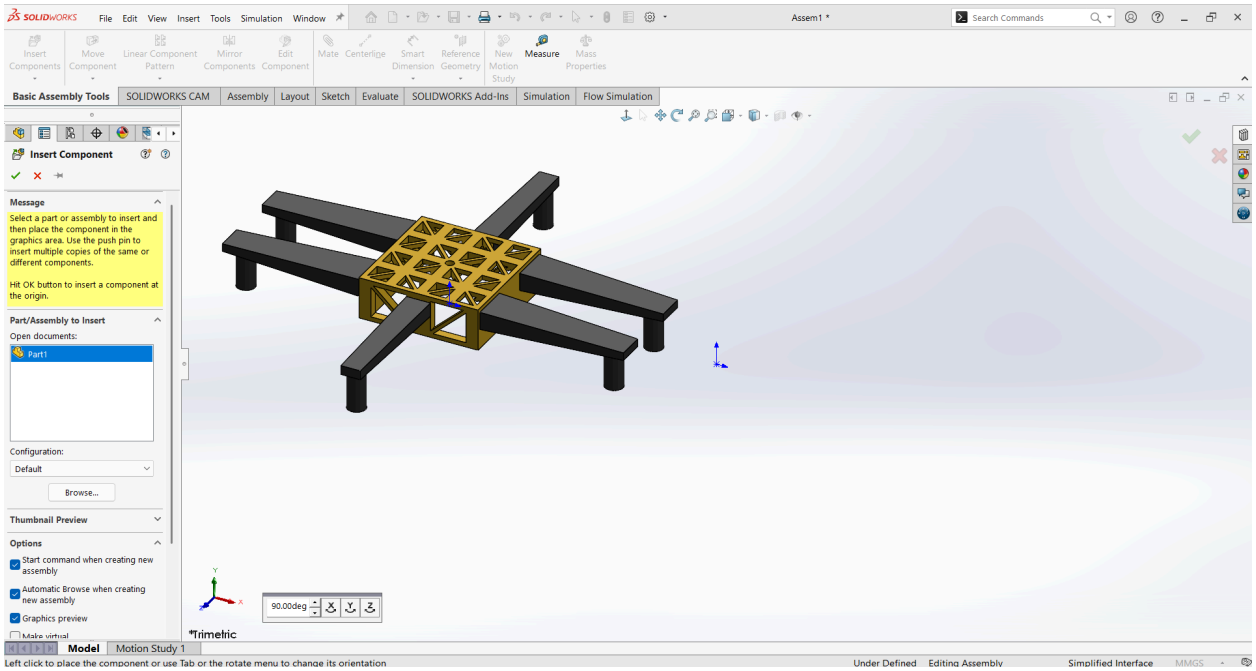
Now the assembly origin is visible.



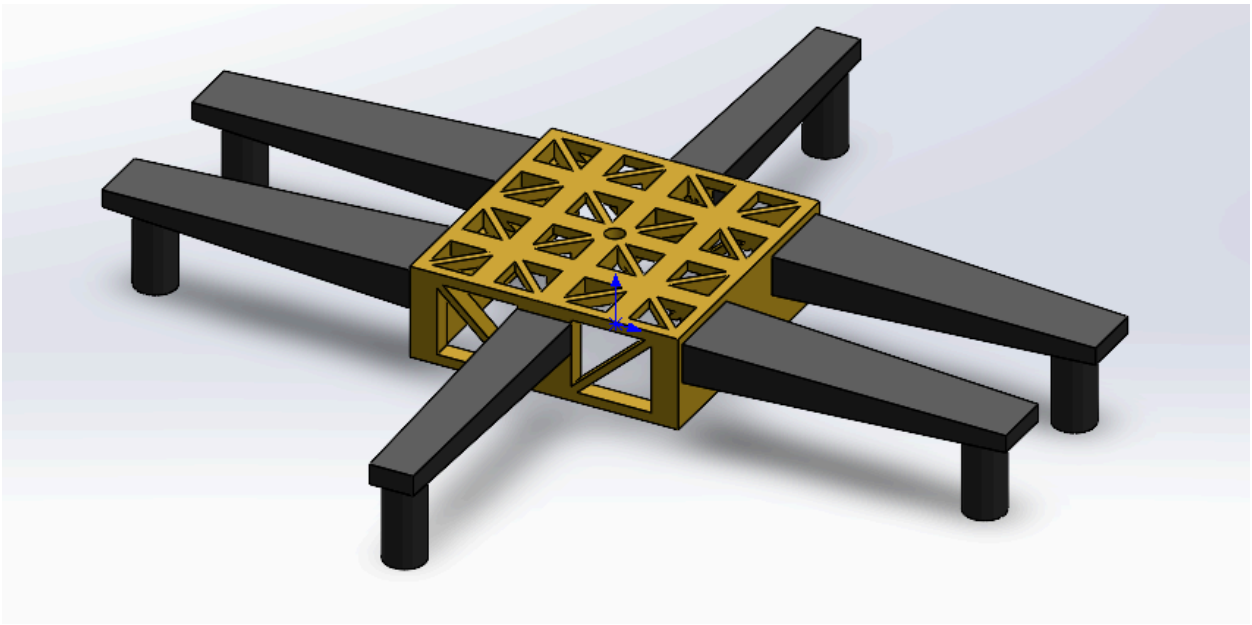
Go to the **Basic Assembly Tools** tab and select the first tool, **Insert Components**. A dialog box opens automatically on the left side in the **PropertyManager**. Another dialog box also opens, allowing you to select the components you want to insert into the assembly.



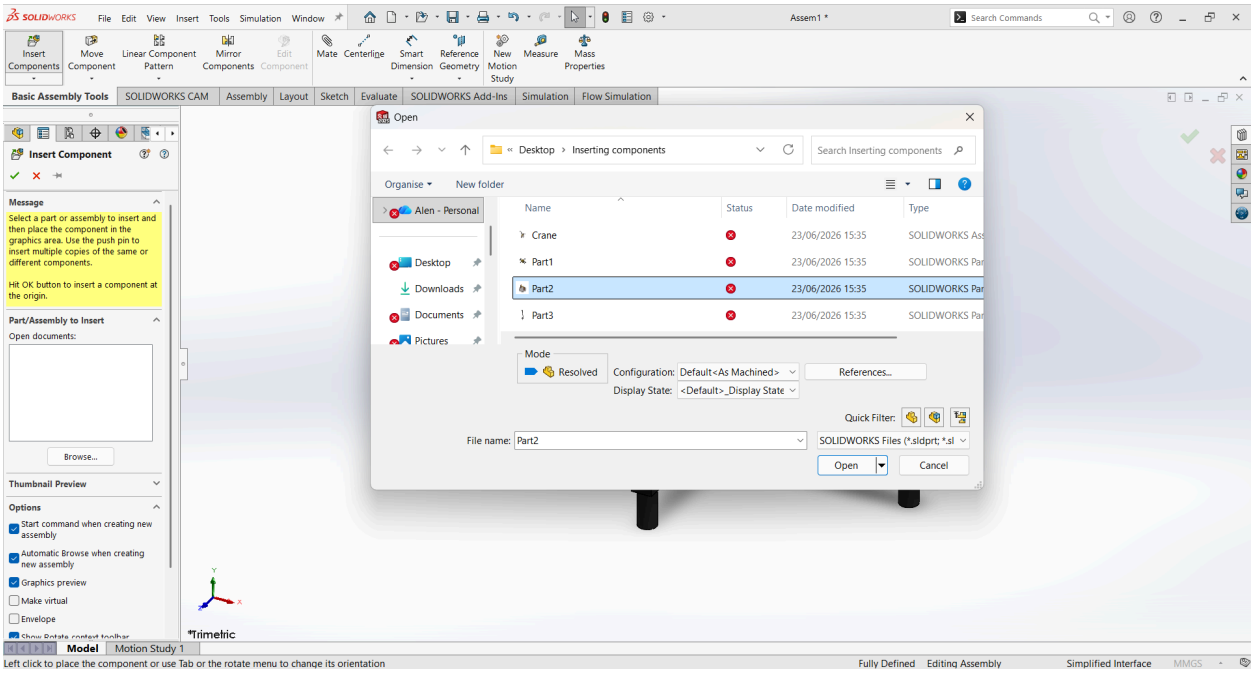
Find the component file on your computer. Select the component, then move your mouse into the SOLIDWORKS graphics area. You will see the component preview appear in the graphics area. As you move the mouse, the component moves with it.



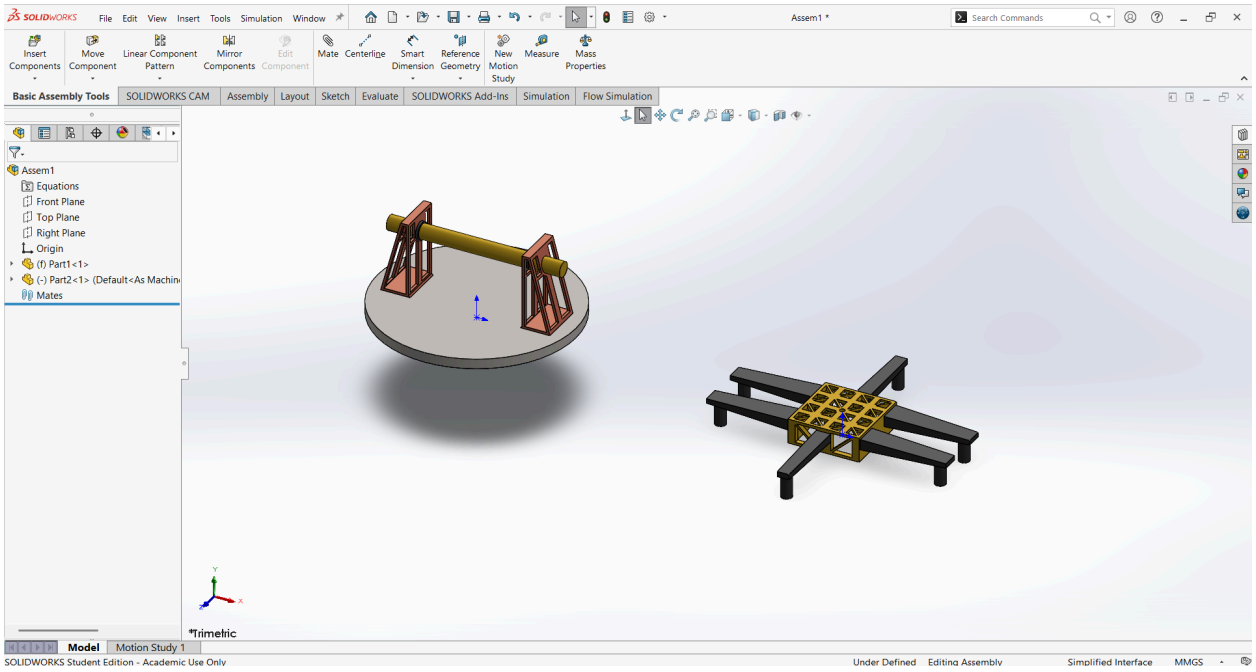
Move the mouse to the assembly origin. The component automatically snaps to the origin. Click there to place the component. Now the component origin and the assembly origin are coincident. The component is also fixed, which means you cannot move it freely in the assembly.



Let’s insert the second component. Go to the **Basic Assembly Tools** tab and select **Insert Components**. When the dialog box opens, select the second component.



The new component appears on the screen. Click anywhere in the graphics area to place the component.



2. Mates

What it does: Defines relationships between assembly components.

Common mates include:

- Coincident
- Concentric
- Parallel
- Perpendicular
- Distance
- Angle

How it works:

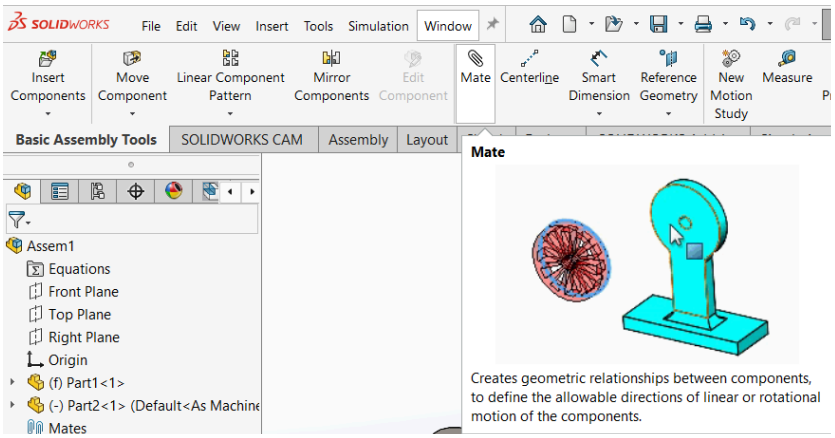
Now we will explain **Standard Mates**.

In an assembly, **mates** are used to define the relationship between components. They control how parts are positioned, aligned, and allowed to move relative to each other.

For example, mates can make two faces touch, align two holes, keep two parts parallel, or allow one component to rotate around another.

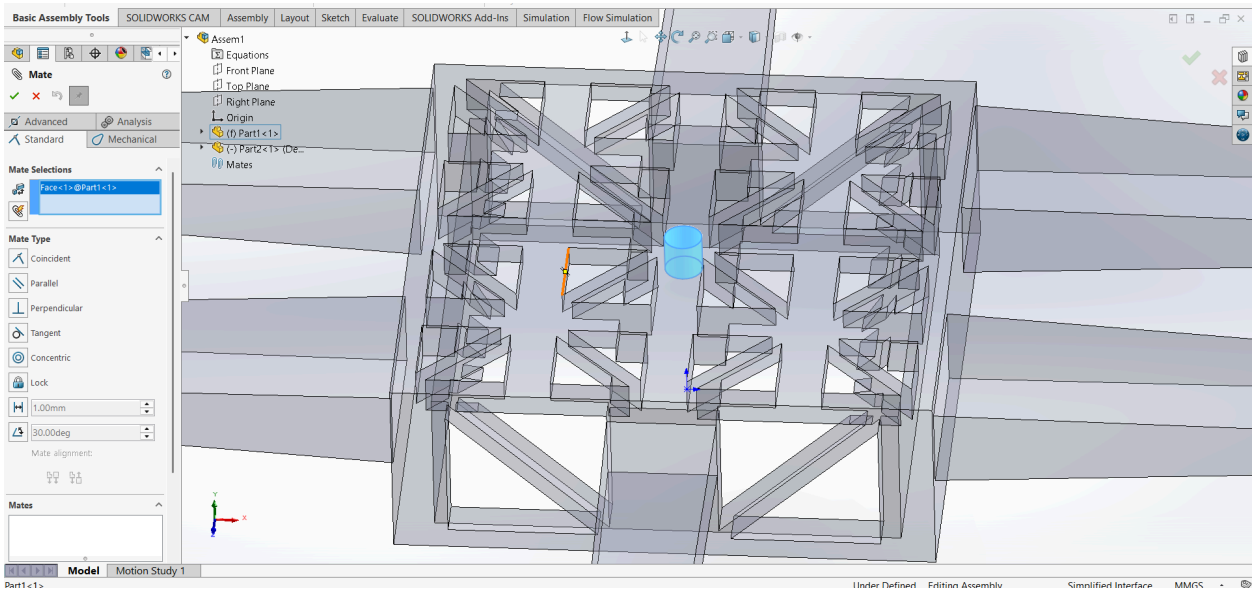
Mates are very important because they turn separate parts into a functional assembly that behaves more like a real mechanical system.

Go to the **Basic Assembly Tools** tab and select **Mate**.

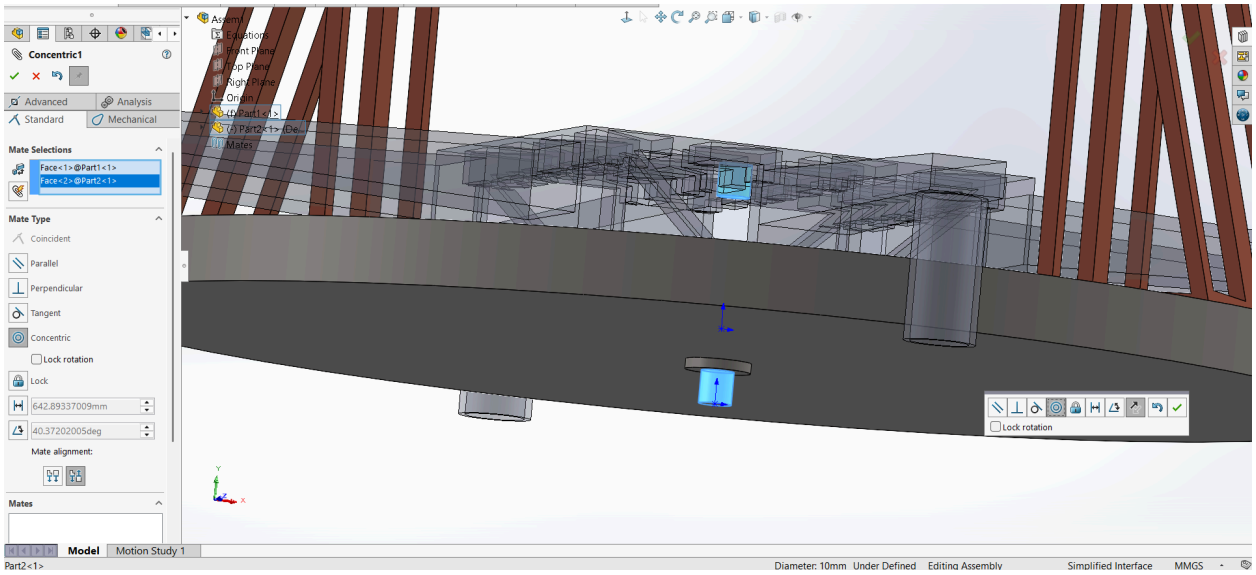


A new dialog box appears on the left side in the **PropertyManager**. Here, you can see the available **Standard Mates**, such as **Coincident**, **Parallel**, **Perpendicular**, **Tangent**, **Concentric**, **Distance**, and **Angle**. In the selection box, we need to select the faces, edges, or other geometry between which we want to define the mate relationship.

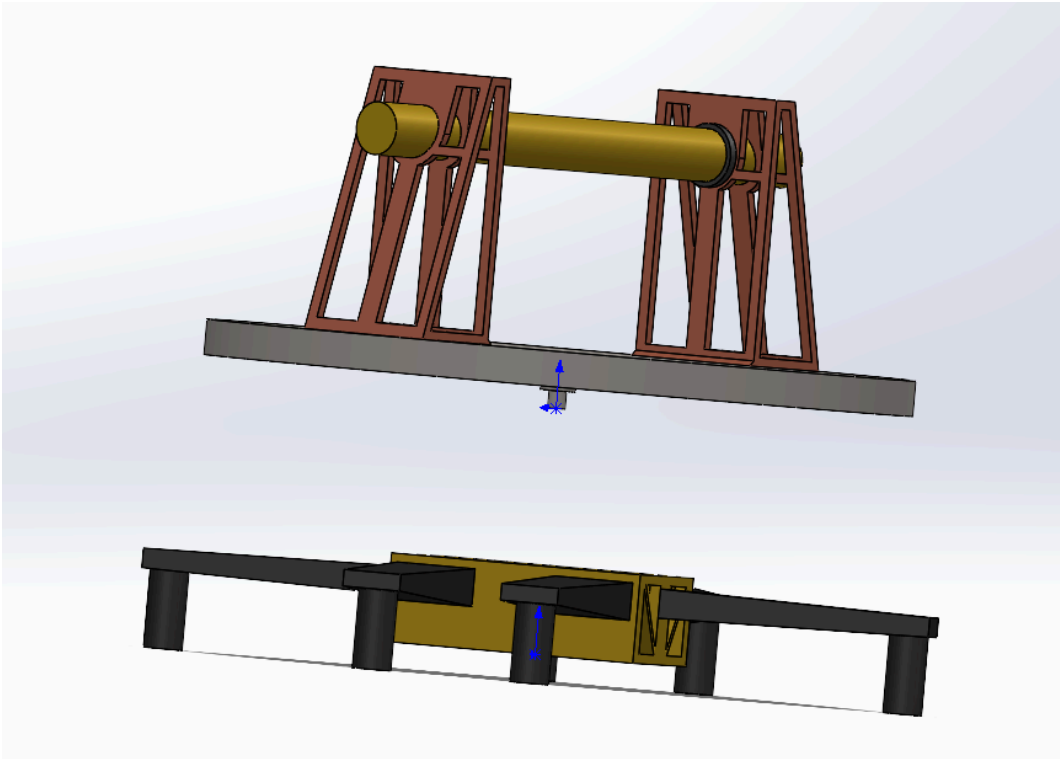
First, select the hole on **Component 1**.



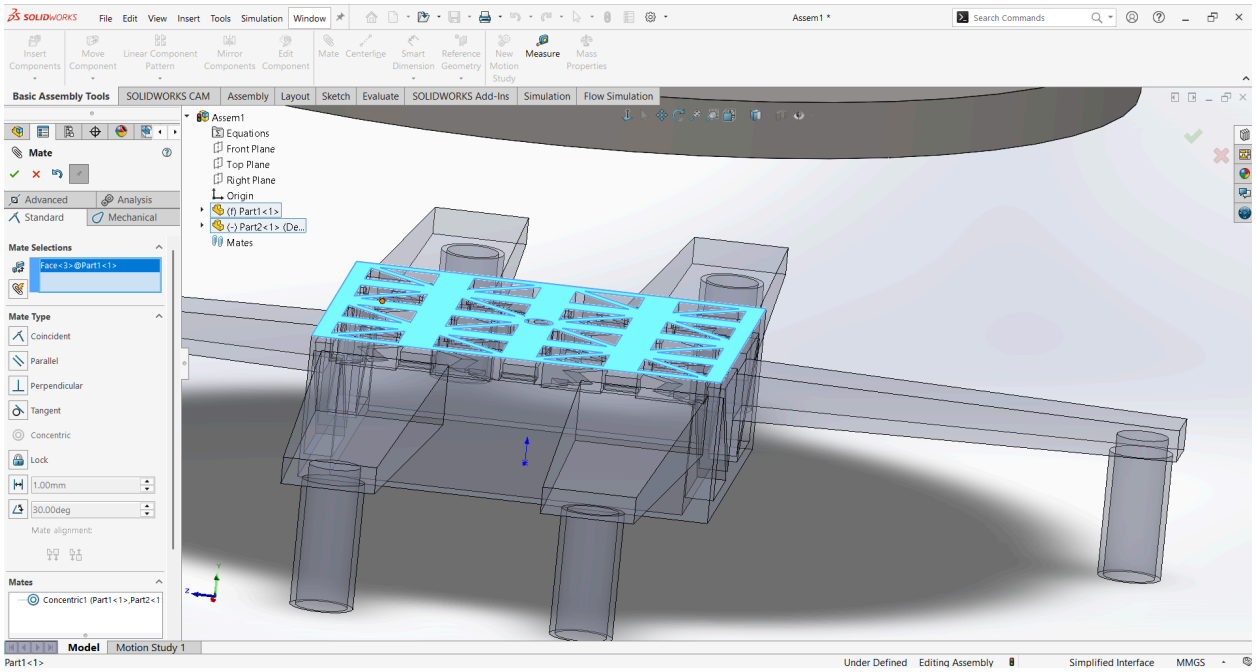
Then select the small pin at the bottom of **Component 2**. SOLIDWORKS automatically suggests the mate type **Concentric**. This is the connection type we want, because the pin should align with the hole. Click **OK**.



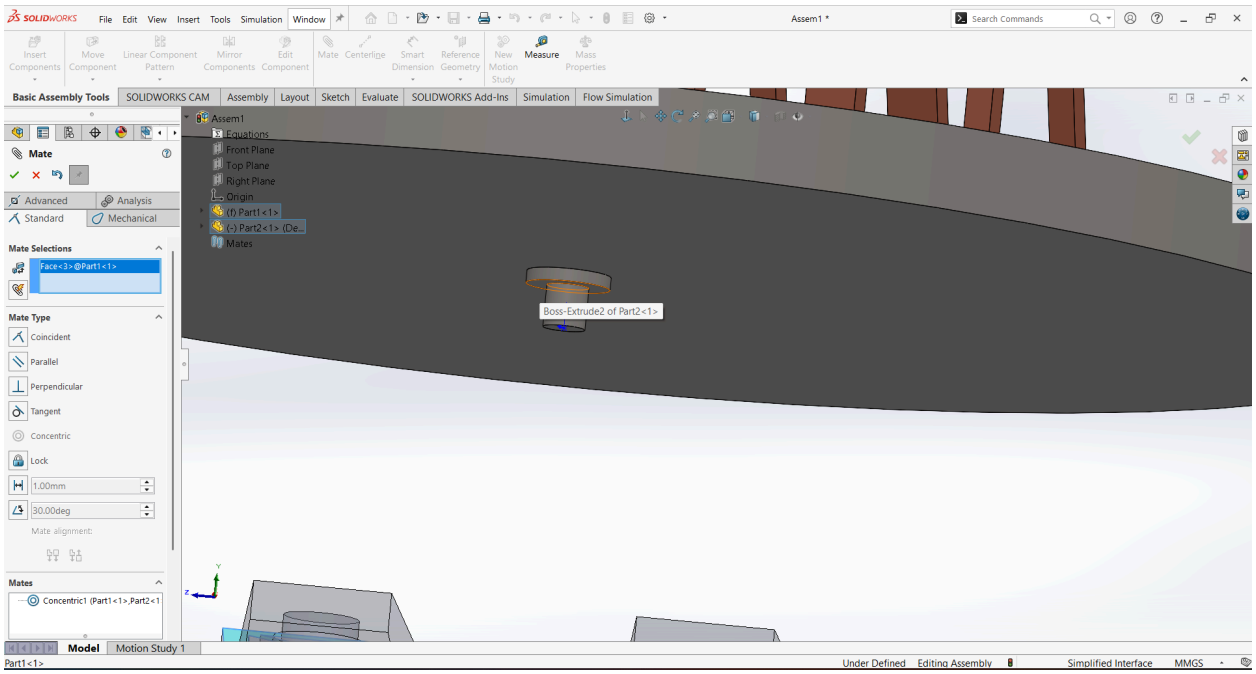
Now we have created a **Concentric** mate. Component 2 is aligned with the hole, but it can still move up and down along the shared axis.



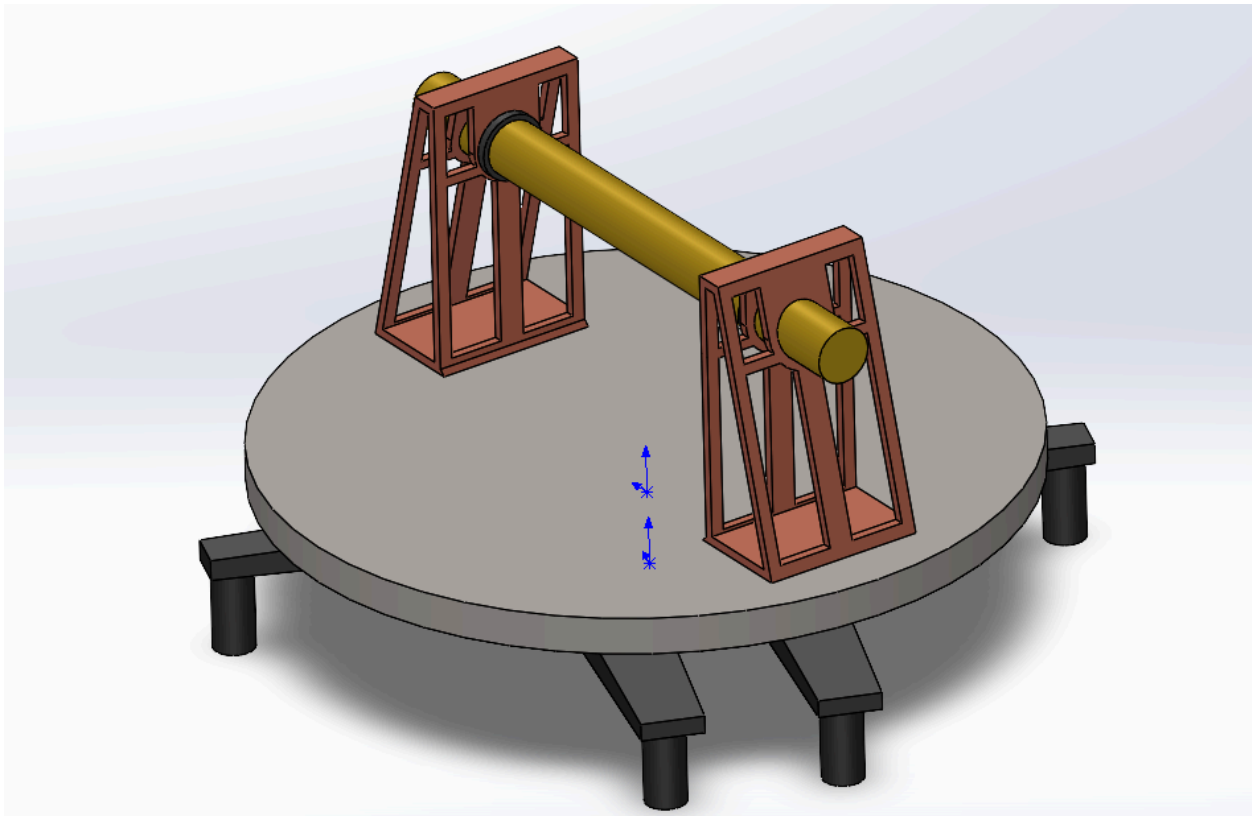
Now we need to prevent the up-and-down movement. Go to **Mate** again and select the upper face on **Component 1**.



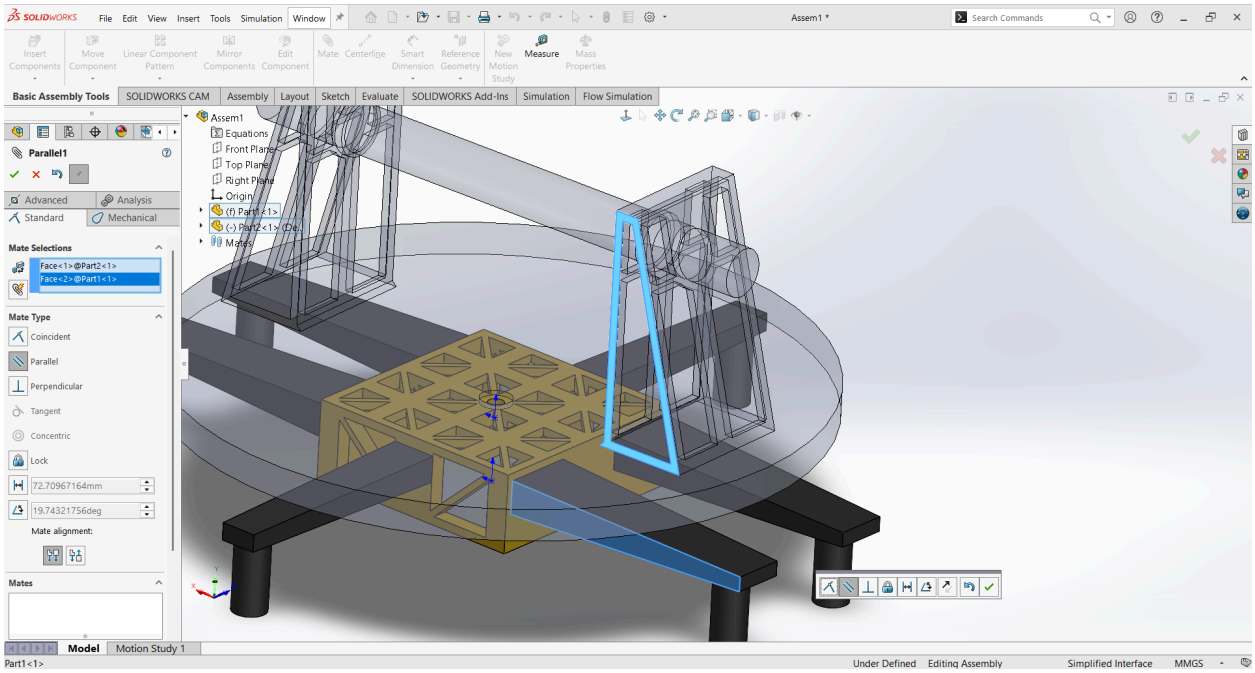
Now select the face on the smaller cylinder of **Component 2**. Click **OK**.



Now **Component 2** can only rotate around the shared axis.

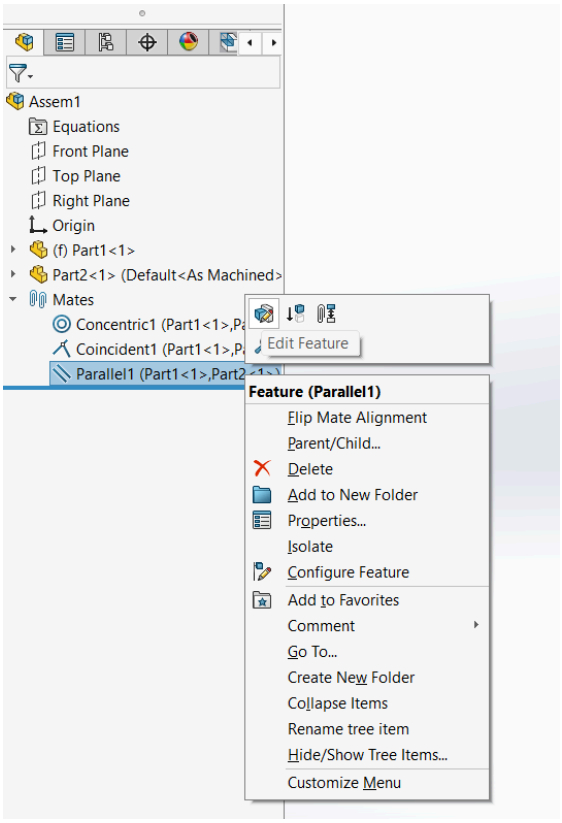


If we want to prevent rotation and fully restrict the movement of **Component 2**, we need to add another mate. Go to **Mate** and select the faces on **Component 1** and **Component 2**, as shown. Select the mate type **Parallel**. Click **OK**.

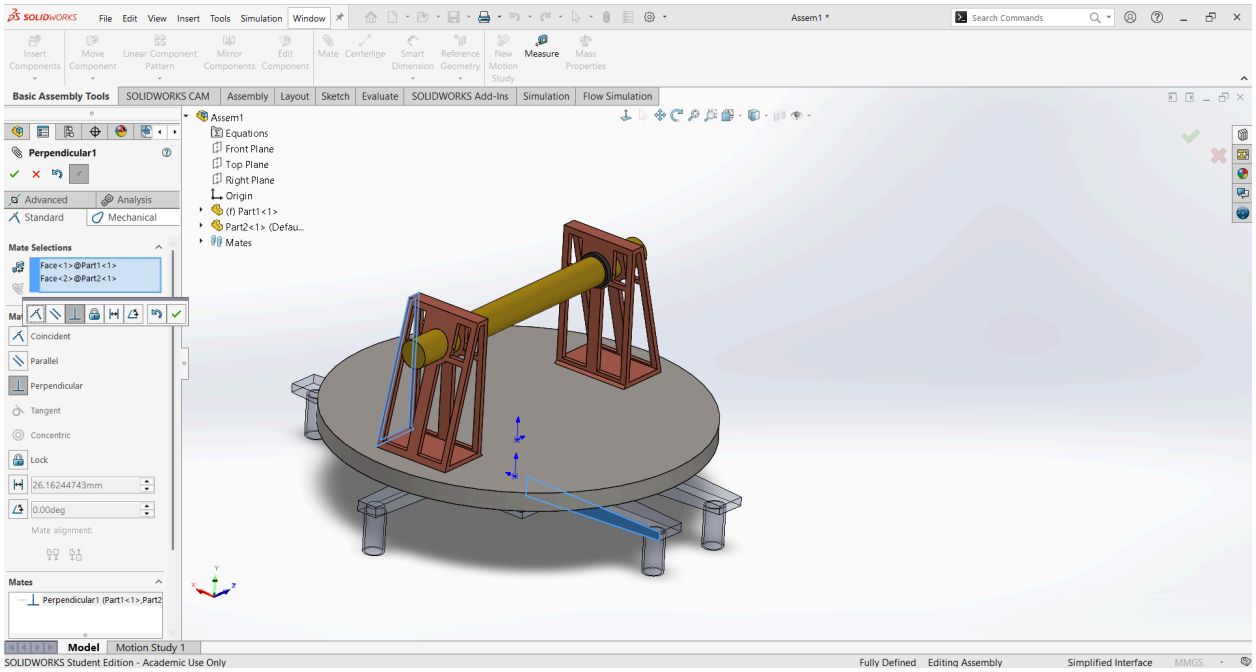


Now these two faces are parallel. This prevents **Component 2** from rotating, so the component is fully constrained.

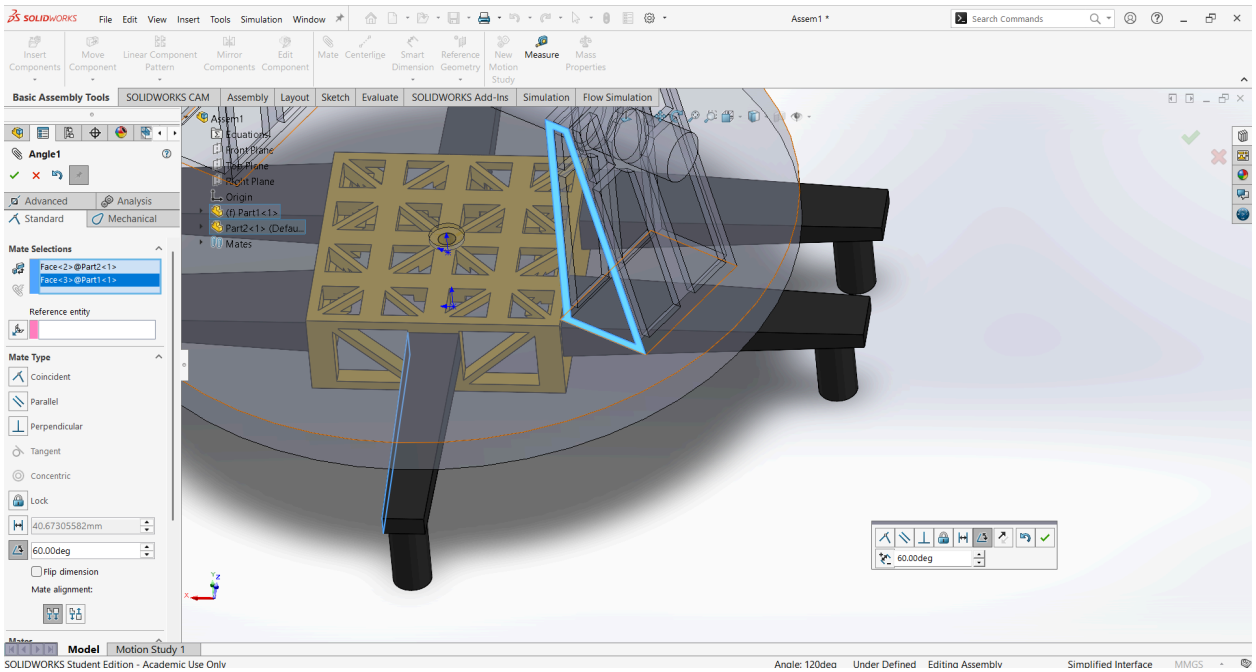
In the **FeatureManager Design Tree**, there is a folder called **Mates**. This folder contains all the mates we have created in the assembly. Go to the last mate, **Parallel**, right-click it, and select **Edit Feature**.



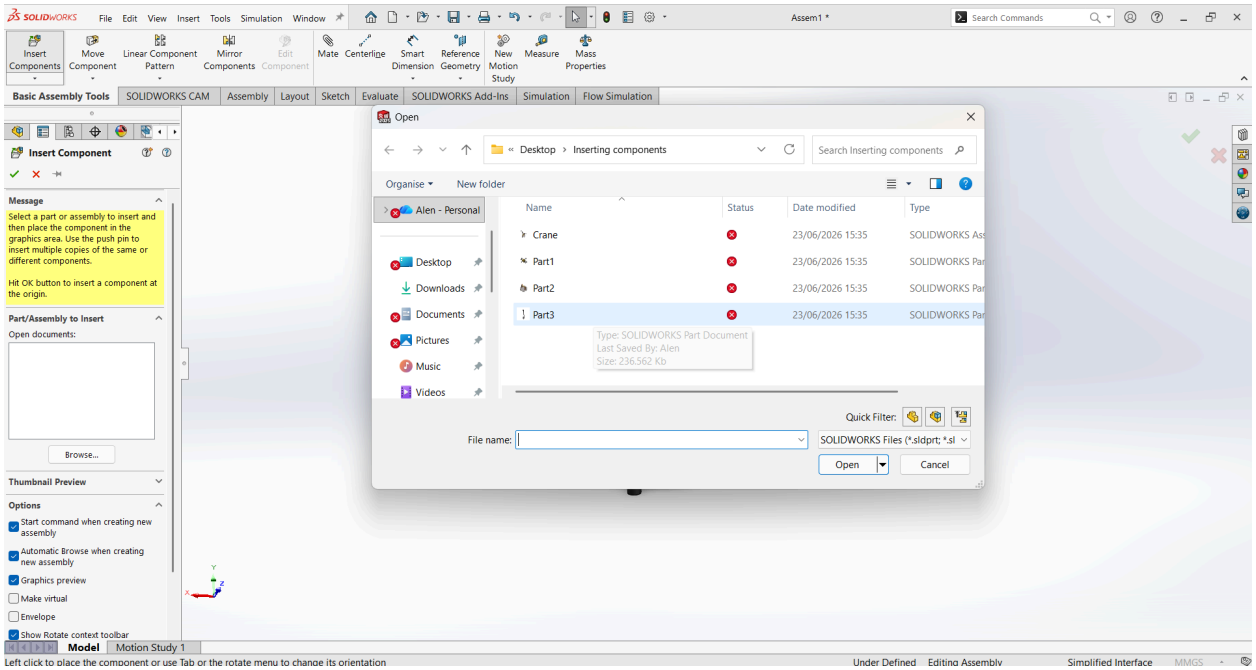
Now change the mate type. Select **Perpendicular**. Now these two faces have a **Perpendicular** relationship, which means they are positioned at a **90° angle** to each other.



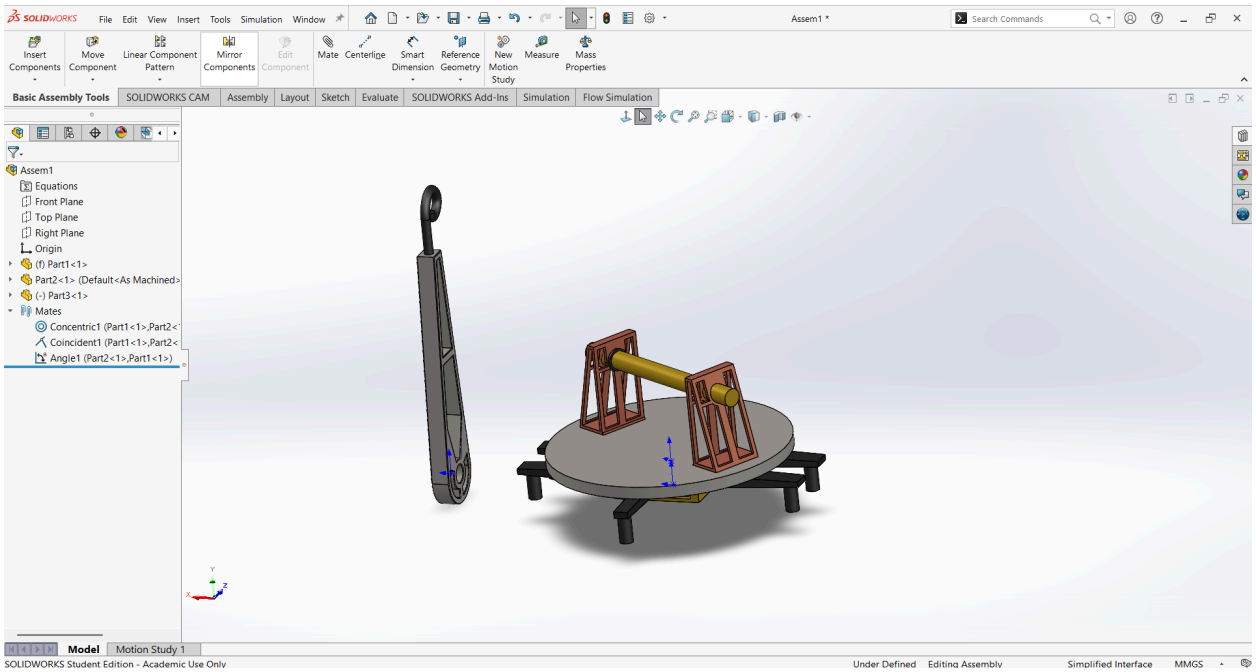
Now select **Angle** from the **Mate Type** options. Set the angle value to **60°**. Now there is a **60° angle** between these two faces.



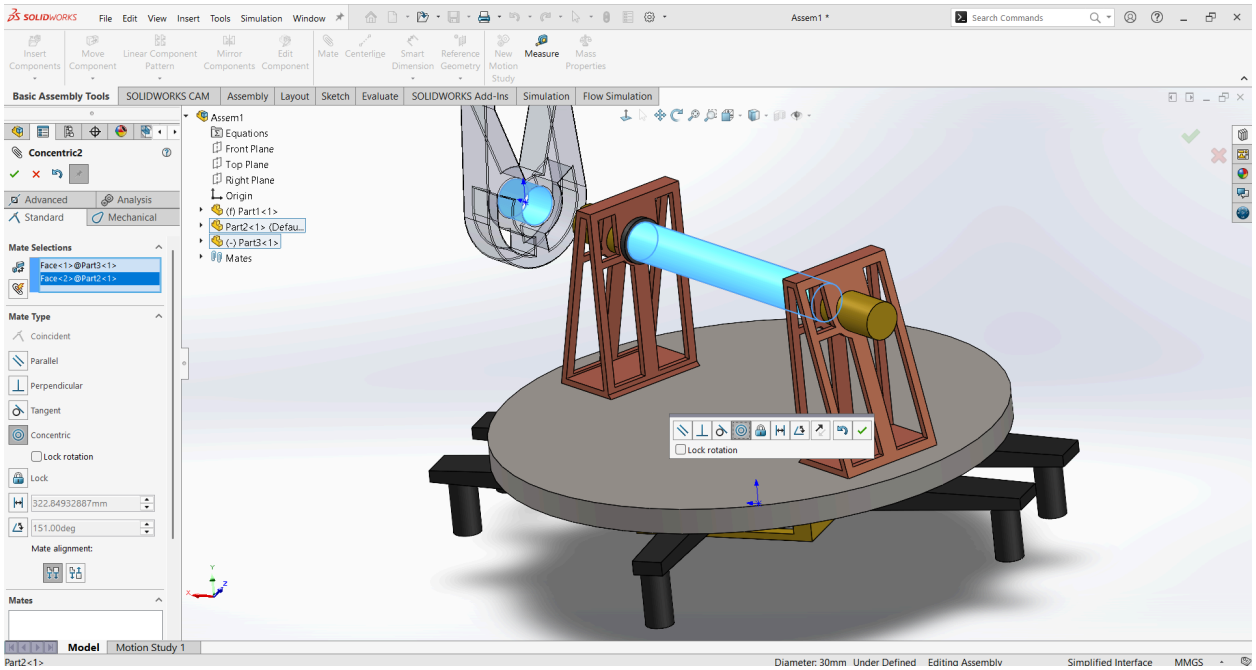
Click **OK**. Now let's insert the third component. Go to **Insert Components** and select **Component 3**.



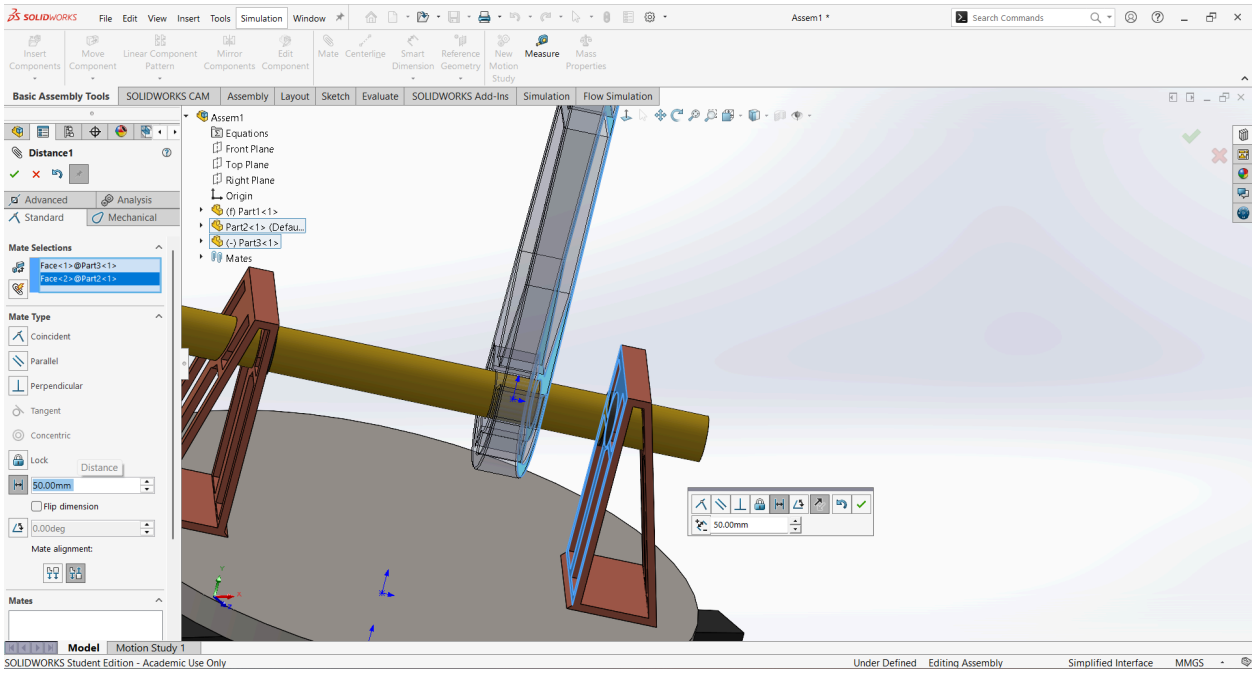
Place it anywhere in the graphics area.



Now we need to connect **Component 3**. Go to **Mate**. Select the shaft on **Component 2**, then select the hole on **Component 3**. SOLIDWORKS automatically suggests a **Concentric** mate. Click **OK**.

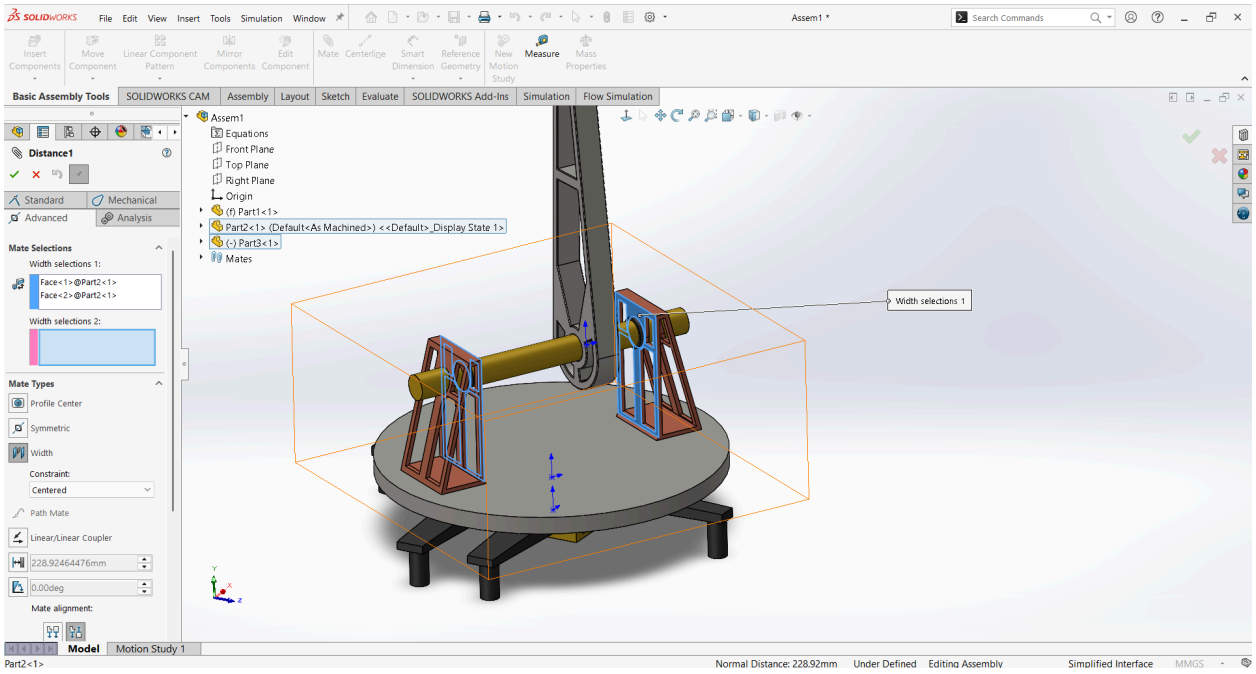


Now go to **Mate** again. Select the face on **Component 2**, then select the face on **Component 3**. Select the **Distance** mate type and set the distance to **50 mm**. Now there is a **50 mm** distance between these two faces.

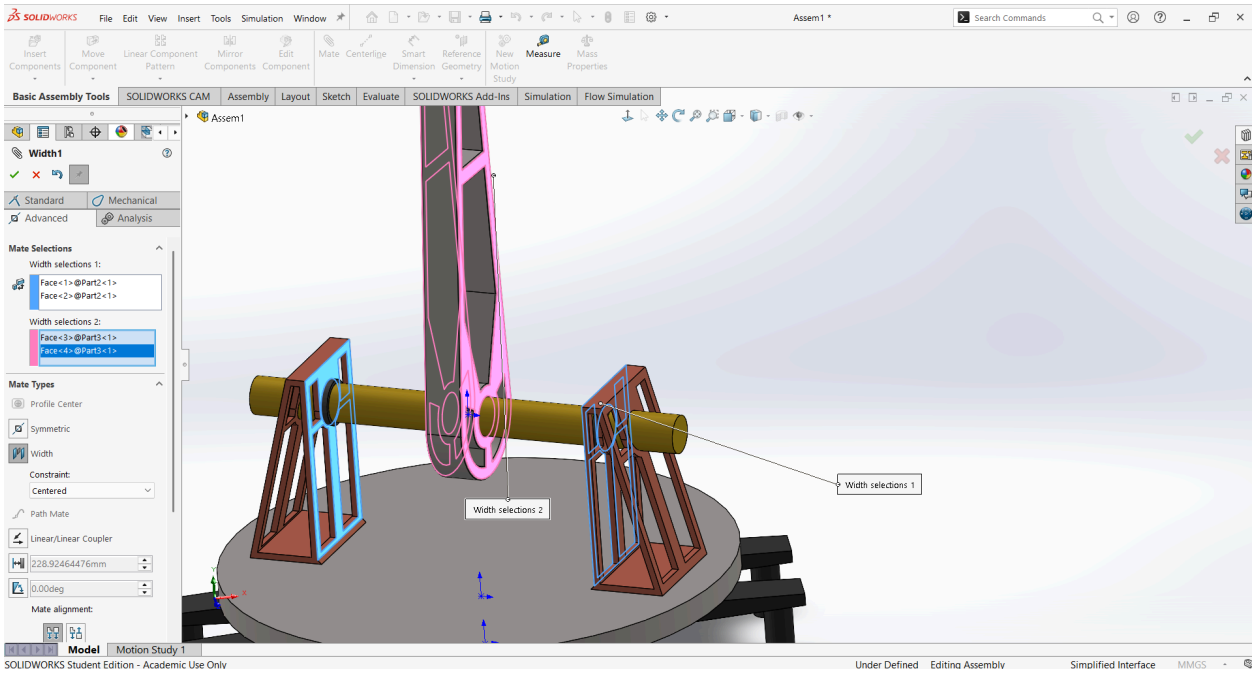


Delete previous Angle mate.

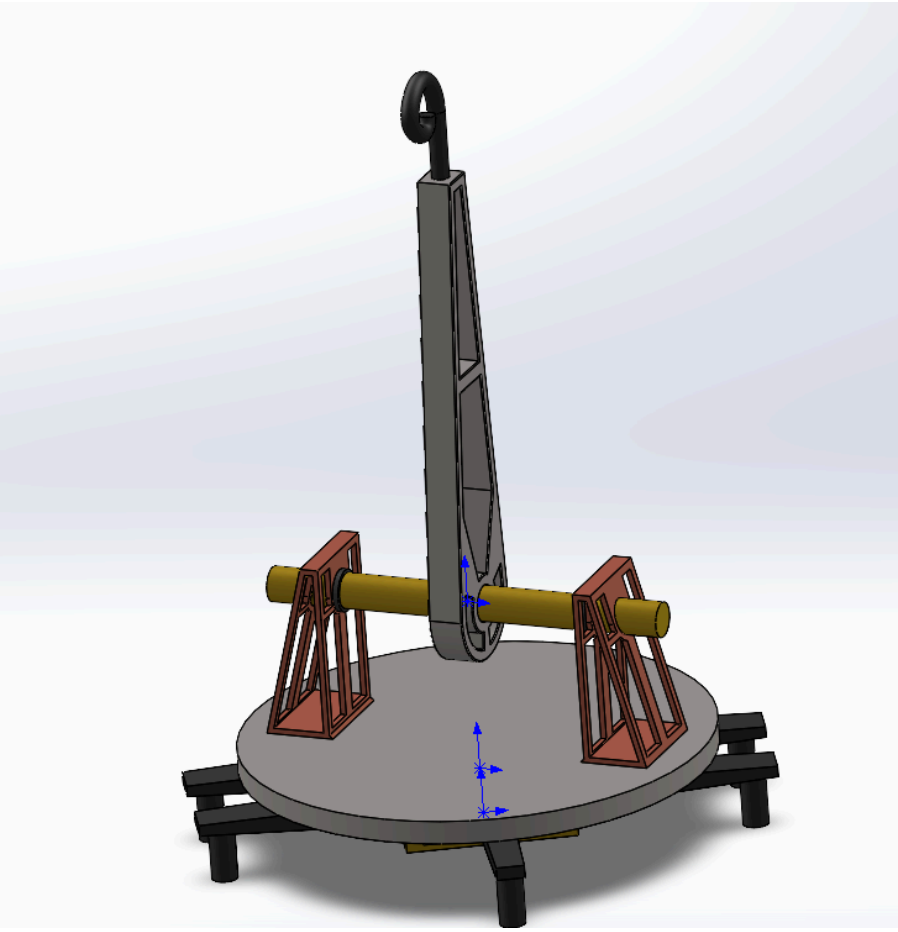
Now let's show one **Advanced Mate**. Go to **Advanced Mates** and select **Width**. In the first selection box, select the two faces of **Component 2**, shown in blue.



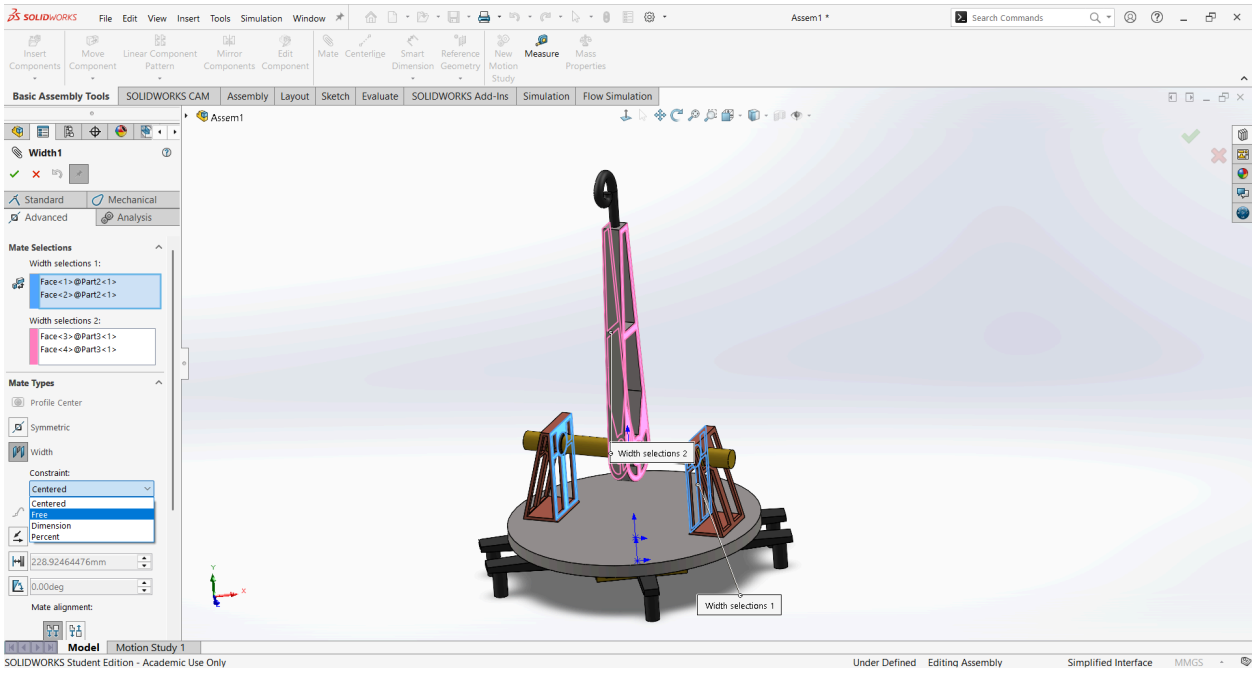
In the second selection box, select the two faces of the other component, shown in pink.



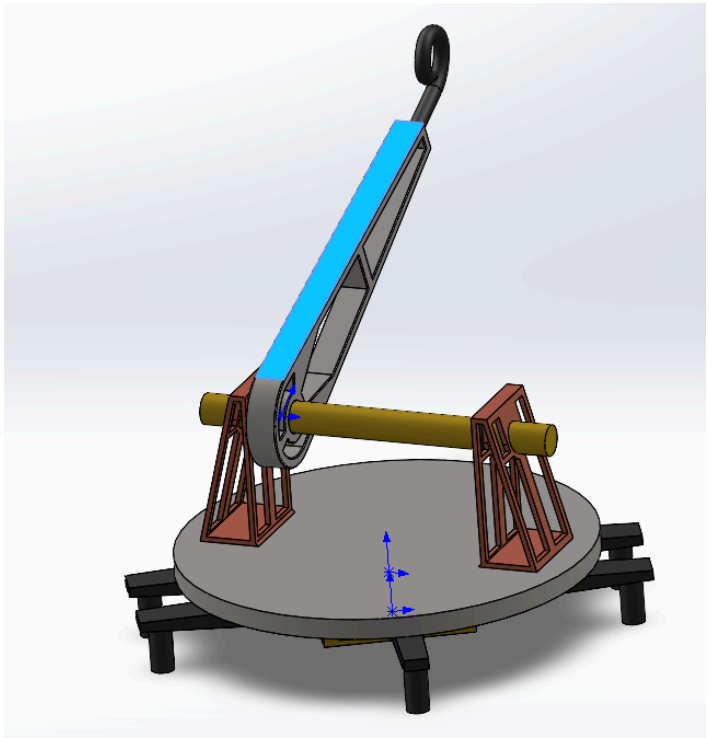
Click **OK**. Now the part is positioned exactly in the middle of the shaft.



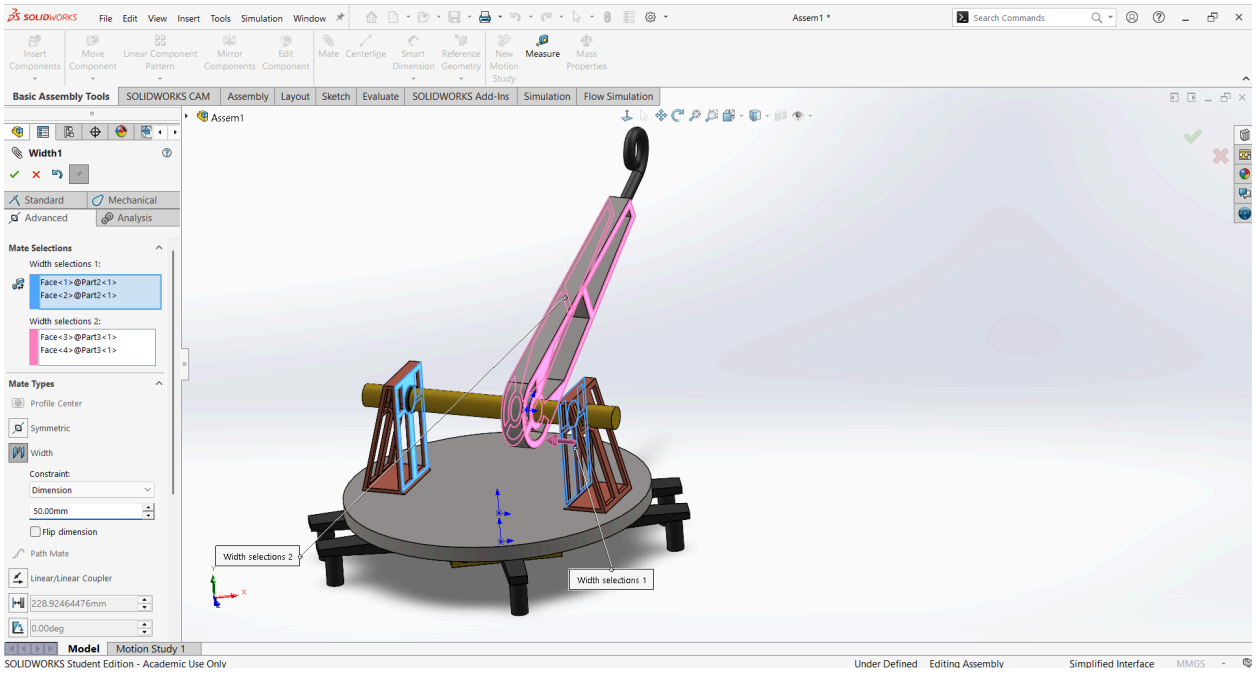
Now go back and edit the **Width** mate we created. Change the **Constraint Type**. Right now, the constraint type is set to **Centered**. Click on it and change it to **Free**.



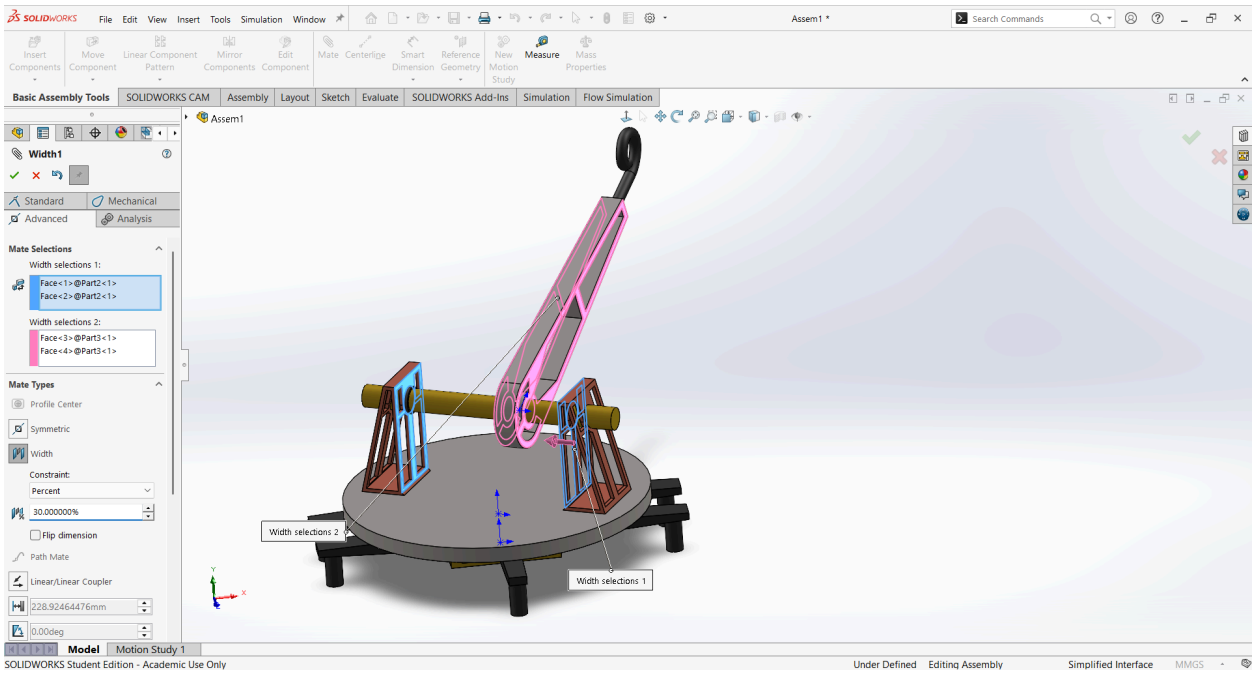
Now we can move **Component 3** freely along the shaft of **Component 2**.



Let’s change the **Constraint Type** again. This time, set it to **Dimension** and enter a value of **50 mm**. Now **Component 2** and **Component 3** are positioned with a **50 mm** distance between them.



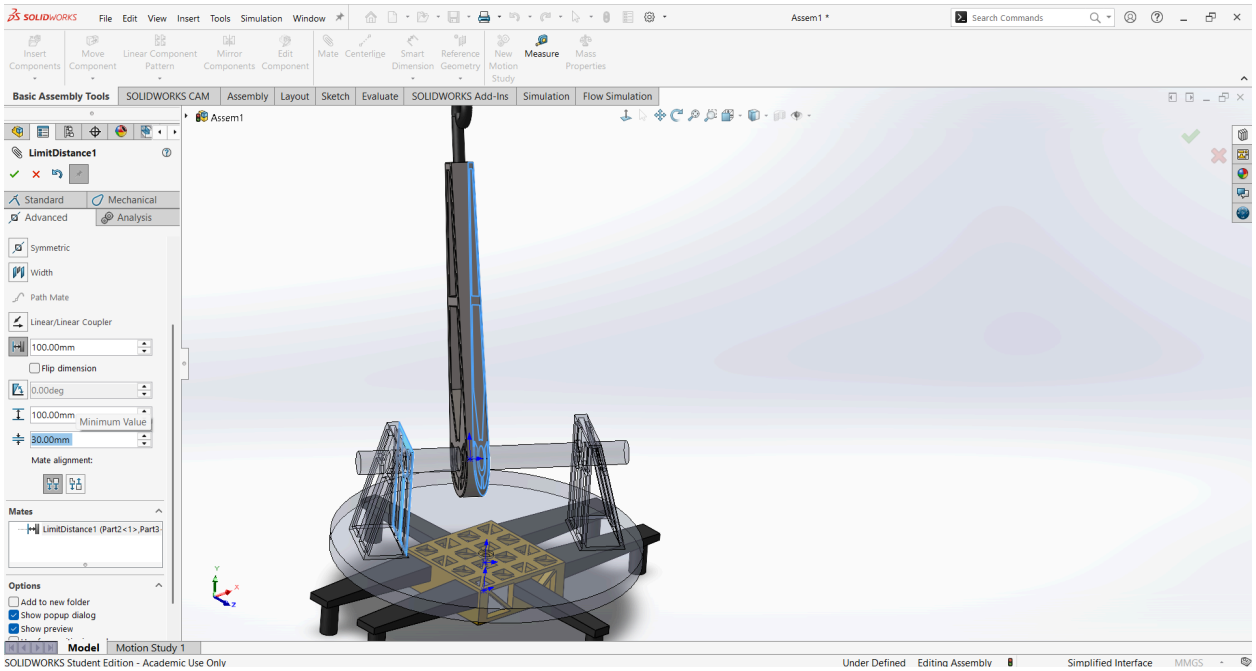
Now change the **Constraint Type** to **Percent**. With this option, you can define the position of **Component 3** along the shaft as a percentage. Set the value to **30%**.



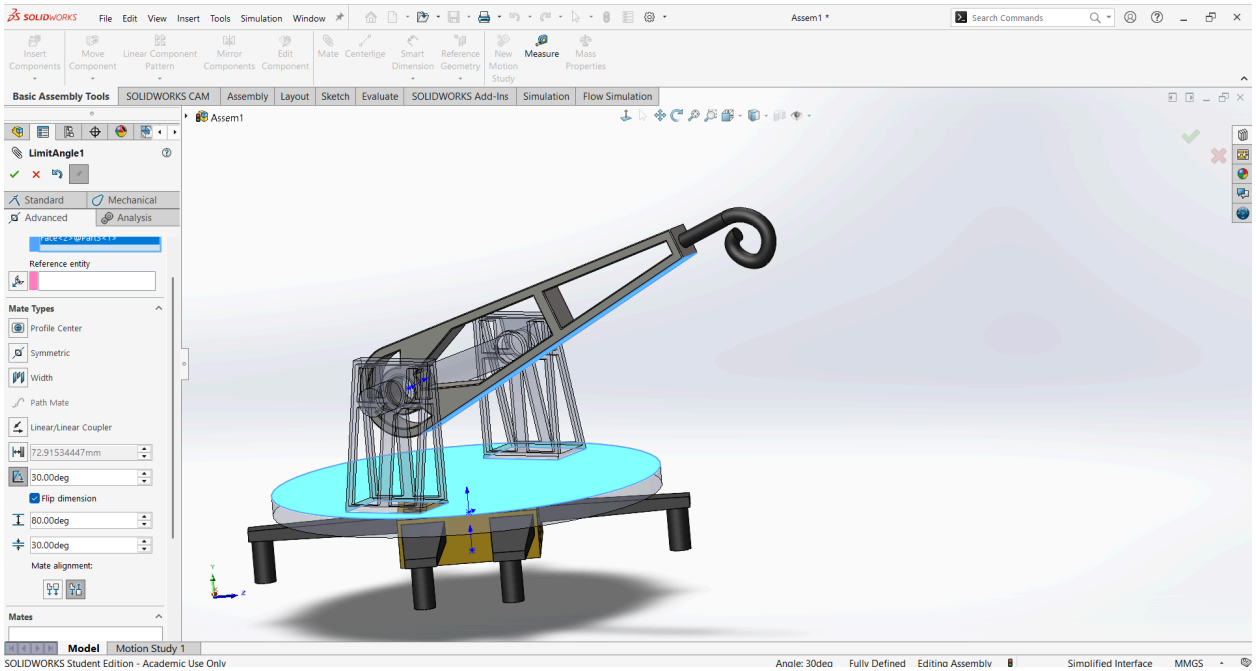
Click **OK**.

Now let's show the second **Advanced Mate** type: **Distance**. With this mate, we can define a minimum and maximum distance, creating a movement range. Select the two faces on

Component 2 and **Component 3**. Set the **Maximum Distance** to **100 mm** and the **Minimum Distance** to **30 mm**. Now **Component 3** can move only within this distance range.



We can do the same thing with an **Angle** mate. Select the faces on **Component 2** and **Component 3**, as shown. Define the **Minimum Angle** as **30°** and the **Maximum Angle** as **80°**. Now the component can rotate only within this angle range. If needed, change **Mate Alignment** or use **Flip Dimension** to position the components correctly.



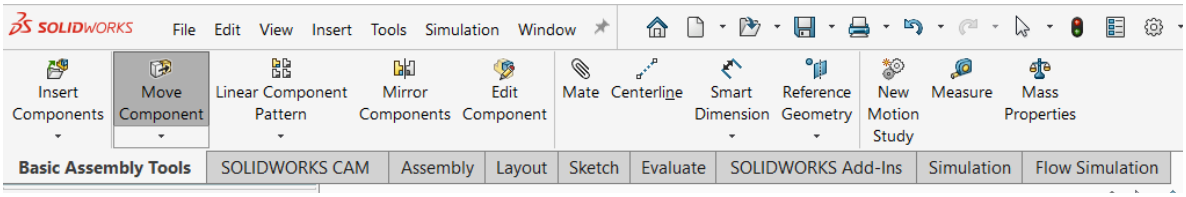
3. Move Component

What it does: Moves components inside an assembly.

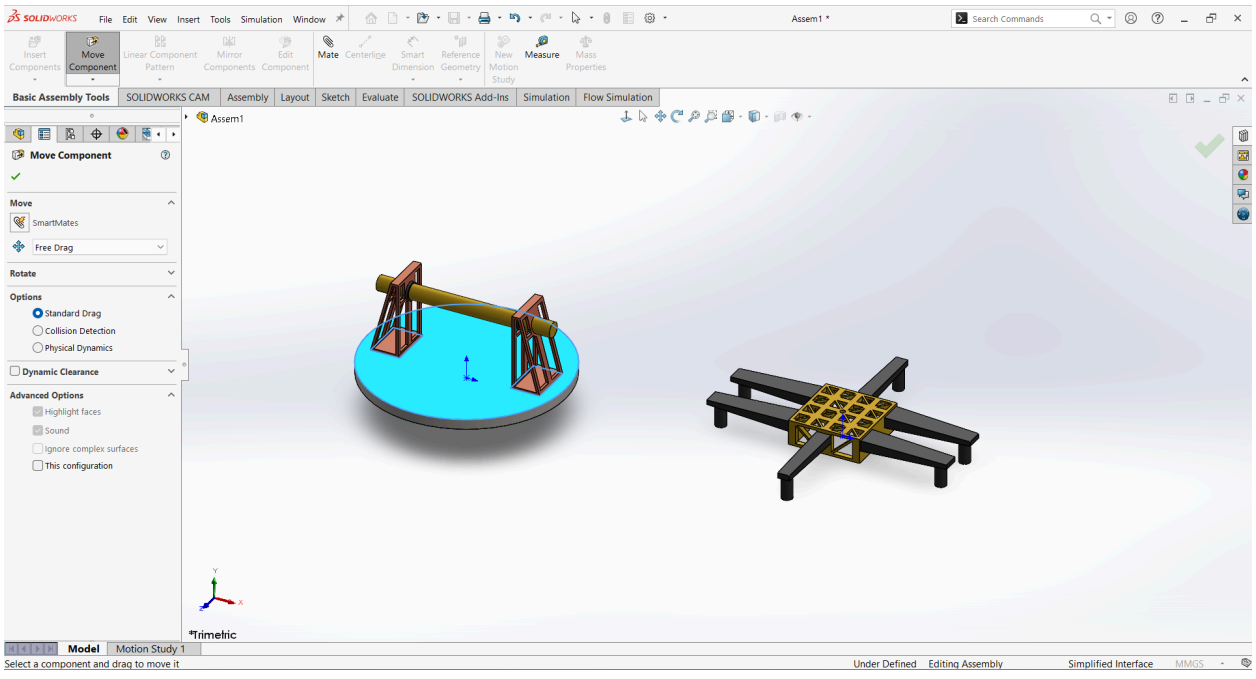
Use it for: Positioning parts and testing remaining degrees of freedom.

How it works:

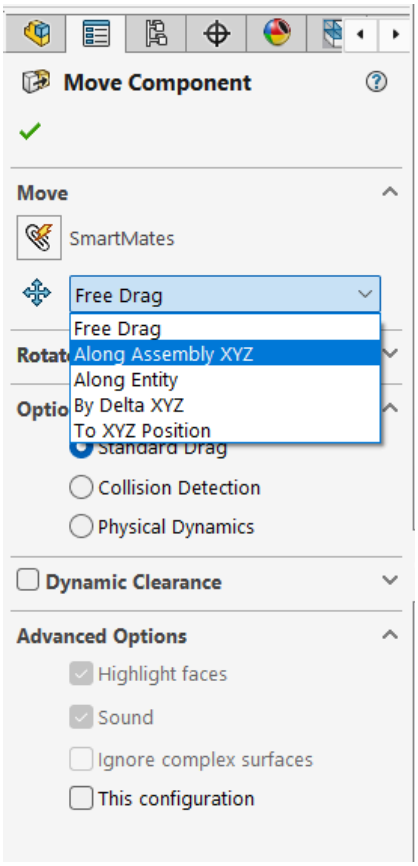
The second component is not fixed, so we can move it freely. Go to the **Basic Assembly Tools** tab and select **Move Component**.



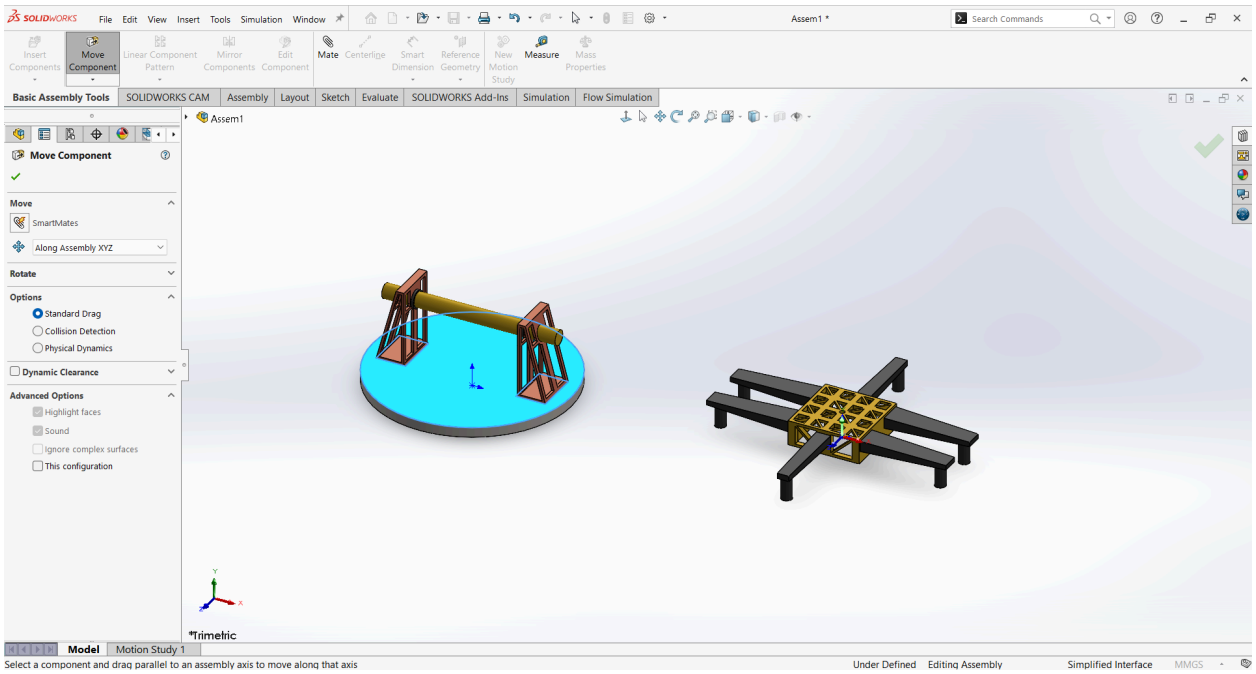
Property Manager appears. By default, the **Move** option is set to **Free Drag**. Now, if you click and drag the component with the mouse, you can move it freely in the graphics area.



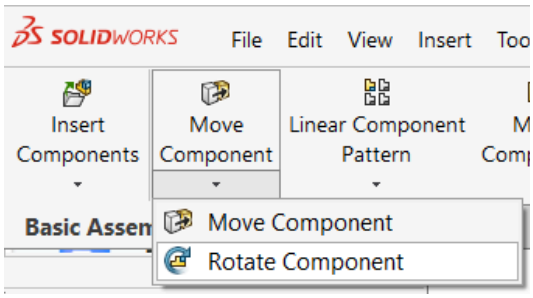
There are also a few other options under **Move**. Select **Along Assembly XYZ**.



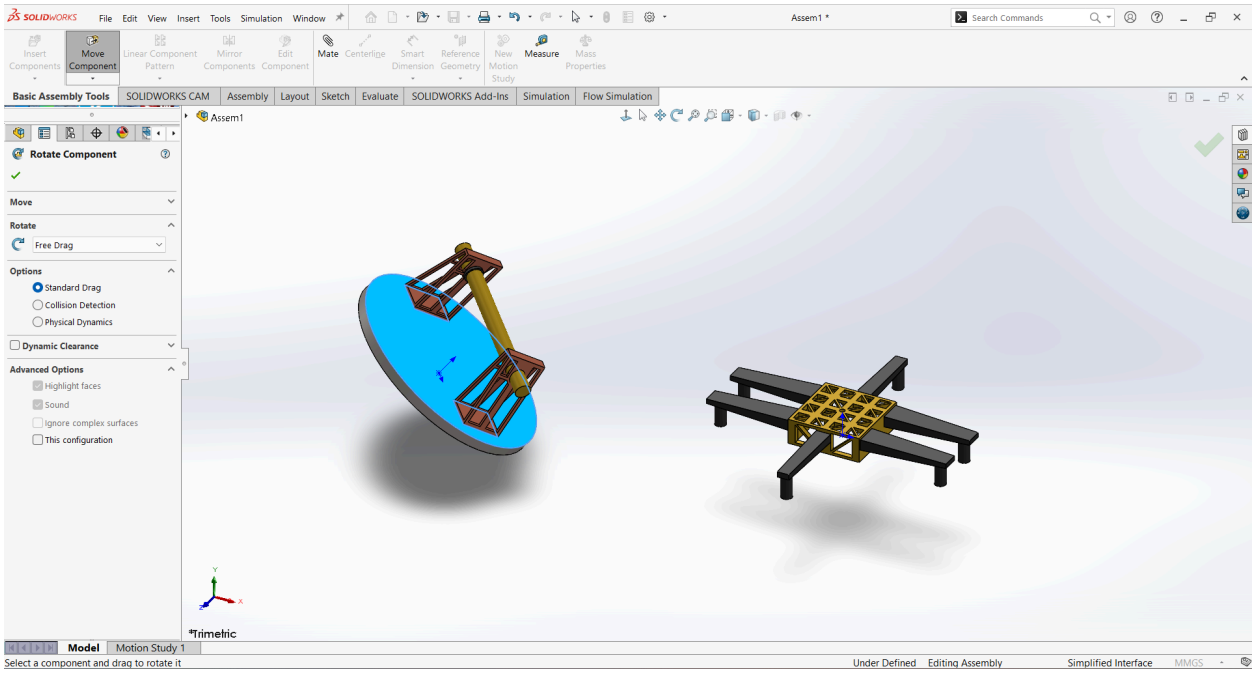
Now you can move the component along the assembly **X**, **Y**, and **Z** directions.



Go to the Move **Component**.



Click the small arrow below the tool, then select **Rotate Component**. Now you can rotate the component using the mouse.



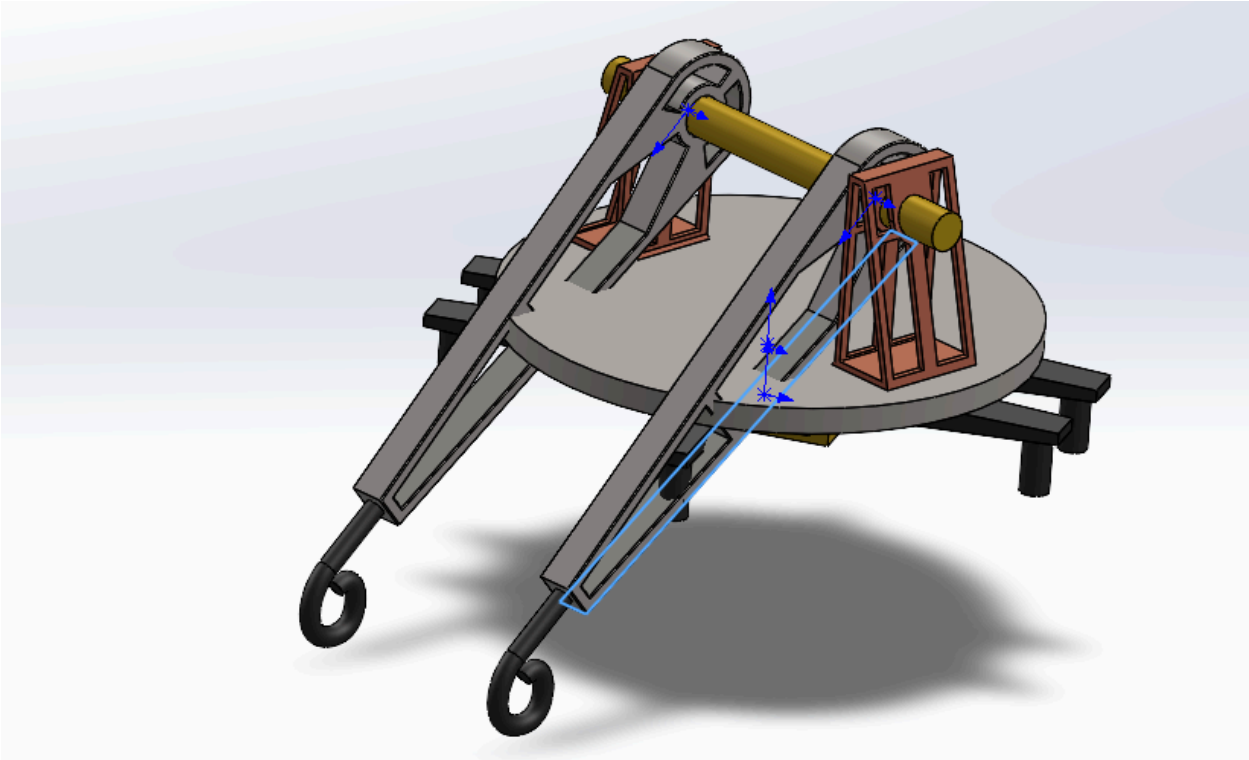
4. Collision Detection

What it does: Checks whether assembly components physically overlap.

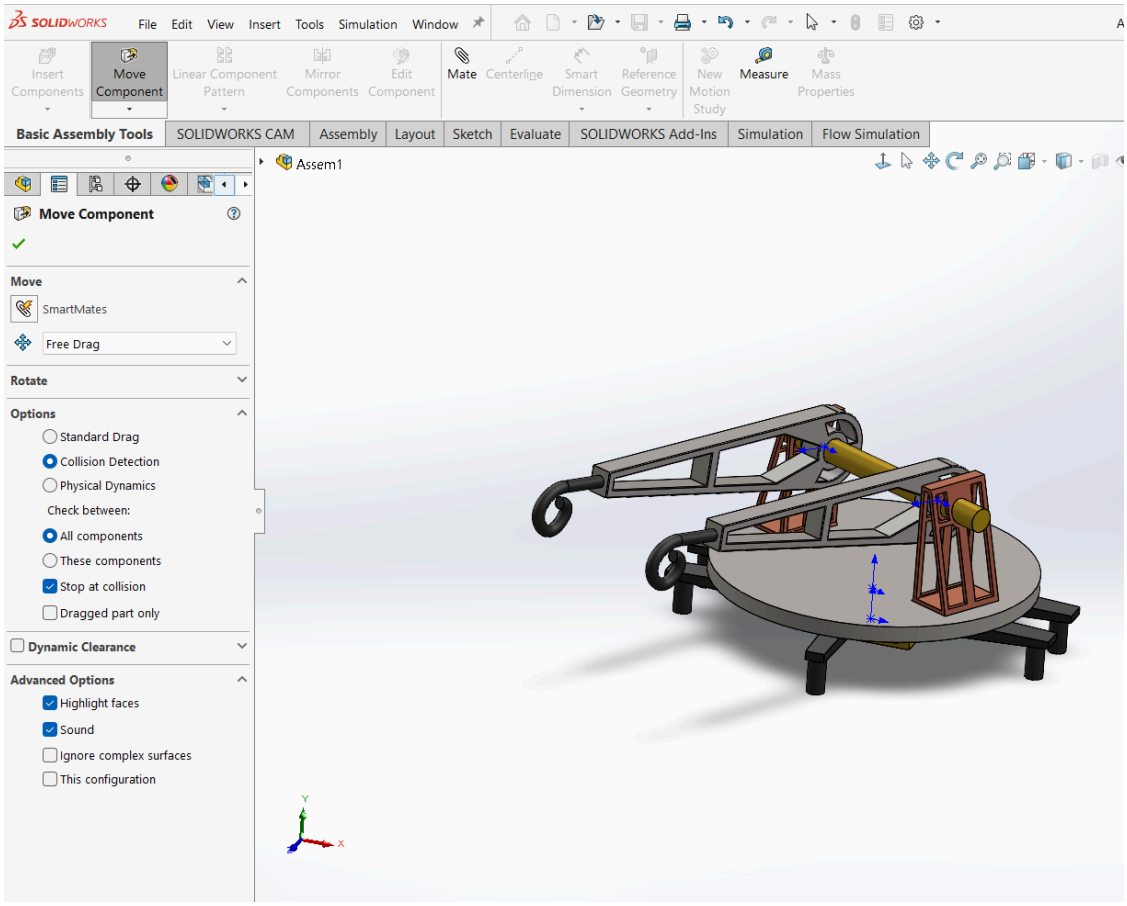
Use it for: Finding design problems before manufacturing or assembly.

How it works:

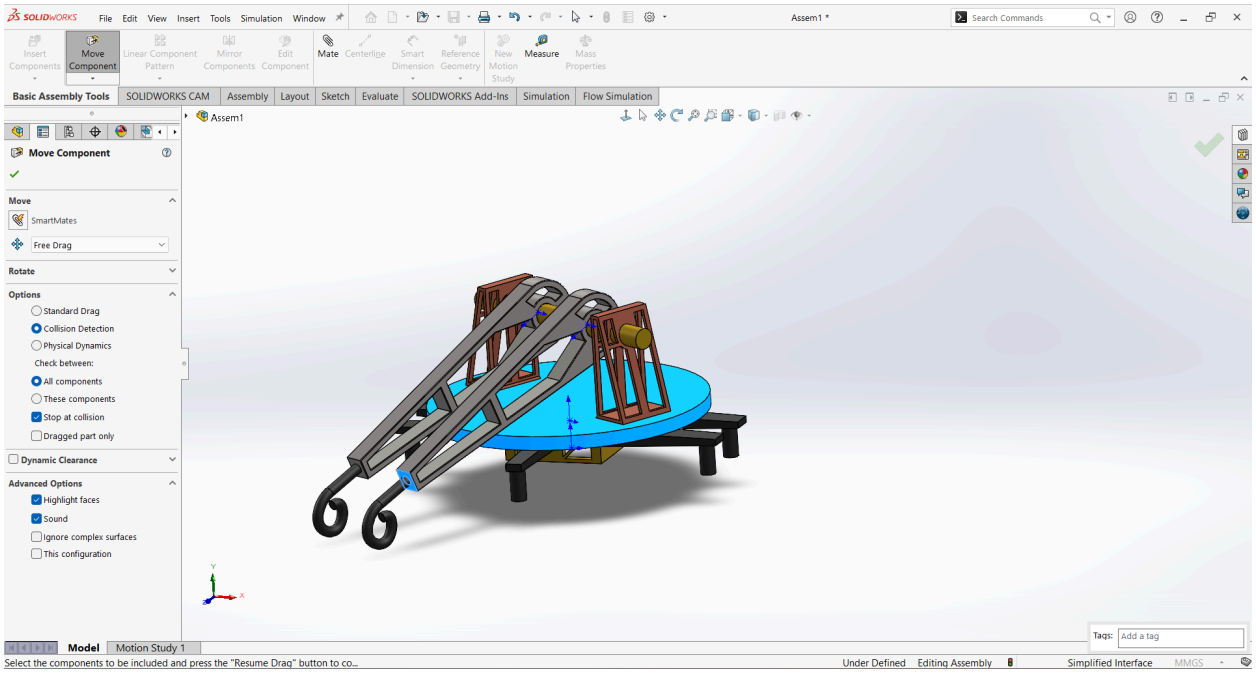
Now you can see that when you move **Component 3**, it goes through the material of **Component 1**. We want to be able to detect the collision.



Go to the **Move Component** tool. Under **Options**, select **Collision Detection**, and also make sure **Stop at collision** is checked.



Now if you rotate component 3 you can see that it will hit component 1.



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Inside, you'll learn the complete SOLIDWORKS workflow step by step:

✓ **Sketching** – Build accurate and fully defined sketches
✓ **Part Design** – Create simple to advanced 3D models
✓ **Assembly Design** – Combine parts, apply mates, and build complete assemblies
✓ **Engineering Drawings** – Turn your 3D models into professional technical drawings

Whether you're completely new to SOLIDWORKS or want to take your existing skills to the next level, the course gives you a **structured path from the fundamentals to advanced modeling.**

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