AUTODESK UNIVERSITY

CP501657

# Design with just 12 commands: The Essential Modeling Commands in Autodesk Fusion 360

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## Learning Objectives

- Learn how to use 12 essential Fusion 360 design commands
- Learn how to put those 12 commands together to create real-world designs
- Gain confidence in your Fusion abilities, without being intimidated by the wide range of Fusion commands
- Explore sketching, solid modeling, and assembly modeling with a minimum number of interactions

## Description

The Fusion 360 team often hears, from beginning users, comments such as: “Fusion is so hard to learn. I don’t know where to start. There are so many commands and workspaces, it’s too intimidating”. We are here to bust that myth. In this class, we will demonstrate, live, creating a real-world design (not a trivial demo design) using just 12 commands in the Design workspace.

Join your expert Fusion instructors, Phil and Jeff for this interactive demo/training class aimed at newer Fusion users who just want to learn the basics. Once you’ve mastered these 12 commands, you will have skills to create moderately complex designs, and, with that confidence, you can expand your knowledge by learning the rest at your own pace. You’ll leave the class with a good foundation built on basic skills along with keys to further explorations in areas that matter to you.

## Speaker(s)

Phil is a Principal Software Engineer for Autodesk on the Fusion 360 team primarily testing design workflows and crashes. Before Autodesk he used Inventor starting at R1. He enjoys working with the online Fusion community and teaching Fusion at a local community college.

Jeff is a Director of Software Development at Autodesk. “I’ve been with Fusion since the very beginning (before parametrics, before XRef…). My focus is on general modeling/sketching. Before that, I was a developer and architect on Inventor, also before R1. So, I’m a long-time CAD guy. When not working with Fusion or its customers, I like to run, cycle, hike, and read science fiction.”

## The purpose of this class

Learning a new CAD tool is a lot like learning a language. It can be intimidating and confusing, but once you know some concepts in areas of grammar and structure, and some simple vocabulary of commonly used words, you'll soon be composing sentences and communicating your thoughts. The very act of composition can begin to drive further integration with the language, to create ideas and combine them. The combining of ideas is more powerful than the singular application of them. This progression is also a way to learn a new CAD application. This class introduces the grammar, basic words, and sentence structure of Fusion 360.

Starting with the basic commands that get the most work done, you can then expand your skillset to combine these commands and others into more complex expressions of your design intent. This class intends to speed this process by providing a firm foundation by which later growth is supported and accelerated.

### How much is enough

Is 12 or 15 commands enough? Including too much would be diluting the impact by adding too many "essential" commands. The choice of what qualifies as essential is debatable by experts, with valid reasoning, from a myriad of backgrounds in design and manufacturing. They may disagree somewhat, but there is an undeniable, and short, list of requirements for what could be called a "CAD Foundation", when applied to learning any CAD product. We're trying to provide that here for Fusion 360. We strived for 12, but in the end expanding to 15 was required.

The design selected was chosen to represent the cornerstone blocks of that CAD foundation - a robot arm with a gripper. It has components, motion, sketches, and a realistic parts list. These basic elements are the things you need to get right in every design, simple or otherwise.

### Assembly structure

The design chosen for this class requires components. Components have motion, part numbers, can be used in exploded diagrams, and so forth. Components are parts. This class shows how they are made, how they are assembled together, and how this assembly structure represents a simple, yet realistic design.

*THE ROBOT GRIPPER ARM ASSEMBLY*

### Sketch essentials

The shapes in the chosen design are simple and easy to create with a handful of the most common sketch commands: lines, arcs, circles, and a few more. Working at this granular level of design, creating the elemental particles of your design, you'll start to see why more complex sketch commands exist.

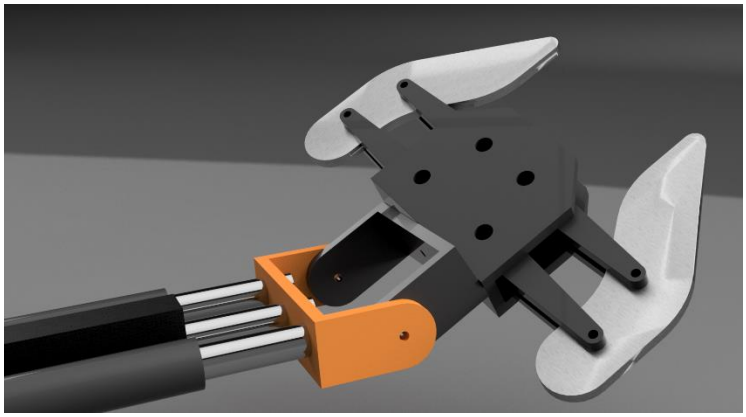
For example, expanding on the analogy of particles and extending it to molecules, you can draw a slot with two circles and two lines (particles), or you can use the slot command, which is just an automated combination of lines and arcs, but is slightly more complicated. In the case of slots, the trade off between complexity and productivity comes when you start to ask "How could this be easier?" Starting with basics, and expanding as opportunity presents is what this class suggests.

### Modeling core commands

Similar to the thinking about sketch commands, there are just 4 commands representing 3D geometry creation used for this design. Extrude, hole, fillet, and offset workplane. If you haven't heard the 80/20 rule, this is it in a nutshell: A fraction of the available tools do the majority of the work.

But each of these workflows, once you know them, will lead you to ask how the more complex commands could be utilized for greater effect, and perhaps increase efficiency. Without the basics, the complex commands are not seen in the right context, and can be a fancy way to "model yourself into a corner". Yes, they are more efficient, but with great power comes great responsibility. You can't skip over the responsibility part, but you're here, so that's a good start.

In the sections below, you'll find a step by step guide to designing this assembly. Along with diagrams and instructions, there will be explanations about expanding the skills shown into more complex workflows and commands.



*THE GRIPPER ASSEMBLY*

## The demonstration

What follows is a step-by-step guide based on the construction of the sample design. The live video will necessarily vary from this guide in a few ways. First, this paper skips no steps. Live, it was necessary to condense the demonstration both for reasons of time, and to avoid needless redundancy. Here you'll find every step to create the model yourself.

### The model

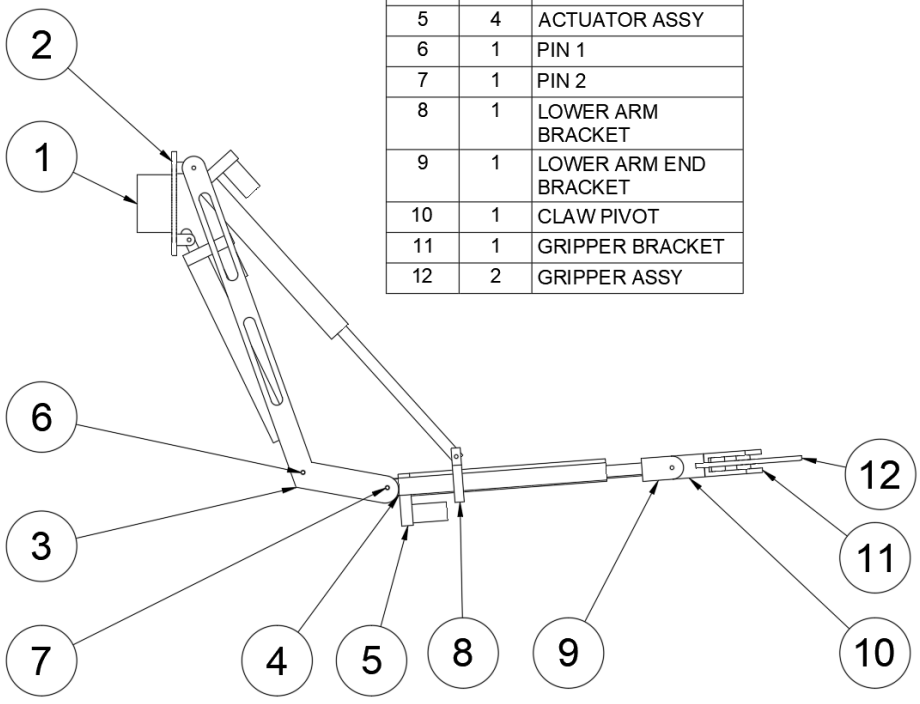
Starting at the same place as in the live class, the model will start with the gear box, and modeling will proceed out along the "arm" to the end of the gripping hand.

All images and diagrams are taken from the reference model. This model is available for download in the Autodesk Gallery using this link.

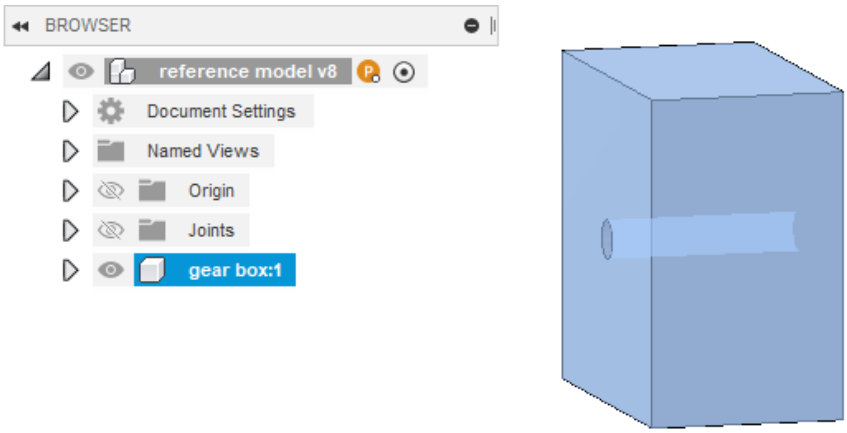
In the tutorial instructions that follow, commands are listed by name in *italics*. Paths to commands are shown by (menu name) > *(command name).* Example: Create > *Create Sketch*

At the end of each section is a list of commands used, with each newly introduced command highlighted. Also you'll find some observations about each stage of design development.

| PARTS LIST | | |
|---|---|---|
| ITEM | QTY | PART NUMBER |
| 1 | 1 | GEAR BOX |
| 2 | 1 | MOUNTING PLATE |
| 3 | 2 | UPPER ARM |
| 4 | 2 | LOWER ARM ASSY |
| 5 | 4 | ACTUATOR ASSY |
| 6 | 1 | PIN 1 |
| 7 | 1 | PIN 2 |
| 8 | 1 | LOWER ARM BRACKET |
| 9 | 1 | LOWER ARM END BRACKET |
| 10 | 1 | CLAW PIVOT |
| 11 | 1 | GRIPPER BRACKET |
| 12 | 2 | GRIPPER ASSY |

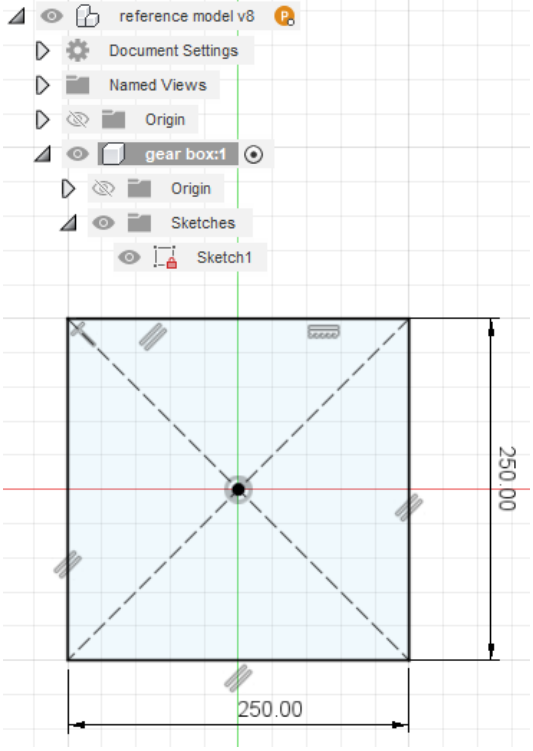


## Step 1 – Gear box component



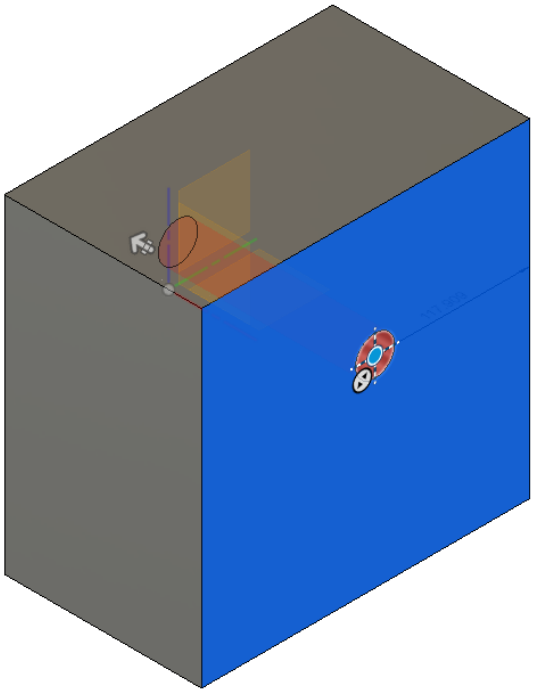

*THE FINISHED GEAR BOX COMPONENT (SELECTED)*

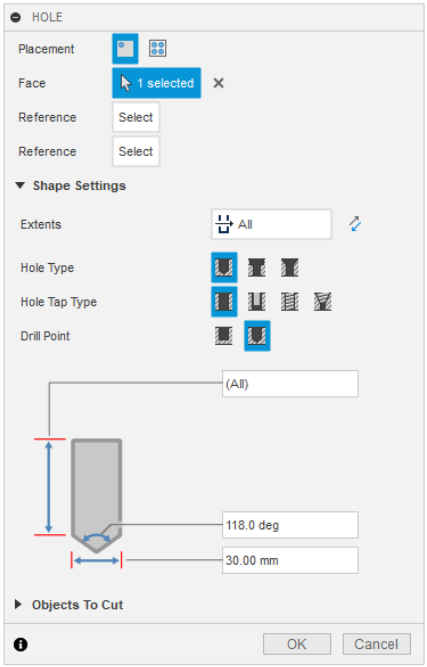
1. Create > *New Component*, name it “gear box” and make it active.
2. Save the design.
3. Create > *Create Sketch* on YZ plane.
4. Create > Rectangle > *Center Rectangle*. Start on origin. 250 x 250 mm size.



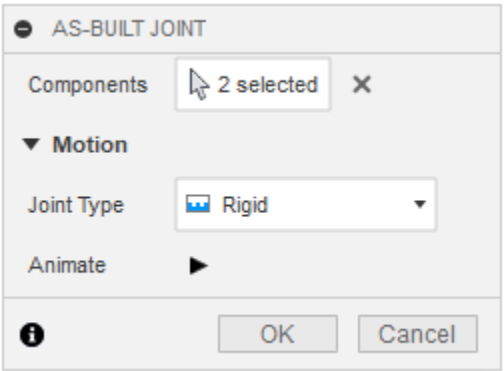

5. Finish the sketch and use Create > *Extrude*. Enter 150 mm for distance.

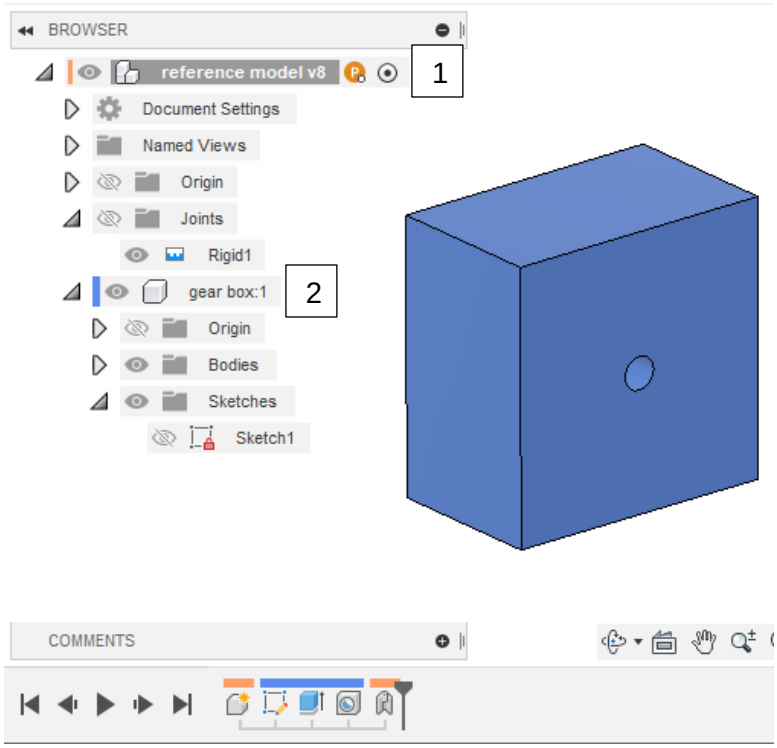
6. Now that you have a 3D shape, you can refine the design by adding features. In this case it needs a hole through the center.
    a. Create > *Hole*. Select the face, and then drag the center to snap with the dot that appears on the center of the face. You'll use this method a few times later in this tutorial.
    b. Make a simple hole, thru all, 30 mm diameter. "Simple" is the default style, you just need to add the diameter and make the termination "All".






7. Finish this component by locking all degrees of freedom to move using a joint command. This is so it will not move as you attach more components to it.
    a. Assemble > *As-BuiltJoint*, rigid type. Select the gear box component and design origin component, in the model browser, as the two components.






**Commands used:**

- New Component
- New Sketch
- Center Rectangle
- Extrude
- Hole
- Fillet
- As-Built Joint or Ground

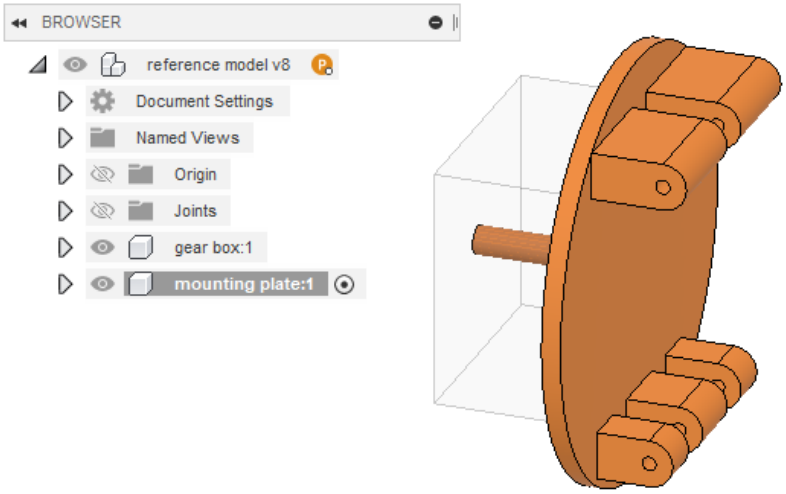
**Insights:**

- The best way to start an assembly design is to create a component, make it active, and then add sketches and modeling features.

- Also, it's critical to connect sketch geometry to the sketch origin and fully define the sketch with dimensions and constraints.

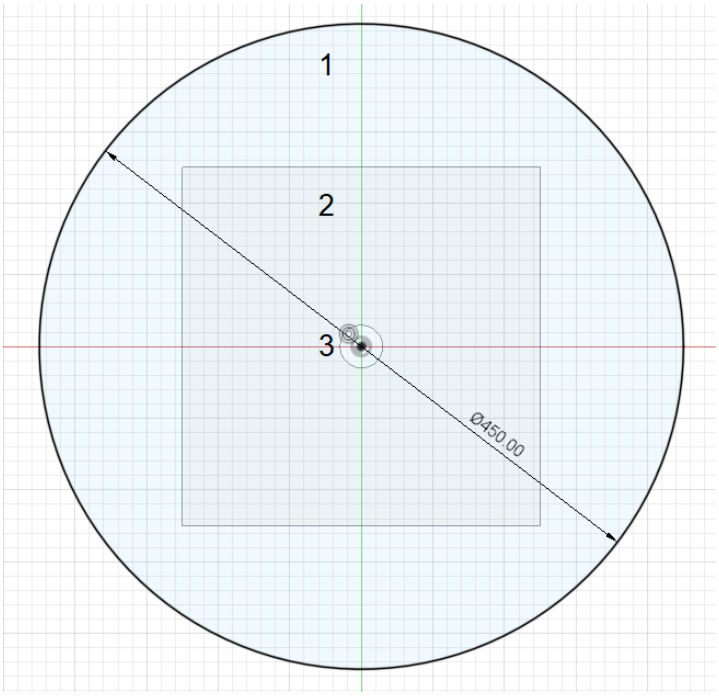
- If the design is symmetrical, use the sketch origin as the center of the diagram. Using *Center Rectangle* makes this easy.

## Step 2 – Mounting plate

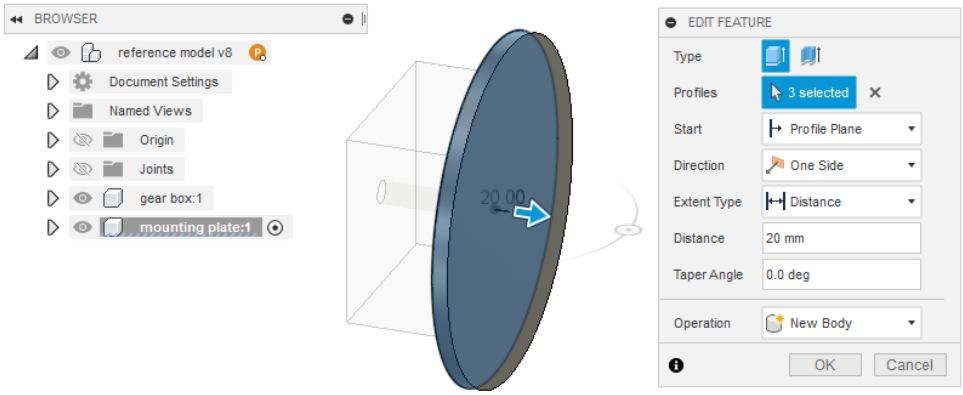



*THE FINISHED MOUNTING PLATE (ACTIVATED)*

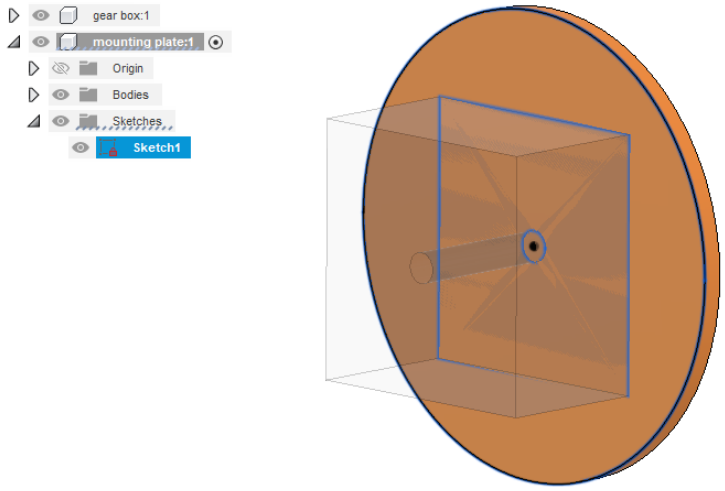
1. Activate the top level browser node (click the small dot next to the name).
2. Create > *New Component.* Name it “mounting plate” and ensure it is activated.
3. Create > *New Sketch* on front of gear box component.
4. Create > Circle > *Center Diameter Circle:* diameter 450 mm.



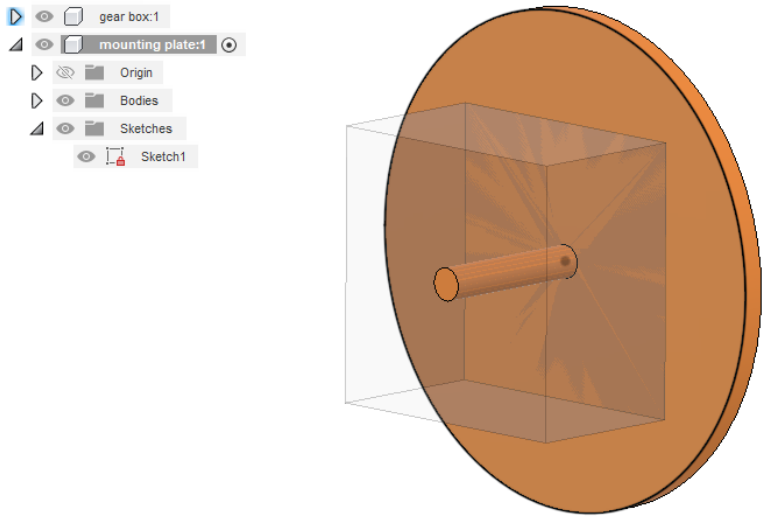

5. Solid > Create > *Extrude*. Select the three sketch profiles inside the circle. Direction and distance is 20mm away from the gear box.



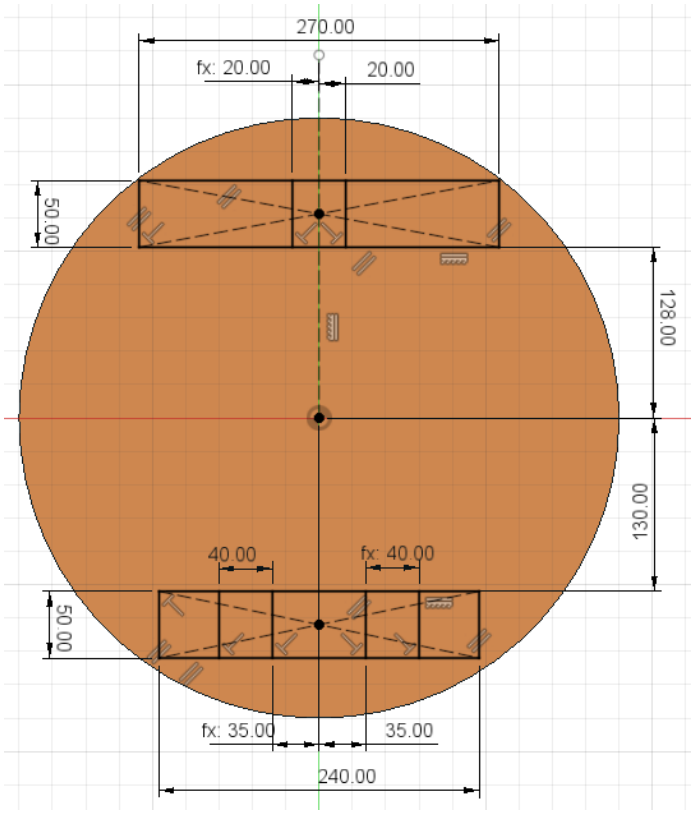

6. Look for Sketch1 in the main mounting plate component in the model browser. Click the eye icon to make it visible.



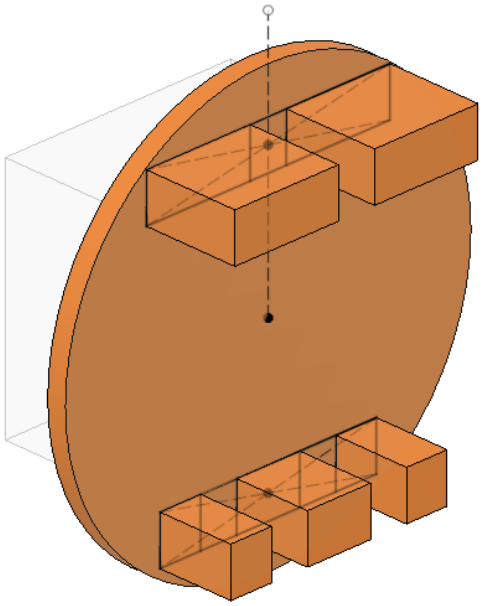

7. *Extrude* the center shaft shape.



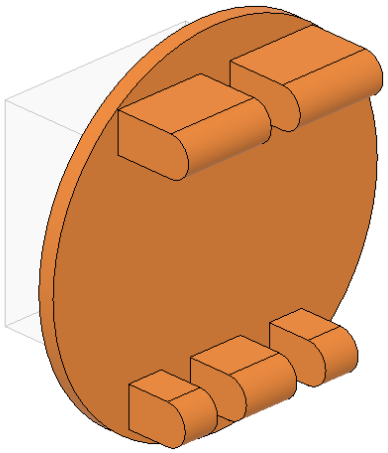
8. On the front of the mounting plate create this *New Sketch*.



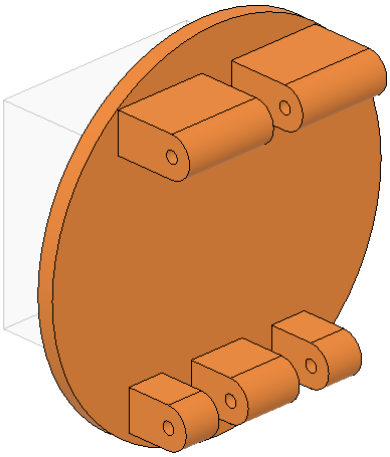

9. *Extrude* the two top rectangles 100 mm.
10. Make the sketch visible again, *Extrude* the three bottom rectangles 80 mm.



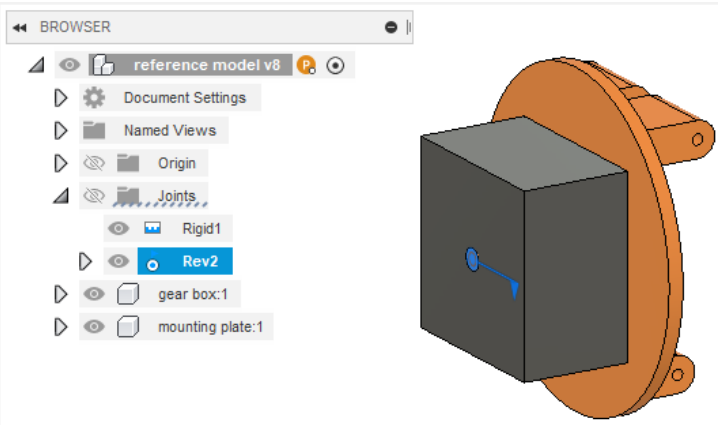
11. Add full-round fillets using Modify > *Fillet* > (type) *Full Round Fillet.*



12. Create > *Hole.* Add Ø 15 mm holes thru the centers of the full-round fillets. Use the technique you used on the gear box, drag the hole center to snap to the center point.



13. Assemble > *As-Built Joint*. Use a revolute joint to the gearbox using the shaft axis.



**Commands used:**

- New Component
- New Sketch
- Sketch line
- Sketch dimension
- Extrude
- Hole
- Fillet
- As-Built Joint

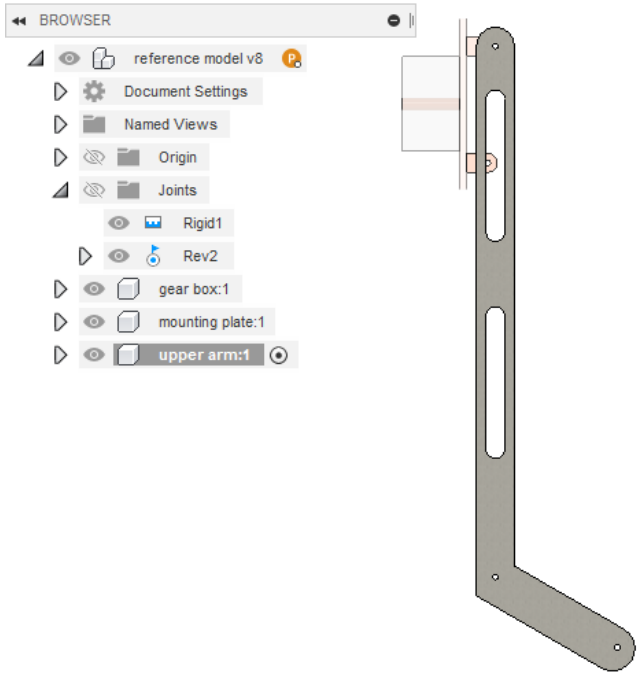
**Insights:**

- Using an existing part as a location to place and create a component that shares geometry with it is a best practice. This creates a parametric relationship between the parts, which is often called “associative” if done this way. Edits to the gear box shape are associatively tied to the first sketch of the plate.

- Sketches are consumed by features, but you can make them visible again and enable more features to consume the same sketch.

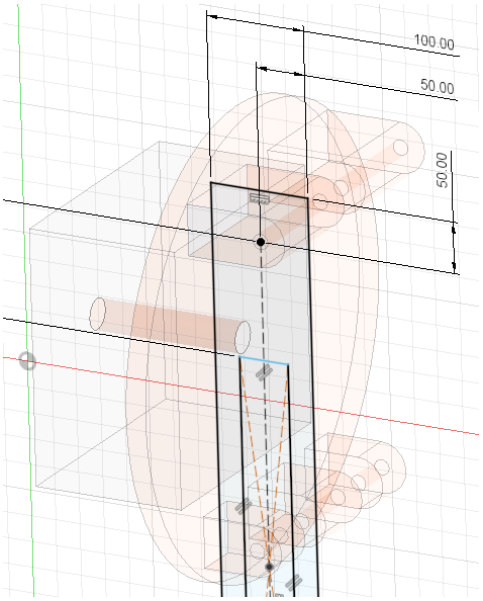
- As-Built joint is intended to make defining motion easy when components are modeled in place together.

## Step 3 – Upper Arm

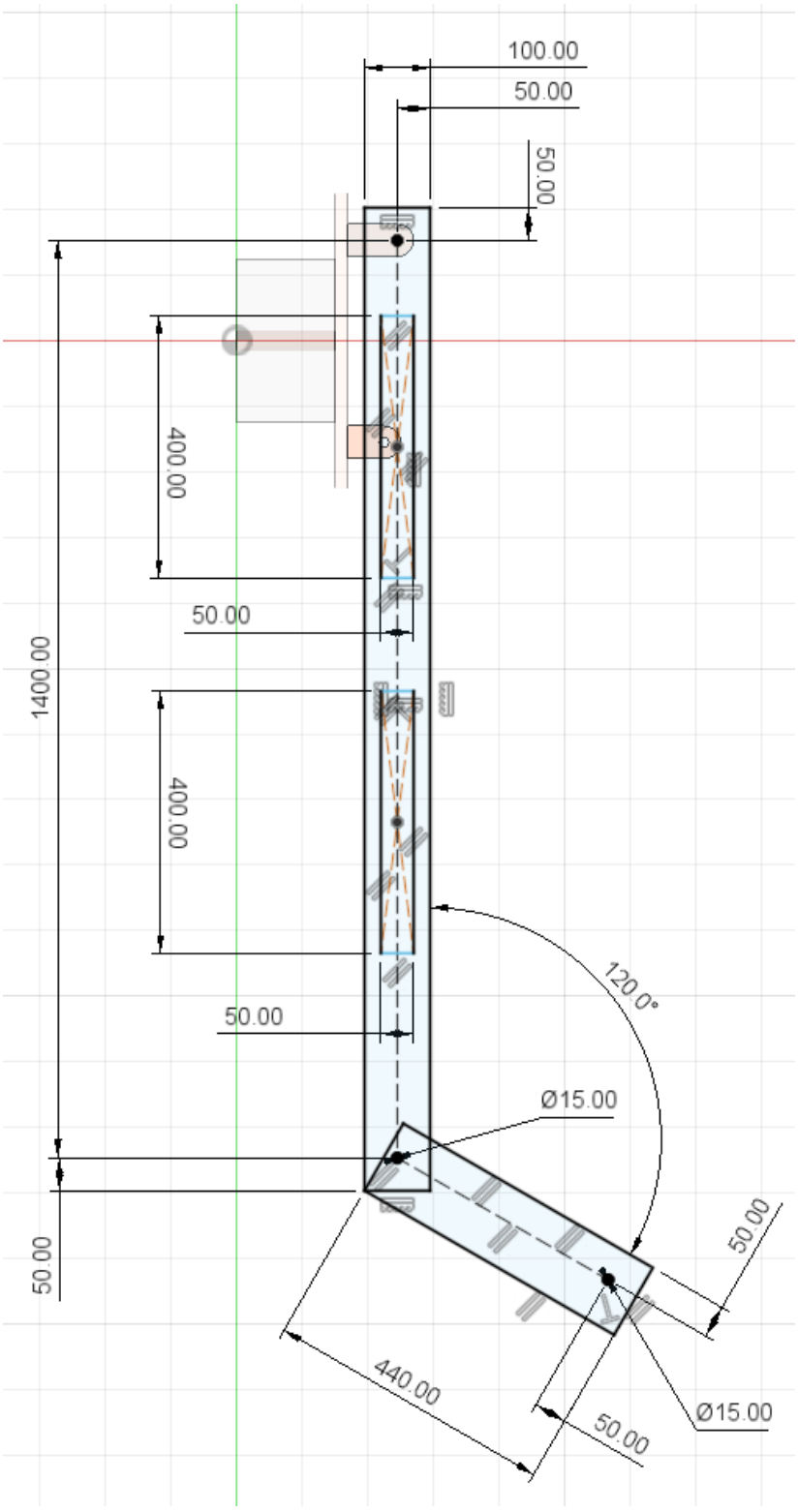



*THE FINISHED UPPER ARM (ACTIVATED)*

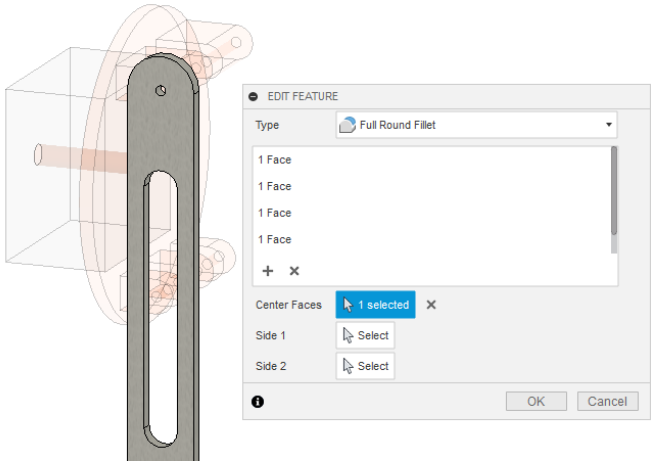
1. Activate the top level browser node.
2. Create > *New Component*, call it “upper arm”. Activate it.
3. Create > *New Sketch* on the outside face of the taller extruded rectangles on the plate.



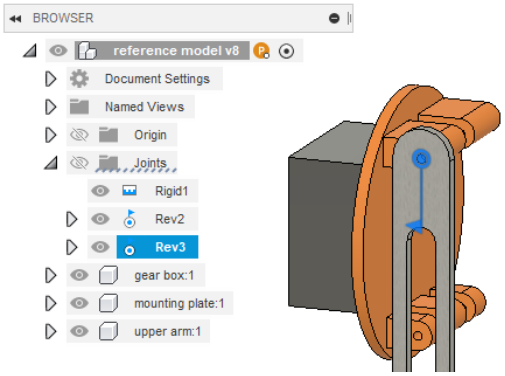
4. Complete sketch:



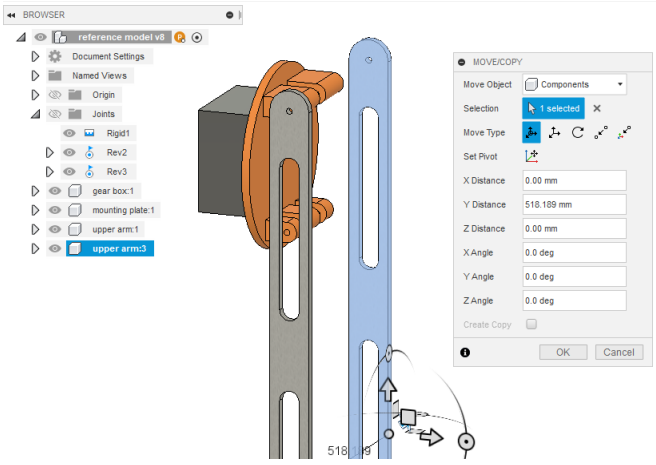
5. Solid > Create > *Extrude* the shape 15mm away from the plate.
6. Modify > Fillet > *Full-round Fillets* at each end of the arm.
7. Add more full-round fillets inside the slot shapes.



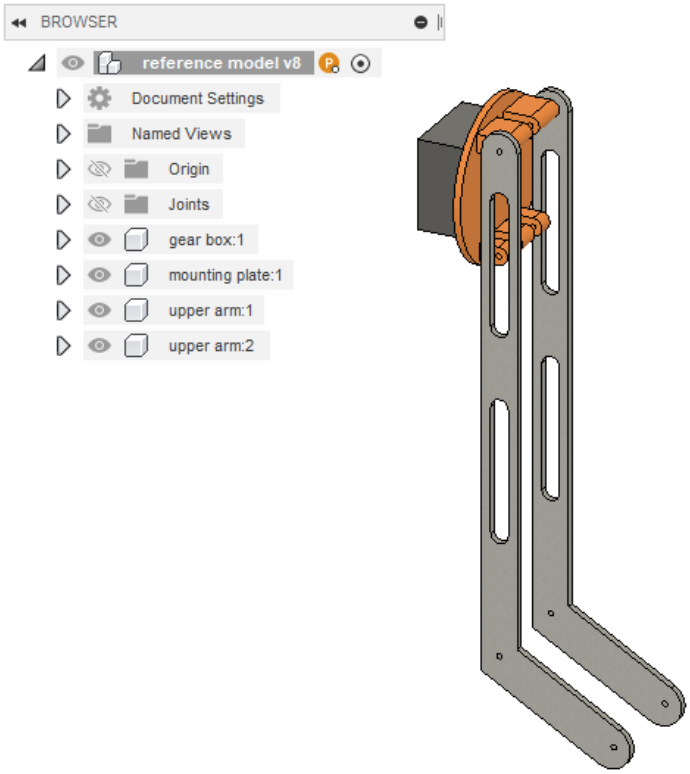

8. Make an *As-Built Joint*, Revolute, between the top of the arm and the plate.




9. Right click the upper arm component. *Copy* and *paste* it to make another instance.



10. Use Assemble > *Joint* to add a revolute Joint to to the second arm to place it on the opposite side of the plate.
11. Use Assemble > *As-Built Joint.* Picking the two arms for a rigid joint. This will make them move together.



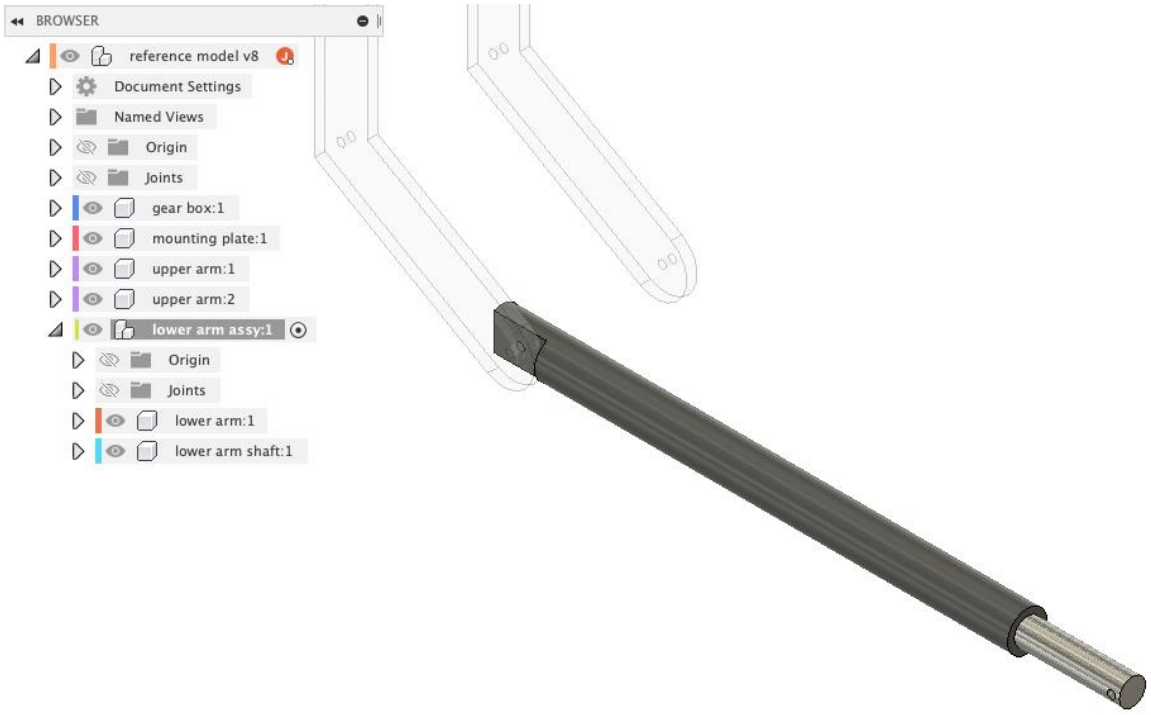

**Commands used:**

- New Component
- New Sketch
- Sketch line
- Sketch circle
- Sketch dimension
- Extrude
- Fillet
- Joint
- Copy/Paste
- As-Built Joint

**Insights:**

- Creating another instance of a component is easily done with copy/paste hotkeys (ctrl+c / ctrl+v) , but the key to success is using the model browser to make the component selection.
- Rigid Group/Rigid As-Built joint is a way to make components move together.

## Step 4 – Lower Arm Assembly

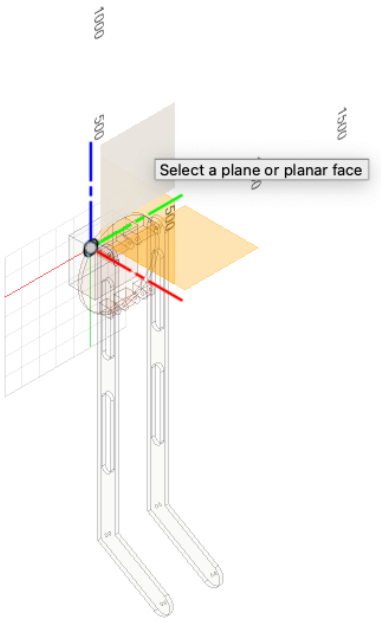



*FINISHED LOWER ARM ASSEMBLY*

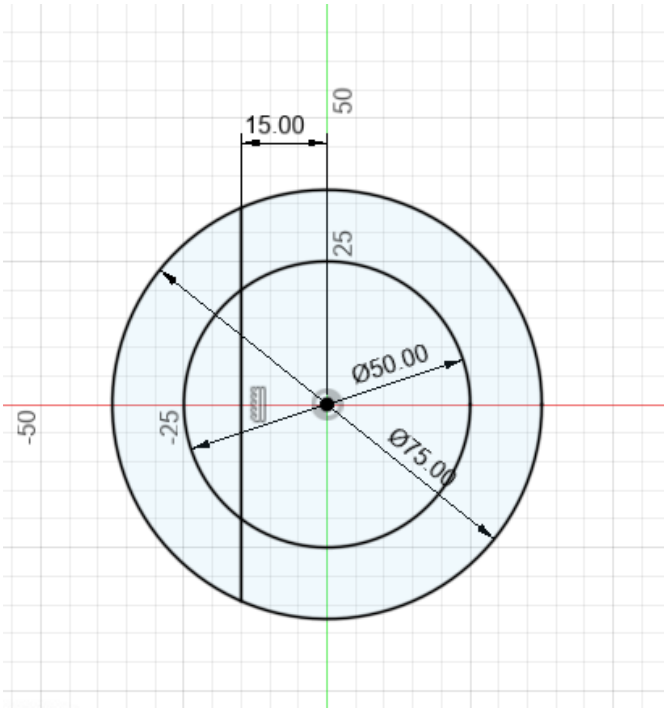
This is our first “out of place” component, and our first sub-assembly. We will be building this based on one of the global origin planes, because there is no convenient plane on the upper arm component to create the base sketch. This step will show the method used to create components that are not built in place, and the use of the Joint command to connect them to the rest of our design.

1. Activate the top level browser node.
2. Create > *New Component* , named “lower arm assy”, activate it.
3. Create > *New Component*, name it “lower arm”, activate it.
    a. This step creates a sub-component inside the one you just made called “lower arm assembly” and makes it into a sub-assembly, which is an assembly inside another assembly. A group of parts that works together is an assembly and these can be nested inside of larger assemblies.

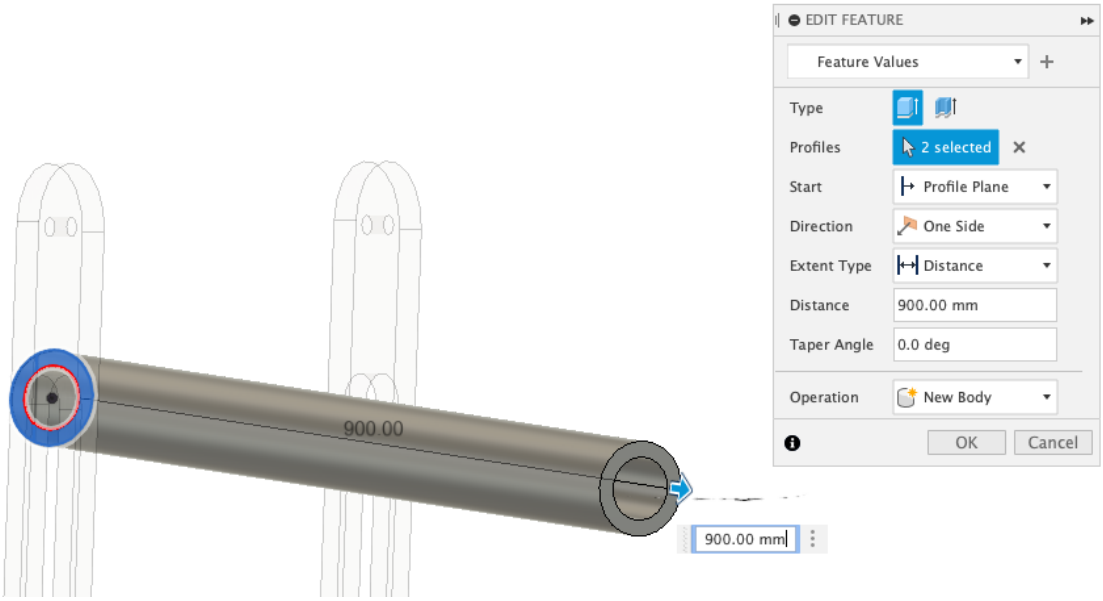
4. Create > *New Sketch*, selecting the global YZ plane.



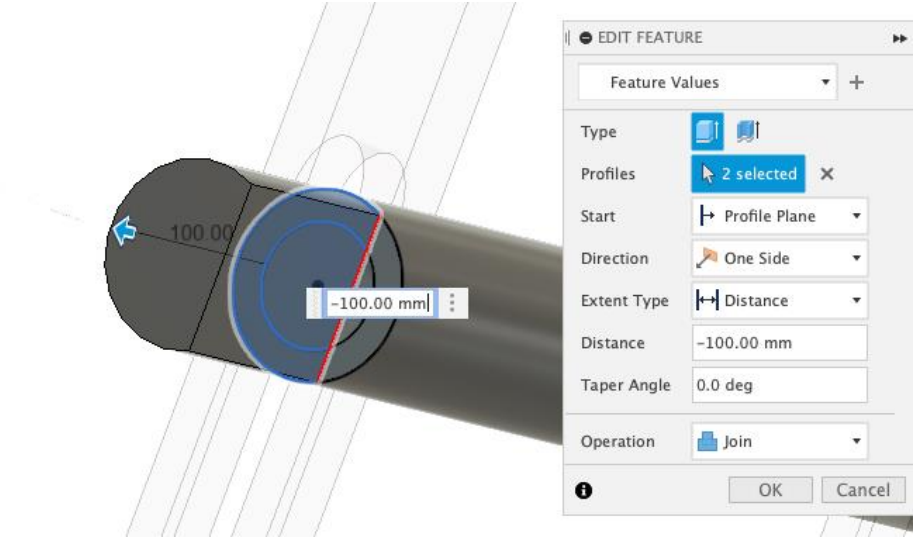

5. Create the sketch geometry: two circles, centered on the sketch origin, dimensioned as shown, and one vertical line to the left of the origin:



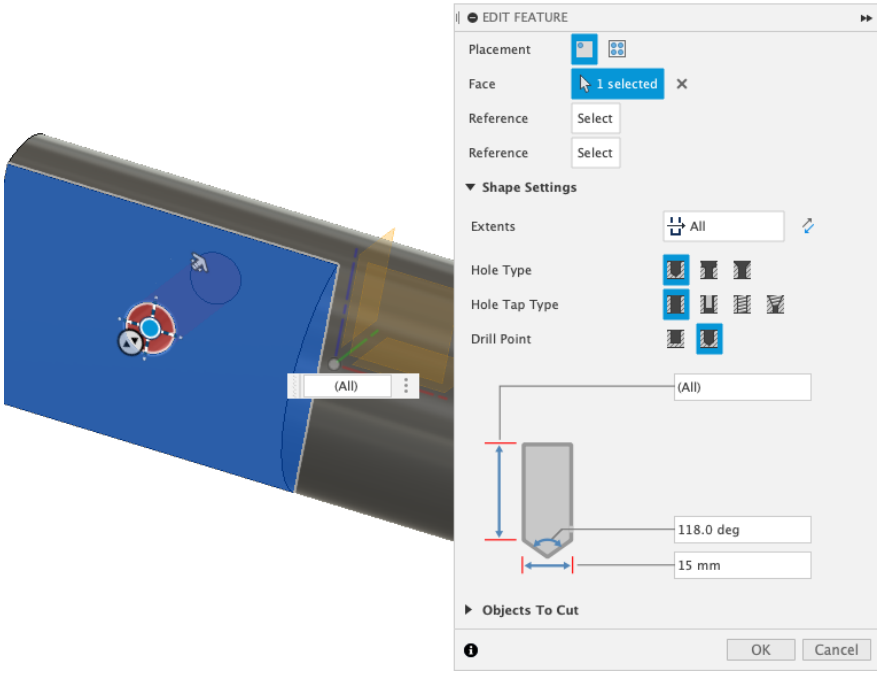
6. *Extrude* the outer cylinder by 900 mm:



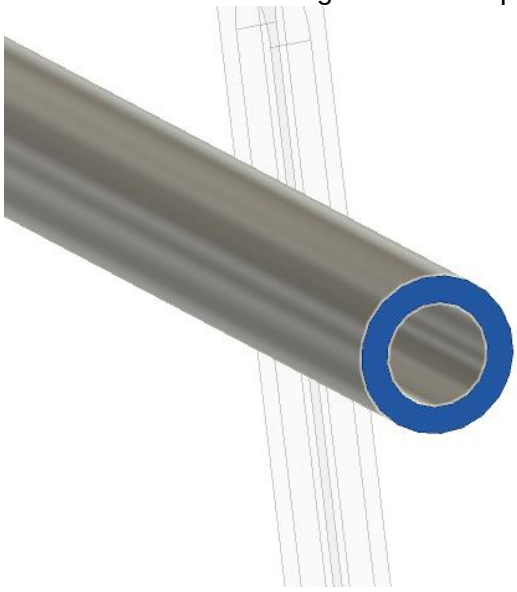

7. Make the sketch visible, and *Extrude* the mounting tab on the back side, -100mm, from the geometry defined by the circles and the vertical line:



8. Add a *Hole* to the center of this mounting face, 15mm in diameter, through the mounting tab:

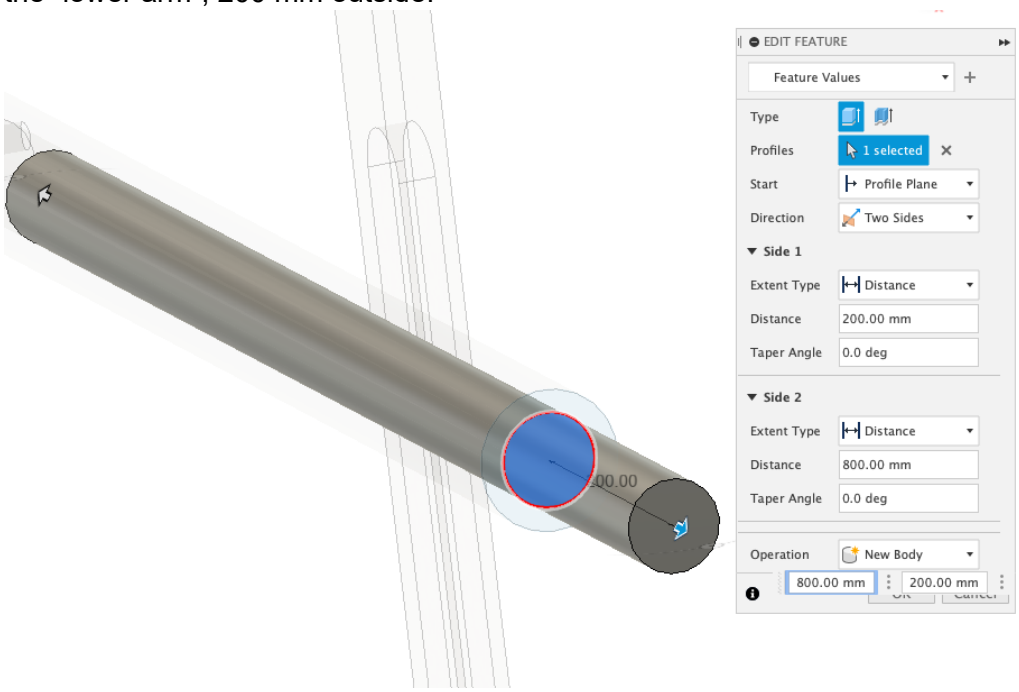



9. Activate the “lower arm assy” component.
10. Create > *New Component* named “lower arm shaft” so that it is a sibling to the “lower arm” component (both nested under “lower arm assy”).
11. Create > *New Sketch*, selecting the end face of the “lower arm” component.
    Note: We will be building this shaft in-place with respect to the “lower arm”.

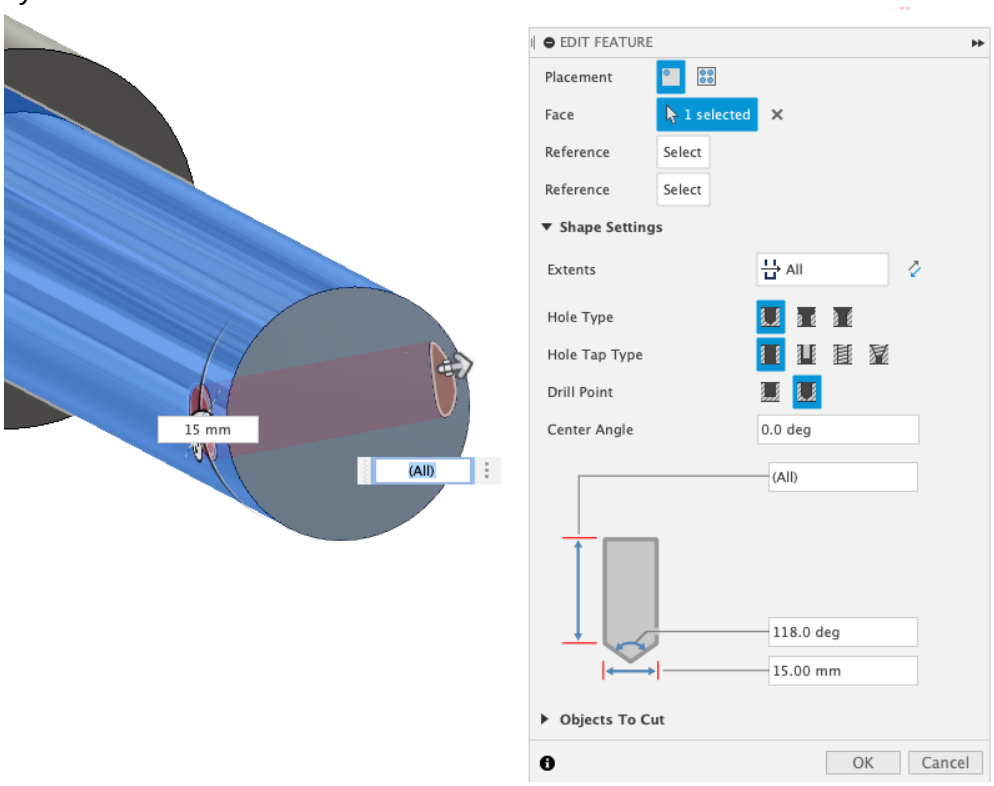


12. No sketch geometry is needed for this component. By sketching on model geometry, we have all the geometry we need!

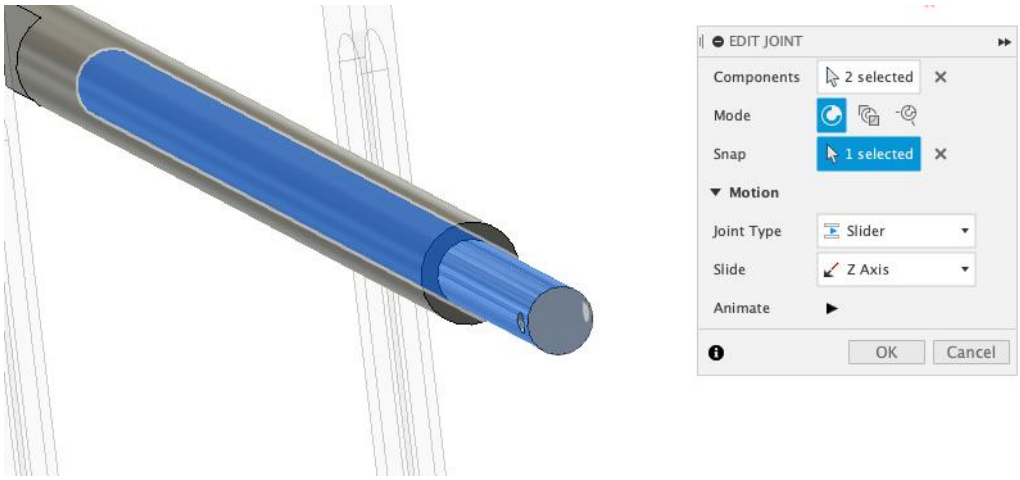
13. *Extrude* the inner circle, using a “Two Sides” direction (this is new). 800 mm into the “lower arm”, 200 mm outside:



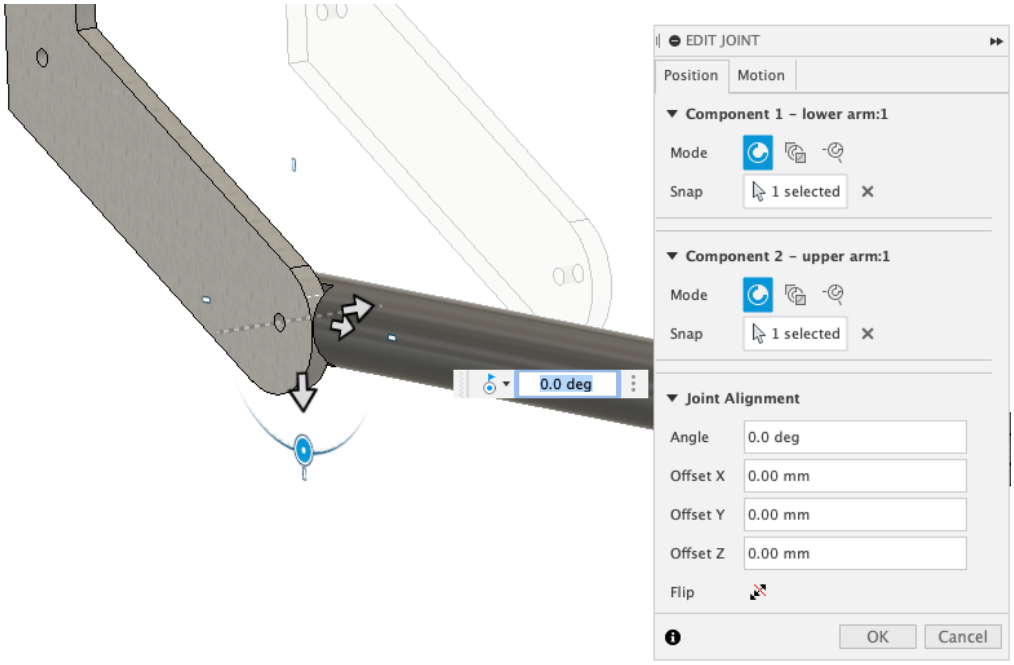

14. Add a *Hole* to the end of the shaft. Key: selecting the 9 o’clock position on the cylindrical face:



15. Place an *As-Built Joint* between the shaft and the arm, motion type of Slider, and choose a point on the shaft cylinder to define the motion to be along the cylinder axis:

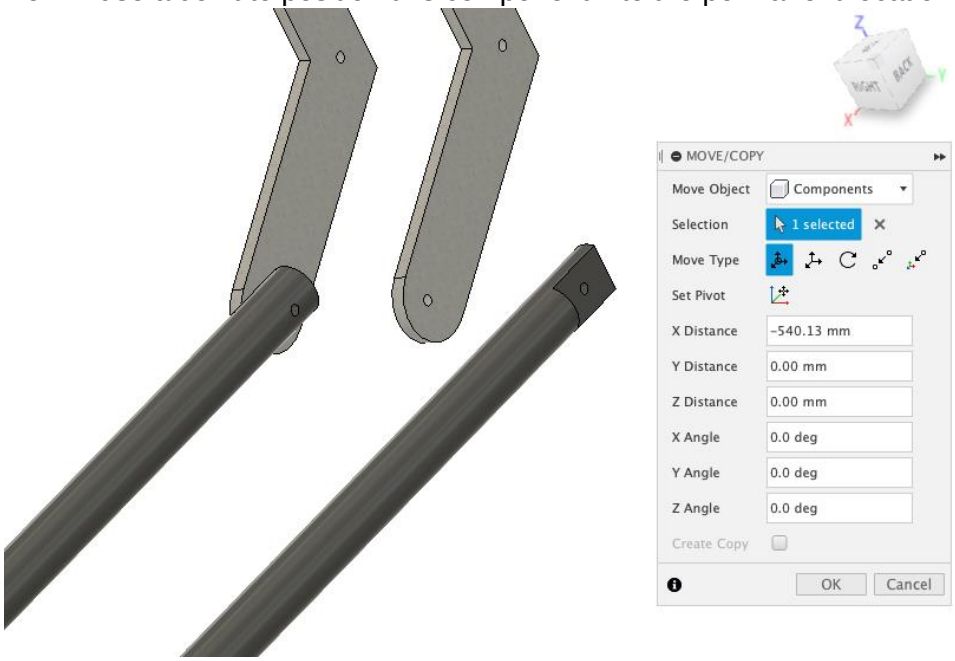



16. Because the sub-assembly was not built in place with the rest of the design, we use a *Joint* to both position the “lower arm” component with respect to the “upper arm” component, and to define its motion (Revolute Joint). Choose the center of the mounting holes on the “lower arm” and “upper arm” components:

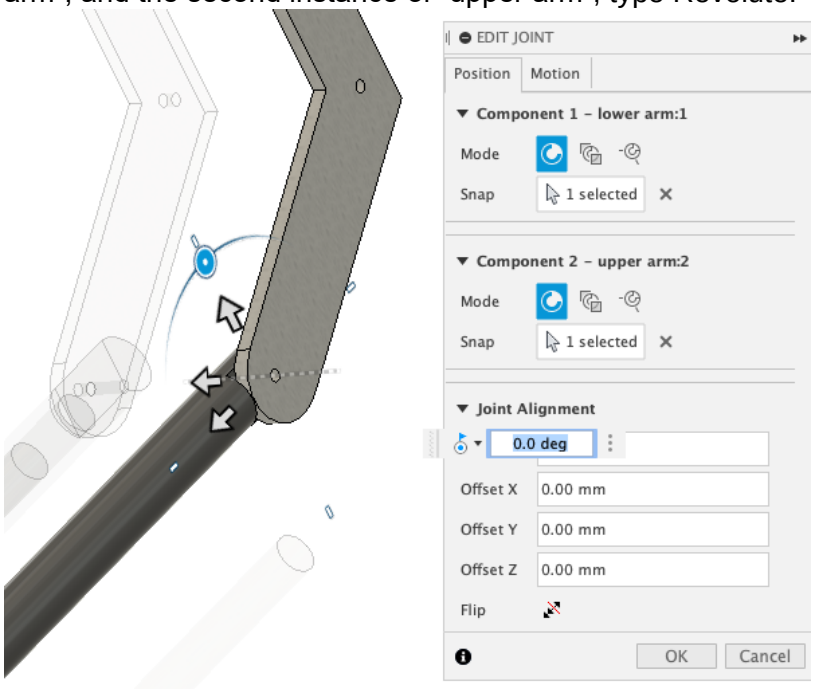



17. Now, we need one more instance of the “lower arm assy” component. Select it in the browser, then *Copy and Paste* it to make another instance.

18. After pasting, the move command starts to allow you to position the newly pasted sub-assembly. This is for convenience, but it helps to get it closer to the place it joins the assembly. We need this to be rotated so that the flat part of the mounting tab is in line with the vertical face of the second “upper arm” component instance.  A temporary position for this component is okay, because we will use a Joint to position this component into the permanent location.




19. Assemble > *Joint* betweent the mounting holes on the second instance of “lower arm”, and the second instance of “upper arm”, type Revolute.



20. Because we want the two lower arms to move in tandem, create an *As-Built Joint* between the two instance of both “lower arm” and “lower arm shaft”, with type Rigid. This assures that they will move together.

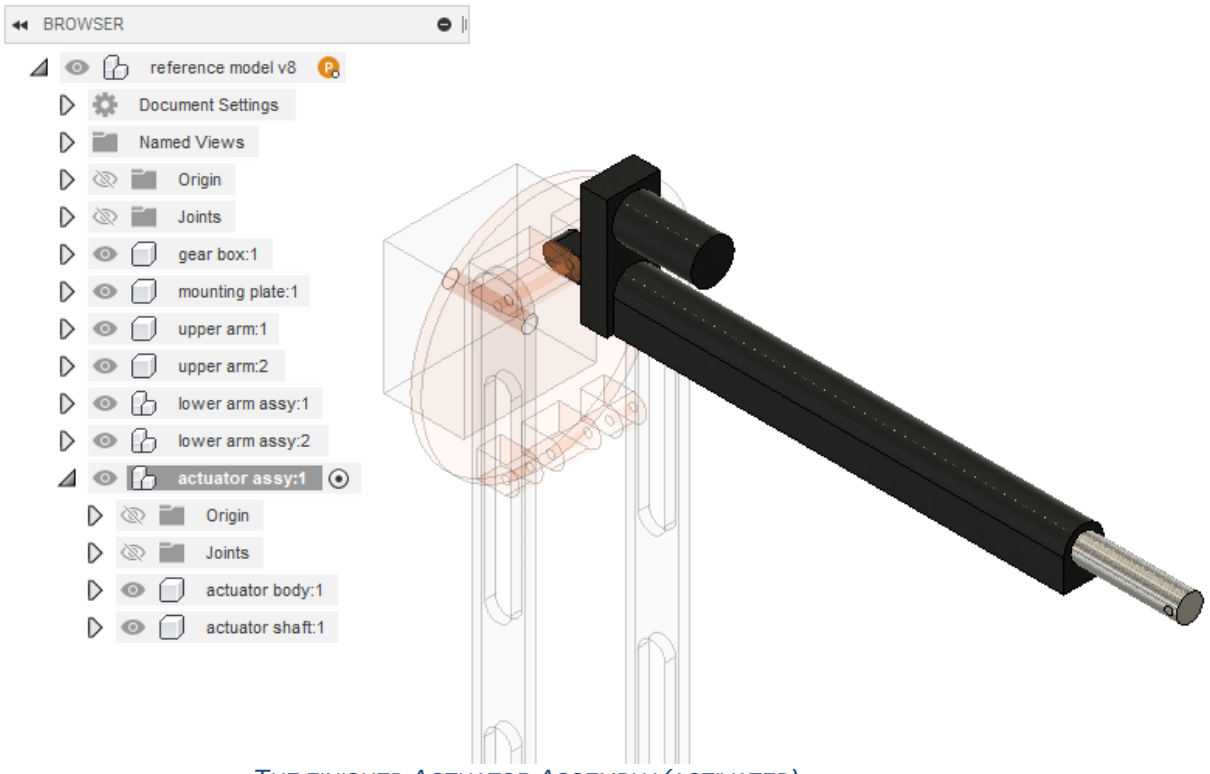
**Commands used:**

- New Component
- New Sketch
- Sketch line
- Sketch circle
- Sketch dimension
- Extrude
- Hole
- Joint
- Copy/Paste
- As-Built Joint

**Insights:**

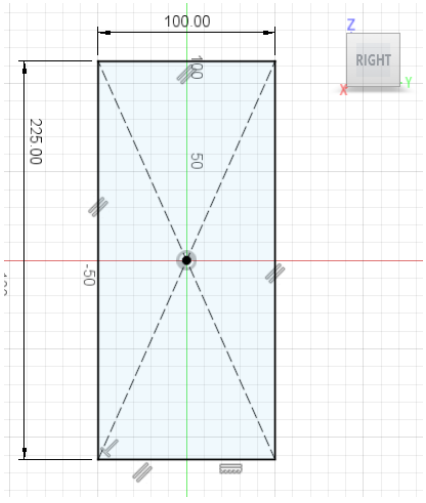
- Creating a component not in position is useful sometimes.
- Creating sub-assemblies with joints internal to the member components means less joints to create later.
- The *Joint* command defines position and motion of two components relative to each other.

## Step 5 – Actuator Assembly

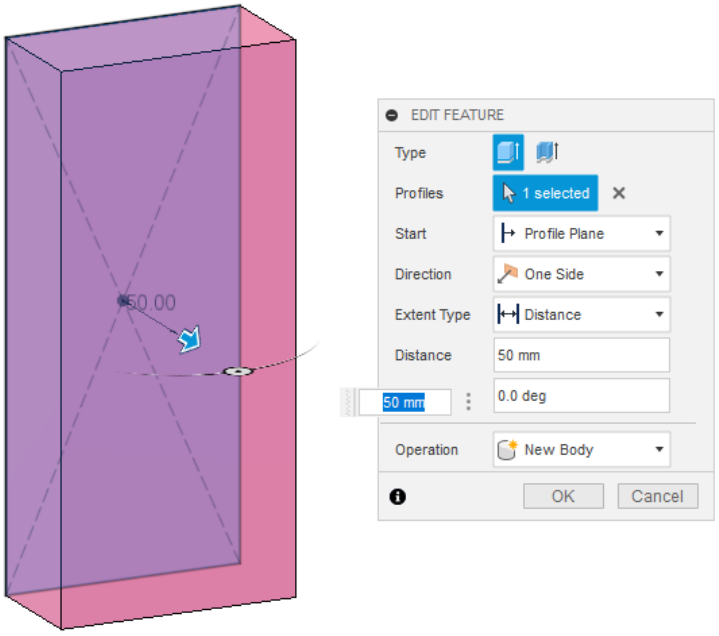



*THE FINISHED ACTUATOR ASSEMBLY (ACTIVATED)*

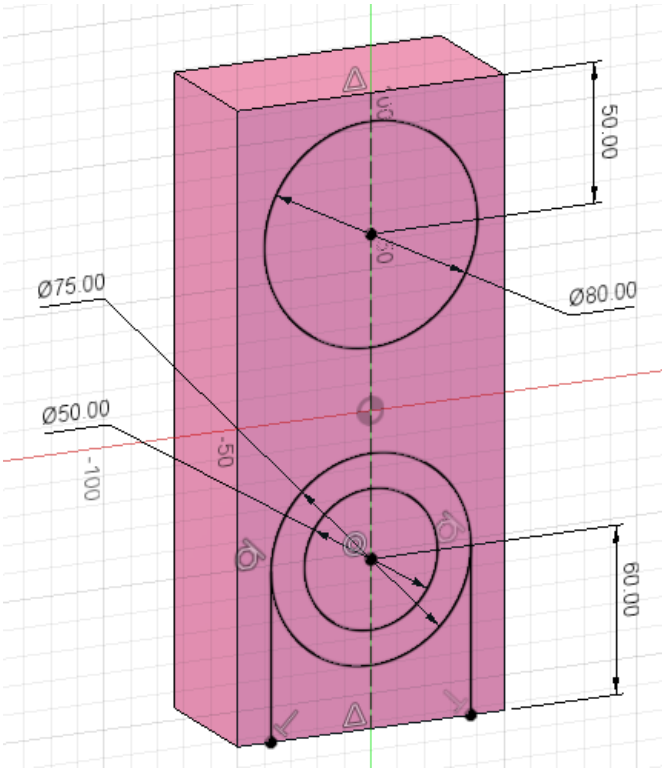
1. Activate the top level browser node. Time for another sub-assembly.
2. Create > *New Component*, named “actuator assembly”, activate it.
3. Create > *New Component* , named “actuator body”.
4. Create > *New Sketch* on the YZ plane.
5. Create > Rectangle > *Center Rectangle*, 100 x 225 mm.



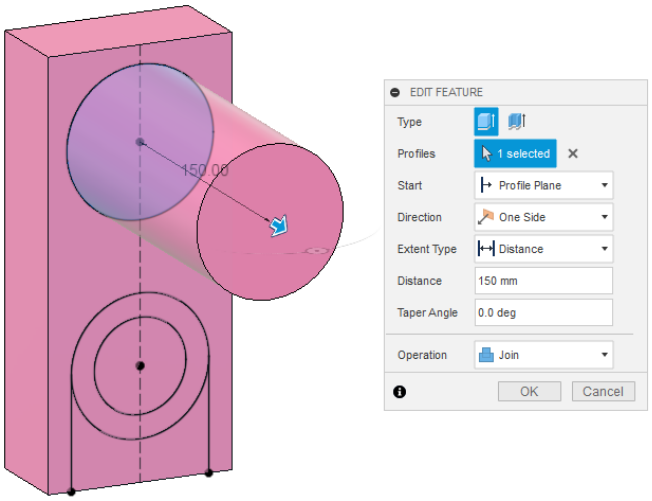
6. *Extrude* the rectangle 50 mm. Note: Inspect > *Display Component Colors* is turned on in the following images for clarity:



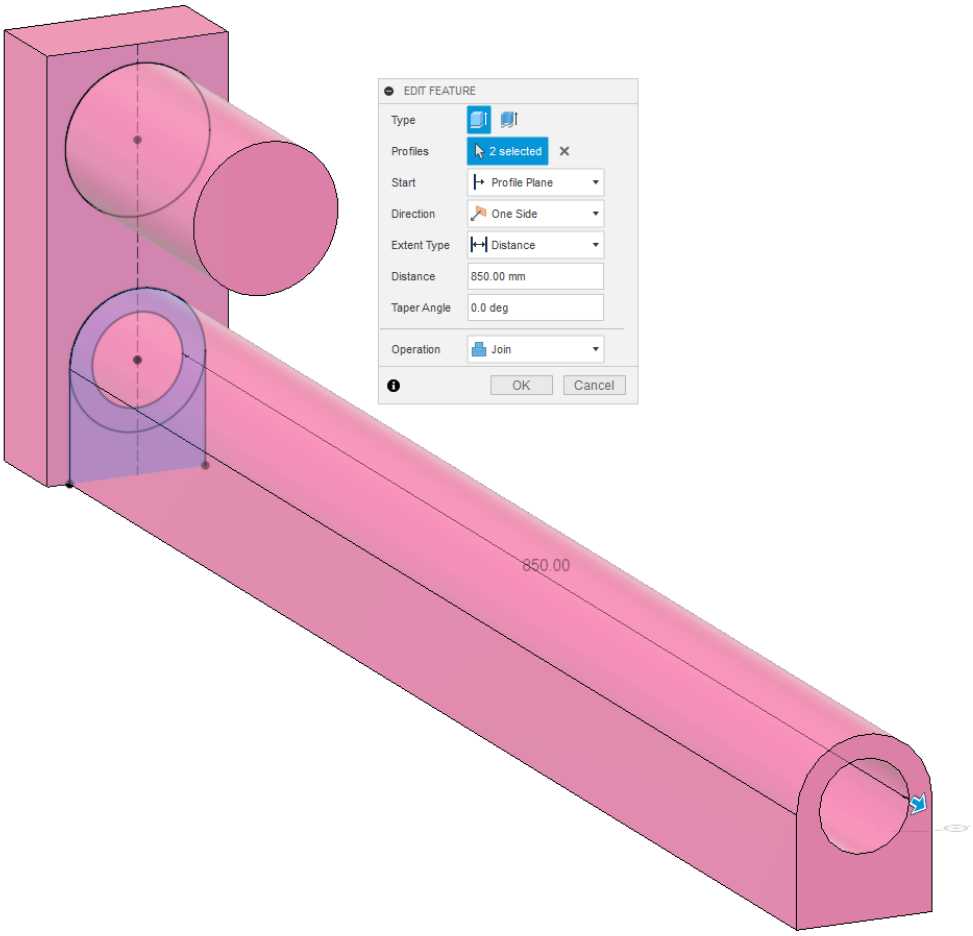

7. *New Sketch* on the front of that shape:



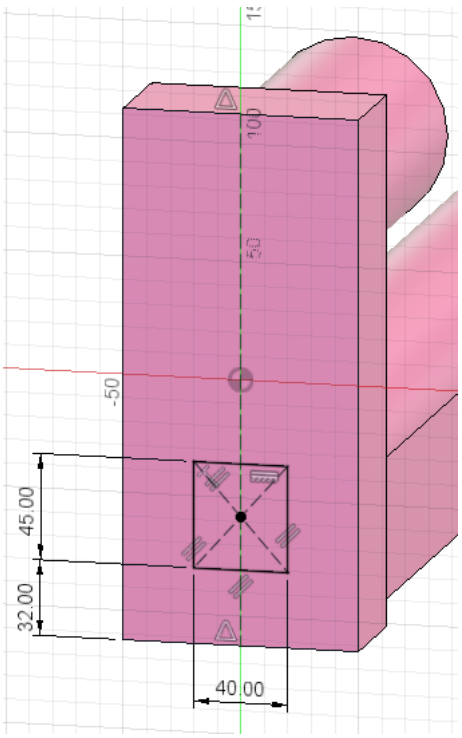
8. *Extrude* the top circle 150 mm:



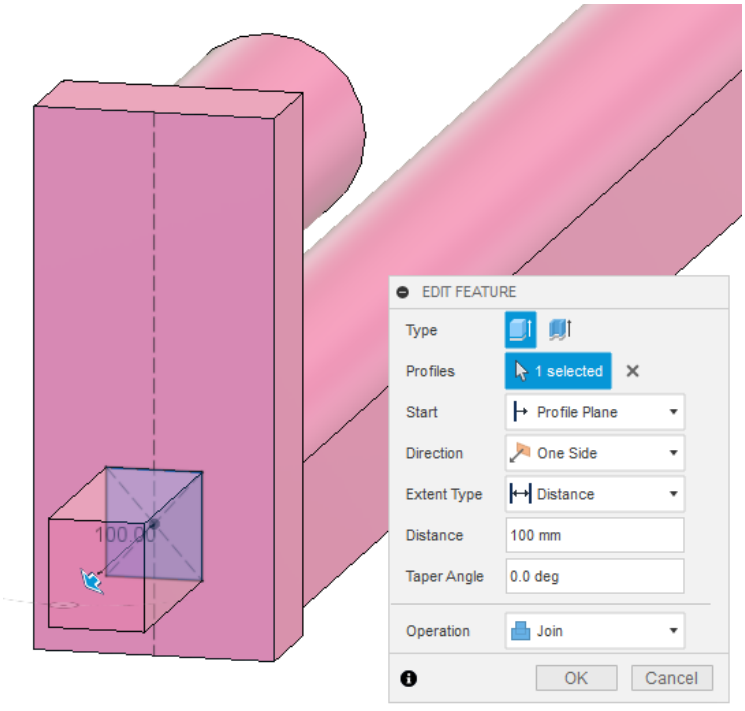

9. Make the sketch visible again using the “eye” visibility control for it in the browser. *Extrude* two of the lower profiles 850 mm to produce this result:



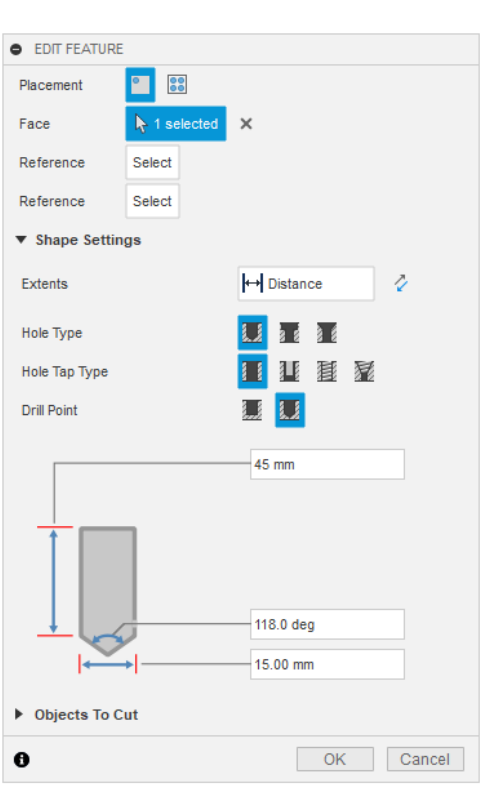
10. *New Sketch* on the side facing away from the last two extrudes and make this sketch:

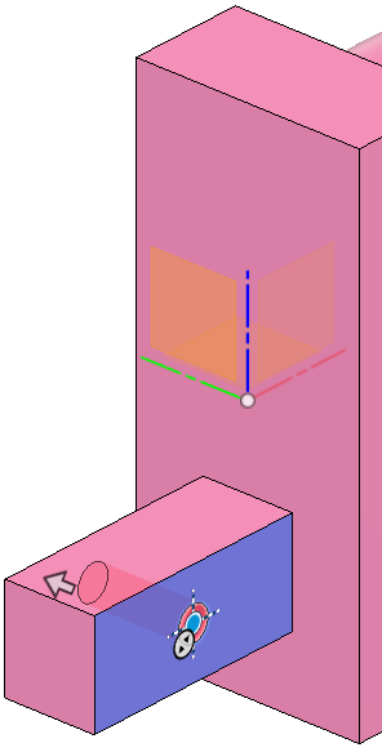



11. *Extrude* the rectangle 100 mm:

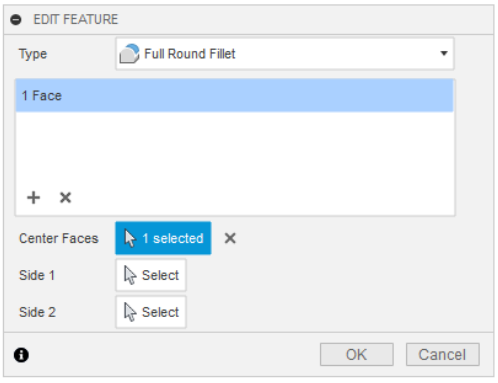


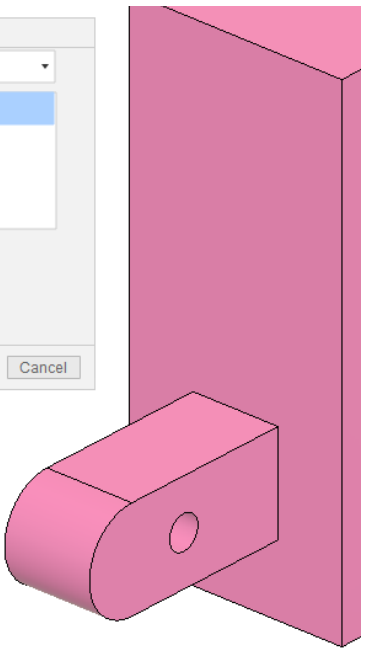
12. Use the *Hole* command to place a Ø15 mm hole, centered on this face, with a distance of 45mm:



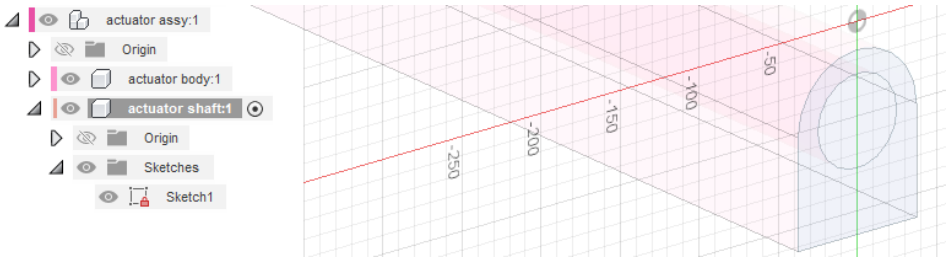



13. Add a *Full Round Fillet* to the shape:

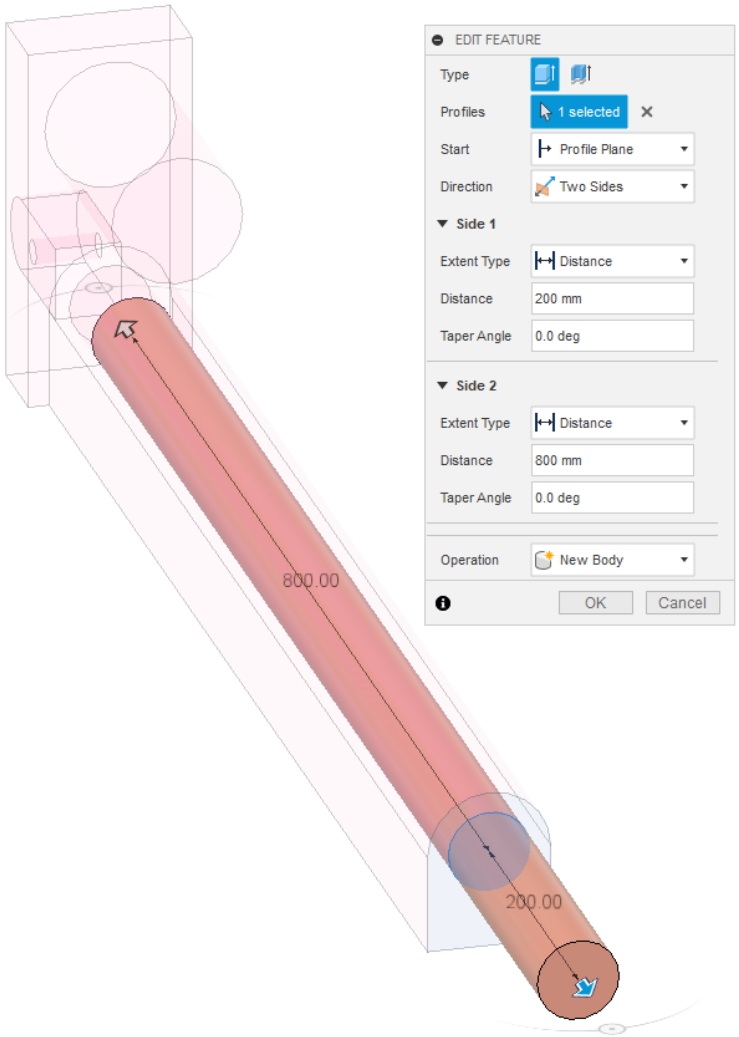


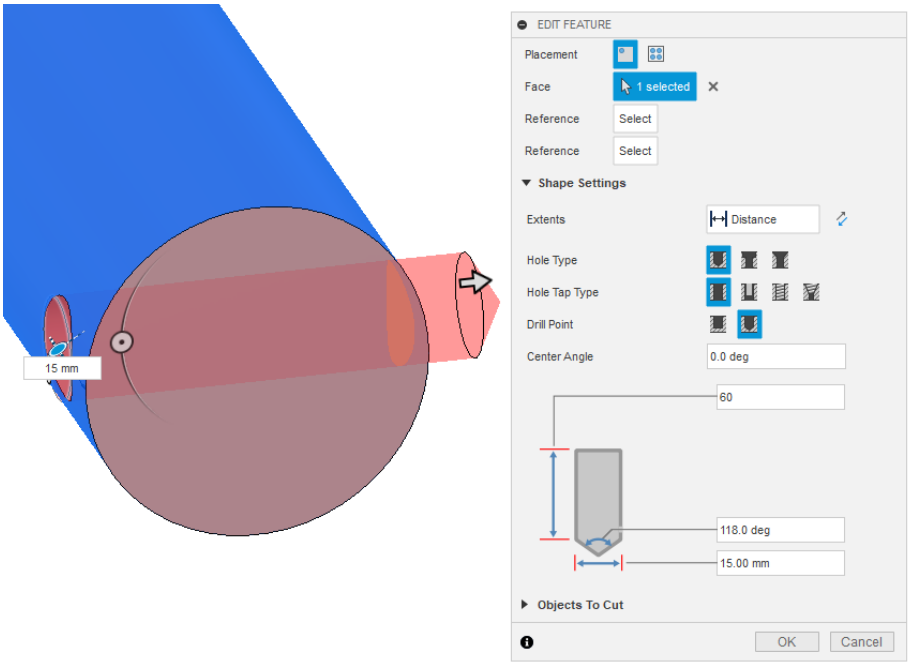
14. Activate the component called “actuator assembly”.
15. Create > *New Component*: “Actuator shaft”.
16. *New Sketch*: on the end of the actuator body component:



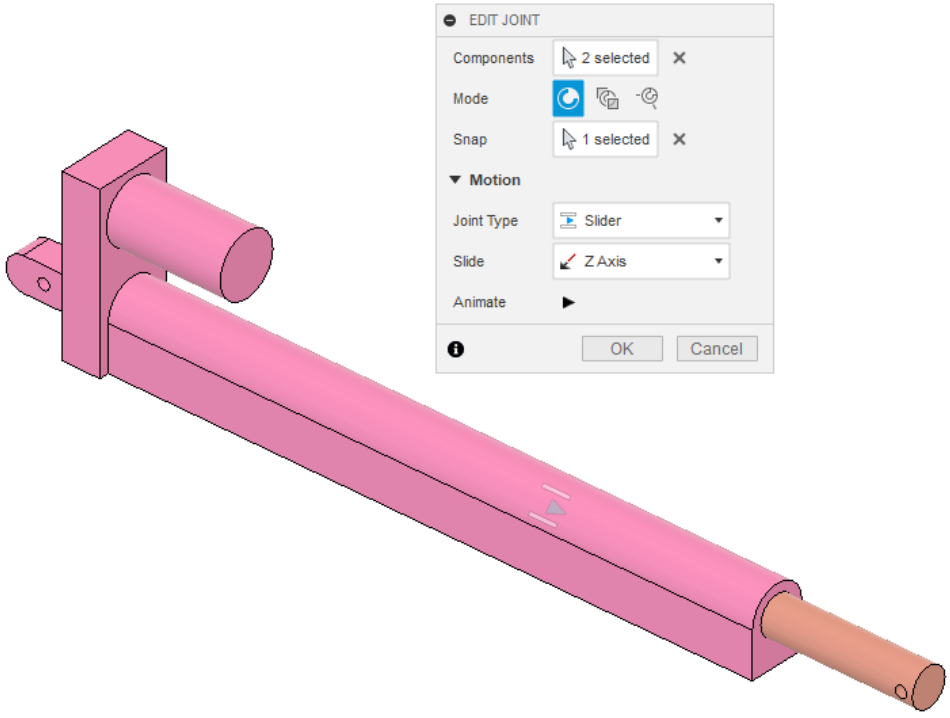

17. Finish the sketch without drawing anything. Fusion is including all the geometry needed, getting it from the face of the other component.
18. *Extrude* the center circle with the settings: Direction: Two Sides, 200, 800 mm:



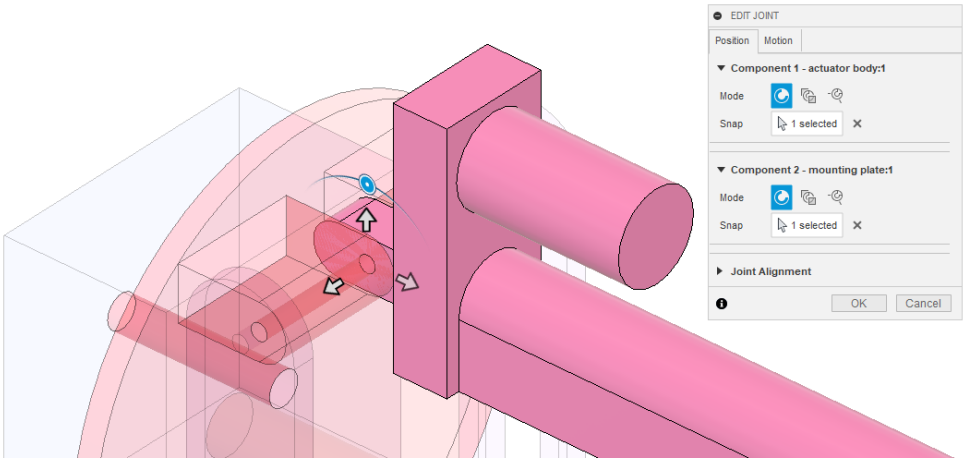
19. Create a Ø15 mm *Hole* near the exposed end of the shaft, using the 9 o'clock snapping position of the hole angle manipulator:




20. Make an *As-Built Joint*, Slider type, between the actuator shaft and actuator body:



21. Use a *Joint* with Revolute motion to attach the actuator body to the mounting plate:



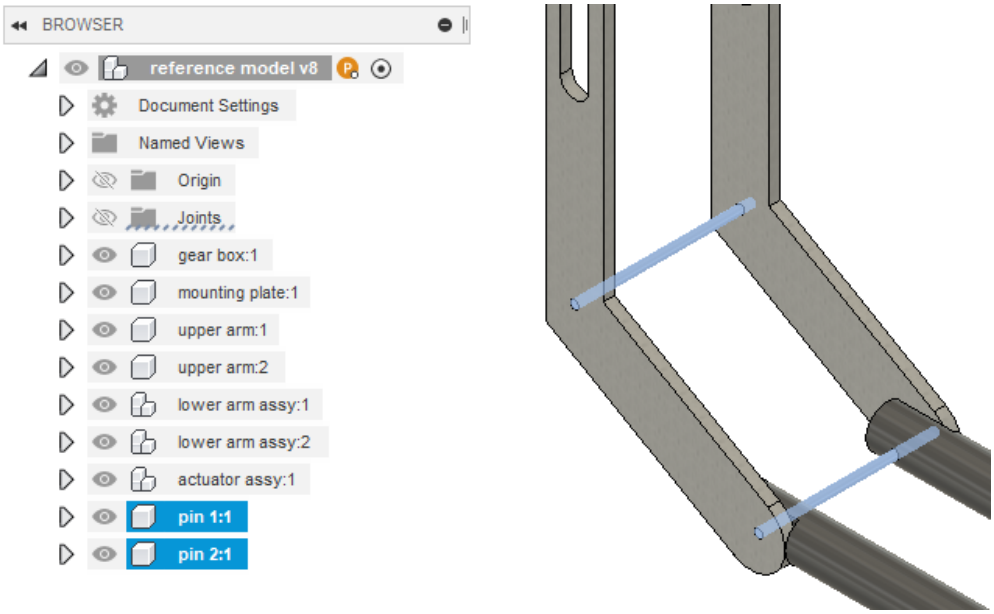

**Commands used:**

- New Component
- New Sketch
- Sketch line
- Sketch circle
- Sketch center rectangle
- Sketch dimension
- Extrude
- Hole
- Joint
- As-Built Joint

**Insights:**

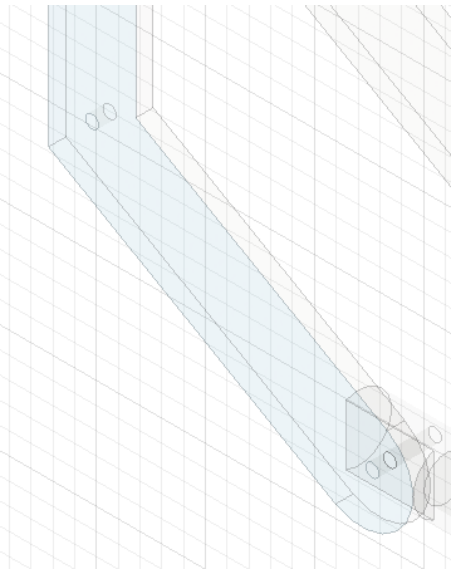
- Sometimes you don't need to add anything to a sketch to extrude with it. The edges are included when you sketch on a face.
- When two components are created from one sketch, you can take advantage of the *As-Built Joint* workflow, defining movement based on As-Built position.

## Step 6 – Pins 1 and 2

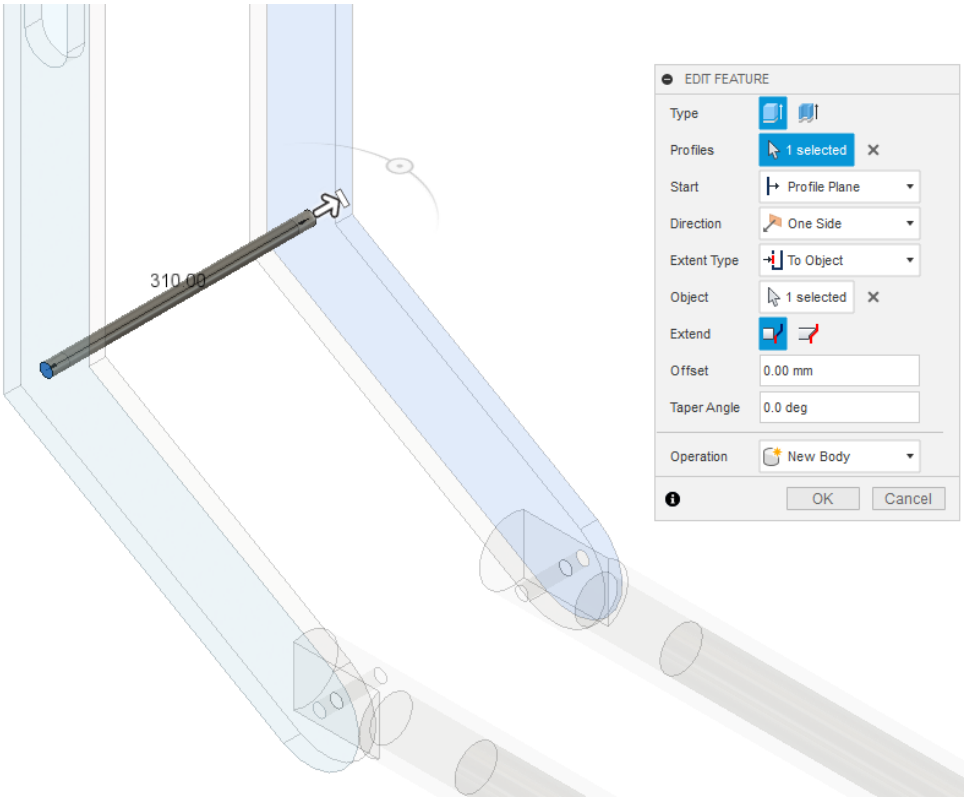



*PINS 1 AND 2 FINISHED (SELECTED)*

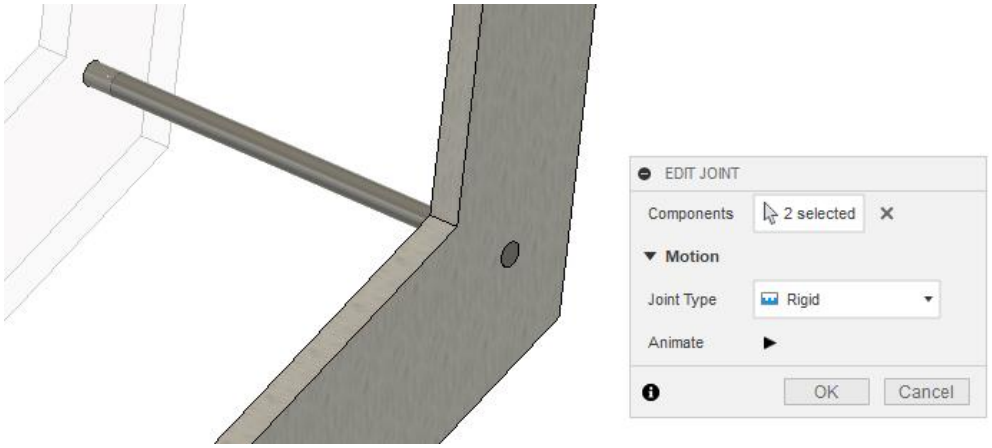
1. Activate the top level browser node.
2. Create > *New Component*: Pin 1, activate it.
3. *New Sketch*, on the outer face of the upper arm:



4. *Extrude* profile that fills the pin hole using extent type: To-Object. Select the outer face of the other upper arm:




5. Create a rigid *As-Built Joint* between the pin and the upper arm components:




6. Activate the top level of the assembly.
7. Create > *New Component*: name it Pin 2, activate it.
8. Repeat the process you used for Pin 1.
    a. *New Sketch* on the outer face of the upper arm
    b. *Extrude* the pin shape using Extent Type: To Object.
    c. Pick the opposite upper arm, outer face as the object.
    d. Finish the extrude and apply a rigid *As-Built Joint* between the pin and the upper arm.

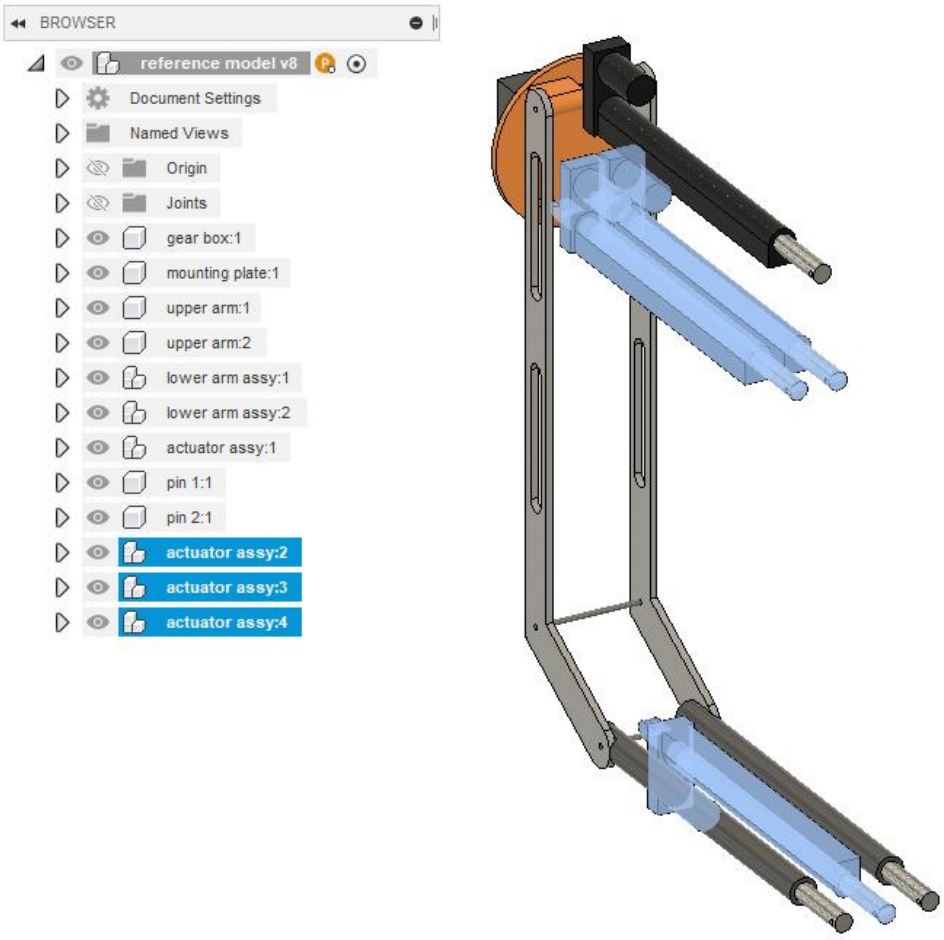
**Commands used:**

- New Component
- New Sketch
- Extrude
- As-Built Joint

**Insights:**

- Sometimes you don’t need to add anything to a sketch to extrude with it. The edges are included when you sketch on a face.

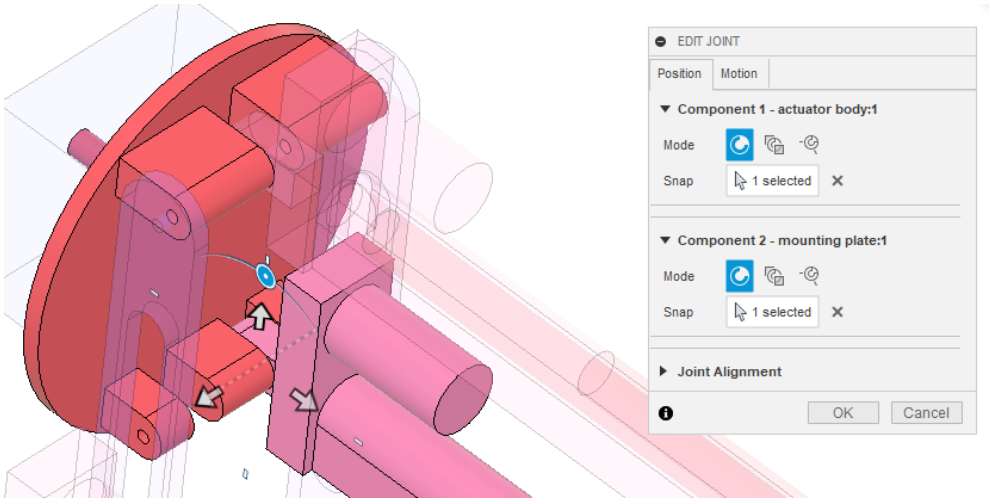
## Step 7 – Additional instances of the actuator assembly

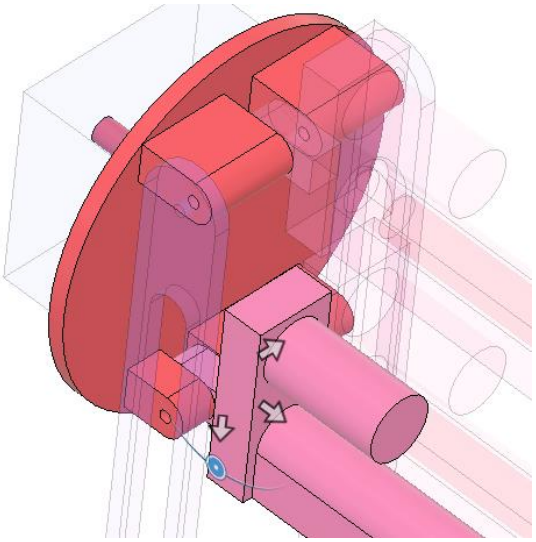


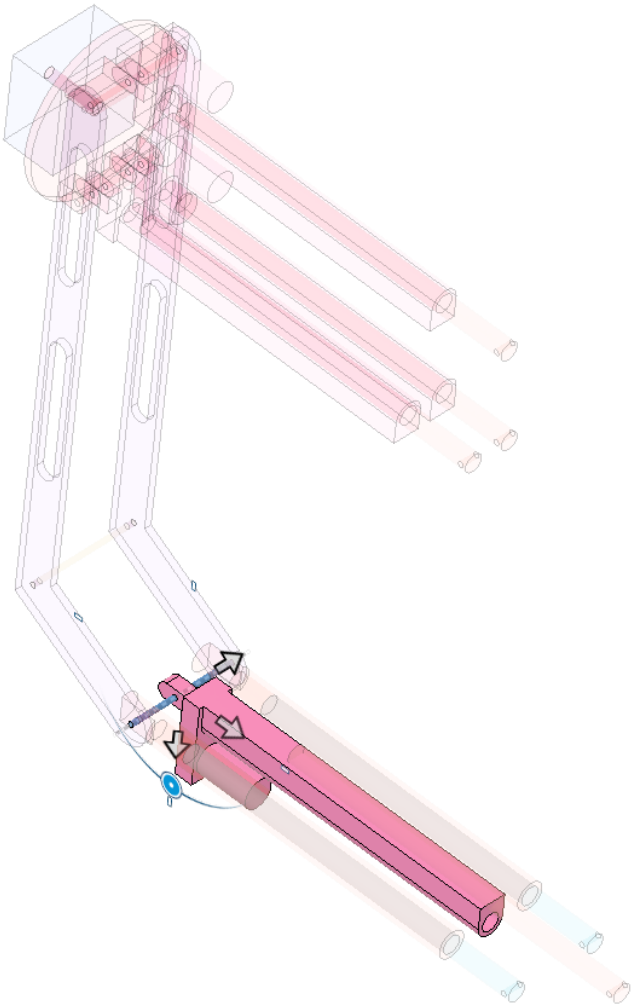

*ADDITIONAL ACCTUATOR ASSEMBLIES AFTER INSTALLATION (SELECTED)*

1. Activate the top level browser node.
2. Select the first actuator assembly in the browser.
3. *Copy and Paste* it to make another instance.
4. After paste, move the new instance to the side. A temporary location is okay for now, we'll use a joint to put it into it's permanent position.
5. Repeat step 4 so that you have 3 new actuator assemblies floating in space nearby, ready to be jointed to the overall assembly.

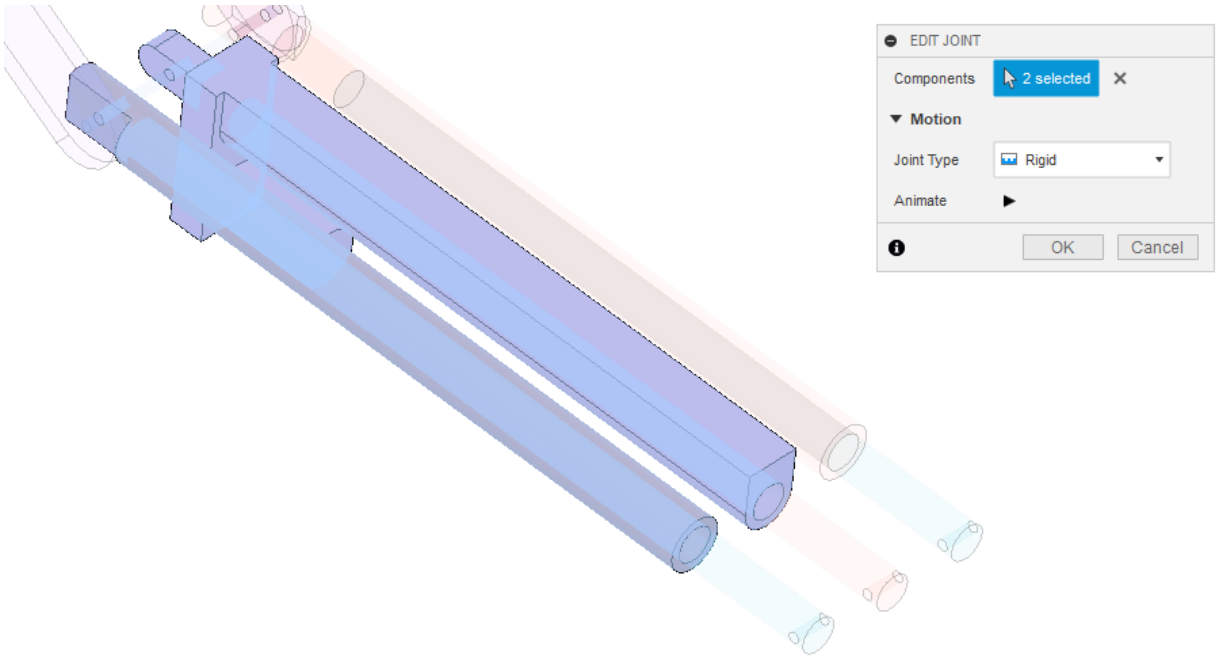

6. Use a revolute *Joint* to position the actuators, one by one into the positions shown in the images below:



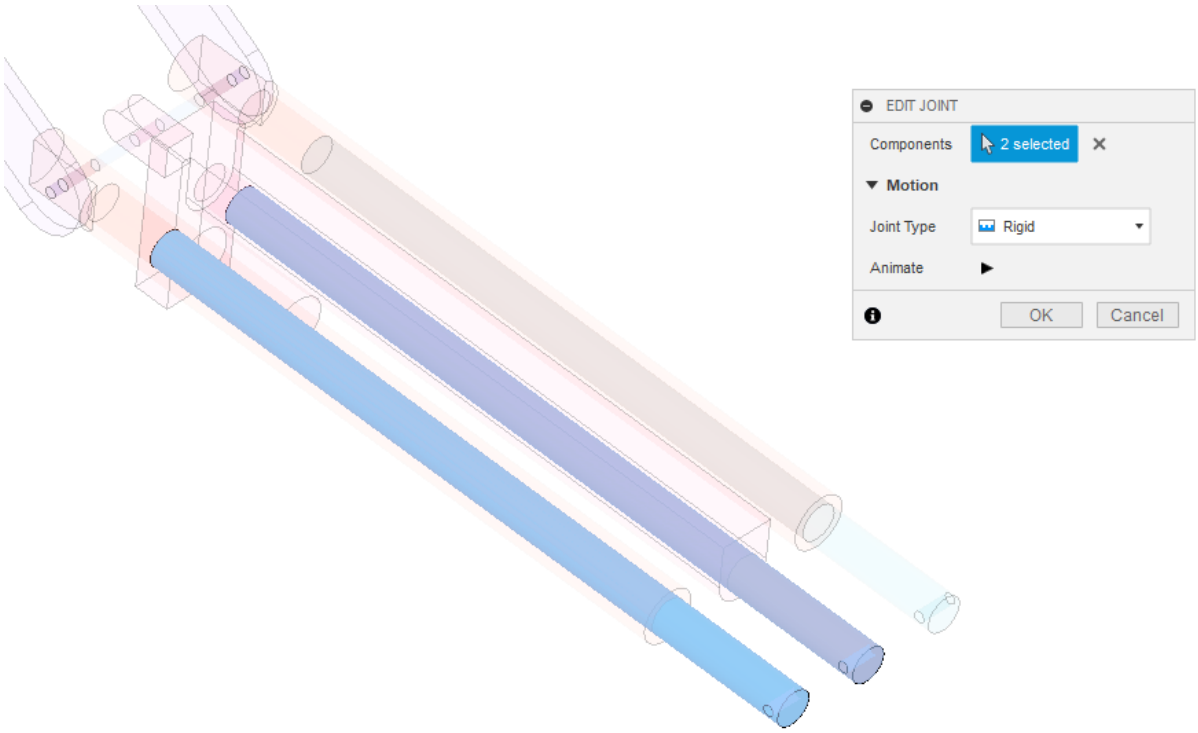




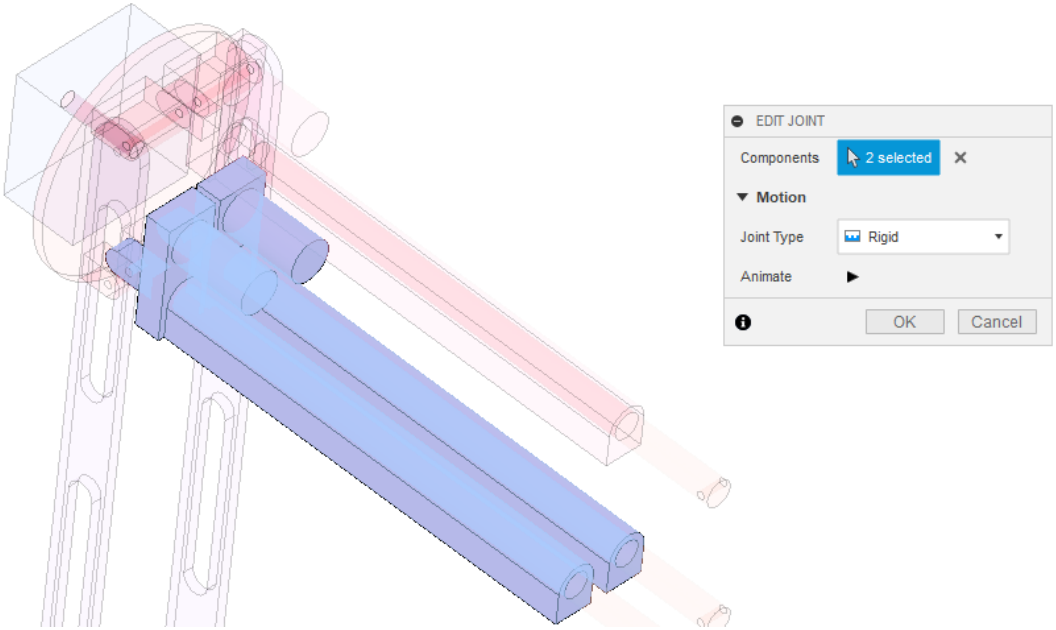
7. Apply an *As-Built* rigid joint to these two highlighted components. This will make the newly installed actuator assembly rotate in unison with the lower arm assembly:



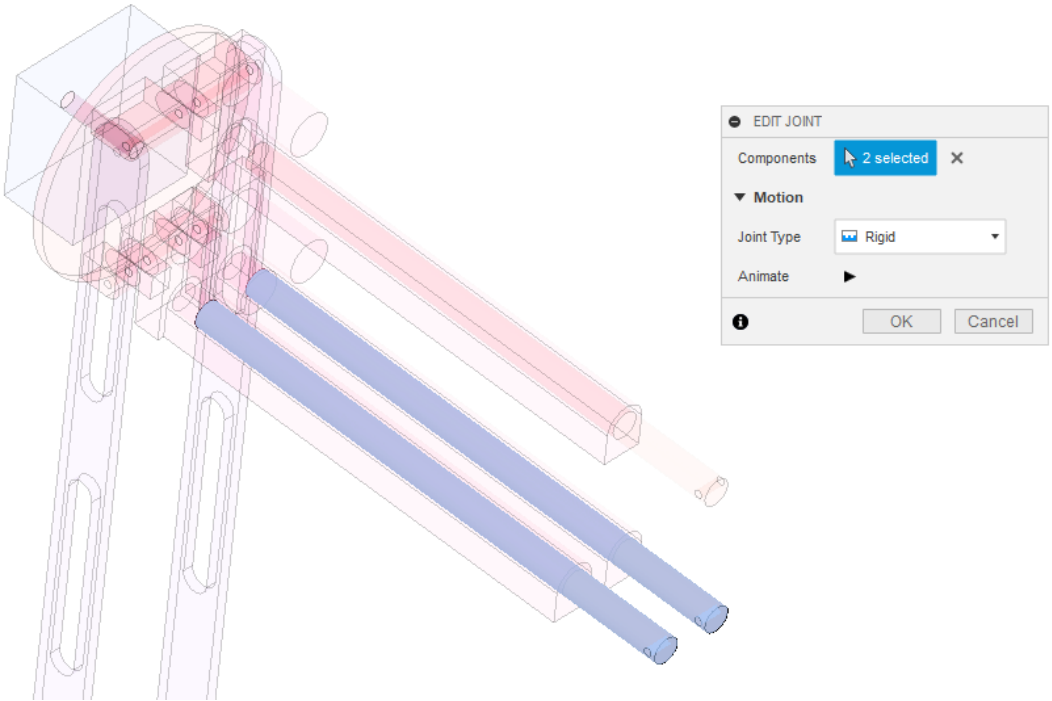

8. Repeat the process for the inner sliding parts. They will slide in/out in unison:



9. Now use a similar process to make the two new actuator assemblies attached to the mounting plate behave in unison. Start by adding an *As-Built* rigid joint between the two newly pasted actuator bodies:




10. Finish by adding an *As-Built* rigid joint between the two corresponding actuator shafts, so they move in/out in unison:



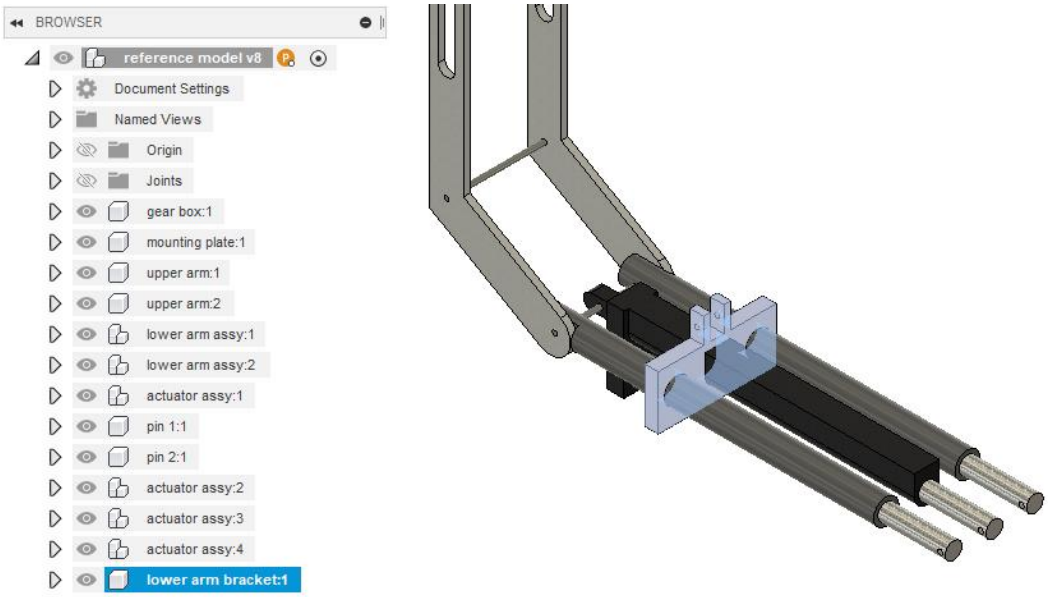
**Commands used:**

- Copy/Paste component
- Joint
- As-Built Joint

**Insights:**

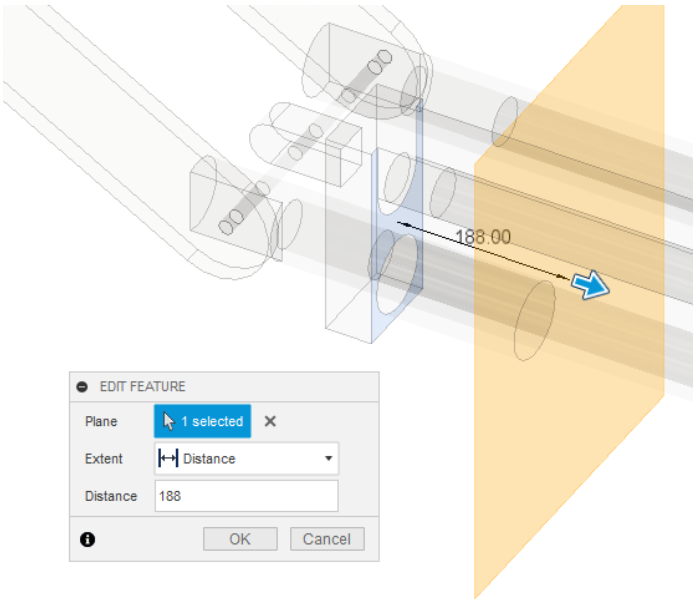
- Joints provide position and motion when you need to provide both.
- As-Built Joints provide only motion between components (or rigid relationships) when the relative position is already established.

## Step 8 – Lower arm bracket

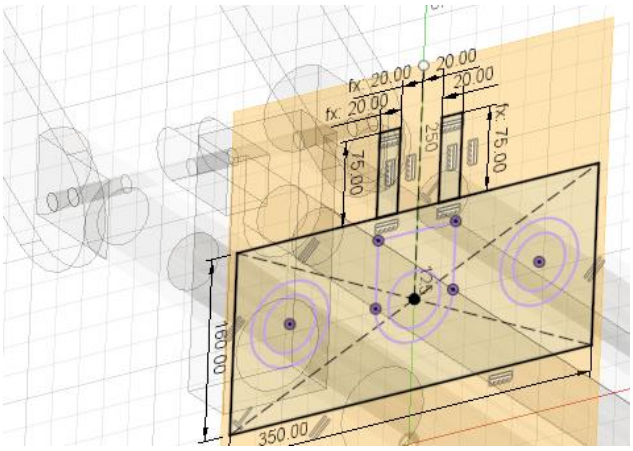



*LOWER ARM BRACKET (SELECTED)*

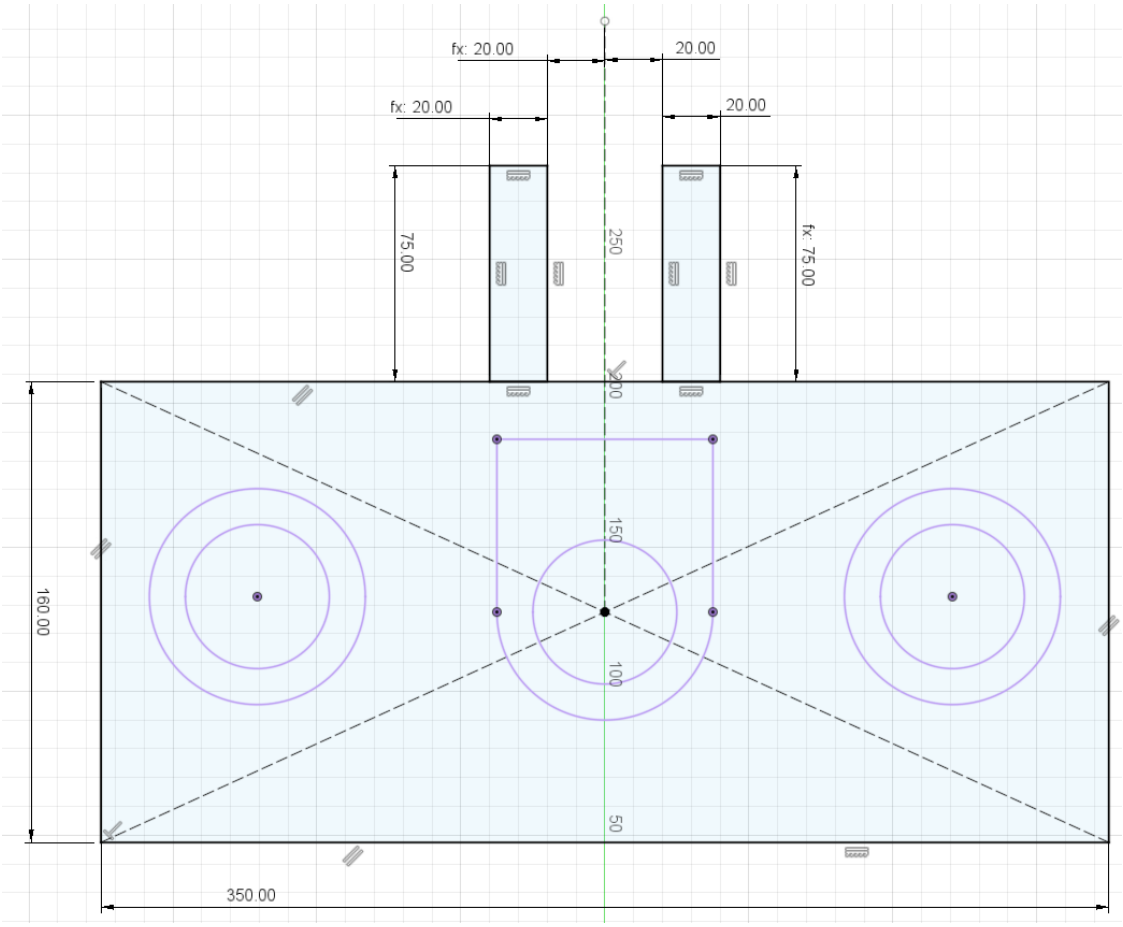
1. Activate the top level browser node.
2. Create > *New Component*, name it “lower arm bracket”. Activate it.
3. New command: *Construct* > *Offset Plane*, from the front of the actuator assembly on the lower portion of the assembly. Give it a value of 188 mm:



4. Create a *New Sketch* on the plane you just created.
5. Use Sketch > Create > Project / Include > *Intersect.*
6. Select the three bodies that intersect the sketch plane:

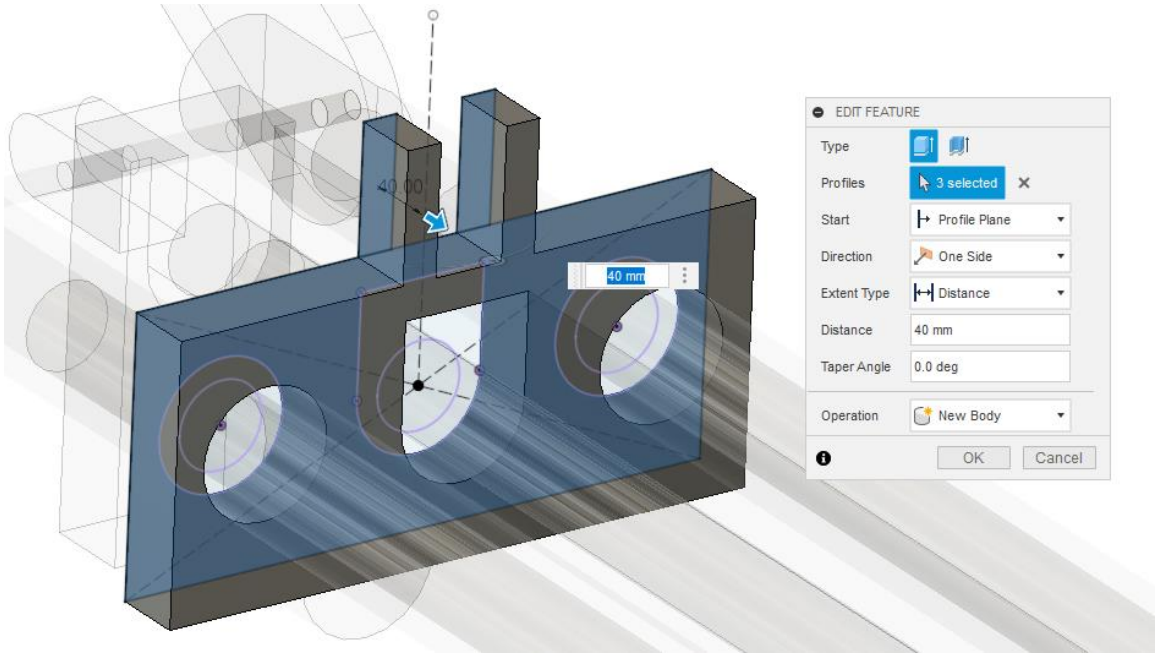



7. Finish the sketch with center rectangle, lines, and dimensions:

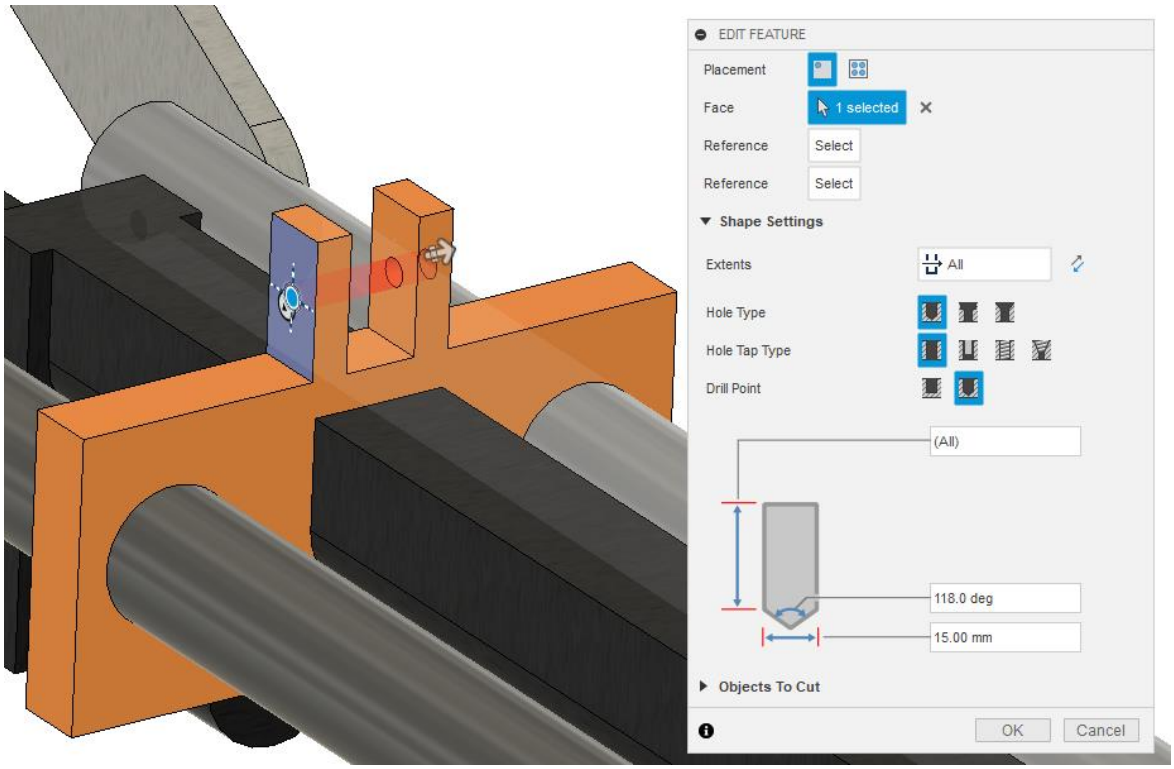



*LOWER ARM BRACKET SKETCH*

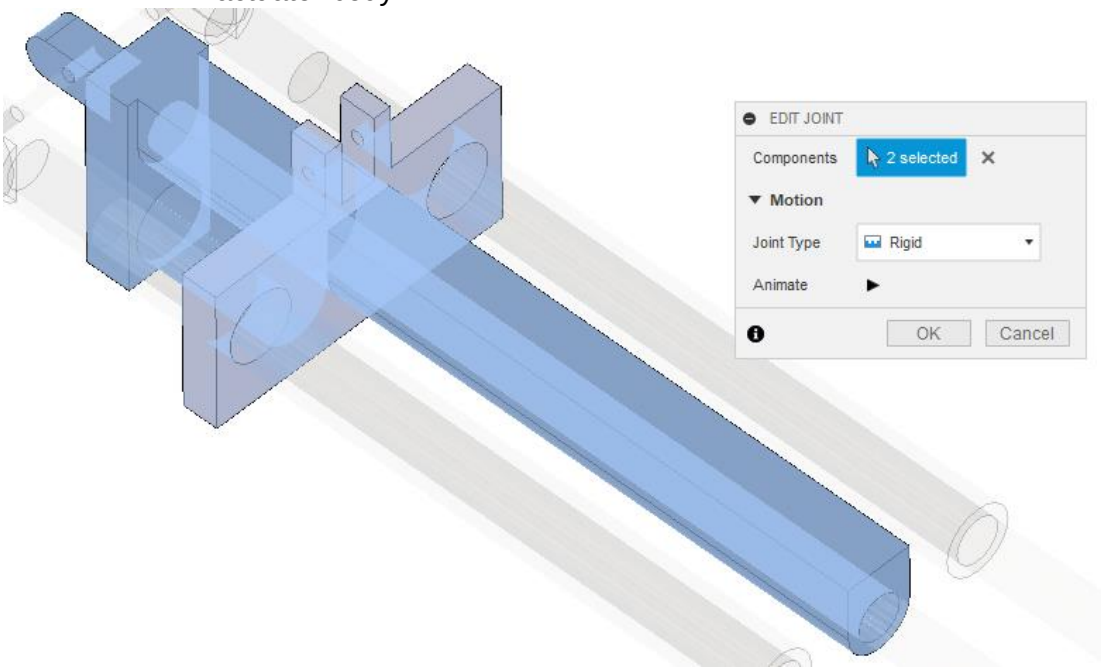
8. *Extrude* the sketch 40 mm:




9. Use the *Hole* command to create a Ø15 mm hole in the center of the face of the top tab, either side will do:



10. Make a rigid *As-Built Joint* between the lower arm bracket and the actuator body:



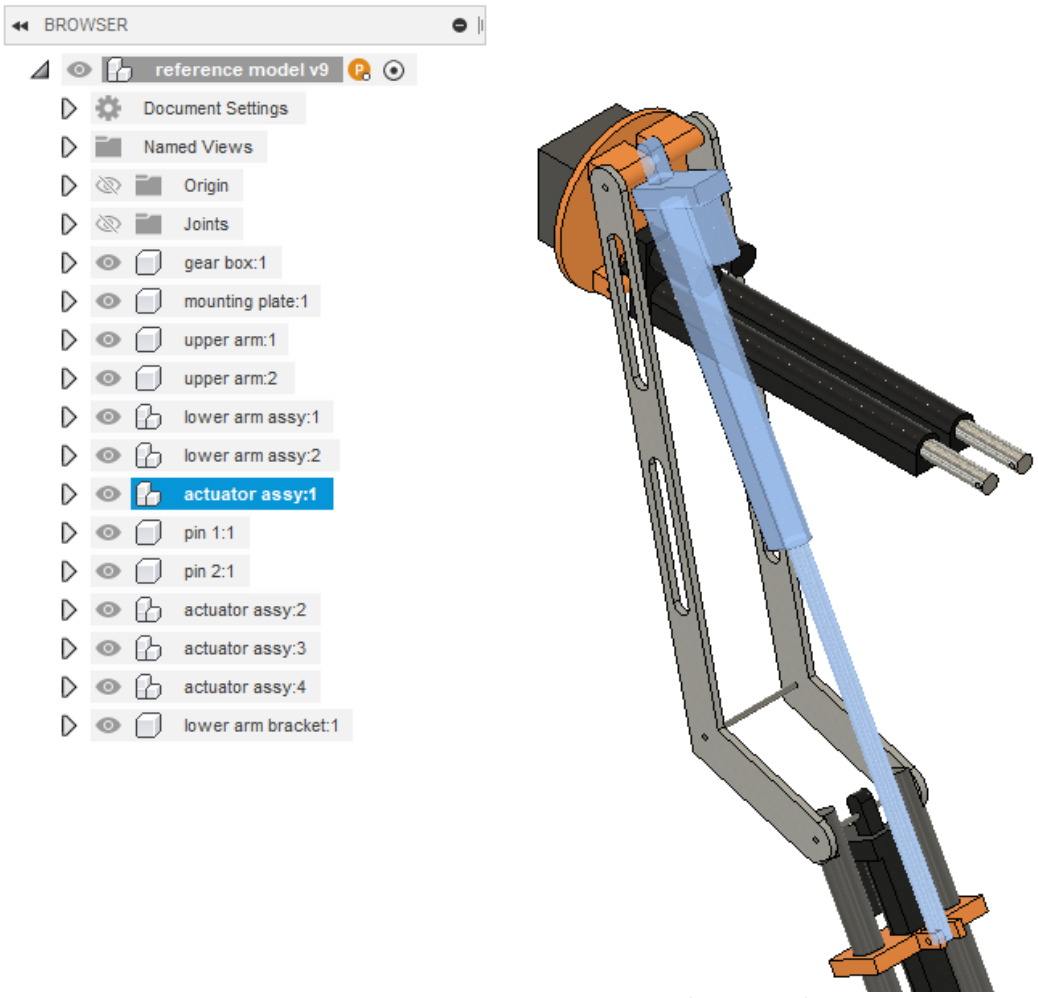

**Commands used:**

- New Component
- Offset Plane
- New Sketch
- Sketch project geometry
- Sketch rectangle
- Sketch line
- Sketch dimension
- Extrude
- As-Built Joint

**Insights:**

- An offset plane is the right way to put a sketch where there isn't any model surface or origin plane to sketch on.
- A rigid As-Built joint is the CAD equivalent of gluing or bolting a part to another.

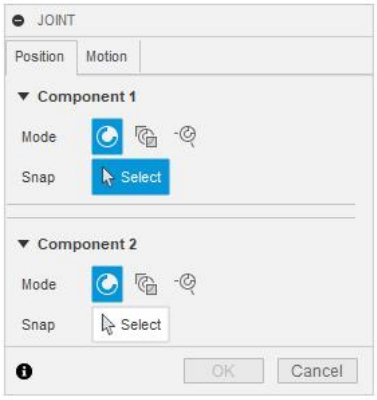
## Step 9 – Connect the top actuator assembly with a joint




*TOP ACTUATOR ASSEMBLY CONNECTED (SELECTED)*

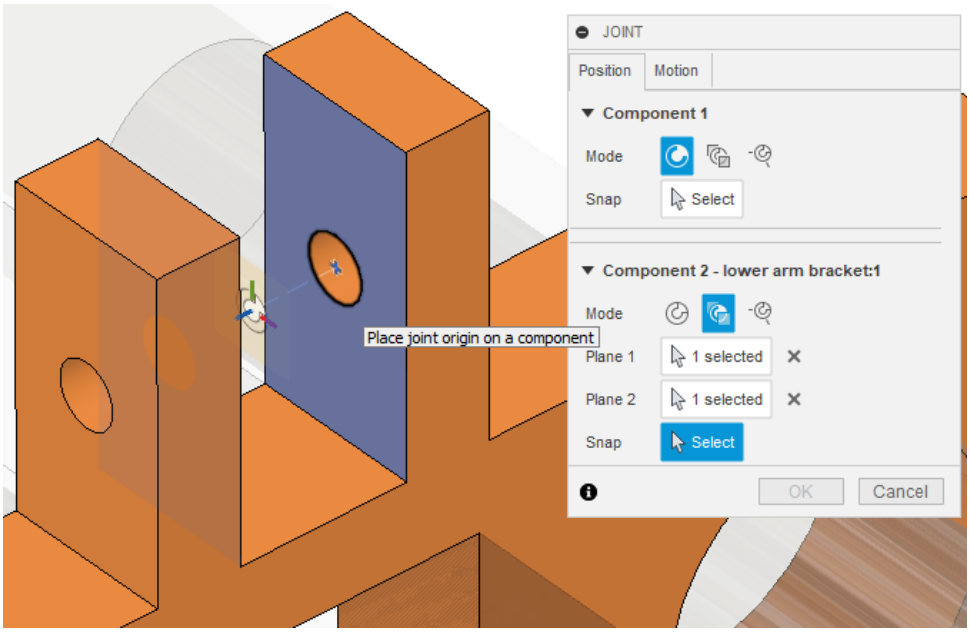
1. Activate the top level browser node.
2. This is not required, but it helps to drag the end of the actuator assembly towards the lower arm bracket.
    a. Start from a view similar to the image above step 1.
    b. Click and drag the end of the actuator shaft down towards the lower arm bracket.
    c. Stop when just above the location of the joint you're about to build.

3. Zoom into the area.
4. Start the *Joint* command.
    a. On the Position tab, the first snap for Component 1 requires getting the mouse onto the cylinder face of the hole, and then pressing ctrl/command to isolate the selectable points. This allows you to pick the middle point triangle icon:

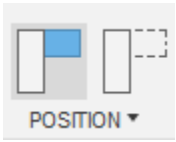




    b. The second snap, for Component 2, is made using the Between Two Faces mode.
    c. Change the mode to Between Two Faces.

d. Select the two faces shown highlighted in blue below for Plane 1 and Plane 2. This will require you to orbit to make the selections.
e. Select the Snap point using the edge of one of the holes.
f. Click OK.




5. Capture the position:



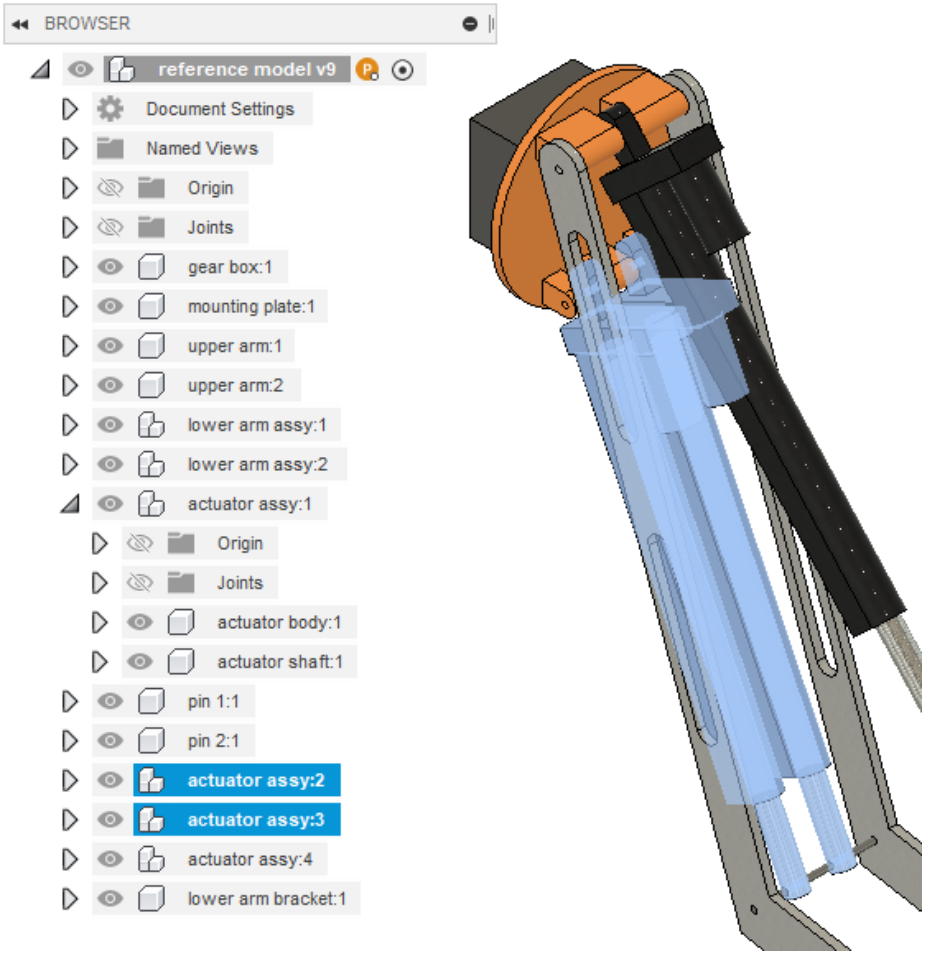

**Commands used:**

- Joint command
- Capture position

**Insights:**

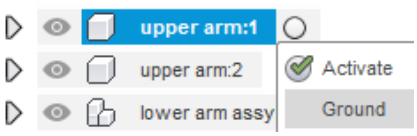
- Similar to the many Extrude command dialog options, you can see in this workflow that there are many options to explore in the Joint command dialog when it comes to connecting geometry on the Position tab.
- *Capture Position* is not on the 12 commands list because it's a passively activated command. Fusion will force you to decide to capture a position when a component has been moved and you start a parametric command. For more information about *Capture Position* please refer to this help page.

## Step 10 – Connect the lower actuator assemblies with a joint

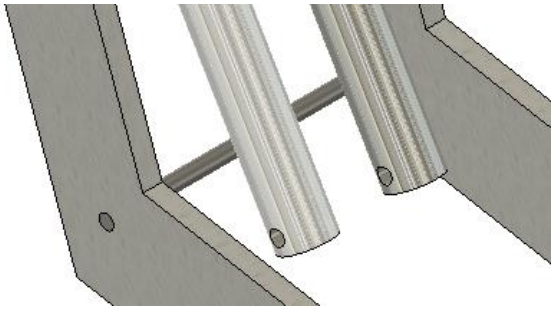



*LOWER ACTUATOR ASSEMBLIES CONNECTED (SELECTED)*

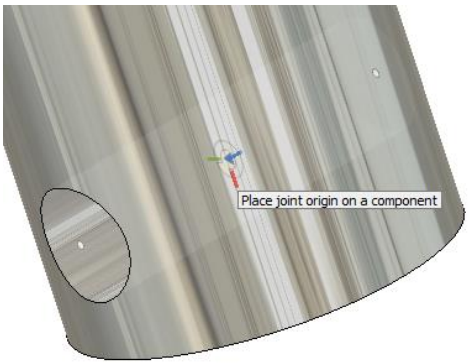
1. Activate the top level browser node.
2. The next step benefits from temporarily grounding the upper arm. *Ground* is not on the 12 commands list because it is not absolutely required. It's called out here as the kind of next level command you would discover by doing these joint assembly workflows. *Ground* works like a clamp to lock a part in place and prevent movement. It becomes a timeline event.
    a. Right click on the upper arm component and pick "*Ground*":

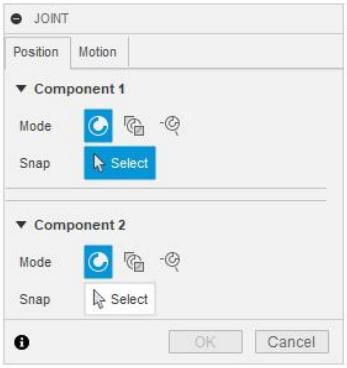


3. Following a similar workflow as before, using the mouse pointer and left mouse button, drag the end of the actuator assemblies towards the pin:

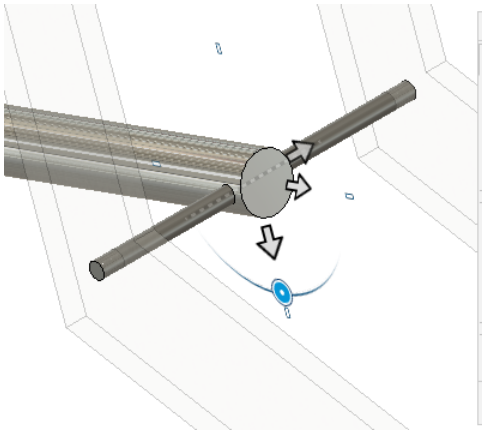


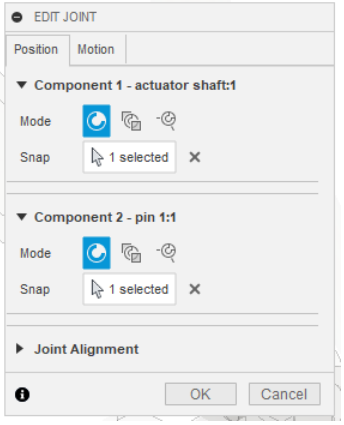
4. Zoom into the area. Focus on just one of the ends, either will do the job.
5. Start the *Joint* command.
    a. On the Position tab, the first snap for Component 1 again requires getting the mouse onto the cylinder face of the hole, and then pressing ctrl/command to isolate the selectable points. This allows you to pick the middle point triangle icon:



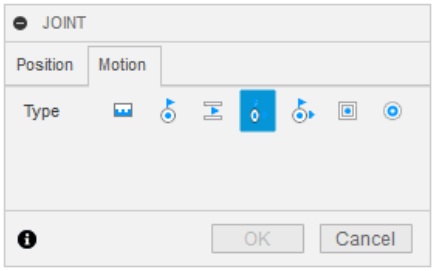




    b. The second snap, for Component 2, is made using Simple mode. Hover the mouse over the mid point of the pin and select the triangle icon when it appears:





c. On the Motion tab, change the Motion type to Cylindrical:




6. Press OK.
7. Right click on the upper arm component and use "*Unground*" to release the temporary grounded state. The timeline shows this sequence when you are done:

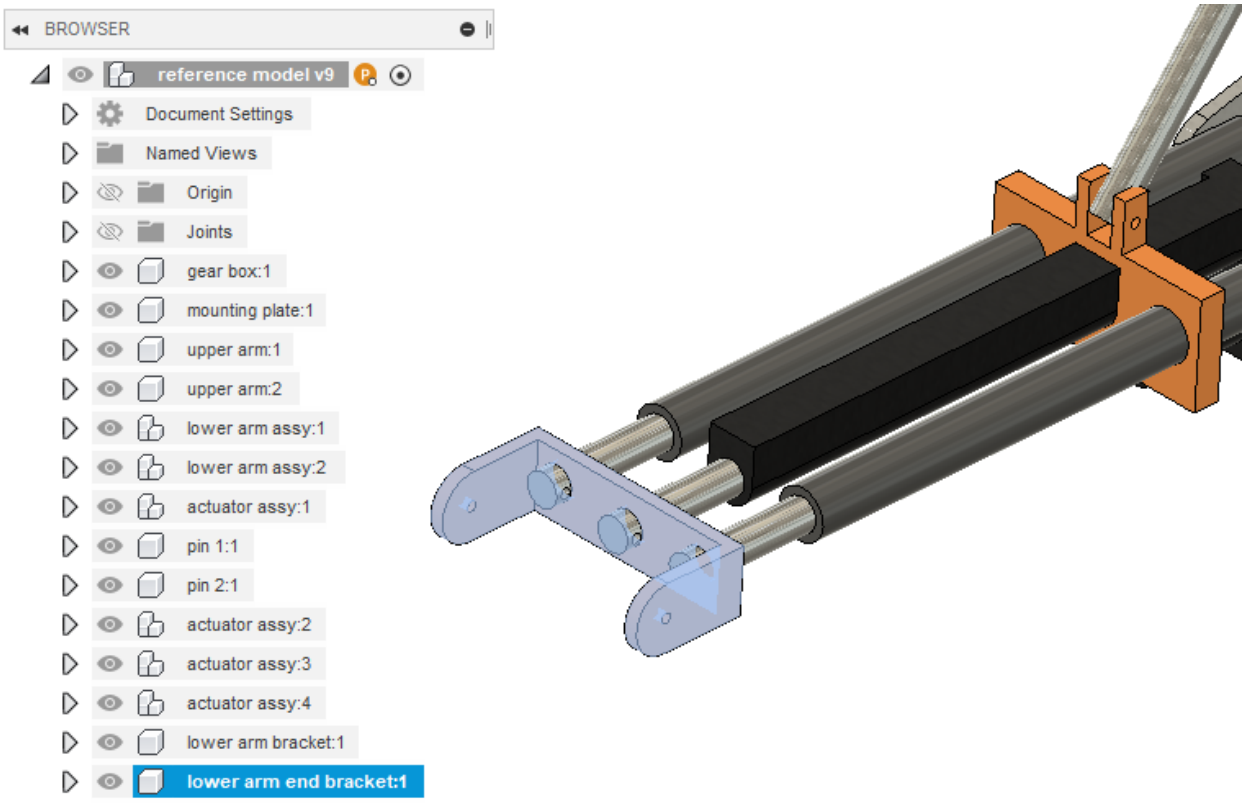
**Commands used:**

- Joint command

**Insights:**

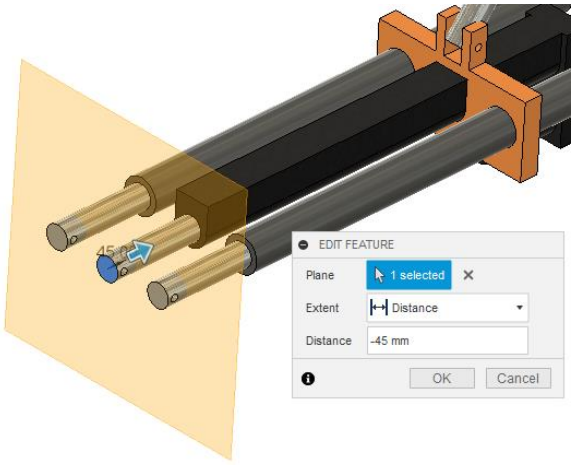
- Right clicking on browser objects, such as components, to find special commands that apply only to that object, such as ground/unground.

## Step 11 – Create the lower arm end bracket

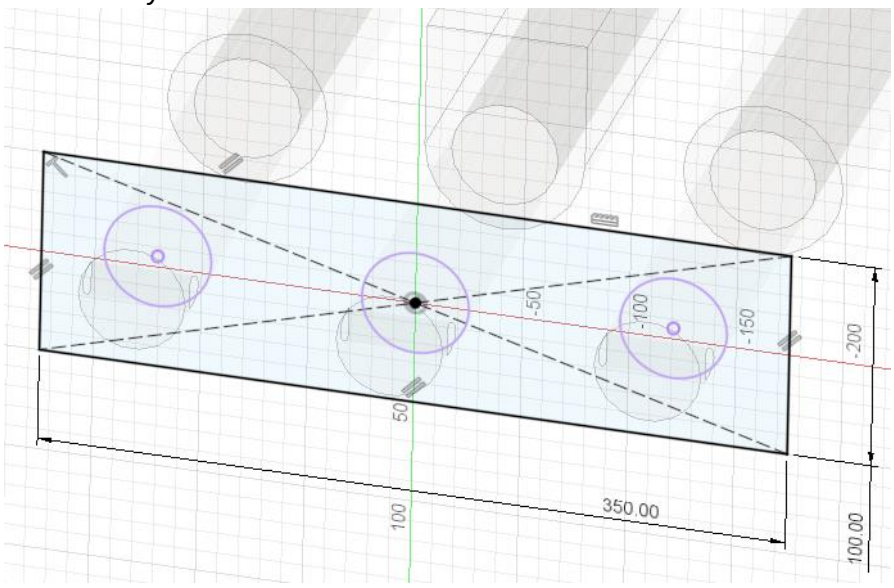



*LOWER ARM END BRACKET (SELECTED)*

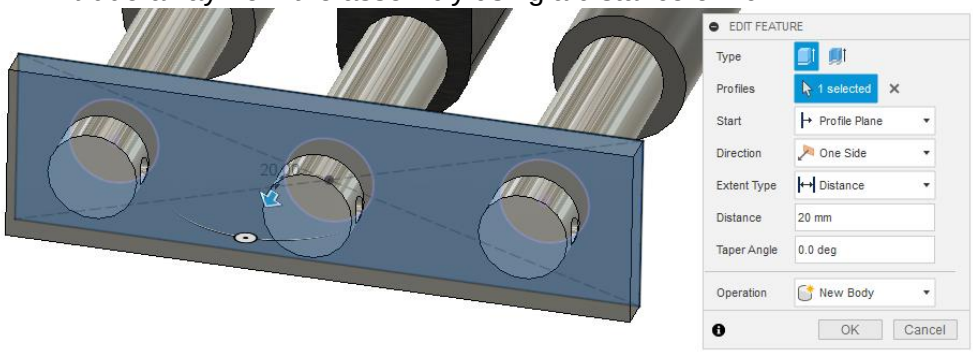
1. Activate the top level browser node.
2. Create > *New Component*, “lower arm end bracket”, activate it.
3. Construct > *Offset Plane*, from the end of the center actuator shaft. The distance is -45 mm:



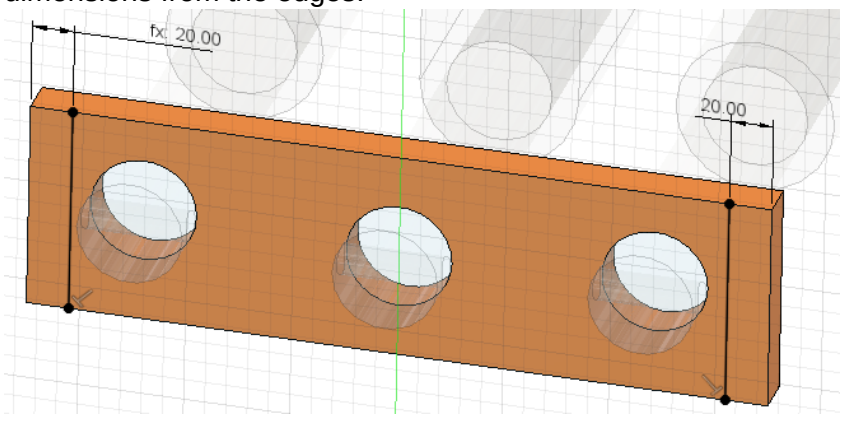
4. Similar to the lower arm bracket, sketch on this plane.
5. Use Create > Project/Include > *Intersect* to project the cross section of the parts that intersect the sketch plane. These will be purple shapes.
6. Create a *Center Rectangle*, centered on the middle projected circle. 350 mm wide by 100 mm tall:



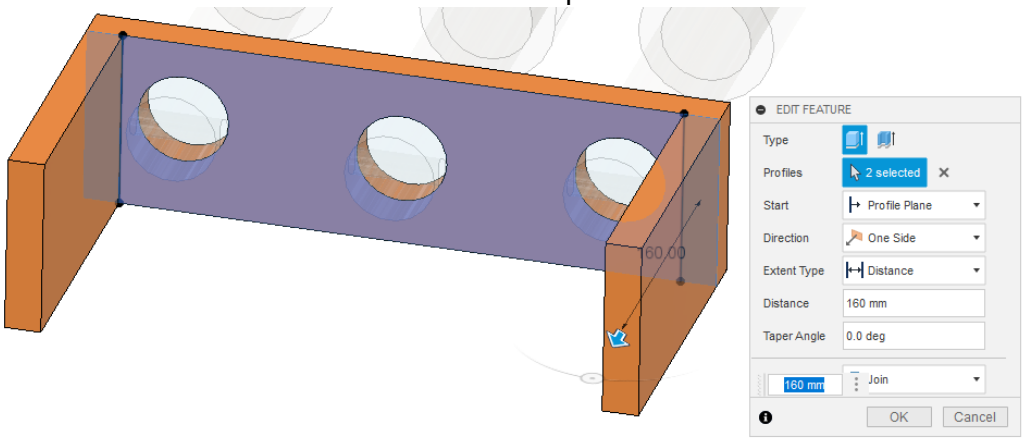

7. *Extrude* away from the assembly using a distance of 20mm:



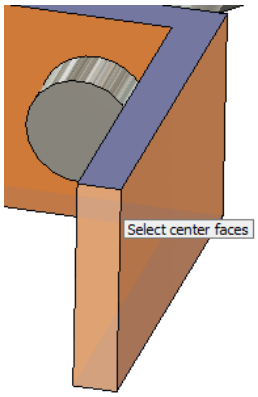

8. *New Sketch* on the resulting face and add two lines as shown, with 20mm dimensions from the edges:



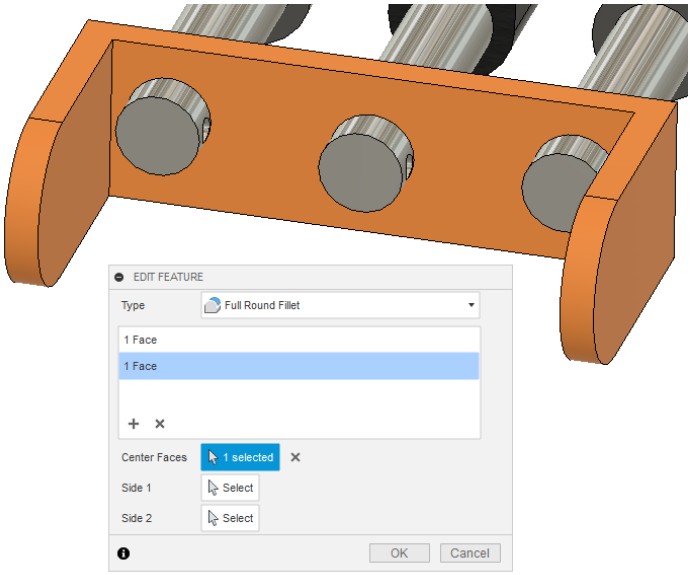
9. Finish the sketch and *Extrude* the shapes 160 mm:



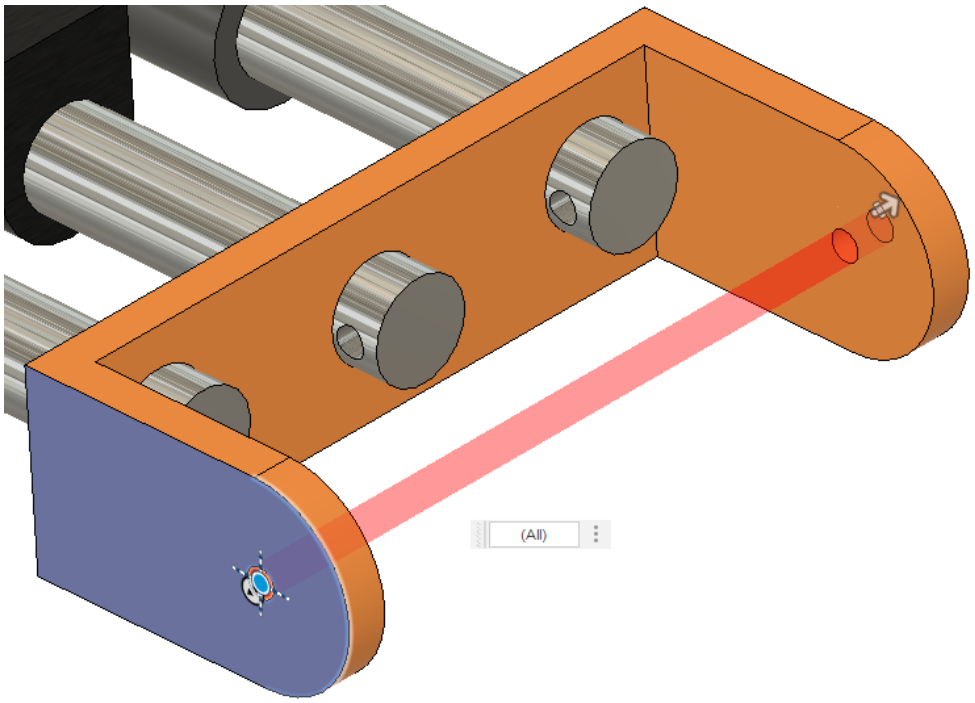

10. Add a *Full Round Fillet* to the ends of the shapes you just extruded. Here is where you get to practice making full round fillets, the key is where you select on the face you wish to be rounded:



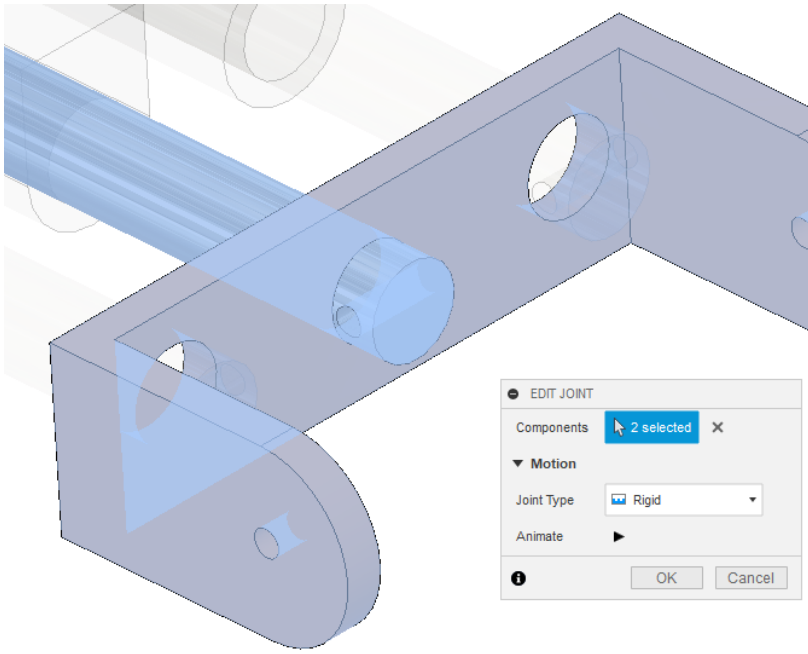

11. This is the intended result:



12. Create another Ø15 mm hole, on this face, and drag the center point to be co-located at the center of the full round fillet arc:




13. Create a rigid *As-Built Joint* between the lower arm end bracket and the center actuator shaft:



14. Click OK, you're done with this part!

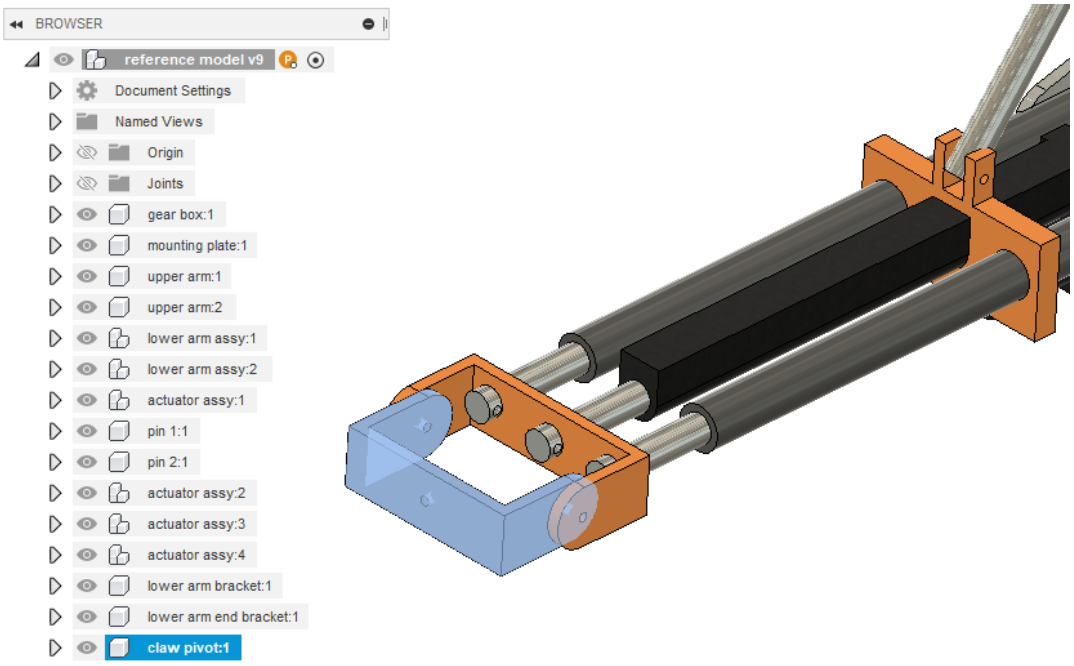
**Commands used:**

- New component
- Offset plane
- New sketch
- Sketch project geometry
- Sketch center rectangle
- Sketch line
- Sketch dimension
- Extrude
- Hole
- As-Built joint

**Insights:**

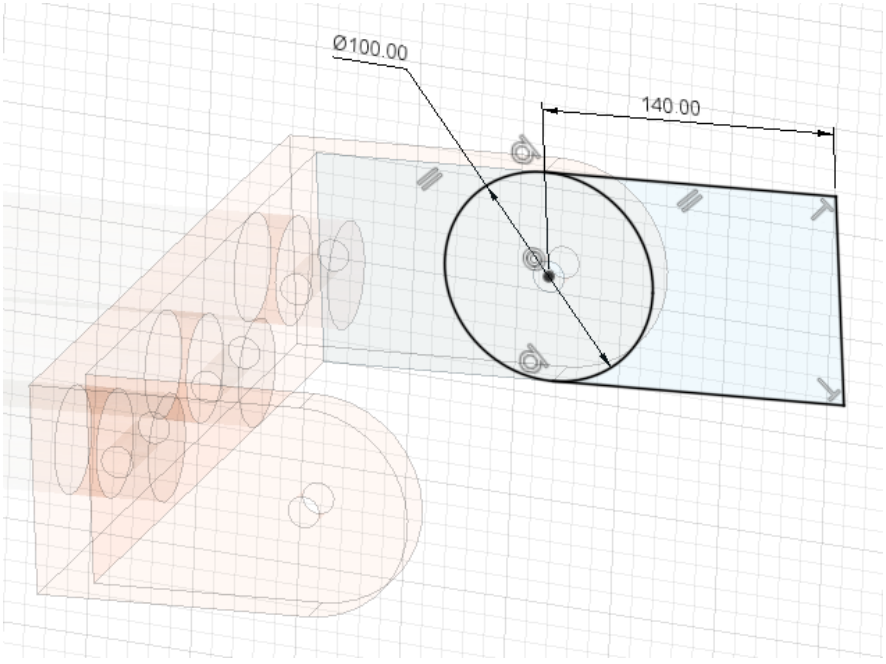
- Modeling in place is supported by offset planes and sketch projections. These help place sketches where you need them and make parametric relationships between the parts in your design.

## Step 12 – Claw Pivot

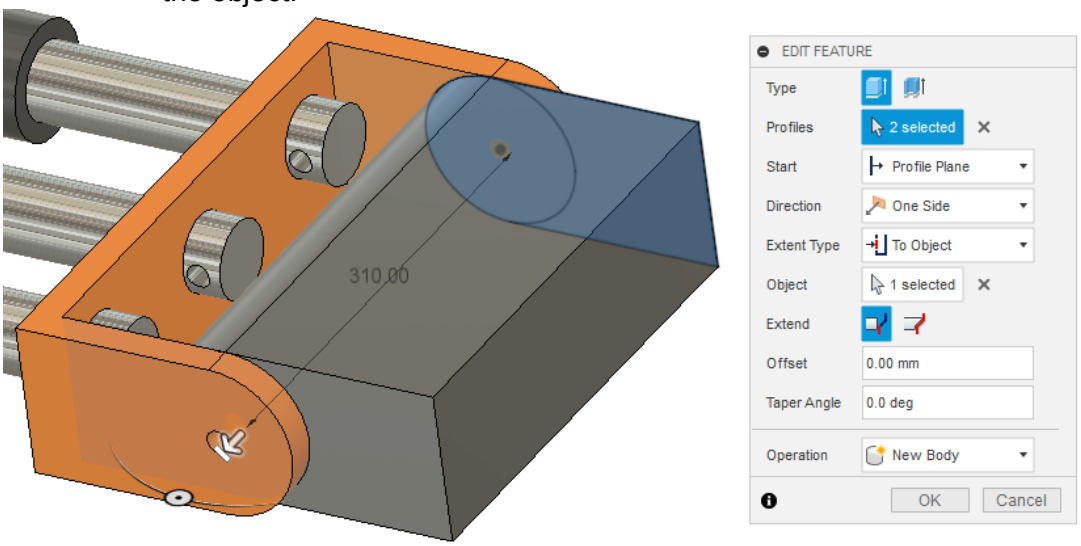



*CLAW PIVOT (SELECTED)*

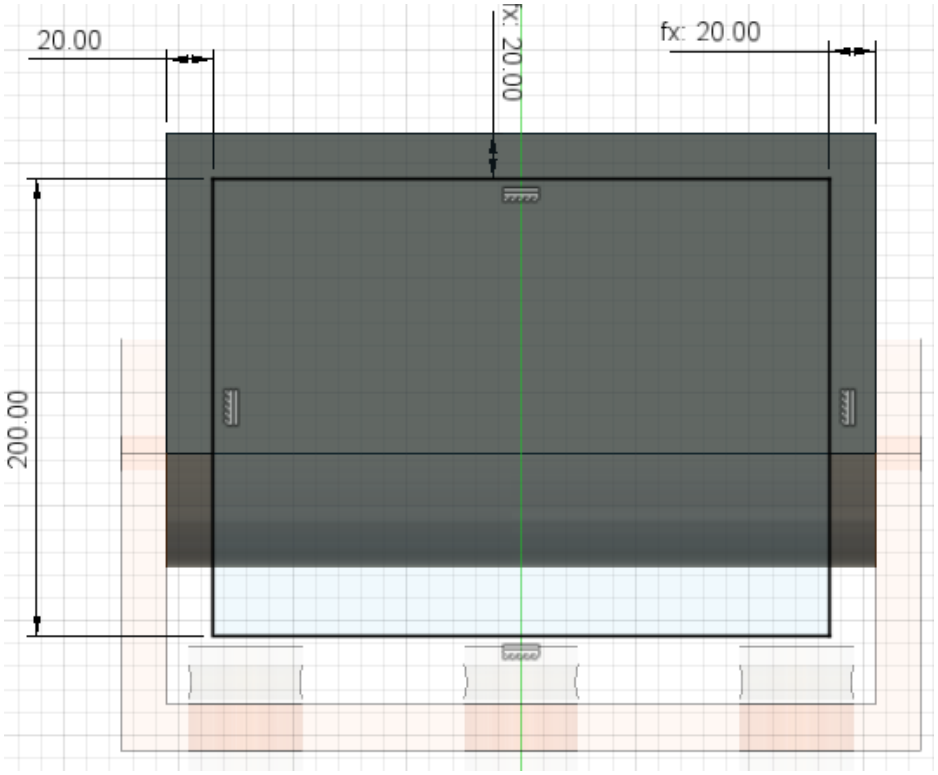
1. Activate the top level browser node.
2. Create > *New Component*, “claw pivot”, activate it.
3. *New Sketch* on the inside face of the lower arm end bracket:



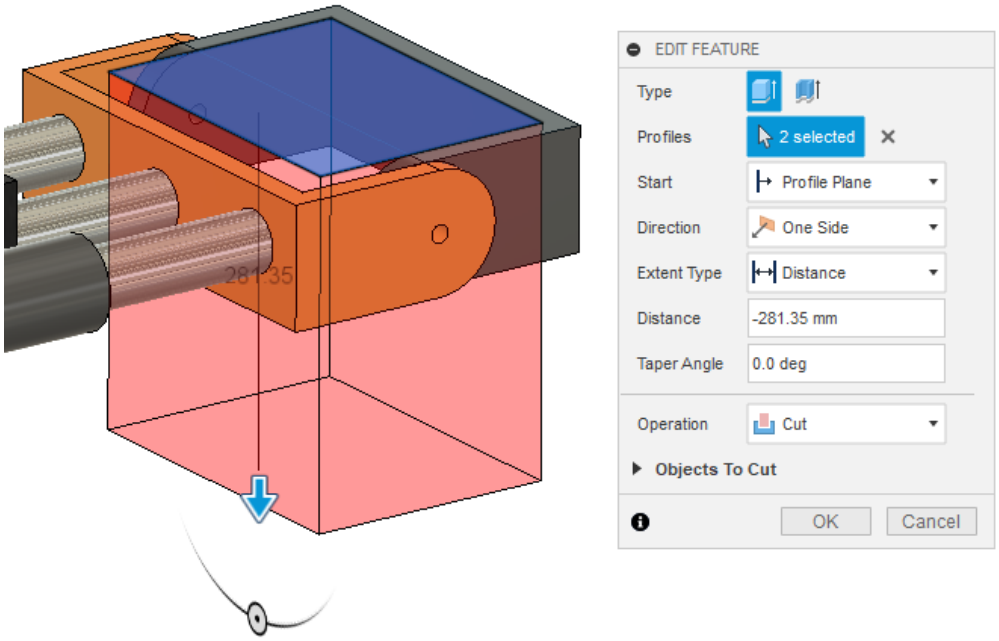
4. *Extrude* the two profiles created in the sketch, using Extent Type: To Object. Select the opposite inside face of the lower arm end bracket as the object:



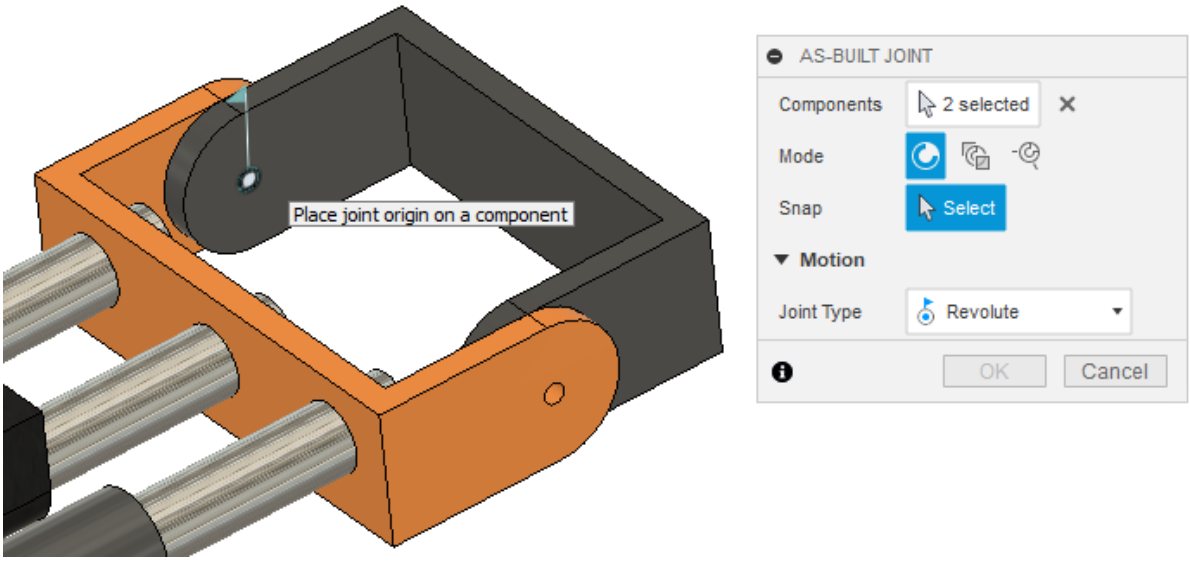

5. Create a *New Sketch* on top of the new body. Add a *2-point Rectangle* and dimensions to create this shape:



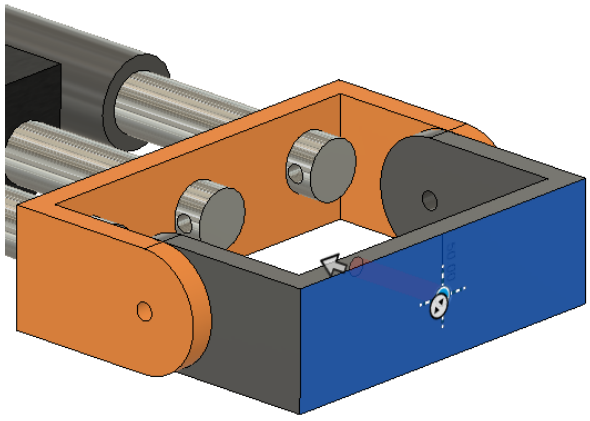
6. *Extrude* the rectangle as a cut through the body:

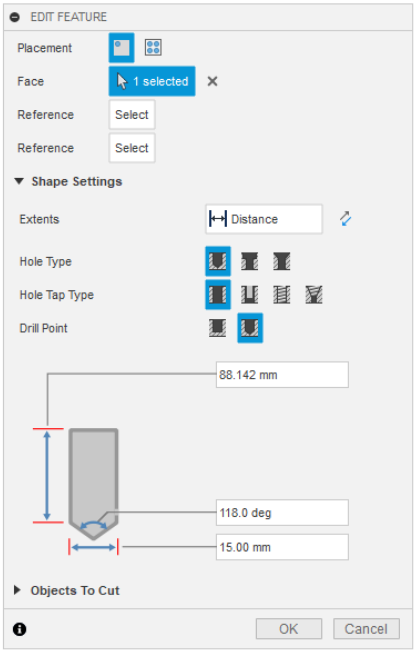



7. Add an *As-Built* revolute joint between the lower arm end bracket and claw pivot:




8. Add a Ø15 mm *Hole* to the center of the outer face on the claw pivot. Remember the method to drag the hole center point to enable the center snap point on the face. When a small white dot appears in the center of the face, drag the hole center to it and observe the snapping action:





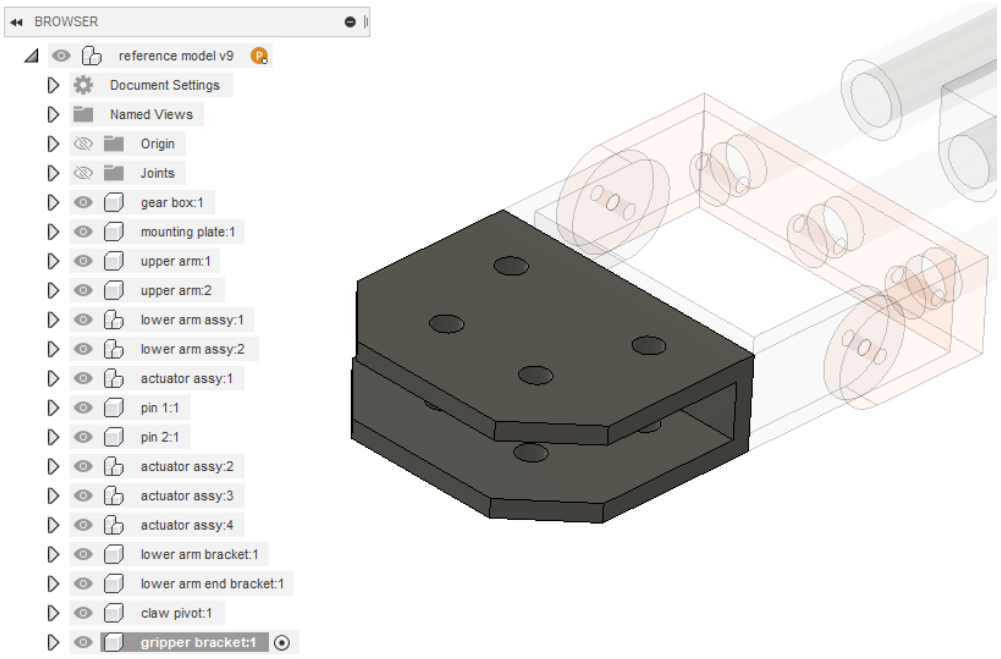

**Commands used:**

- New component
- New sketch
- Sketch circle
- Sketch line
- Sketch dimension
- Extrude
- Hole
- As-Built joint

**Insights:**

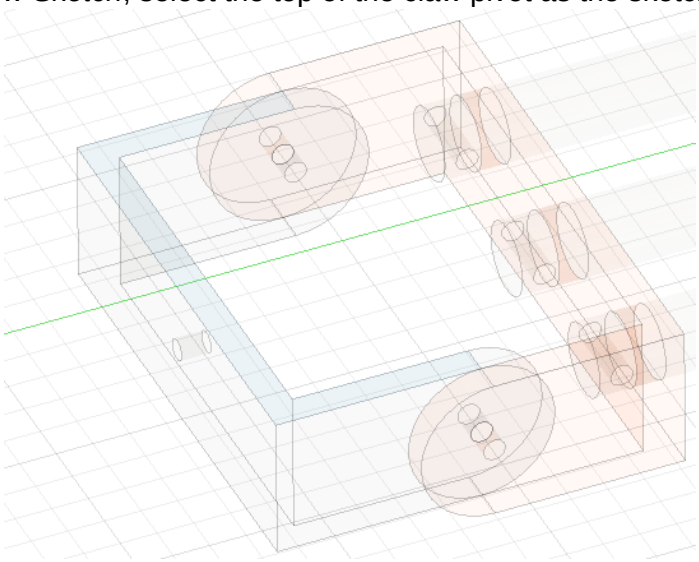
- For extruded parts, placing the sketch plane on another component surface and terminating the extrude using To Object will create a parametric relationship. In this case, if the lower arm end bracket changes size, the claw pivot will adjust to fit inside of it, by design. All you had to do is pick the start and end conditions for the extrude by simple modeling techniques.

## Step 13 – Gripper Bracket

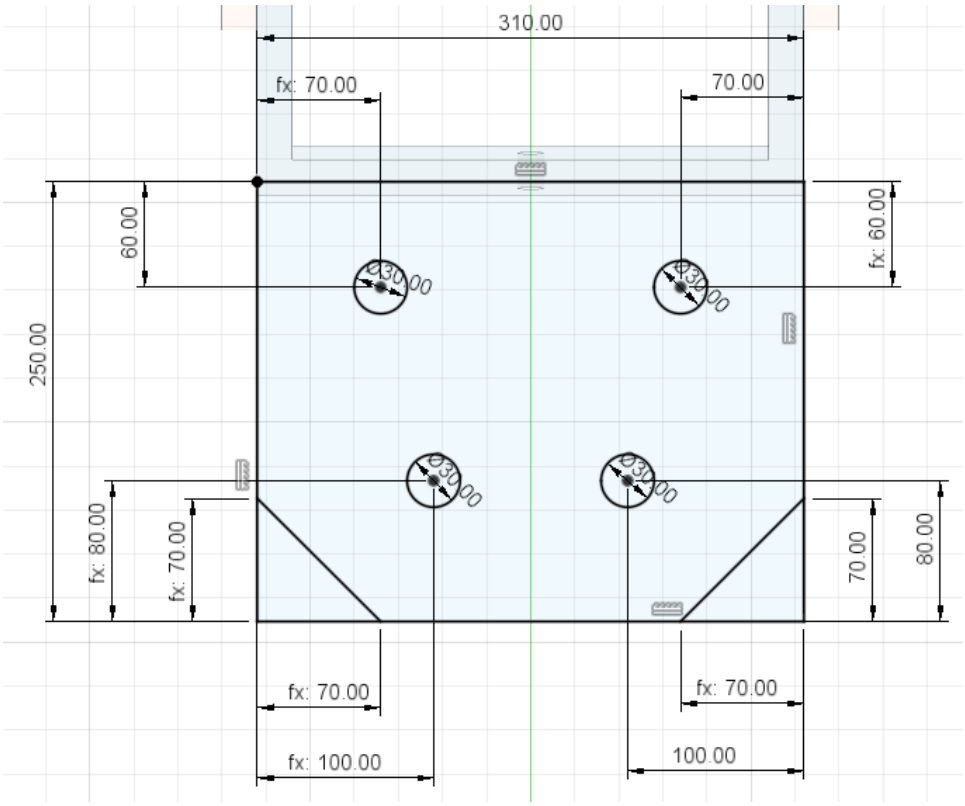



*GRIPPER BRACKET (ACTIVATED)*

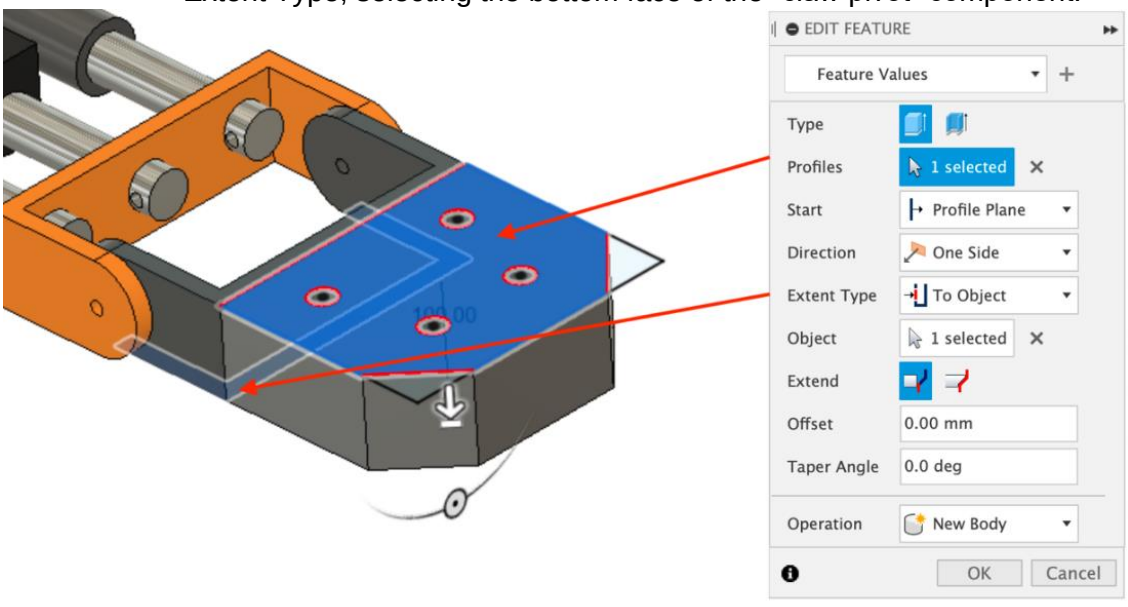
1. Activate the top level browser node.
2. Create > *New Component*, “gripper bracket”, activate it.
3. *New Sketch*, select the top of the claw pivot as the sketch location:



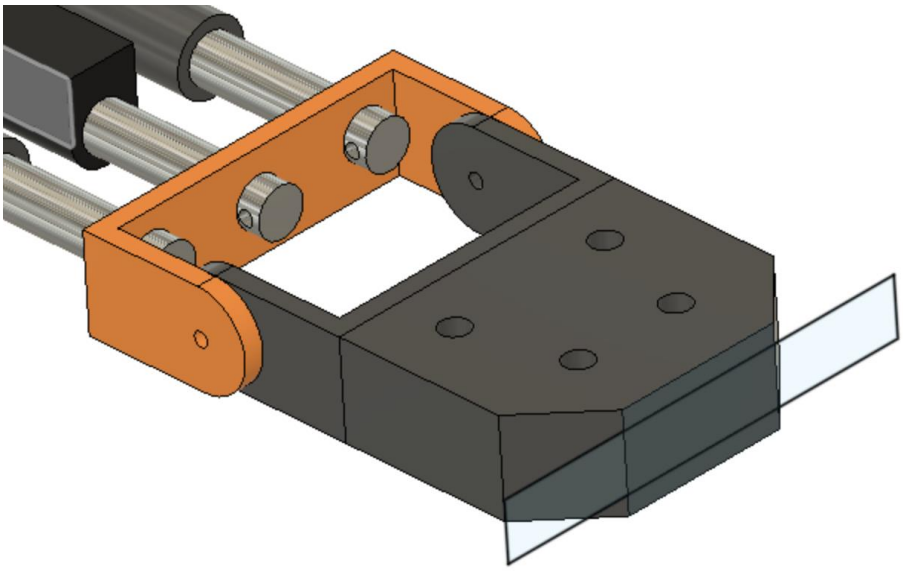
4. Using *2-point Rectangle*, lines, circles, and dimensions, create this figure in the sketch:

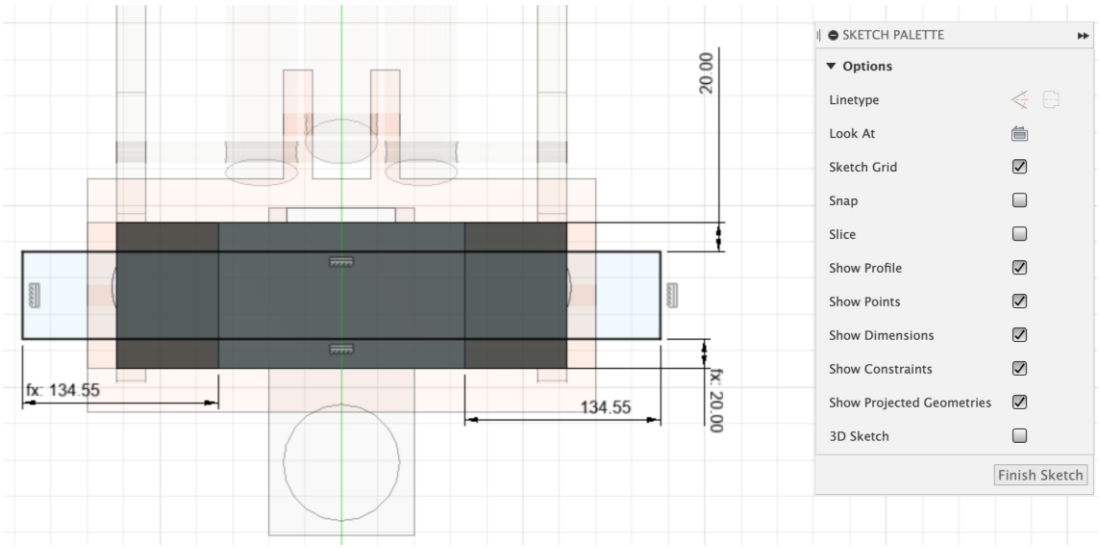



5. *Extrude* the inner profile defined by this sketch. Use “To Object” as the Extent Type, selecting the bottom face of the “claw pivot” component:

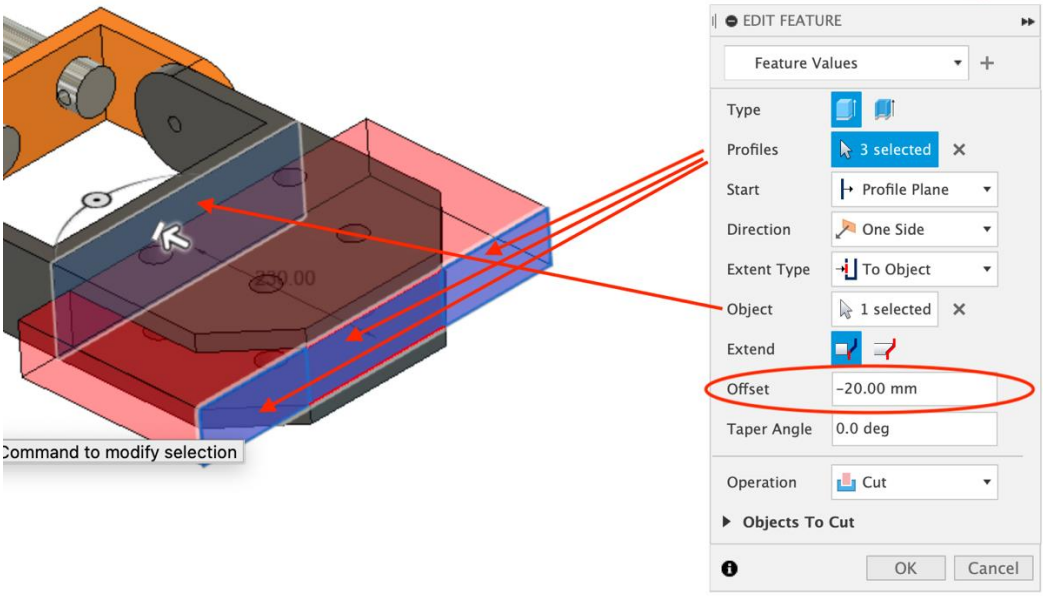


6. We will need to add a cut to this bracket, to allow room for the gripper claws. Create a *New Sketch* on the end face of the extruded body, and draw a *2-Point Rectangle* that is bigger than the body:

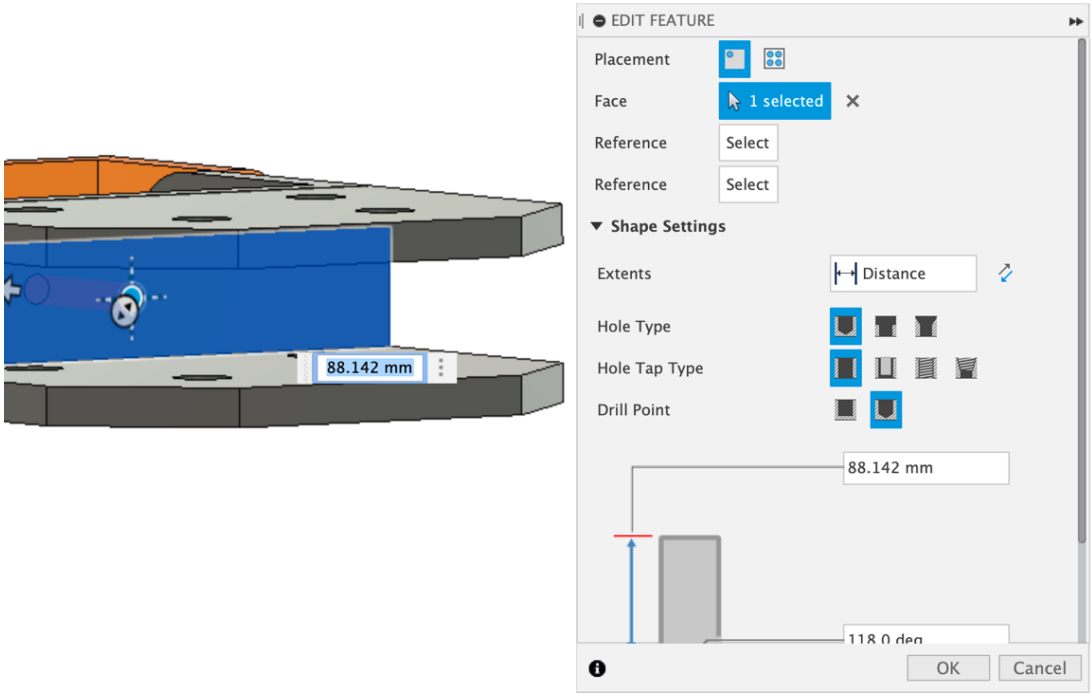




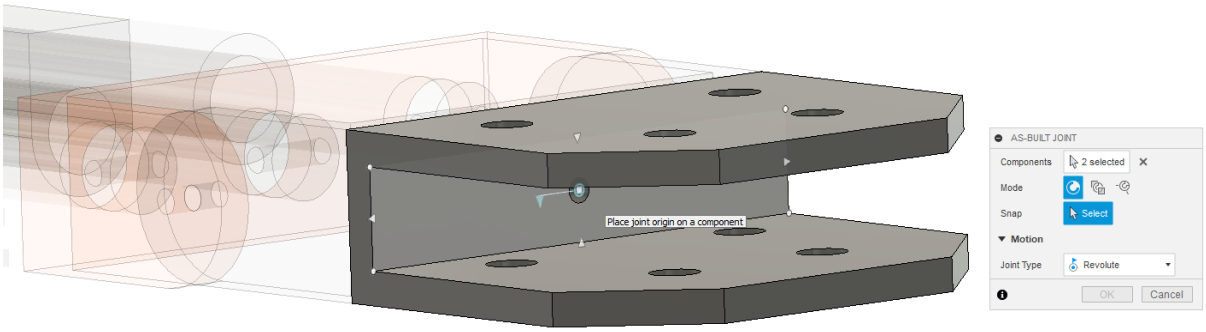

7. Create a Cut *Extrude* feature, selecting all 3 profiles from the sketch. Use To Object for the Extent, and use an Offset of -20 mm to guarantee material thickness on the back side of the bracket:




8. Add a Ø15 mm *Hole* to the back of the bracket, in the center of that face:




9. Add an *As-Built Joint*, revolute, between the new “gripper bracket”, and the previous “claw pivot” component, selecting the Hole as the Snap:



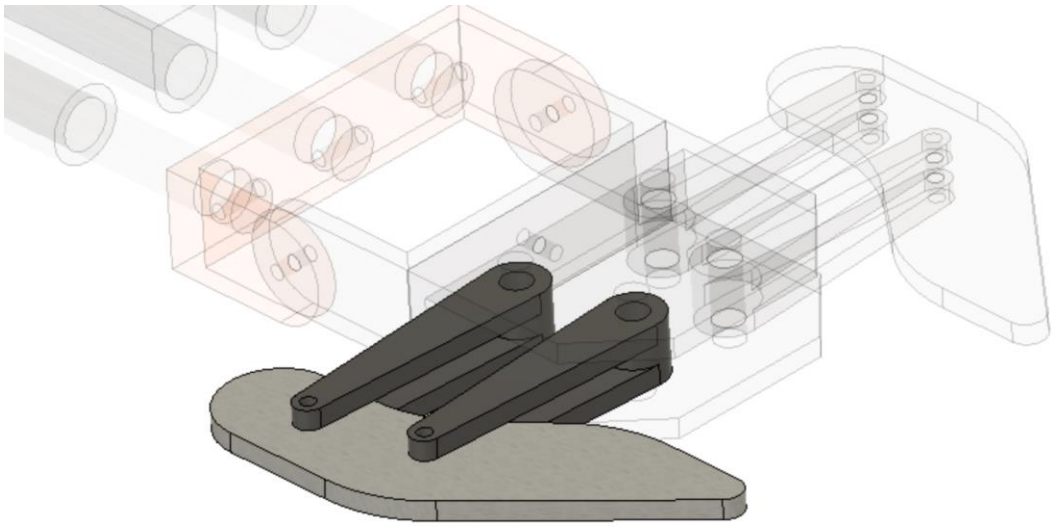

**Commands used:**

- New component
- New sketch
- Sketch circle
- Sketch line
- Sketch dimension
- Extrude
- Hole
- As-Built joint

**Insights:**

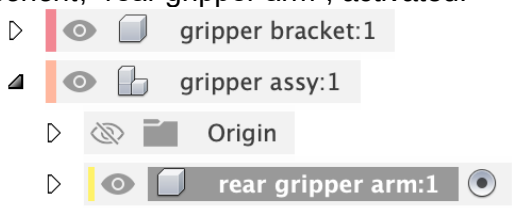
- When using *Extrude* to cut out some geometry in a body, use To Object + an Offset value to maintain material on that side of the body. Extending your understanding of each command starts by exploring and practicing the options in each command dialog box.

## Step 14 – Gripper Assembly

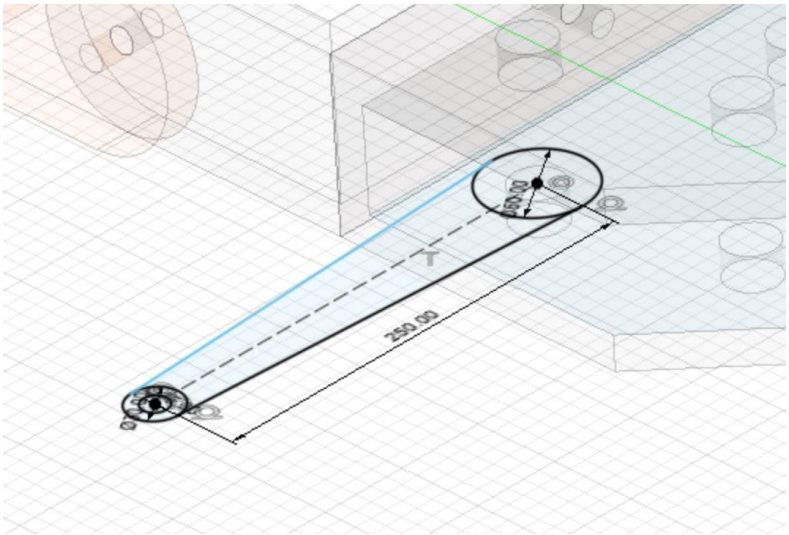


*GRIPPER ASSEMBLY (ACTIVATED)*

1. Create > *New Component,* “gripper assembly”, activated.
2. Create > *New Component*, “rear gripper arm”, activated:

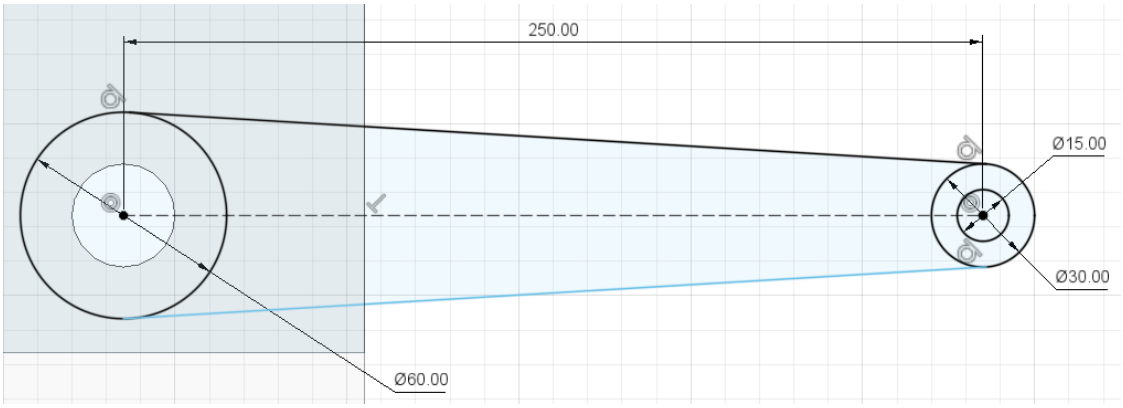



3. Create > *New Sketch* on the lower inside face of the “gripper bracket”:

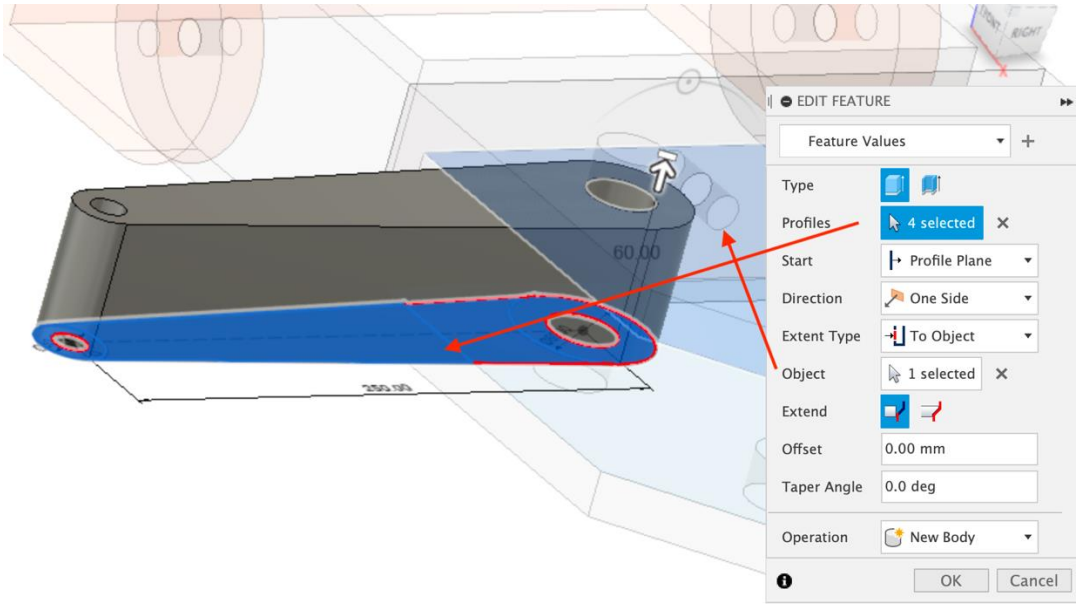


4. Draw this sketch – a dimensioned construction line from the existing hole in the bracket, horizontal, plus two circles, one at each end of this line, with tangent lines between the outer circles.

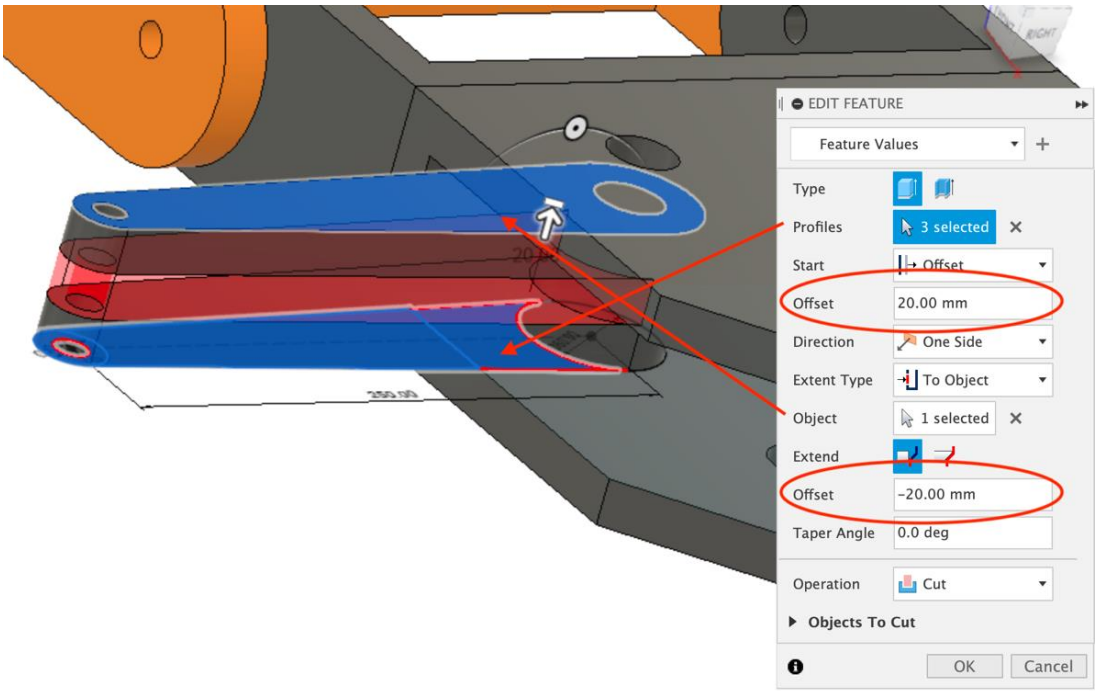
    a. Note: These lines can be created by mouse down on the circle, and dragging to the other circle, snapping to tangent points. This is a great skill to learn. Make sure to add a circle at the small end of the arm to form a hole:




5. *Extrude* the profile, using To Object, and selecting the upper face of the "gripper bracket":

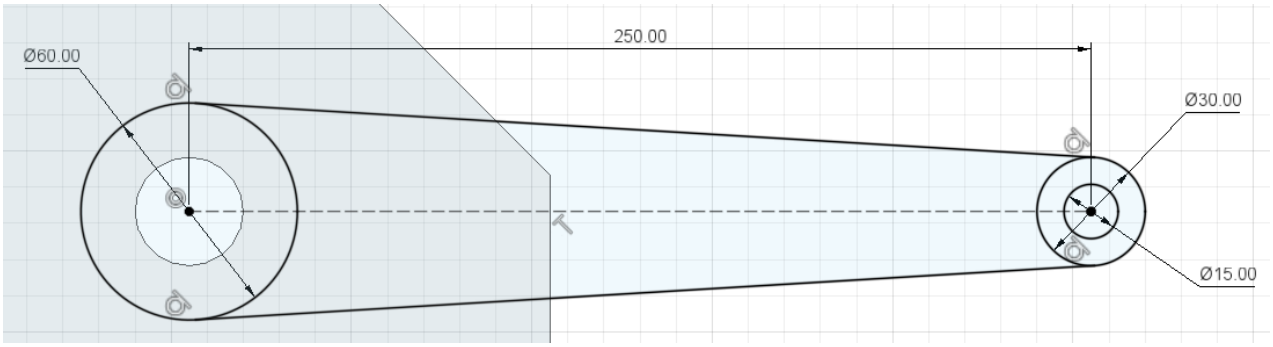



6. Just like we needed to hollow out the "gripper bracket" component, we need to do the same for this arm, to allow room for the actual gripper component. But, we will do this one differently, using the sketch for the previous *Extrude*. Make it visible, then start *Extrude*. Select a subset of the profile from the sketch, as shown. Use To Object again, selecting the top face of the extruded body. But, we will use 2 offsets this time, to keep material on both sides of the body. This is another useful trick to know:

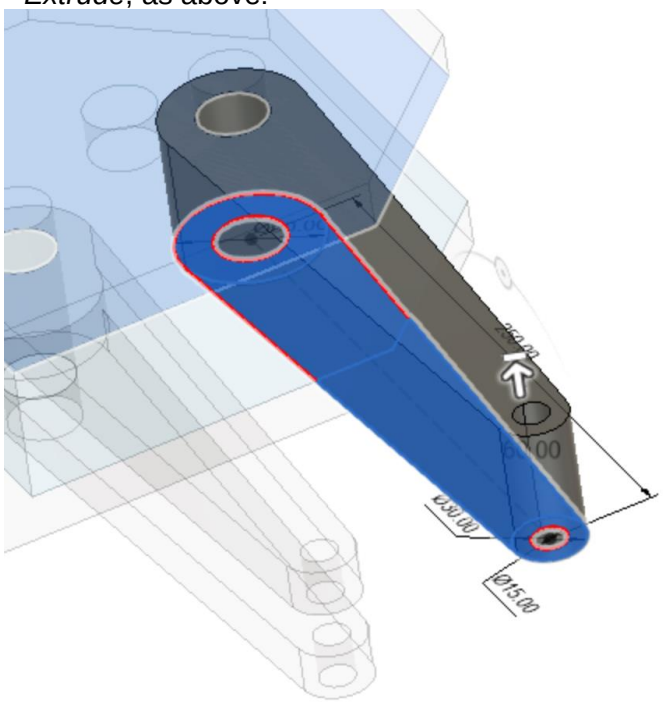


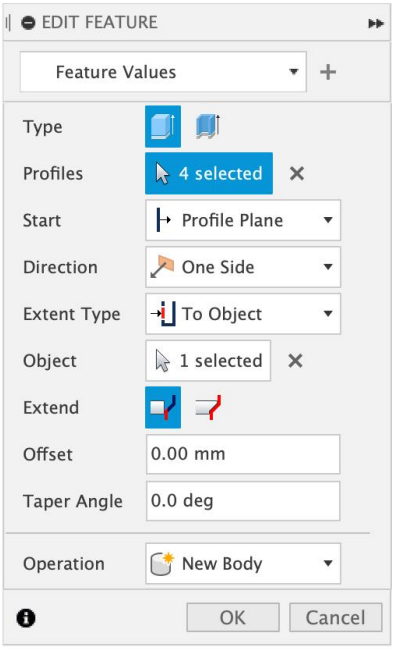

7. Activate the “gripper assy” component, then Create > *New Component*, to create the “front gripper arm” component. Construction of this arm is exactly the same process as the rear arm, just different sketch dimensions:


8. Create > *New Sketch*, again on the lower surface of the “gripper bracket”. Create this sketch:

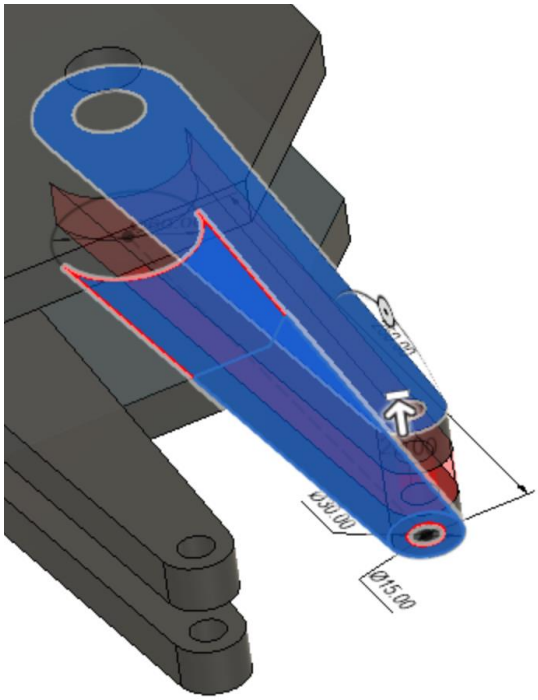


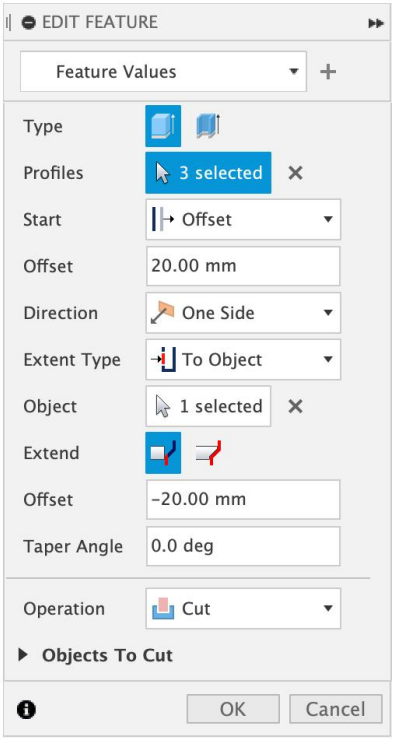
9. *Extrude*, as above:





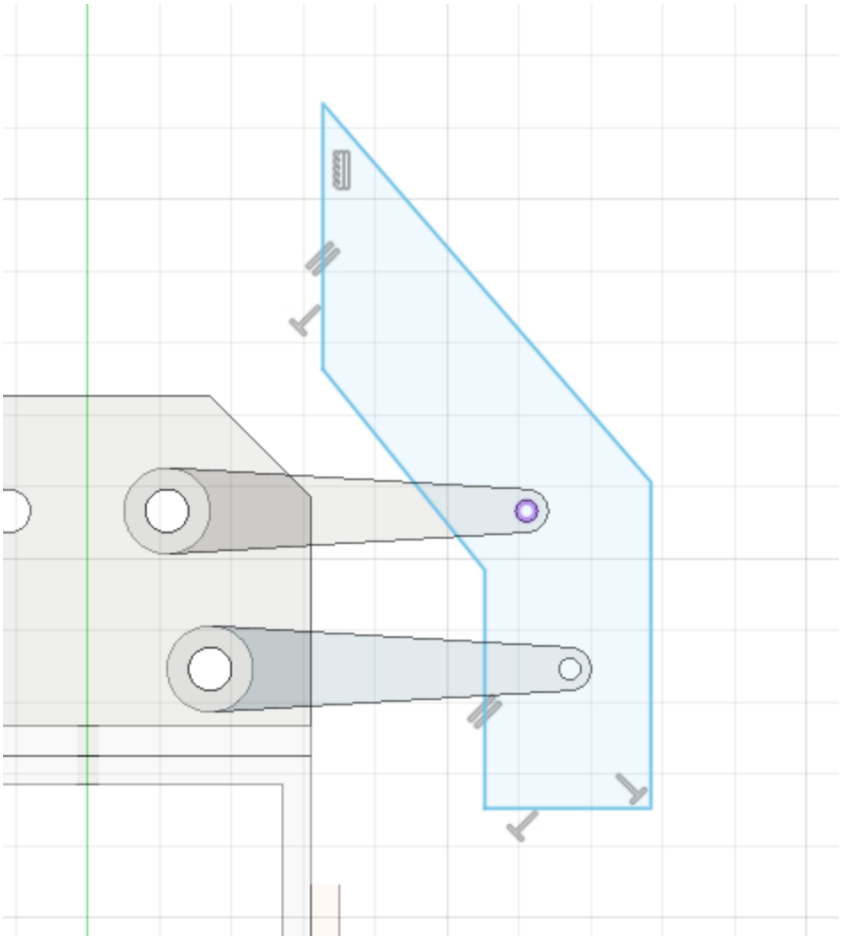

10. And, again, as above, *Extrude* Cut, using the sketch, To Object, and two offsets:





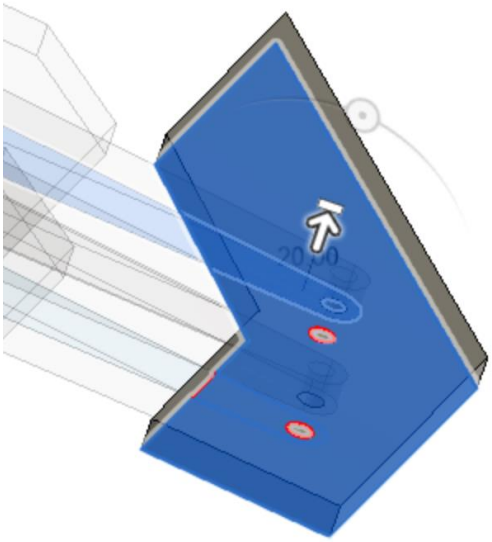
11. Activate “gripper assy” again, and Create > *New Component*, to create the “gripper” sub-component:

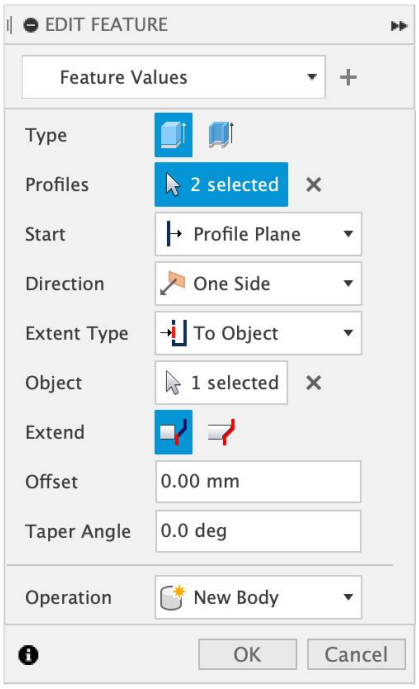

12. Create > *New Sketch*, choosing the inner face of “rear gripper arm”, and create a sketch like this:



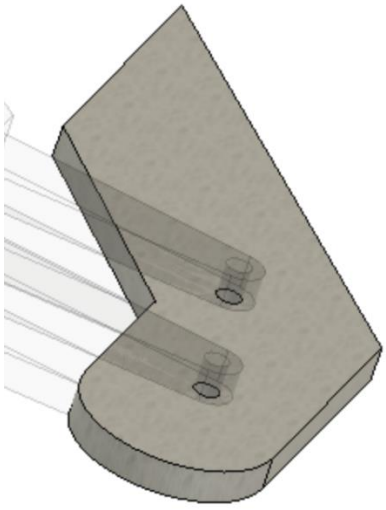
a. Here is where you can show some creativity: Make your gripper look however you want, at whatever size (you’ll notice that there are no dimensions on our version…)

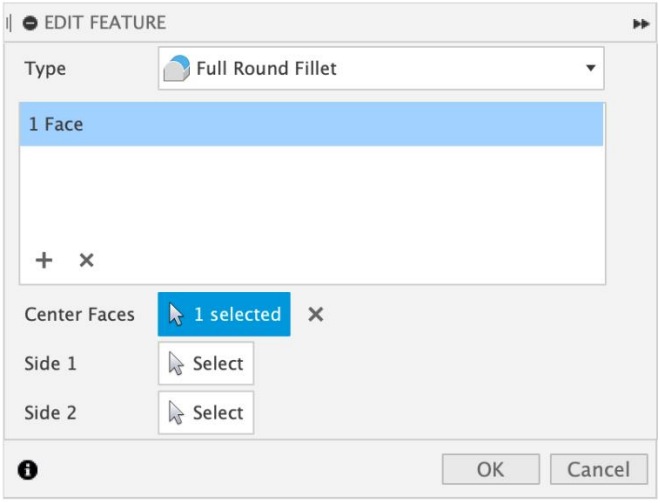
13. *Extrude*, using To Object, selecting the upper face of either of the arms:



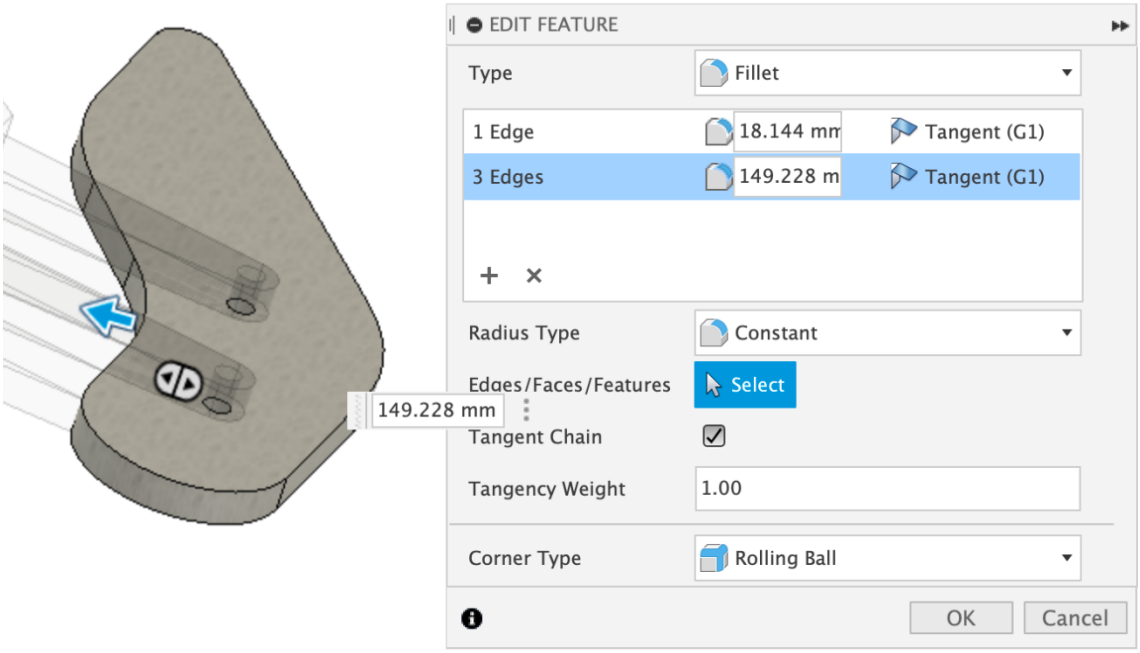



14. Add a *Full Round Fillet* to the back of the gripper:

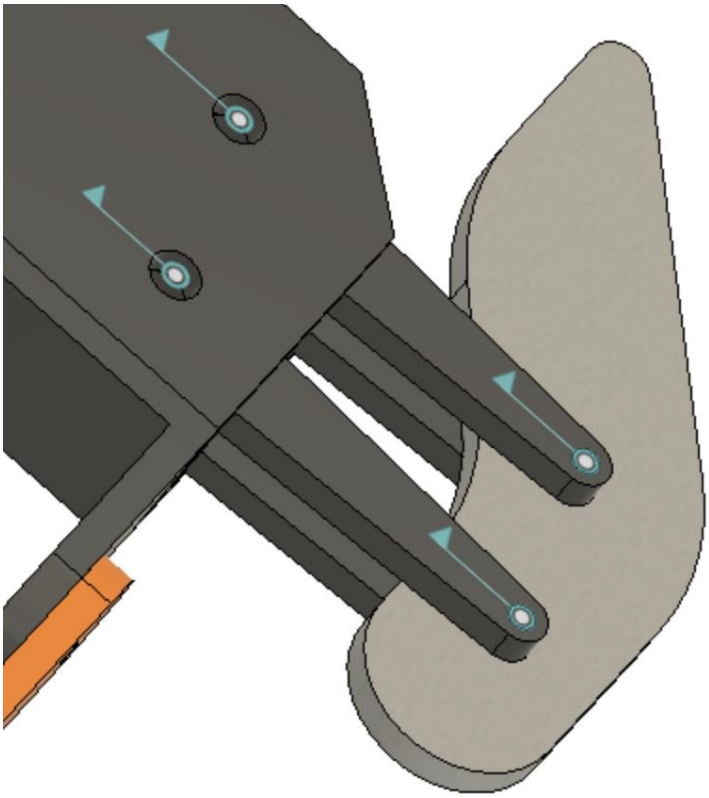





15. And, add another *Fillet* feature (this time: Edge Fillet) to fillet the remaining sharp edges. Use whatever radii you feel looks best. Again, room for creativity here:




16. Add Joints. Four Joints are needed. All 4 are *As-Built Joints,* Revolute:



Two of these Joints are internal to the “gripper assy” component, and two are owned by

the root component. We will see the effect of this in the next step, since only two will be needed for the other instance of this assembly.

**Commands used:**

- New component
- New sketch
- Sketch circle
- Sketch line
- Sketch dimension
- Extrude
- Hole
- As-Built Joint

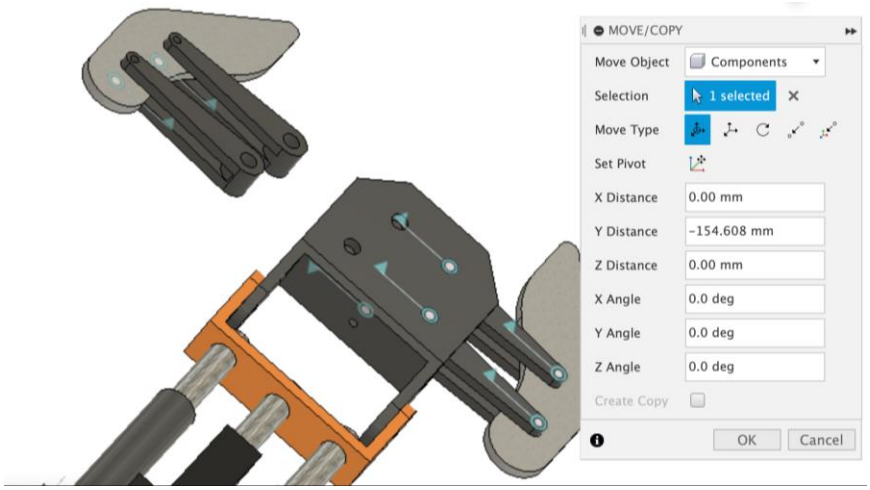
**Insights:**

- We learned another way to add a cutout region to a body, along the extrusion direction of the first *Extrude*. Re-use the profile sketch (but with an Offset from the profile plane), and use To Object + an Offset value to maintain material on both sides of the body.

## Step 15 – Second Instance of the Gripper Assembly

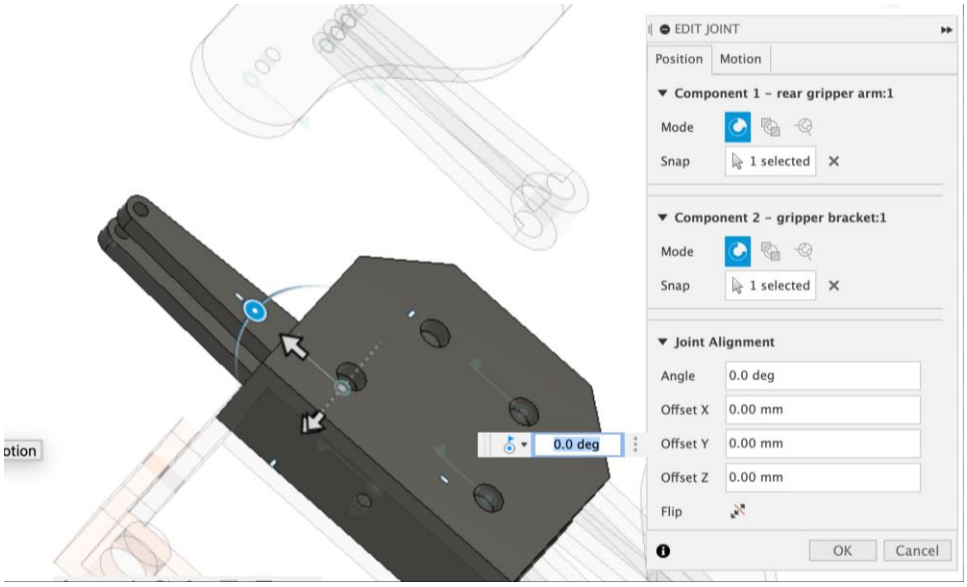
*SECOND INSTANCE OF GRIPPER ASSY*

1. Now, as with the other sub-assemblies used in this model, we need a second instance. This one, also, needs to be rotate. So, the first step is to *Copy* "gripper assy", and use *Paste*. When the move dialog appears, rotate the assembly by 180 degrees, and translate it to the other side of the gripper bracket. You do not have to get it exact, just close:

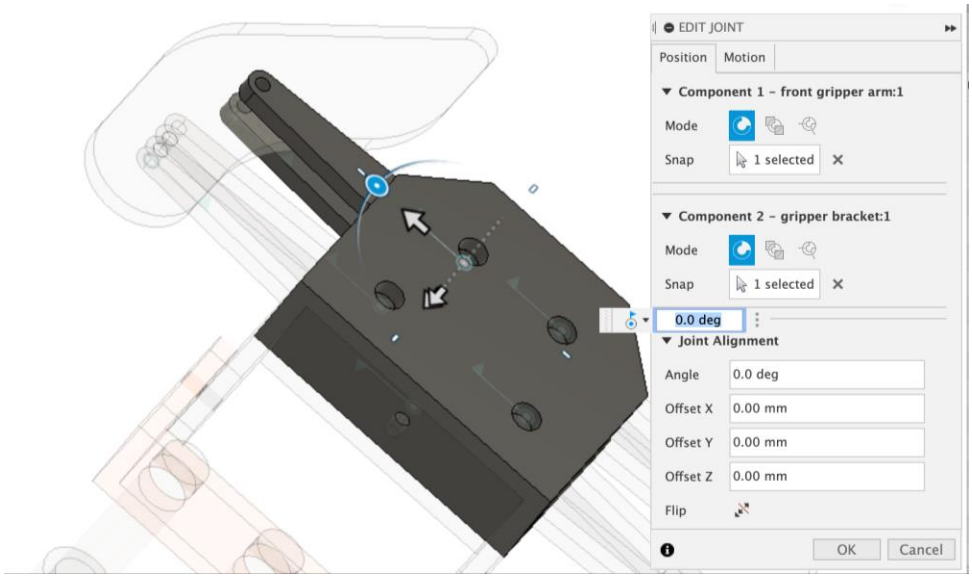



2. Make the Joints. The first will be a "regular" *Joint* (not As-Built), because the second instance is not in position. You can choose either the front or rear arm. We chose

the rear arm. Choose the top of the hole on the arm, and the bottom of the hole in the bracket as the snap points:




3. Because the rest of the assembly will move with that joint, the second revolute joint can be an *As-Built Joint*:




Important: The revolute joints between the arms and the gripper do not need to be created in this instance. Because those joints are owned by the “gripper assy” component, they come along for the ride.

**Commands used:**

- Copy/Paste of a Component
- Joint
- As-Built Joint

**Insights:**

- We learned that Joints that are internal to a sub-assembly will come across when a new instance of that sub-assembly is created, and that creating a “regular” *Joint* can also position a sub-assembly so that an *As-Built Joint* can be added.

## Appendix 1 - Command list

Click on each item to see the reference documentation for that command.

Create component
Create Sketch
Sketch Line
Sketch Circle
Sketch Dimension
Sketch Rectangle
Sketch Project
Sketch Project->intersect
Extrude
Hole
Fillet
Offset plane
Copy/Paste Component
Joint
As-Built Joint