LINDEN ROBOTICS
FRC-3568 Linden RoboEagles
FTC-7031 Linden Battle Blazers
Autodesk Fusion 360 Training

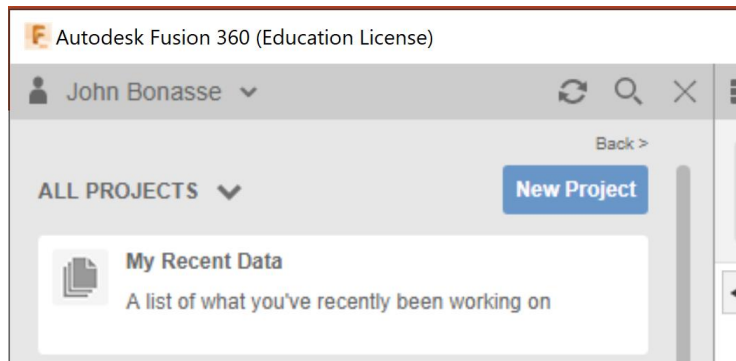# We need to thank our FIRST sponsor -

You should have already downloaded and installed Fusion 360.
If you haven't please see the email previously sent with details.

Open the software. You may be prompted to sign into Autodesk.
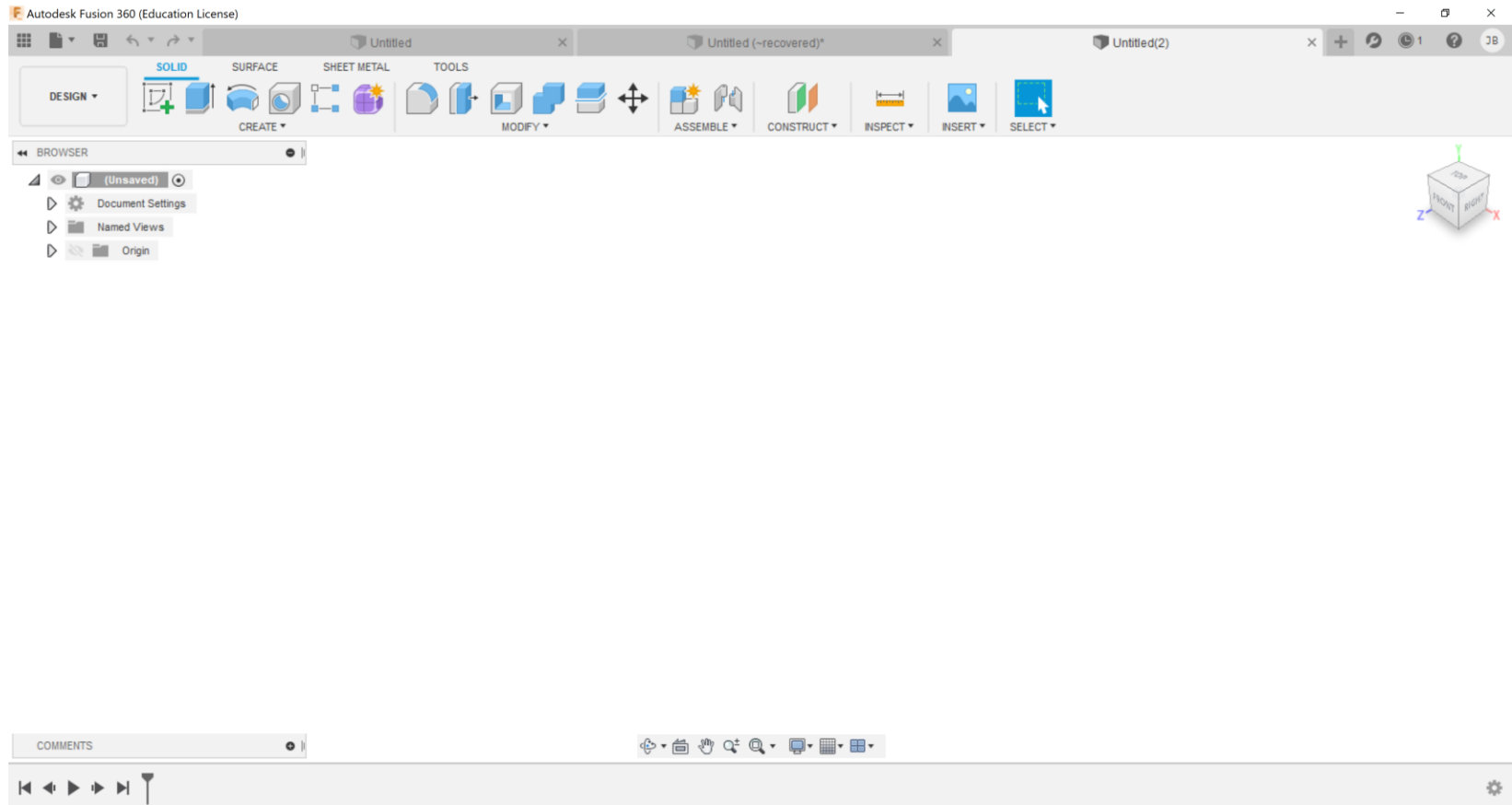
Click on New Project and name it RoboEagles.

Your screen should look similar to the one shown below.

We learned how to Create a SKETCH

Autodesk® Fusion 360™

SOLID   SURFACE   SHEET METAL   TOOLS

CREATE ▾   MODIFY ▾

- New Component
- Create Sketch
- Create Form
- Derive
- Extrude   E
- Revolve
- Sweep
- Loft
- Rib
- Web
- Emboss
- Hole   H
- Thread
- Box
- Cylinder
- Sphere
- Torus
- Coil
- Pipe

Use Sketch mode to create basic geometry profiles that define the design. First select a construction plane. Then create lines, arcs, or points and use dimensions to constrain the boundaries. Sketches are used with 3D creation commands such as Extrude.
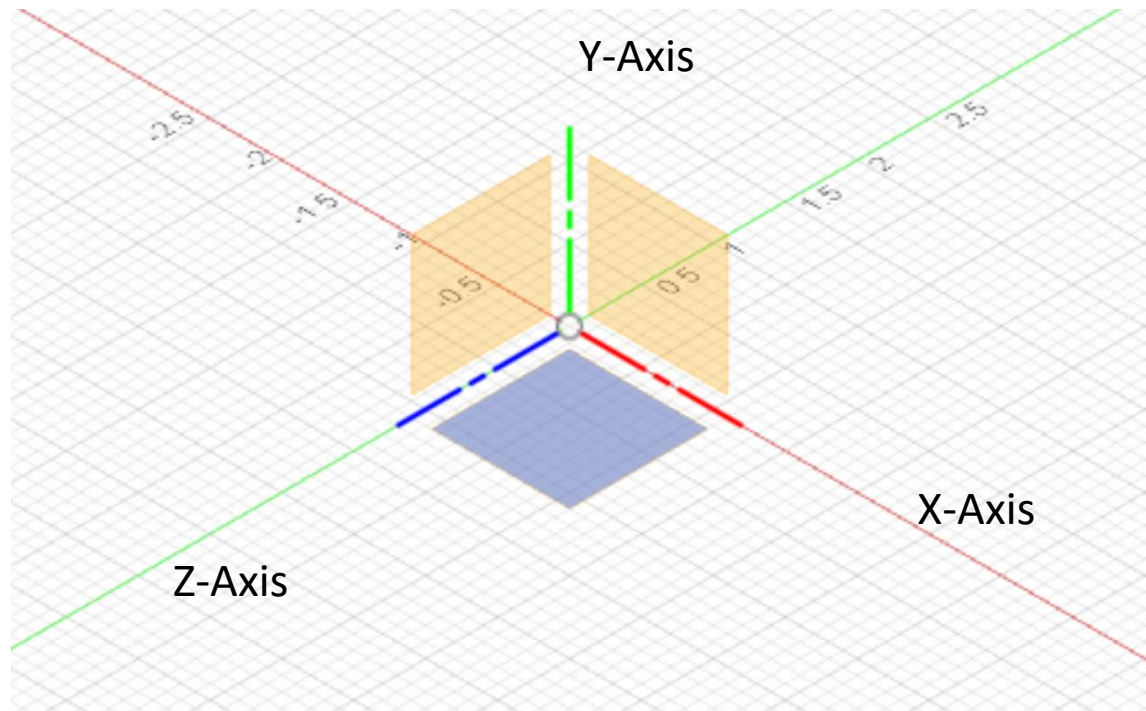
After creating the boundary, select Finish Sketch to exit Sketch mode.
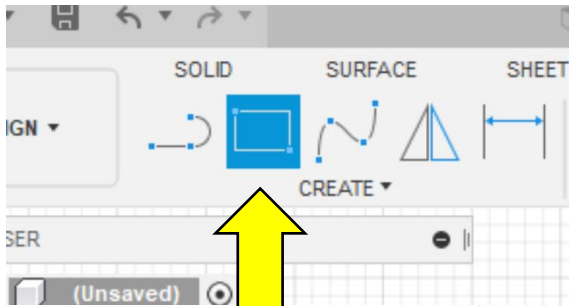
We learned about 3 dimensional planes

There are 3 drawing planes – X, Y, & Z.

We used the Rectangle Tool
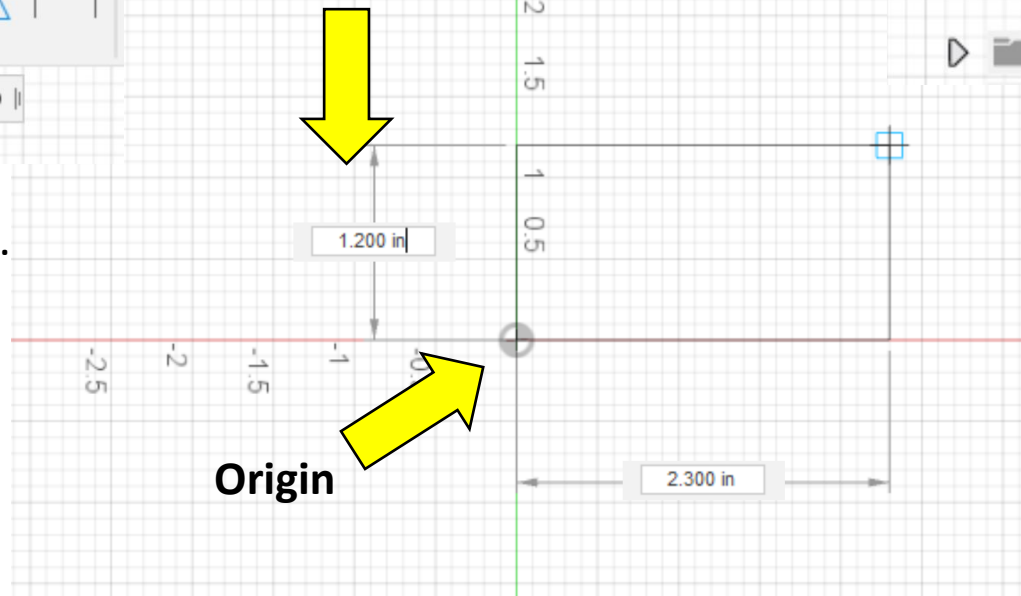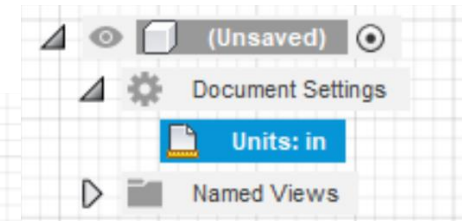
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Dimensions can be changed to inches or mm using Document Settings.

(Unsaved)

Document Settings

Units: in

Named Views

SOLID   SURFACE   SHEET

IGN ▾

CREATE ▾

SER

(Unsaved)

Select the **Rectangle Tool**.

1.200 in

2.300 in

Origin

We used the Press Pull Tool
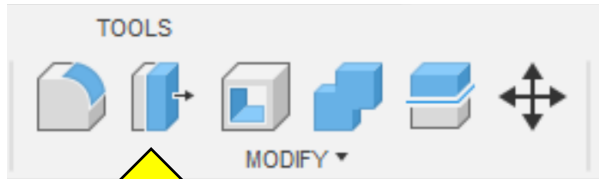
B - Choose **Select**.

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TOOLS

MODIFY ▾

● PRESS PULL

Selection   Select

ⓘ   OK   Cancel

A - Select the **Press Pull Tool**.

● EXTRUDE

Profile	1 selected ✕
Start	→ Profile Plane ▾
Direction	One Side ▾
Extent	↤↦ Distance ▾
Distance	0.00 in ▾
Taper Angle	0.0 deg ▾
Operation	New Body ▾

ⓘ   OK   Cancel
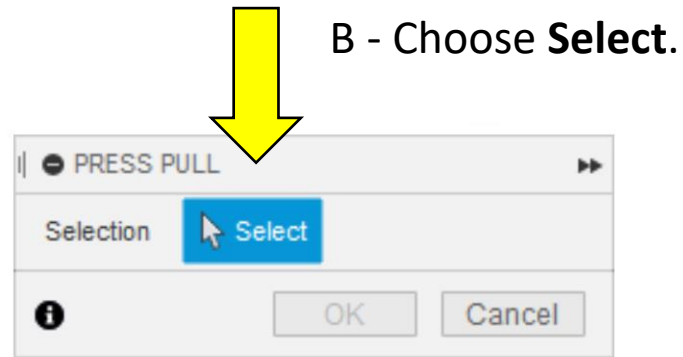
C - Click on the
Square.

0.00 in ▾
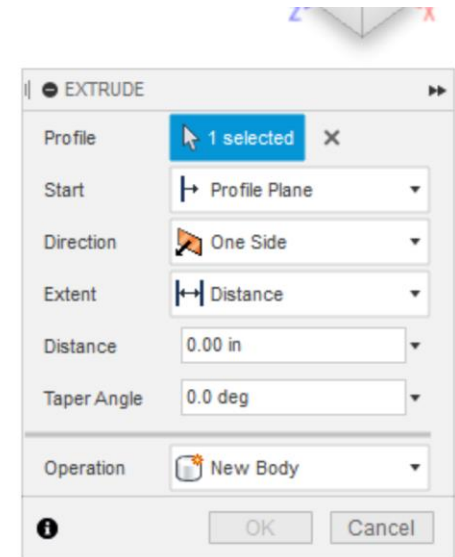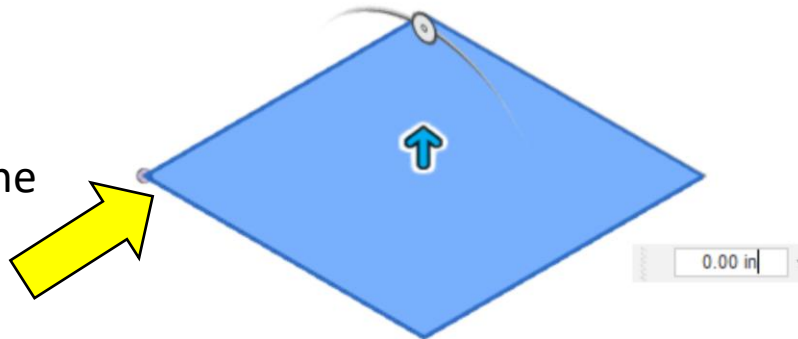
We used the Line Tool to make Construction Lines
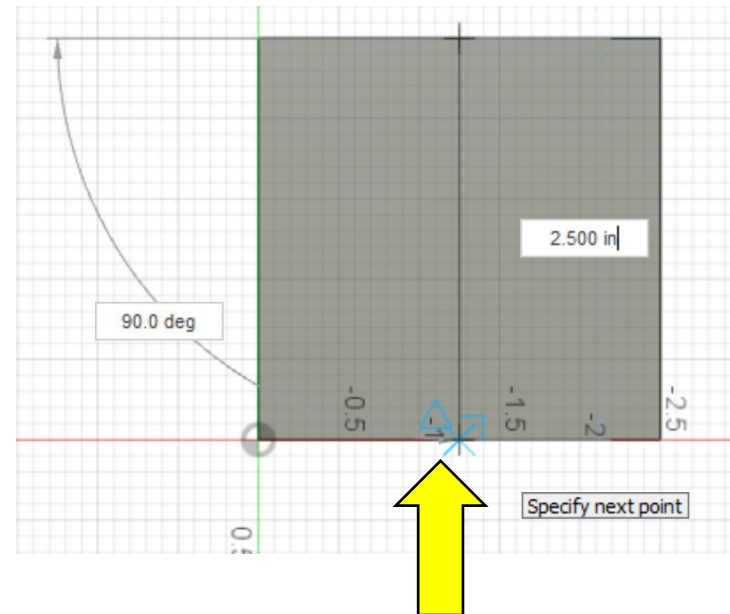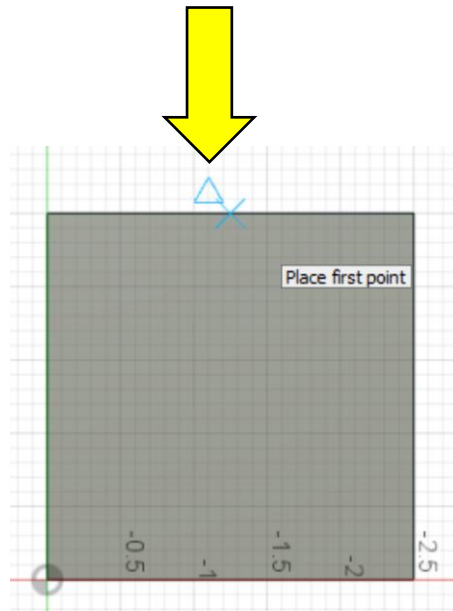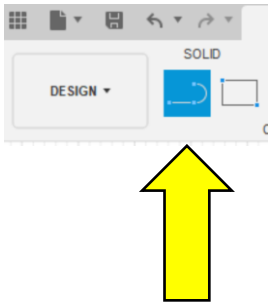
We used the Offset Tool
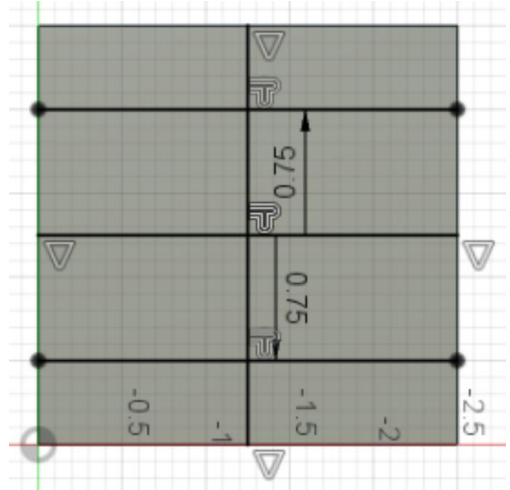
We used the Center Diameter Circle Tool

We used the Chamfer Tool

We used the Fillet Tool

We finished with a completed Bracket

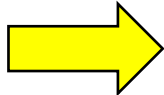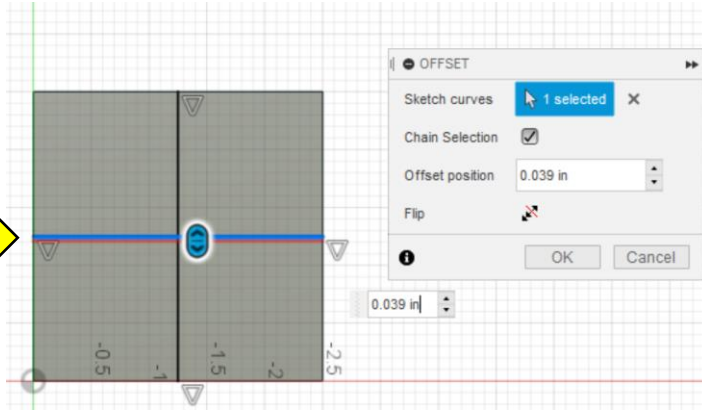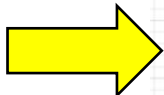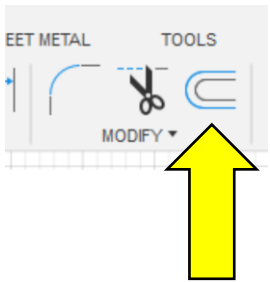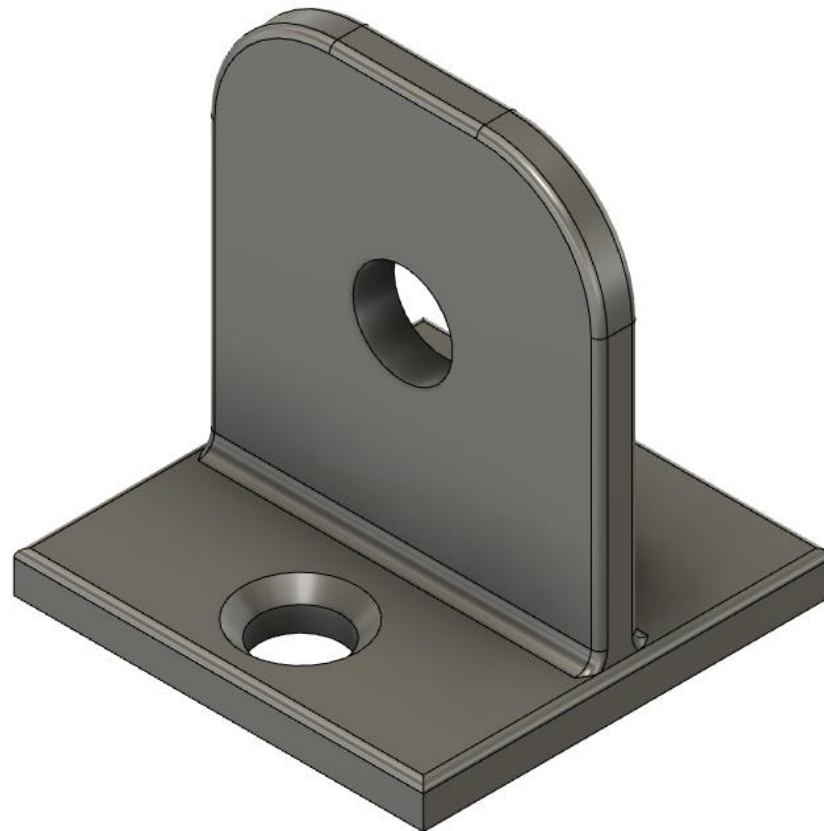## Assemblies, Bodies, & Components

In Fusion 360, **a Body is any continuous 3D shape**. Whenever you create a 2D sketch in Fusion 360 and then extrude it into a 3D shape, you automatically create a Body.

So therefore, **the Bracket we made is considered a Body**.

There are **four types of Bodies**; Solid, Surface, Mesh, and TSpline (Polygon).

Bodies are shown as a **solid cylinder** in the project tree.
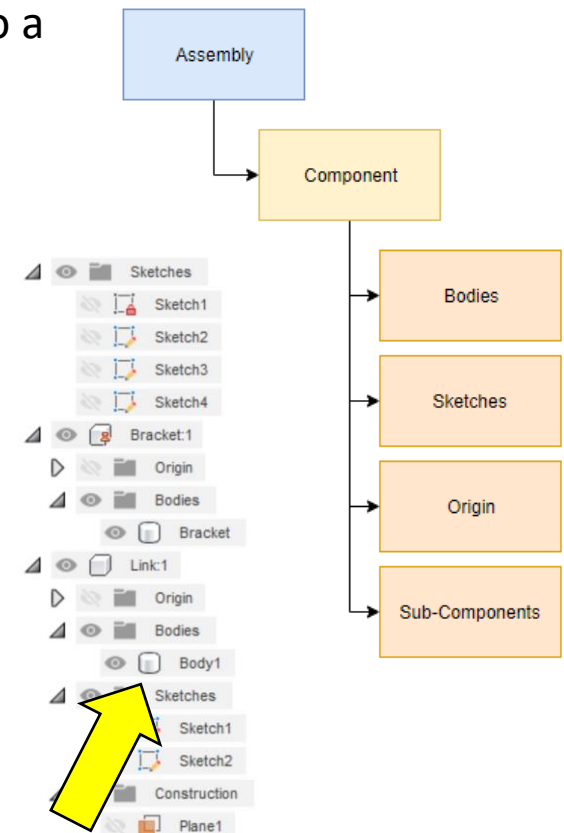
Assemblies, Bodies, & Components

A **Component is a part that is capable of motion and has its own unique origin.**

Components are "containers" that can contain Bodies, and **Bodies can be converted into Components**. They can also contain Construction Planes, Sketches, Canvases, Origin Planes and other Components.

Components are shown as a **solid cube** in the project tree.
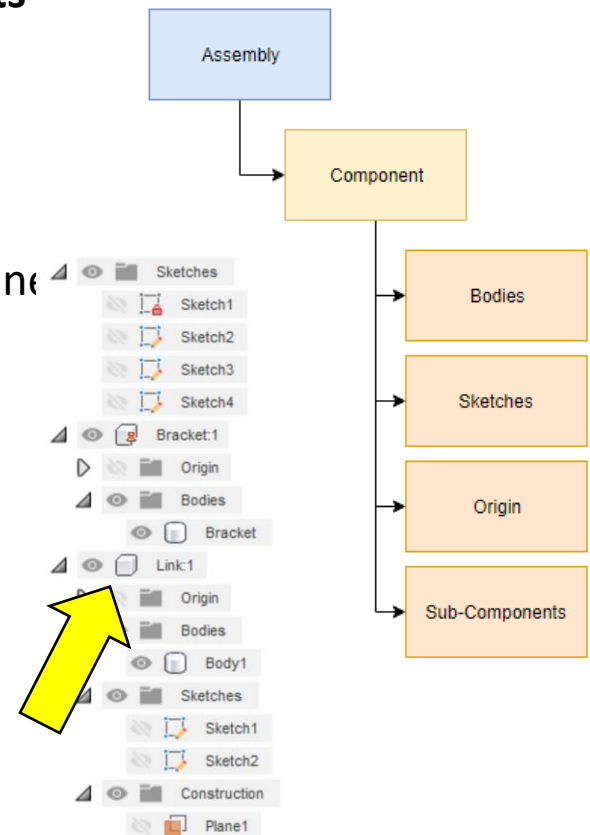
Assemblies, Bodies, & Components

An **Assembly can contain one or more Components.**

Bottom-Up Assemblies are when each piece of the assembly is a separate file and imported into the assembly. **Top-Down Assemblies** are where each piece is a component within the assembly. This is important because of the relationships between each piece. Top-Down creates links that automatically adjust based on the changes made.

Assemblies are shown as a **three stacked component cubes** in the project tree.

Components can be combined within a main assembly to create **Subassemblies**. This helps to organize your project and makes it easier to work within it.
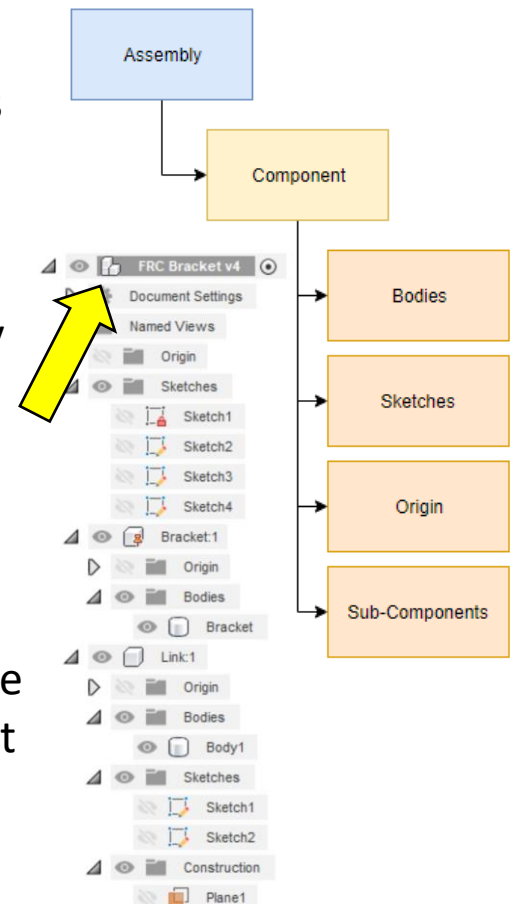
Assemblies, Bodies, & Components

The Unofficial Rules –

#1 Always start your design with a new component.

   This ensures that you always start out with the right file
   structure.

#2 Always rename assets right after creation.

   This enables you to easily navigate through your assembly
   and its components.
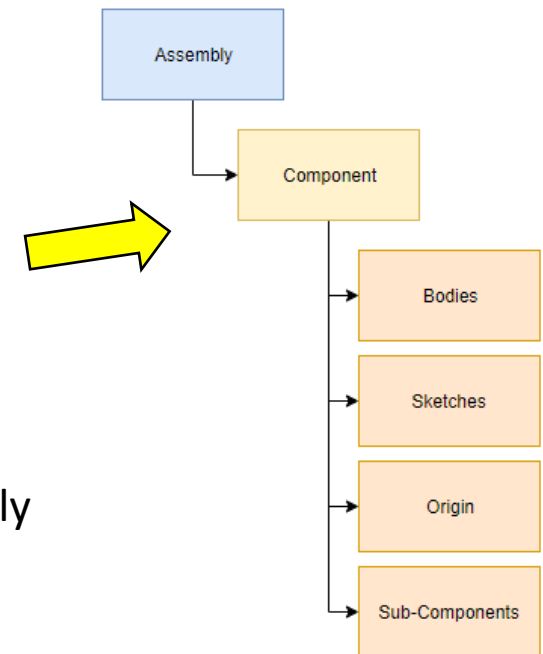
We didn't follow these rules…

Let's Convert Our Bracket Into a Component

1 - Select **Body1** in the Project Browser.

2 - The bracket will become highlighted.

3 - Right-Click and select **Create Components from Bodies**.
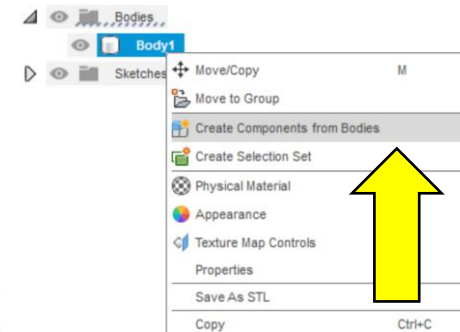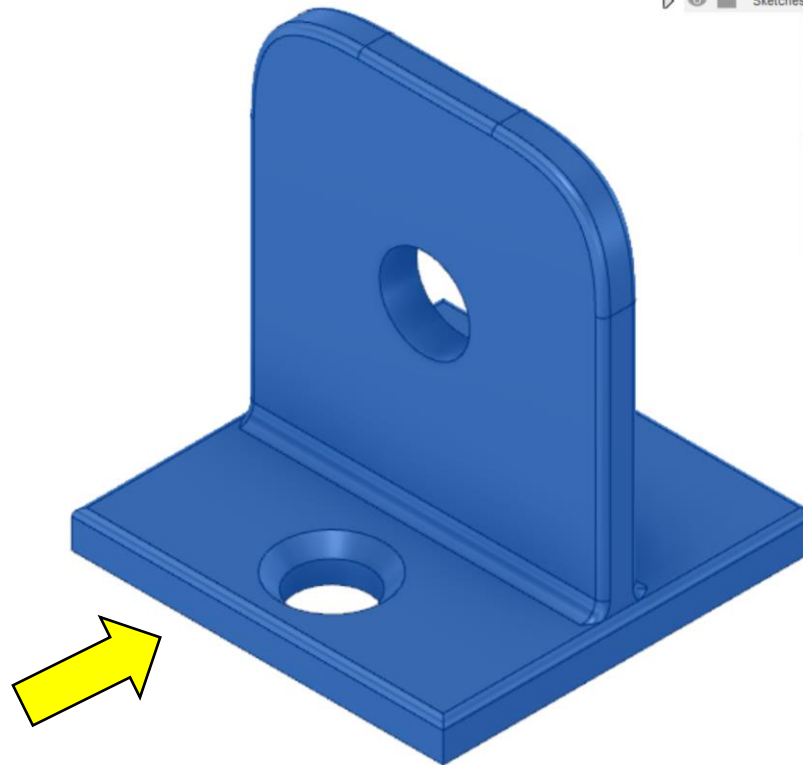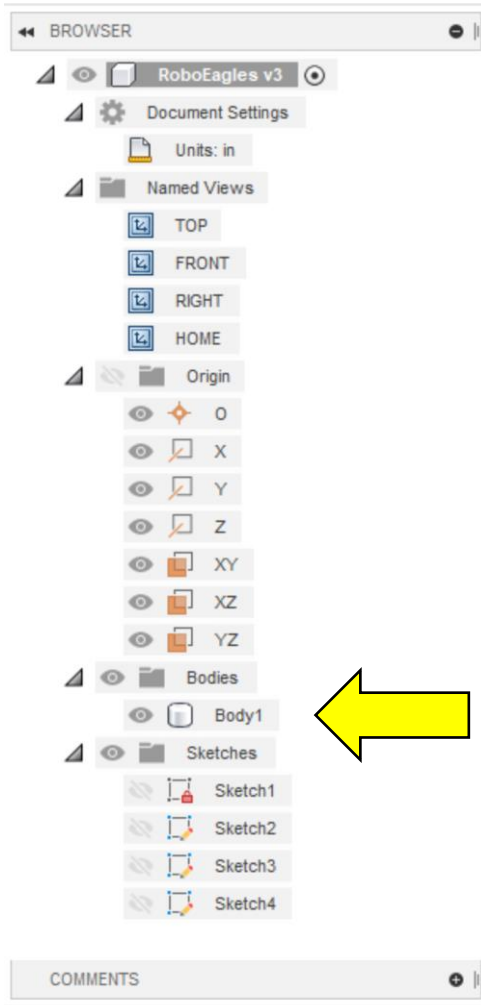
Let's Convert Our Bracket Into a Component

4 – Double click on Component1:1 and rename it "**Bracket**".

5 – The name will change. Open the Component information by clicking on the arrow to the right of the name. Notice that the Component now contains the Body and its Origin information. Rename the Body as "**bracket**".
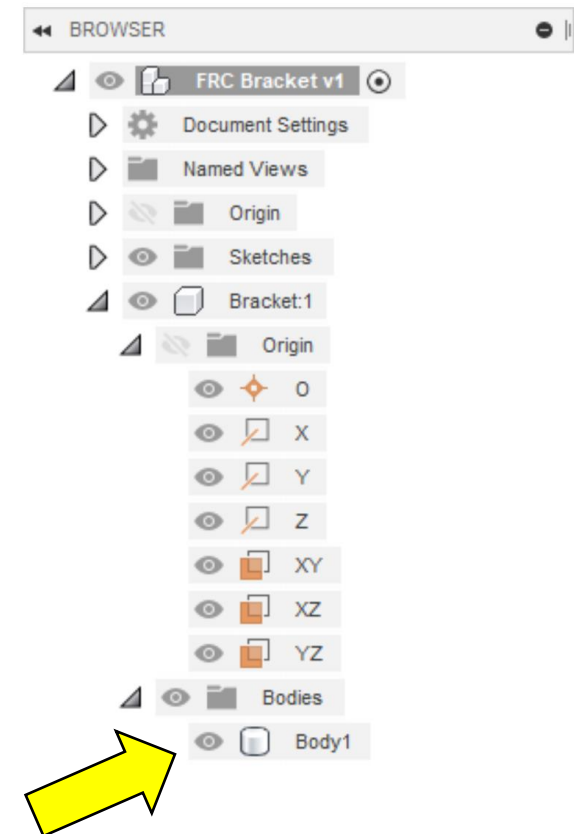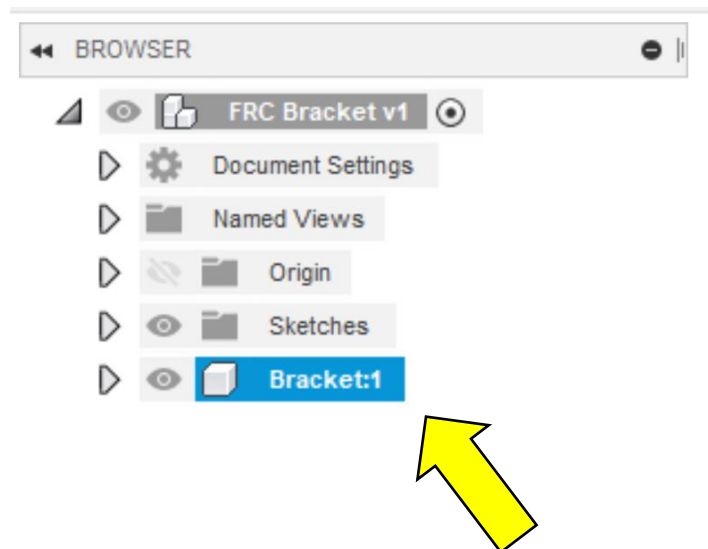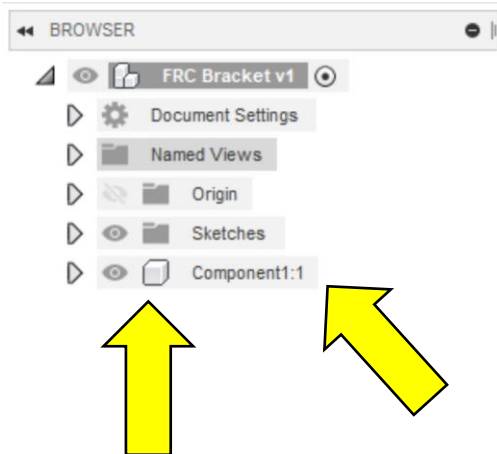
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Let's Create a New Component

6 – Under **Create** choose **New Component**.

7 – Rename the component to "**Link**". Note that the component named "Link" appears in the **Project Browser** but it is empty and only has Origin information.
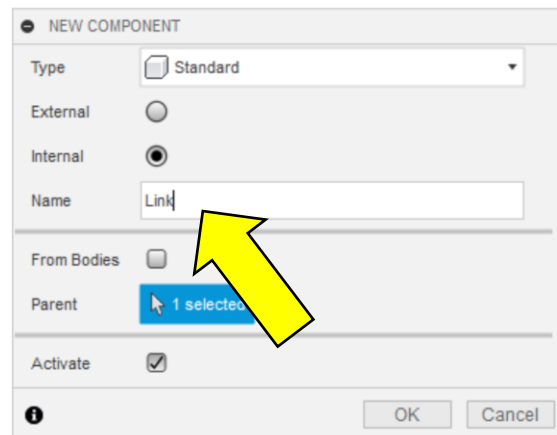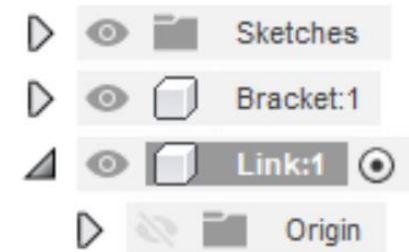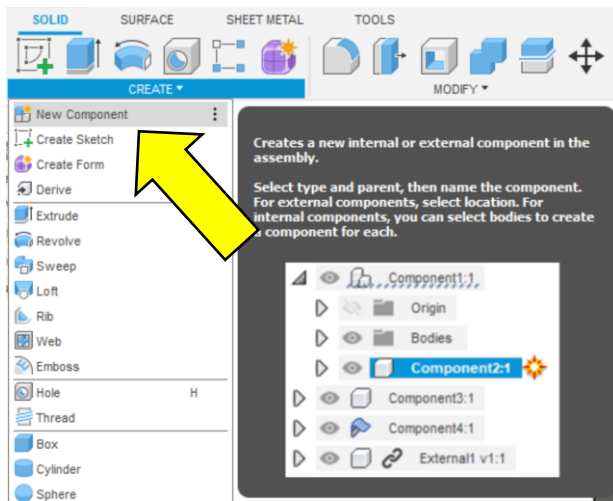
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Let's Create a New Component

8 – We need to create a **Construction Plane** between the bracket's flange surfaces. To do this select **Midplane** under the **Construct** menu.

9 – Select the two outside surfaces of the flange and click OK. You should see an orange **Construction Plane** through the middle of the flange.

Note: Use the Orbit Tool or View Cube to rotate the Bracket to see the back side.
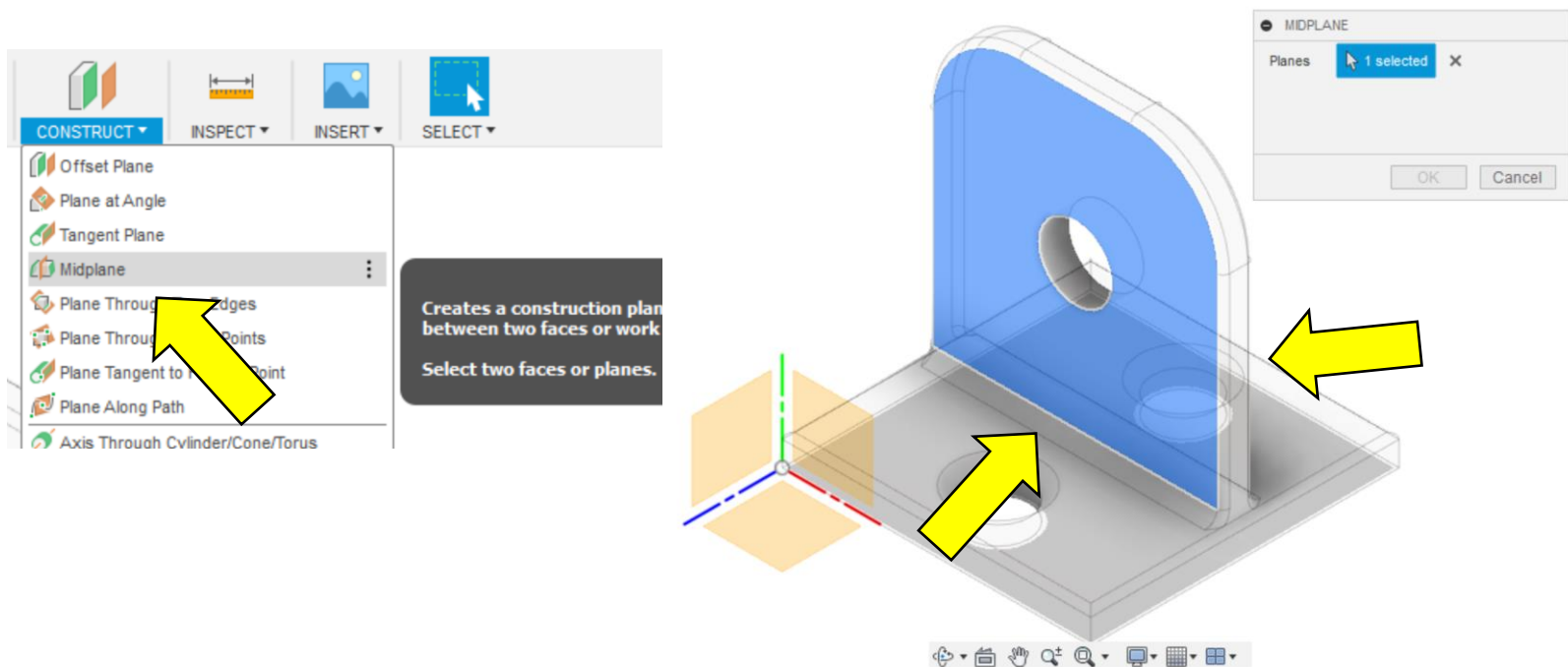
Let's Create a New Component

10 – **Create a Sketch** on the **Construction Plane**.

11 – We need to Project the hole to enable us to create that relationship with the
Link Component. To do this select **Project** from the **Create Menu**.
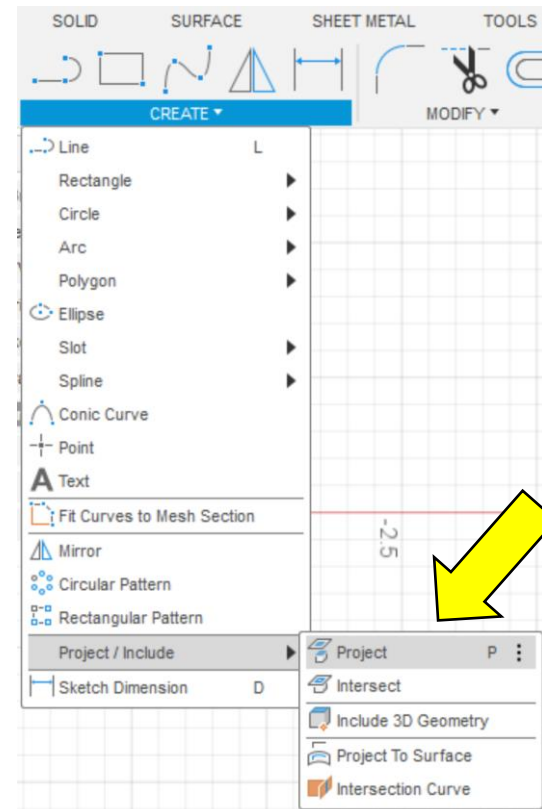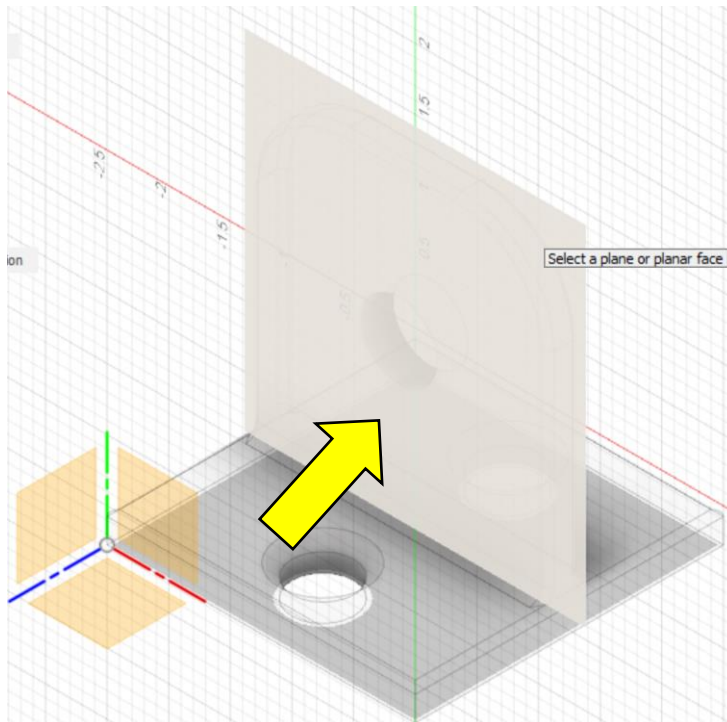
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Let's Create a New Component

12 – Select the hole and click OK. It will now be highlighted on the sketch plane. This will create a **Parametric Relationship** between the two Components. So for example, if the bolt size changes and we modify the Bracket hole size the Link's hole will change as well.

Note: This is a good reason for **Top-Down Assemblies**.

Let's Sketch the Link

13 – Draw a circle using the **Center Diameter Circle Tool** any place on the sketch. Set the diameter to be **1.50"**.

14 – Now we need to set **Constraints** to anchor the Circle to the Bracket. To do this select the **Concentric Tool** from the **Constraints Menu** either by clicking on the icon or selecting it from the menu. Choose both circles and click OK. The circle should **Snap** to the **Center Point** of the **Projected Circle**.

Let's Sketch the Link

15 – Now we need to draw a **Construction Line** from the center of the hole using the **Line Tool** and set its length to **6"**. You can set the **Line Type** to **Construction** by right clicking on the line and selecting the **Normal/Construction Tool**. The line will change to a **Dotted Line** and is ignored by Fusion 360 but allows us to use it for drawing purposes.
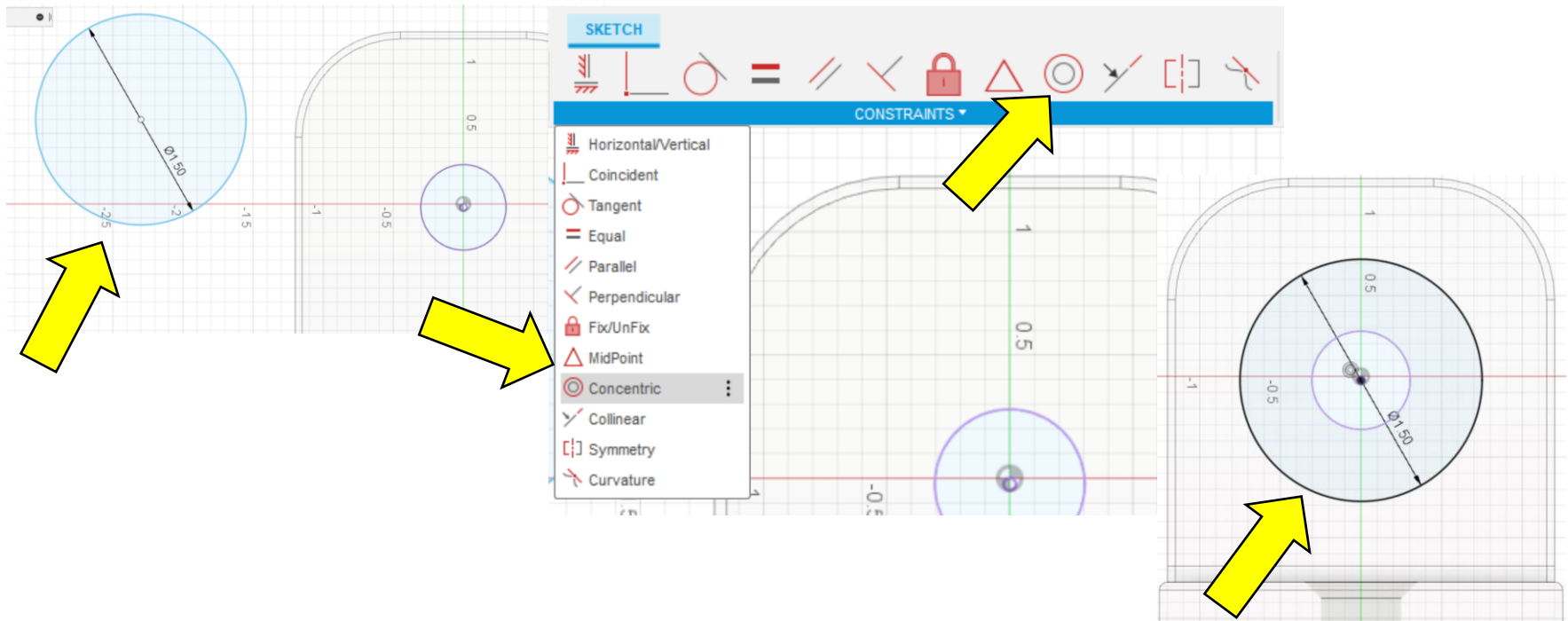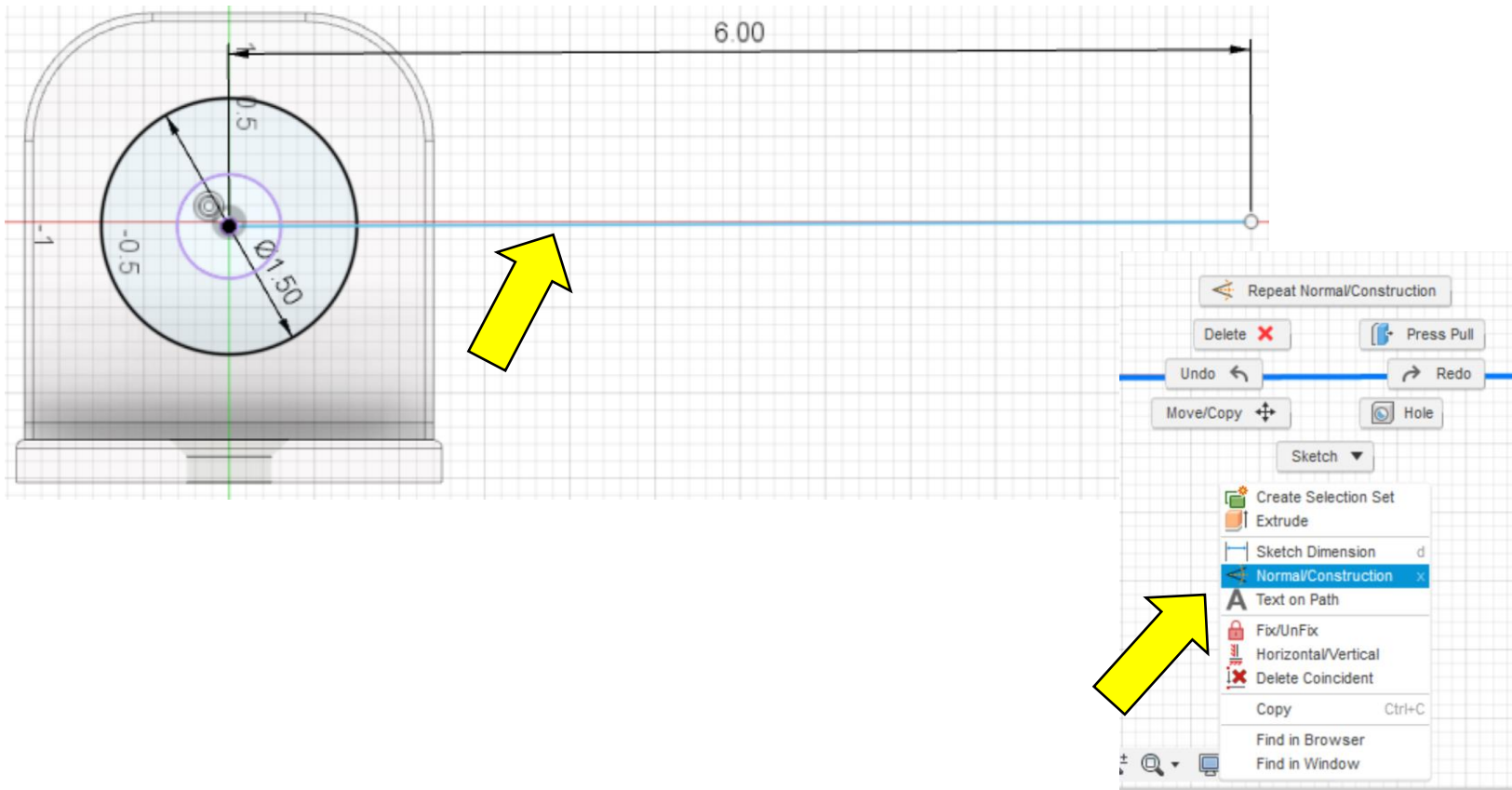
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Let's Sketch the Link

16 – Now **Offset** the **Construction Line** by **.50"** to either side of it using the **Offset Tool**.

17 – We need to **Trim** the lines where they touch the circle. To do this use the **Trim Tool** be selecting its icon or from the **Modify Menu** and clicking on the ends of the lines.

Let's Sketch the Link

18 – Connect the ends of the lines opposite the circle with the **Line Tool**.

19 – Finish the Sketch using the **Finish Sketch Tool**. You will see the Sketch on the plane through the center of the Bracket Flange.

Let's Sketch the Link
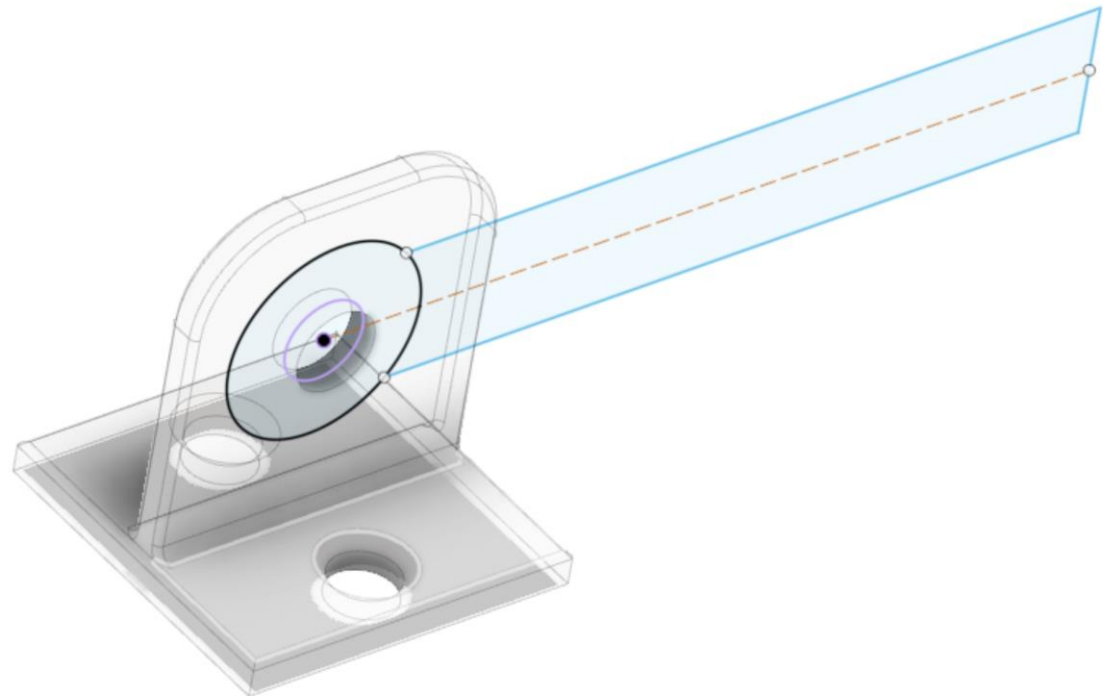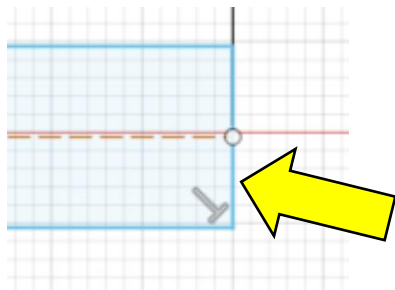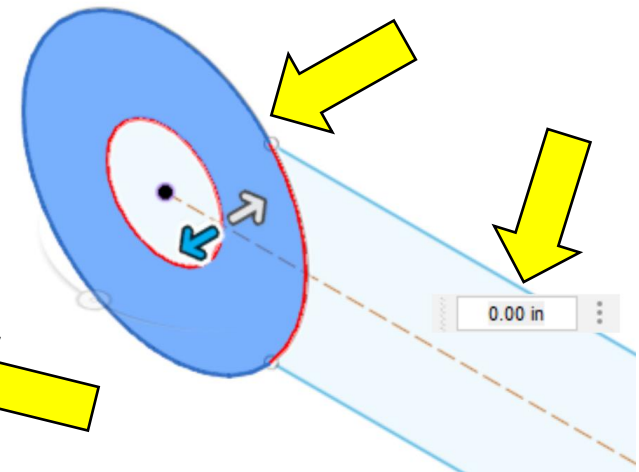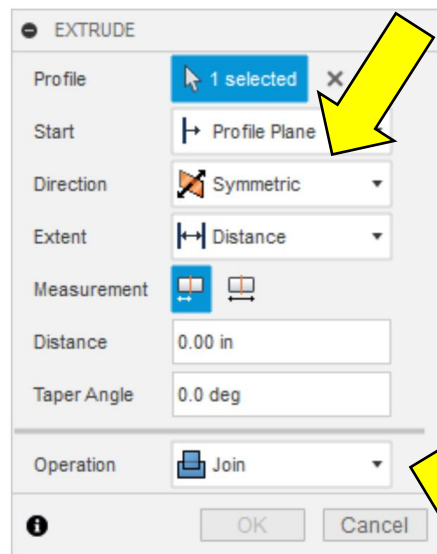
20 – We need to Press Pull the Sketch to create the Link. Start by turning off the Bracket visibility. Open the **Bracket Component** and **Body Folder** and click on the **Eye Icon**.

21 – Select the **Press Pull Tool** and the **Circle**. Change the **Direction** to **Symmetric** and the **Operation** to **Join**. Set the **Distance** to **.50"**.

Let's Sketch the Link

22 – The Sketch has disappeared and will need to be turned back on. To do this click on the **Eye Icon** for **Sketch1** under the **Link Component** and **Sketches Folder**.

23 – Press Pull the Link Arm the same way as Step 21 to a **Symmetric Distance** of **.375"**.

Let's Sketch the Link

24 – Use the **Fillet Tool** to soften and strengthen the Top and Bottom edges. Set the two edges to a **3"** Fillet.

25 – Set the two outside edges to a Fillet of **.125"**.

Let's Sketch the Link

26 – Soften the rest of the edges to a **.10" Fillet**.

Let's Sketch the Link

27 – We need to lighten the Link by removing some uneeded material. To begin we need to **Create** a **Sketch** on the surface of the Link Arm.
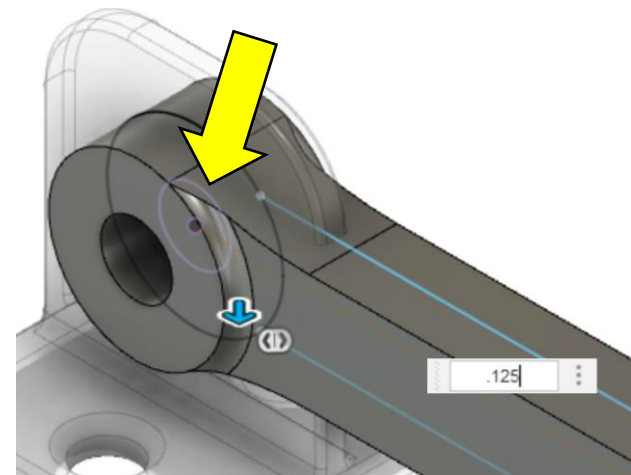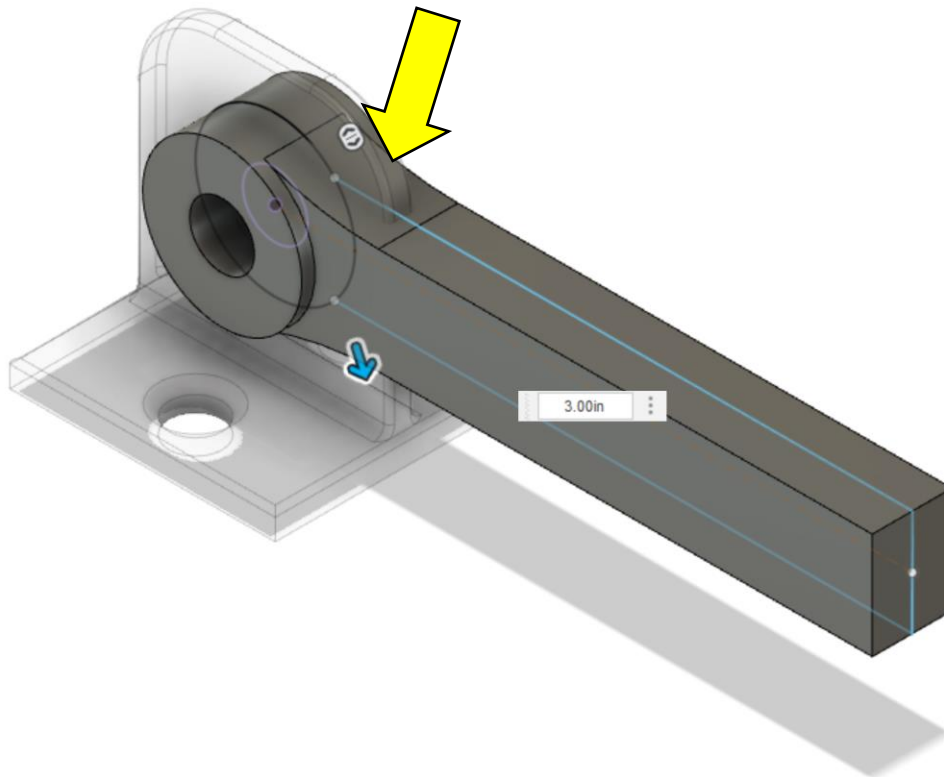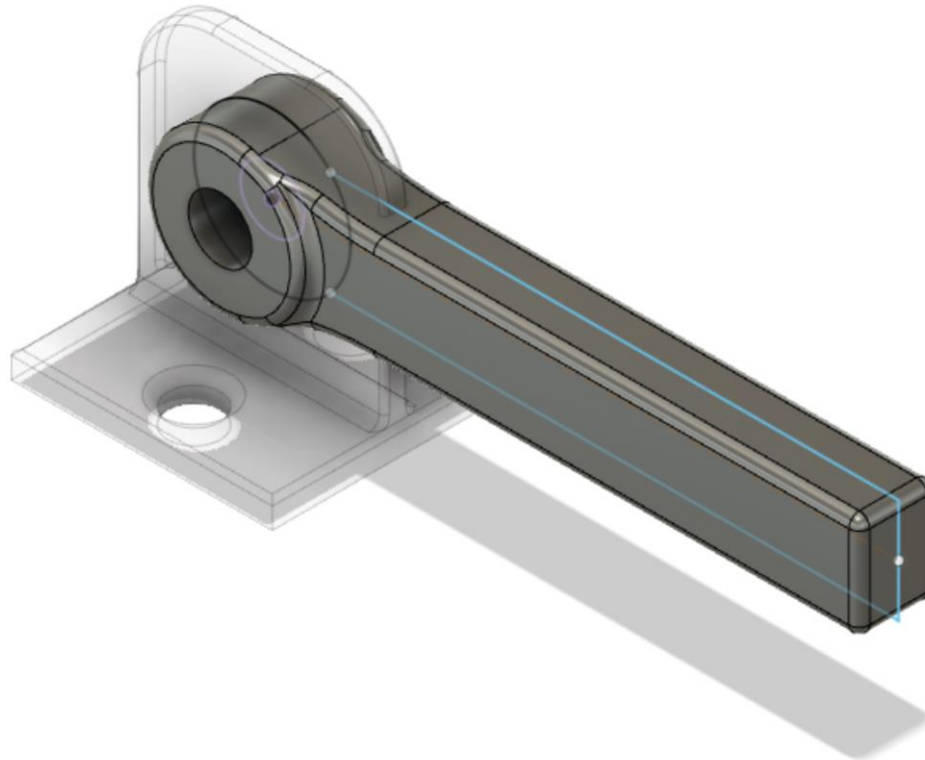
28 – Select the **Line Tool** and pick a starting point for the straight part of the recess to begin. Select the end point and click again and hold to form an **Arc** until the end of the Arc is reached. Click again and start a line back. Place the cursor on the starting point and slowly drag down until the readout says zero degrees. Click to end the straight line and make another Arc to the starting point. **Finish** the **Sketch**.
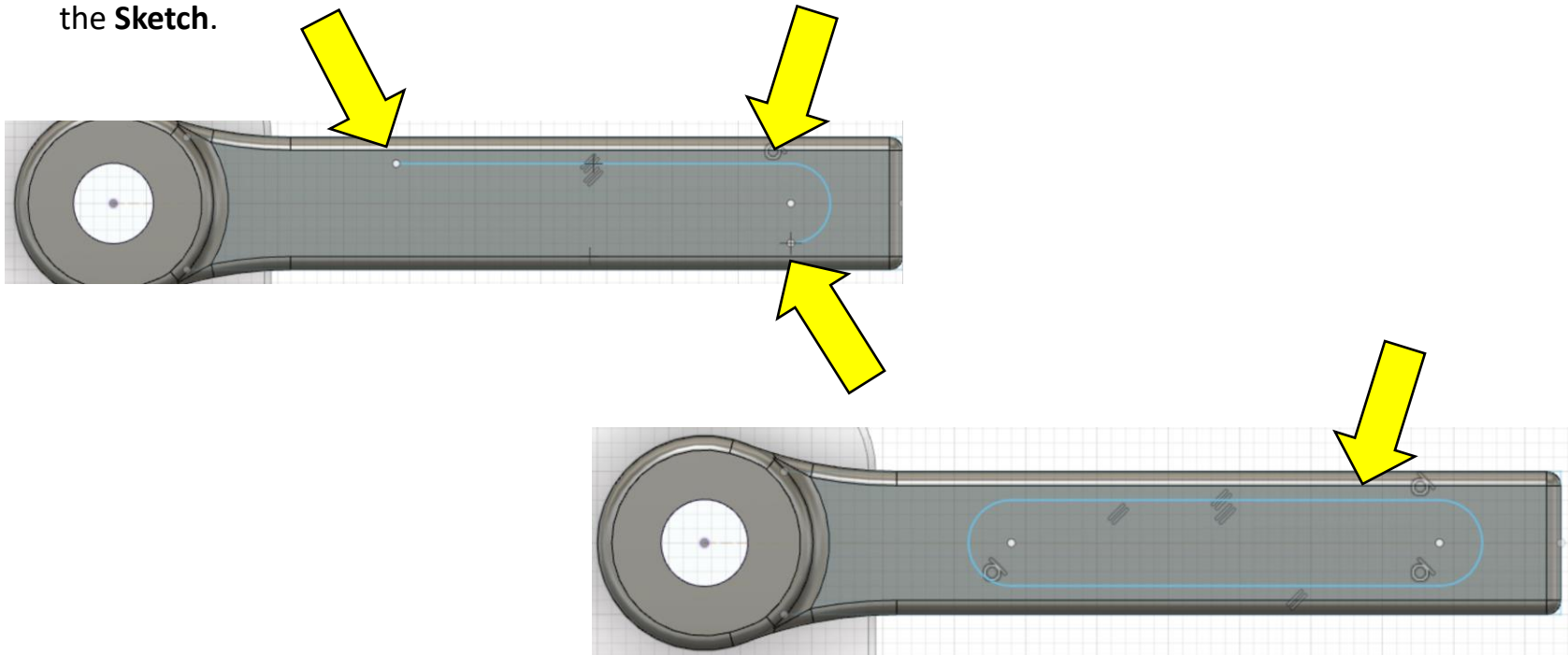
Let's Sketch the Link

29 – Use the Press Pull Tool to recess the form .25" in.

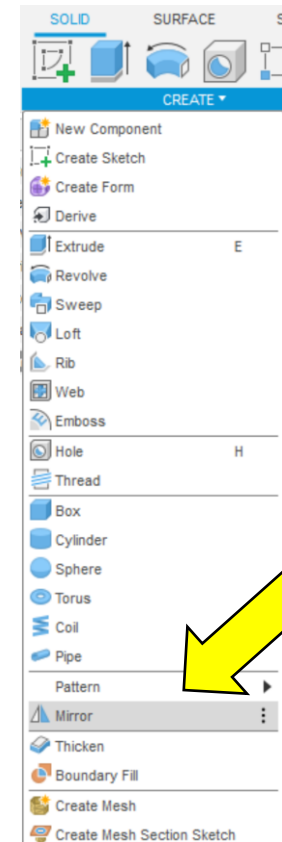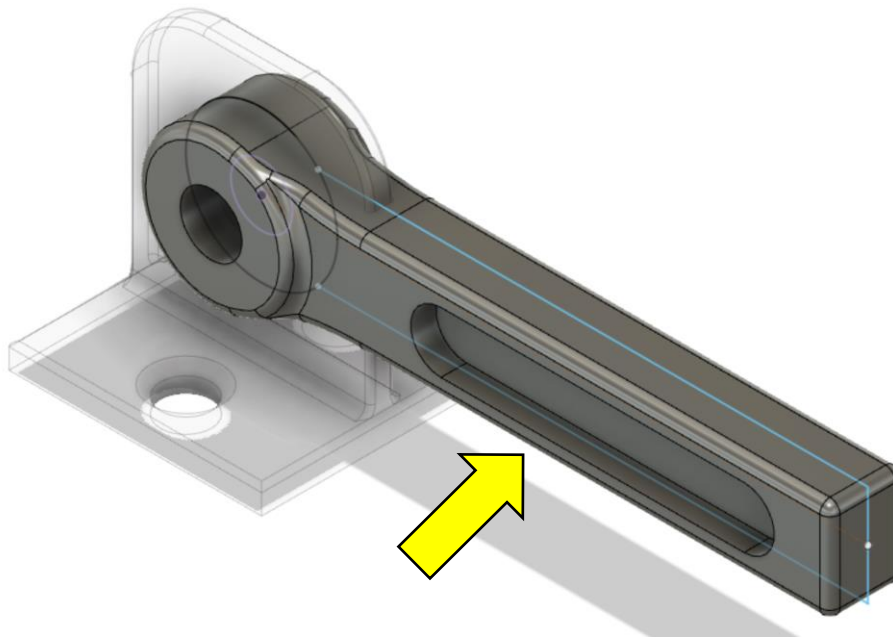30 – Use the **Mirror Tool** to duplicate the shape on the other side. To do this select **Mirror** from the **Create Menu**.
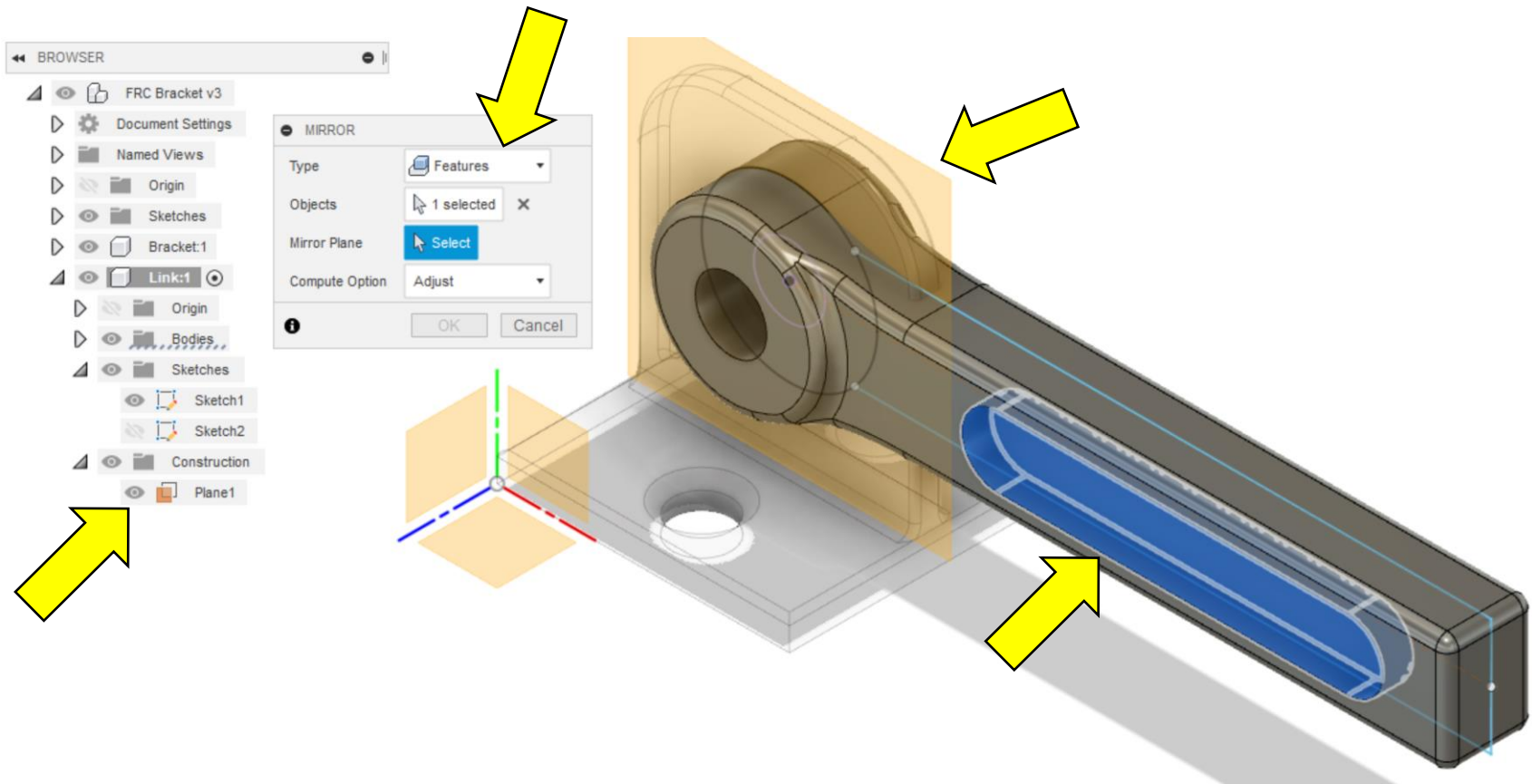
Let's Sketch the Link

31 – Change **Type** to **Features**, Select the Recess Feature and then select the **Construction Plane** we created in Step #9. To select the plane you will need to make it visible by clicking the Eye Icon in the Link Assembly under the Construction Folder. Click OK and the feature will be added to the opposite side of the Link.
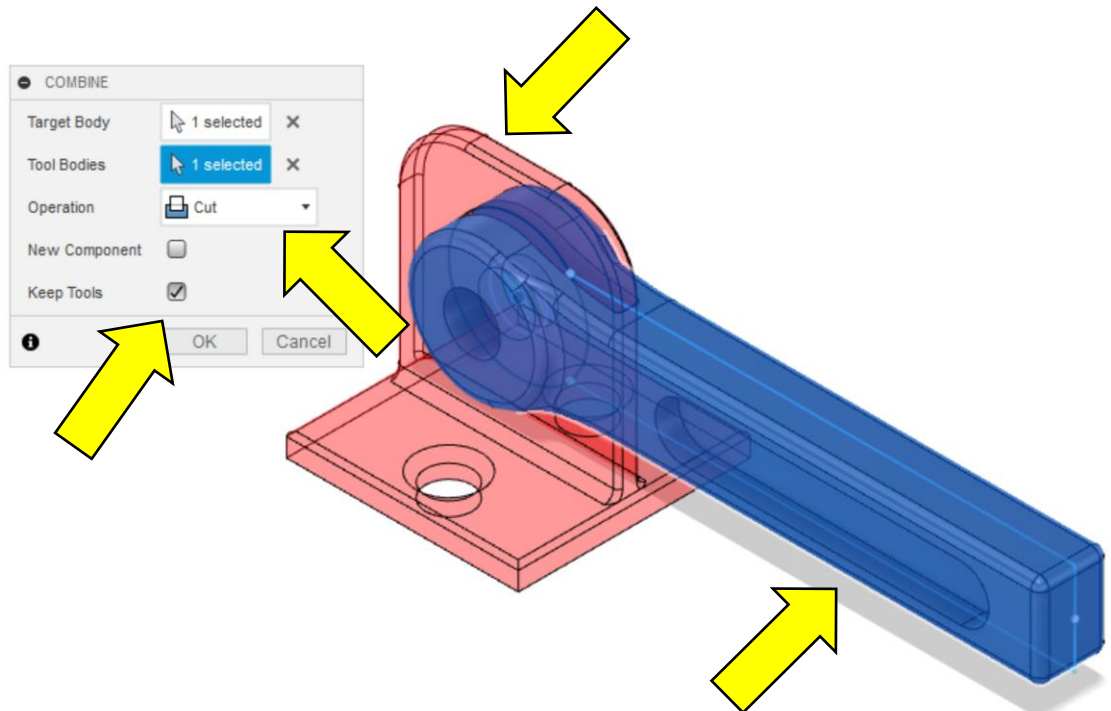
Let's Sketch the Link

32 – Now we need to remove the material where the two components overlap. To do this we will use the **Combine Tool**. Select it under the **Modify Menu**. The **Target Body** is the **Link** and the **Bracket** is the **Tool Bodies**. Make sure to change the **Operation** to **Cut** and check the **Keep Tools** box. Click **OK** to remove the overlap.
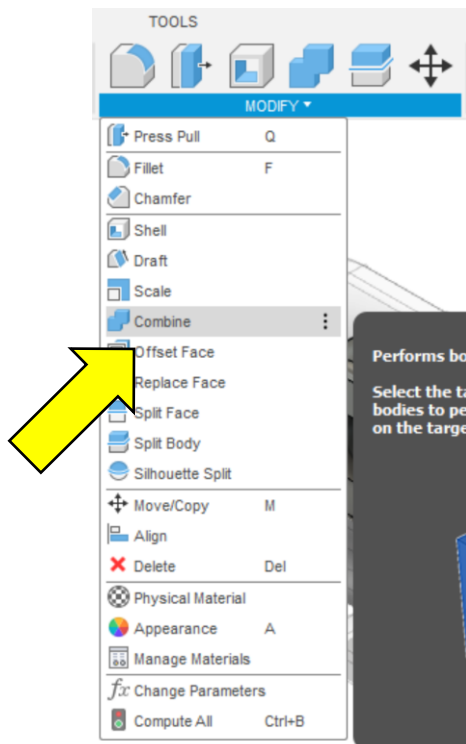
Let's Sketch the Link

The next step will be to add motion to the Link.

Let's Add a Pivot Joint to Allow Motion

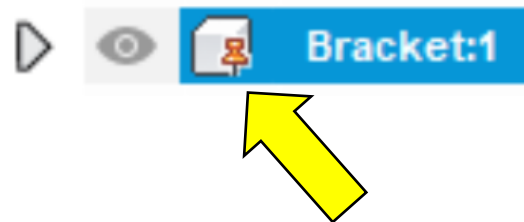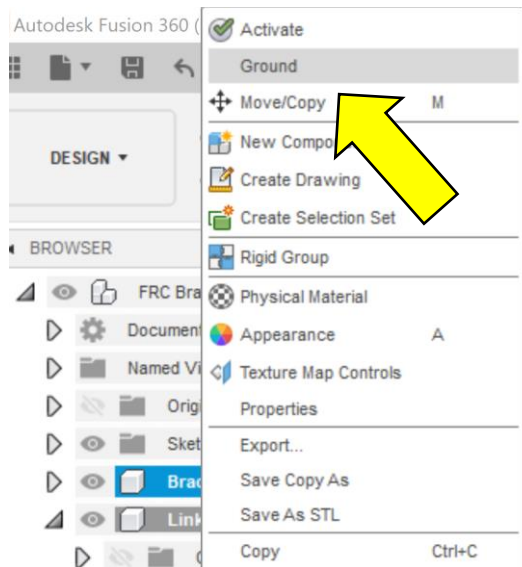33 – To begin we need to **Ground** our **Bracket** to keep it from moving. To do this right click on the Bracket and choose **Ground** from the menu. You will see a push pin appear in the Component cube.
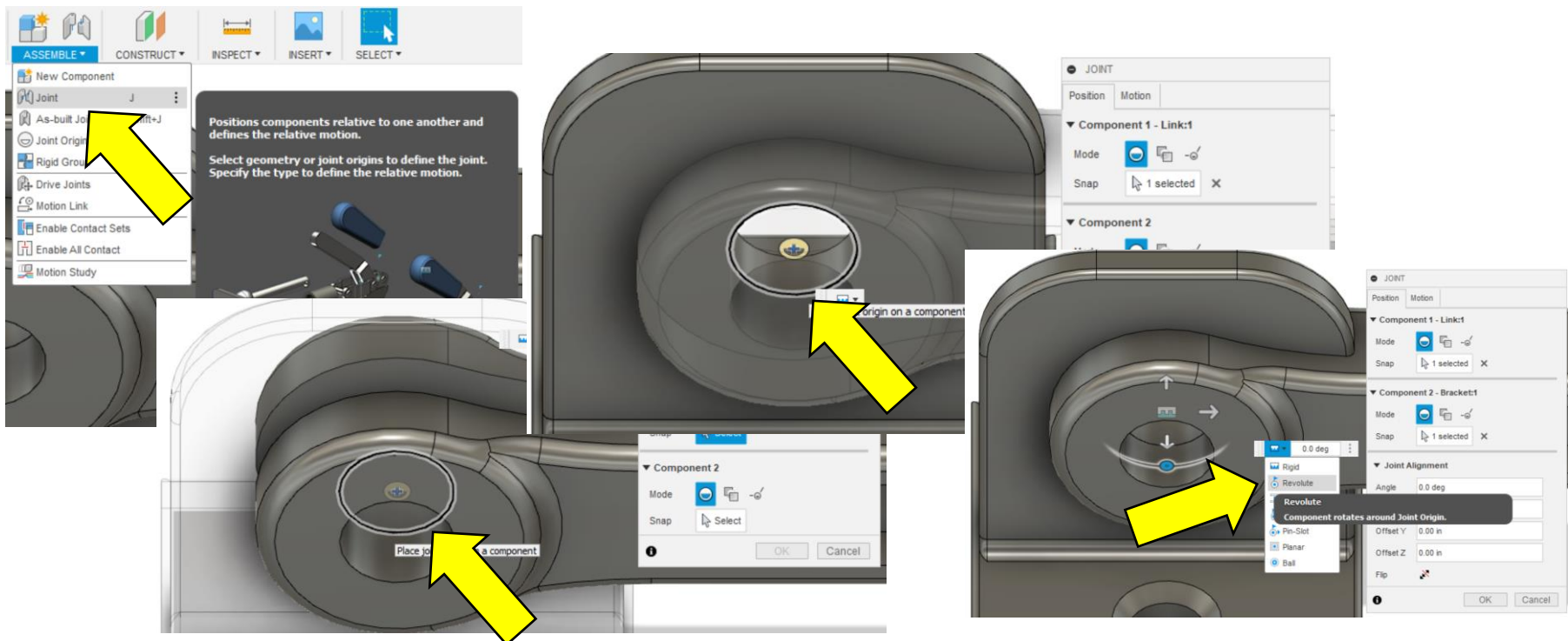
Let's Add a Pivot Joint to Allow Motion

34 – Next we need to create a Joint. Select the **Joint Tool** from the **Assemble Menu**.

35 – The **Joint Tool Window** will appear an the **Bracket** will turn transparent. Choose the inside edge of the **Link**.

36 – The **Link** will turn transparent and **Bracket** will turn solid. Choose the outside edge of the **Bracket**.

37 – Select the **Motion Type** as **Revolute** by clicking on the drop down or using the **Motion Tab** in the **Joint Tool Window**. The **Link** will animate and rotate around the joint. Click **OK**.
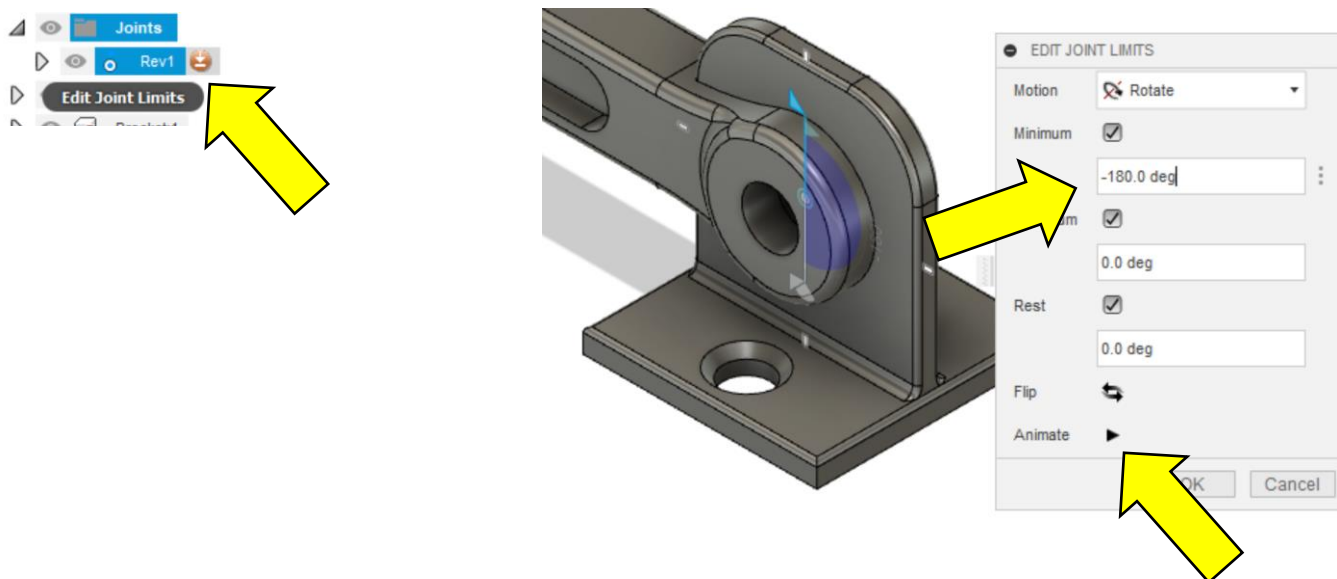
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Let's Add a Pivot Joint to Allow Motion

38 – Click on the Red Down Arrow to edit the Joint Limits. The Edit Joint Limits Window will appear.

39 – Click the 3 check boxes to enable the limits and enter **-180 deg** as a **Min**, **0 deg** as **Max**, and **0 deg** as the **Rest** position.

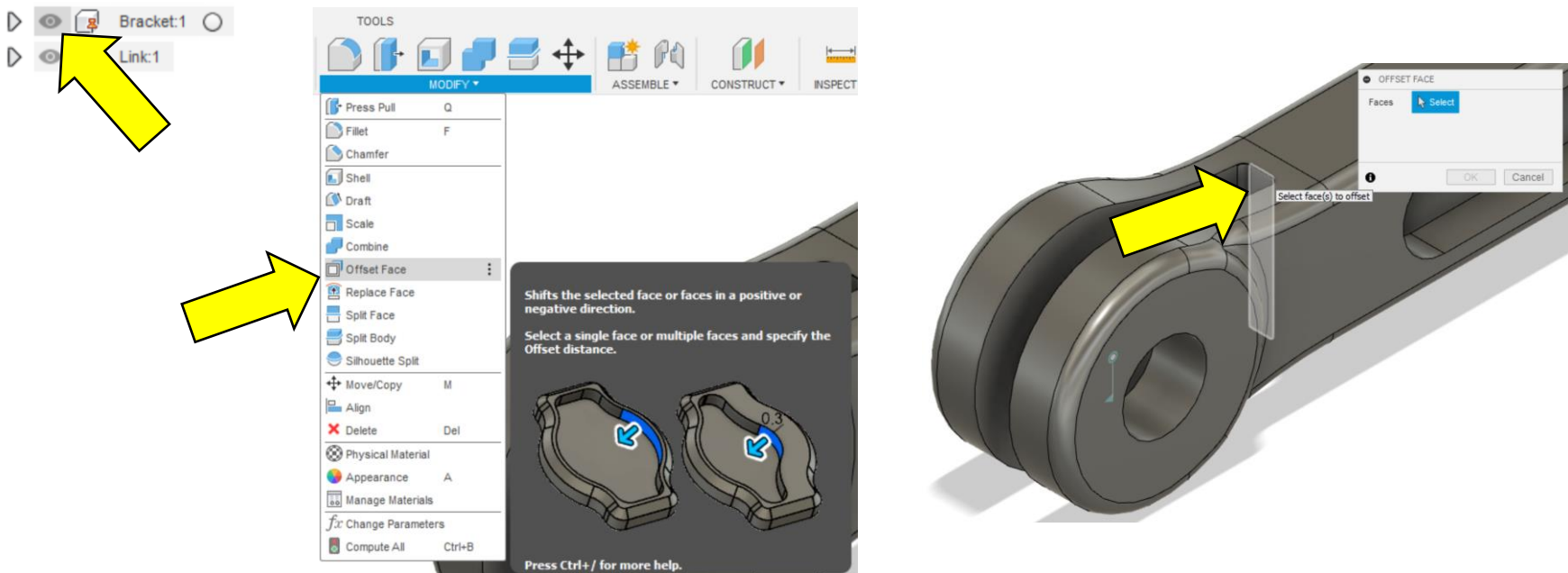40 – Click **Animate** to verify limits are set and if good click **OK**.

Let's Add a Pivot Joint to Allow Motion

41 – We need to cut some clearance in the **Link** to eliminate the interference with the **Bracket**. Start by hiding the **Bracket** by clicking on the **Eye Icon**.

42 – No we need to move the inside surface of the **Link**. Choose the **Offset Face Tool** from the **Modify Menu**.

43 – Click on the surface that is in interference with the **Bracket** and enter **-.150"** for an offset distance.
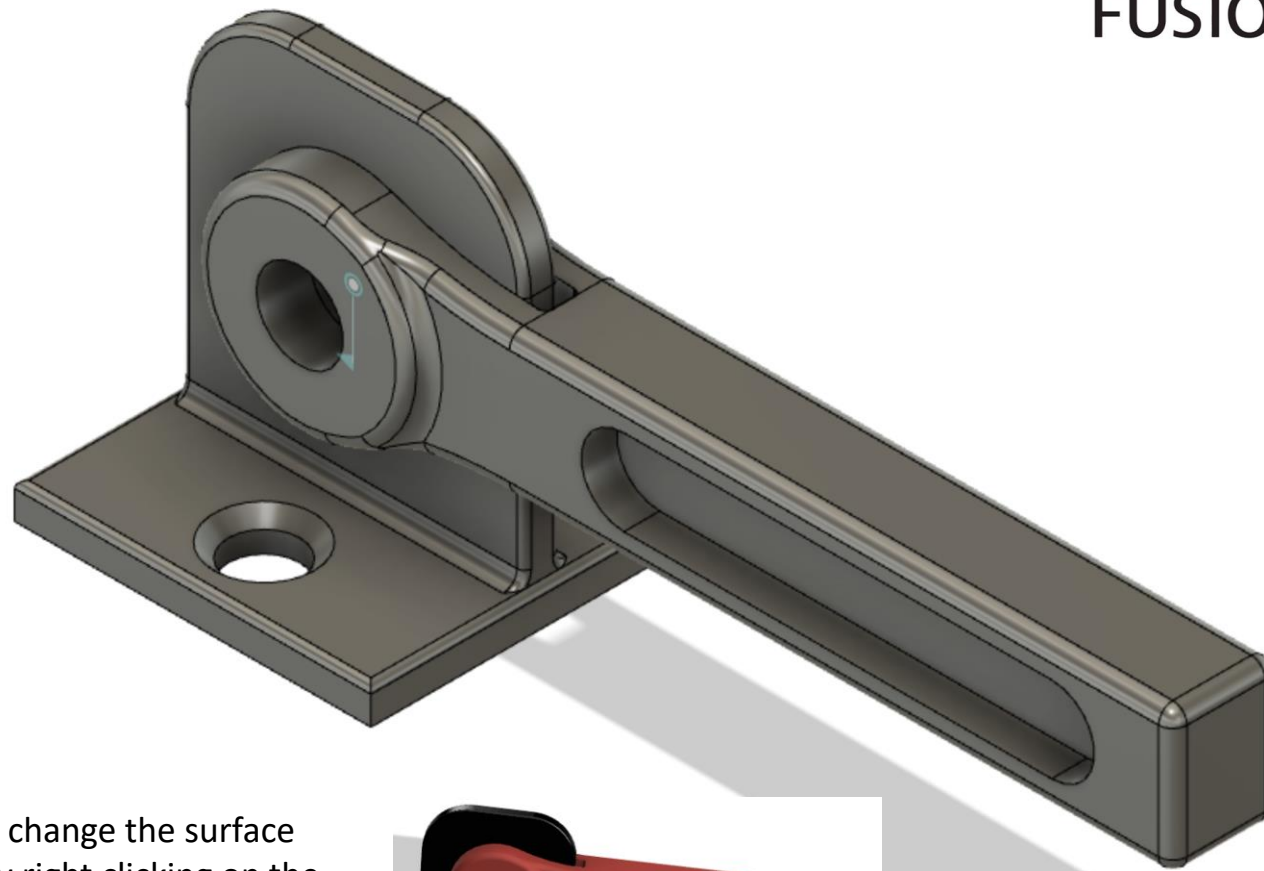
Let's Add a Pivot Joint to Allow Motion

The finished project with motion.

Note: You can change the surface appearance by right clicking on the component and chossing Physical Material.