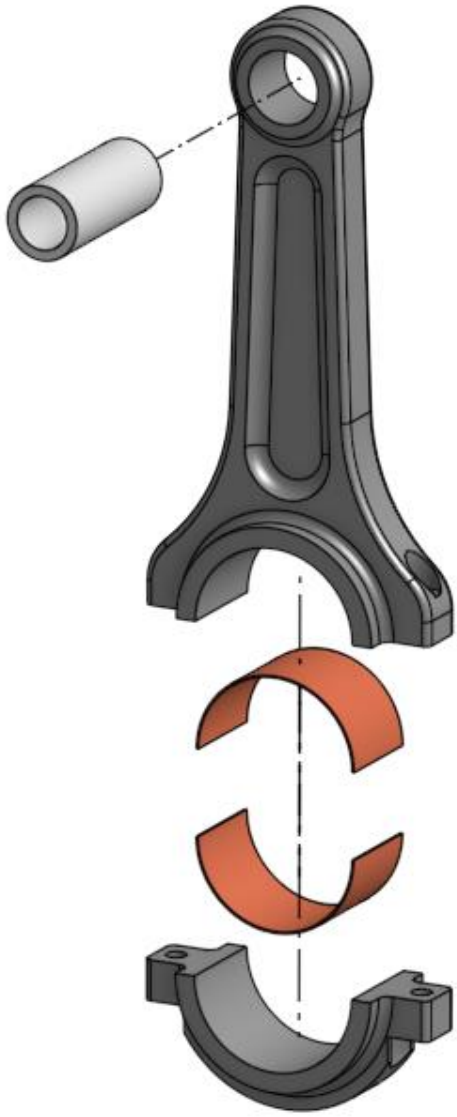
onshape™

Step-by-Step Tutorial

The Connecting Rod – by TooTallToby

In this 30-minute tutorial we will cover:

- Time saving shortcuts
- The SPLIT command
- Modeling for Symmetry
- Exploded Assemblies
- …and more!



Target Experience Level:

Intermediate

Some Onshape fundamental experience will be helpful

Toby's Onshape playlist for beginners:

https://www.youtube.com/playlist?list=PLzMIhOgu1Y5dY8DD5vnOPJ0a3d8F2TWHe

NOTE: This document is meant to function as a companion to the video tutorial published by TooTallToby in April of 2025 and not as a stand-alone document.

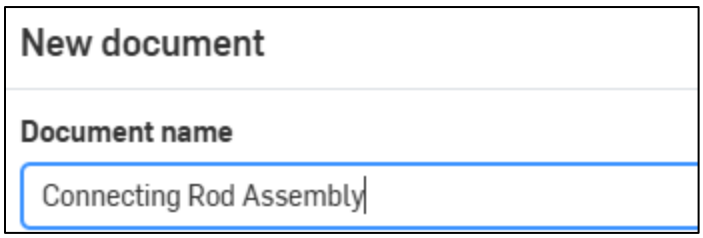
NEW DOCUMENT & SHORTCUT KEYS

1. Visit **Onshape.pro/TooTallToby** to create a new account
   Sign up your engineering/design team to try Onshape Professional for 6 months for free!

   - OR -

   Sign into your existing Onshape account

2. Create a new document in ONSHAPE called “Connecting Rod Assembly”



3. **4 time-saving shortcuts** we’ll be using throughout this tutorial

   a. 

   The “S” key brings up a useful SHORTCUT menu
   Toby’s Video from March 2023 - https://www.youtube.com/watch?v=qdRPOJ9U6KY

   b. **Using Auto Dimensions**
   Toby’s Video from August 2023 - https://www.youtube.com/watch?v=BmBK_9H5BaU

   c. 

   Using “N” for “Normal To”
   Pressing N changes the sketch plan to be perpendicular to your line of sight (aka “Normal To”)

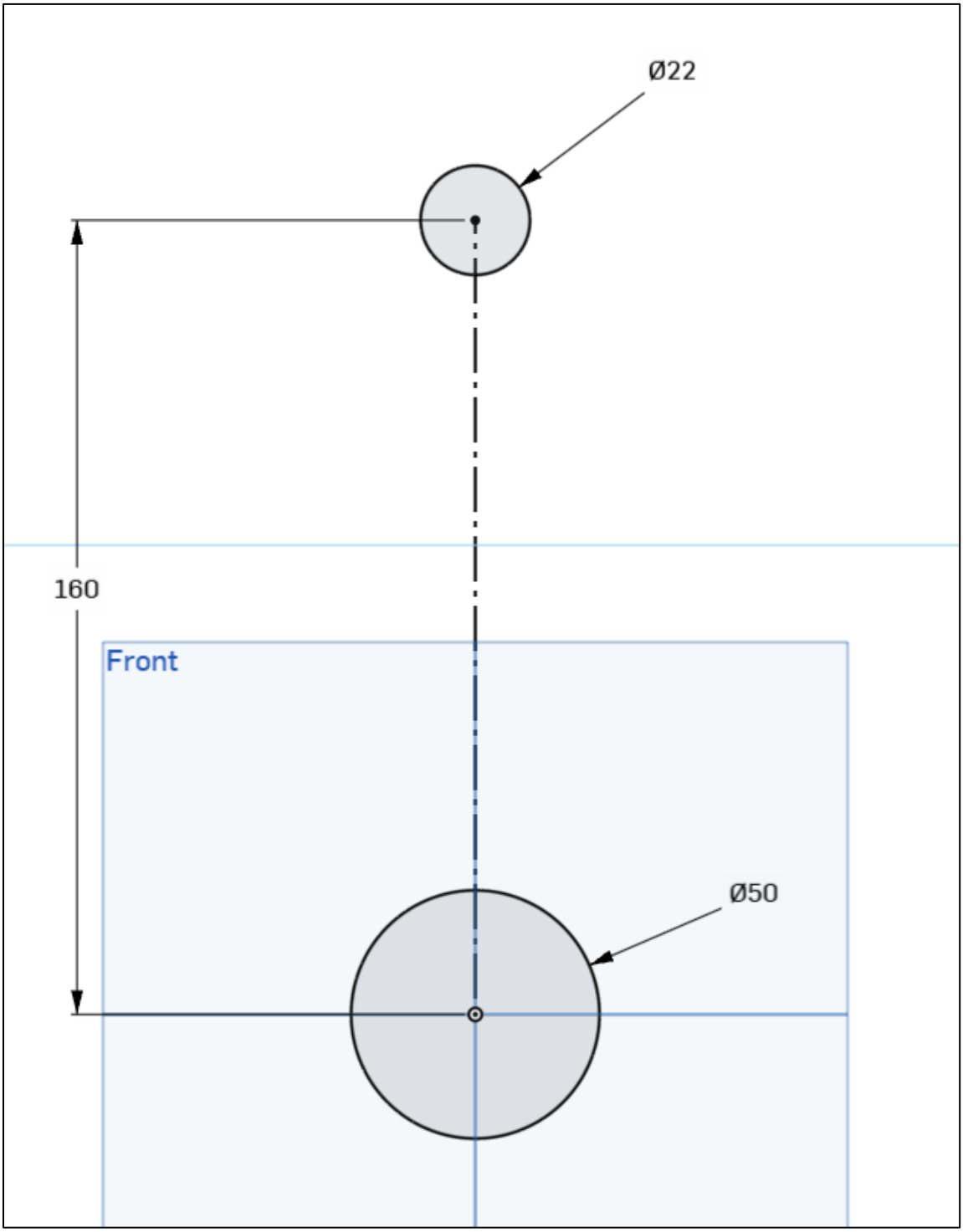
   d. 

   Using “Q” to toggle for construction geometry
   Toby’s Video from May 2023 - https://www.youtube.com/watch?v=mnIwvlvuGaA

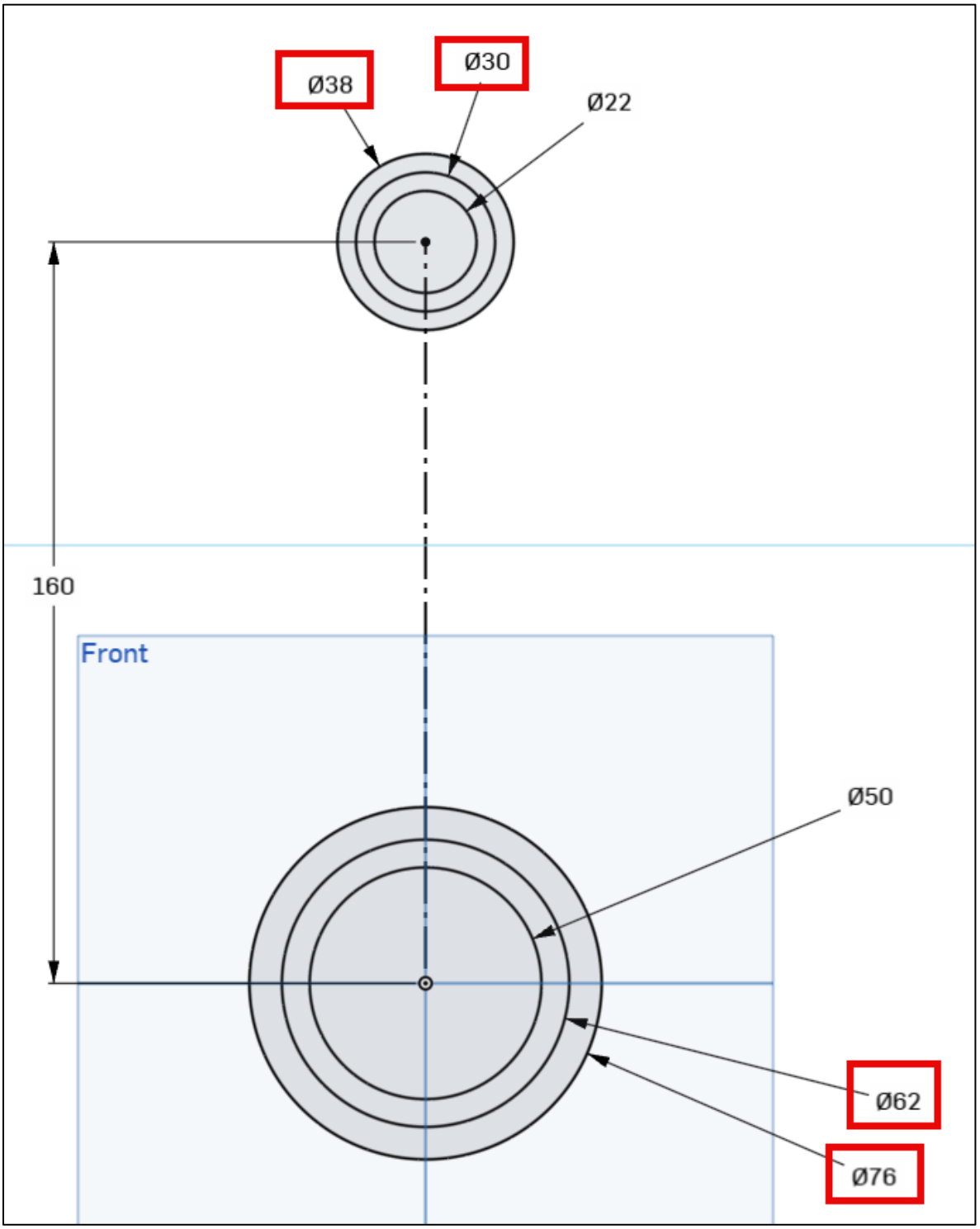
Sketch1

1. Create a **NEW SKETCH** on the **FRONT PLANE**:

    a. Begin with the following geometry:



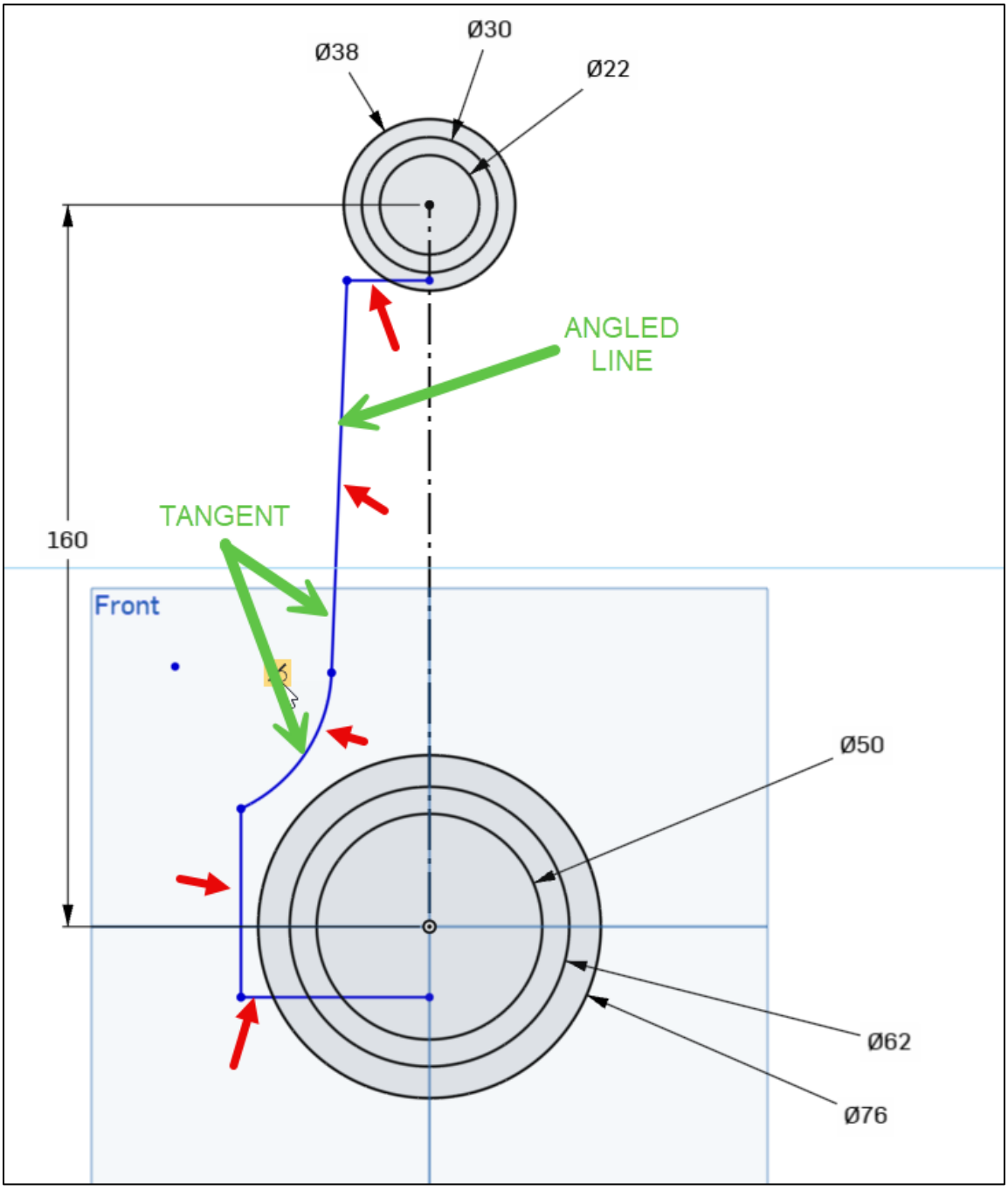
b. Add the following **4 circles**:



c. **Add these 4 lines and 1 arc**.

Make the arc tangent to the angled line.

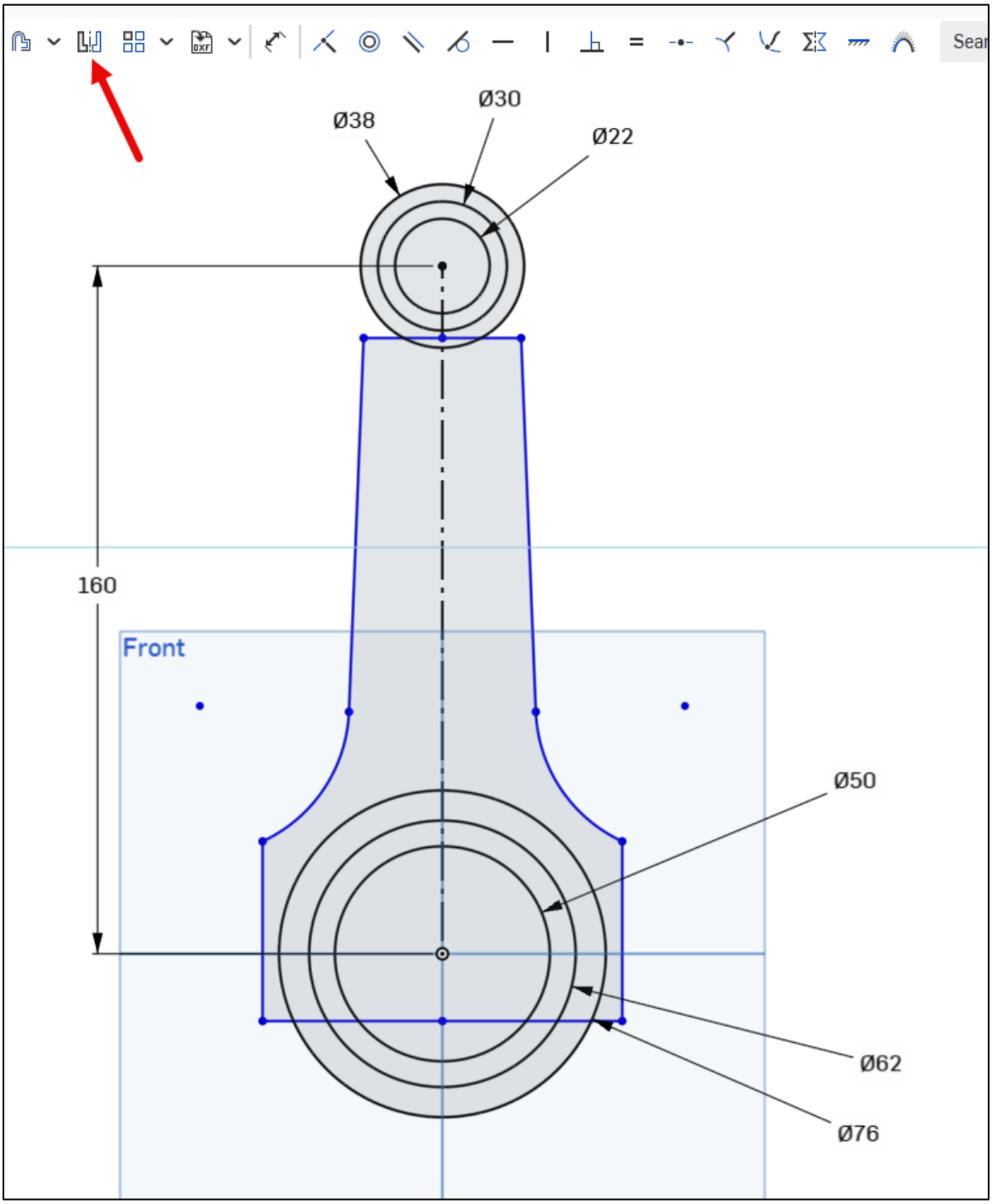
The remaining lines are horizontal and vertical.



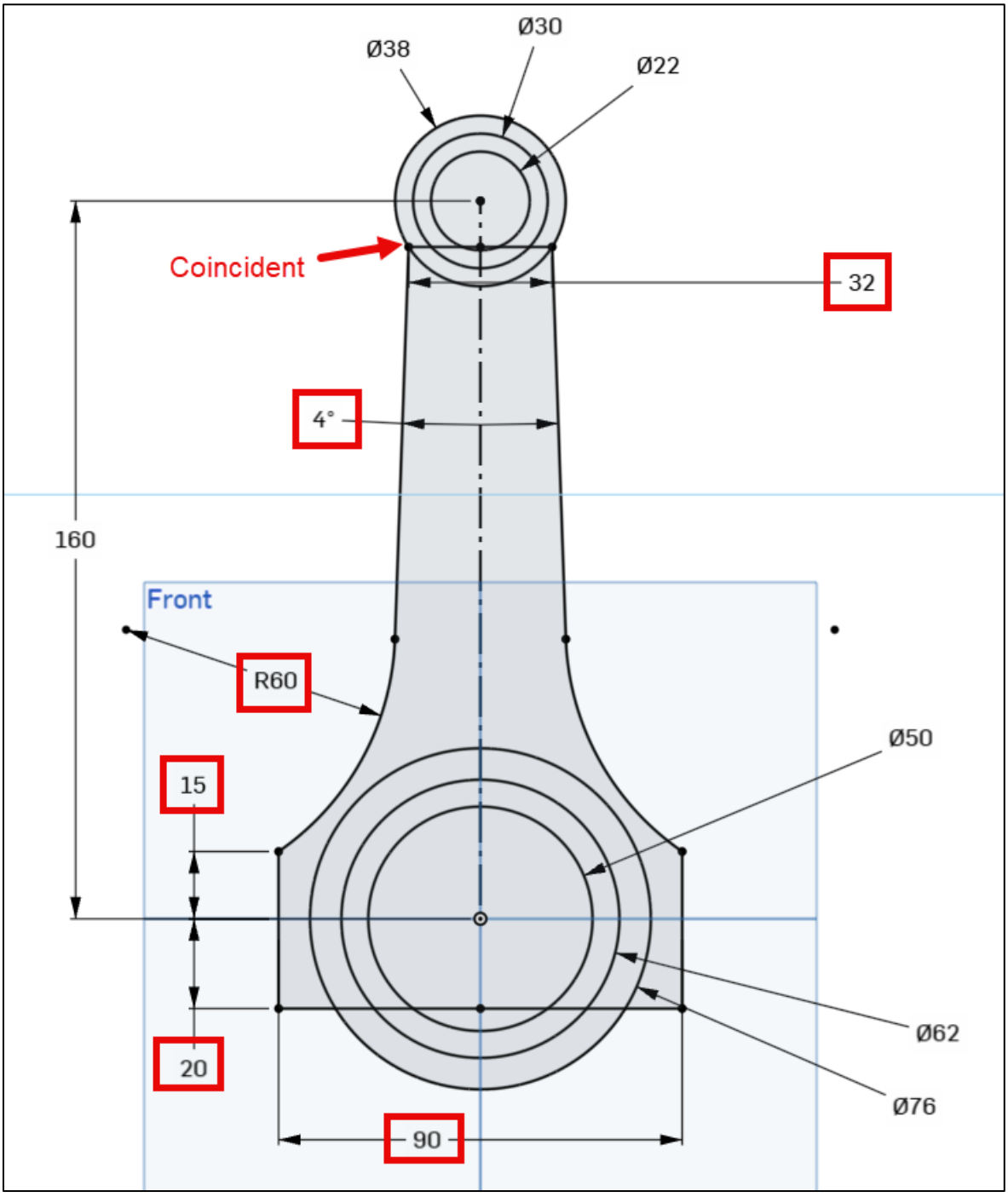
PRO TIP:

Using the **LINE-ARC-LINE** workflow: https://www.youtube.com/watch?v=gYfyXnAN0Cc

d. Use **SKETCH MIRROR** to mirror the 4 lines and he arc:



e. Add these **6 DIMENSIONS** and add this **CONICIDENT RELATIONSHP**

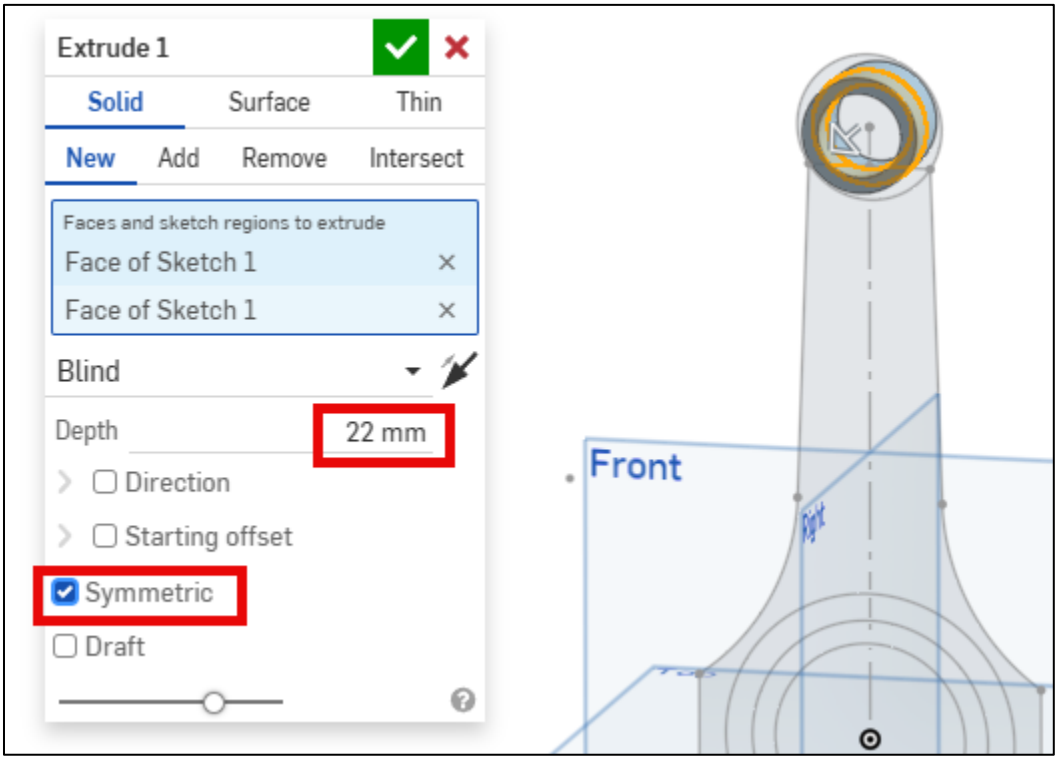


f. Use the GREEN CHECKMARK to exit this sketch:

MULTIPLE EXTRUDES from Sketch 1

2. Use the **EXTRUDE** command 3 times, to extrude the following 3 features:

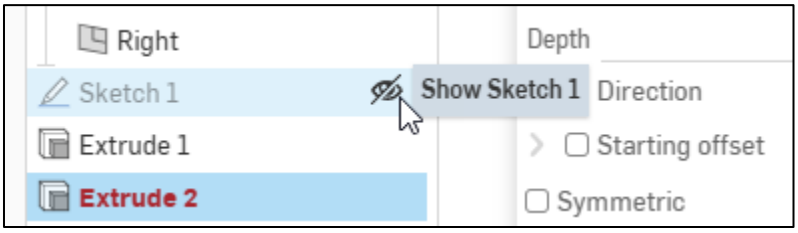
a. **SMALL END**: (22mm circle and 30mm circle)



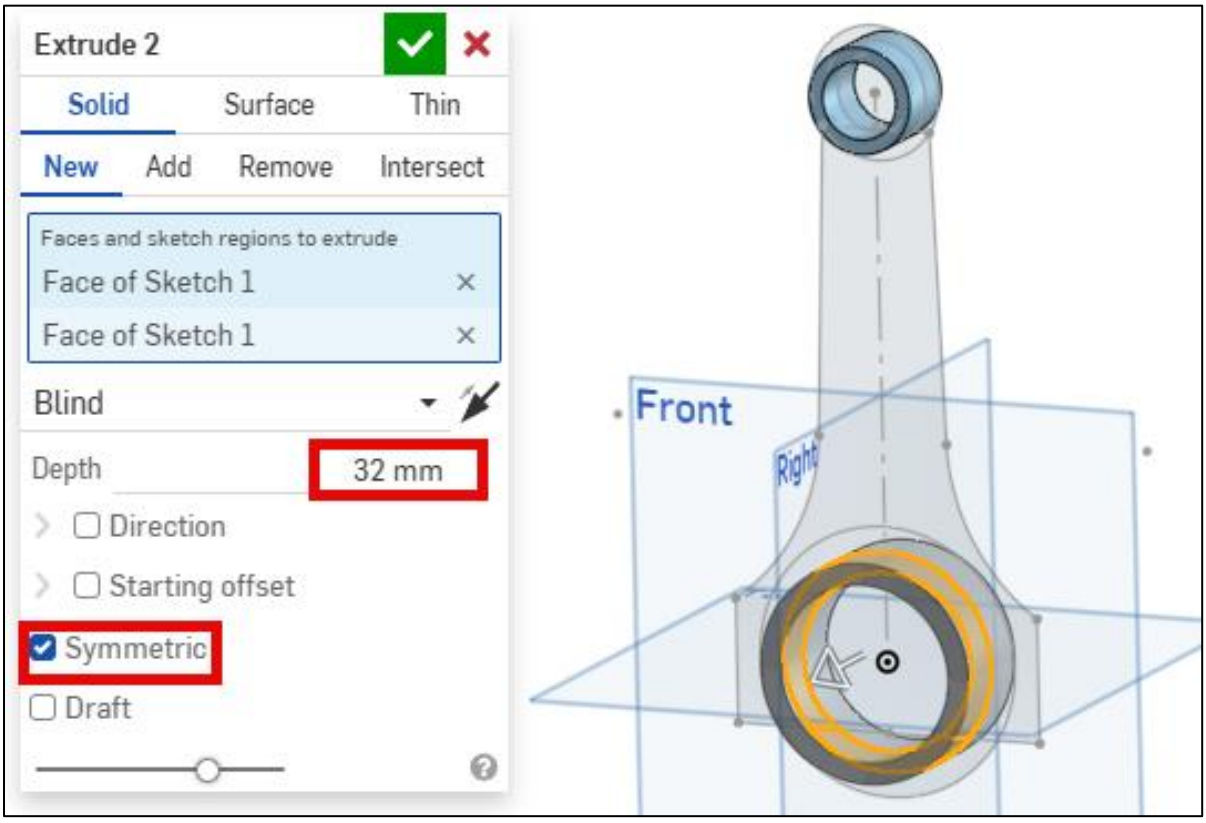
PRO TIP:

Use the **SHIFT+ENTER** to activate the "Repeat Last Command" function

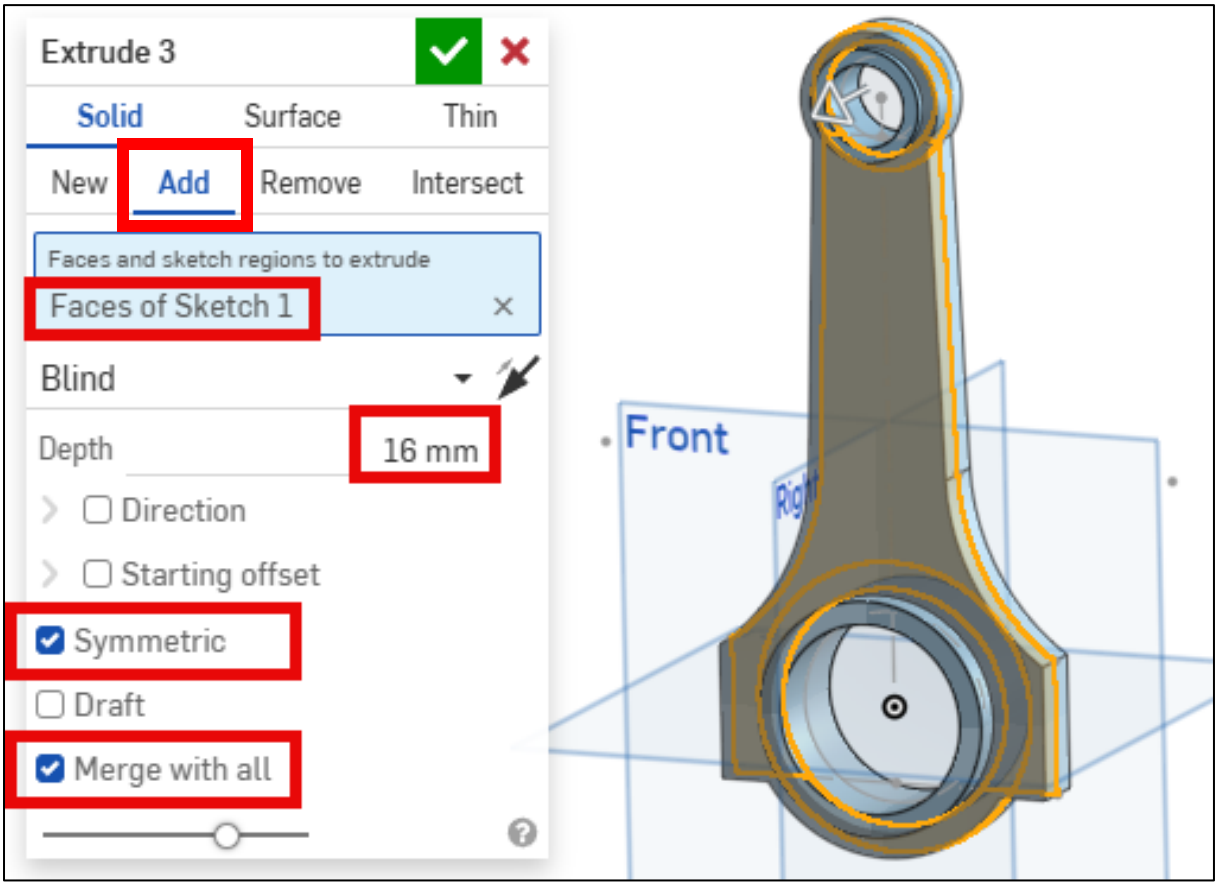
After pressing SHIFT+ENTER, **show Sketch 1** in the tree



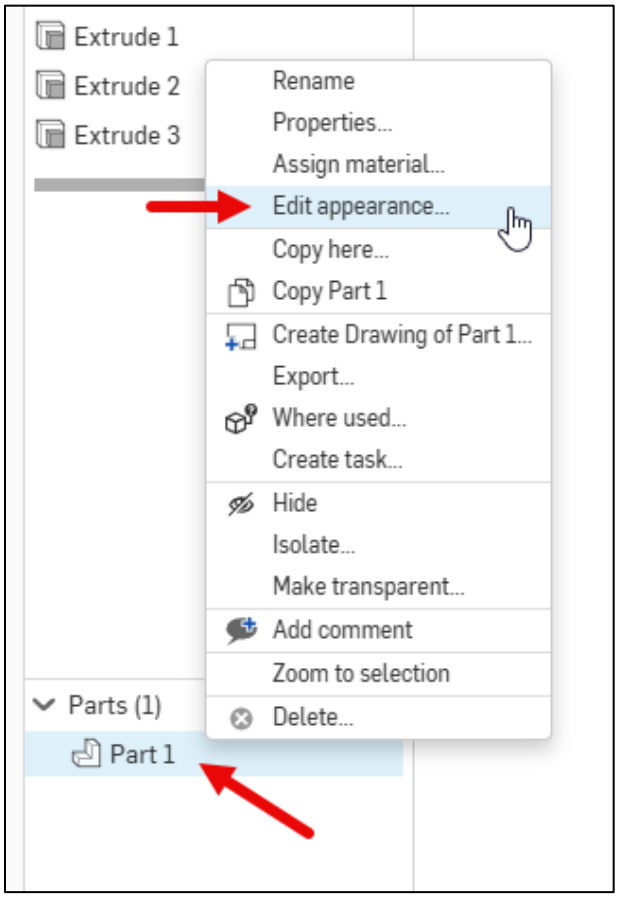
b. **BIG END**: (50mm and 62mm circle)

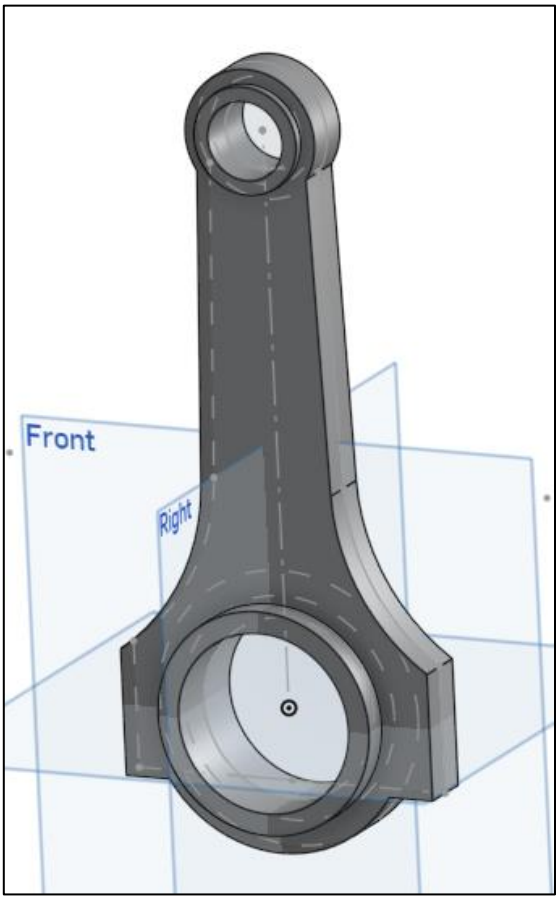


c. **MAIN CONNECTOR ROD** (pick entire sketch from tree):



3. Use **EDIT APPEARANCE** to change the part color to **GREY**:



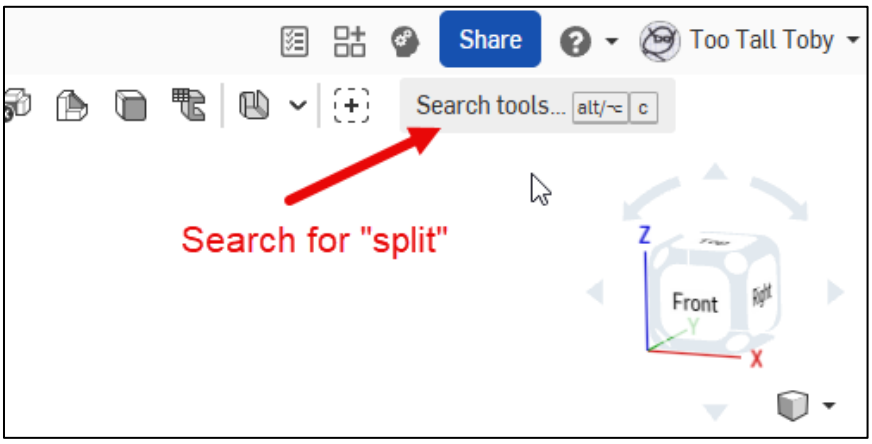


CUT THE MODEL into ¼ of its original shape

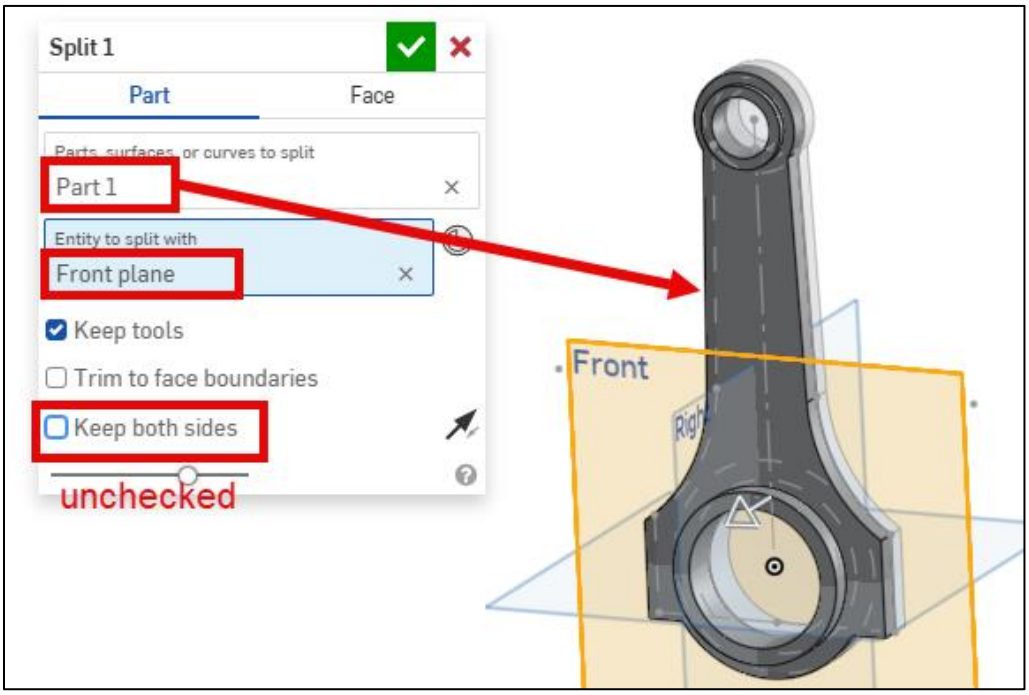
4. To model more efficiently, we now realize that we should cut the model in half, then cut the model in half again. Then, later in the modeling process, we will MIRROR the model.

To cut the mode we will use the **SPLIT** command.

a. **Search for the command** called “split”

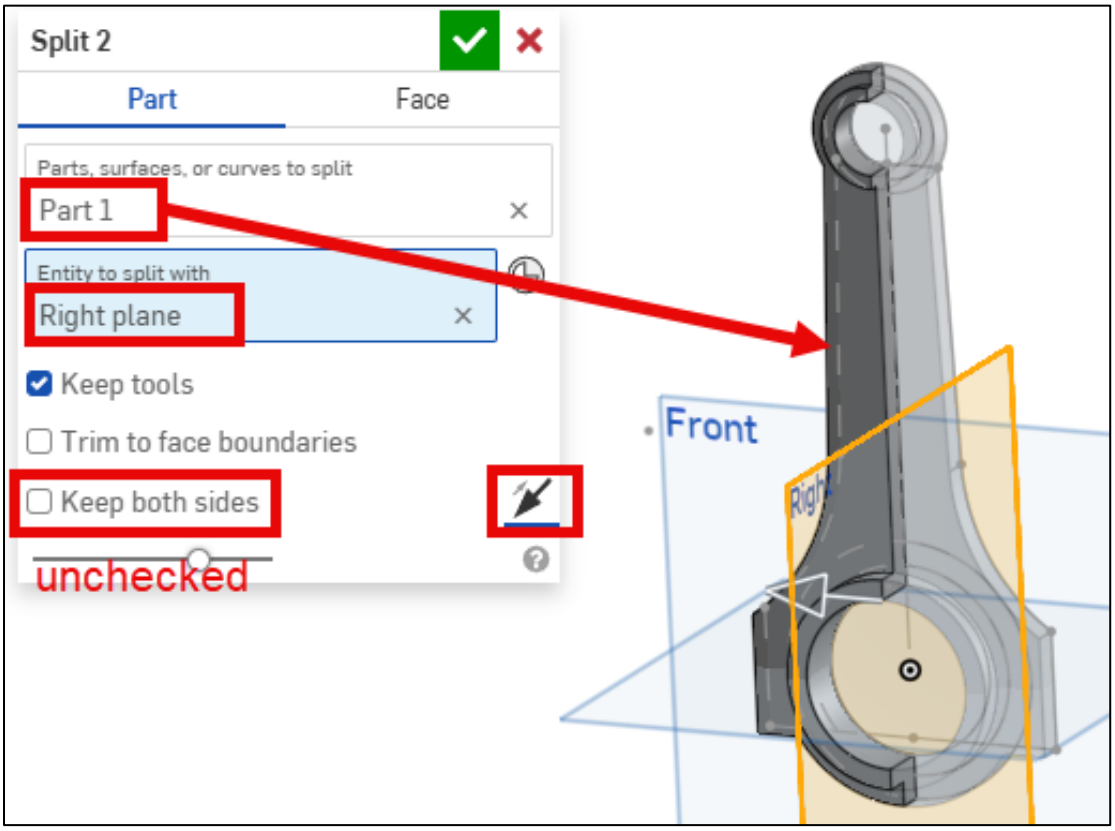


b. Use the following settings:



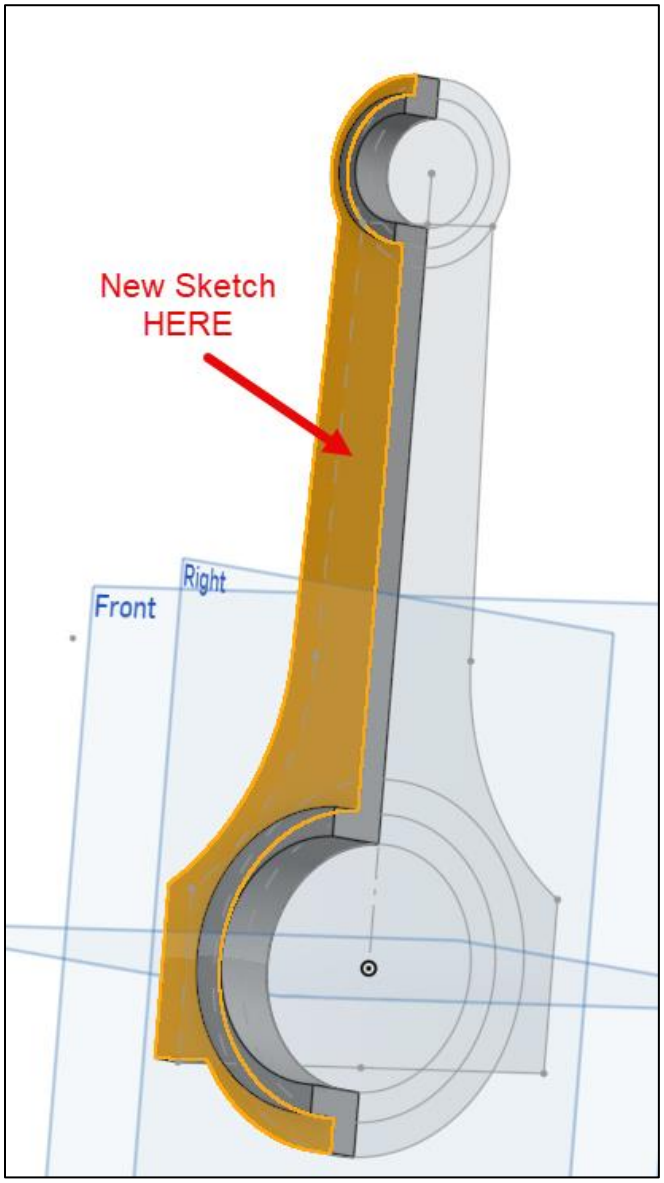
Hit the GREEN CHECKMARK to cut away ½ of the part

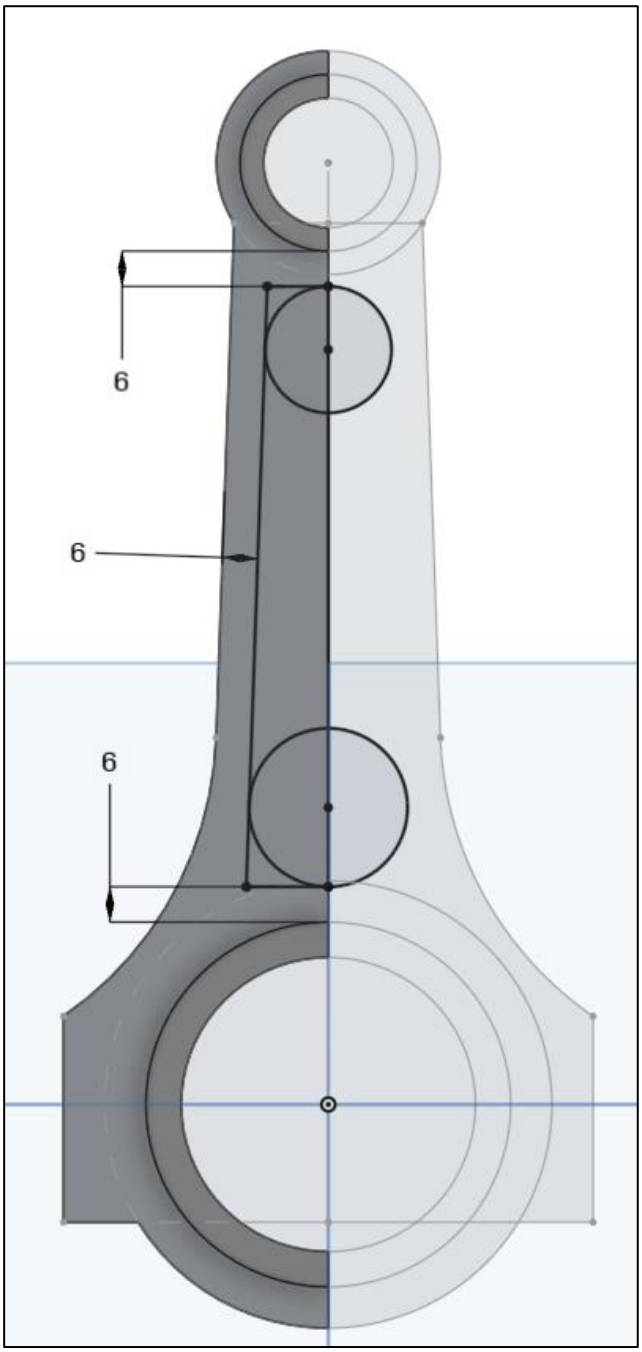
c. Repeat the **SPLIT COMMAND** remove the other half of the part:



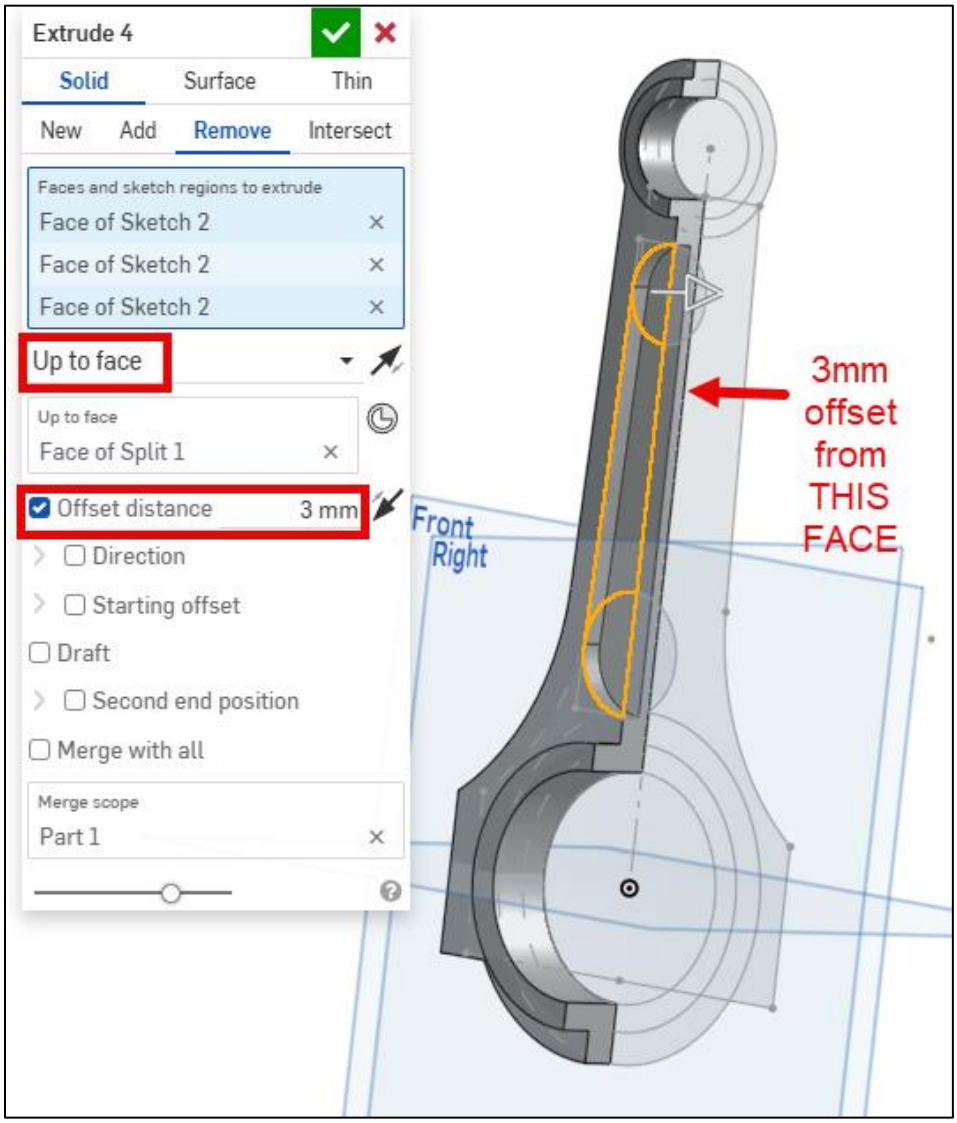
Hit the GREEN CHECKMARK and we are left with ¼ of the original part

5. Create a new sketch for the I-BEAM section, using **4 LINES and 2 CIRCLES** as shown

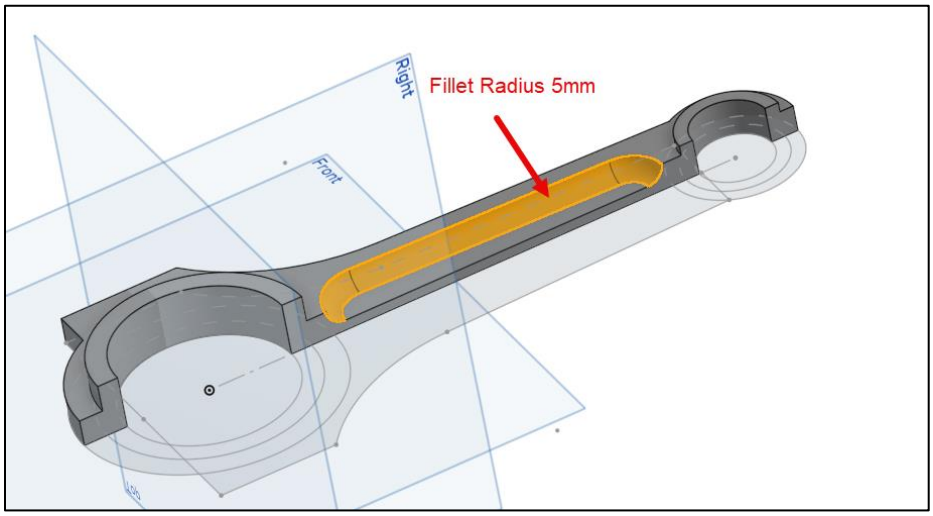




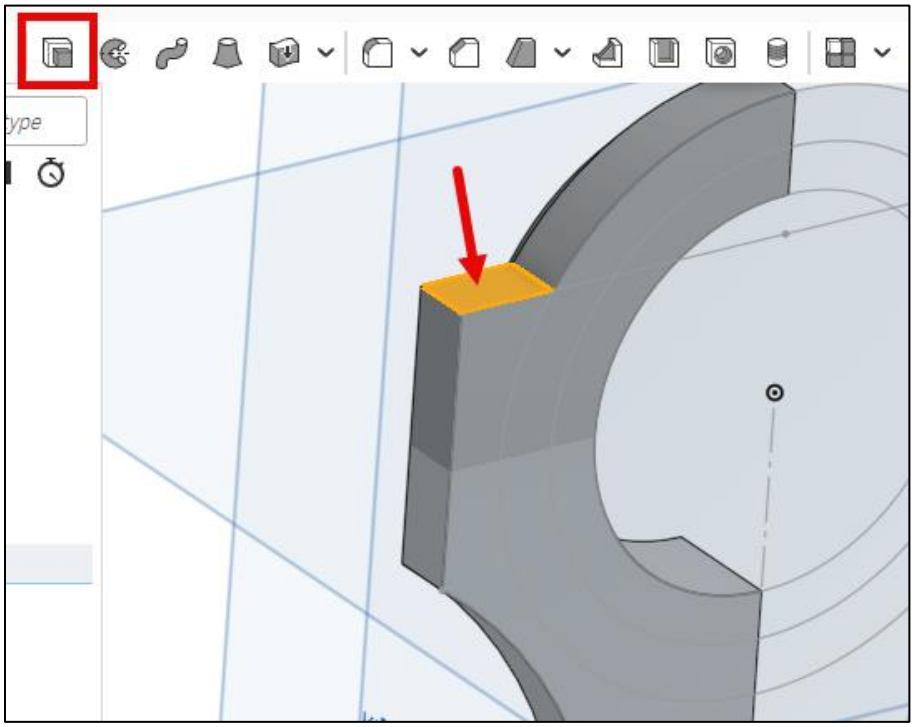
6. Use **EXTRUDE / REMOVE** to create the I-BEAM section as follows:



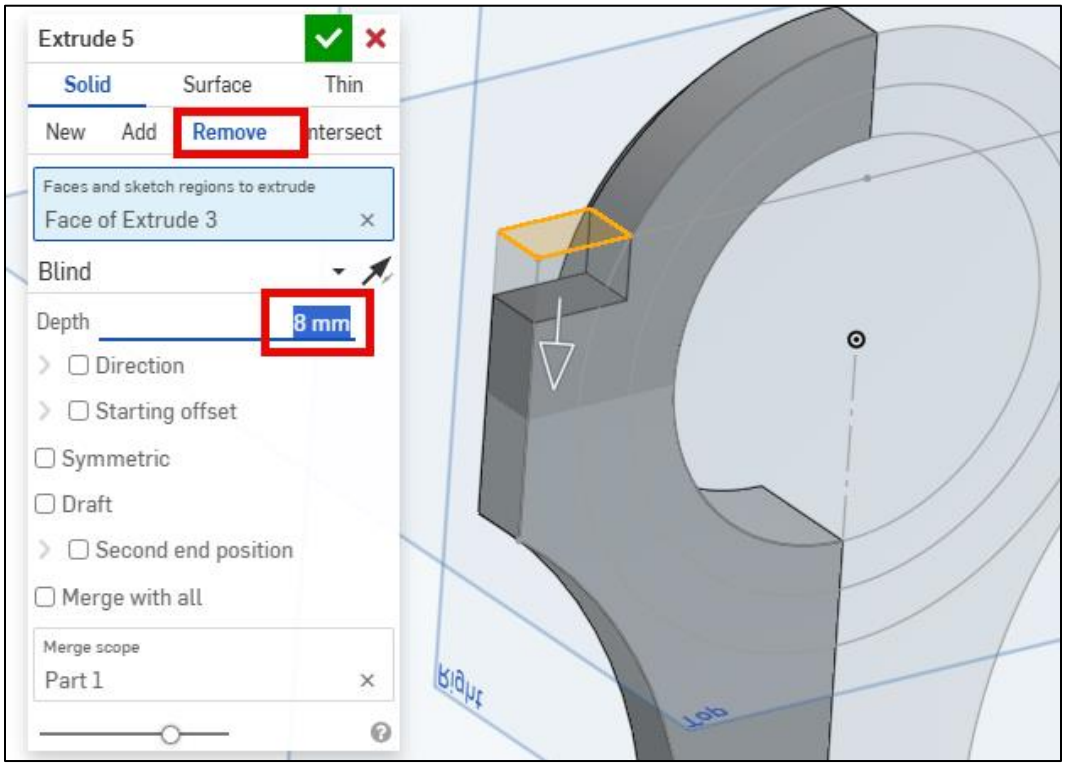
7. Add a **FILLET** to this pocket with a radius **5mm**



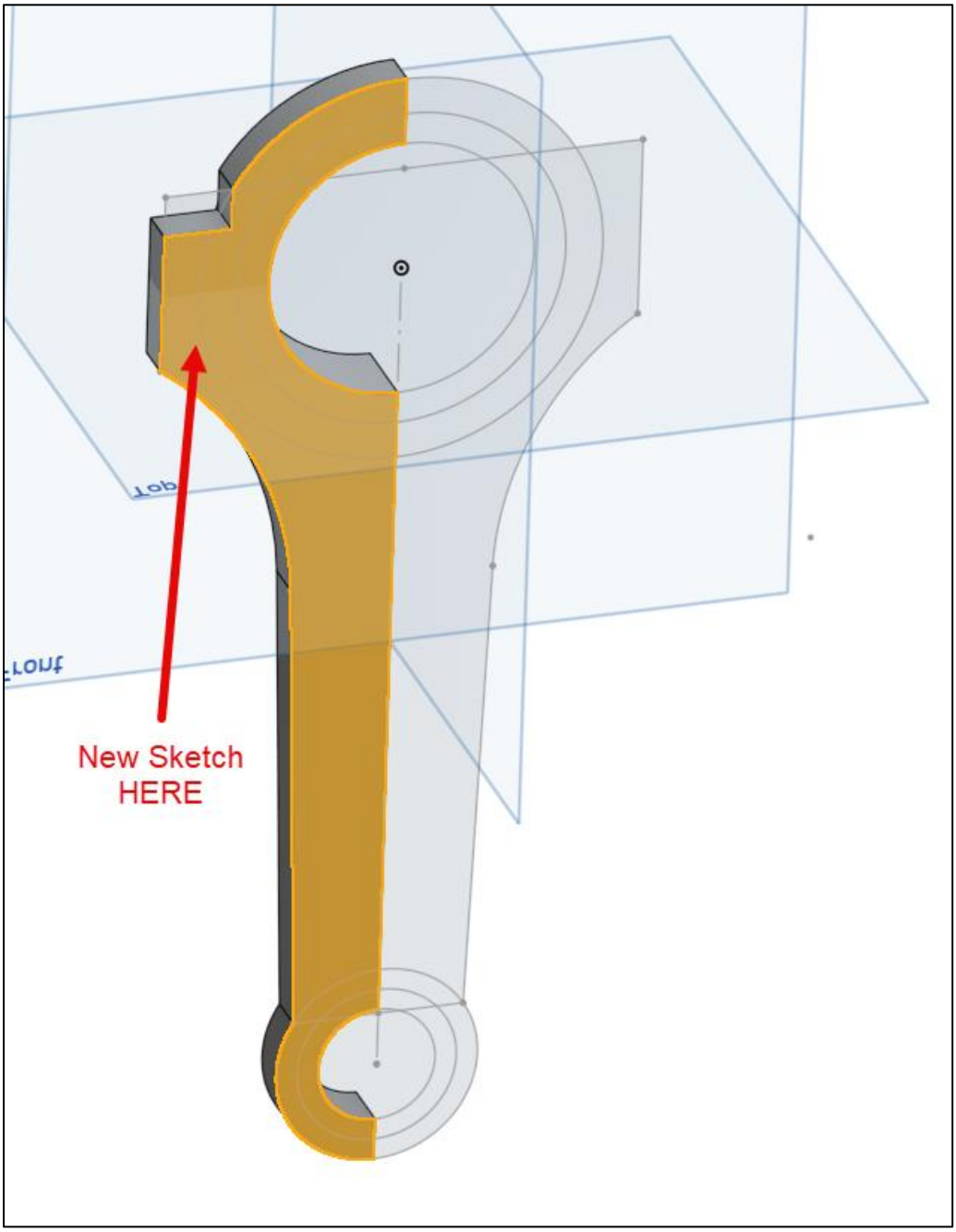
8. Pick this face and choose **EXTRUDE**:



Choose **REMOVE** and set the depth to 15mm



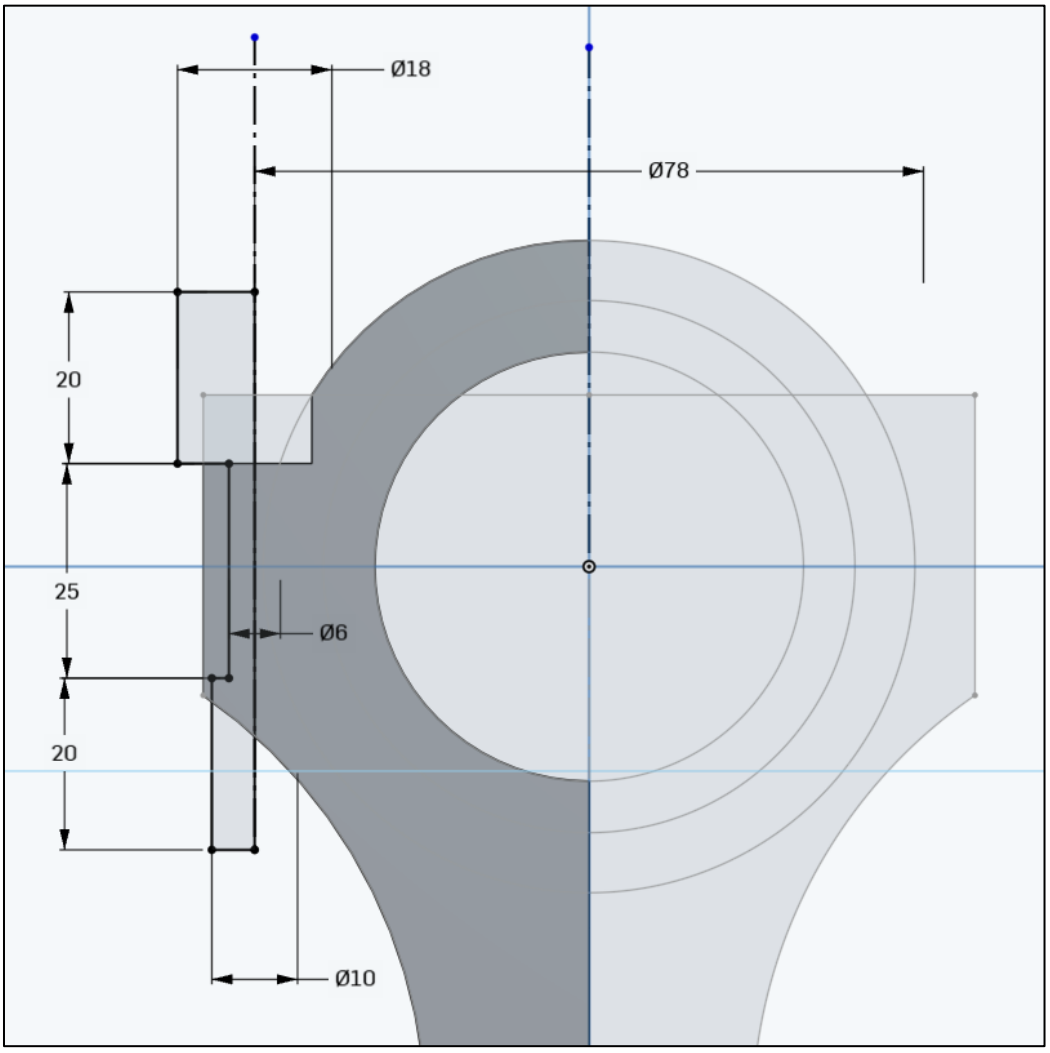
9. **Create a REVOLVED CUT for an M6 Socket Head Cap Screw and Nut**
    a. Begin a **NEW SKETCH** on this face:



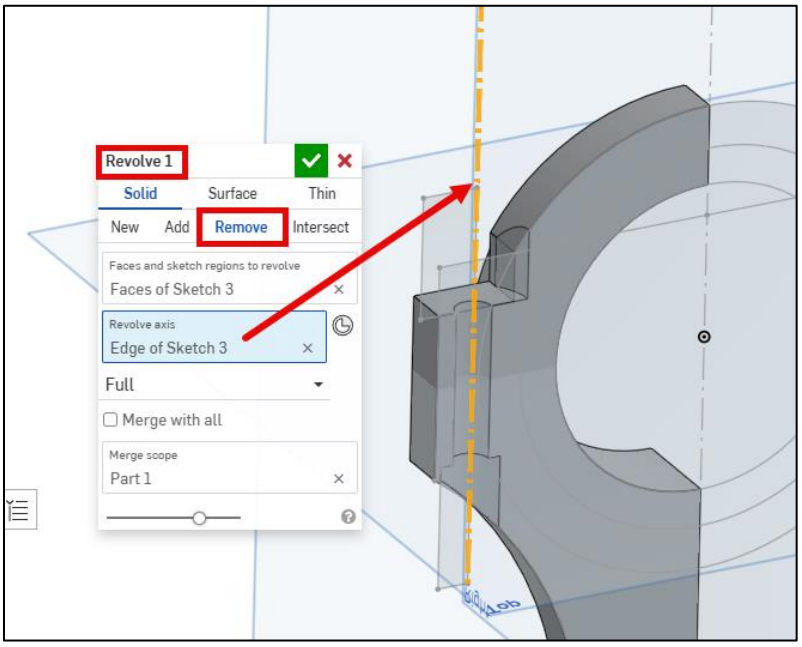
PRO TIP:

Use the  key to activate the “SELECT OTHER” function

b. Create the following sketch geometry using **CENTERLINES and DOUBLED DIMENSIONS:**



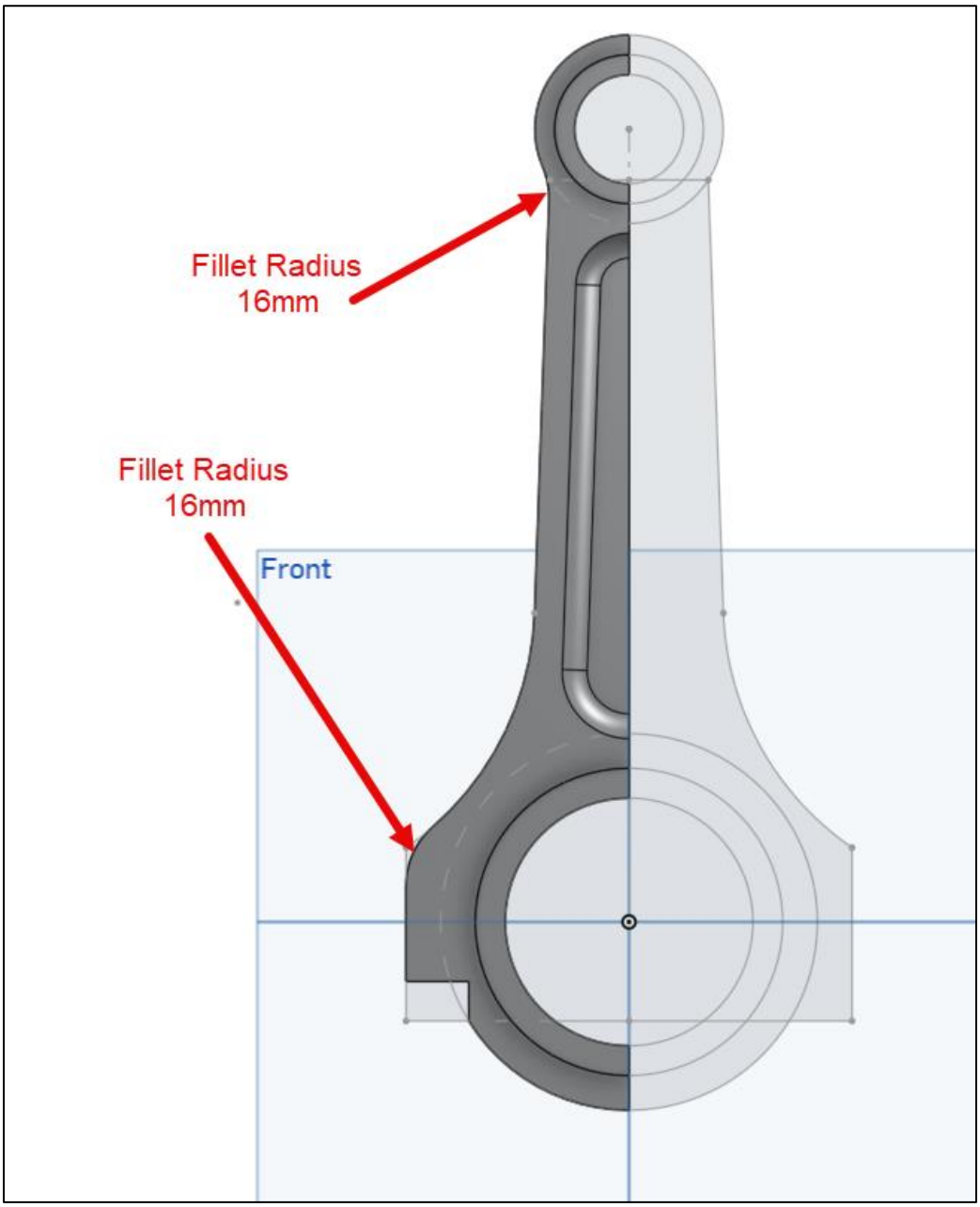
c. Launch the **REVOLVE** command and choose to **REMOVE** the following material:



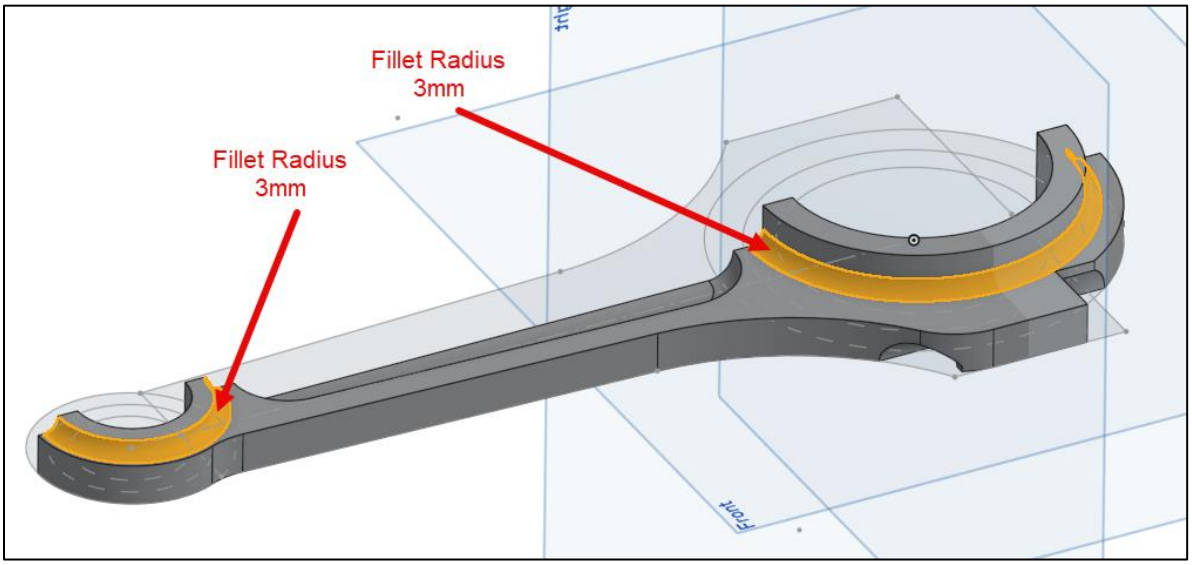
Final Fillets

10. Add the following **3 FILLET FEATURES** to the mode:

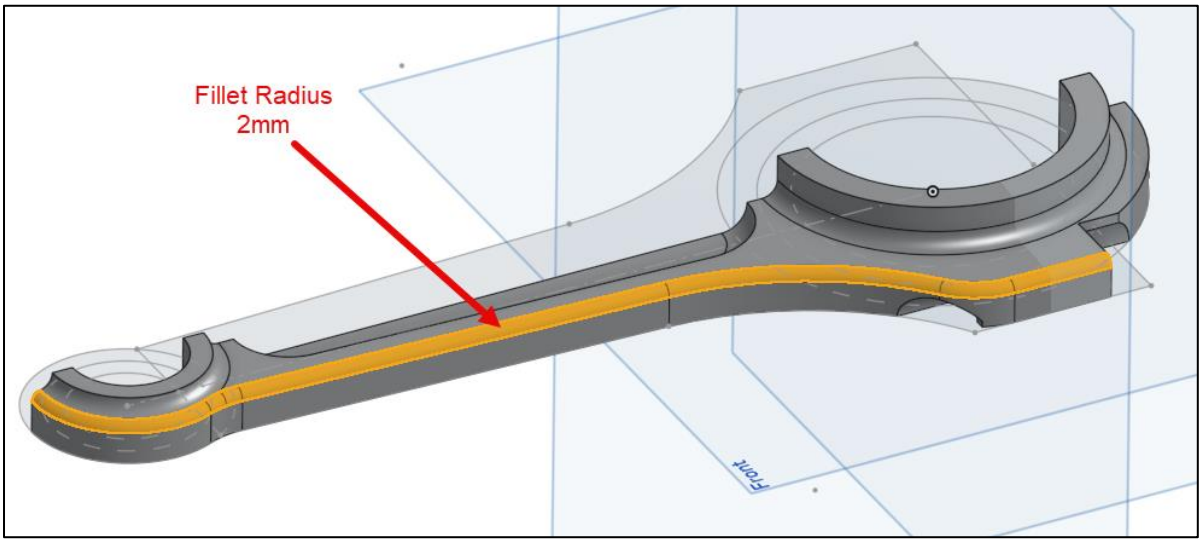
a. Add **R16mm Fillets** to these 2 edges:



b. Add **R3mm Fillets** to these 2 edges:



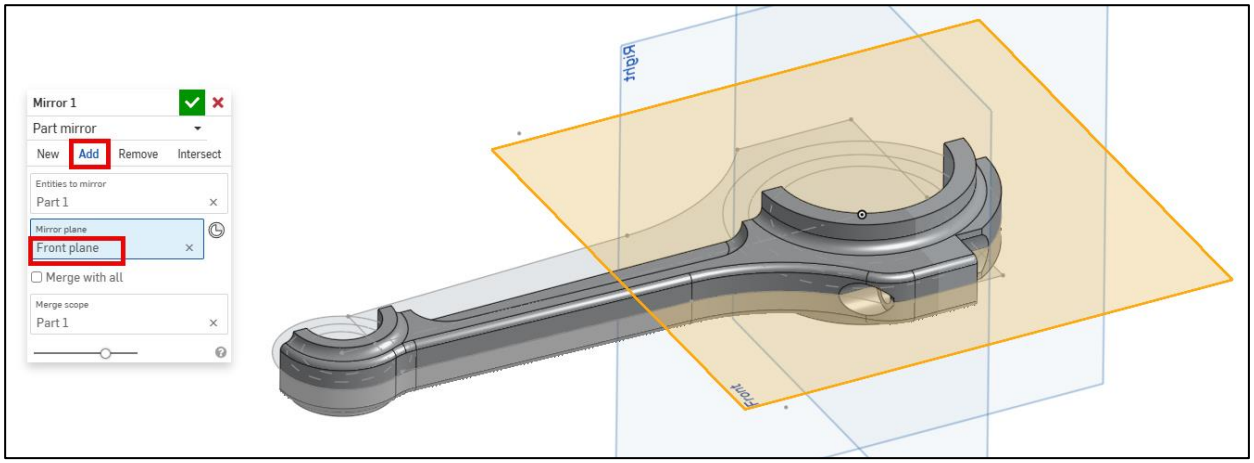
c. Add an **R2mm Fillet** to this edge:



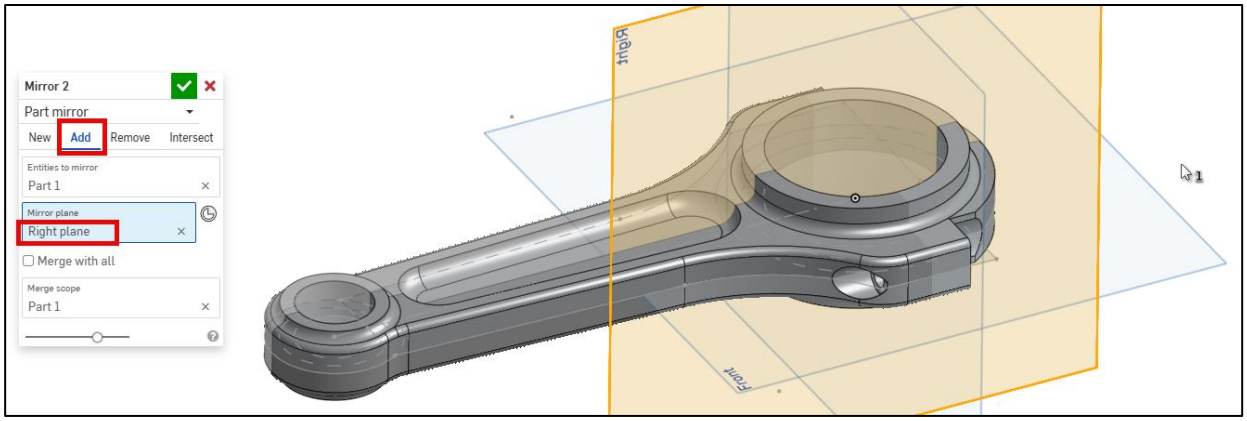
MIRROR PART (twice)

11. Now let's turn the ¼ part back into a FULL MODEL

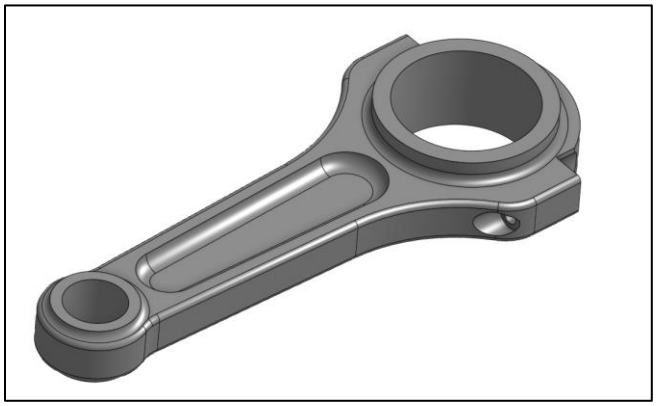
a. **MIRROR PART** once – using the Front Plane



b. **MIRROR PART** again – using the Right Plane



12. Use **SHIFT + P** to hide all planes and reference geometry. Our model looks like this:

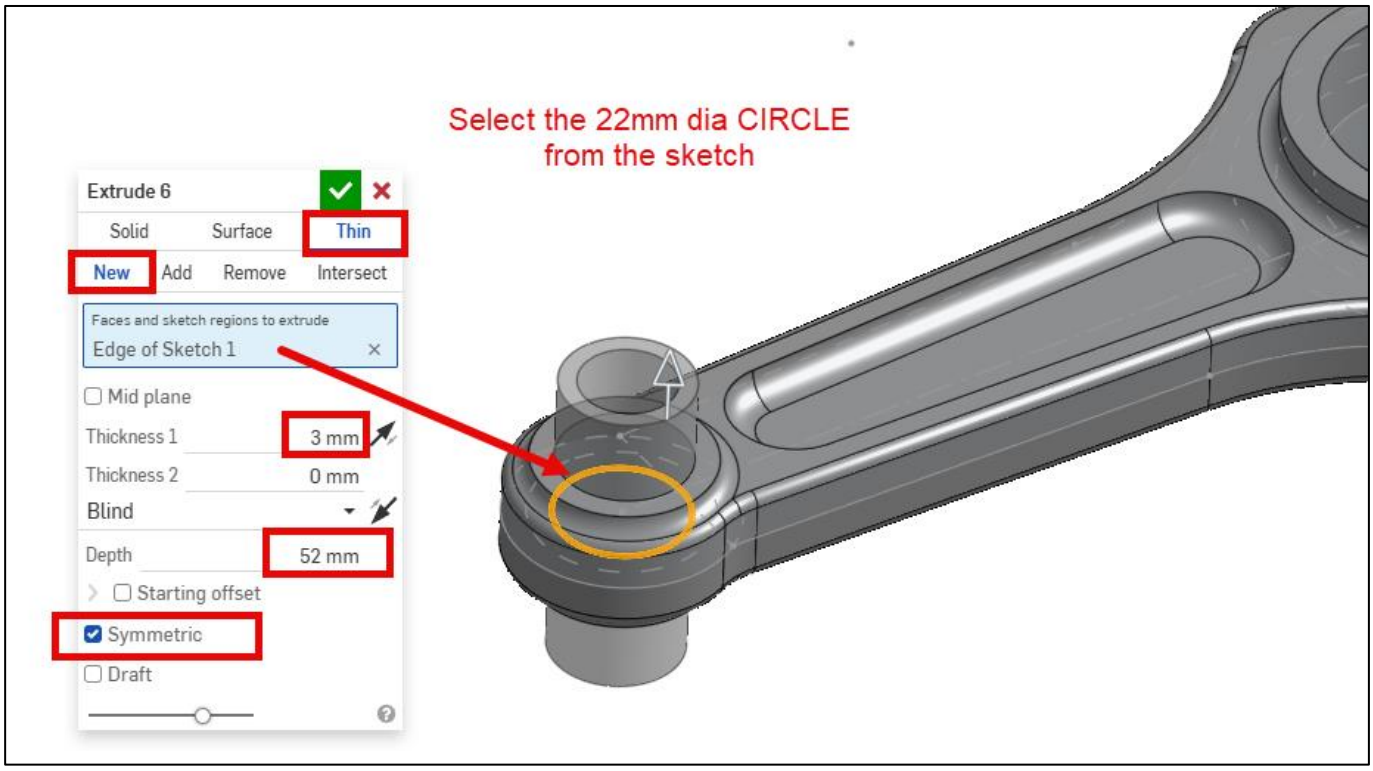


13. We will now create the geometry for the WRIST PIN and the ROD BEARING INSERTS

WRIST PIN:

a. **SHOW** the first sketch in the tree (Sketch1)

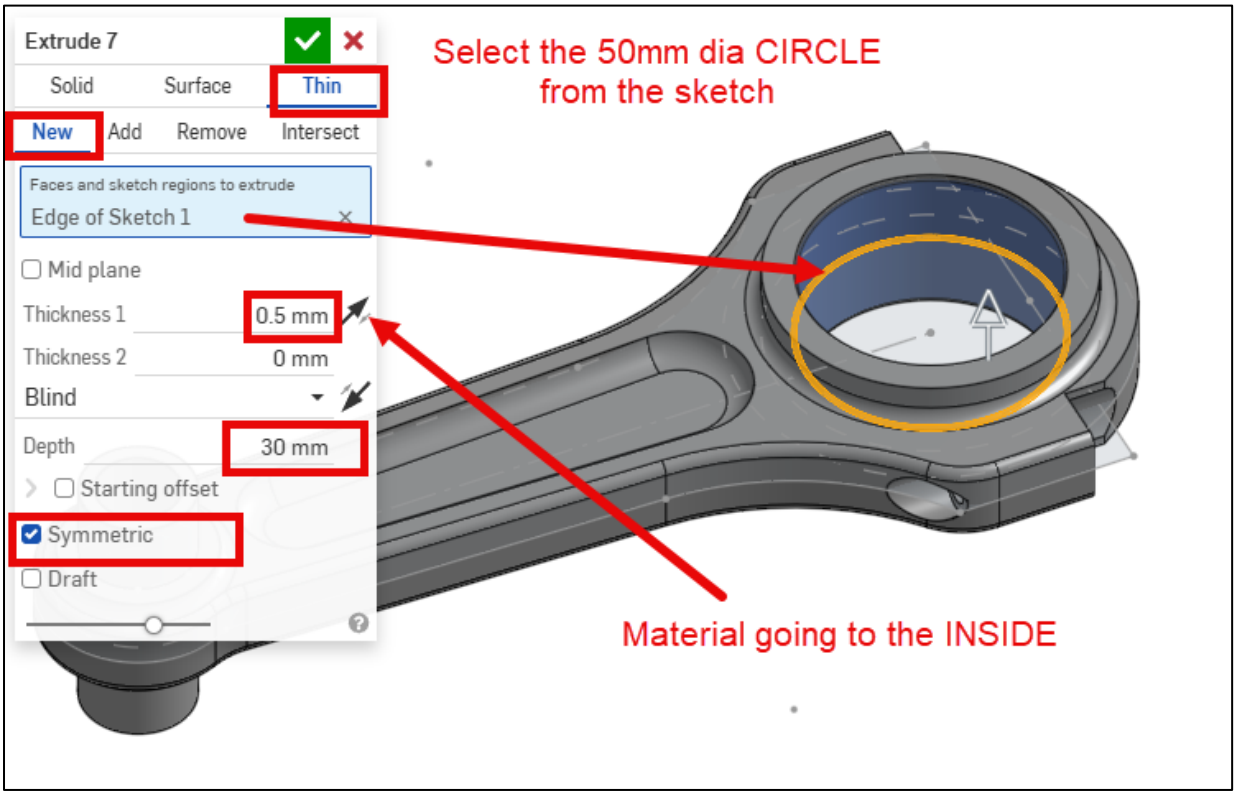
b. Click **EXTRUDE** and choose **THIN**, then choose the following settings:



By using the option for NEW, we're creating a new part in the parts list

ROD BEARING INSERTS:

a. Click **EXTRUDE** and choose **THIN**, then choose the following settings:



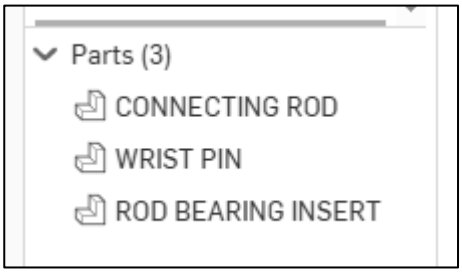
Again, by using the option for NEW, we're creating a new part in the parts list

14. We now have 3 parts in our parts list. Let's rename these to:

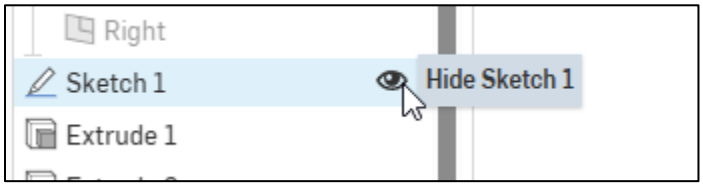
- CONNECTING ROD

- WRIST PIN

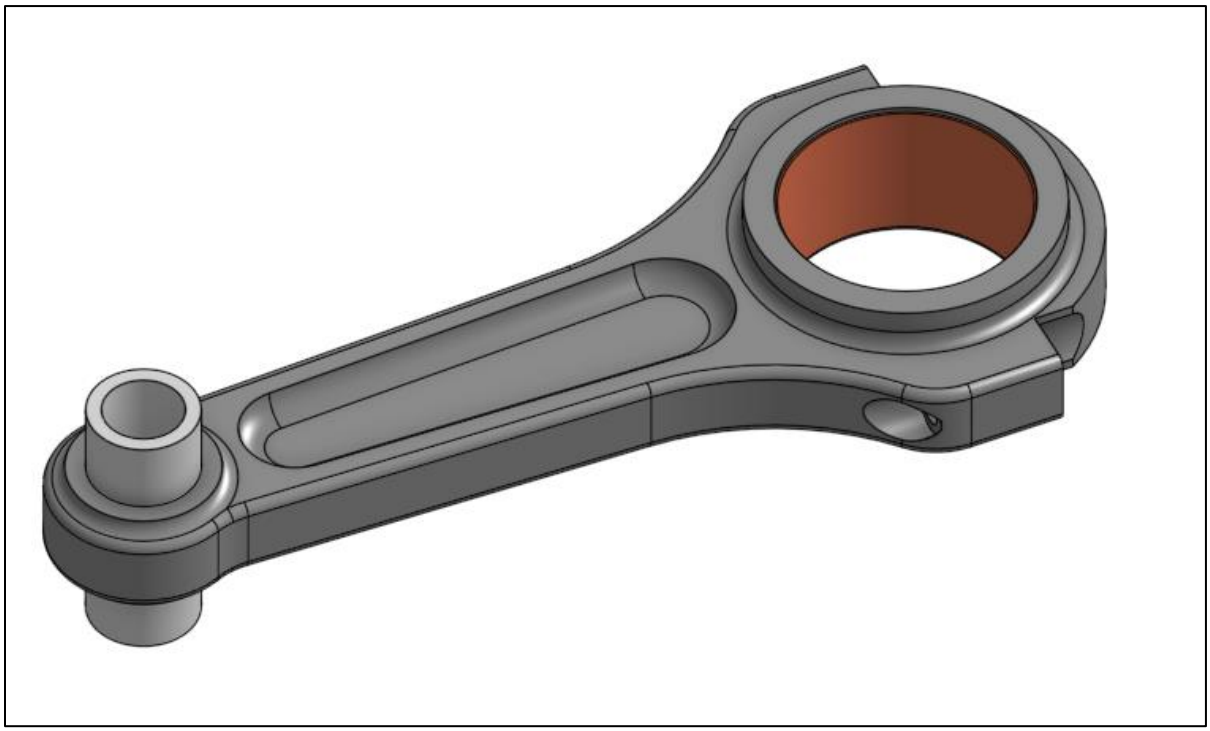
- ROD BEARING INSERT



15. HIDE Sketch1 in the tree



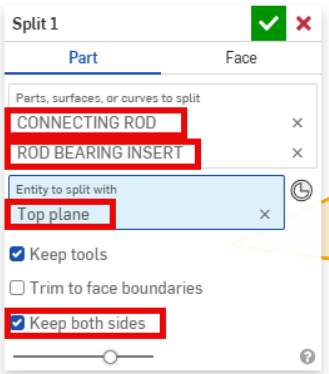
16. Change the appearance of each part as shown below:

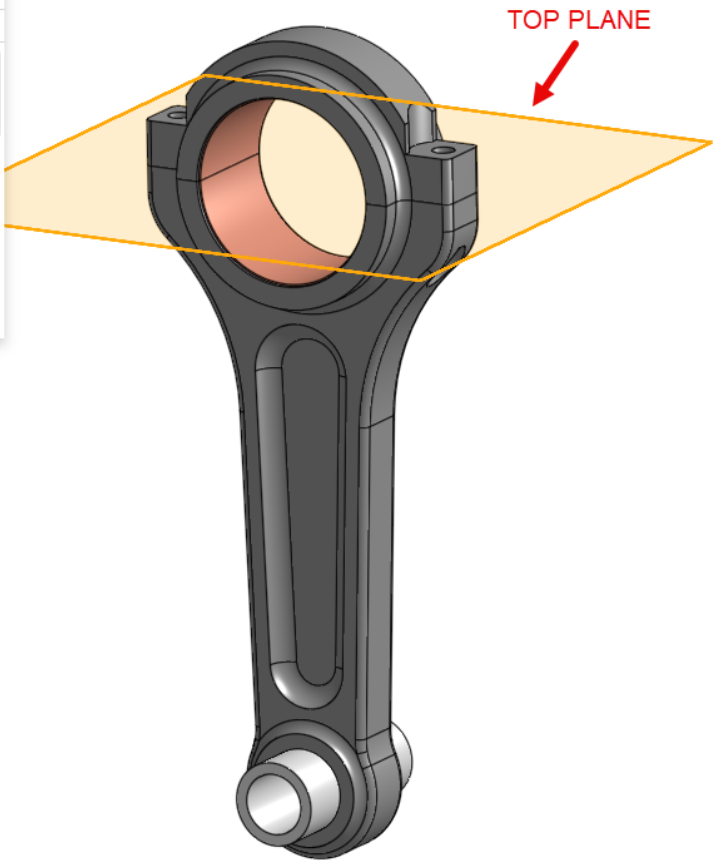


Use SPLIT to cut the Connecting Rod into multiple parts

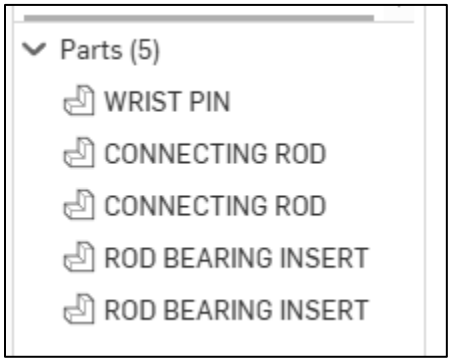
17. Use the **SPLIT COMMAND** to split the CONNECTING ROD and ROD INSERT BEARING into separate halves:

a. Launch the **SPLIT** command
b. Use the following settings:

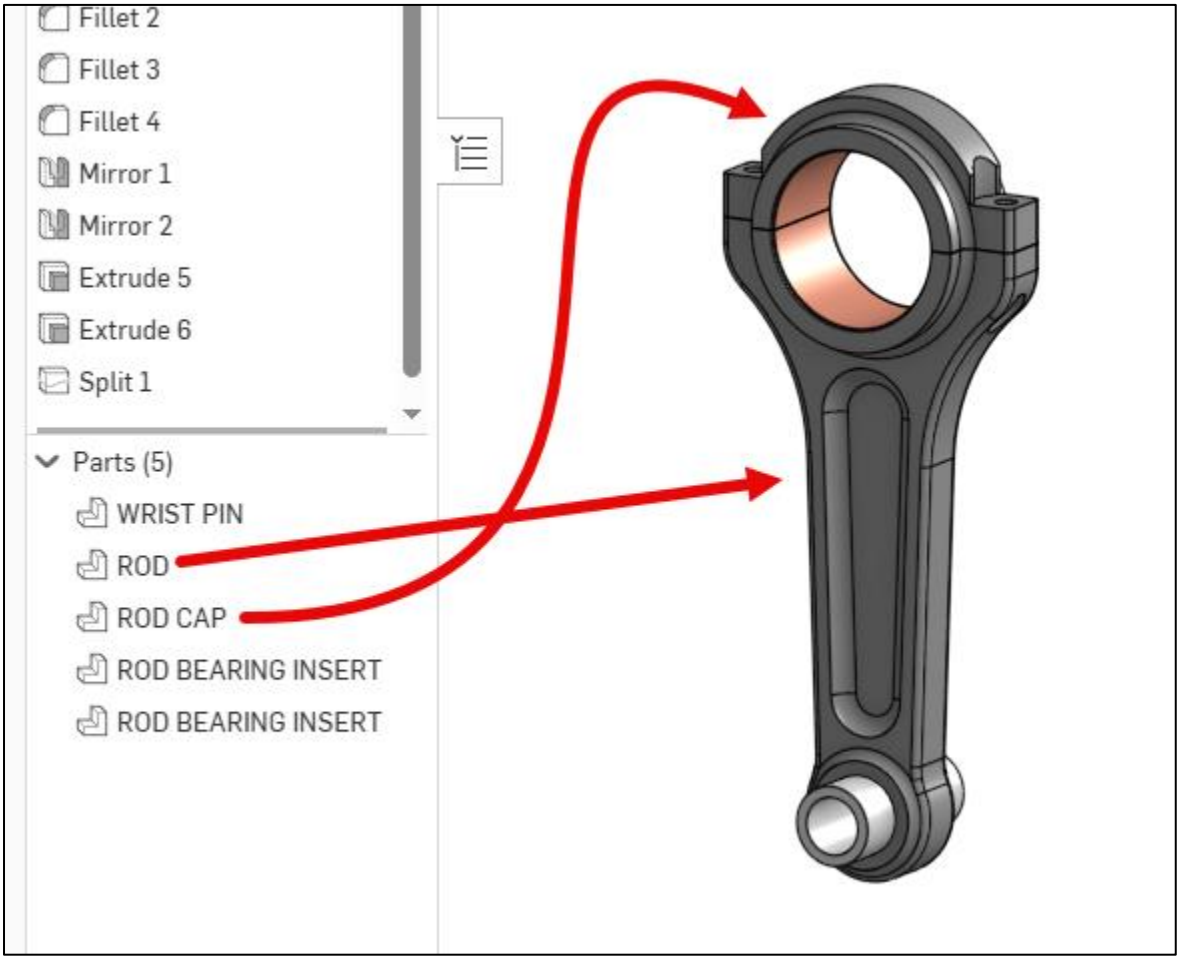




18. Examine the parts list. We now have 5 total parts:

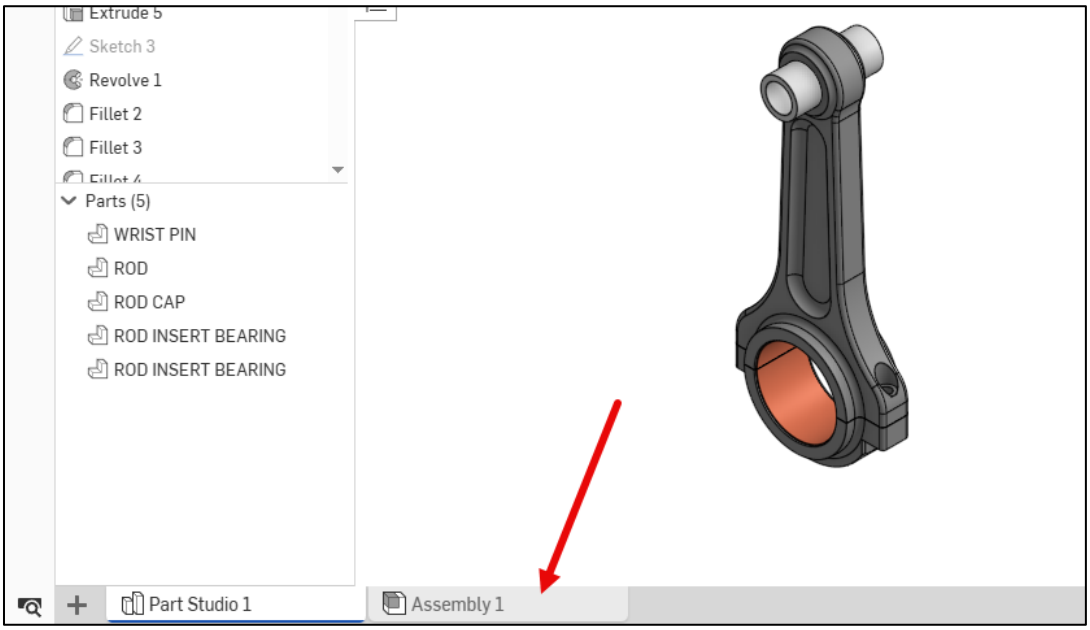


a. Rename the **CONNECTOR ROD** and **CONNECTING ROD** parts to:
- ROD
- ROD CAP

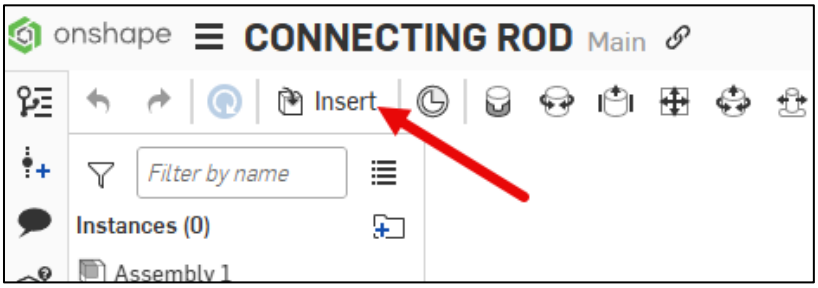


19. Now let's add these parts to an assembly:

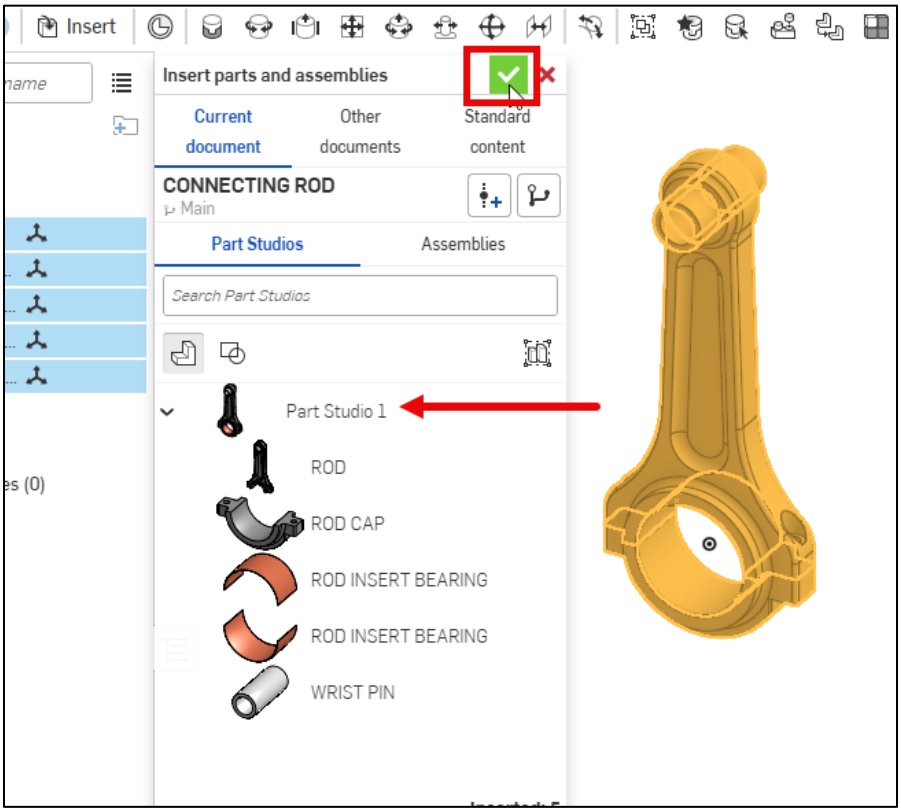
a. Click on the **Assembly1 tab** (or create an Assembly)



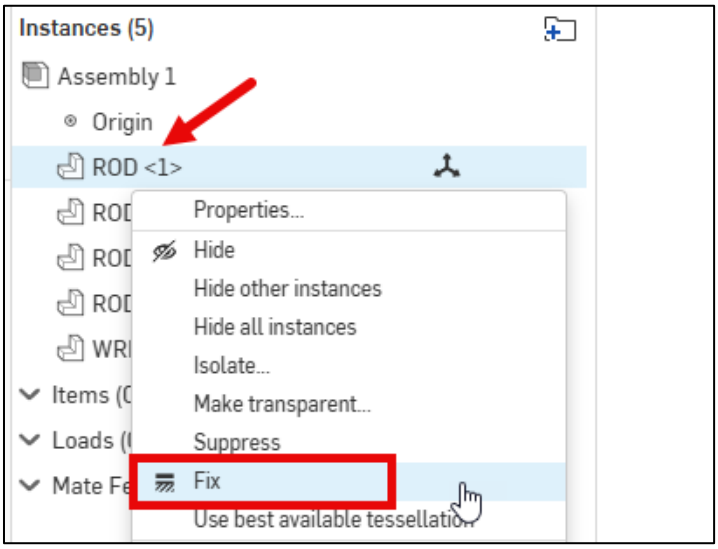
b. Choose the **INSERT** command in Assembly mode:



c. Click the name **Part Studio1**, then click the **GREEN CHECKMARK**

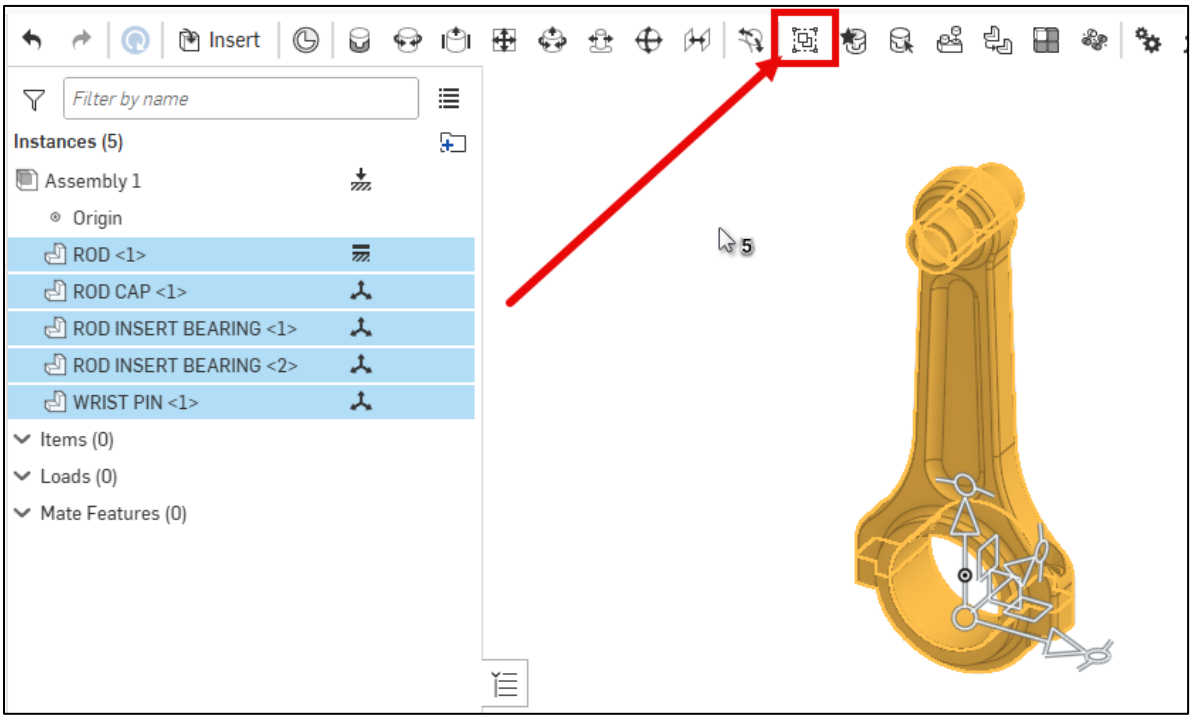


d. **Right Click** on the part called **ROD**. Choose **Fix**

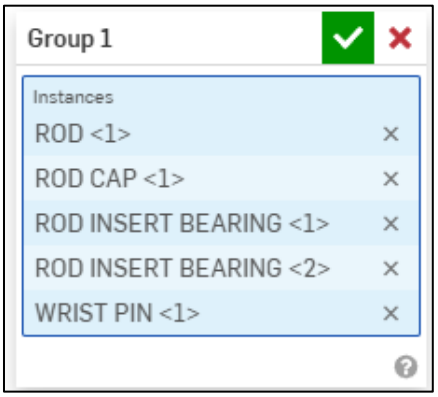


This part is now **immovable**

e. **Select the 5 parts** from the tree
then select the GROUP mate type



Click the green checkmark and these parts will be grouped together. This means the parts will have zero degrees of relative motion, which means that all the parts in this assembly will become immovable.



To learn more about mates and assemblies on Onshape:

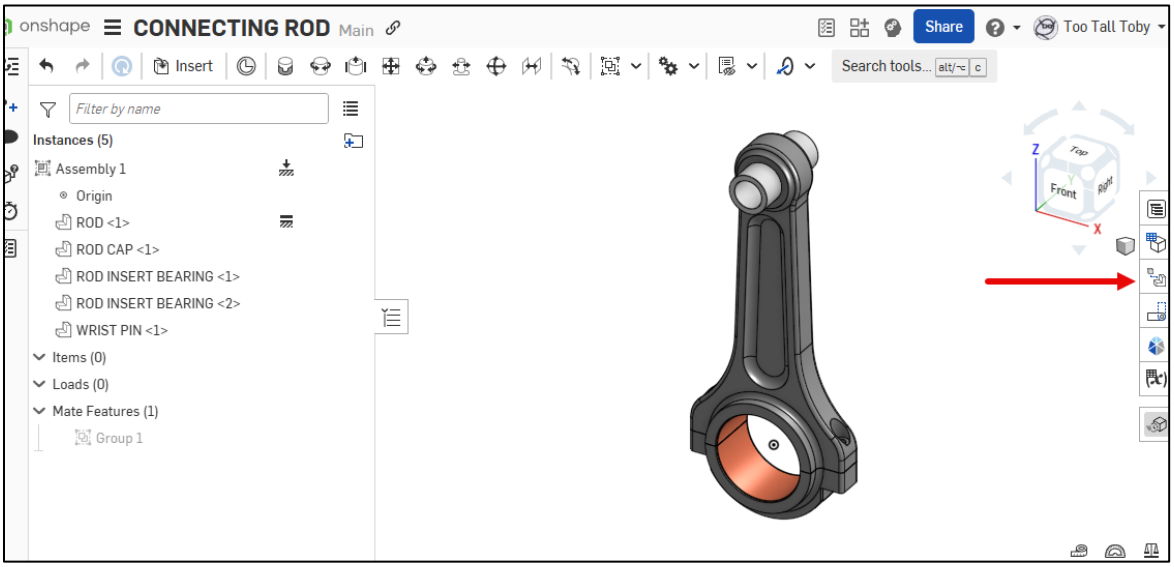
Check out **Toby's Step-by-Step tutorial** on how to build a **STIRLING ENGINE**:

https://www.youtube.com/watch?v=GYkZmE_6MpY

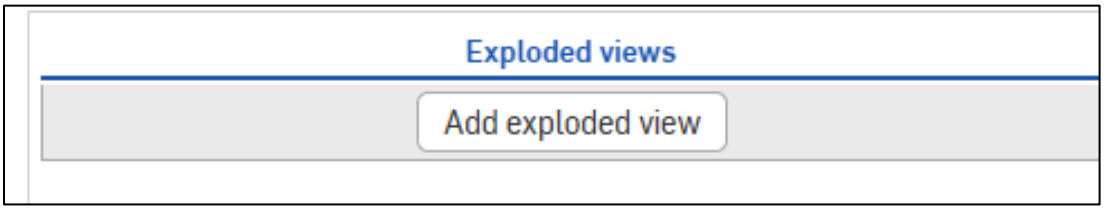
EXPLODED VIEW

20. Exploded views can be helpful to explain how to assemble these components. Exploded views can also be displayed in a 2D drawing in Onshape.

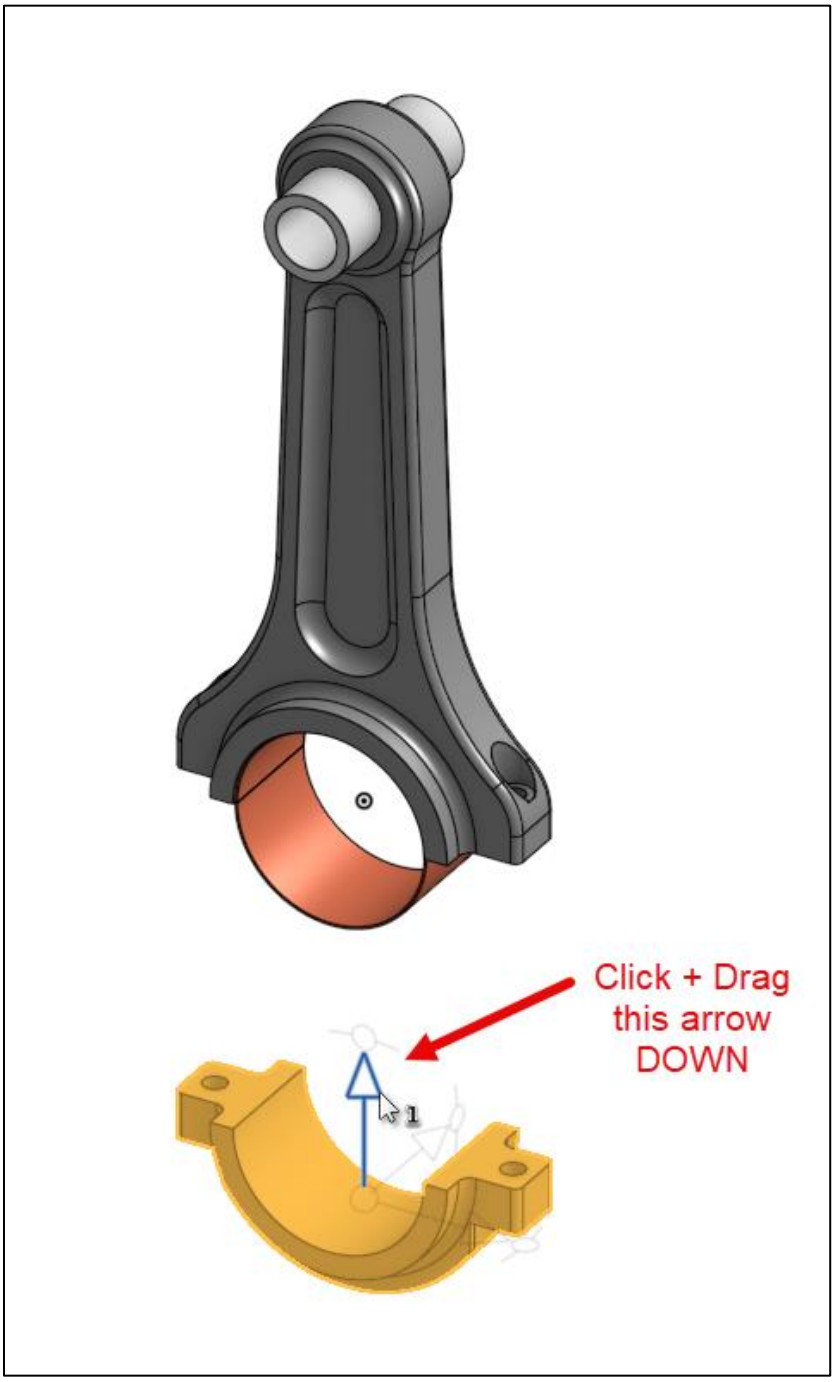
a. Click on the **Exploded Views** Panel



b. At the top of the panel, choose **Add Exploded View**



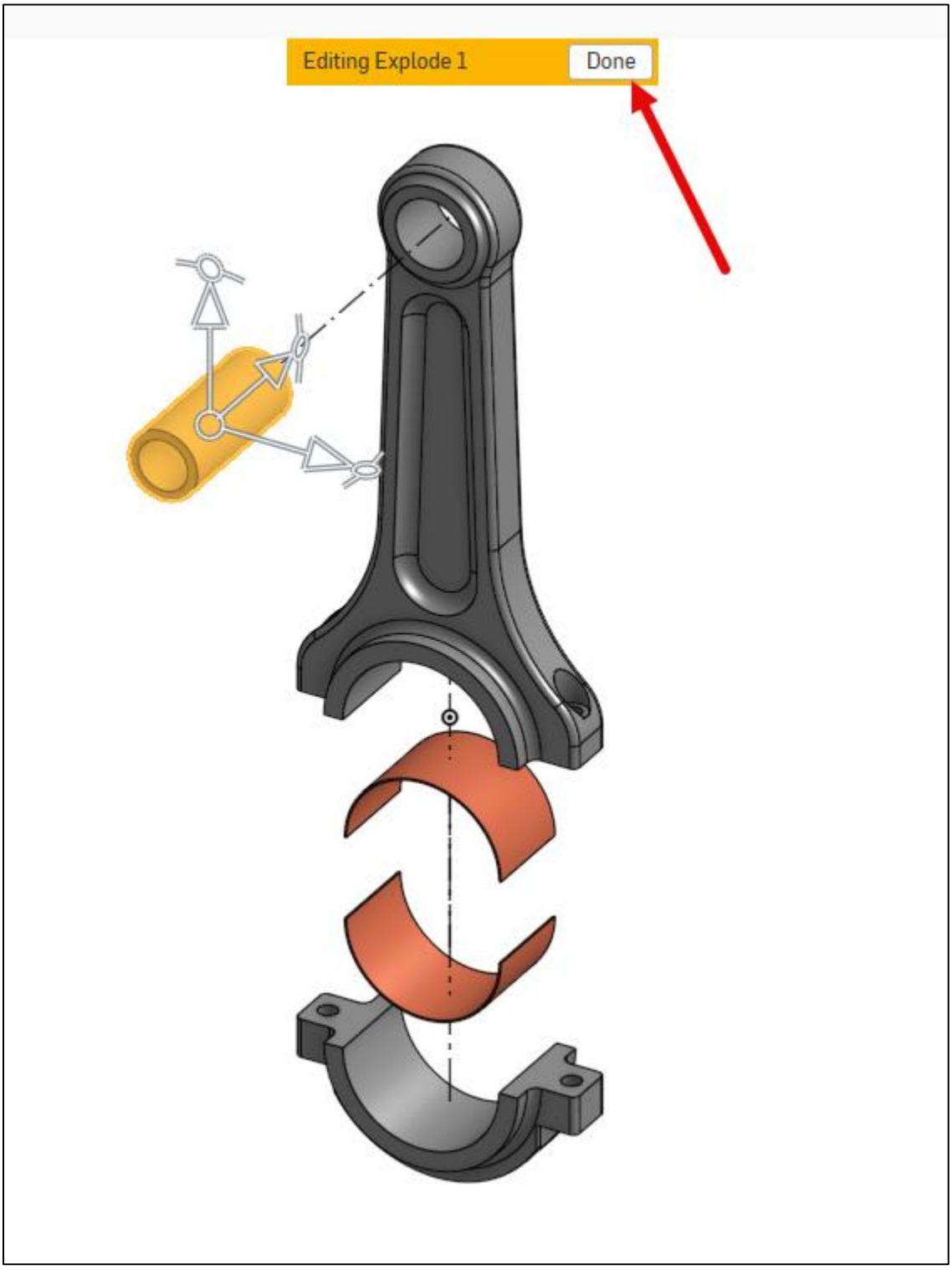
c. Single **left click** on the **ROD CAP**, then **click and drag** this arrow down:



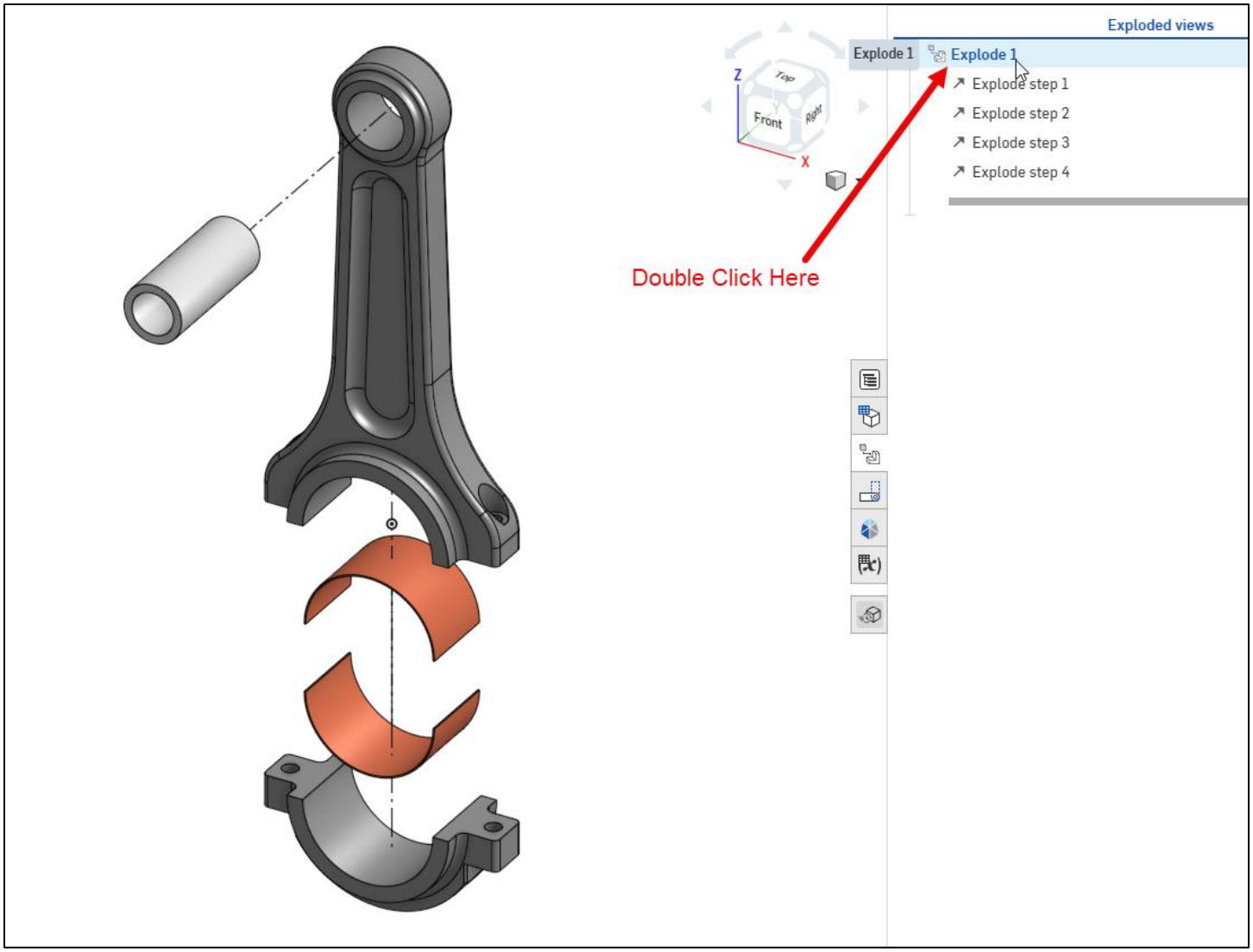
d. Click the **Green Checkmark**

e. **REPEAT for the remaining items** in the assembly

When finished, click DONE:



f. To activate / de-active the exploded view, **double click on EXPLODE 1**



Conclusion

In this short tutorial, you learned how to design multiple parts together in one single Onshape part studio, and how these parts can be brought into an assembly.

Along the way, we learned about:

1. How to use the S-Key at access shortcuts
2. Using 1 sketch for multiple extrusions
3. How to use SHIFT+ENTER to repeat a command
4. Using the SPLIT command to CUT AWAY material from a model
5. Creating a cut that stops once it gets a certain distance from a face
6. Using a Face (with no sketch) to do an extrude cut
7. Using the SELECT OTHER tool
8. Using Doubled Dimensions for symmetry
9. Using a Revolve feature to remove material
10. Adding Fillets of different sizes
11. Mirroring the part
12. Creating additional parts in our parts studio
13. Renaming parts and changing appearances
14. Using the SPLIT command to split a single part into multiple parts
15. Adding parts to an assembly
16. Creating an exploded view from an assembly

Please remember to like this video, subscribe to the channel, and leave us any questions/comments below!

Thanks for watching, and good luck on your CAD journey!

- Toby

Visit www.Onshape.com for more Onshape learning resources

Visit www.TooTallToby.com for 2D to 3D Practice Models and live CAD vs CAD competitions