DE500018

Landing Gear Design Optimization Using Generative Design

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Learning Objectives

- Learn about Designing Landing Gear using Autodesk Fusion 360.
- Learn about applying Generative Design to the designed landing gear for weight reduction and Design optimization.
- Study and Compare the performance of landing gear with Generative Design applied under various forces and factors.
- learn about 3D printing of small scale landing gears with or without generative deisgn applied for testing purpose.

Description

How cool it is to watch the Landing gears of an Aircraft getting retracted back to its resting position and opening during landing and taxing! But a landing gear of an aircraft contributes to 20 percent of the total aircraft weight as they are very heavy and they should be as they have to support the entire aircraft weight while landing, takeoff, or taxing. So with this in mind, we've come out with an innovative solution of applying generative design to the aircraft landing gear for weight reduction and design optimization for drag reduction and more, so with this approach, we will be designing a landing gear in two variants in which we'll apply generative design to one variant and second will be designed like the conventional landing gears and then we'll 3D print both of them and we'll take our models to wind tunnel and static structural testing. and then at last we will study and compare the performance of both the landing gears in various loads and wind tunnel conditions.

Speakers

Hello, my name is Harsh and I am from Delhi, India. I'm currently a senior at Dayananda Sagar University, Bengaluru. where I am studying BTECH in Aerospace Engineering. I am a success-driven detail-oriented individual. I have a passion for learning and applying newfound knowledge. I pride myself on my time management skills and on the quality of work I submit.

I look forward to joining the Aerospace industry upon the completion of my degree. I am expected to finish my undergraduate degree in May of 2023.

Hey there! My name is Kushal, and I am from Bangalore, India. I am currently in my third year of a BE in Mechanical Engineering at The National Institute of Engineering, Mysuru, and I am interested in the field of aeronautics. With a strong desire to learn new things, I am constantly investigating and experimenting with new ideas, involving integration of multiple fields at a single place.

Hello! Greetings! My name is Abhishek Chavan. I reside in Pune, India. I am pursuing my Third Year of BE in Mechanical (Sandwich) Engineering. I am passionate about Automotive, Aerospace, and Defense domains with Sustainability being the main focus. I find myself in Experimentation, Product Design, and Development eventually becoming a Master of multiple trades.

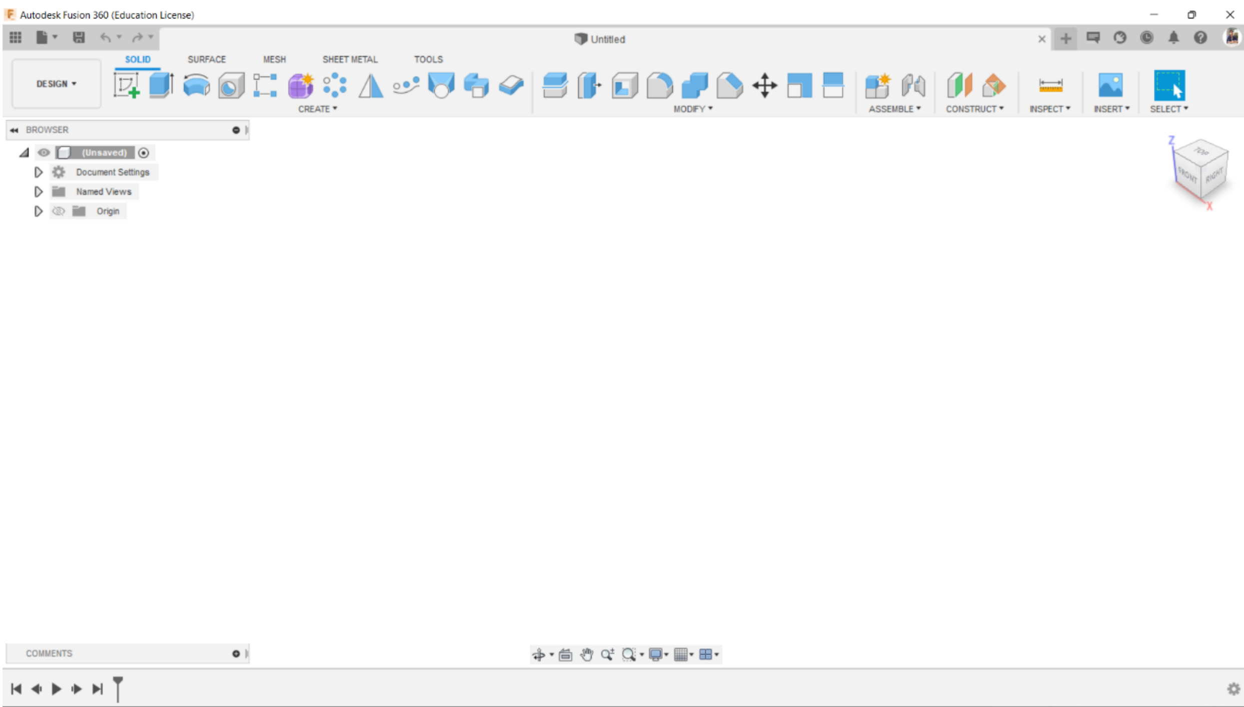
Table of Contents

Introduction

Modelling and Prepartion for GENERATIVE DESIGN

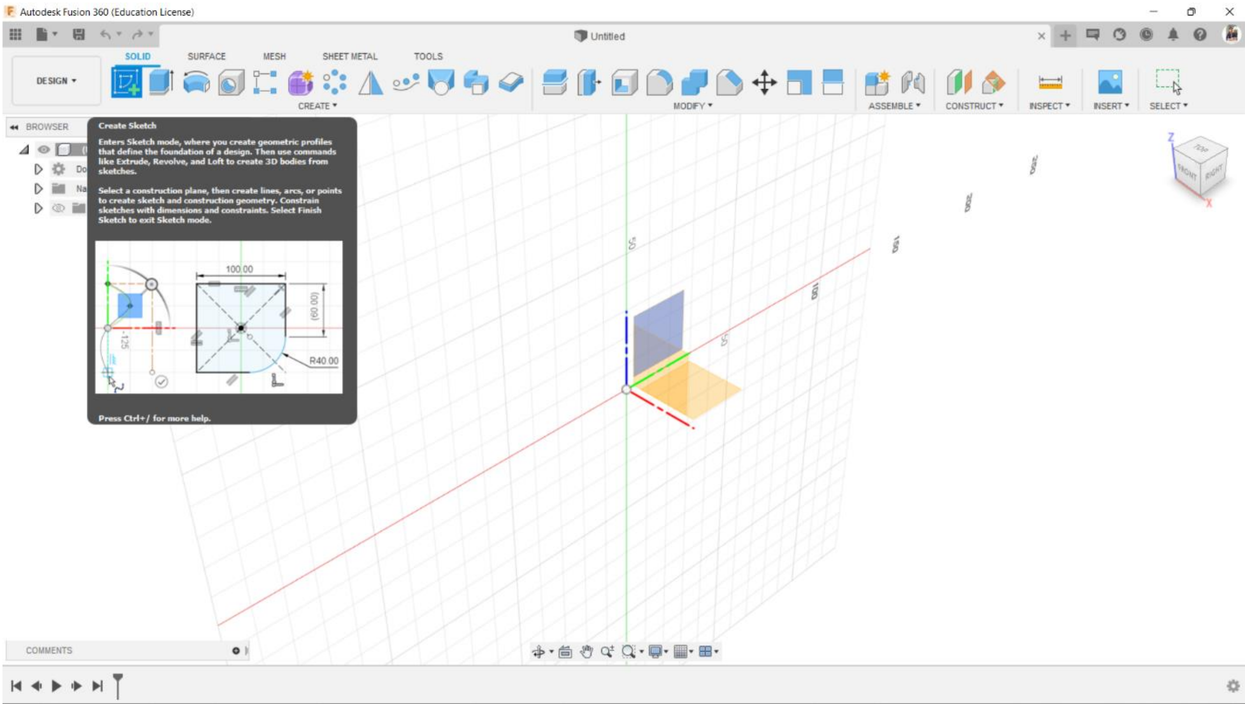
Designing a Starting Shape for Generative Design landing gear Component

To create the necessary parts for Generative Design Landing Gear Component, we will use Solid Enviroment of Fusion 360.

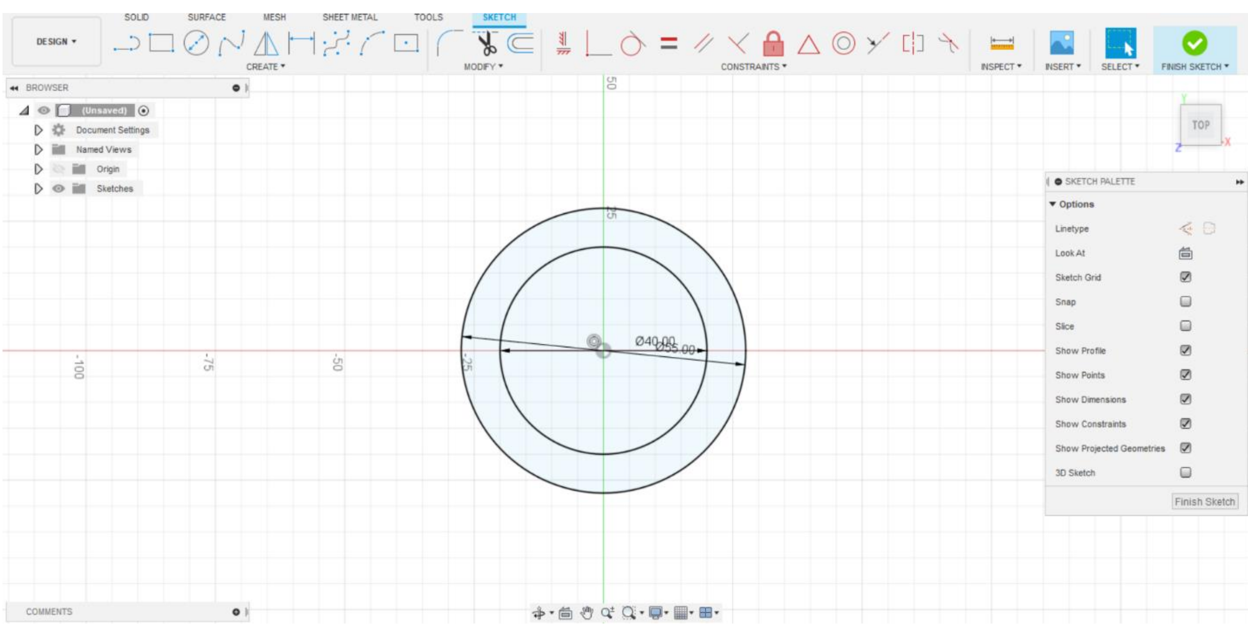


Follow the Next Steps:-

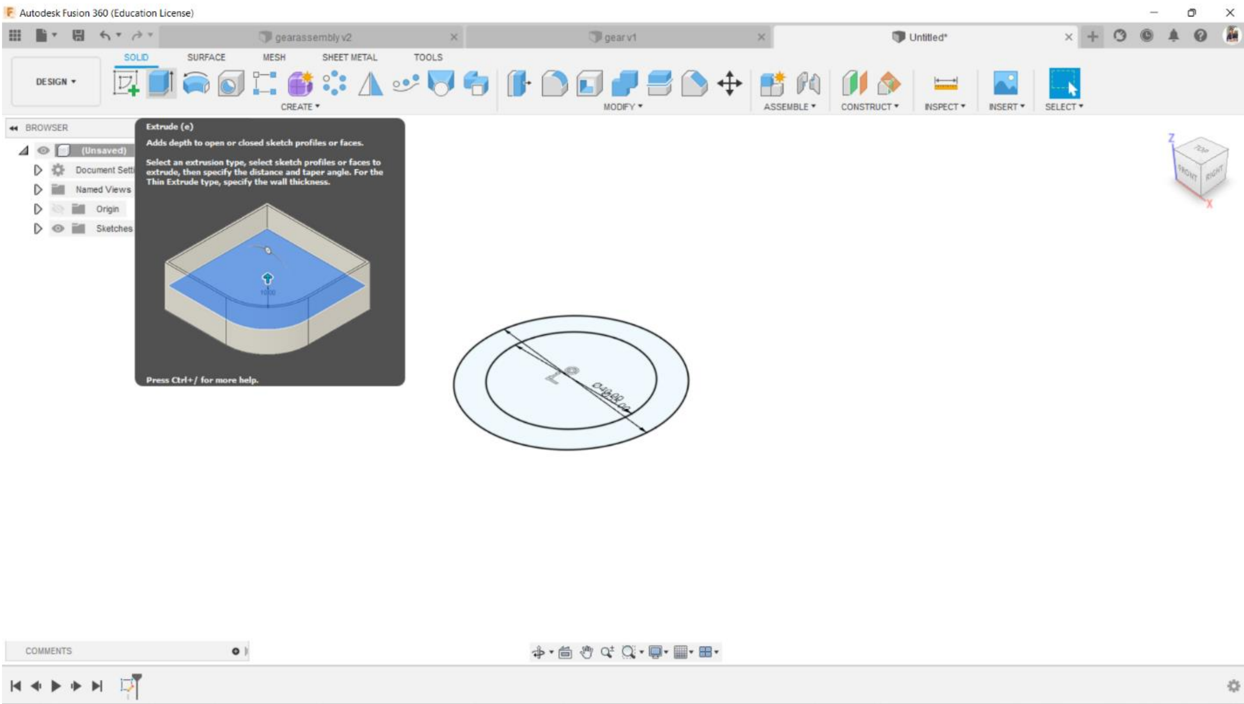
1. After you open the **Fusion 360**
2. Select the Create Sketch Enviroment and after that choose the bottom plane to work on.



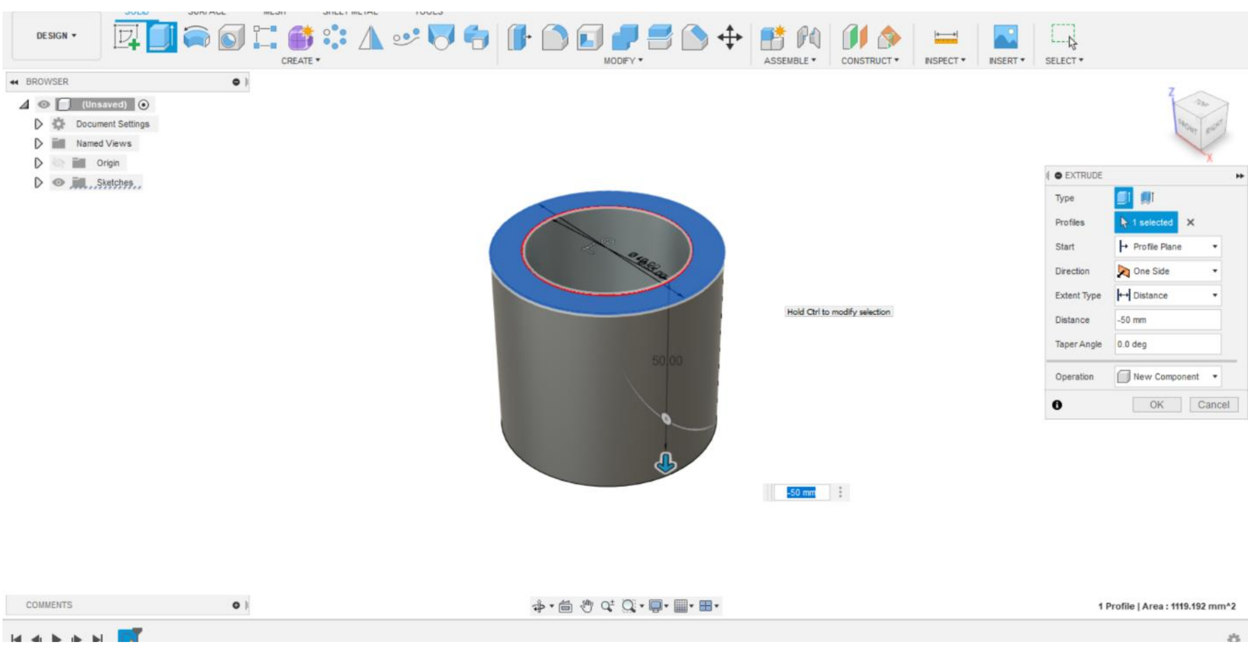
3. After that, Make two Concentric center diamemter circles of **diameter 55 mm and 40 mm.**



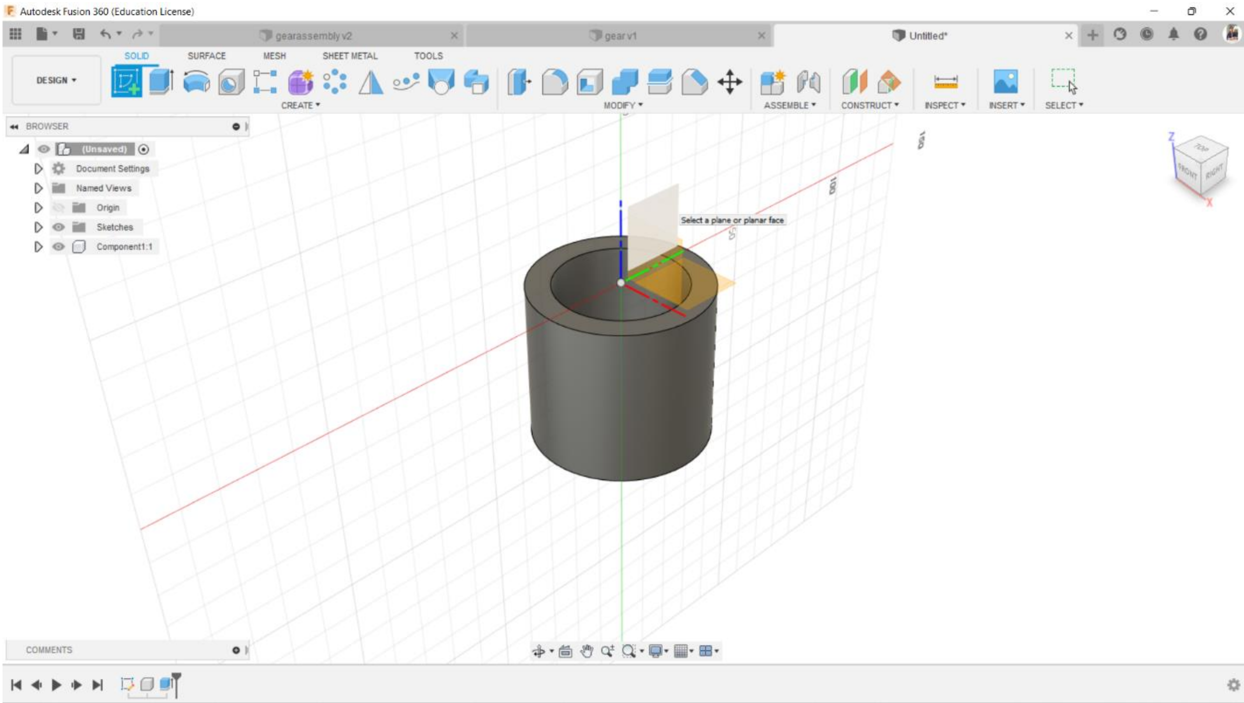
4. Now exit the Create Sketch Enviroment by pressing **Finish Sketch** from the top Right corner colored in green. And choose the **Extrude** tool from the Create Enviroment.



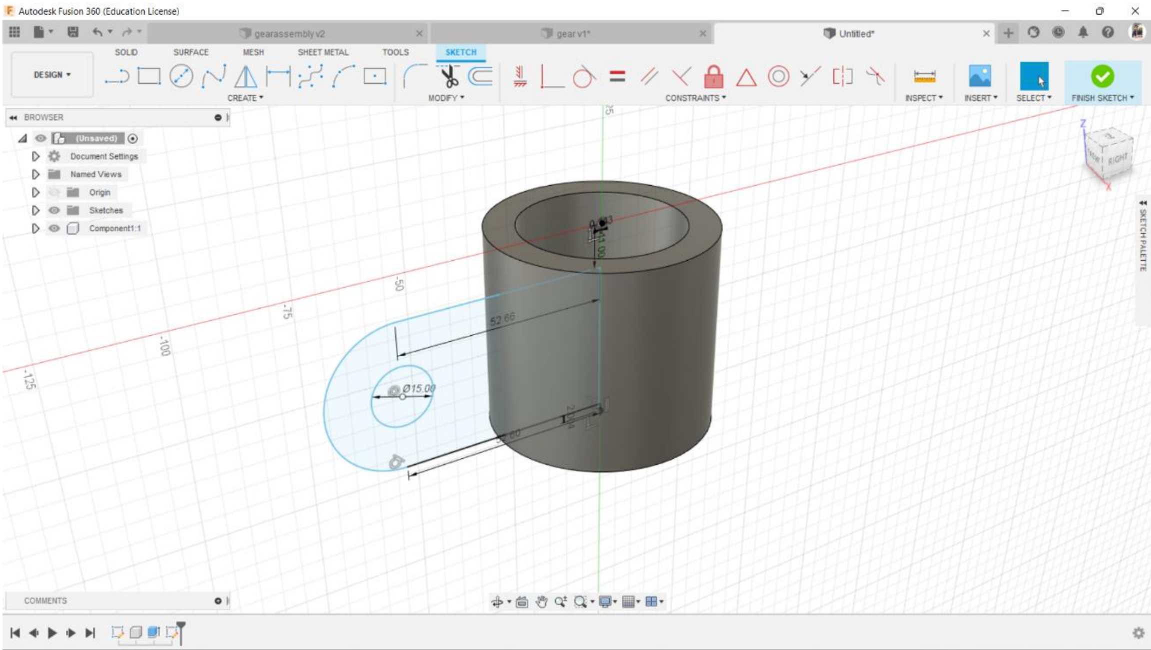
5. After selecting Extrude tool, click on in between the outer and inner circle and give the **Extrude dimensions as -50 mm** (50mm can also be given **)** and give operation **as New Component** and press ok.



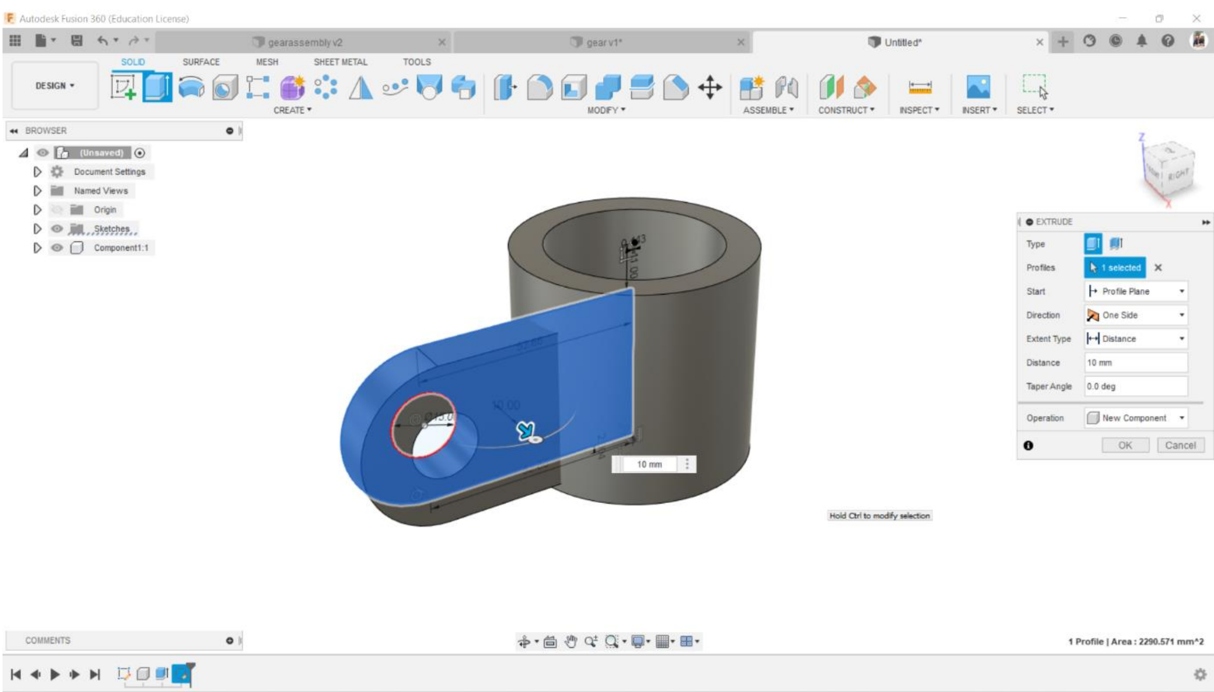
6. After that again go back to Create Sketch by choosing **Right Plane** this time.



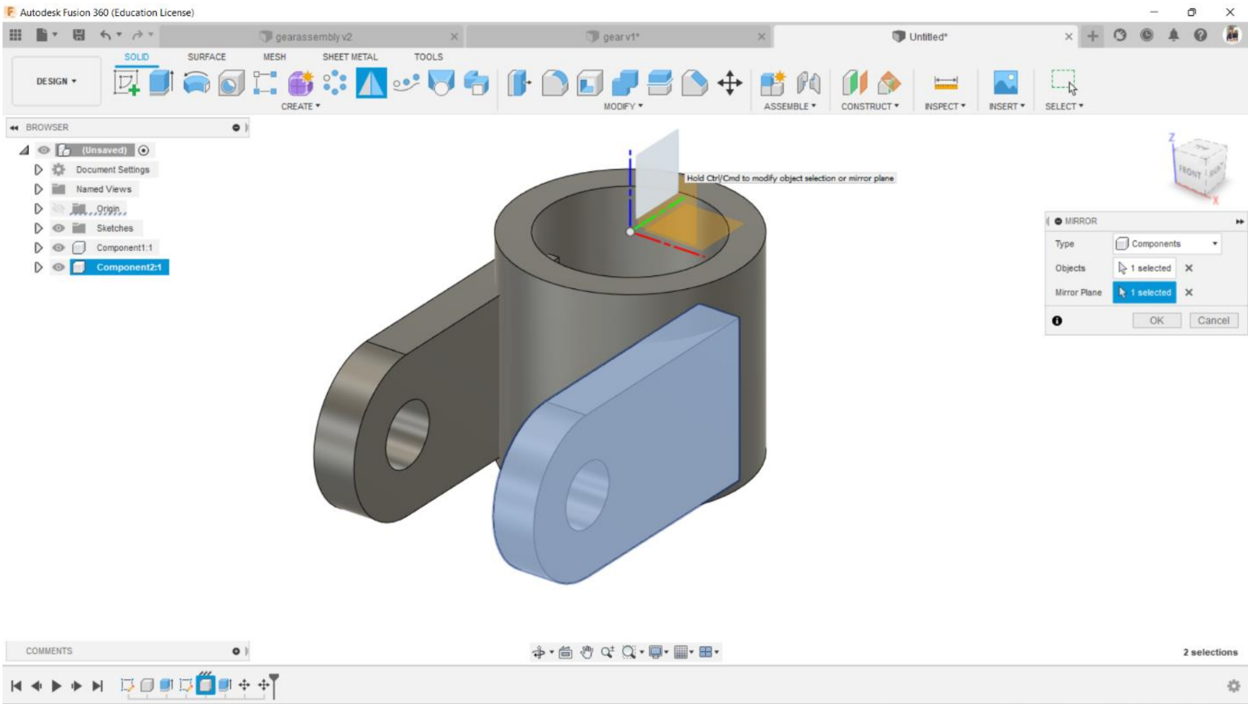
7. Make the Sketch exactly as shown in the picture, make a center line of dimension 35 mm, then draw two lines each of 52.60 mm horizontally starting from the top and end point of the center 35 mm line. Then make a circle of diameter 15 mm at the end of 52.60 mm line and close both the lines with a 3 point Arc of radius 17.50 mm.

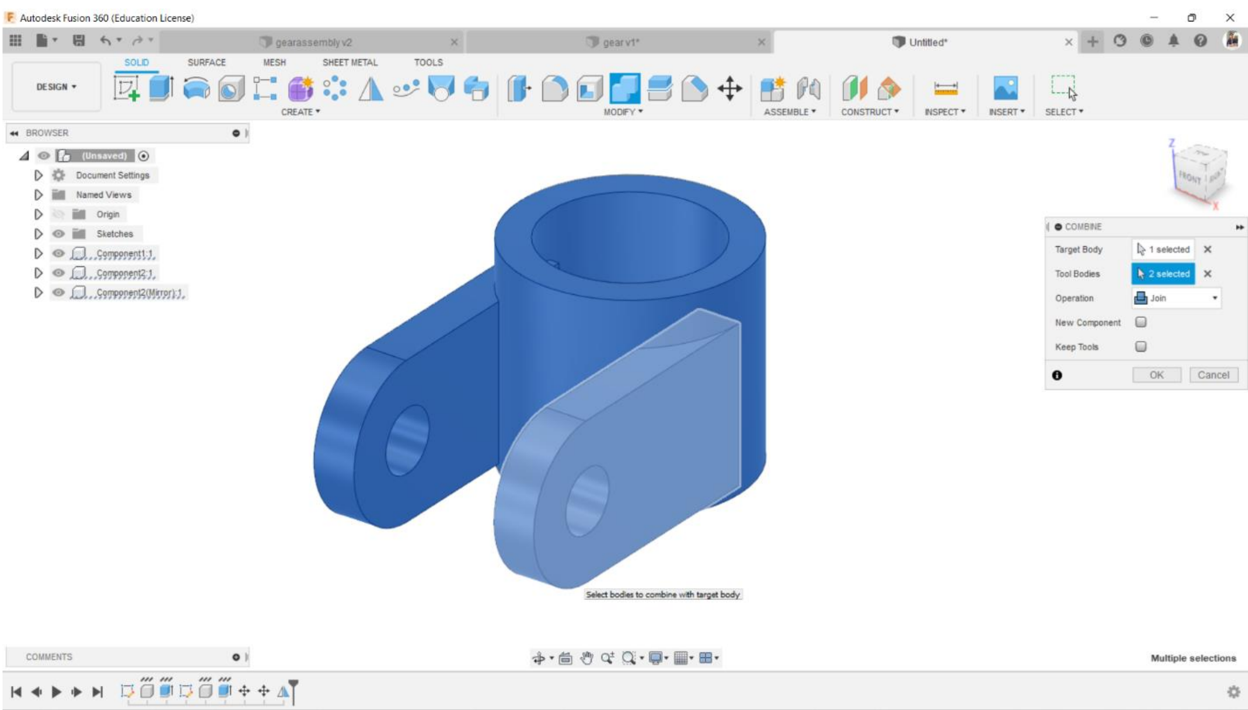


8. Now finish the sketch and click on **Extrude tool** and extrude the sketch upto **10 mm** and give It as new component.



9. After extruding , choose mirror option down from the create environment and choose the component to mirror then choose mirror plane as right plane and press ok. Then click on combine option on the modify environment and combine all three components and press ok.

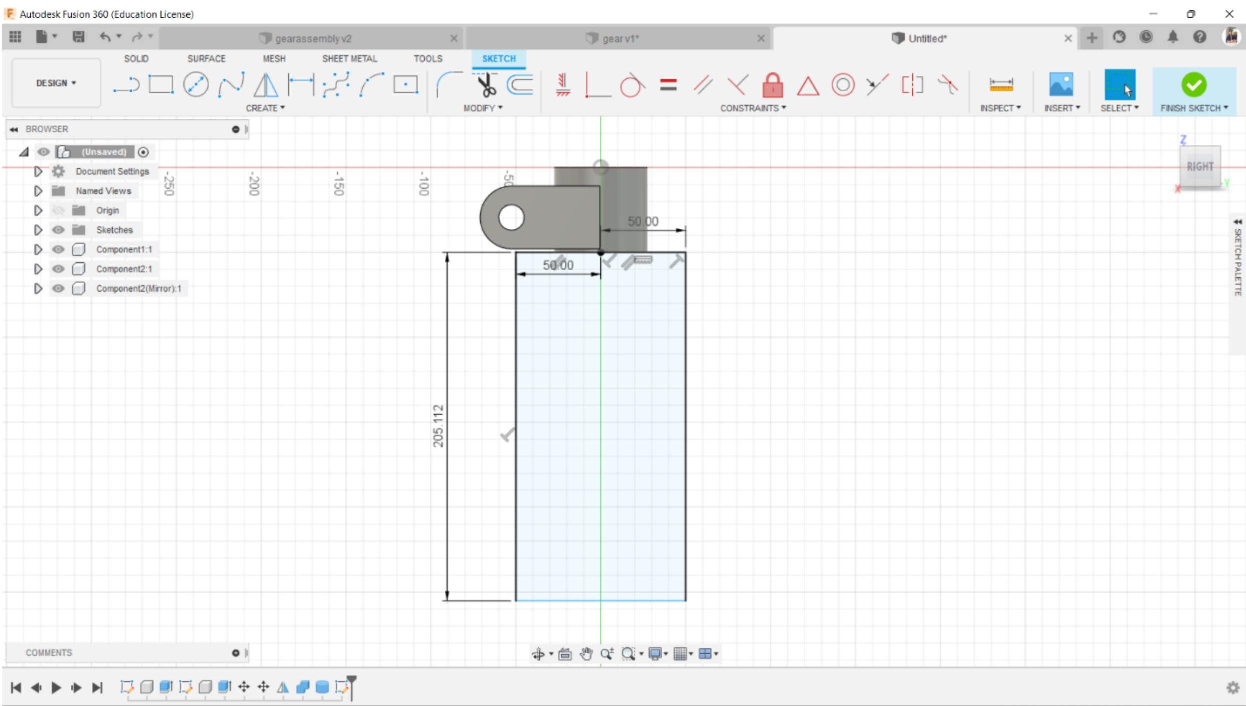




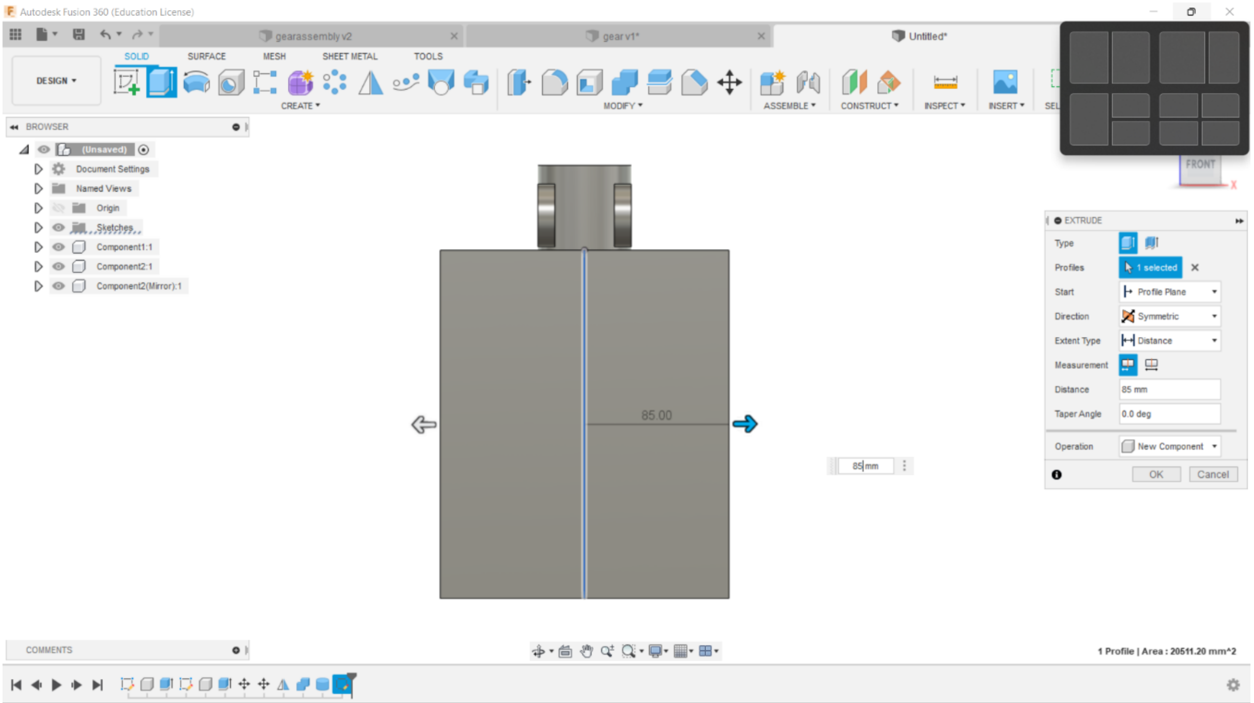
10. Upper part of the Landing Gear Component is now ready, and Lower part we'll do now.

11. Go to Create Sketch and choose the Right Plane.

12. Draw the Sketch as shown in the picture, make a **100 mm line horizontally** (50 mm each on both sides making a in total of 100 mm) then make a **205.112 mm line vertically** from the end points of **50mm line** and repeat the same procedure on the other side as well.

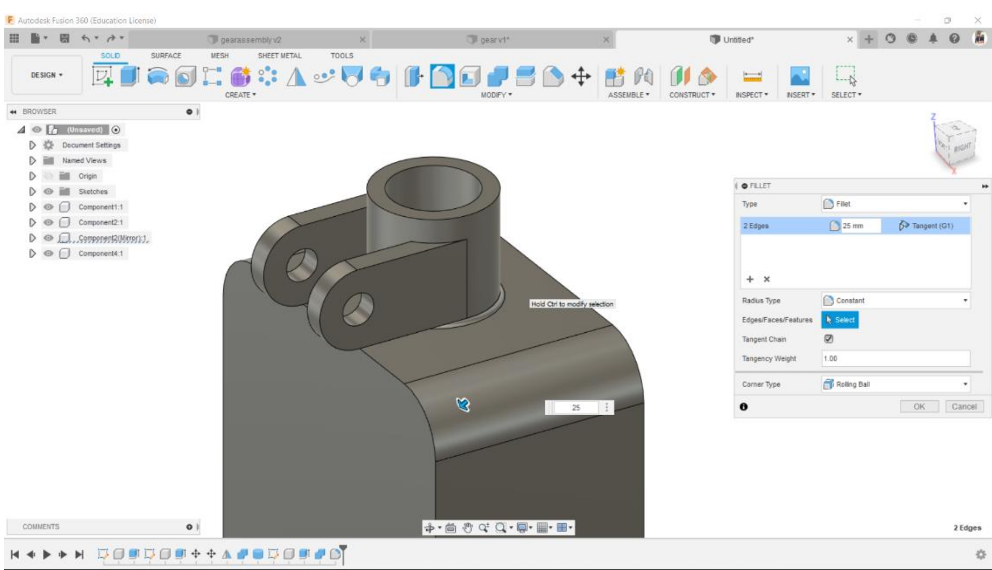


13. Finish the sketch and extrude the sketch as **85mm** by giving Direction as **Symmetric** and Operation as **New Component**.

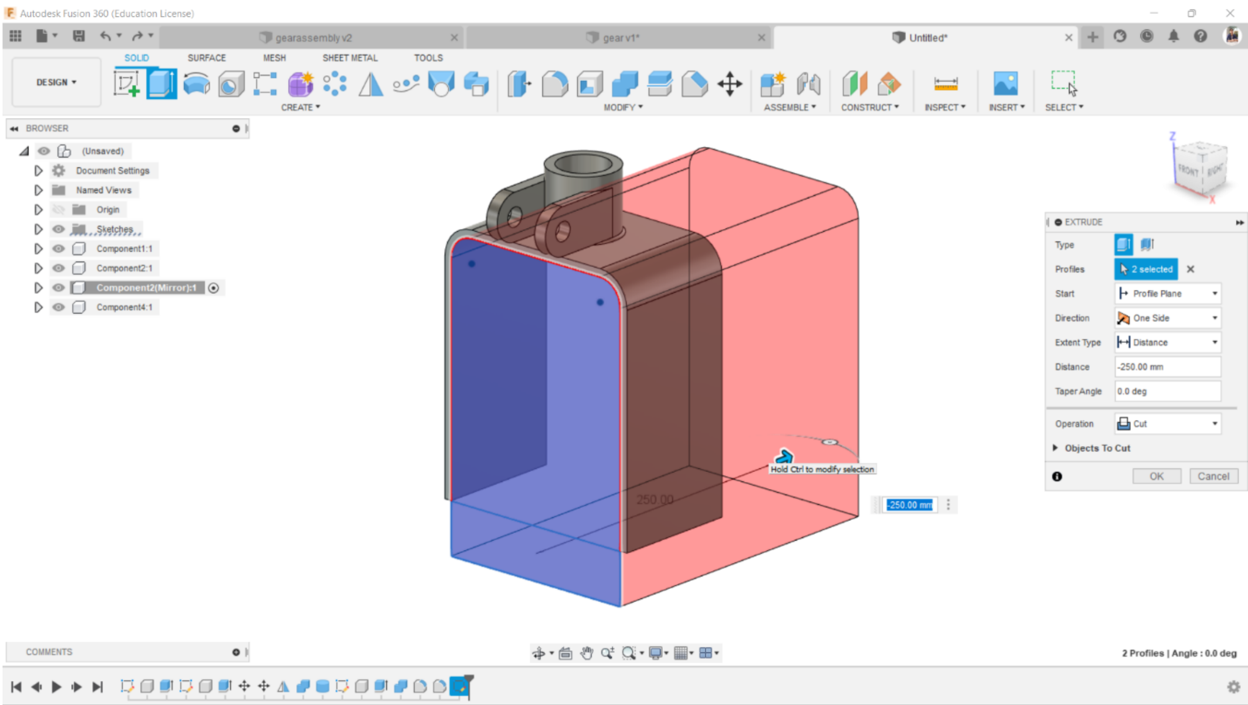


14. Now Combine the newly created Component with the upper body by choosing **Combine** tool from the **Modify Enviroment.**

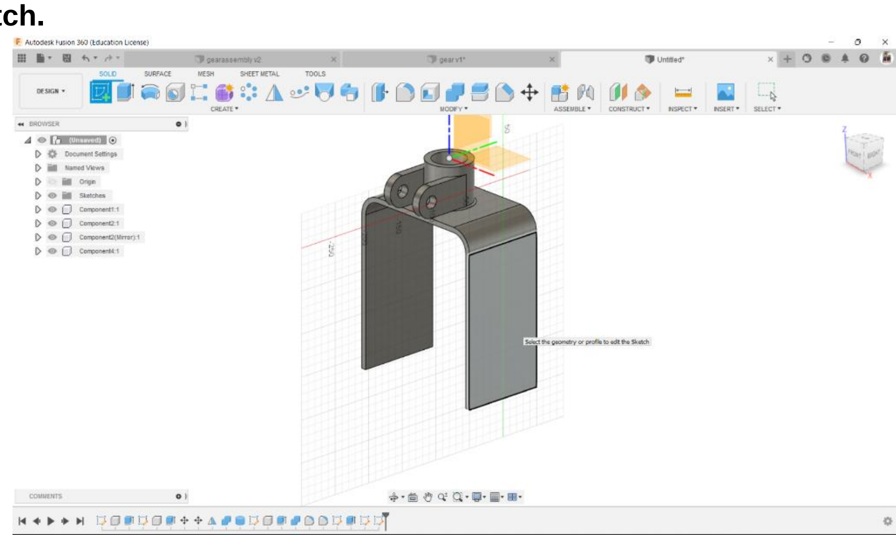
15. After that choose the **Fillet tool from Modify Enviroment** and select the upper ends of the square and give the radius as **25 mm**



16. After Filleting the edges, go to **Create Sketch Enviroment** and choose front portion of the lower component as plane and offset the sketch boundaries of the lower comopent by selecting **offset** from the **Modify** option and select the sketch boundaries and give **offset position as 6 mm**. and Finish the sketch and choose **Extude tool** ,choose operation as **Cut** and remove the inner portion of the component.

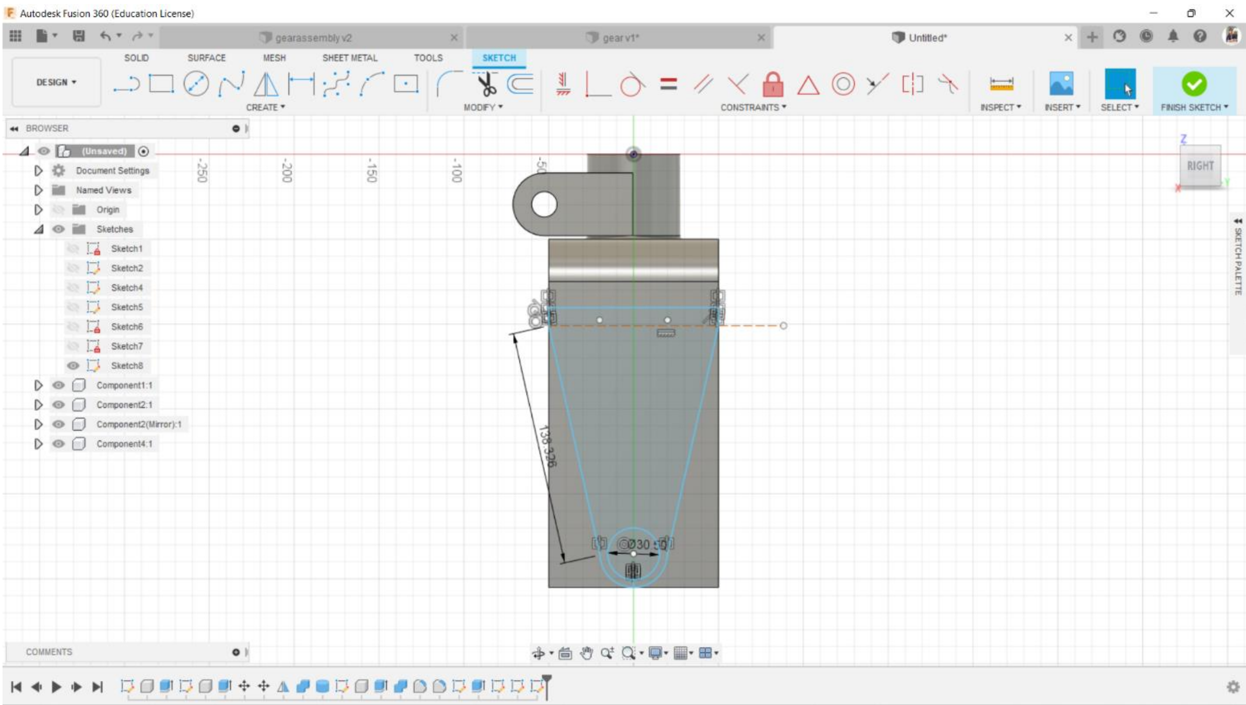


17. After that click on the outer **Right Face** of the component and then click on **Create Sketch.**

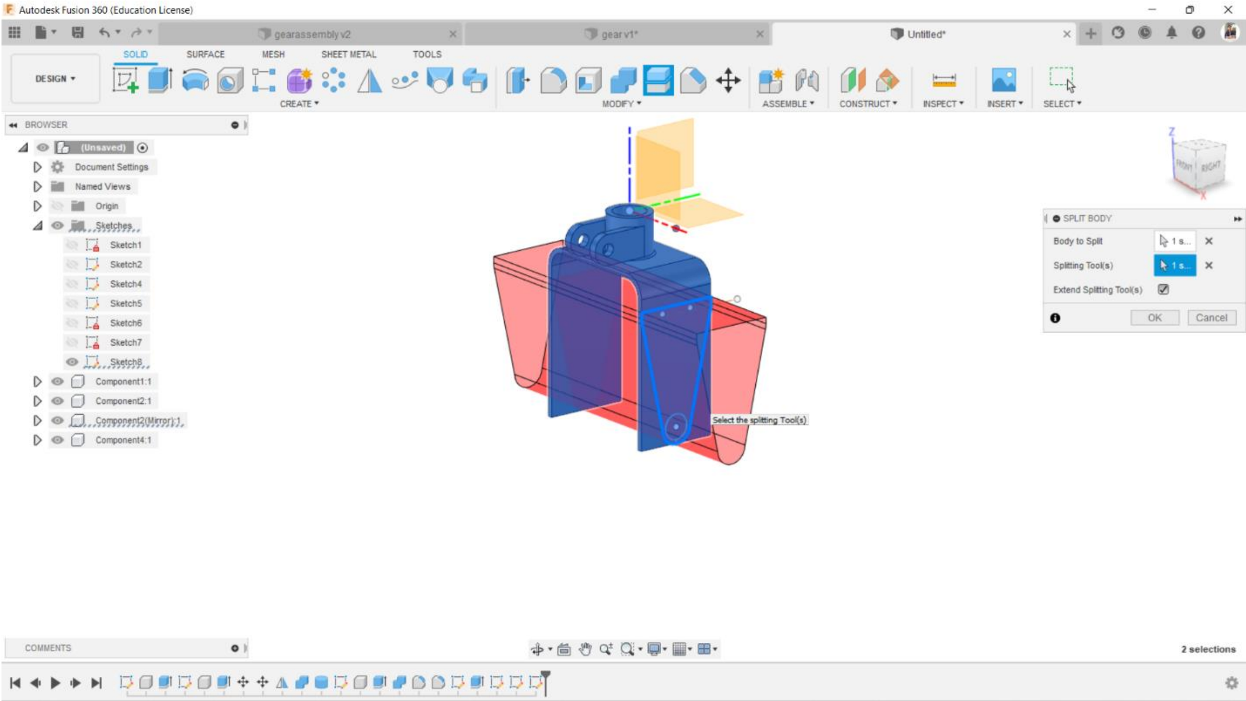


18. After landing on Sketcing Enviroment , Draw the sketch as Follows:-

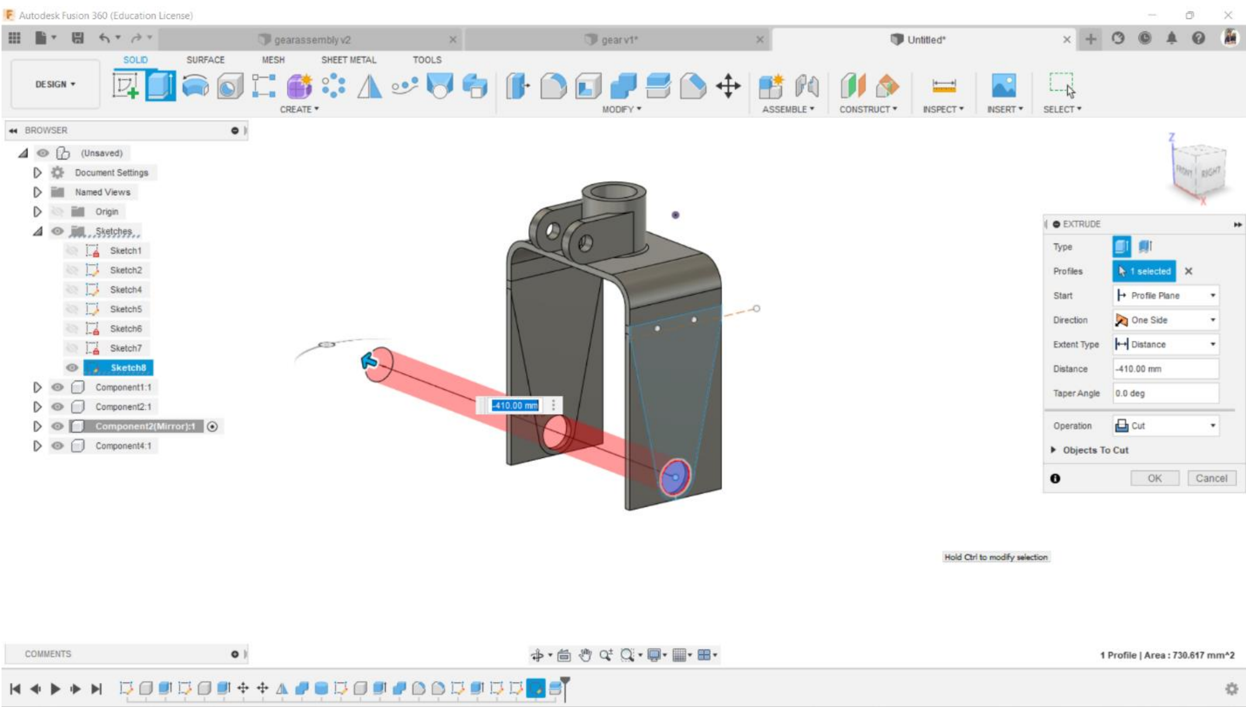
Inclined line **:- 138.326 mm** incined at **102 Degrees** draw it from both the ends and join them using a **3-Point Arc** and make a **30mm diameter circle** in the lower part of the sketch as shown in the picture.



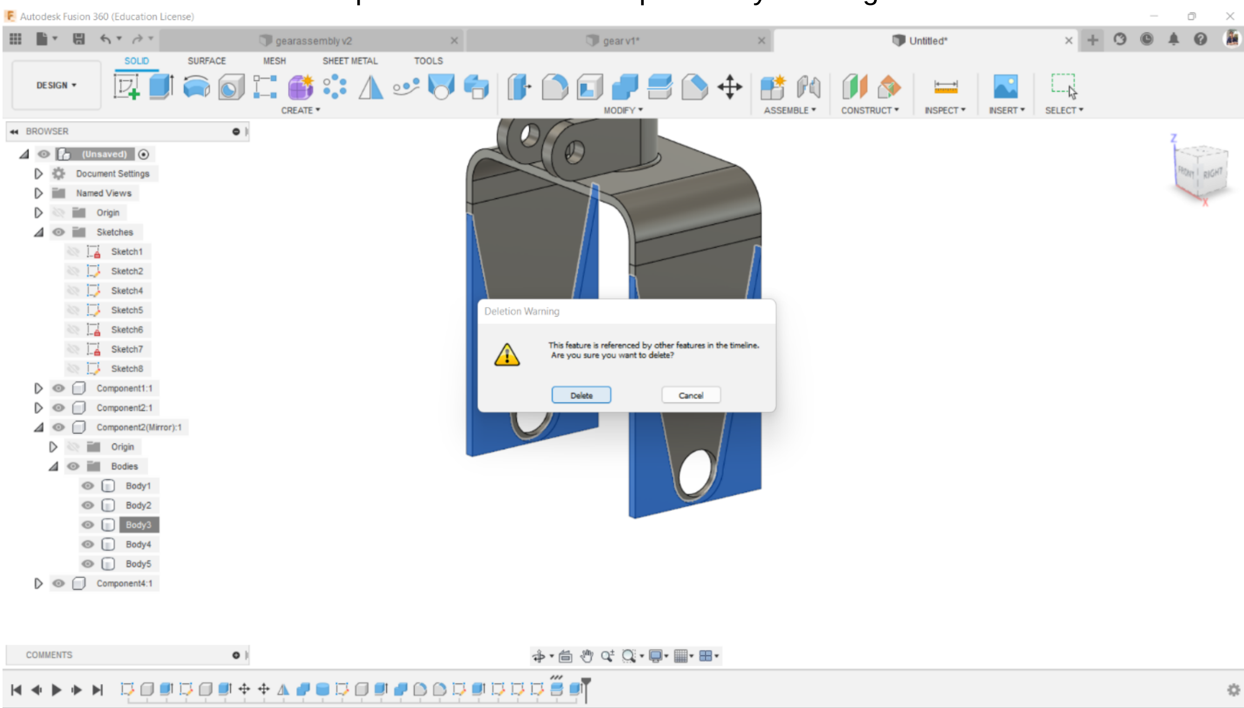
19. Exit the Sketching Enviroment and choos**e Split Body tool from the Modify Enviroment**, and choose body to split as the Designed Component and choose split tool as the sketch that we have just drawn and press ok.



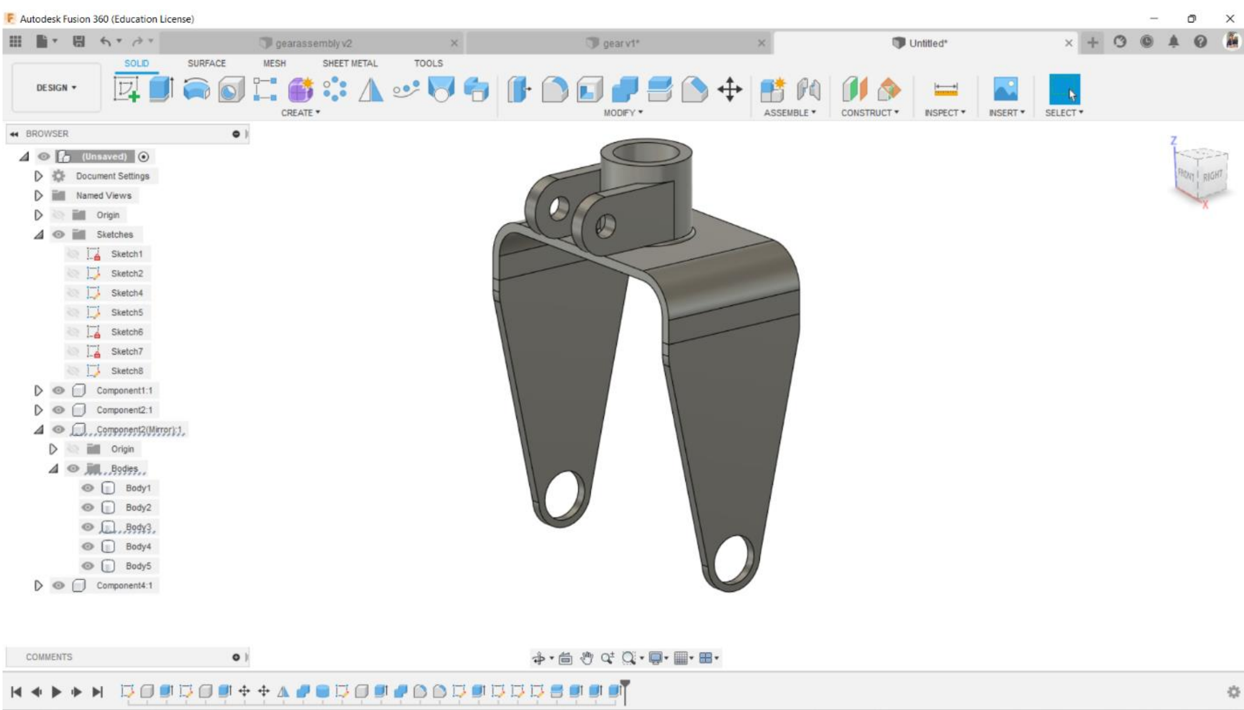
20. Then Cut the lower Circle that we left before using **Extrude Tool** and select option as **Cut** as shown in th Picture.



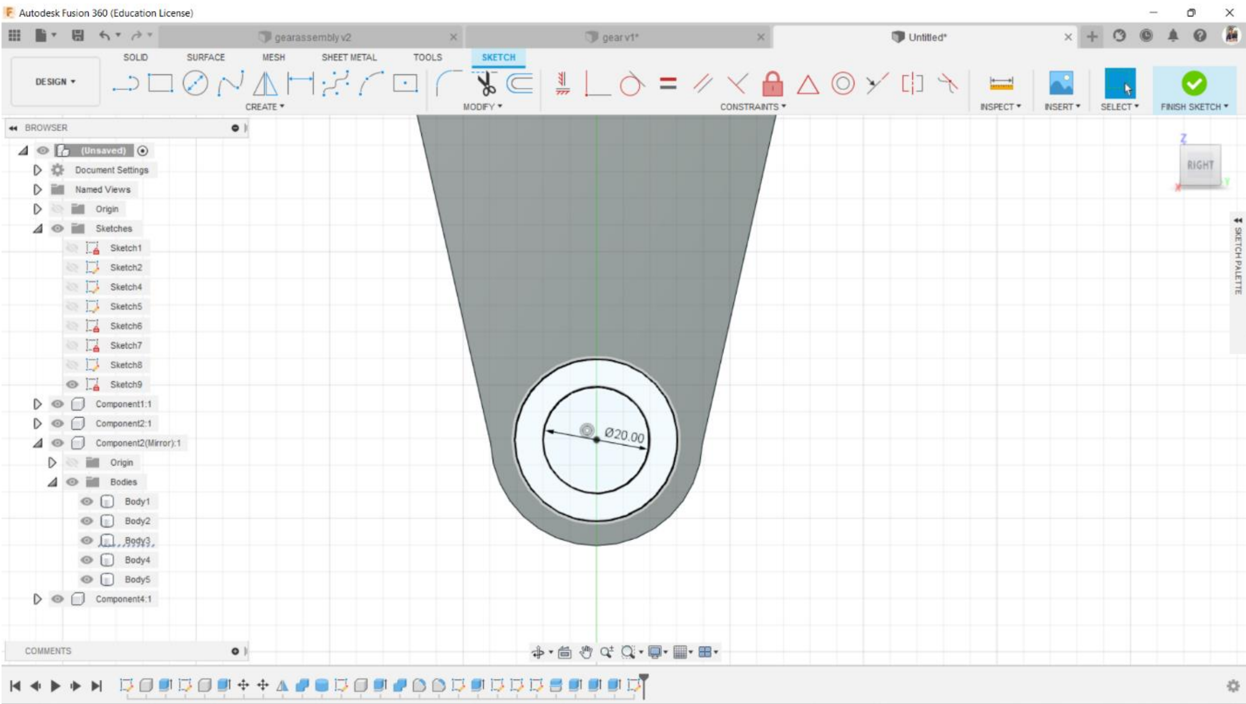
21. Now Remove the splitted unwanted Component by deleting it from the **browser**.



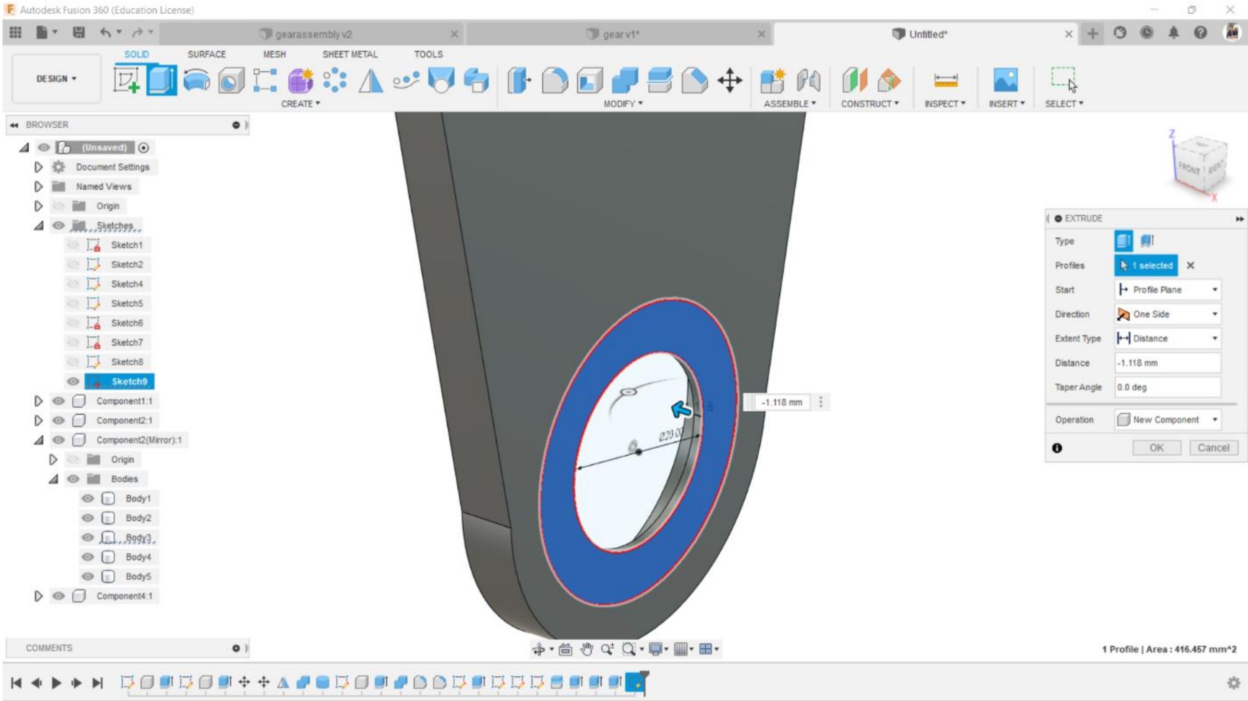
22. We have Completed the 90 percent of the Component now. Some minor details are yet to be done .



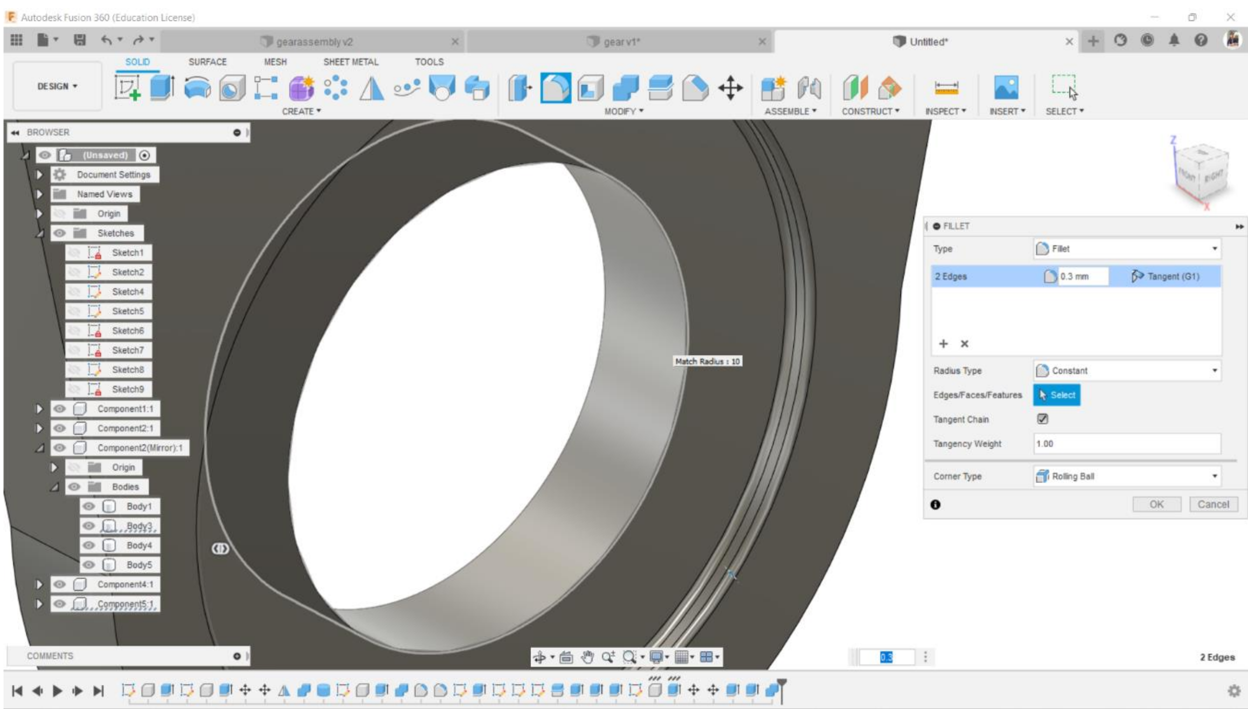
23. Choose the Right Face of the inclined portion of the component and click on create sketch then draw a 20mm diameter circle as shown in the picture.



24. Now exit the sketching environment and **Extrude** the portion between inner and outer boundary of the sketch to **-1.118 mm** press give it as **New Component** and drag it till it alignes with the back portion and leaves some space in the front using **Move and Copy Tool** under the **Modify Enviroment.** Then mirror the component to other side using Mirror Tool available under Create Enviroment and then combine all the componets, making it one Part using Combine Tool.

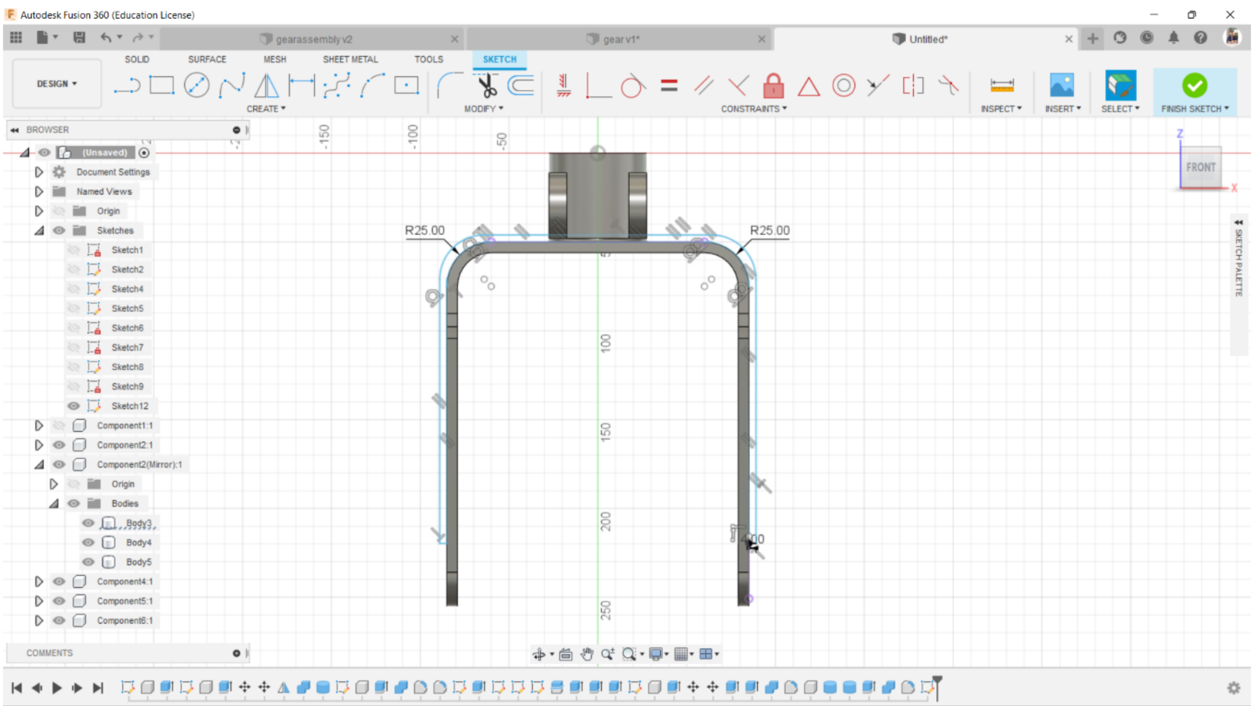


25. Now choose the edges of the circle and give them a fillet of **0.3 mm** using **Fillet Tool** undr the **Modify Enviroemnt** and repeat the procedure to other side as well.

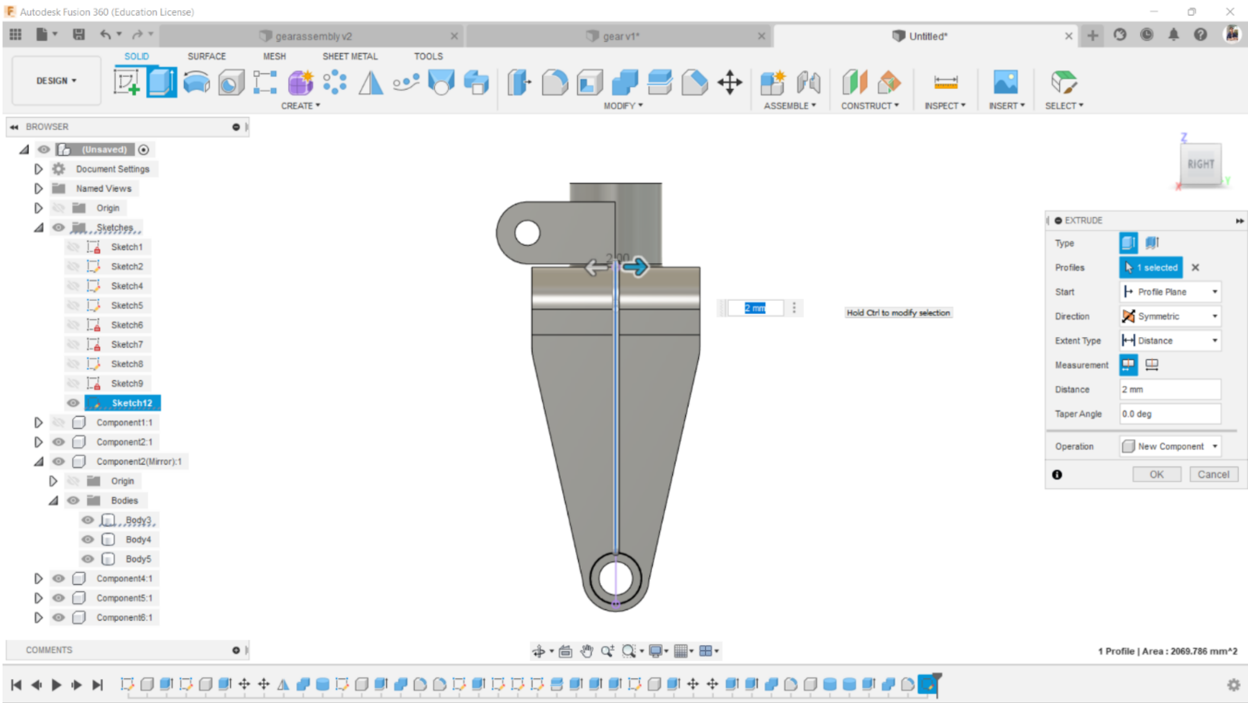


26. After that click on Create Sketch and choose the Front Plane to draw on it.

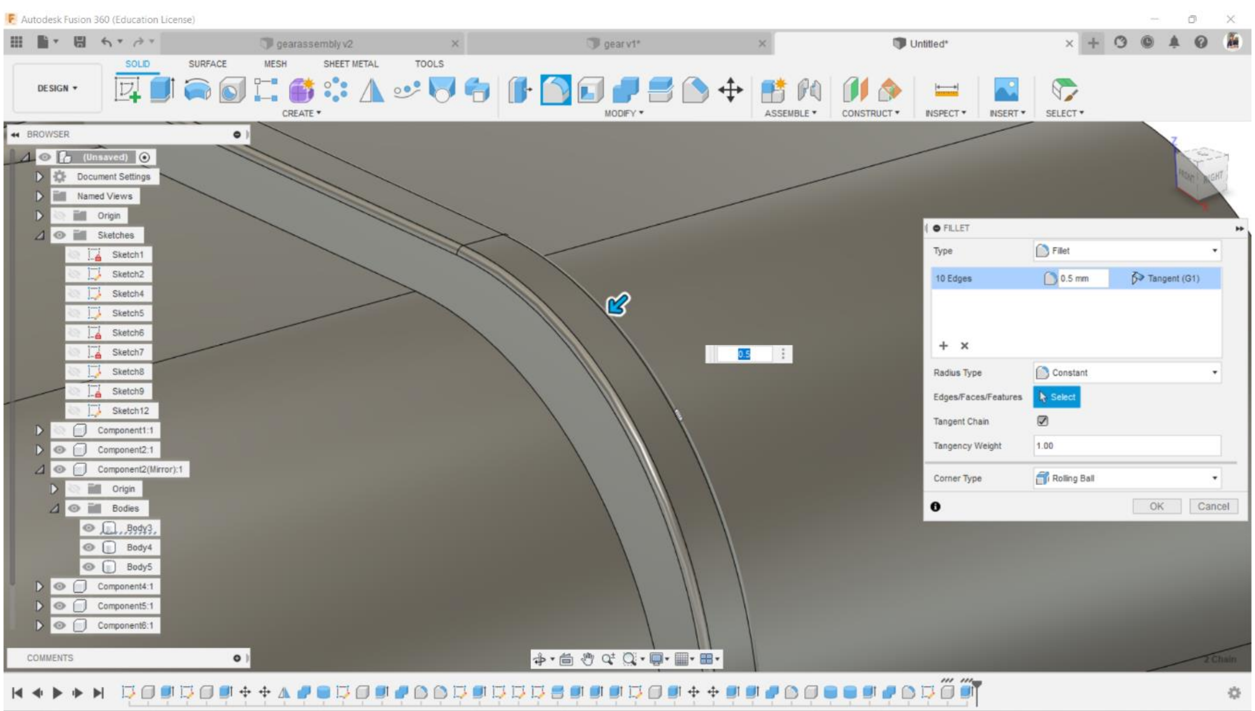
27. Make a **2 Point rectangle** sketch extended **4 mm** from the component and give **Dimesions as 175 mm x 170 mm** then close the sketch by **offseting** it to **4 mm** and give sketch fillet of Radius 25 mm as shown in the picture.



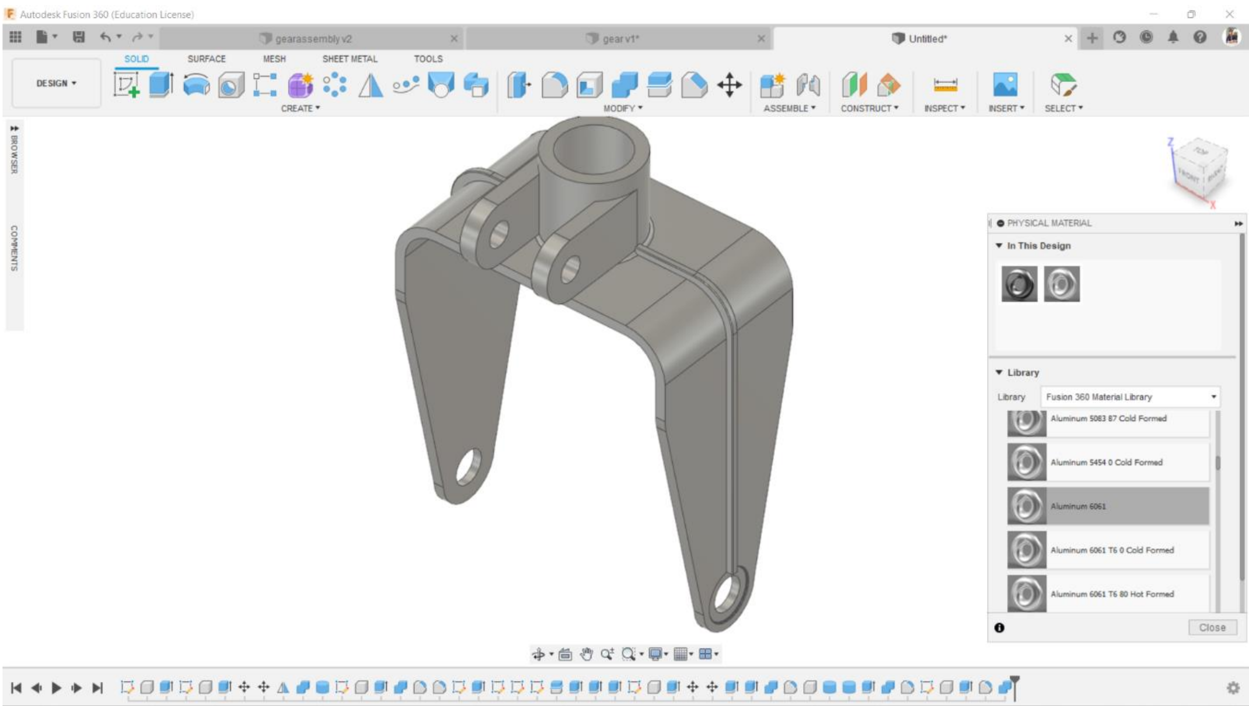
28. Finish the Sketch and **Extrude** the Drawn Sketch by giving **Direction** as **Symmetric** and **Distance** as **2 mm** and Press Ok.



29. Fillet the Edges of Newly Extruded Component by giving Fillet Radius as 0.5 mm and combine the Extruded Component with Parent Component.



30. Our Landing Gear Part is now ready , now we'll apply **Physical material** From the **Modify Enviroemt** and select **Aluminium 6061** to it . and our part will be ready for **Generative Design**.



Generic Design Review

Let's take a short look at some of the properties of the landing gear component we just created.

From the properties of the design, the weight of the part is 963.27 grams. Choice of material is Aluminium 6061, and the manufacturing method used is Die Casting.

Now that we have a good understanding of the standard design approach, let us move on to the generative design approach to design.

Generative Design

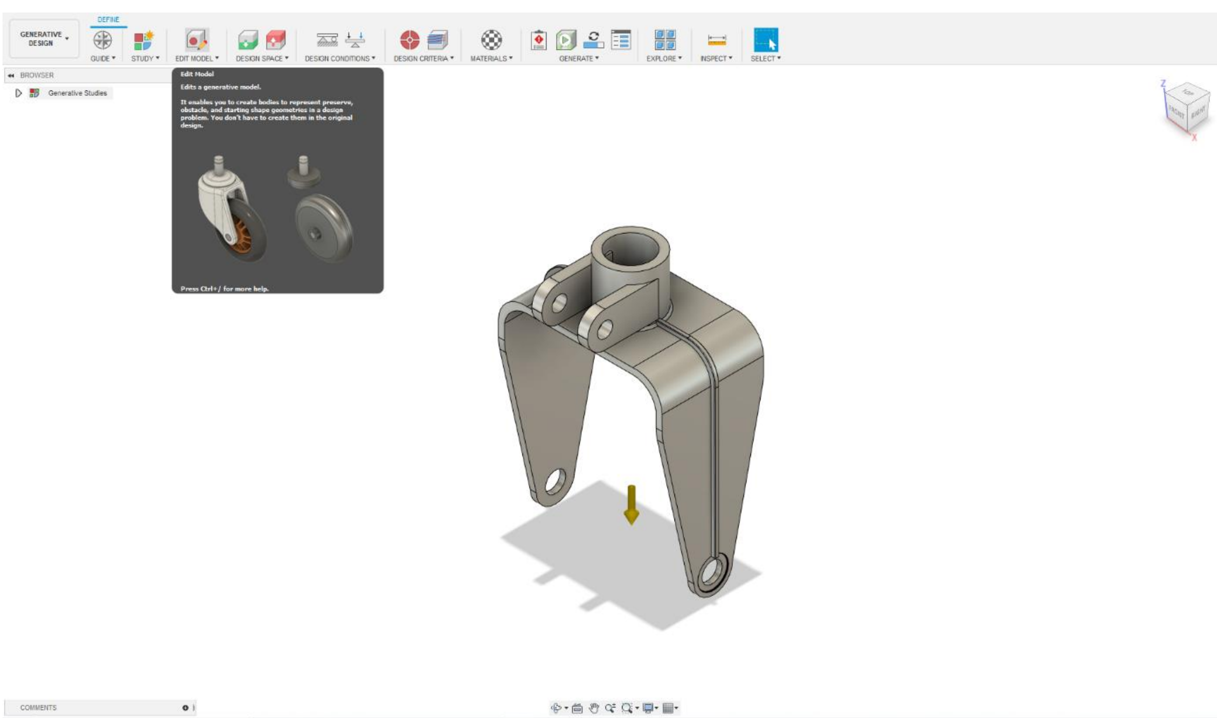
Now that we have our starting form (or first model), we can go on to the Generative Design process. There are several inputs that the software requires from the user in order to continue with the generative design process; some of these are required, while others are optional. In the sections that follow, we will look at what each of these factors (or inputs) signifies and how they are defined in the software.

There are a few changes that need to be made to the model before we begin the generative design process. As we progress through the processes, the importance of these adjustments will be emphasized. One factor to bear in mind is that from now on, all of our edits and definitions will be done in the "Generative Design" workspace.

Editing the model

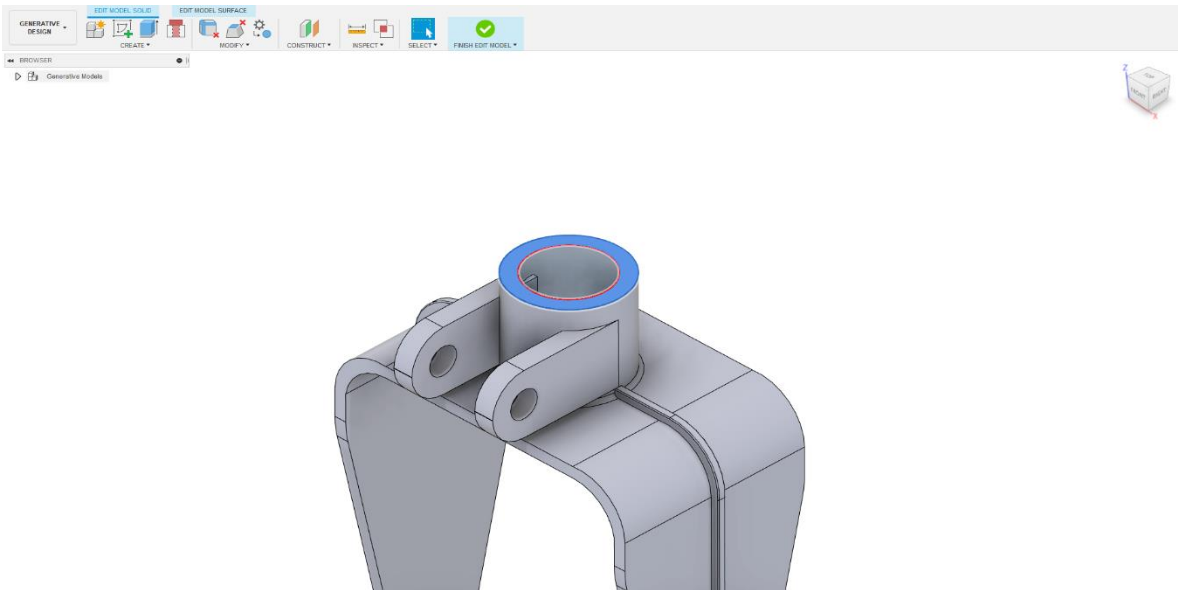
By editing the model, we intend to add or remove a few bodies or features that are required for the main procedures.

To start with, click on **Edit Model** tool, to enter the Editing environment.

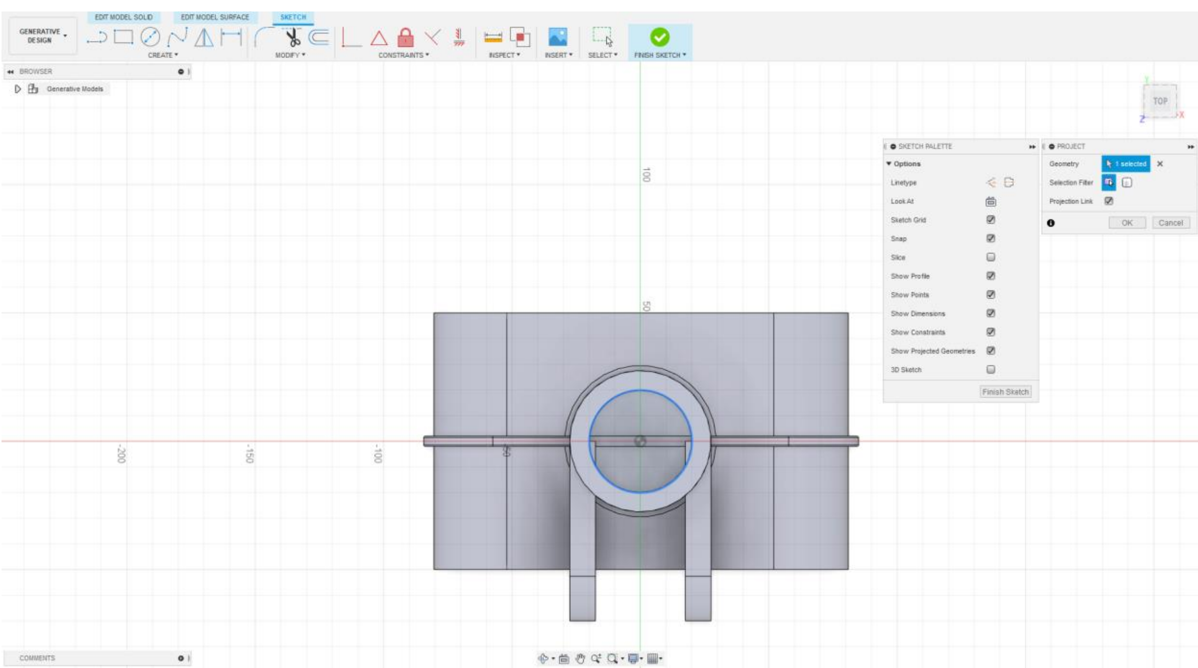


Follow the next steps to edit the model –

1. Let us make a cylinder using the normal design method of sketching and extruding. Select the topmost face, and **Create Sketch**.

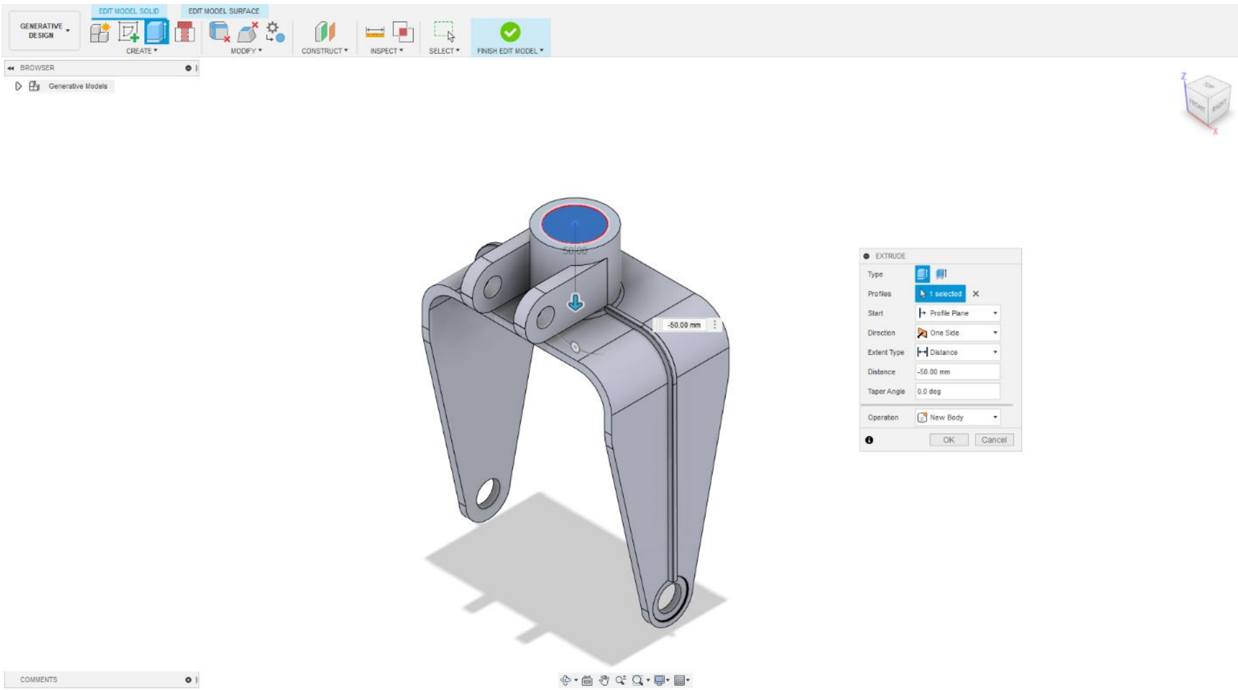


Project the circle (highlighted in blue) using **Project** option (shortcut – P on keyboard).



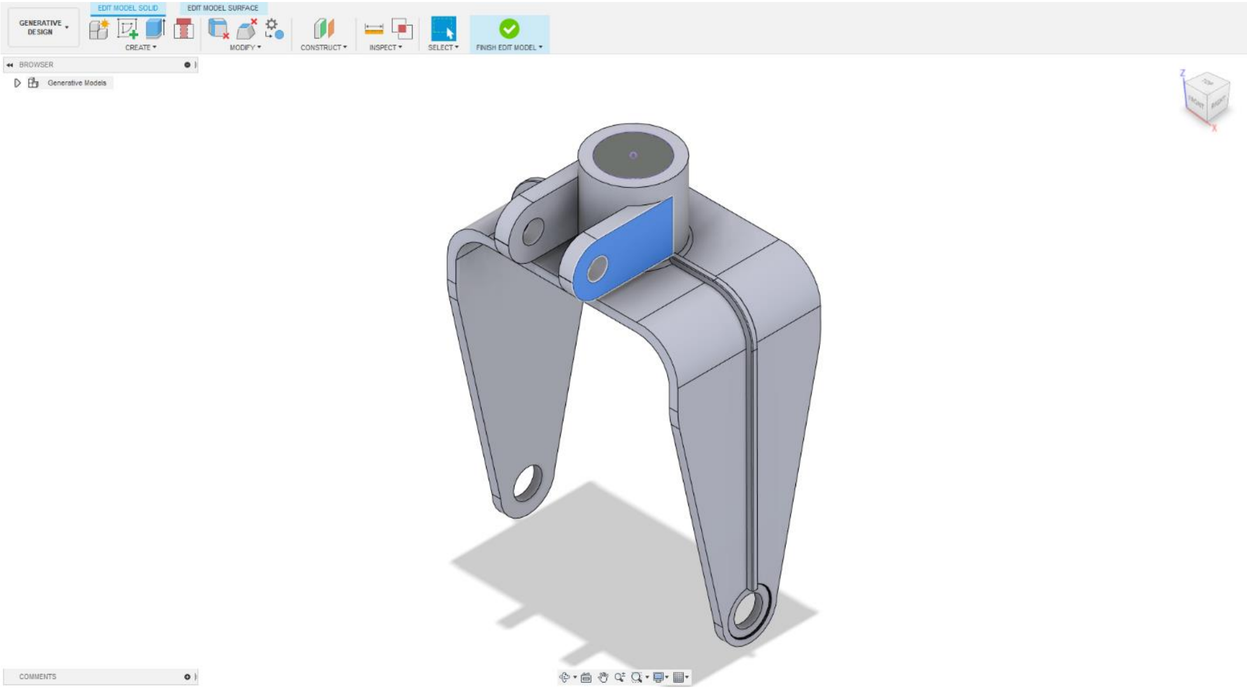
Press OK, and exit from the sketch.

2. Extrude the newly created sketch using the **Extrude** tool with a dimension of -50 mm (50 mm in the negative Z direction). Change the operation to “New Body”, and click on OK.

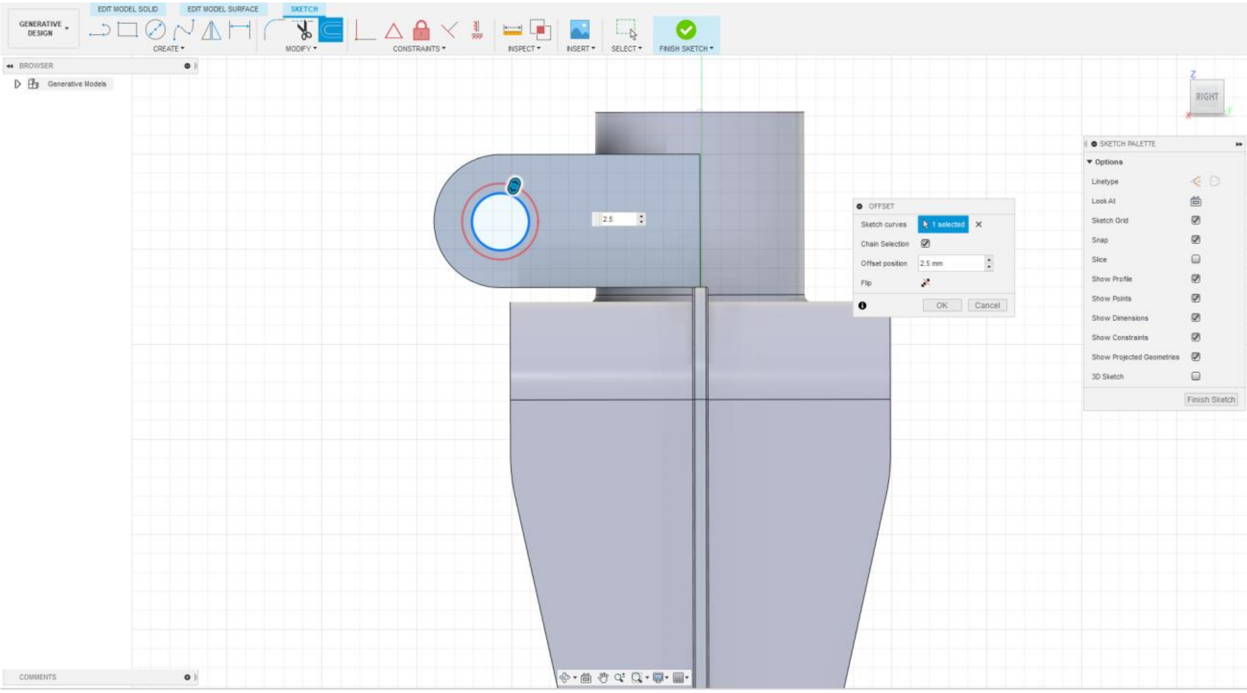


We will be using this body during Obstacle Geometry definition.

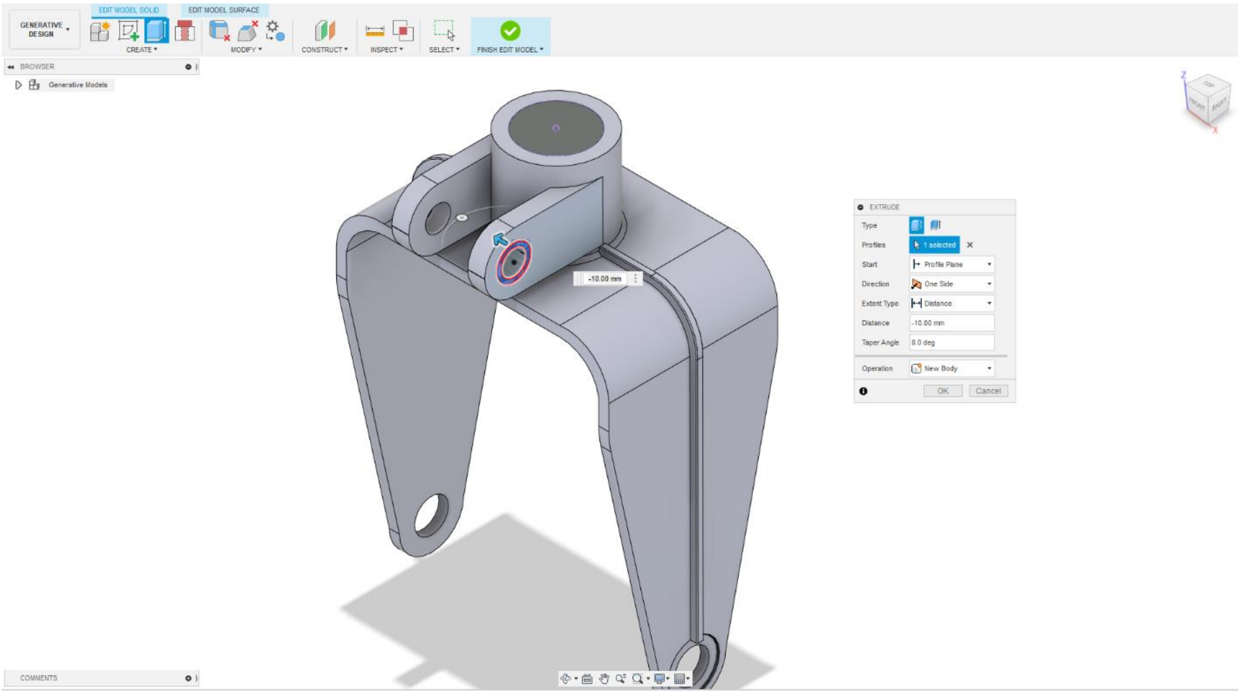
3. Enter Sketching environment with the highlighted face (in the picture) selected.



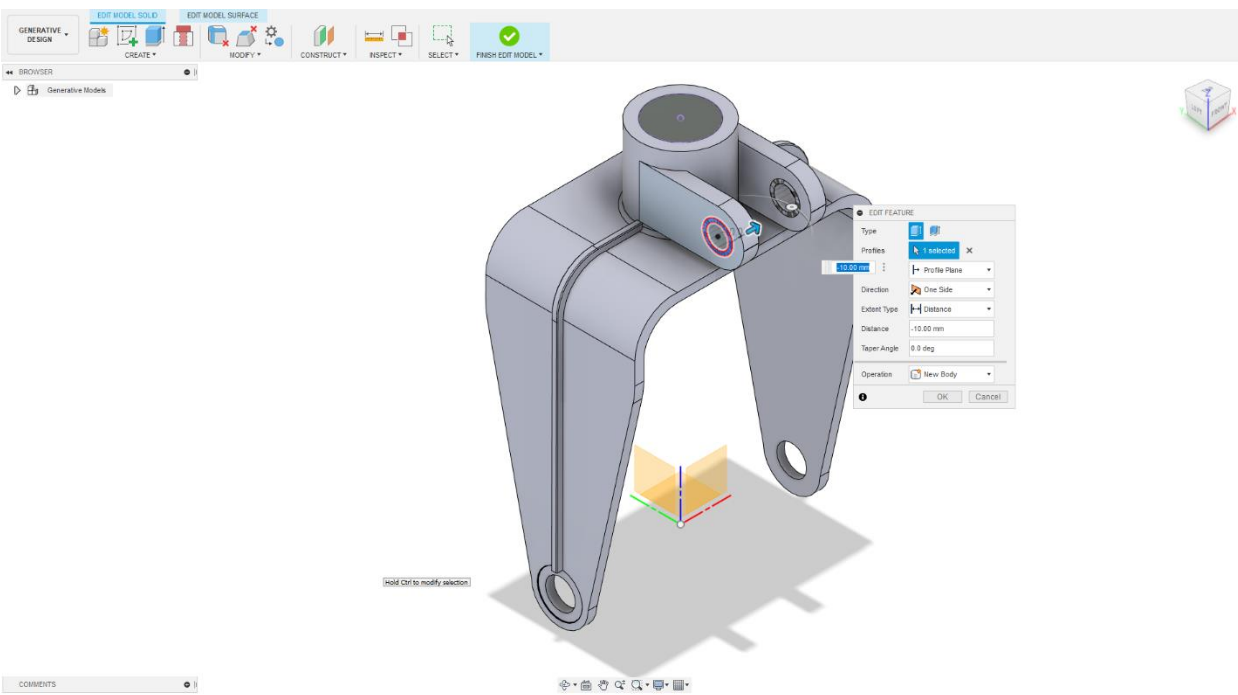
4. Create an offset sketch of the circle, with the offset distance being 2.5 mm outward.



5. Finish the sketch, and select **Extrude** and extrude the sketch upto -10 mm (10 mm along negative X direction), with the operation changed to New Body.

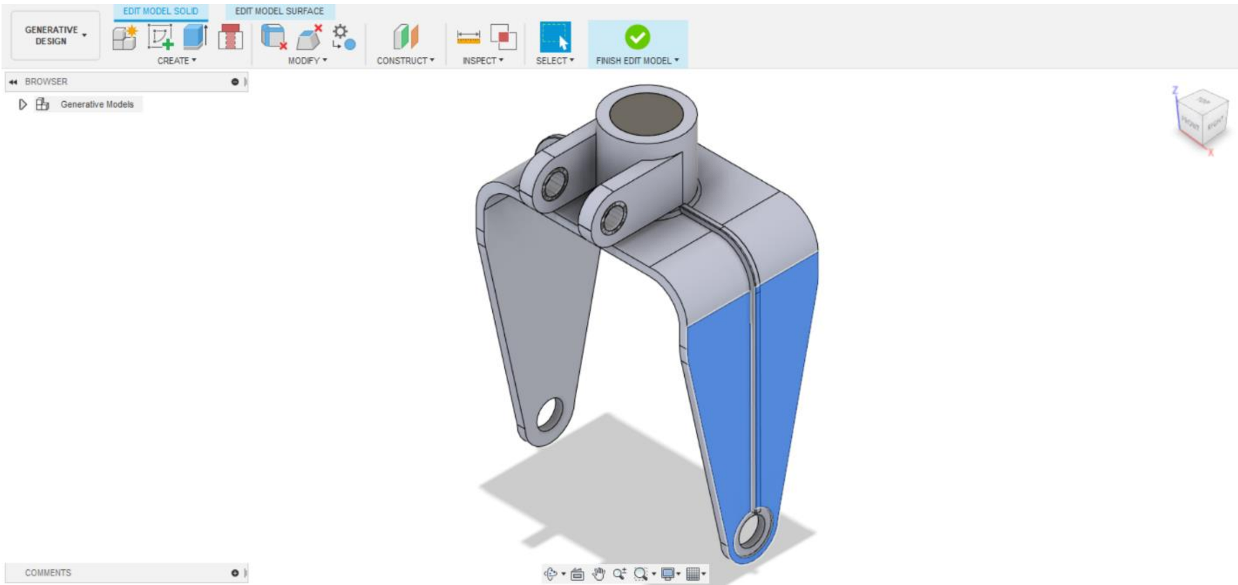


6. Repeat the previous two processes to construct a body on the opposite side.

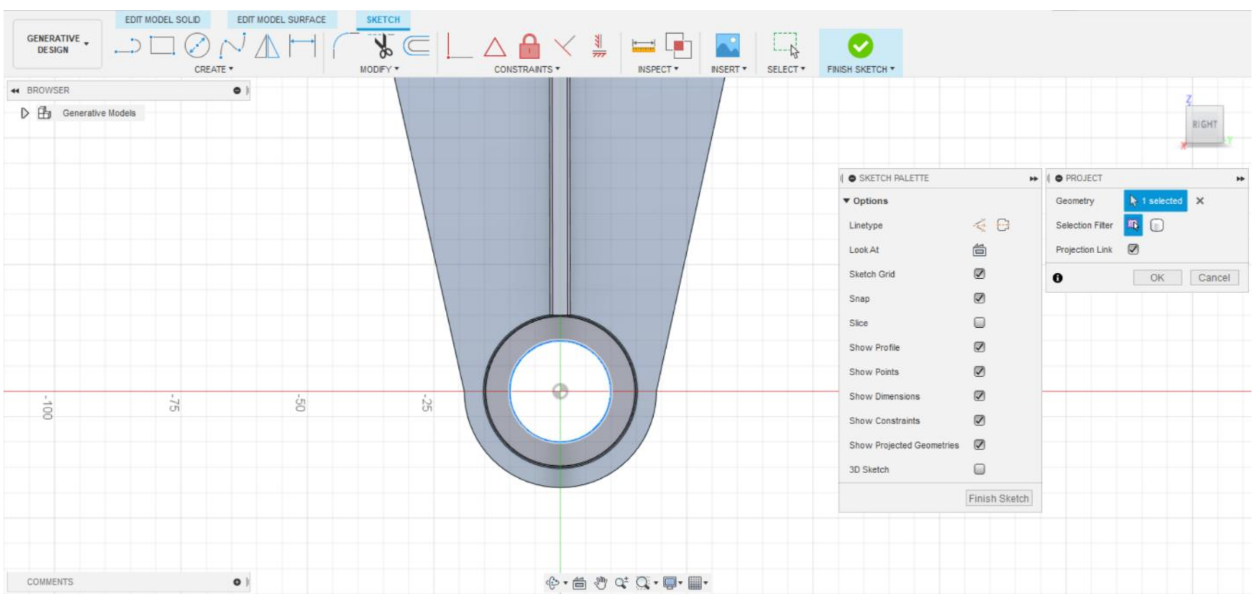


Bodies created in these two steps will be <u>used later in the Preserve Geometry definition</u>.

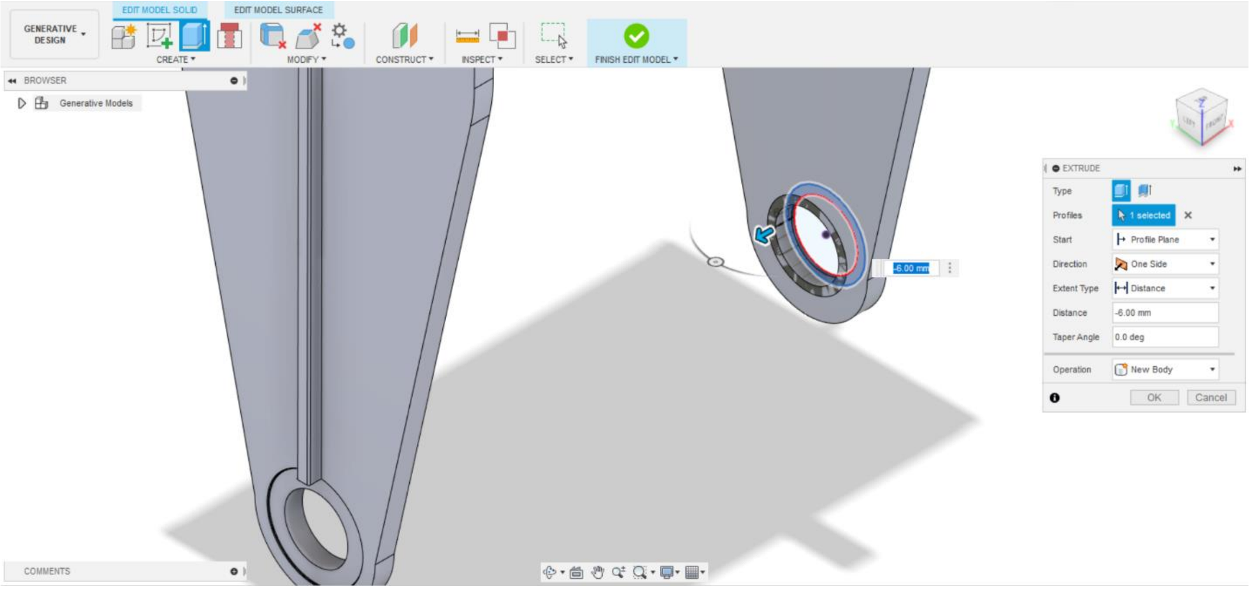
7. Enter the sketch environment with the sketching face to be the face highlighted in the picture.



8. Using **Project** tool, project the innermost circle. After projecting the circle, offset it outward by a distance of 2.5 mm.

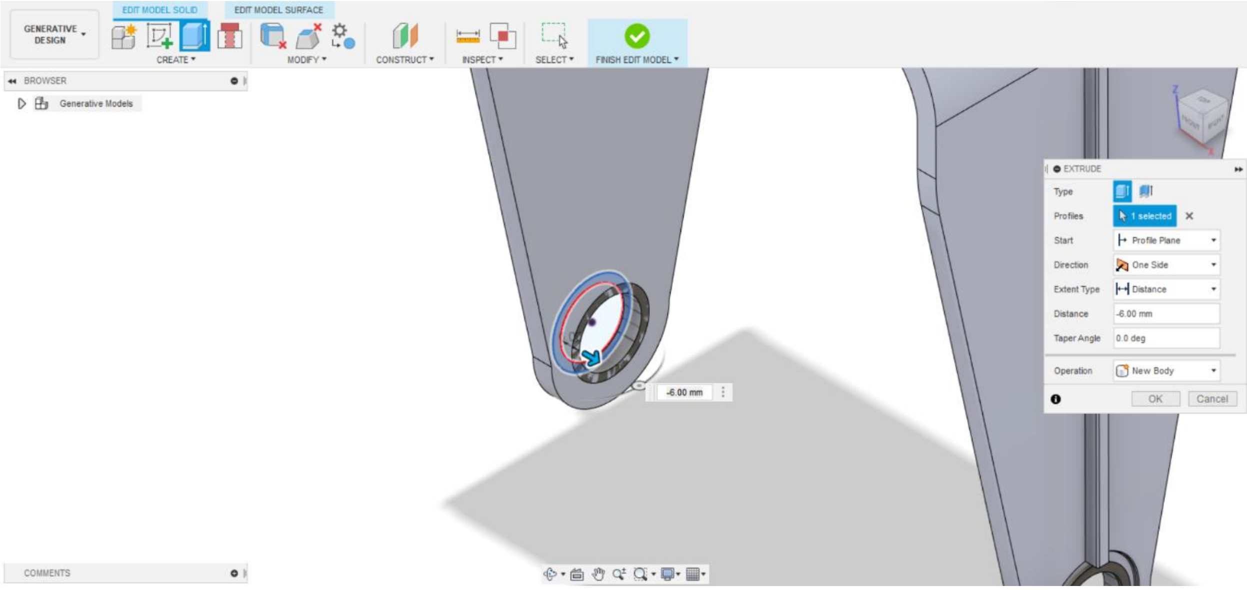


9. **Extrude** the sketch by a distance of 6 mm inward, with the operation being set to “New Body”.



» **PROTIP**: Selecting the face up to which the extrude must be performed is an alternative to providing the distance dimension.

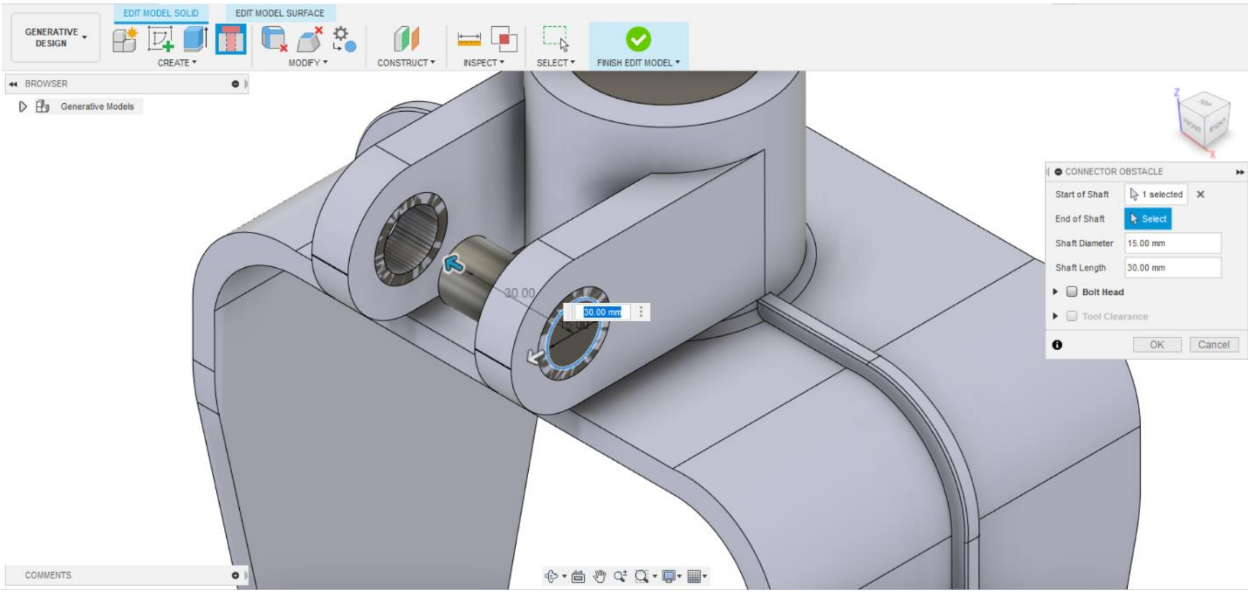
10. Repeat the above step on the other side.



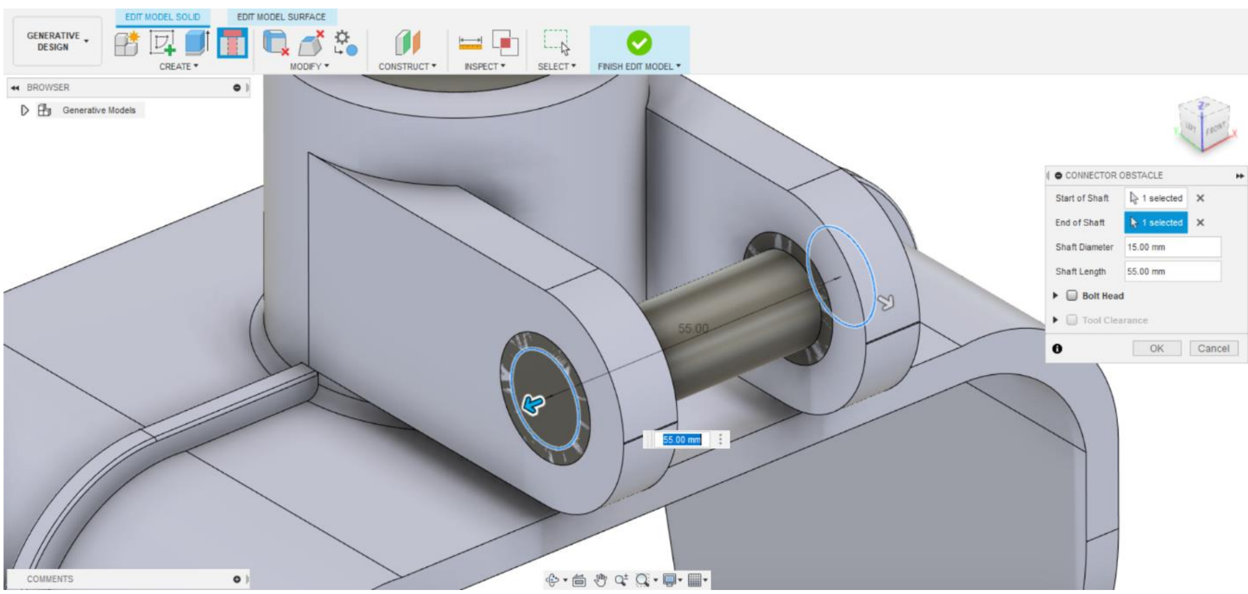
Let us now create some connector obstacles. Connector obstacles creates obstacle geometry in place for bolts or connectors in an assembly.

11. To start with, choose **Connector Obstacles** option.

12. For Start of Shaft, select the circle as highlighted in the picture.

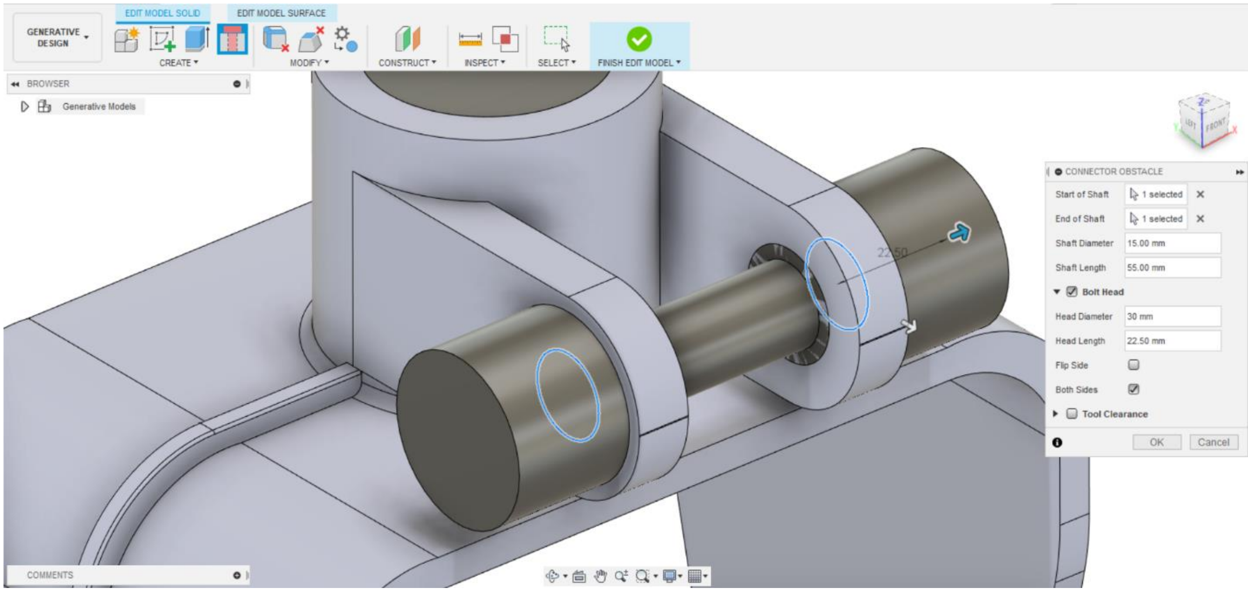


For the End of Shaft, select the circle edge on the other side, as shown:

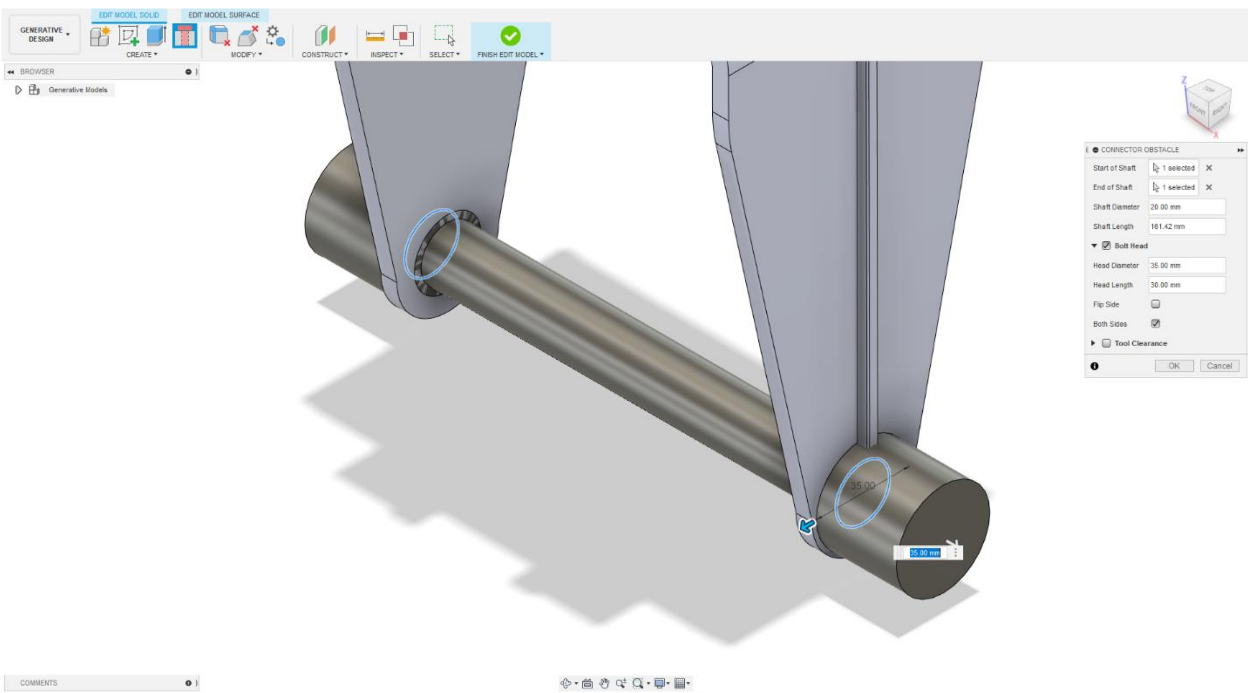


Activate “Bolt head” by clicking on the checkbox, and under Bolt Heaad activate “Both sides”.

We keep the head diameter around two times the shaft diameter as a thumb rule. Hence, change the Head diameter value to 30mm. Click on OK.

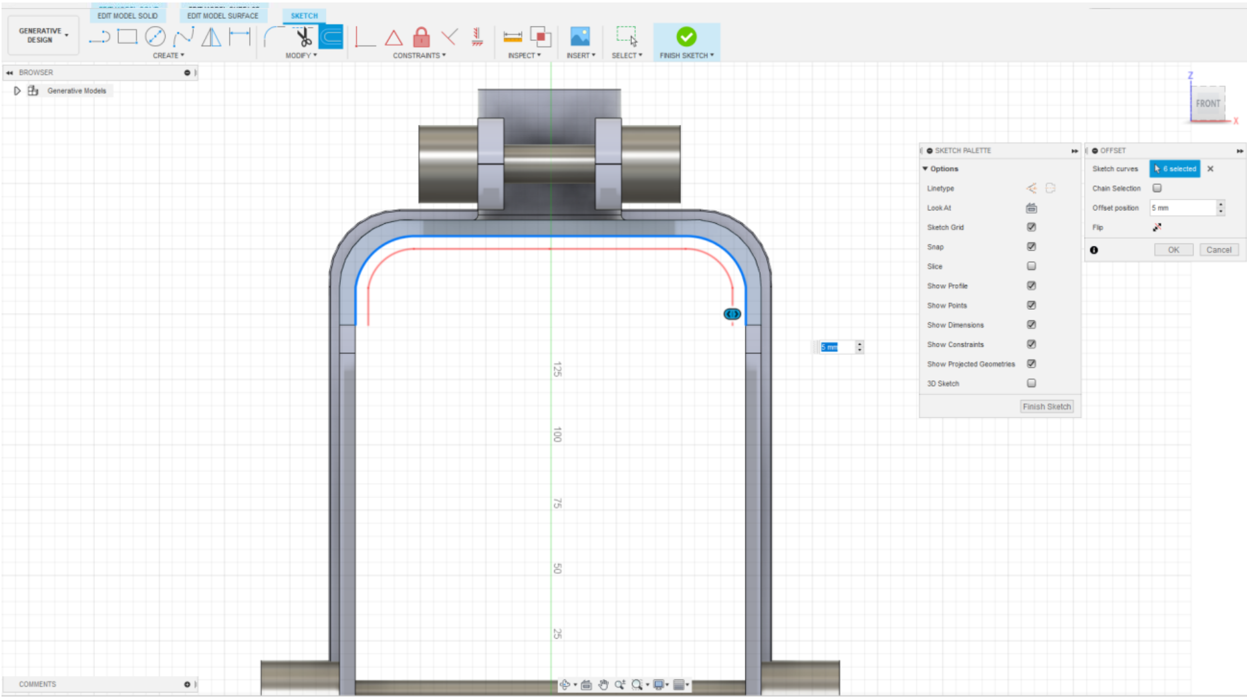


13. Repeat the above process to create another Obstacle for wheel shaft.

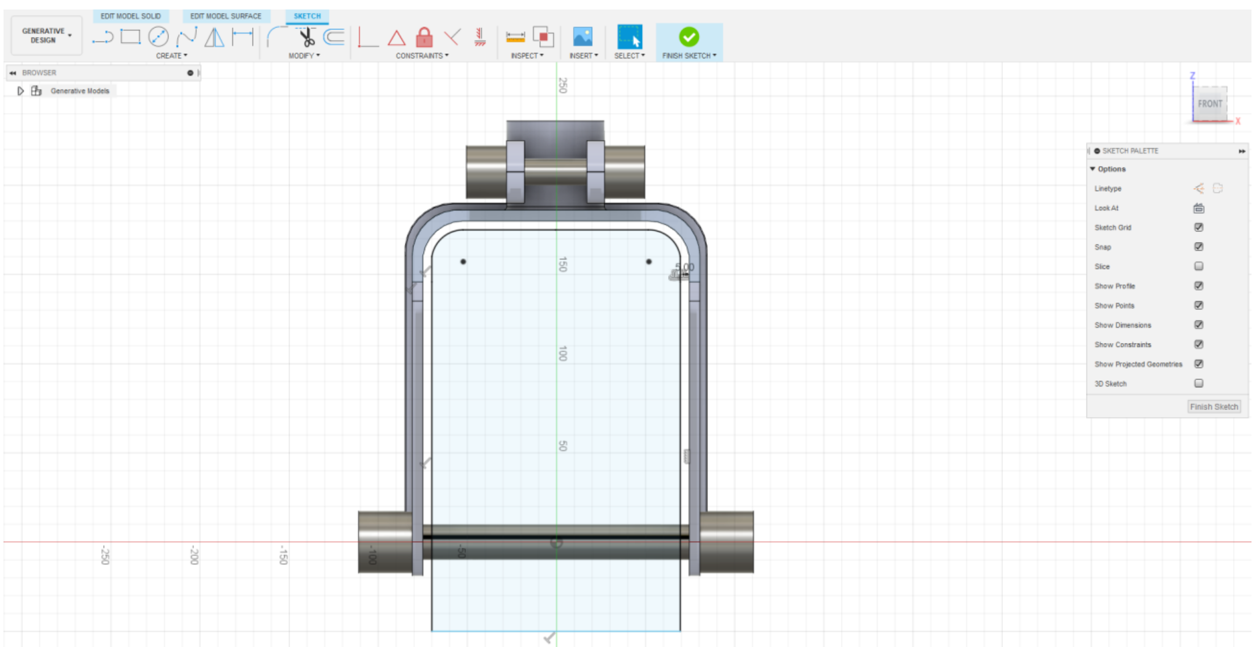


Note here that we have given the Bolt Head diameter as 35 mm (and not 40mm as per our thumb rule), since 40 mm diameter head becomes too huge and goes beyond the edges of our part.

14. Enter Sketch environment with the following face selected:

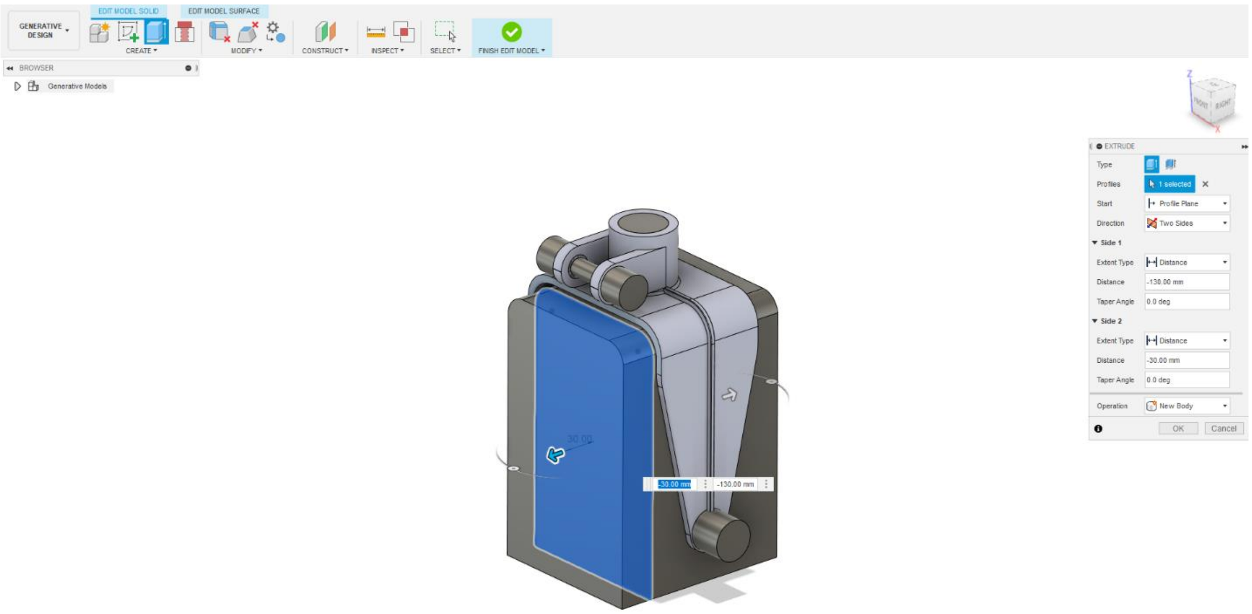


Start by offsetting edges as shown below.

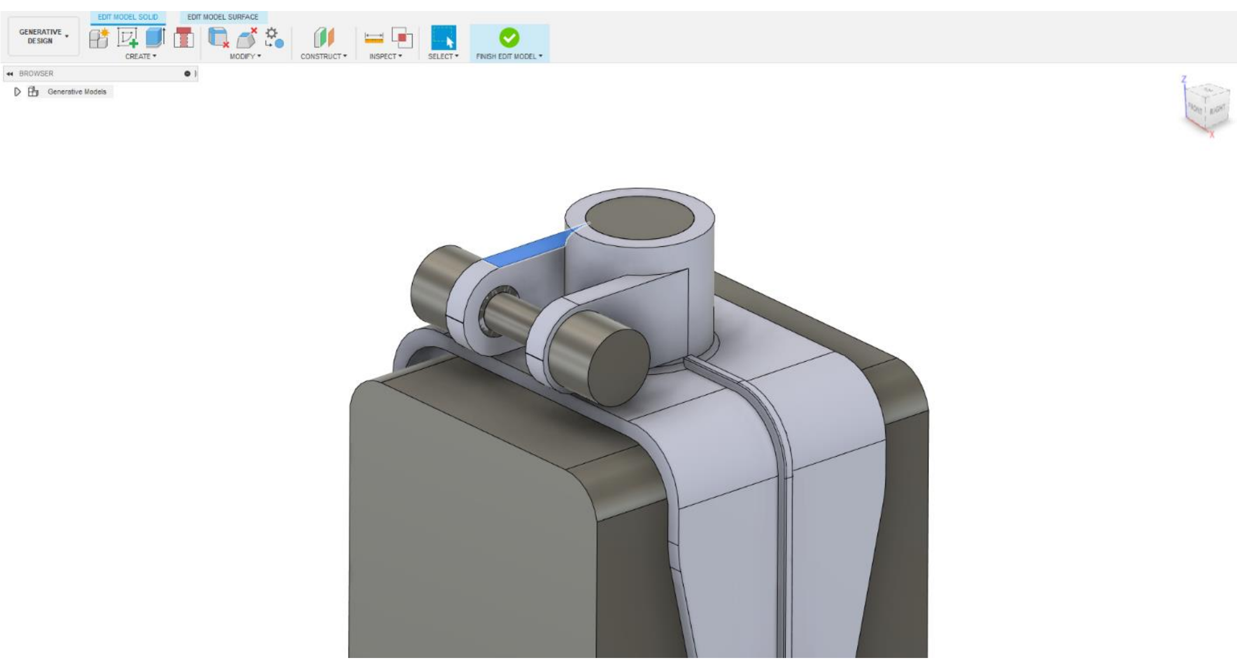


Complete the profile using lines. All the lines are to be horizontal/vertically constrained.

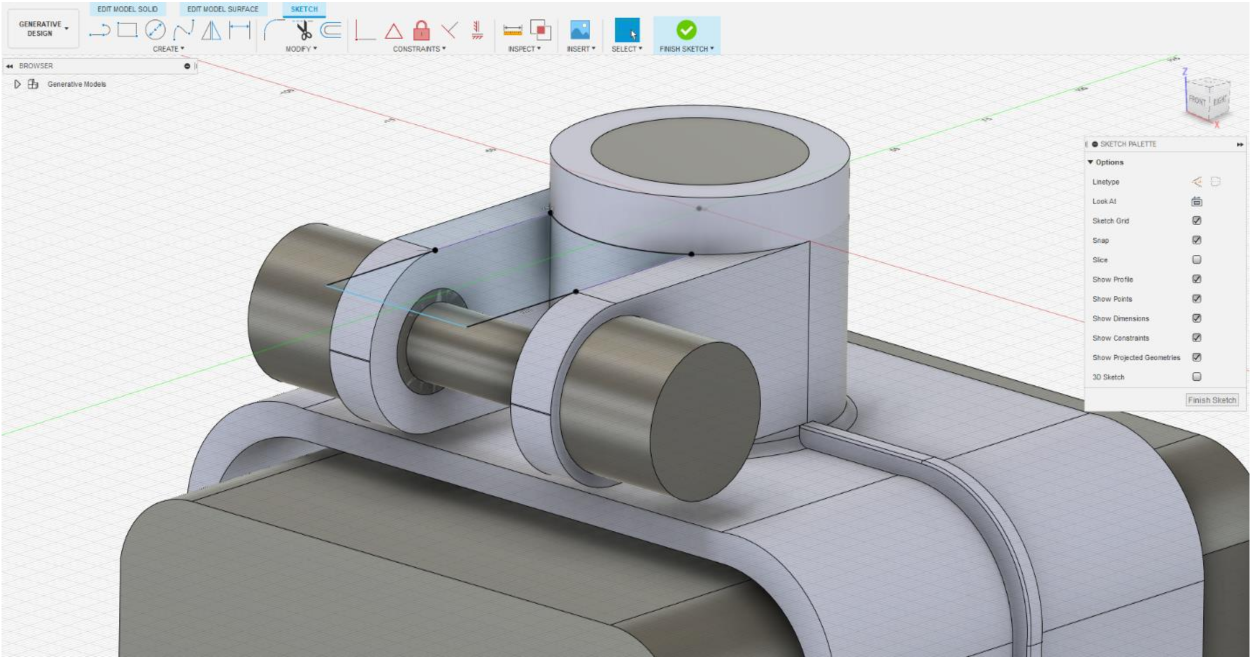
15. Finish the sketch and extrude it to any length necessary to cover the entire void space. Set the operation to "New Body".



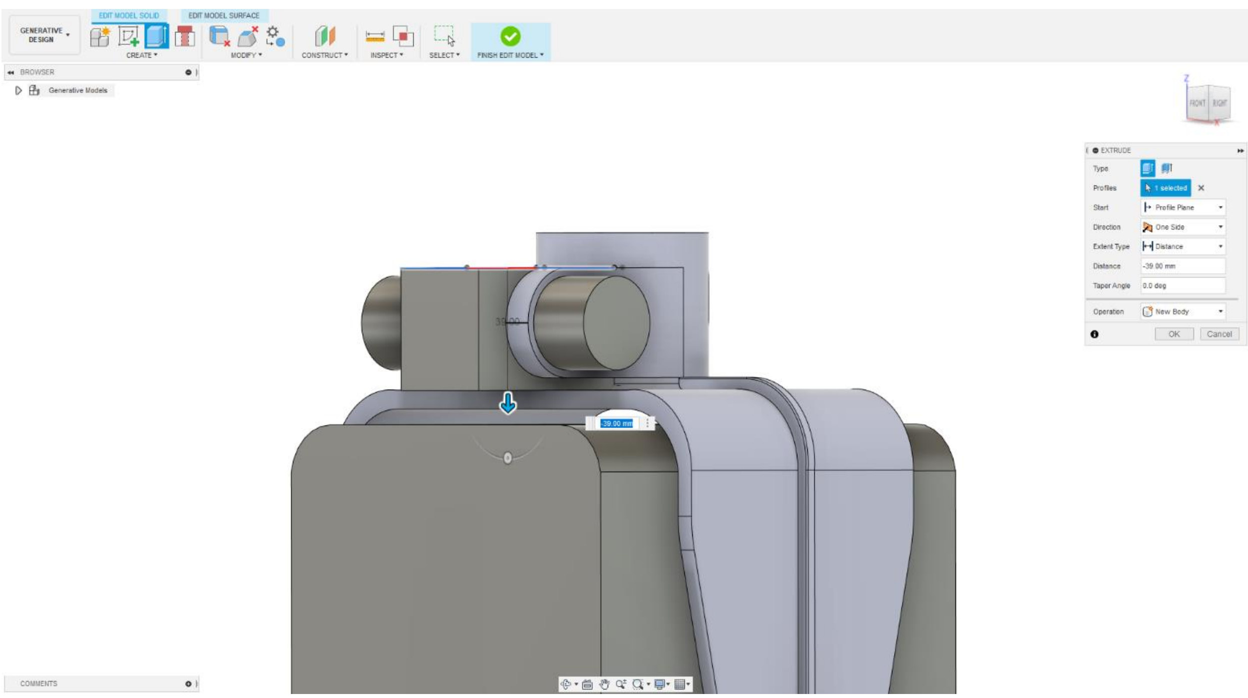
16. Enter the sketch environment with the following face selected.



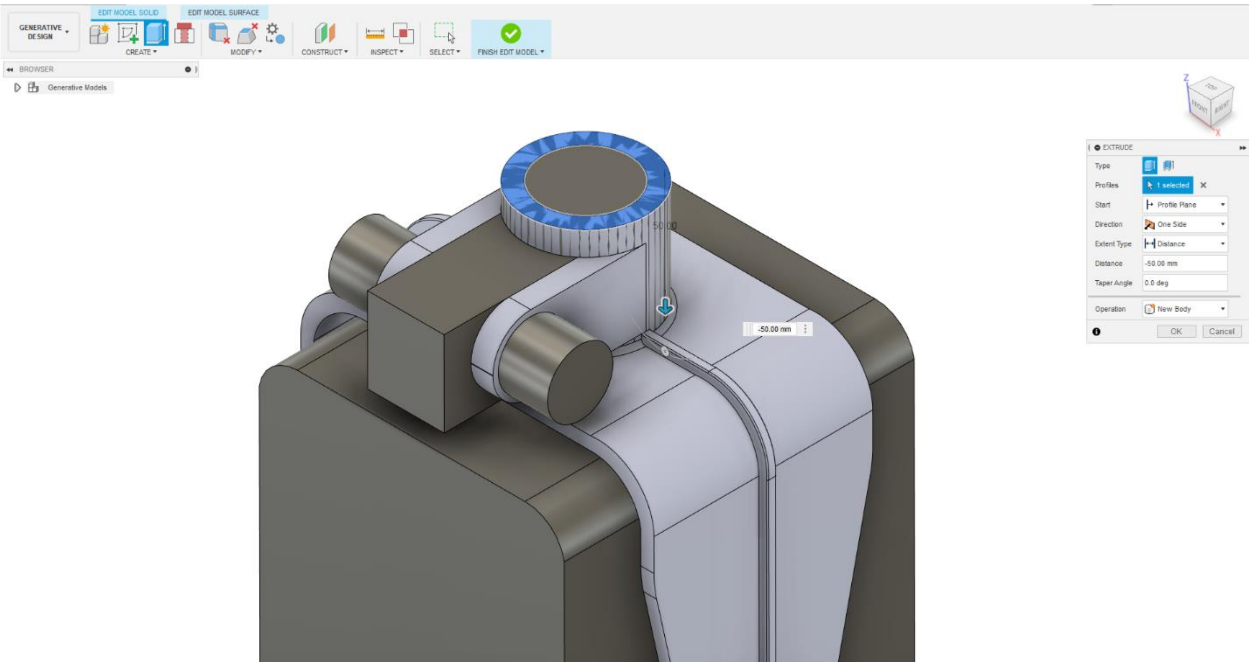
Create a sketch that fits the void space as shown.



17. Finish the sketch, and extrude it until it touches the next face.



18. Select the topmost face, and **extrude** it for a distance of 50 mm inwards.



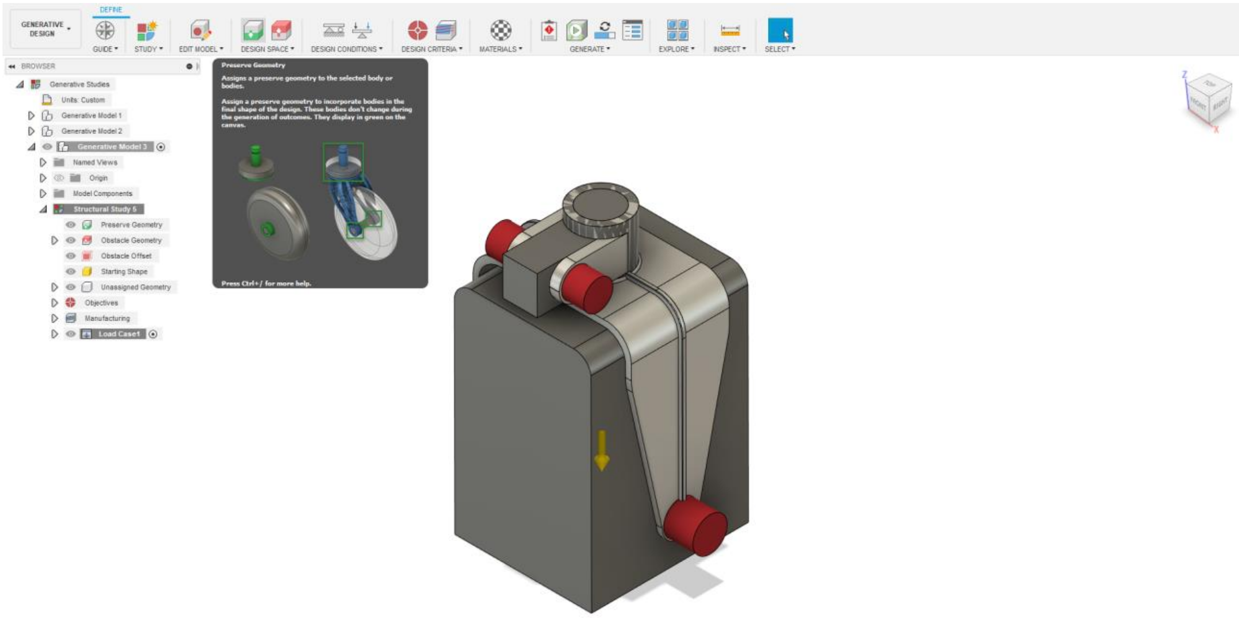
We are now finished with our model edits. We can now proceed with the Generative Designing process.

Preserve Geometry

Preserve Geometry, as the name implies, is a tool used in Generative Design to define the portions of the model that will be retained in the final design. Openings for shafts, bolts, and other parts must be preserved for a range of reasons, the most important of which is that mating parts must have the same geometry, and connectors and these parts are made all over the world.

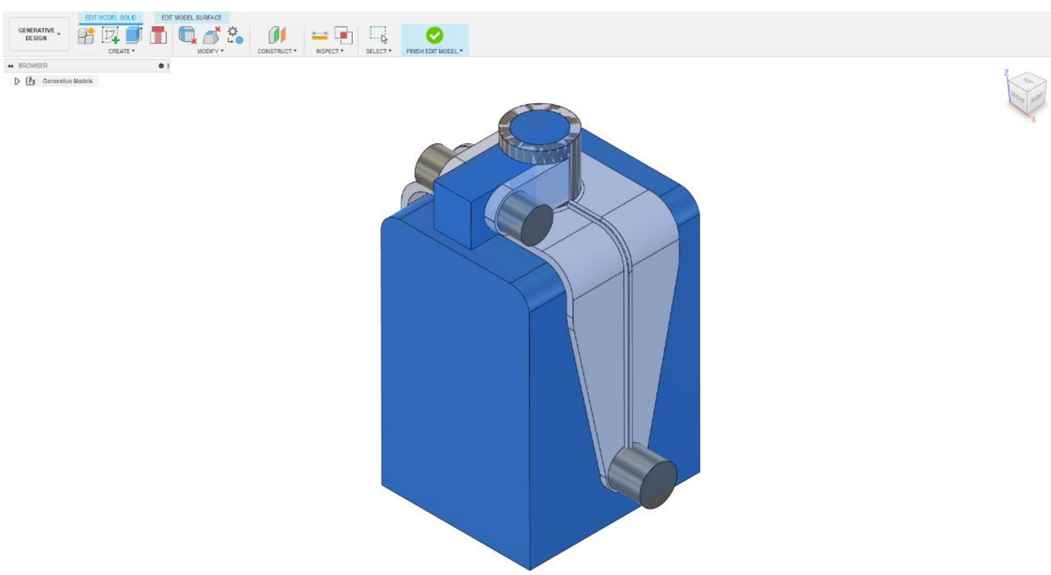
To define inputs for preserve geometry,

- Click on **Preserve Geometry** under “Design space” section.

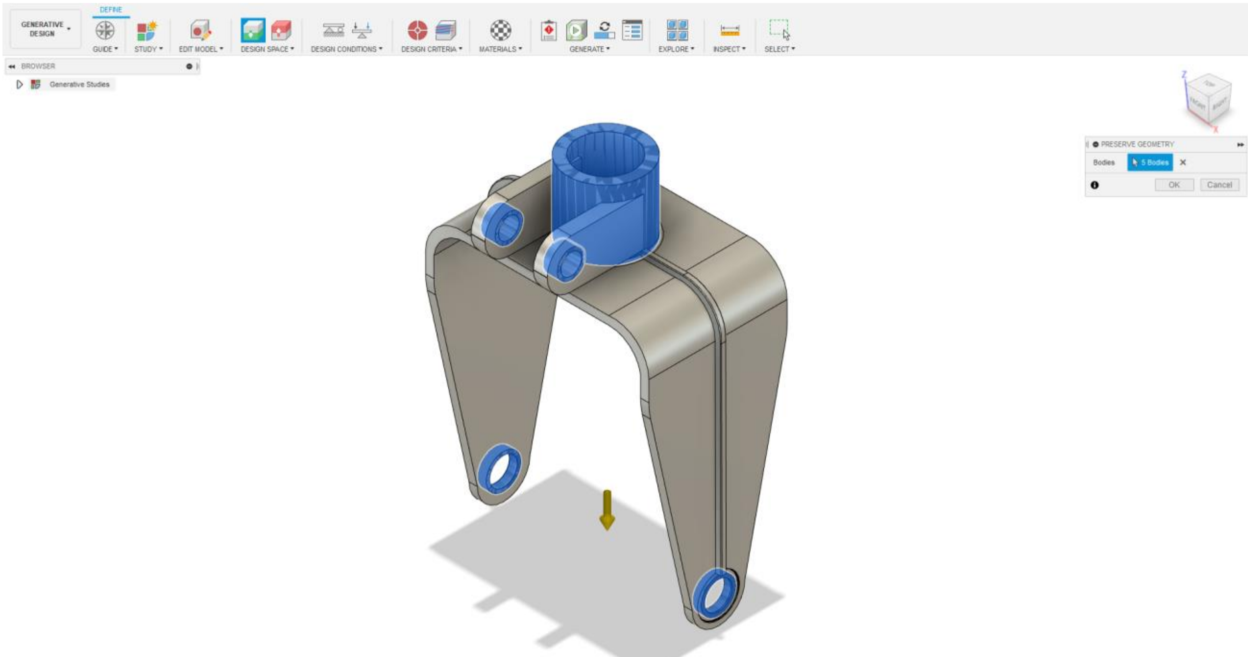


- To avoid visual confusion while choosing bodies for Preserve, we shall hide few bodies during the selection process. This has to be done by going back into **Edit Model** environment.
  Hide the Obstacle Geometry from the Generative Design workspace before going into Edit model by clicking on the eye icon next to Obstacle Geometry.
  » NOTE: The connector obstacles (what we earlier created) are automatically considered as Obstacle Geometry by the software.

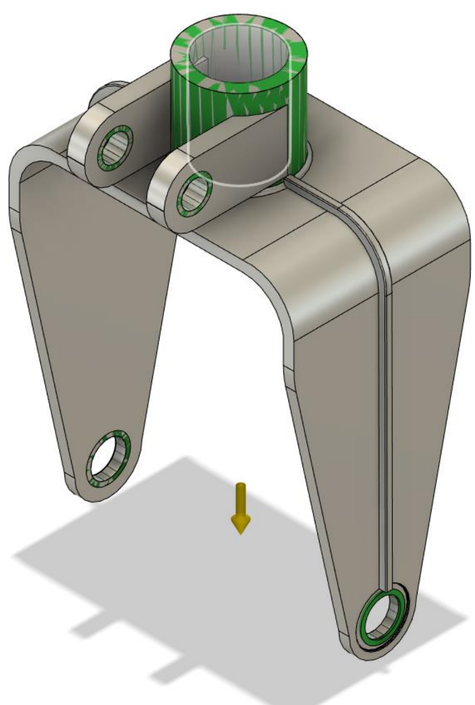
  Going into Edit model, hide the 3 models highlighted in the below picture.
  Do not forget to unhide these bodies after Preserve Geometry definition.



- Going back to **Preserve Geometry** tool, select these three bodies and click on OK.



Once you click OK, the selected bodies turn green.

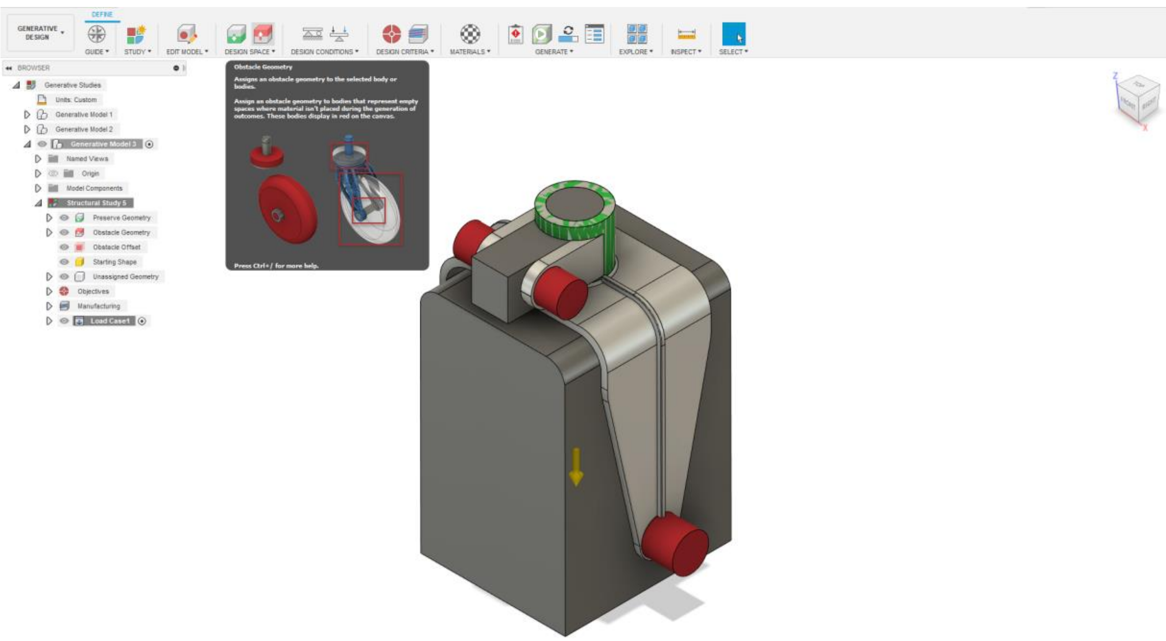


Obstacle Geometry

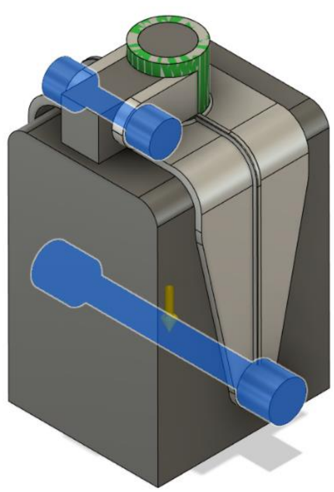
Obstacle Geometry is the go-to tool for defining model regions that the software must avoid throughout the generative design process. The obstacle geometry tool comes in handy when we need to avoid material in certain places in our design when other parts, such as connecting rods and dampening components, are in contact with this present part.

Defining Obstacle Geometry regions is similar to defining Preserve Geometry regions. Before moving ahead, make sure that all the hidden bodies are made visible.

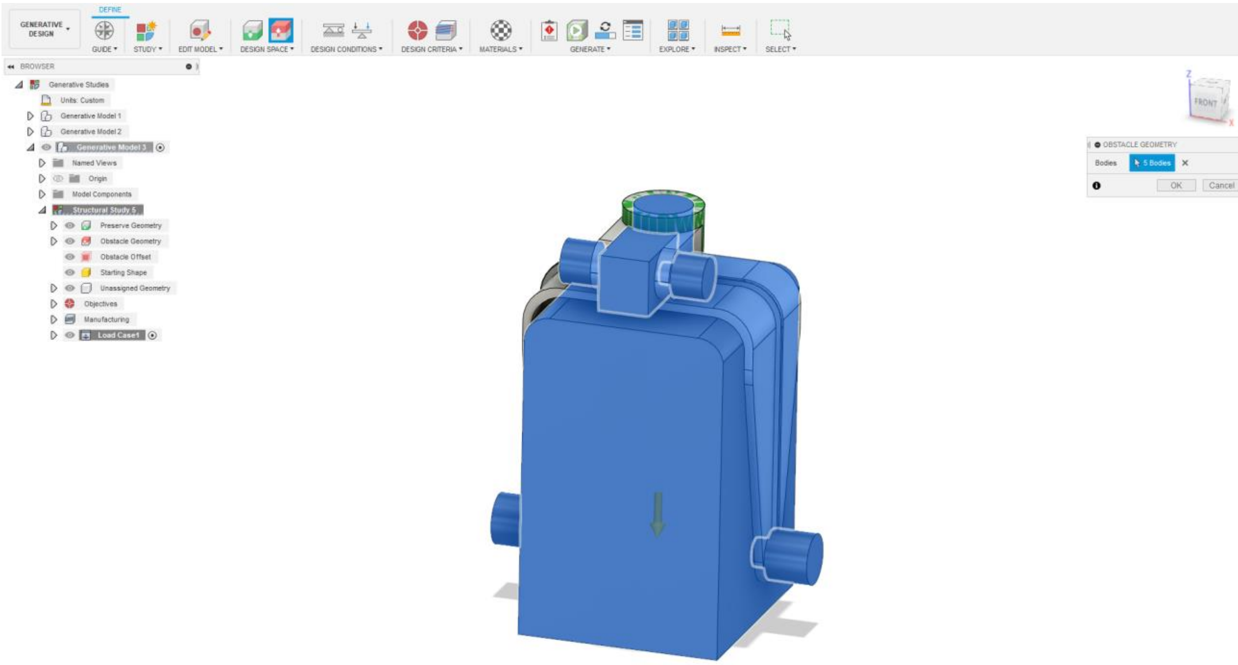
Start by clicking on **Obstacle Geometry** tool under “Design Space” section.



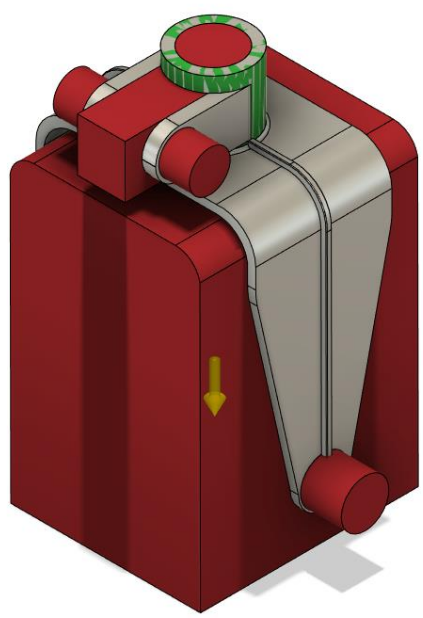
As mentioned earlier, the software automatically links the Connector Obstacle Geometry to the Obstacle Geometry here.



Select three more bodies displayed in the image to add to the bodies for Obstacle Geometry. Click OK.



Now the bodies part of Obstacle Geometry are visible in Red color.



Starting Shape

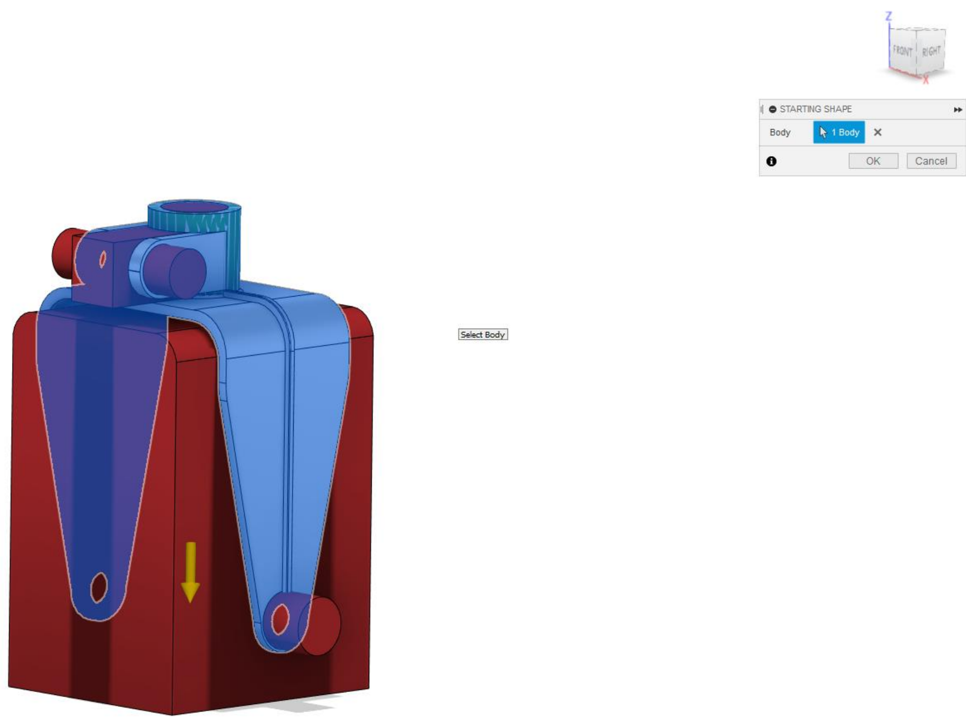
This is one of the few optional inputs to the software.

In general, the generative design algorithm identifies the preserve and obstacle geometries before adding material in the remaining space while performing a simultaneous linear static stress analysis of the part. Generative Design is an "iterative" approach, which means that each design goes through several iterations during which the software learns what works and what doesn't from the preceding iteration.

Our intention is to improve on an existing design. We aim to improve the design, thus we're using the keyword "**optimize**." This is accomplished using the Generative Design algorithm's "*design exploration*" approach, in which the software generates multiple variants of the same design that we would not have imagined or been able to achieve.

Using the Starting shape tool in the Generative Design workspace, you can key in the starting design. This is the starting stage for the machine, from which it will advance through enhancement.

Click on the pencil icon next to **Starting Shape,** and select the landing gear part (as shown).



Once defined, the landing gear component turns yellow.

Structural Loads and Constraints

Now we apply the loads and constraints that the machine would consider during Generative Design process.

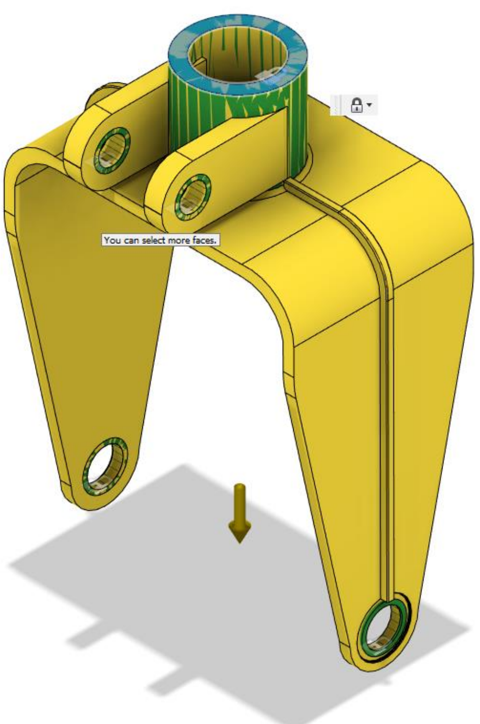
Constraints

Constraints are used in locations where linkages with other sections of the assembly exist or are fixed in nature.

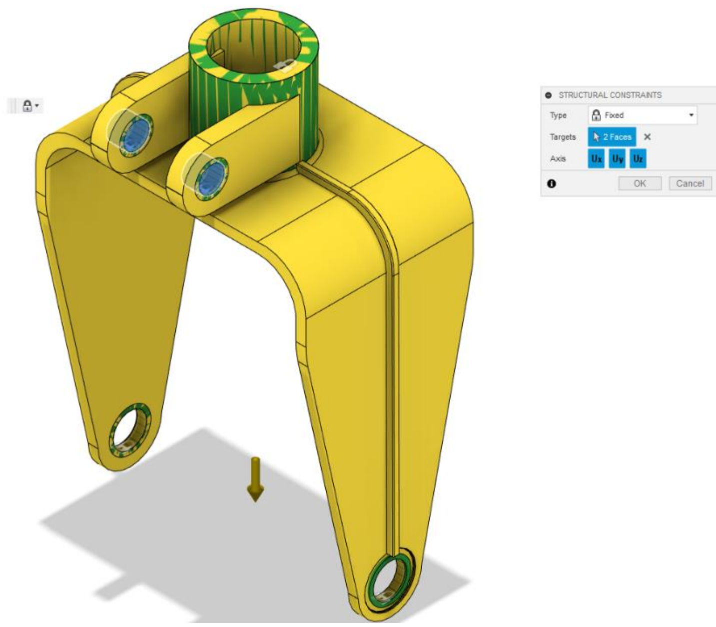
Going back to our model,

- Let us start the step by hiding our Obstacle Geometry.
- Open **Structural Constraints** tool under "Design conditions" section, and apply three separate fixed constraints in the following manner:

  a. Fixed1 – Topmost Circular face
  b. Fixed2 – Inner faces of two holes in upper part



FIXED1

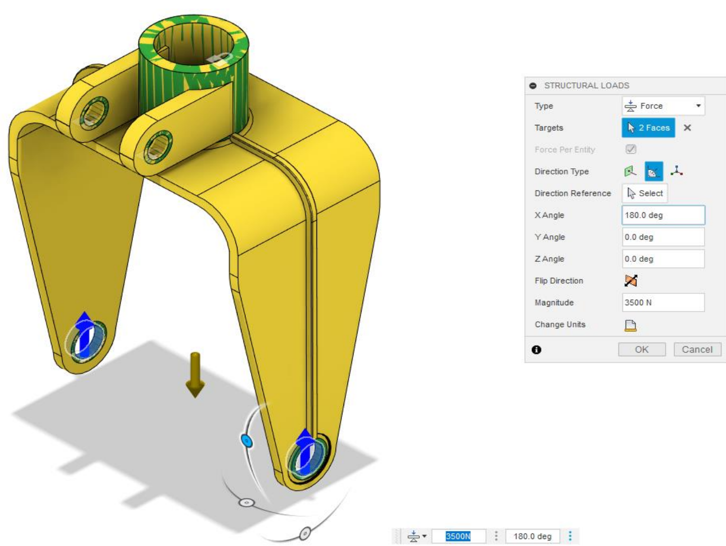


FIXED2

» **NOTE:** Loads and constraints are only allowed for Preserve Geometry. As a result, in order to use these, that specific region must be defined as Preserve Geometry.

<u>Loads</u>

We apply 3500N force on each of the two holes in the lower part of the Landing gear component, by making use of the **Structural Loads** tool.



Design Criteria

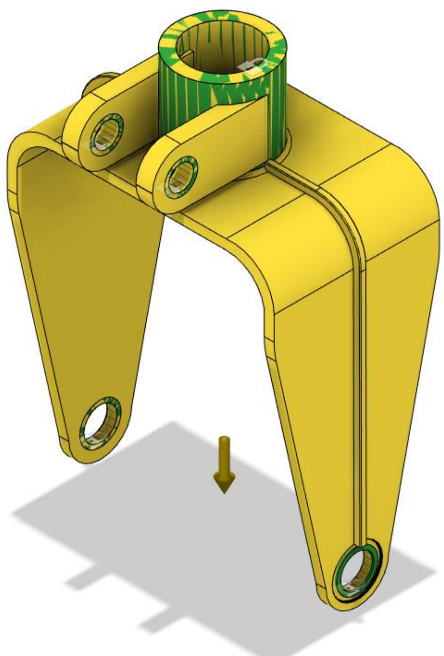
Two inputs are given under Design Criteria:

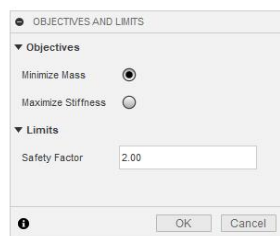
- **Design objective**

  We can specify our goal, which can be either to minimise mass or to maximise stiffness. Following the minimization of mass, the focus will shift to obtaining a specified Factor of Safety (FOS) as defined by the designer. The default minimum FOS is 2.

  With the maximising stiffness option, we may give a certain mass target, and the software will develop designs that fulfil both the stiffness criterion and the mass requirement.

  We look to minimize mass with the default FOS of 2.





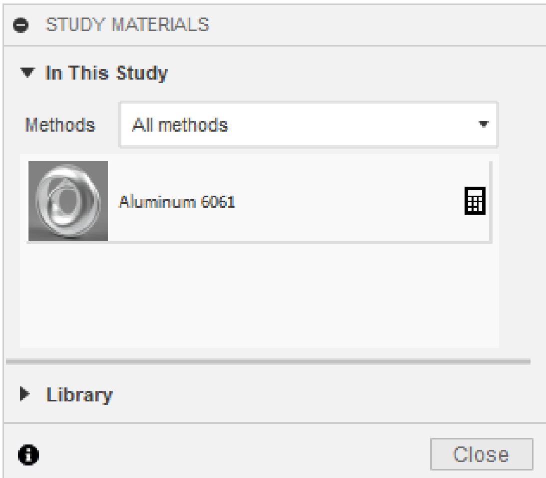
- **Manufacturing methods and Materials**

  We can additionally specify one or more manufacturing methods for generating our part. This is because the system generates outcomes that are tailored to the specific manufacturing method (s).

  In addition, the system expects few materials as input in order to provide the optimum outcomes for each of the given materials.

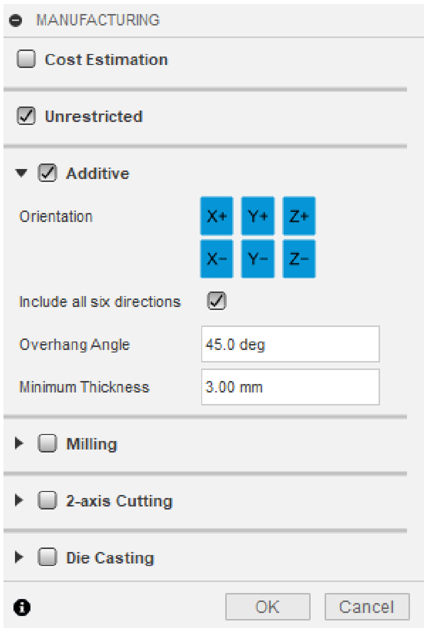
Since we are aiming to just optimize an existing design, we choose the same material from the original design, i.e., **Aluminium 6061.**

» **Note** that we can add upto 7 materials to each of the manufacturing methods we specify.



We want to adopt innovative manufacturing methods instead of standard machining processes with optimization. Because processes such as additive manufacturing can handle complex geometries, this allows us flexibility in the design geometry.

For our case study, we focus on **Additive Manufacturing** as our main manufacturing method. On a side note, if you want to observe how the designs differ for different manufacturing methods, you may add Milling as another option.



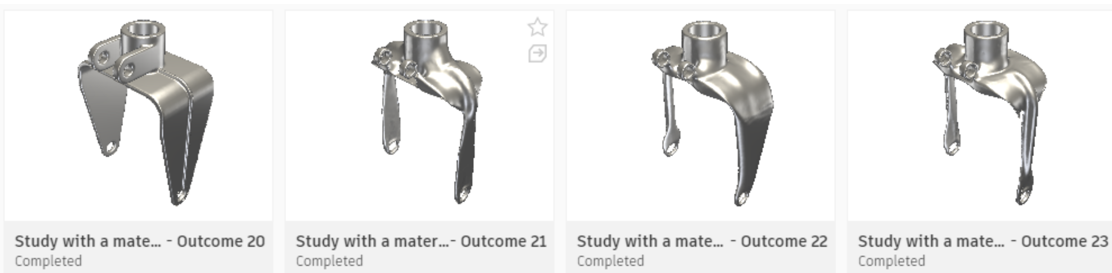
One important aspect to remember is that when we have our goals in mind, keeping the requirements (such as manufacturing processes and materials) to a minimum is always recommended because it saves a lot of time.

Pre-Check and Generating designs

When all the inputs given are correct as per the system's requirements, green tick appears in the **Pre-check** tool (highlighted in the picture).

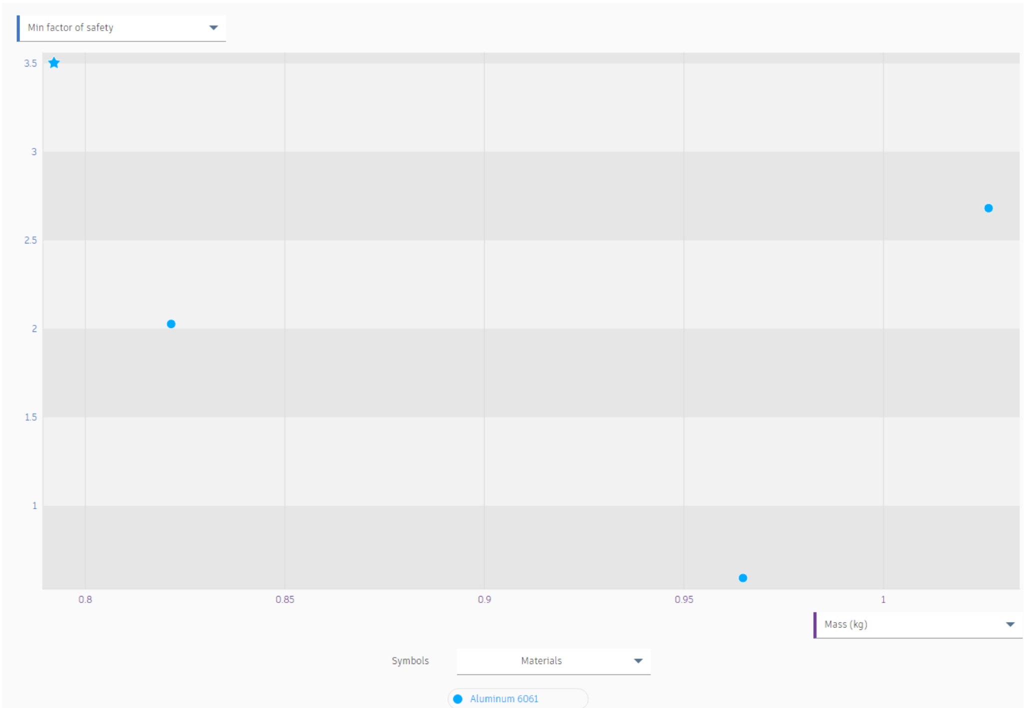
We can proceed with the design generation at the cost of cloud credits. One advantage of Generative Design in Autodesk Fusion 360 is that it is cloud-based — it solves in the background even when the software is closed.

Once the results are obtained, we can select from a variety of alternatives in the **Explore** box, having multiple ways in which to see the data.



Properties	Outcome 20	Outcome 21	Outcome 22	Outcome 23
Status	Completed	Completed	Completed	Completed
Material	Aluminum 6061	Aluminum 6061	Aluminum 6061	Aluminum 6061
Orientation	-	Z+	X+, Y+, Z-	Unrestricted
Manufacturing method	Unrestricted	Additive	3 axis milling	5 axis milling
Visual similarity	Group 5	Group 3	Group 7	Group 2
Production volume (pcs.)	-	-	-	-
Piece part cost				
Range (USD)	-	-	-	-
Median (USD)	-	-	-	-
Fully burdened cost				
Range (USD)	-	-	-	-
Median (USD)	-	-	-	-
Volume (mm^3)	357,333.68	293,373.49	380,131.02	304,261.22
Mass (kg)	0.965	0.792	1.026	0.822
Max von Mises stress (MPa)	465	78.6	102.6	135.7
Factor of safety limit	2	2	2	2
Min factor of safety	0.59	3.5	2.68	2.03
Max displacement global (mm)	5.76	0.52	1.06	0.95

PROPERTIES VIEW



SCATTER PLOT VIEW

The choice of the best outcome can be done based on various factors – *Minimum Factor of Safety, Least weight, Maximum von Moises Stress, Maximum displacement, Material, Manufacturing method, etc.*

We compare alternative results based on the following criteria – Minimum Factor of Safety achieved, Maximum von Moises Stress generated and Maximum Global Displacement.

From the Scatter Plot view, it is evident that one outcome has the highest minimum FOS of around 3.5, least weight of around 0.79 kg and a maximum global displacement of 0.52 mm, which is relatively the least among all the other outcomes. Hence, this is the one outcome we choose.

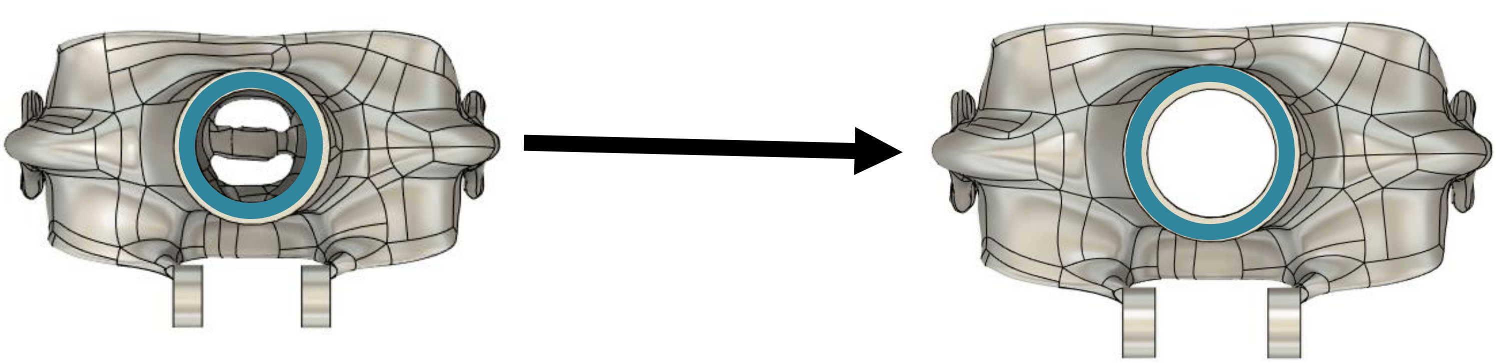
Once decided with the outcome, we generate design from the outcome and move ahead with the further processes. By further processes we refer to **post-processing work** – which is not a part of the Generative Design workspace, but considered a part of the overall Generative Design process.

CHOSEN OUTCOME

Post-Processing

As have said, post-processing is a part of the Generative Design process. Here, we return to our previously created design to focus on key areas that require attention and rework. However, it should be noted that this rework is not always required — many times, we receive a really nice one and can proceed with the next phases (manufacturing or analysis).

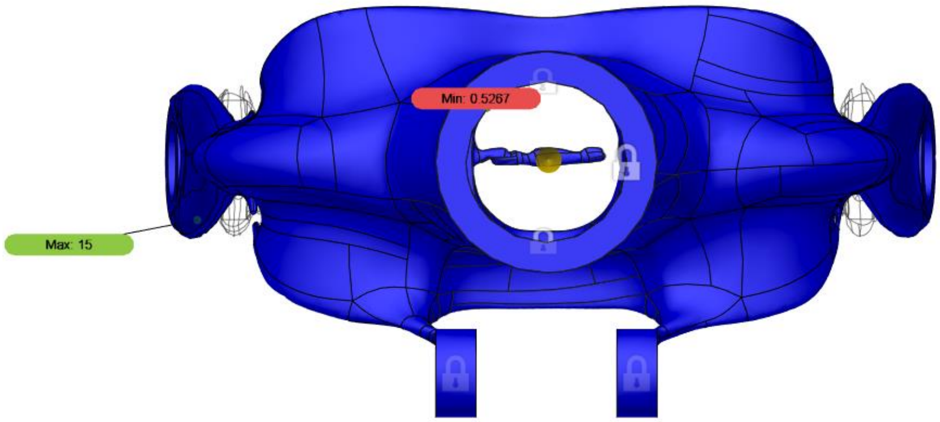
In our design, we have done the post-processing work. We focused on removing some unnecessary material; the area where the material was removed is shown in the next picture.



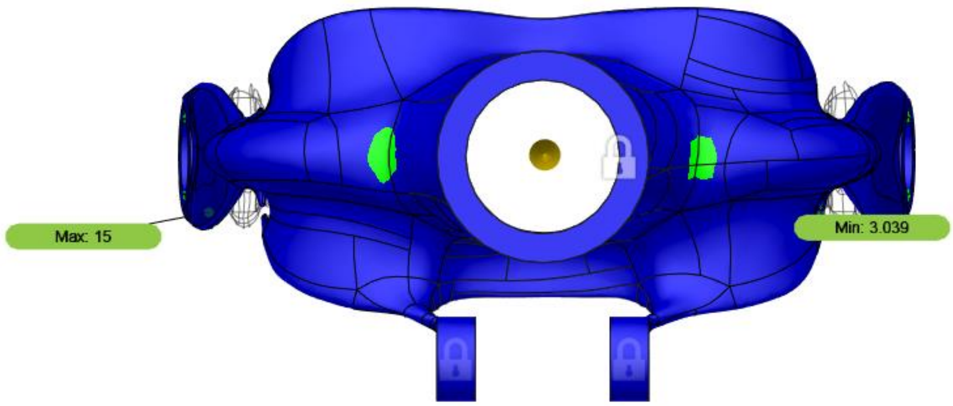
We chose to use post-processing for two key reasons:

- Unnecessary material adds up to the weight of the part, and costs during manufacturing. To solve this cost issue, this material is removed.

  These unnecessary materials have a significant impact on the outcomes of the part analysis. Here are two images from a study of the part before and after the material was removed.



BEFORE REMOVING MATERIAL



AFTER REMOVING MATERIAL

  The above results are for the Factor of Safety constraint. This is a good example of how crucial post-processing can be.

Simulation

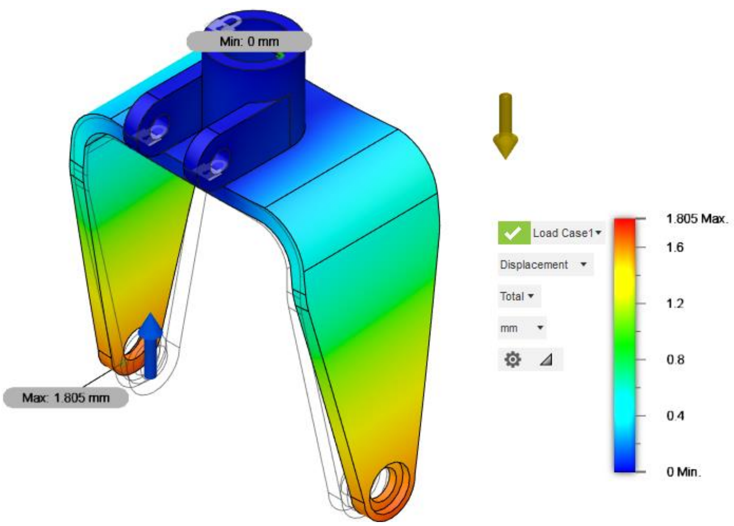
By performing analysis, we will see how the part performs under load.

Standard Engineering Design

The initial design was subjected to **Linear Static Stress Analysis**, and the results for the parameters evaluated are shown in the table:

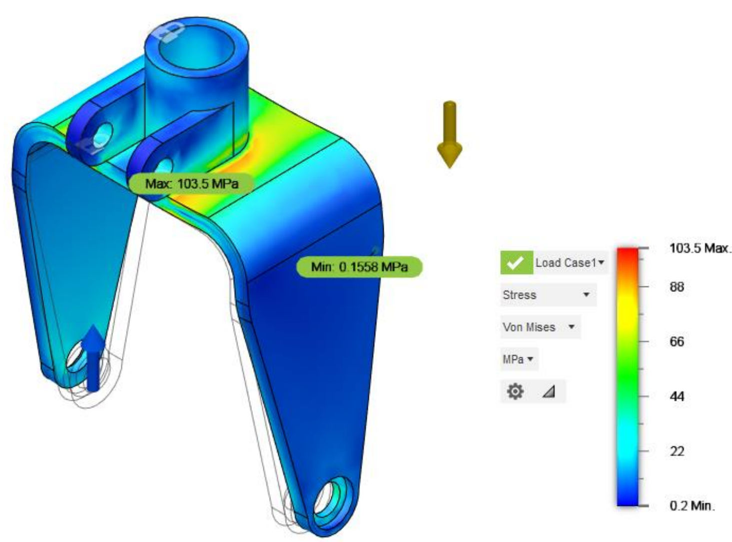
Factor	Value
Maximum Global Displacement	1.805 mm
Maximum von Moises Stress	103.5 MPa

The following two images are visual representation of the analysis results:



DISPLACEMENT PLOT

STRESS PLOT

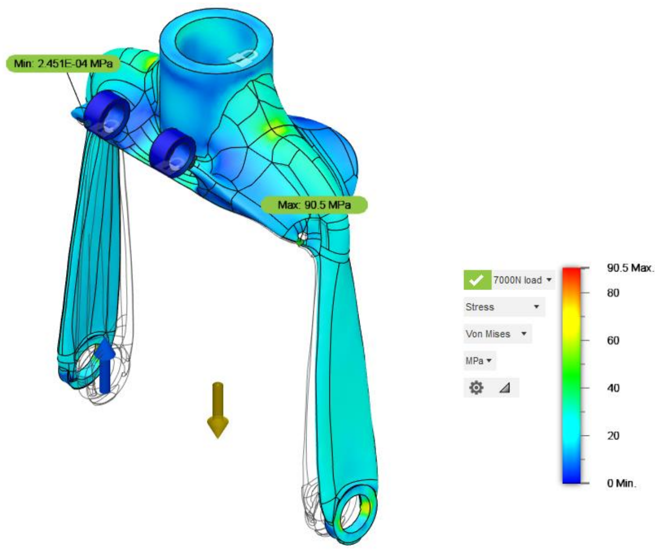


Generative Design

The final design from the Generative Design process was subjected to an analysis study similar to what was performed in the earlier case (Linear Static Stress Analysis). The results for the parameters evaluated are shown in the table:

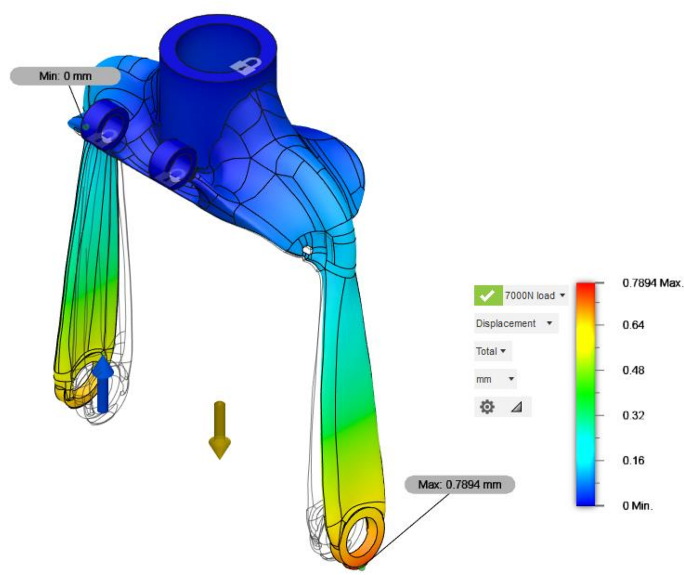
Factor	Value
Maximum Global Displacement	0.7894 mm
Maximum von Moises Stress	90.5 MPa

The following two images are visual representation of the analysis results:



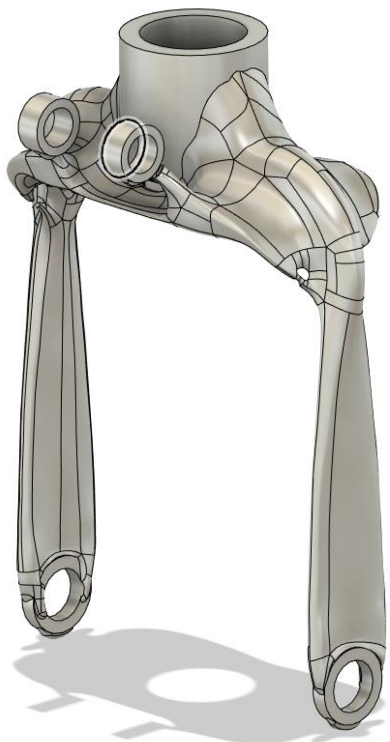
STRESS PLOT

DISPLACEMENT PLOT



Generative Design – Final Part review

Let's go over the final design for a few parameters, just like we did with the standard engineering design, and compare them here.

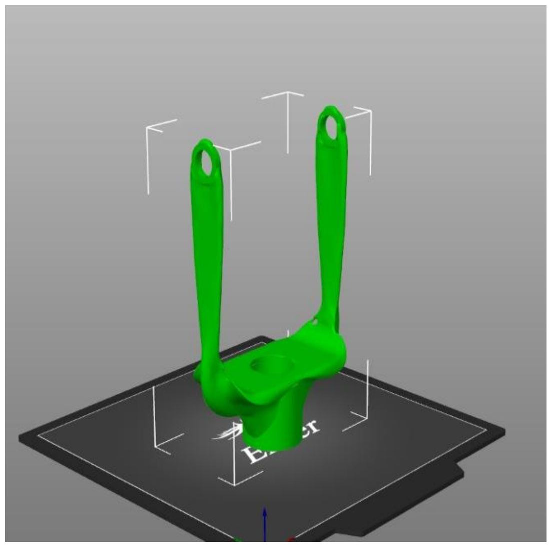


PART FROM GENERATIVE DESIGN

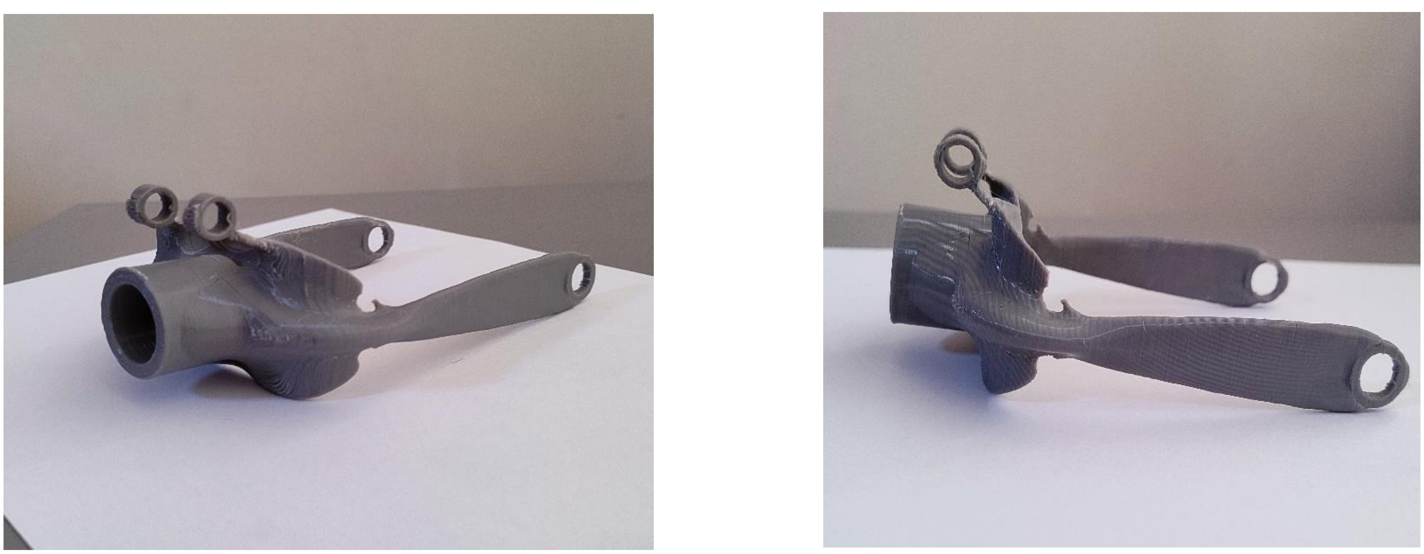
Parameters	Standard Engineering Design	Generative Design
Weight	963.27 grams	741.45 grams (23% lighter)
Displacement	1.805 mm	0.7894 mm (56% reduction)
Manufacturing Method	Die Casting	Additive Manufacturing
Material	Aluminium 6061	Same

3D Printing

Once the design was complete, we had a 3D prototype printed to see how it would look in touch. This is a prototype; hence the **material used was PLA** (Polylactic Acid) – a thermoplastic.



PROCESSING OF DESIGN FOR 3D PRINTING IN SLICER SOFTWARE



FINAL 3D PRINTED PROTOTYPE