Introduction

This comprehensive tutorial will guide you through the steps of re-creating Jamie-Hubo shells. In this tutorial, we will cover a wide range of topics, including opening 3D model files using Autodesk Inventor, exporting them as step files (.stl), and importing these files into a slicer software. We will also explore the process of generating G-code, which is essential for preparing the models for 3D printing. Additionally, we'll provide you with useful tips and insights for successful 3D printing. Furthermore, if you're interested in recreating the models, we'll showcase how to accomplish that using SolidWorks.

Hardware

- 3D printer (Prusa, Creativity, Lulzbot, X-max…)
- 3D printing filament (PLA, PETG, ABS…)

Software

- Autodesk Inventor
- Slicer software (PrusaSlicer or other depending on your 3D printer)
- SolidWorks

Resources

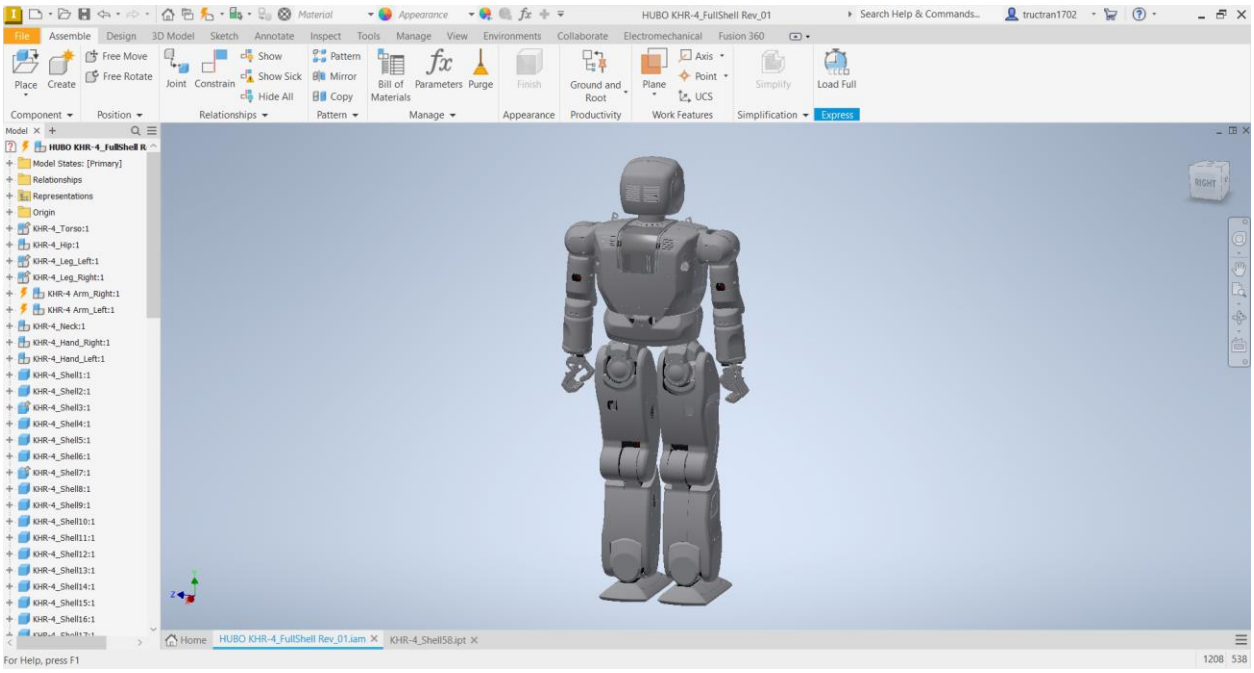
- Autodesk Inventor for student: https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual
- Autodesk Inventor base files: https://www.dropbox.com/s/bwtxddvrvtx8anl/HUBO.zip?dl=0

Procedures

PART I – 3D printing

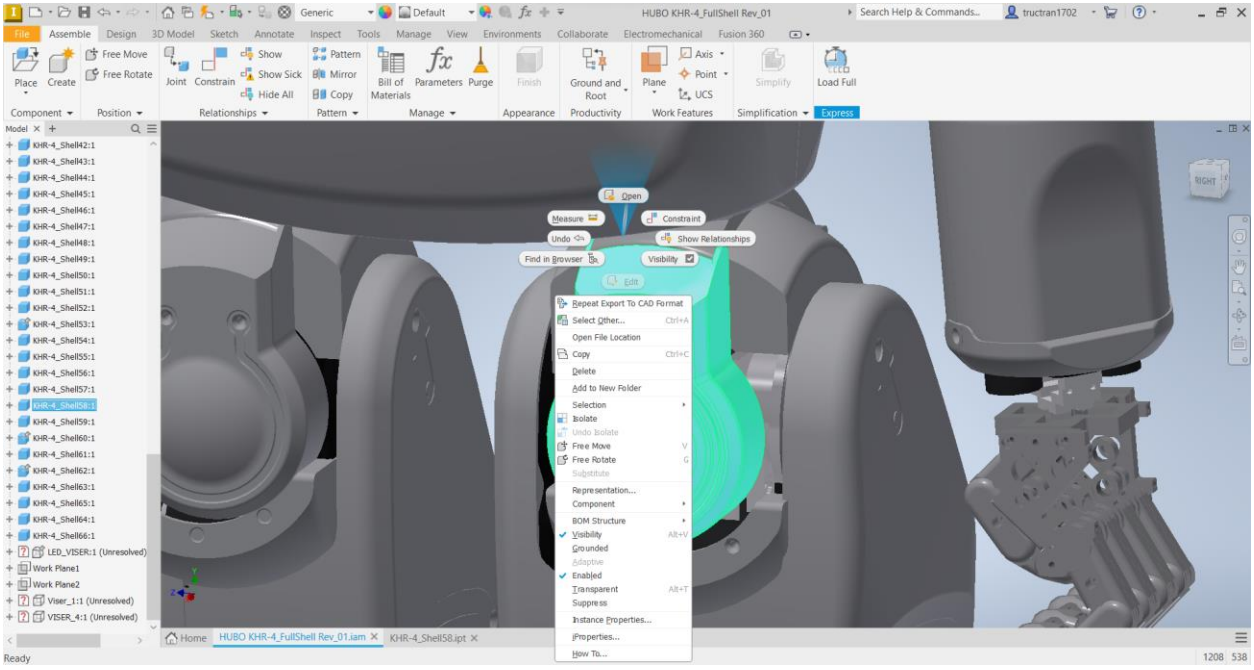
Step 1

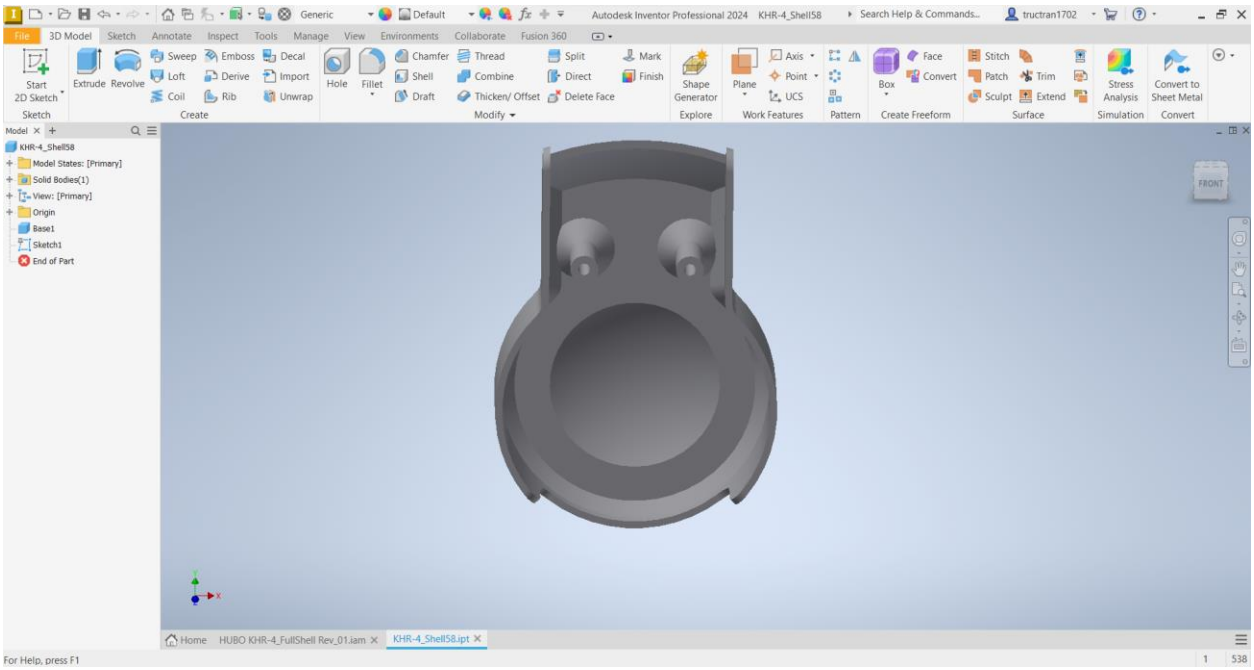
Open the assembly in Autodesk Inventor (..\HUBO\HUBO\Inventor\Finish\Shell Assembly\HUBO KHR-4_FullShell Rev_01.iam). There are 66 parts of the entire shell assembly.



Step 2

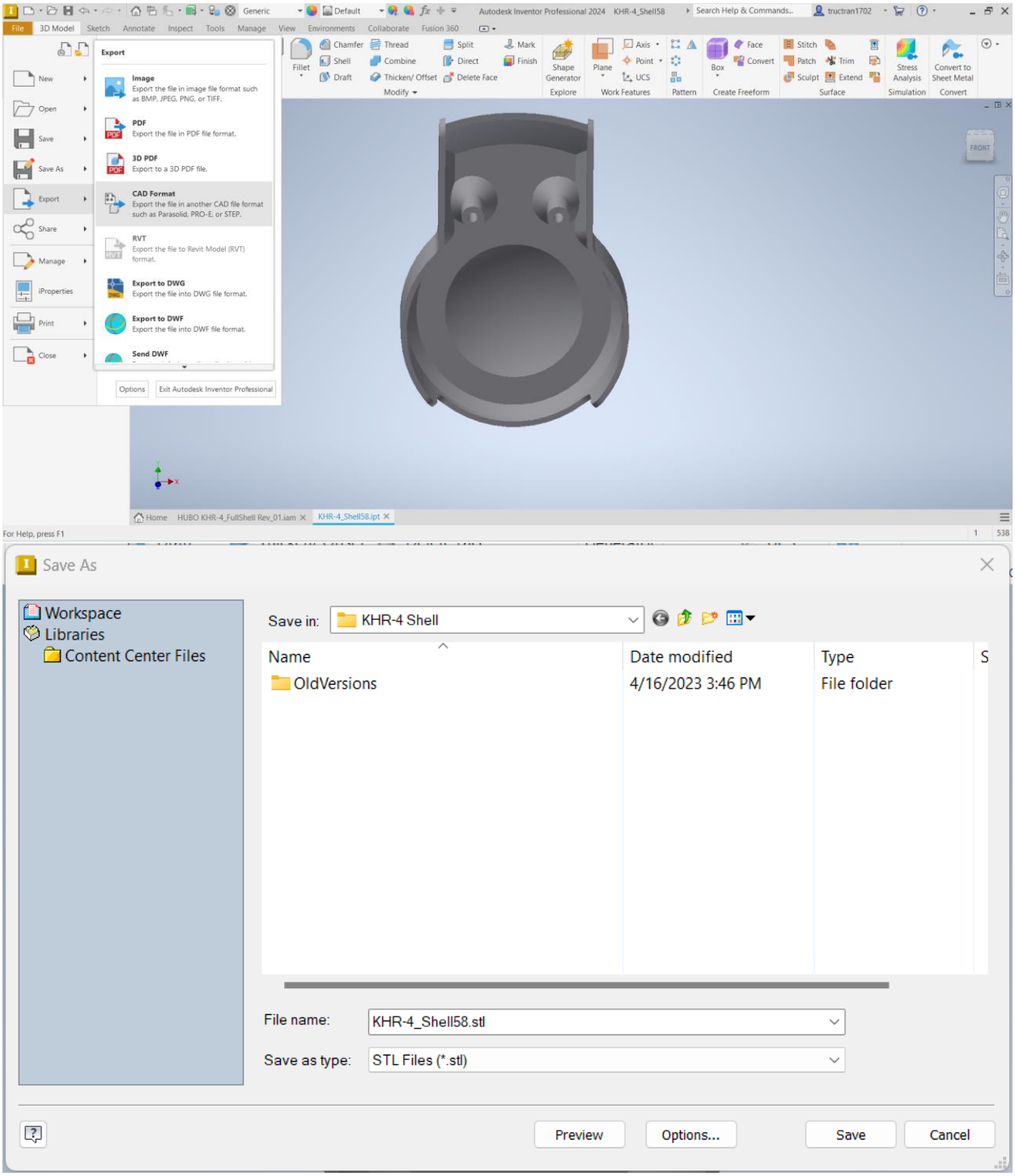
Right click and open the part you would like to 3D print. In this demonstration, we will be using part 58.





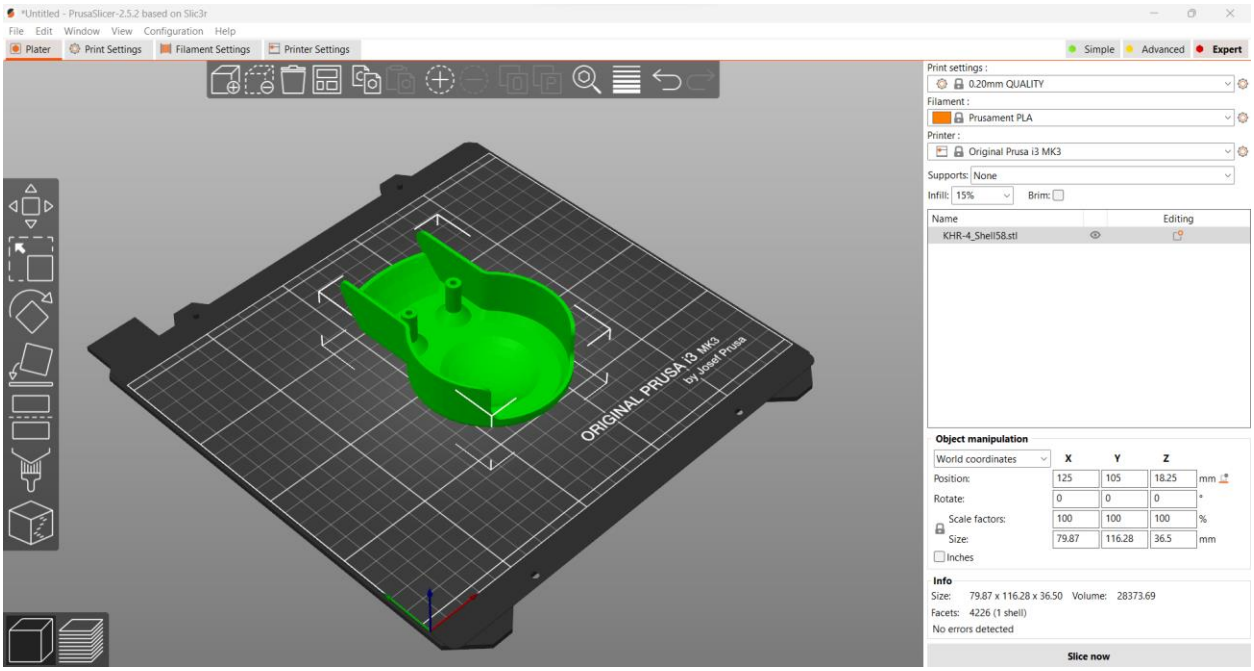
Step 3

On the top right corner, click on File > Export > CAD Format. Save the files as .stl files.



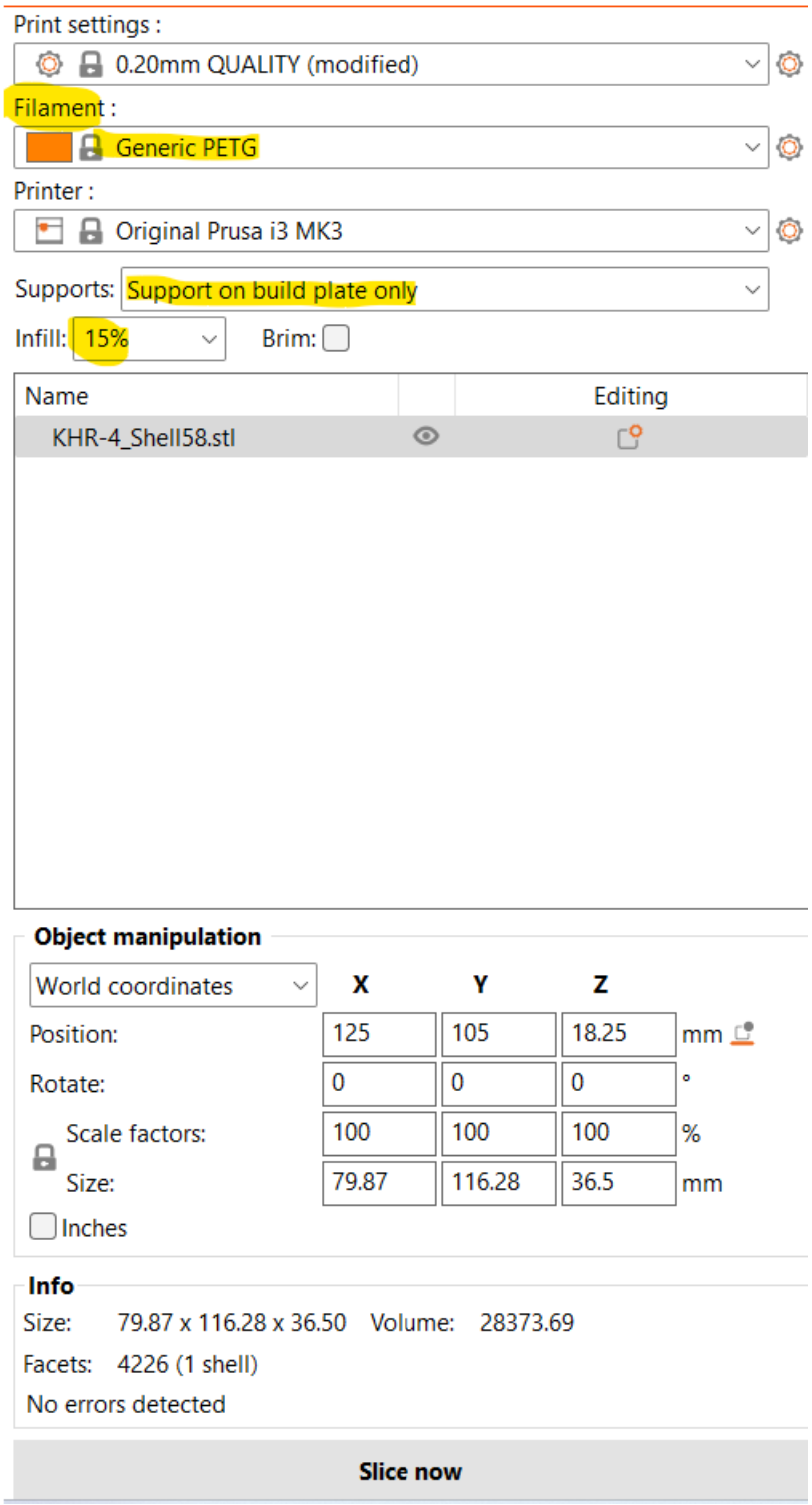
Step 4

Open PrusaSlicer, and draf the .stl files into the slicer software.



Step 5

There are three key slicing settings that are essential for 3D printing with any slicer software. Firstly, we have the filament setting. Common materials include PLA, PETG, ABS, and others. Each material possesses distinct properties such as toughness, flexibility, and melting points. For this particular demonstration, we will be using PETG as our material of choice. The next important setting is printing supports. Ideally, we would prefer to avoid using supports altogether, as they require additional material and often compromise the smoothness of our printed parts. However, in many cases, supports are necessary because 3D printing cannot create structures in thin air. In this demonstration, we will configure the supports to be "Support on build plate only. Lastly, the infill setting determines the internal structure of the printed part and ranges from 0% to 100%. A higher infill percentage results in stronger parts, but it also consumes more material and increases printing time. A commonly used infill percentage for most 3D printed parts is 15%.



Step 6

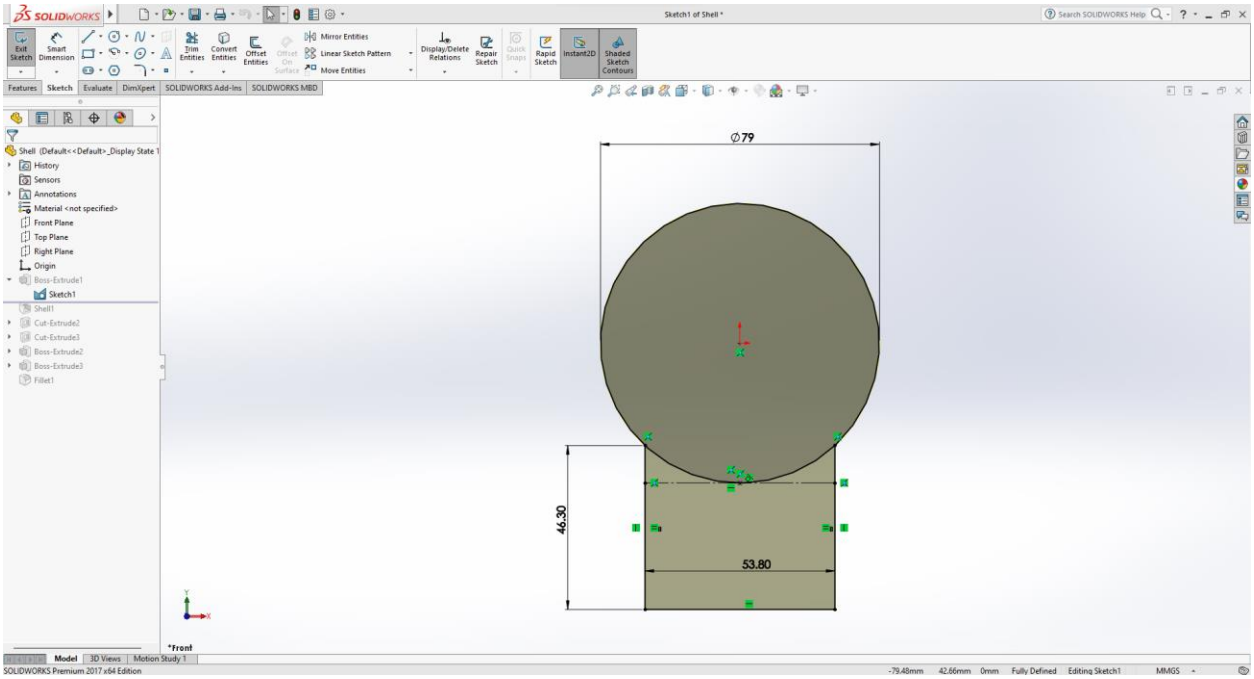
Checking the result of the print and remove the support materials.

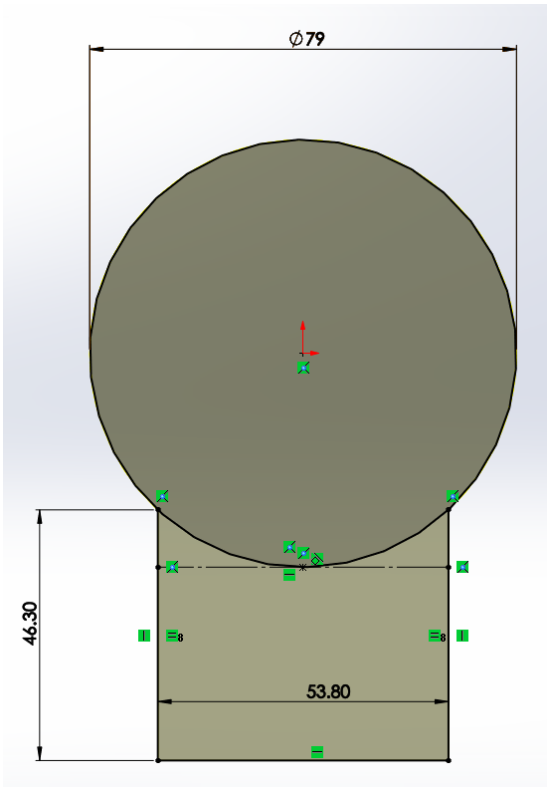
ORIGINAL PRUSA i3 MK3
by Josef Prusa

PART II – SolidWorks recreation

Step 1

In this step, you will use SolidWorks to create a sketch of the base shape of the part. You can use various sketch tools such as lines, arcs, circles, or rectangles to define the geometry. Then, you will use the dimensioning tools to specify the exact dimensions of the part based on your requirements or design specifications.




⌀79
46.30
53.80

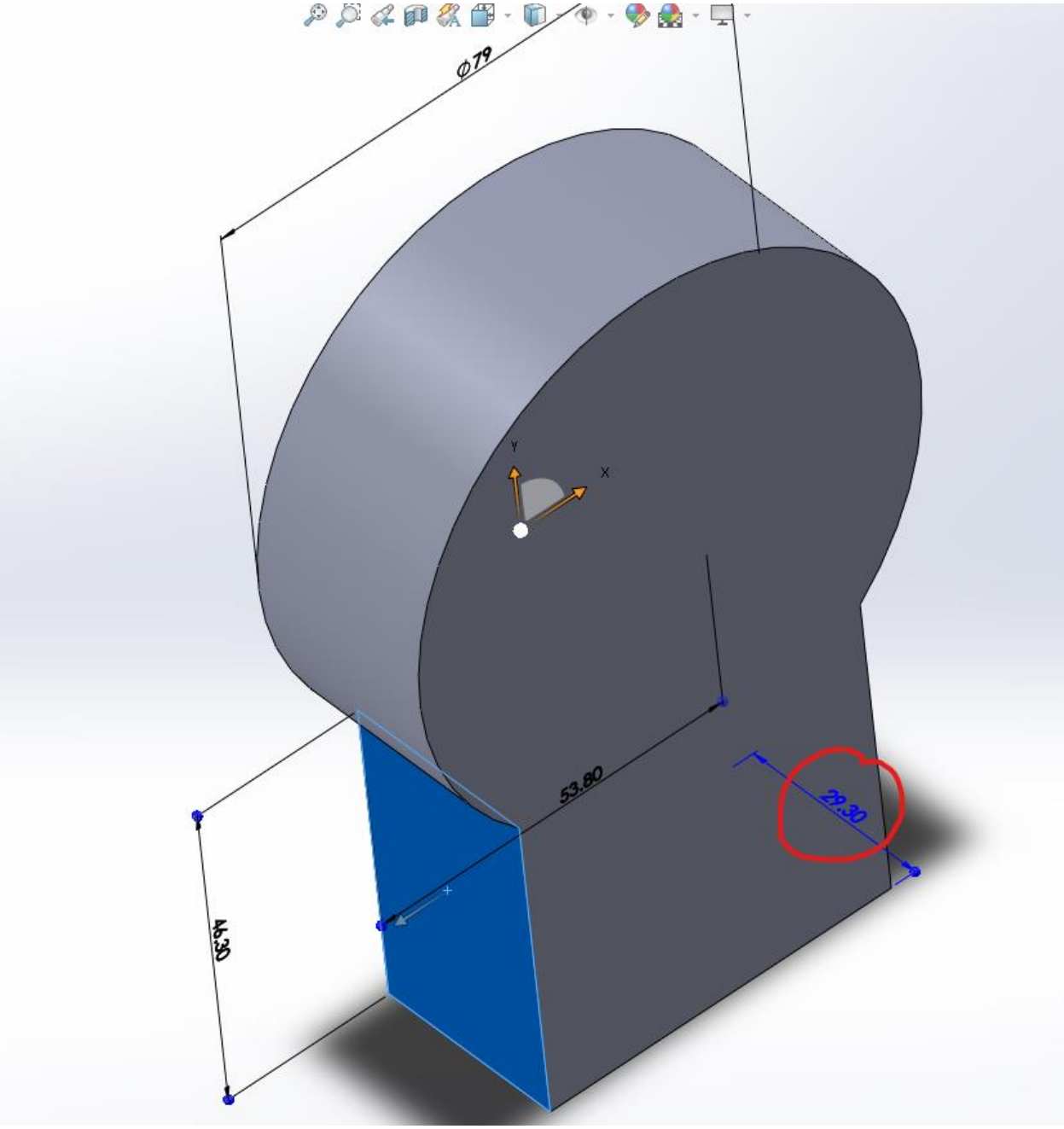
mm/inch
OFF
ON
ZERO

mm/inch
53.8
OFF
ON
ZERO

mm/inch
46.3
OFF
ON
ZERO

Step 2

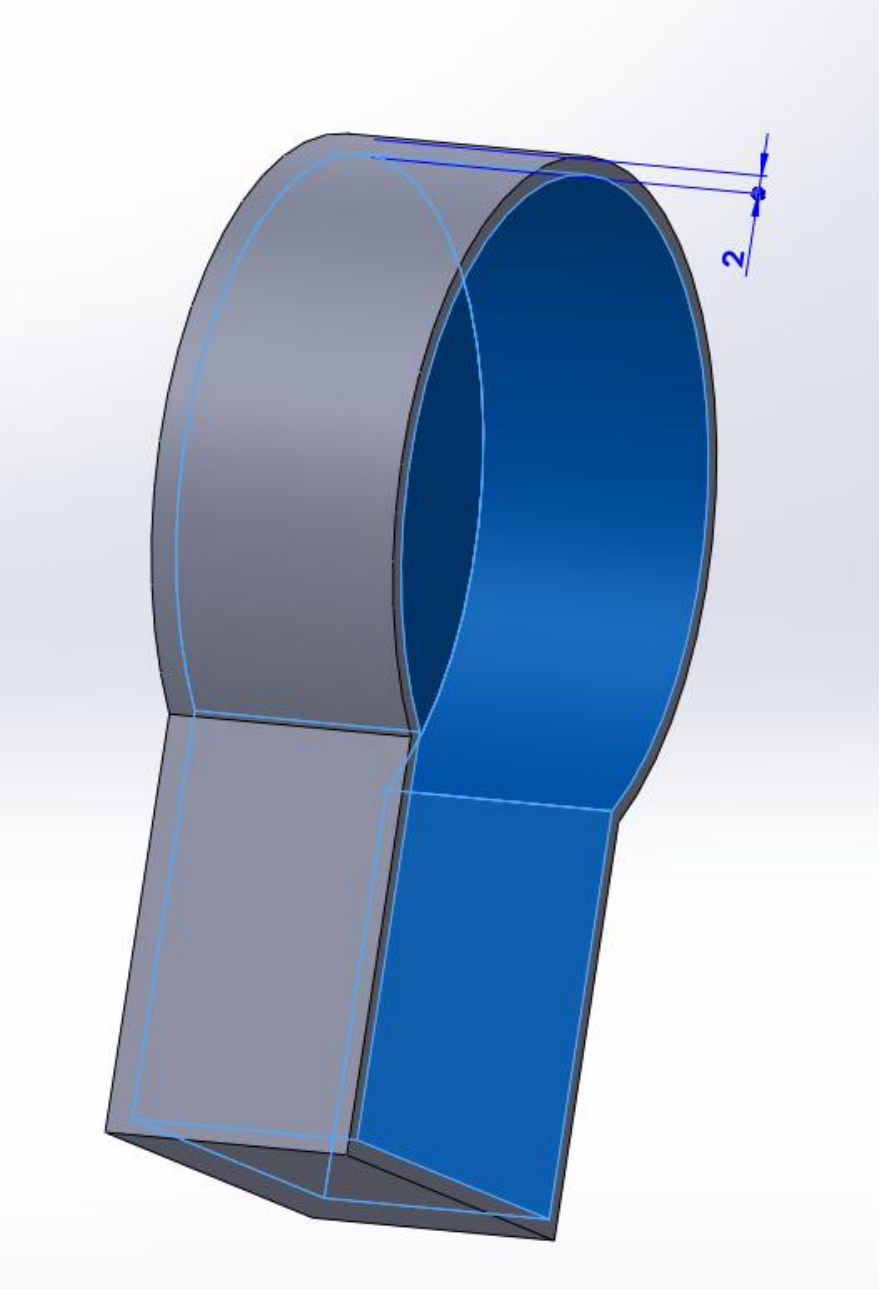
After creating the sketch of the base, you will use the extrude feature in SolidWorks to give it depth or thickness. The extrude feature allows you to extend the 2D sketch into a 3D object by specifying the desired thickness value. You can choose to extrude symmetrically or in a specific direction based on your design needs.



mm/inch
293
OFF
ON
ZERO

Step 3

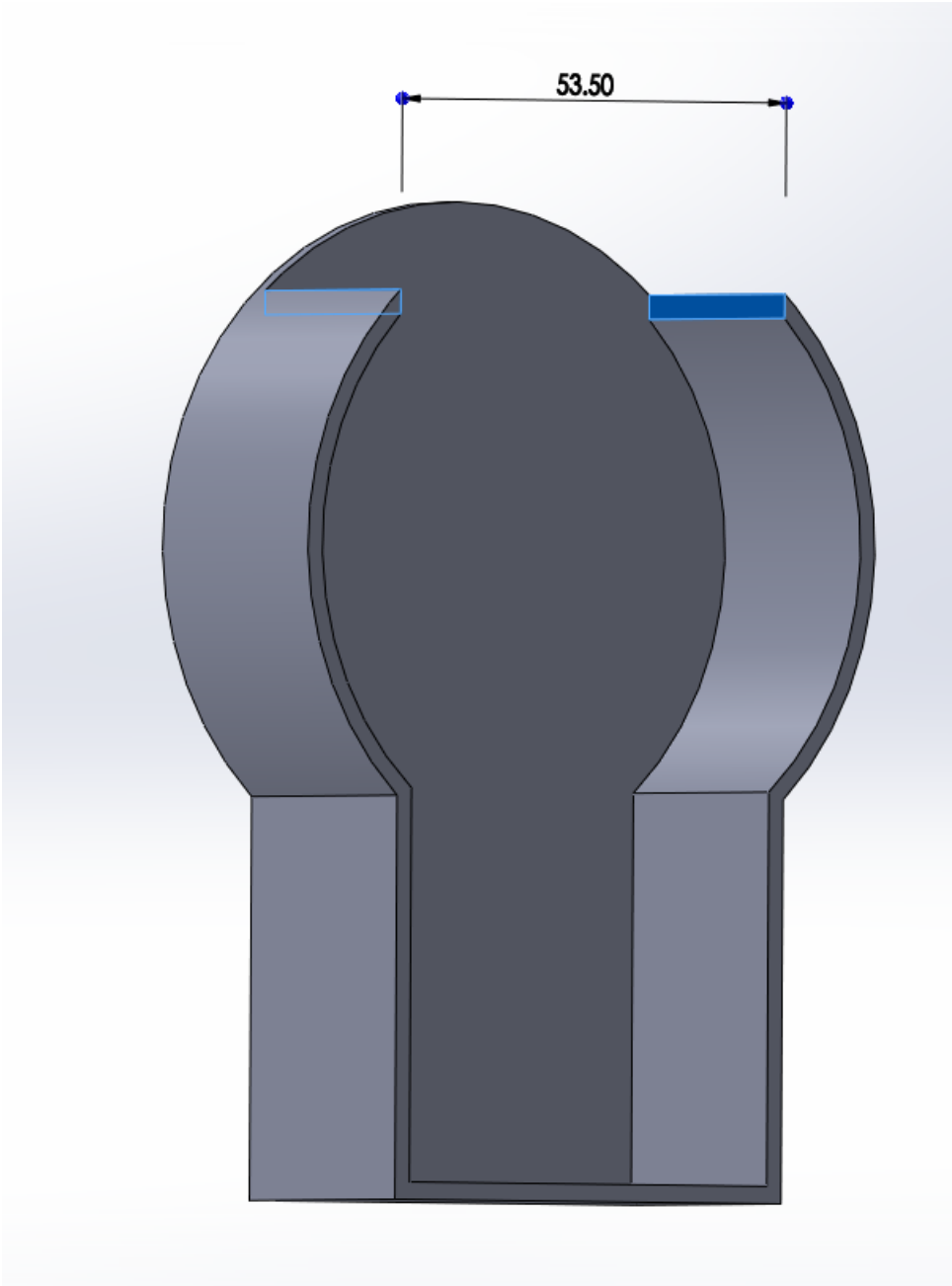
The shell feature in SolidWorks allows you to hollow out the solid part by specifying a wall thickness. By selecting the faces or surfaces of the part, you can create an empty space within the solid body. This step is useful when you want to create a hollowed-out area within the part, such as a cavity or chamber.



mm/inch
20
OFF
ON
ZERO
70
80

Step 4

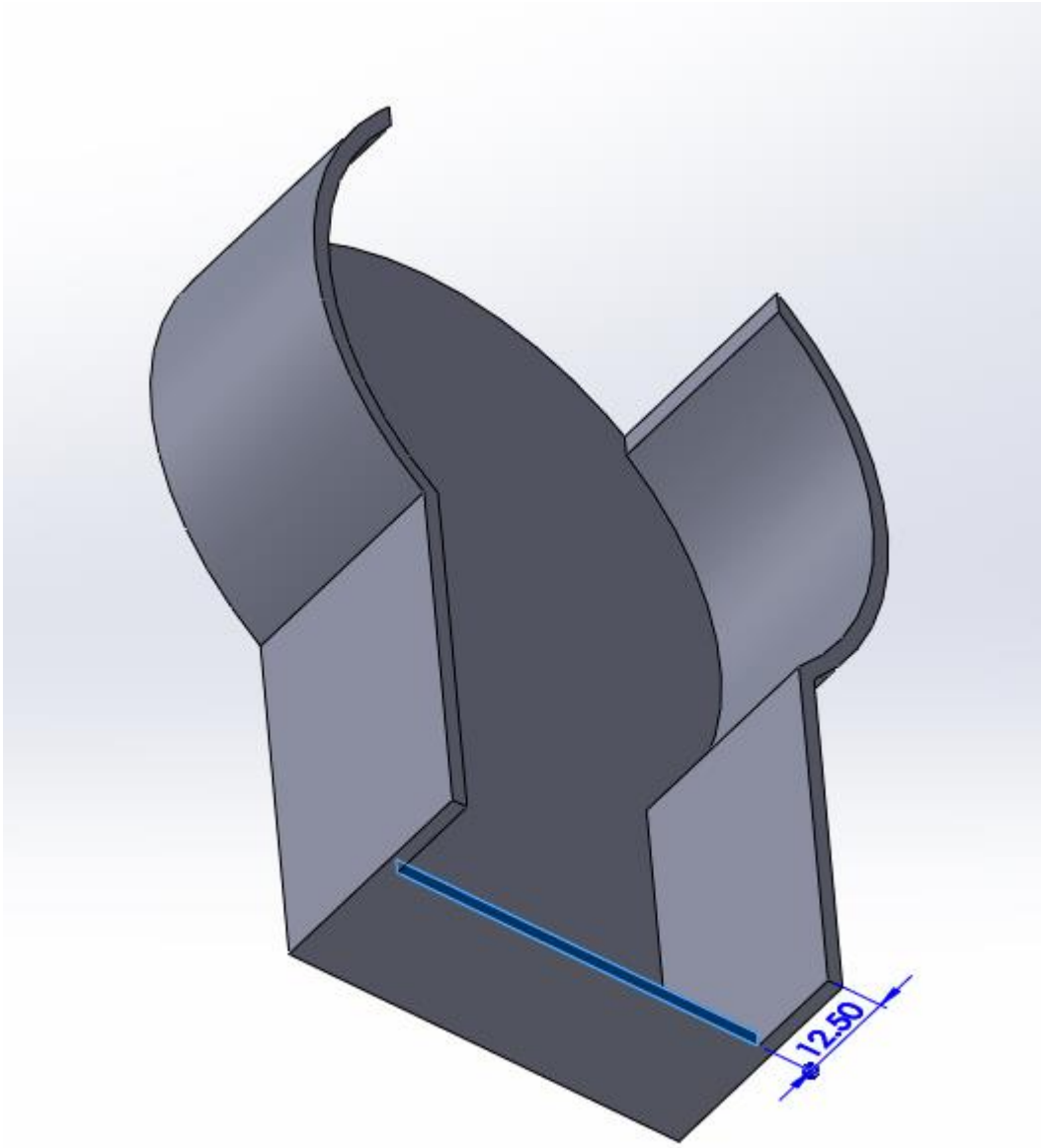
The cut feature in SolidWorks enables you to remove material from the solid body. By creating a sketch on a selected face or surface, you can define the shape of the cutout. The cutout can be through all or a specific portion of the part, depending on your design requirements.

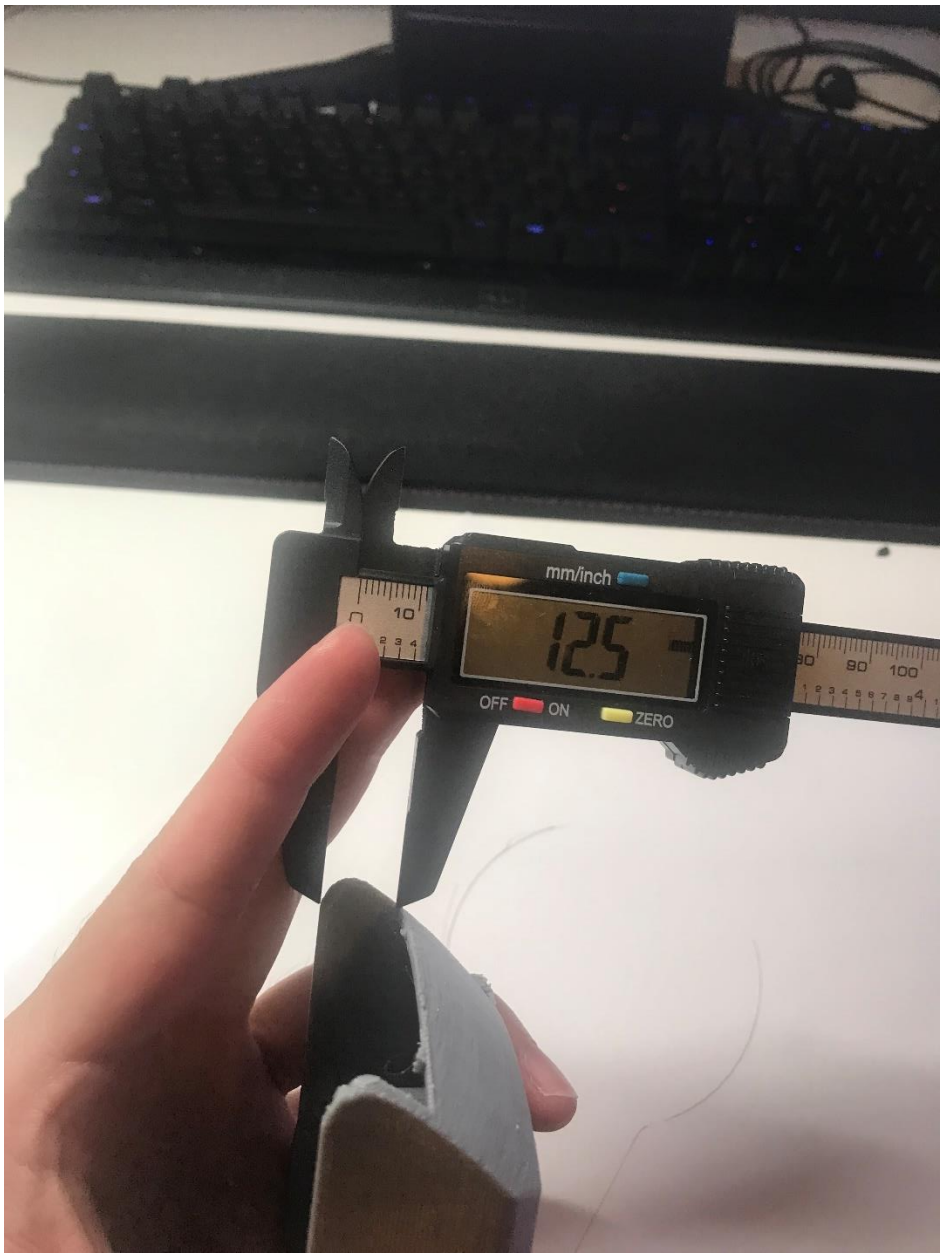


mm/inch
535
OFF
ON
ZERO

Step 5

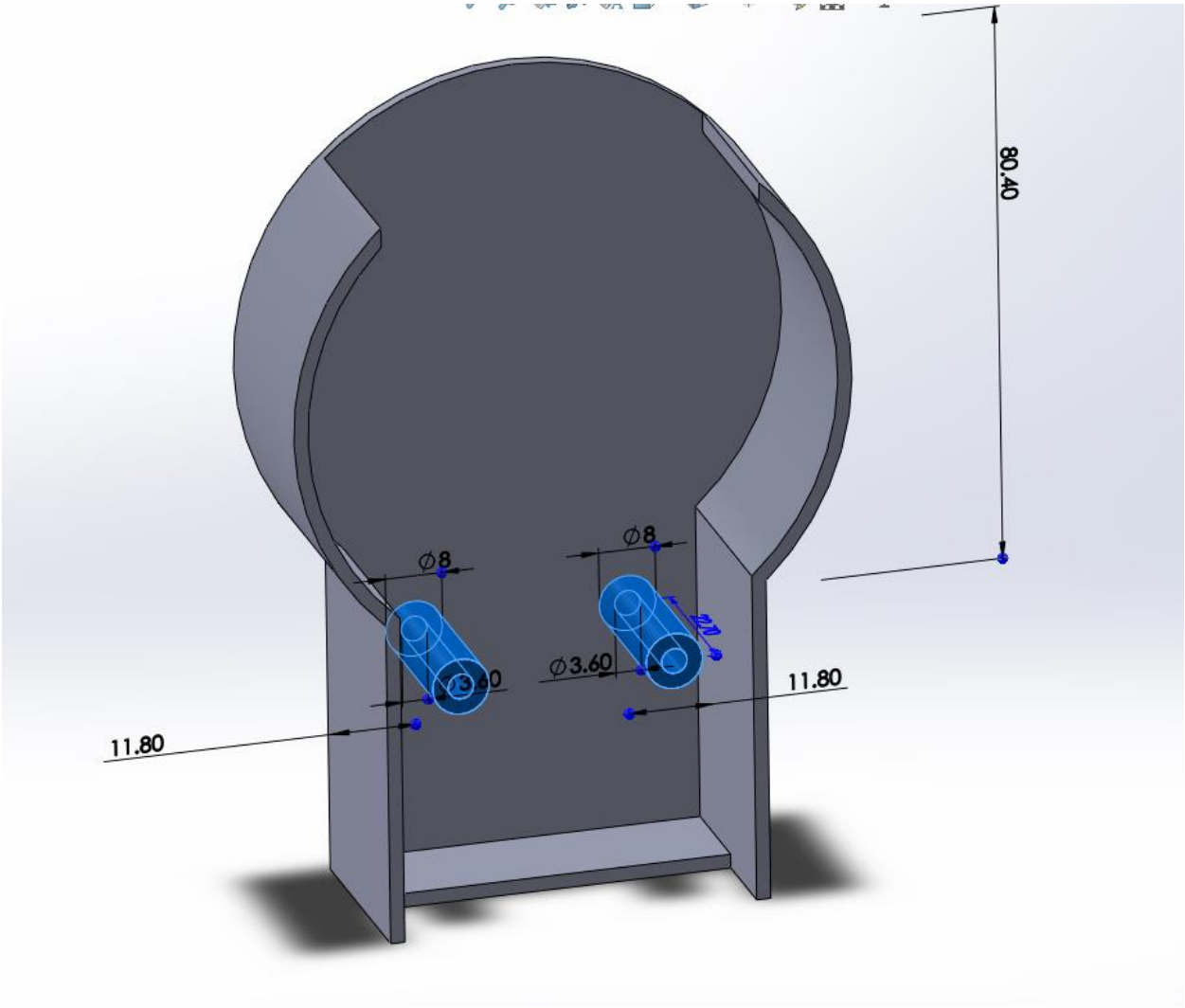
In this step, you will create a cut feature specifically at the bottom of the part. By using the cut feature, you can remove material from the desired area, leaving an open or modified section.




mm/inch
12.5
OFF
ON
ZERO

Step 6

This step involves creating features that will hold screws or fasteners in place. Using sketching tools, you can define the geometry and location of the screw holders. Make sure to accurately measure the required distance between the screw holes to ensure proper alignment and functionality.



mm/inch
36
mm
OFF
ON
ZERO

mm/inch
8.0
mm
OFF
ON
ZERO

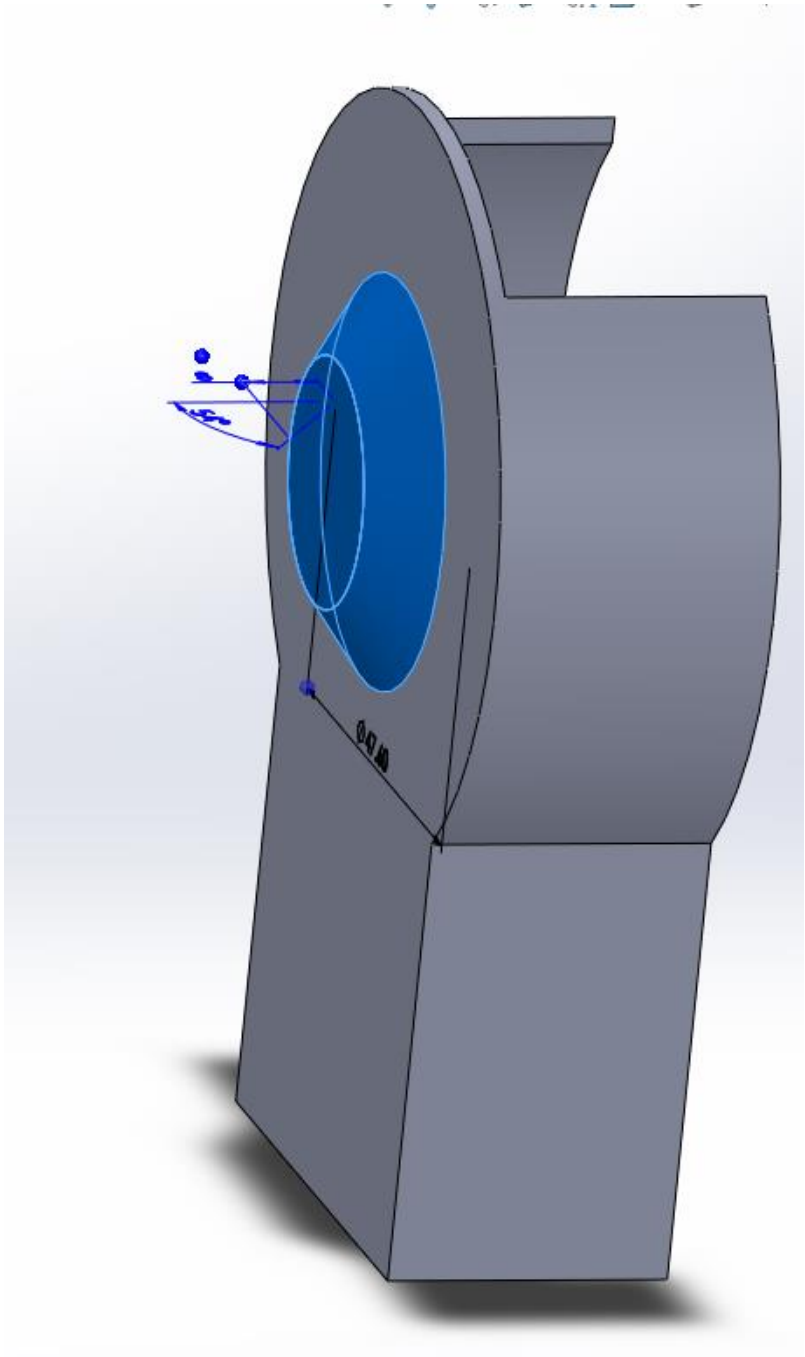
mm/inch
11.8
mm
OFF
ON
ZERO

mm/inch
80.4
OFF
ON
ZERO

OFF
ON
ZERO
mm/inch
ELECTRONIC
DIGITAL CALIPER

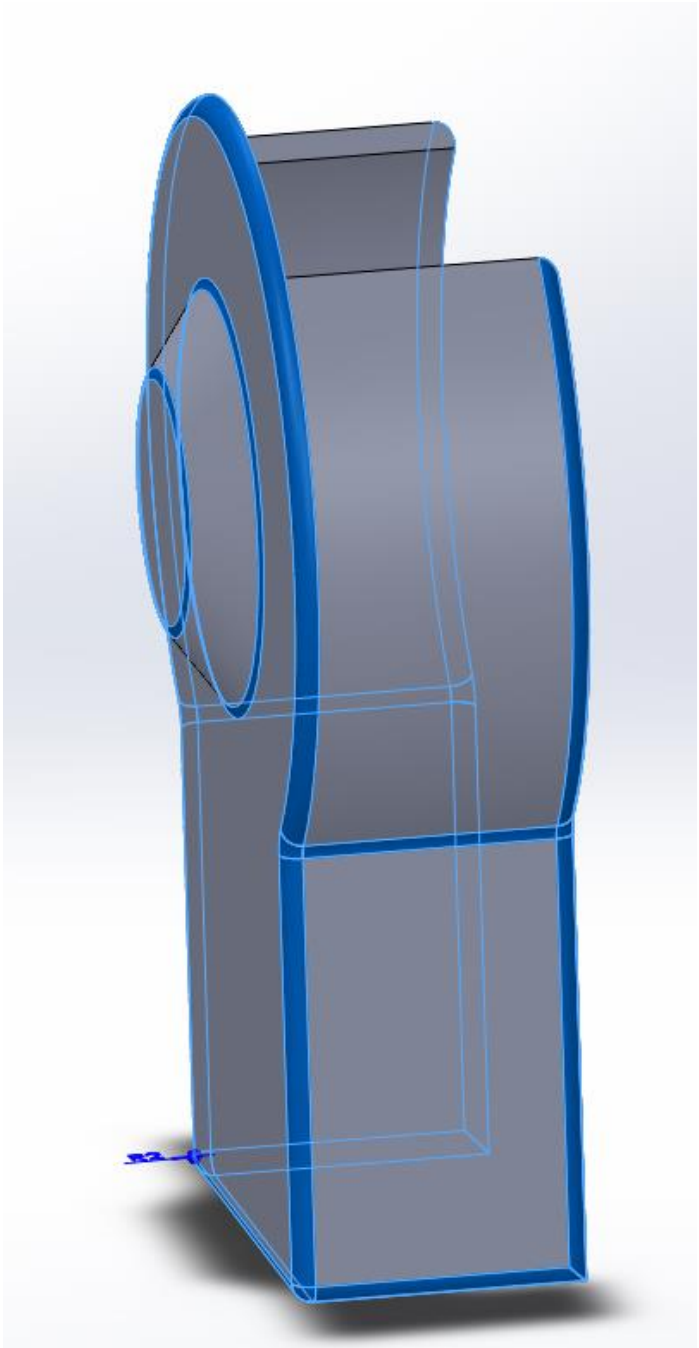
Step 7

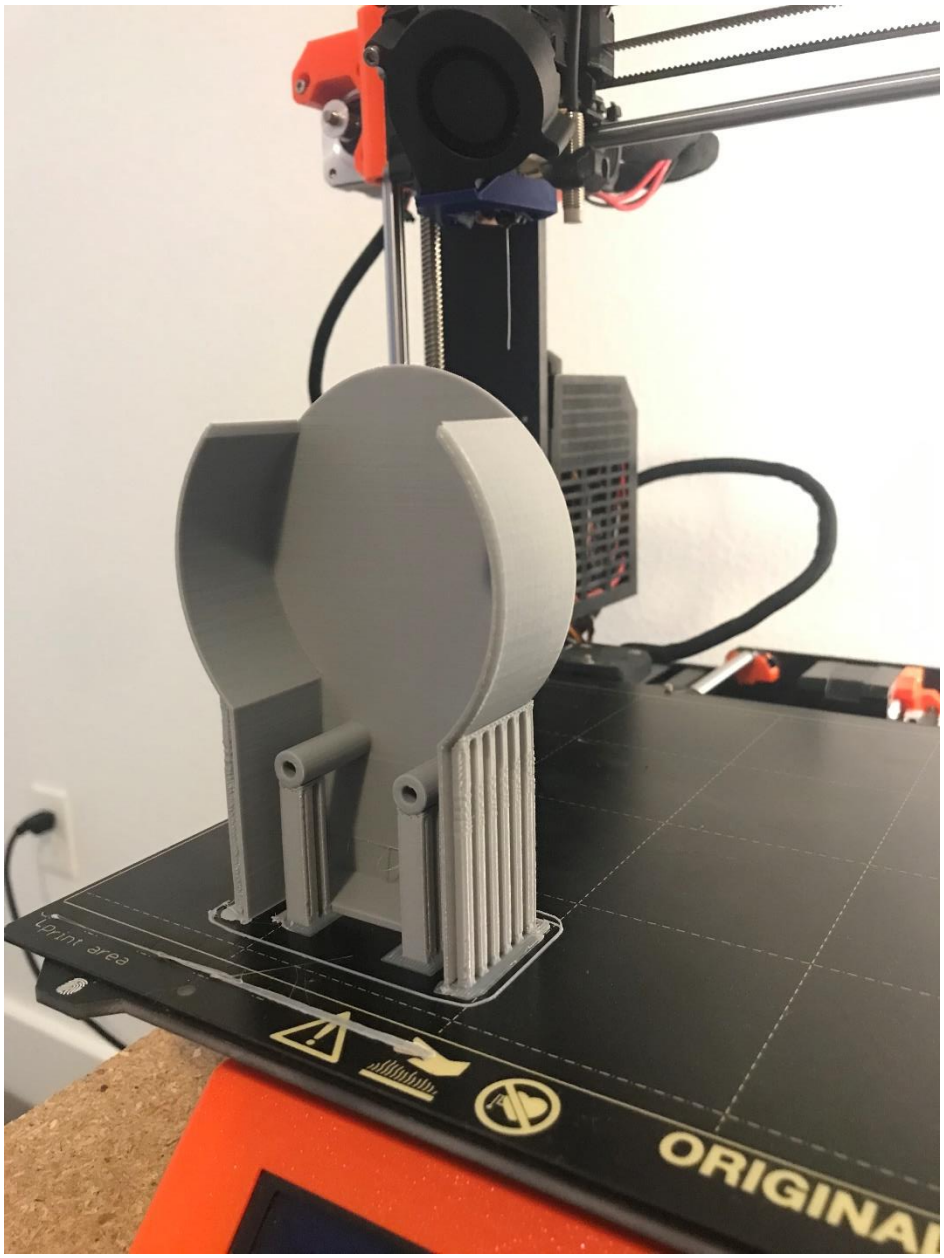
After completing the essential features of the part, you can focus on adding aesthetic details to enhance the visual appeal. This can include adding additional features, such as fillets, chamfers, or decorative elements, to make the part more visually appealing or to meet specific design requirements.



Step 8

In this final step, you will add fillets to the edges of the part. Fillets are rounded transitions between two surfaces, and they help to avoid sharp edges, which can be uncomfortable or potentially dangerous. By specifying a fillet radius of 1mm, you will create rounded edges on the part.



Step 9

Once the model is finalized, go to the "File" menu and choose "Save As." Select the file format as .stl (Standard Tessellation Language). Choose a location to save the file and click "Save." Use the exported .stl file with appropriate 3D printing software to prepare the model for