ENGG 100 Design and Measurement Lab

Introduction to Computer-Aided Design (CAD) and 3D printing

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PHYSLAB

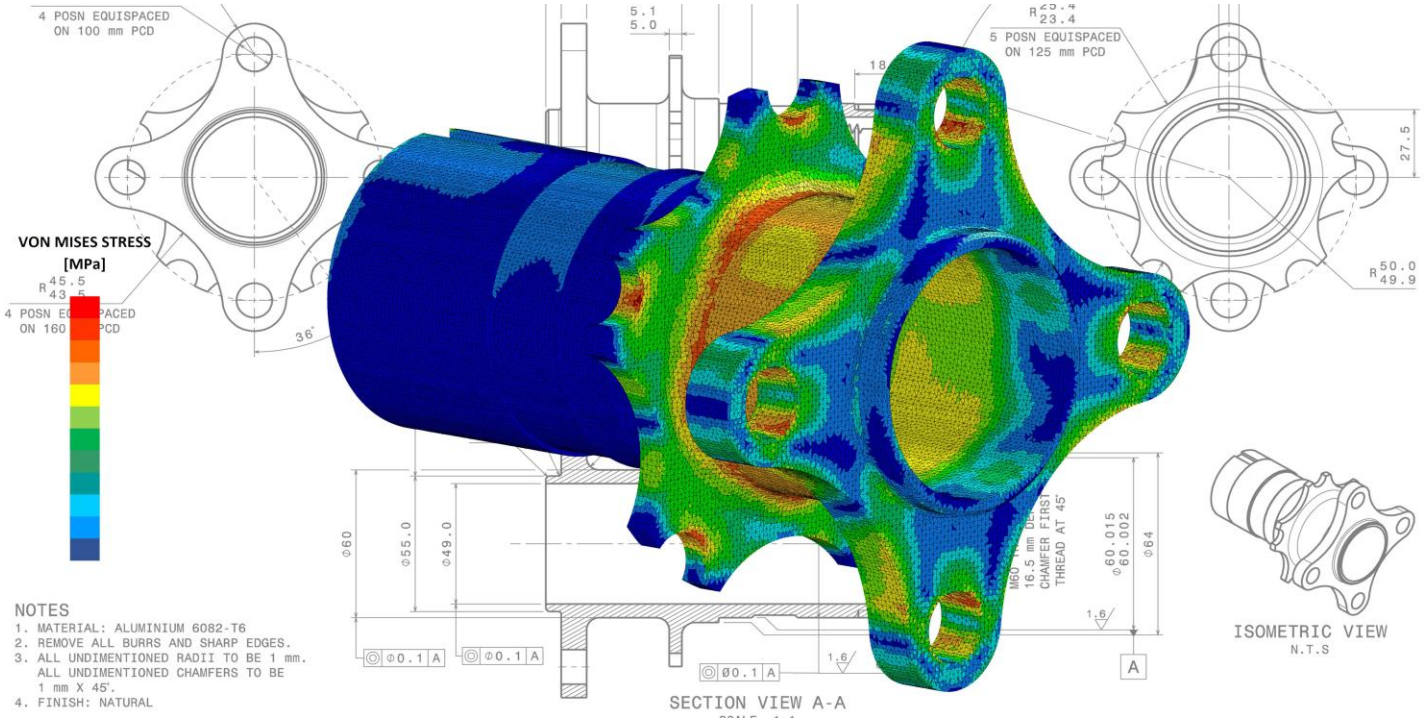
Computed Aided Design

Computer Aided Design (CAD) is defined as the way of creation, modification, and optimization of a design by employing a computer system.

A set of methods and tools to assist product designers in,

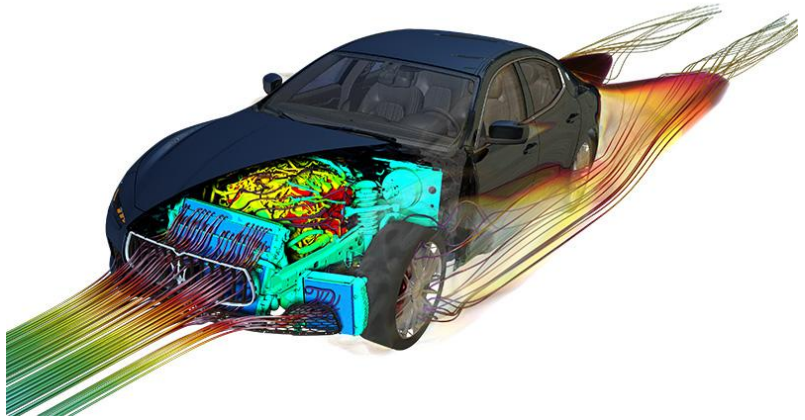
- Creating a geometrical representation of the artifacts,
- Dimensioning, tolerancing,
- Configuration management (changes),
- Feeding subsequent design steps,
  - Analysis(CAA),
  - Manufacturing (CAM),

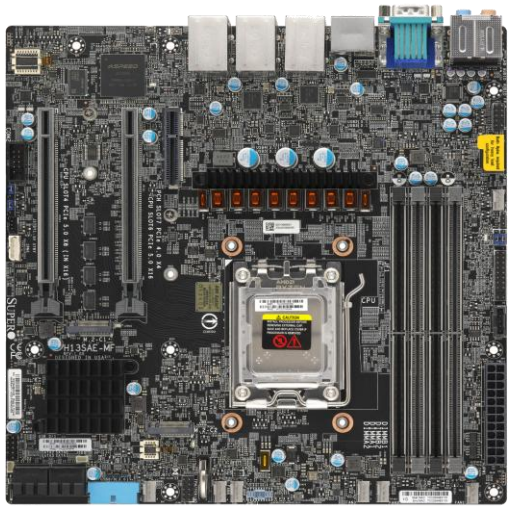
by means of a computer system.

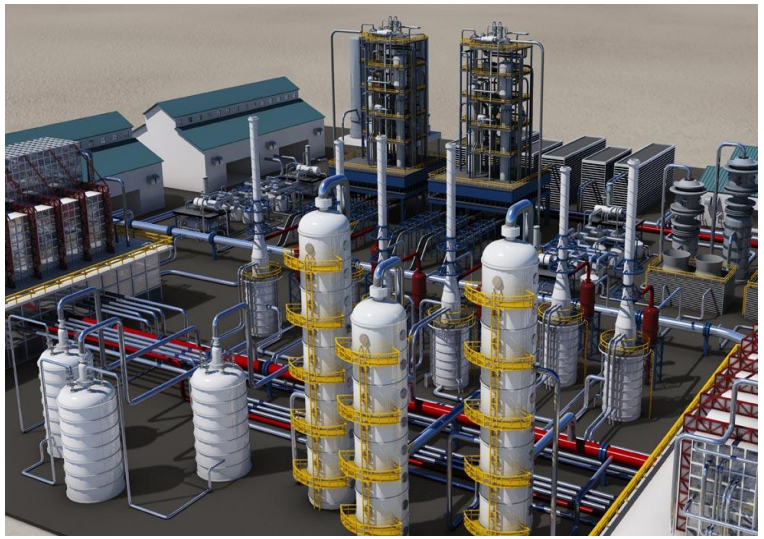


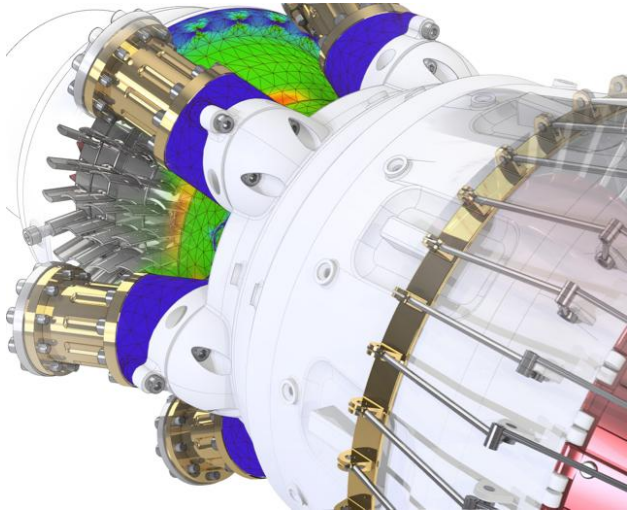
*The figure is taken from an open source repository.

Applications of CAD



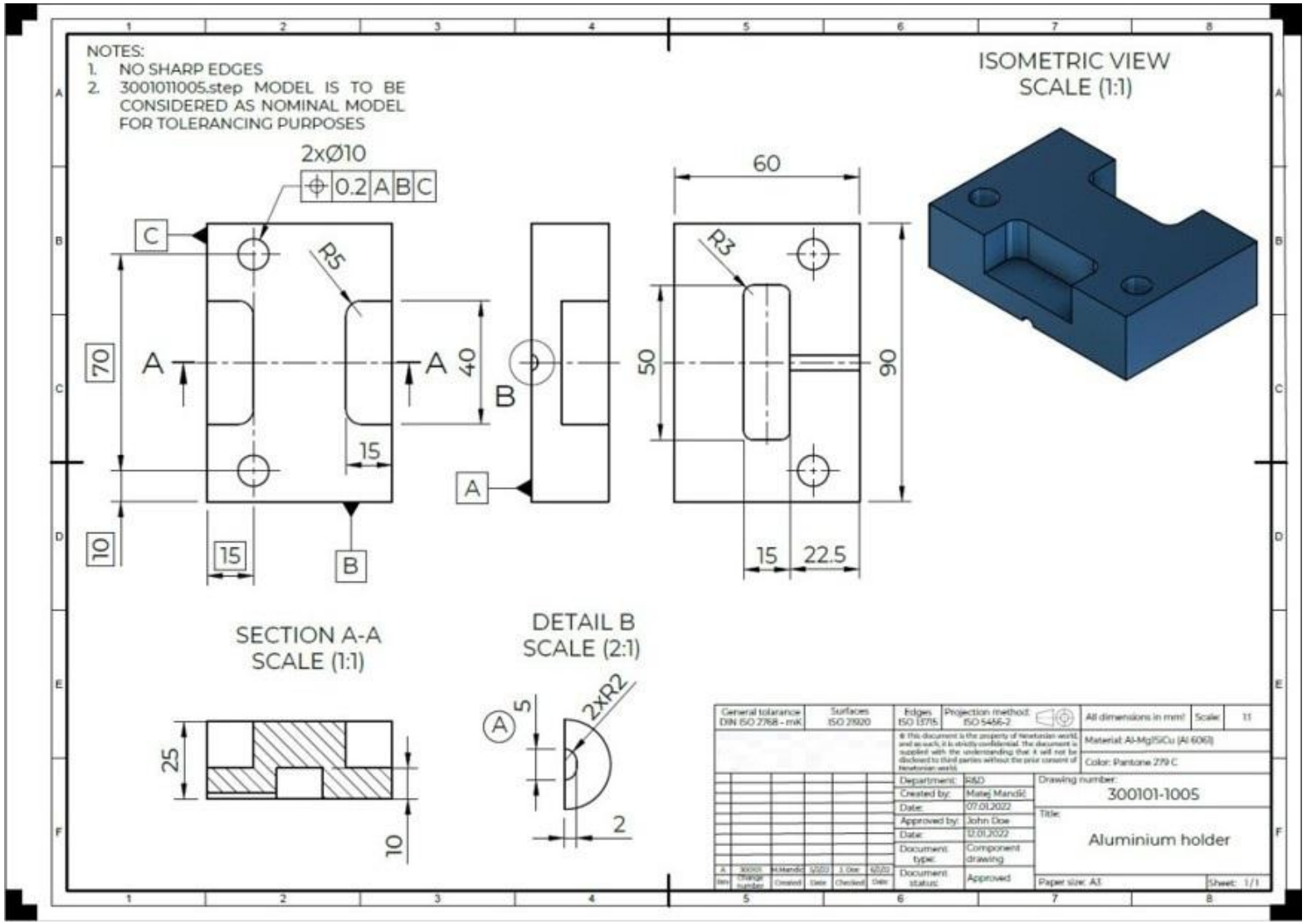






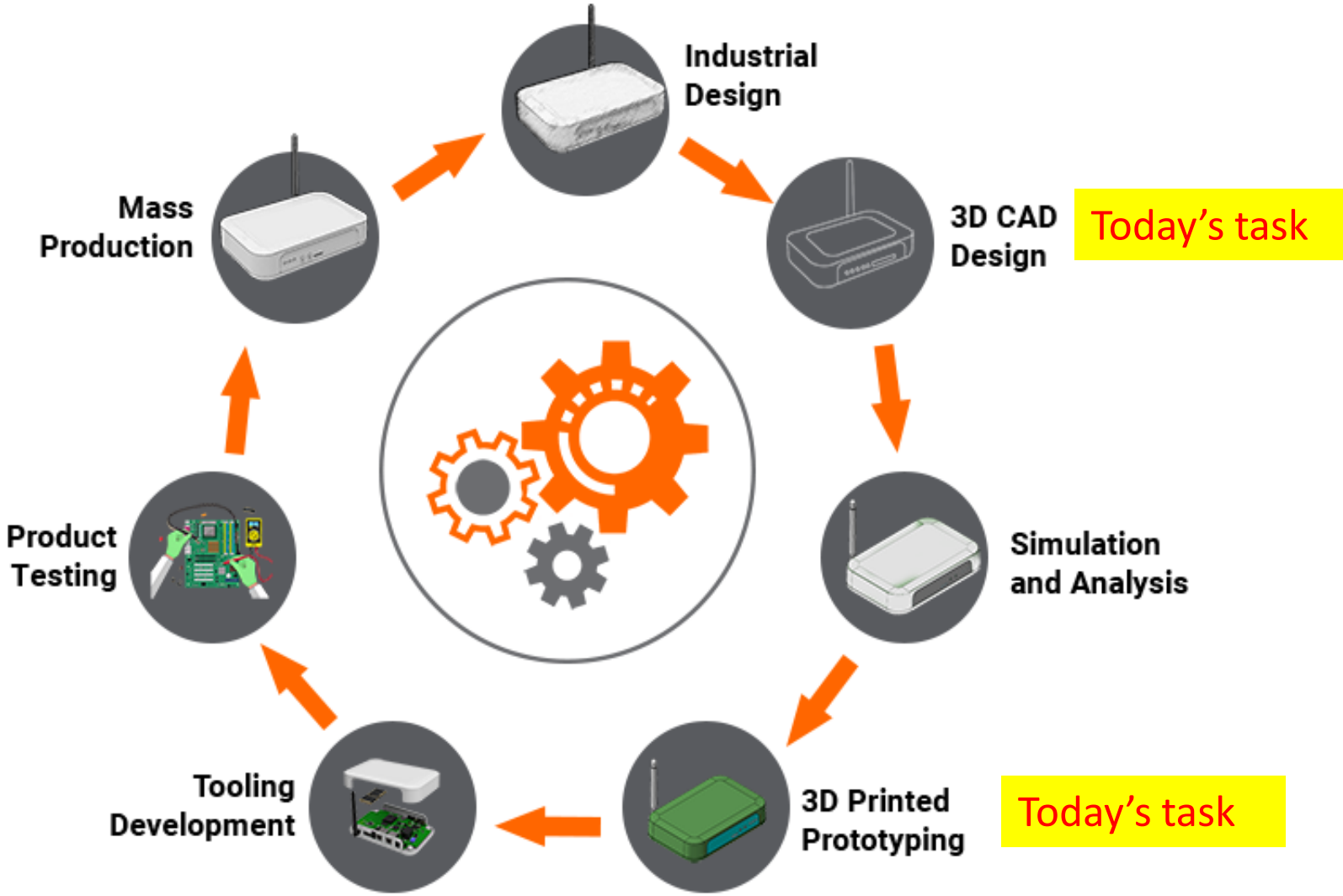
*Figures have been taken from open source repository.

Engineering drawing



*This is taken from open source repository..

Stages in product development



*The diagram has been taken from the open access source and can be accessed by clicking here.

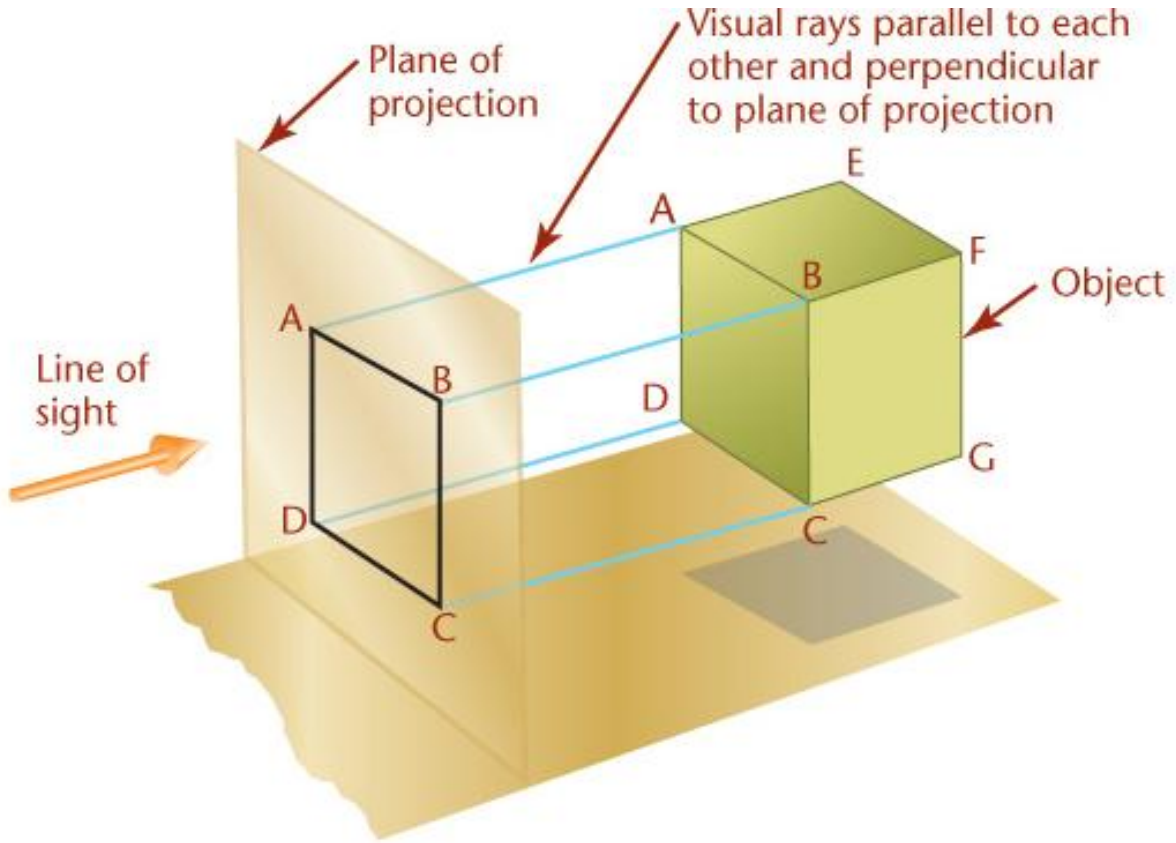
Key Advantages of CAD

- Productivity (=speed) increase
  - Automation of repeated tasks,
  - Insert standard parts (e.g. fasteners) from database.
- Supports changeability
  - Don’t have to redo the entire drawing with each change,
  - Keep track of previous design iterations.
- Communication
  - With other teams/engineers, e.g. manufacturing, suppliers,
  - With other applications (CAA/FEM, CAM),
  - Marketing, realistic product rendering,
  - Accurate, high quality drawings.
- Analysis
  - Mass Properties (Mass, Inertia),
  - Collisions between parts, clearances.

Views in engineering drawing

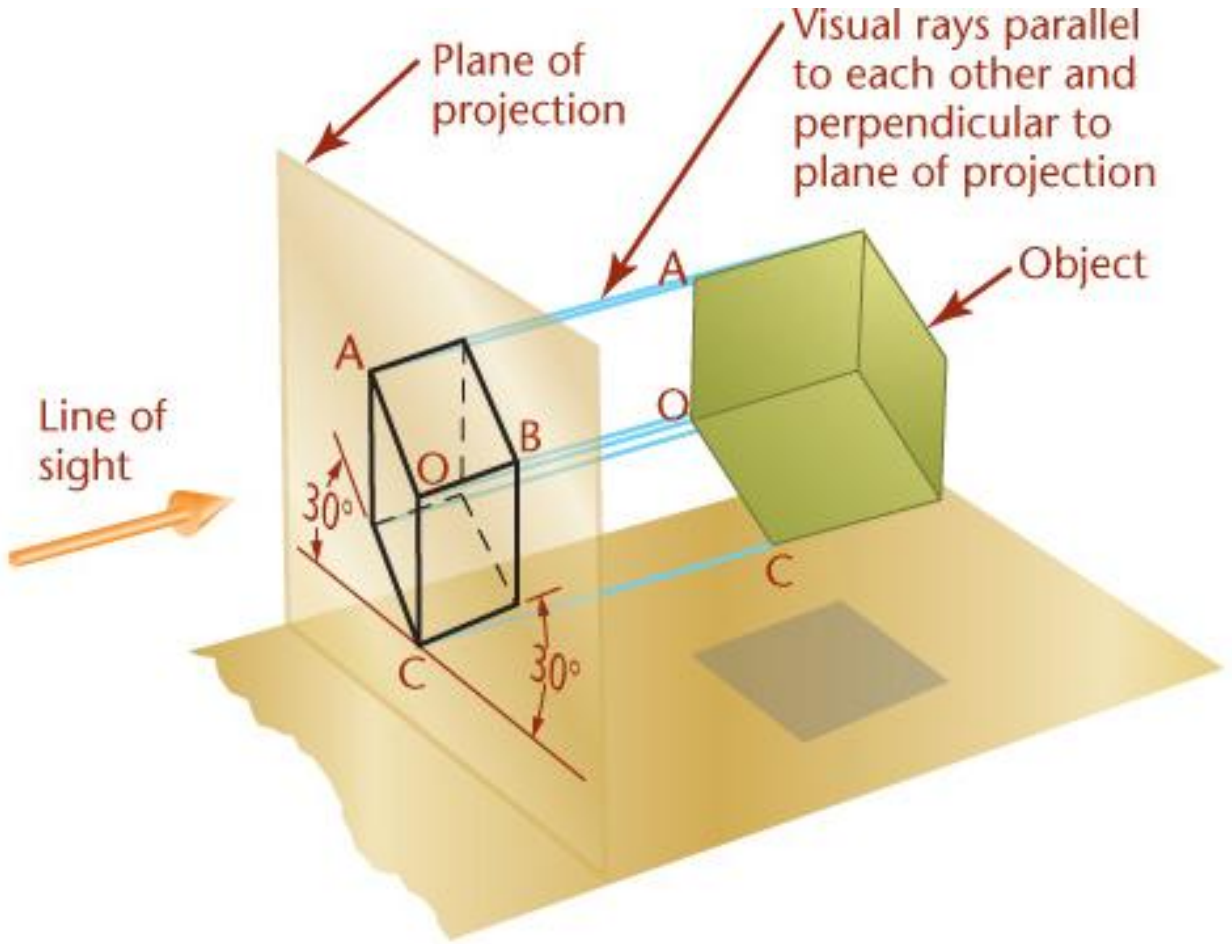
(1)Orthographic , (2) Isometric.

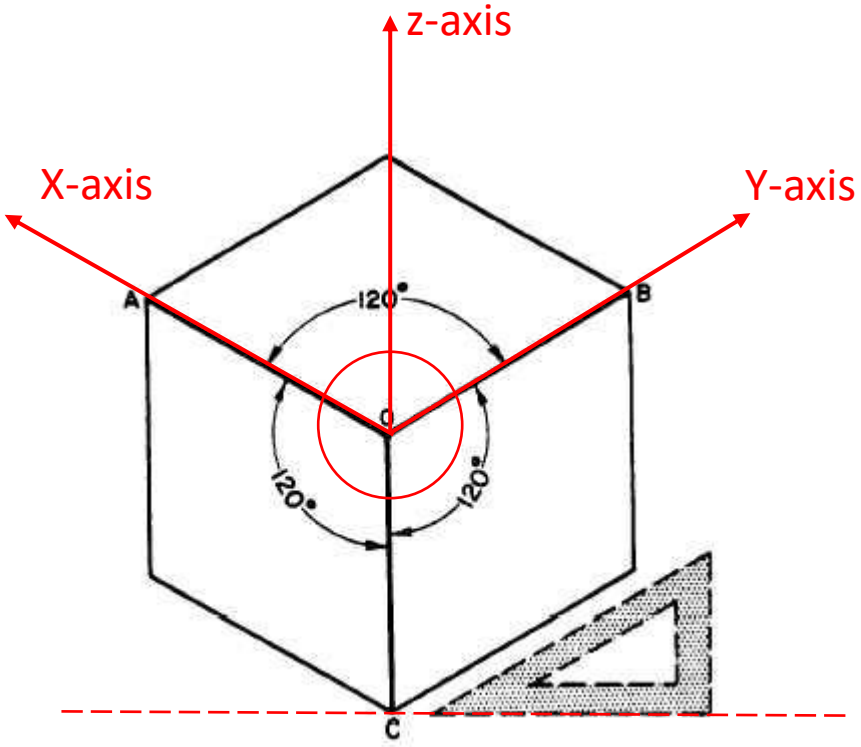
In orthographic view, an object is represented in flat-looking individual planes. The angle between each plane is 90 degrees.



Gill, Carolina. "Visualizing Continuity between 2D and 3D Graphic Representations." *DS 33: Proceedings of E&PDE 2004, the 7th International Conference on Engineering and Product Design Education, Delft, the Netherlands, 02.-03.09. 2004*. 2004.

In isometric projection, the angle between any two planes is 120 degrees, while the third plane is at 90 degrees. We can see the object's depth in isometric projection.

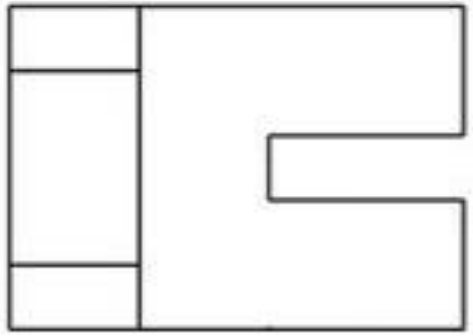




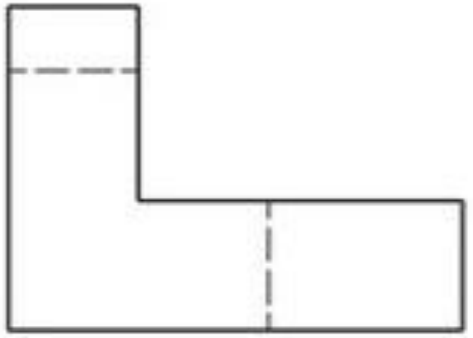
Gill, Carolina. "Visualizing Continuity between 2D and 3D Graphic Representations." *DS 33: Proceedings of E&PDE 2004, the 7th International Conference on Engineering and Product Design Education, Delft, the Netherlands, 02.-03.09. 2004*. 2004.

Example

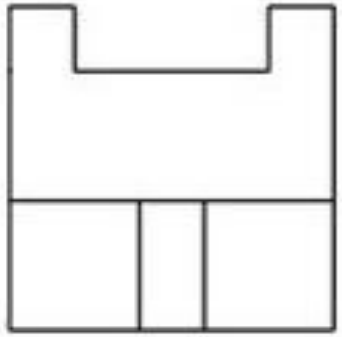
Top



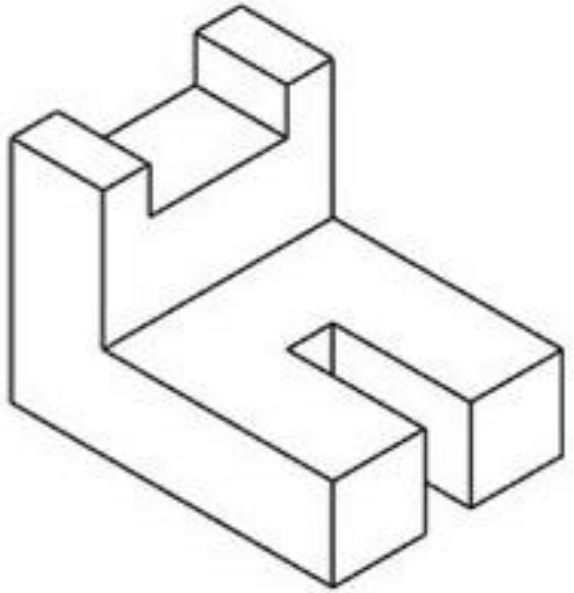
Front



Right



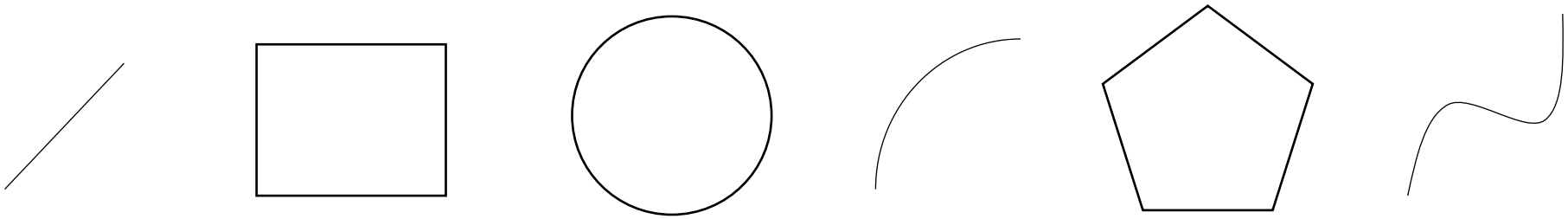
Orthographic view



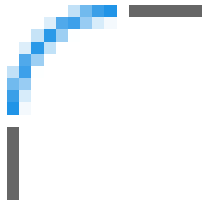
Isometric view

Wire frame geometrical approach for designs

- The wire frame allows the use of two-dimensional geometric constructs like lines, circles, arcs, points, and splines to realize the drawings.
- You can create sketches on a plane or planar face on a body, and create geometry in relation to the **XY**, **YZ**, and **ZX** planes, or at any arbitrary point in 3D space.
- Wire frame geometric profile forms the foundation of 3D geometry.

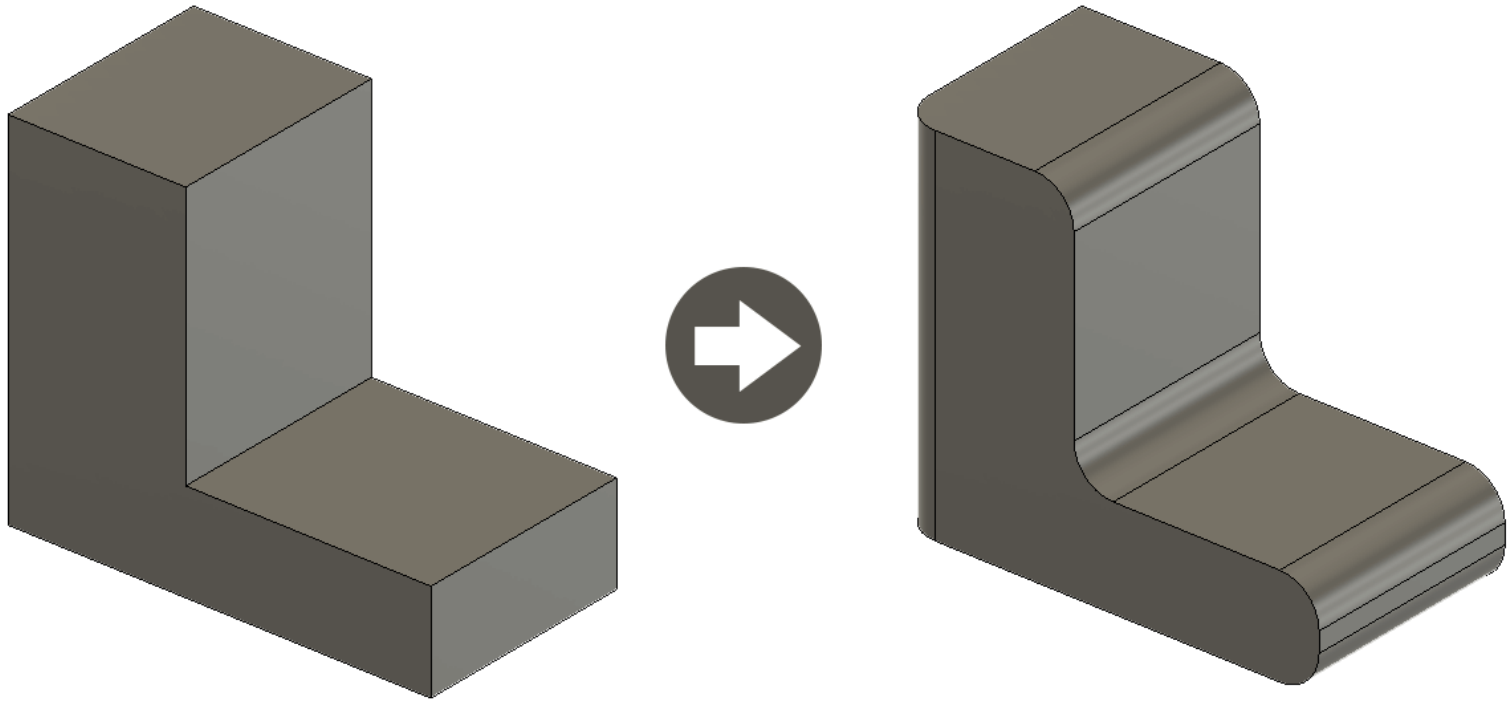


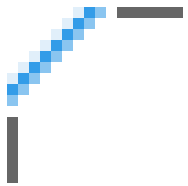
Commonly Used Commands to Modify the Drawing



Fillet

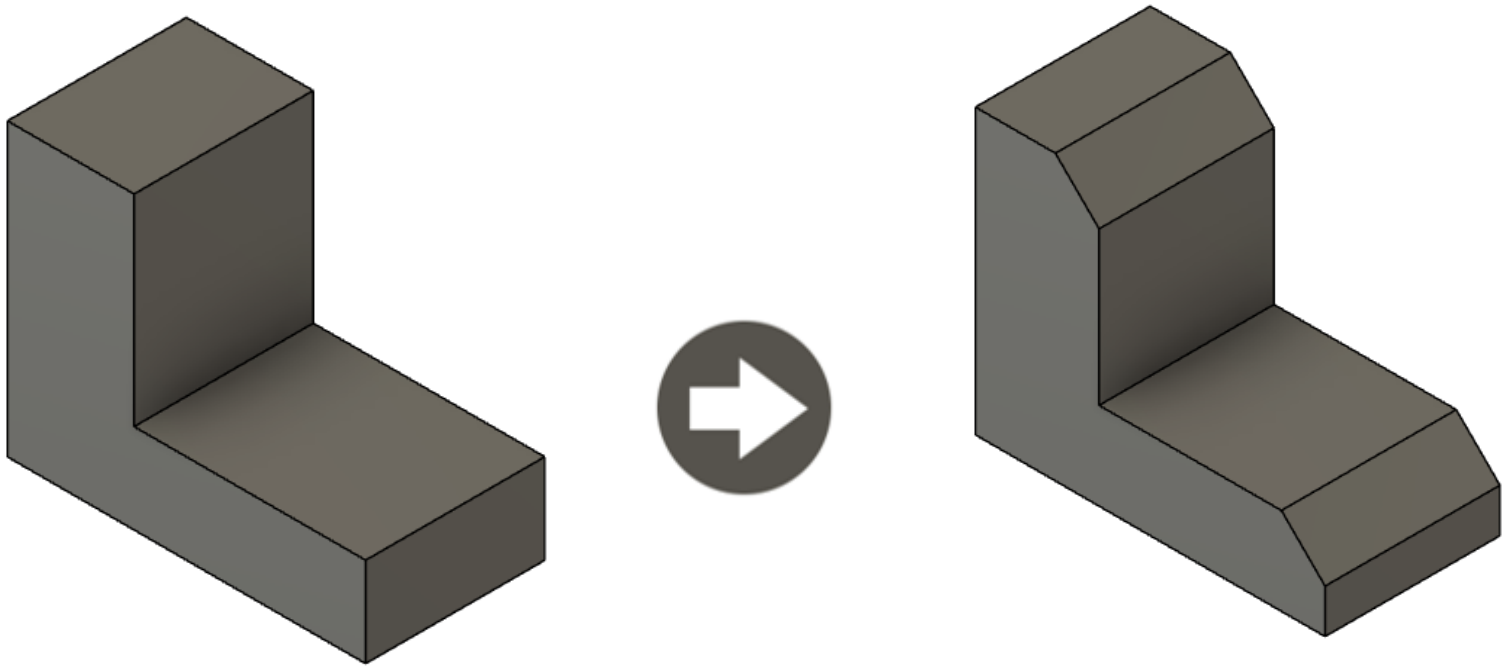
- Rounds the edges of two 2D objects.
- A round or fillet is an arc that is created tangent between two 2D objects.

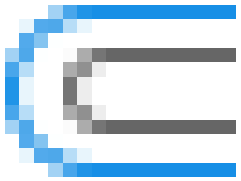




Chamfer

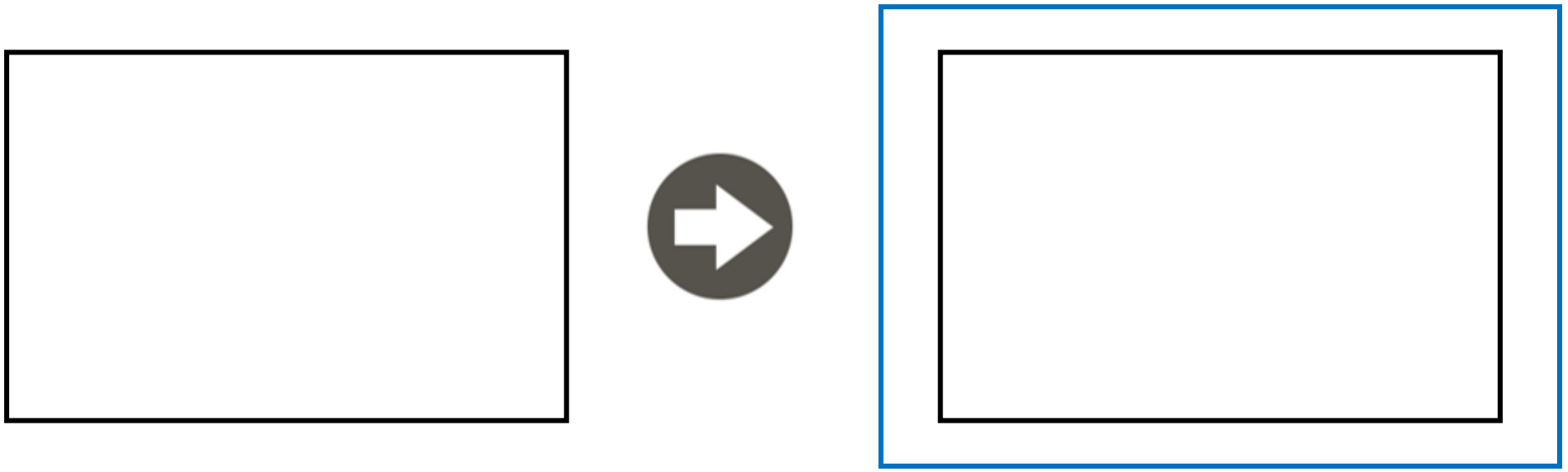
- Bevels or chamfers the edges of two 2D objects or the adjacent faces of a 3D solid.
- A bevel or chamfer is an angled line that meets the endpoints of two straight 2D objects.





Offset

- Creates a geometric object that is parallel or concentric to the selected object at a specified distance.



STEP-Universal CAD software readable file

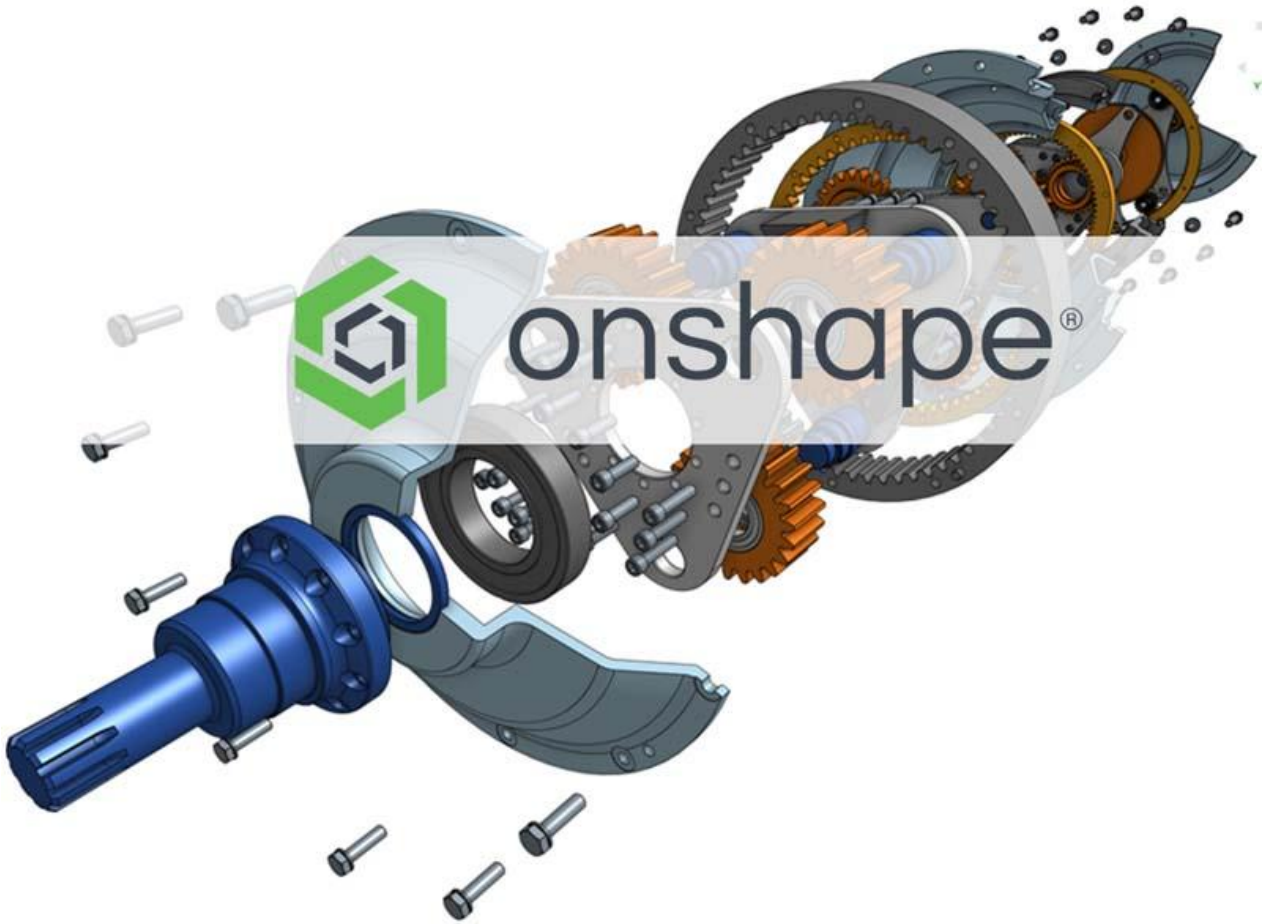
- STEP stands for *Standard for the Exchange of Product Data* and is also known as *ISO 10303*.

- STEP files make it easy to create, share, and edit 3D models across many different programs and software in a standardized format.

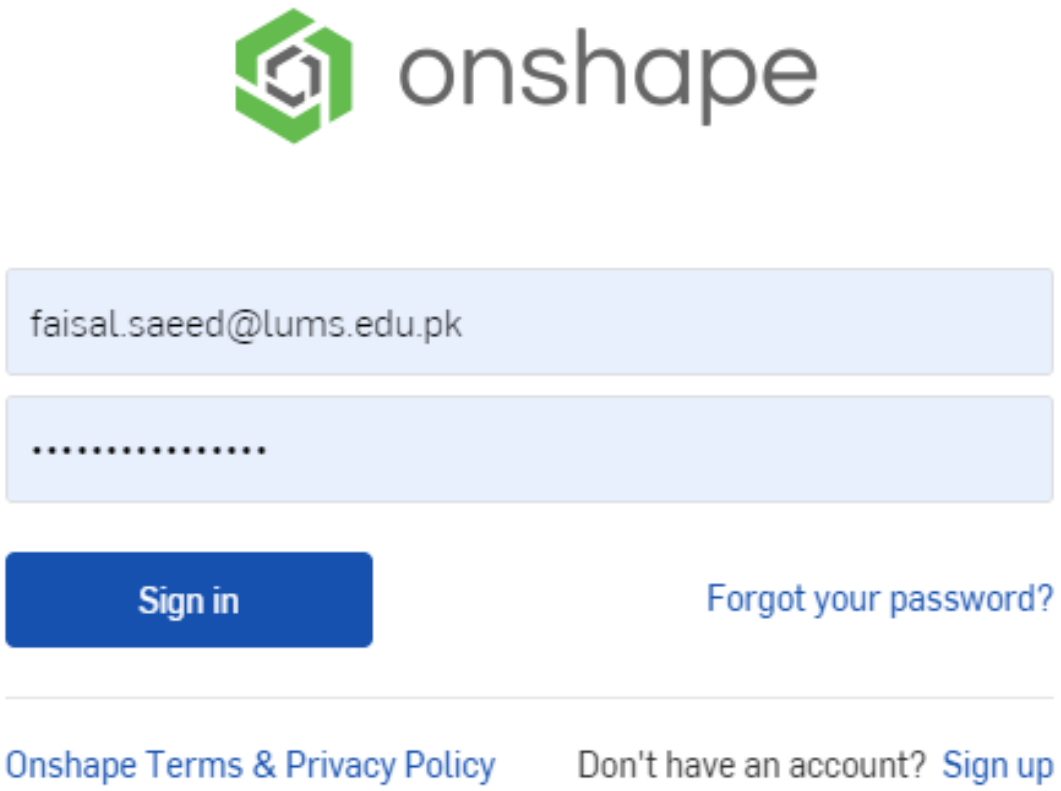
- STEP files go by several different names, including:

  - ISO 10303,
  - STP File,
  - P21-File,
  - .STEP; physical file extension.

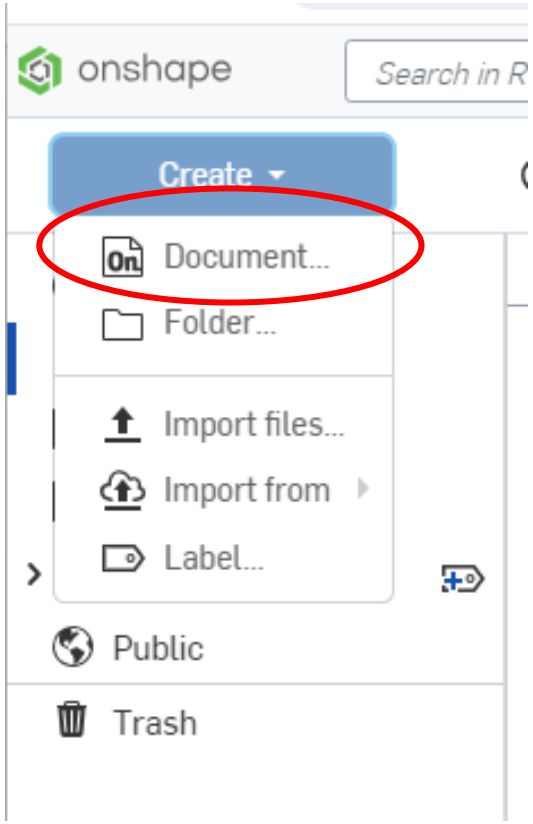
Getting Started CAD in onshape

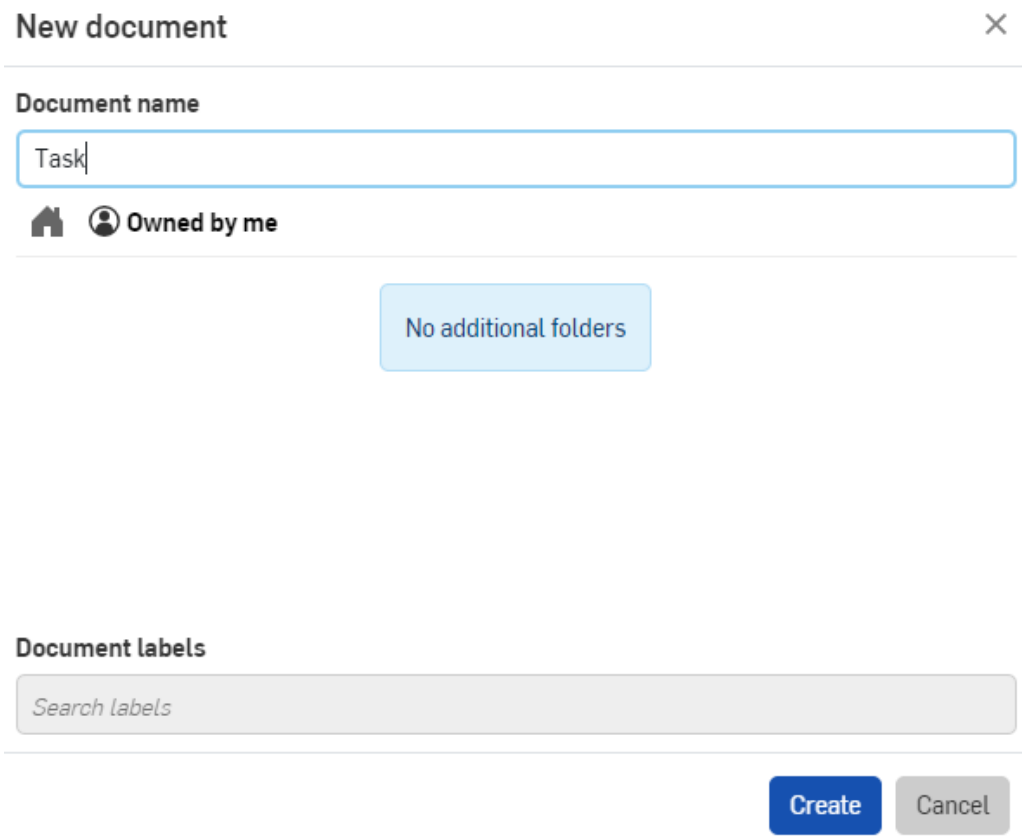


Go to the Onshape, a cloud based CAD application and Sign In with your onshape login credentials.

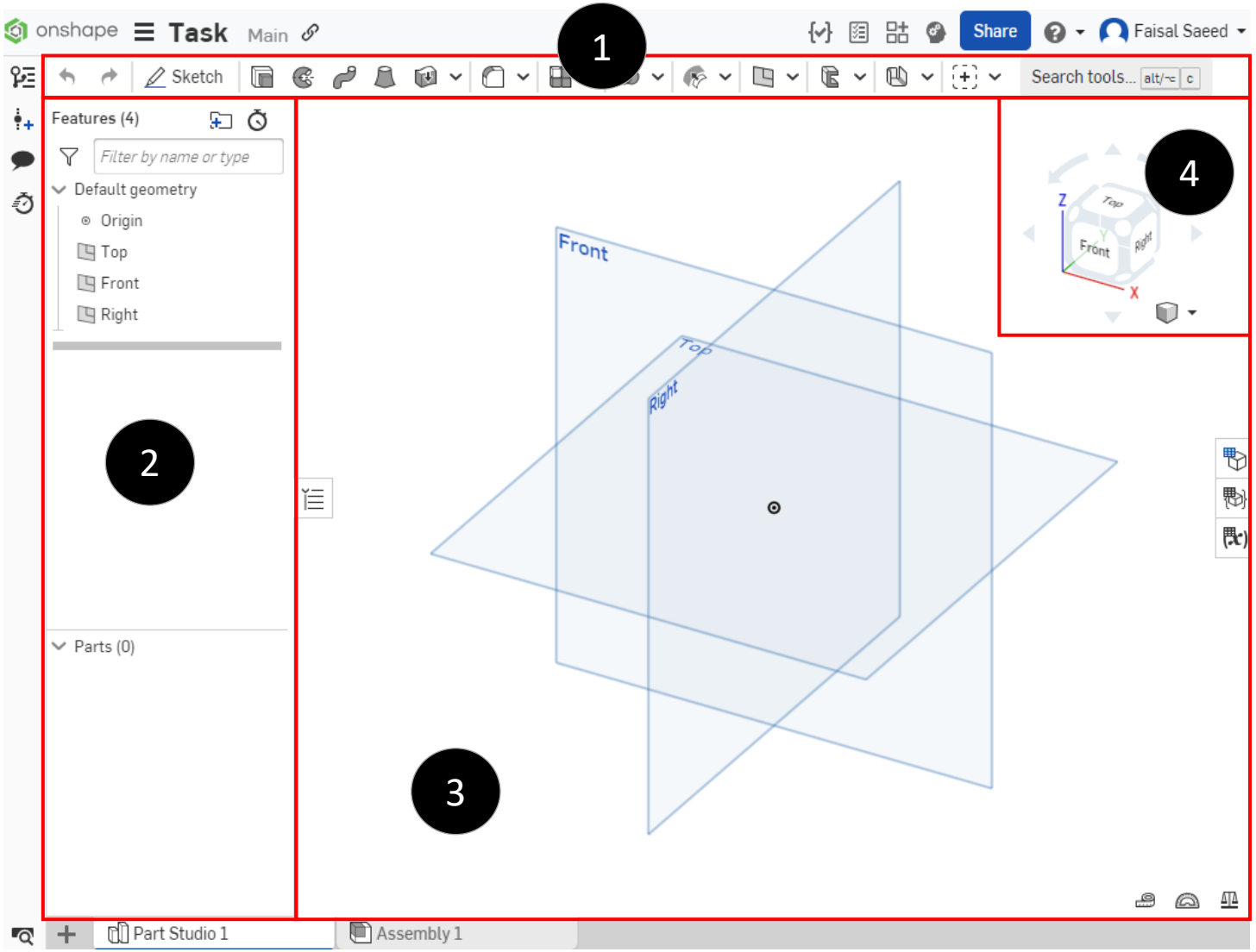


1. To begin CAD design, Go to Create and select Document.
2. Give the document a name (in the dialog that opens).
3. Click OK to create and open the document.





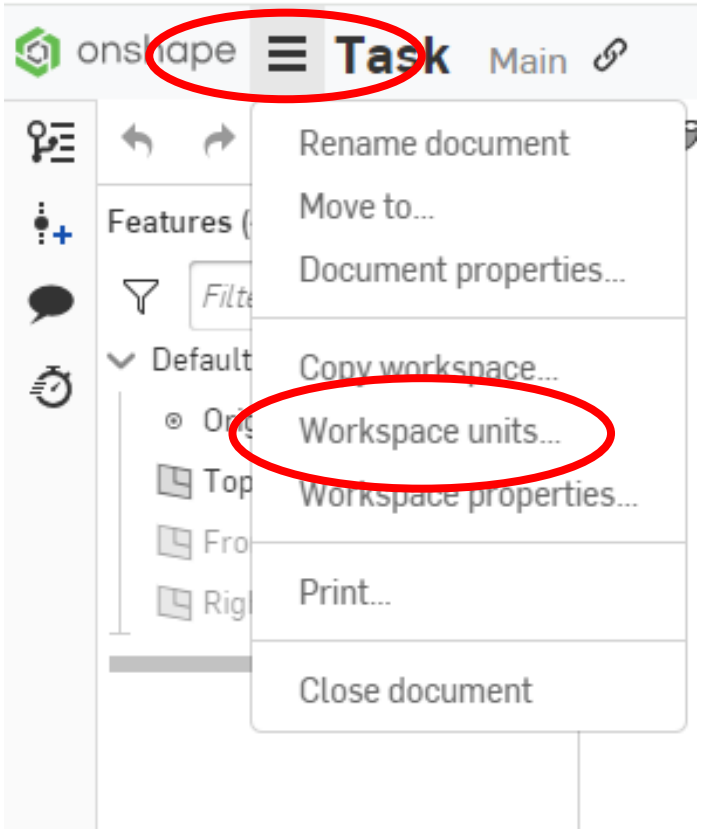
User Interface

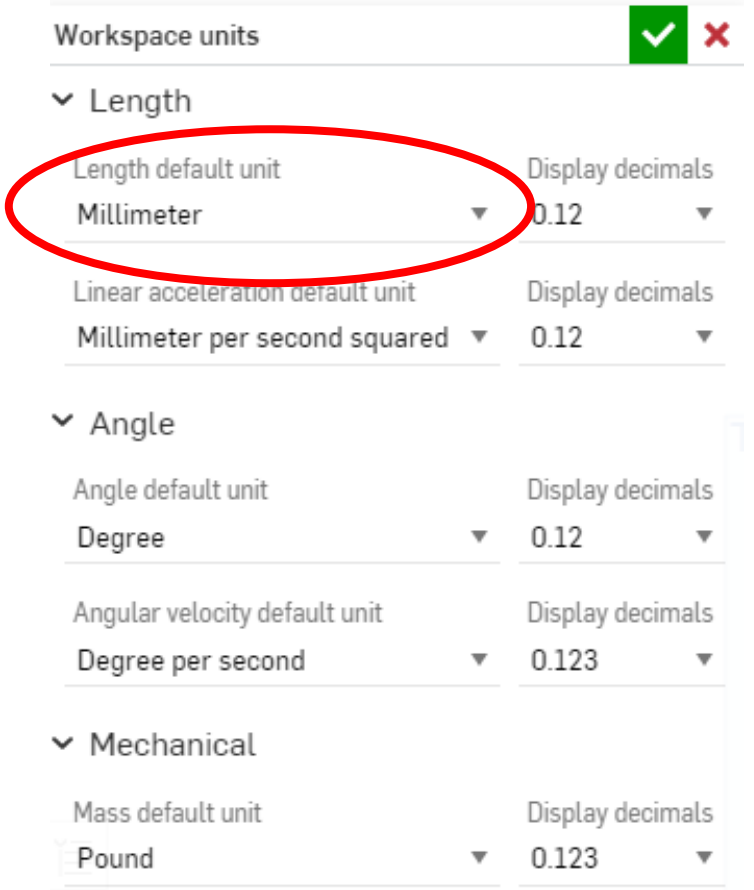


1. **Tool Bar**
2. **Features bar**
3. **Parts/Assembly**
4. **View Cube**

Setting the Units (2D or 3D Drawings)

1. Go to *MENU tab* ☰ .
2. Choose Workspace units, Workspace units window pop-ups. Change the drawing units according to your requirements.

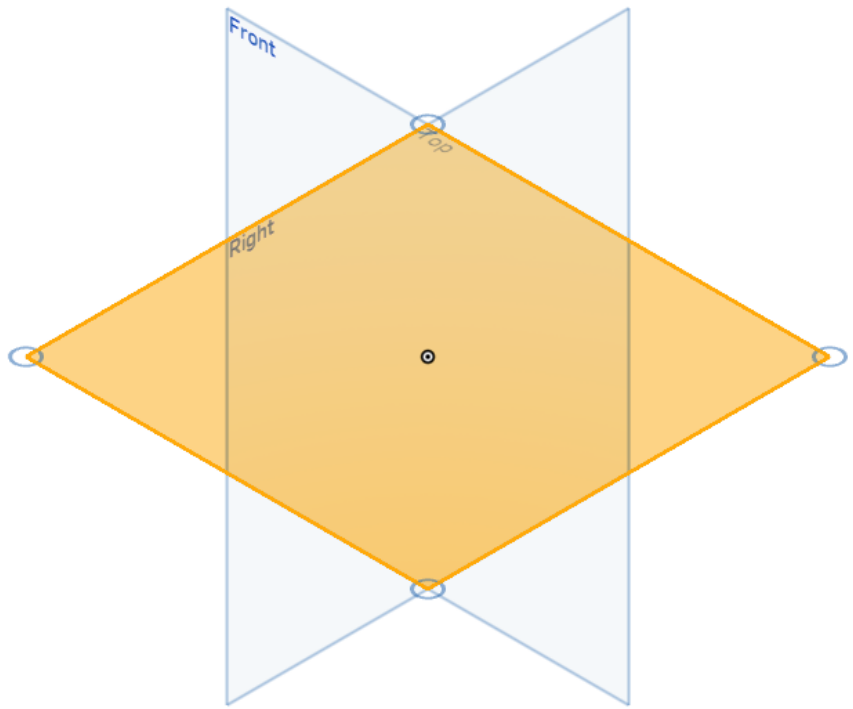




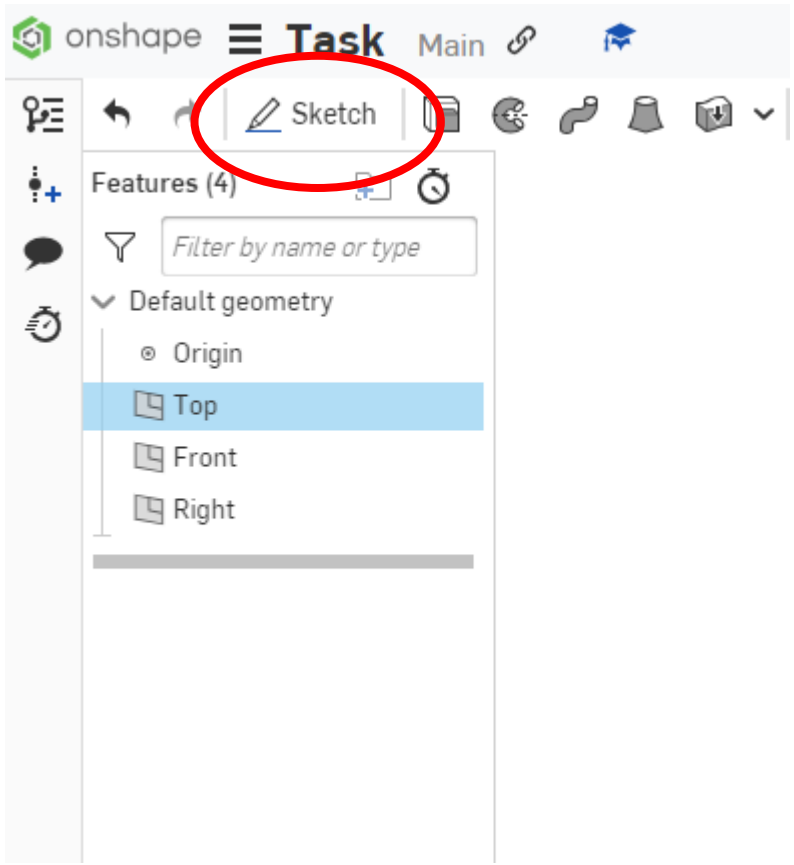
2D CAD Design (1)

1. Select the Plane or the Planar Surface.
2. Go to Sketch in Tool bar to create a sketch file.

1

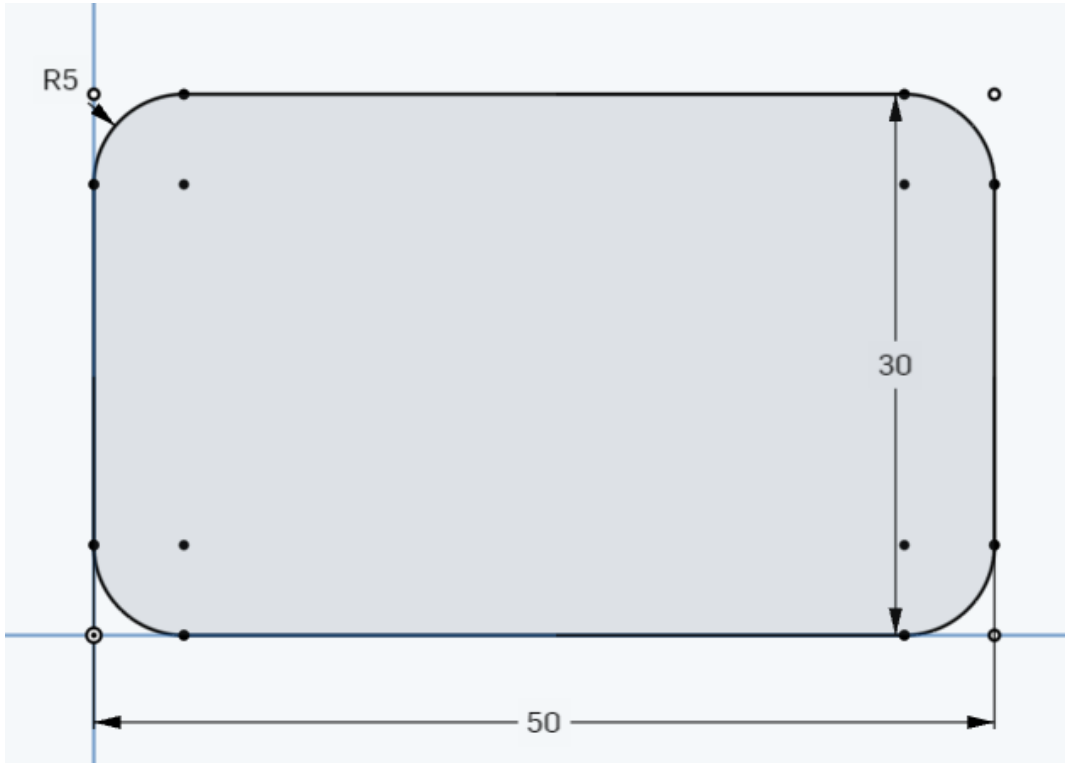


2



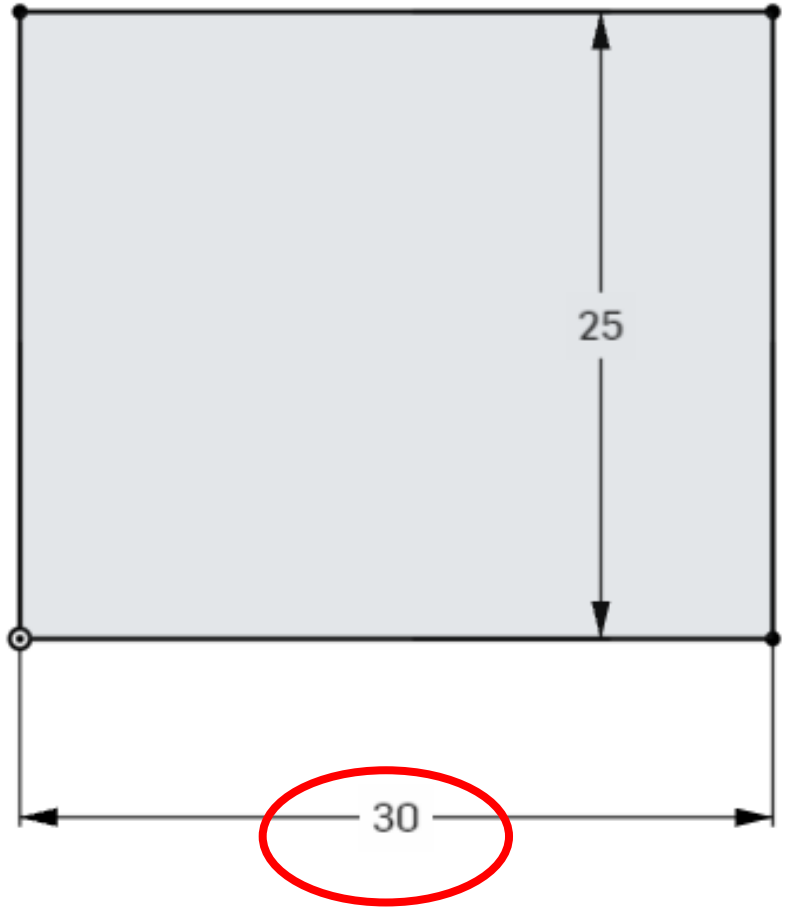
2D CAD Design (2)

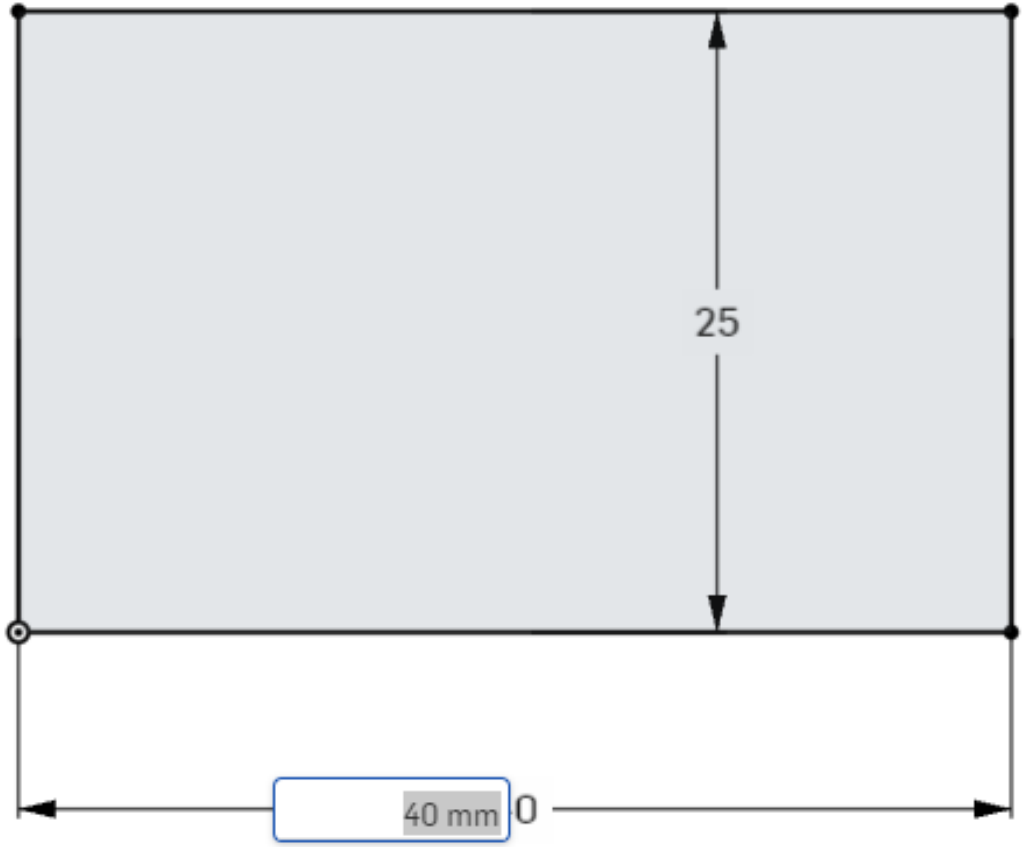
3. Use appropriate geometric constructs according to the drawing requirements.



2D CAD Design (3)

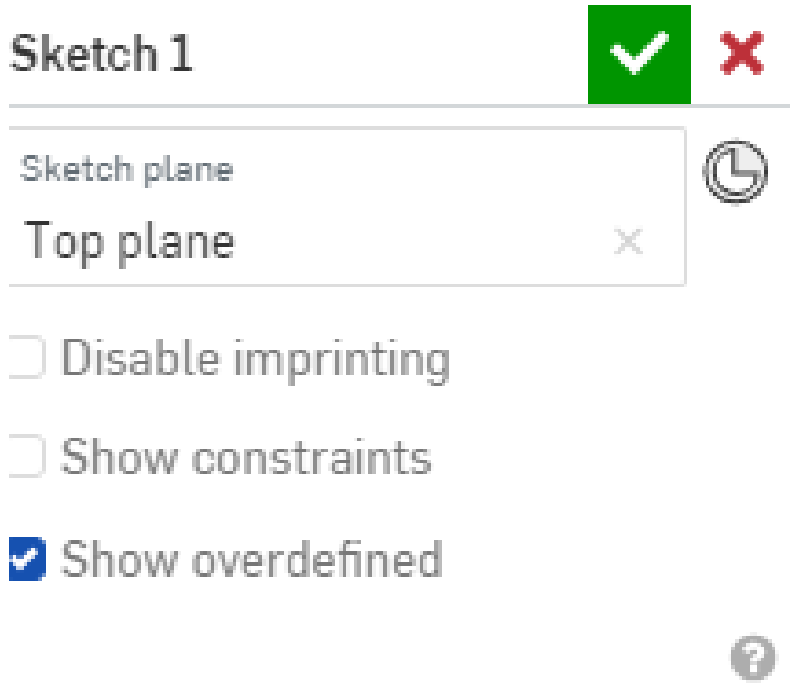
4. Double click the intended dimension to be changed. Press ENTER key to lock the changed measurement.





2D CAD Design (4)

Once finalized, click on FINISH SKETCH ✅.



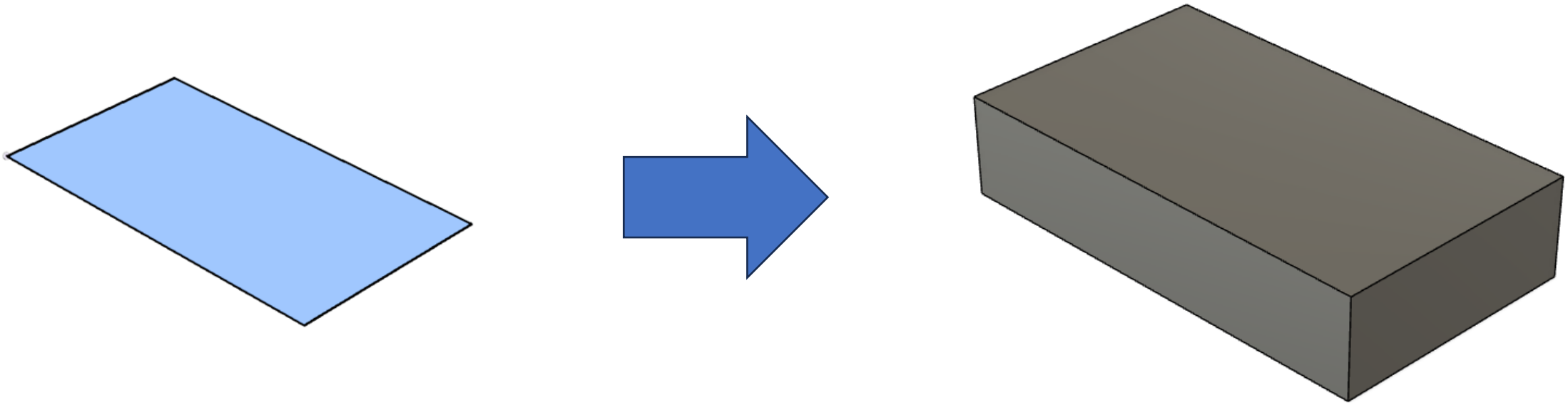
How to Measure the Dimensions in Design

If in the Sketch Environment

Sketch Dimension tool

If Finished the Sketch

Show Measure details

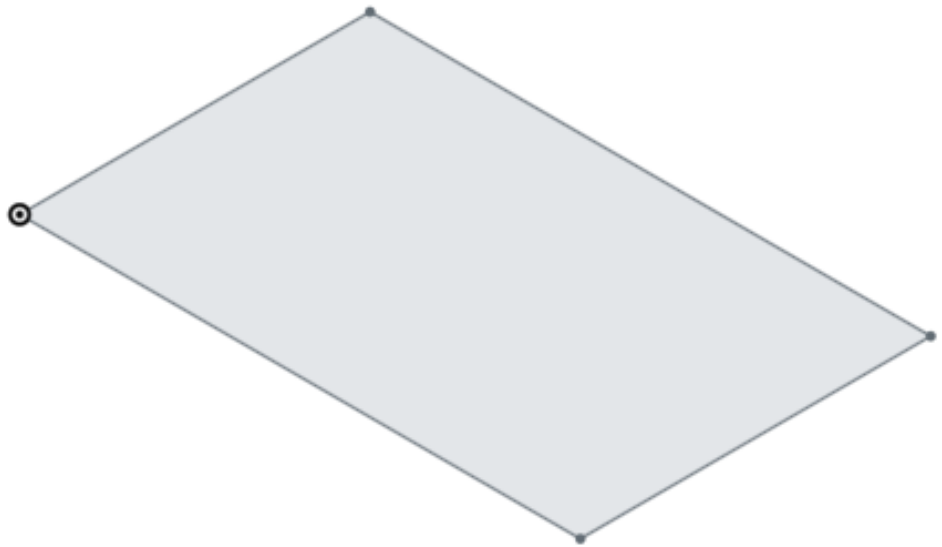


2D to 3D

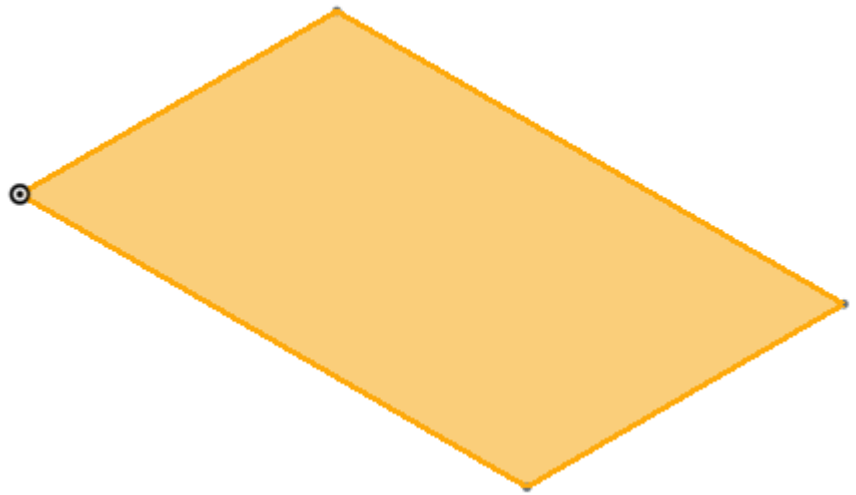
Steps for creating a 3D CAD Model

Implementing some Solid structures from sketches/wireframes

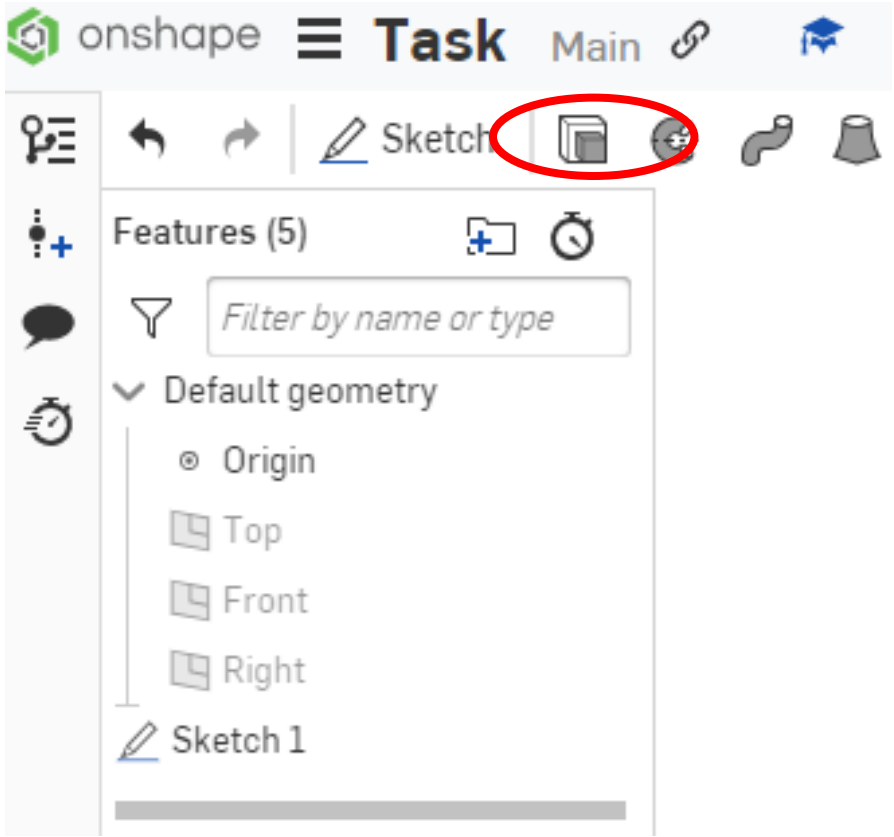
1. Construct an arbitrarily shaped 2D drawing.



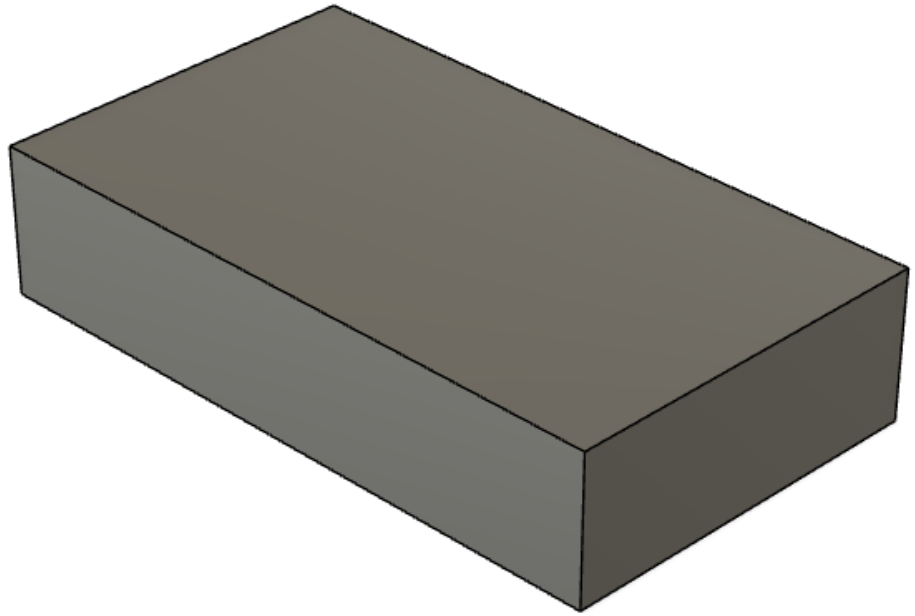
2. Click on the drawing face.



3. Use *Extrude* command in the *Tool bar* to extrude (or pull out) the enclosed area within the closed boundary.

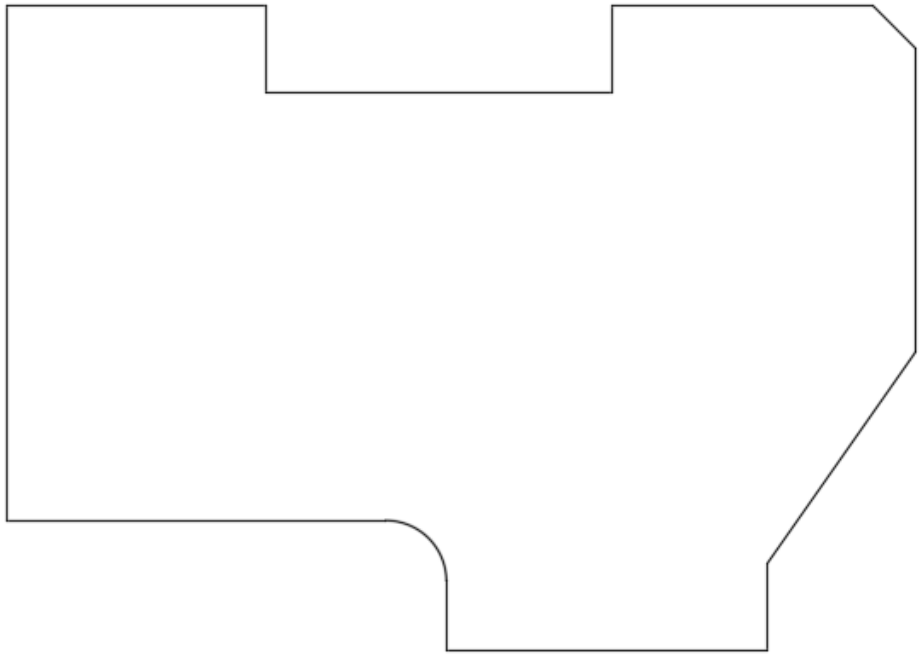


4. Three dimensional Solid Body.

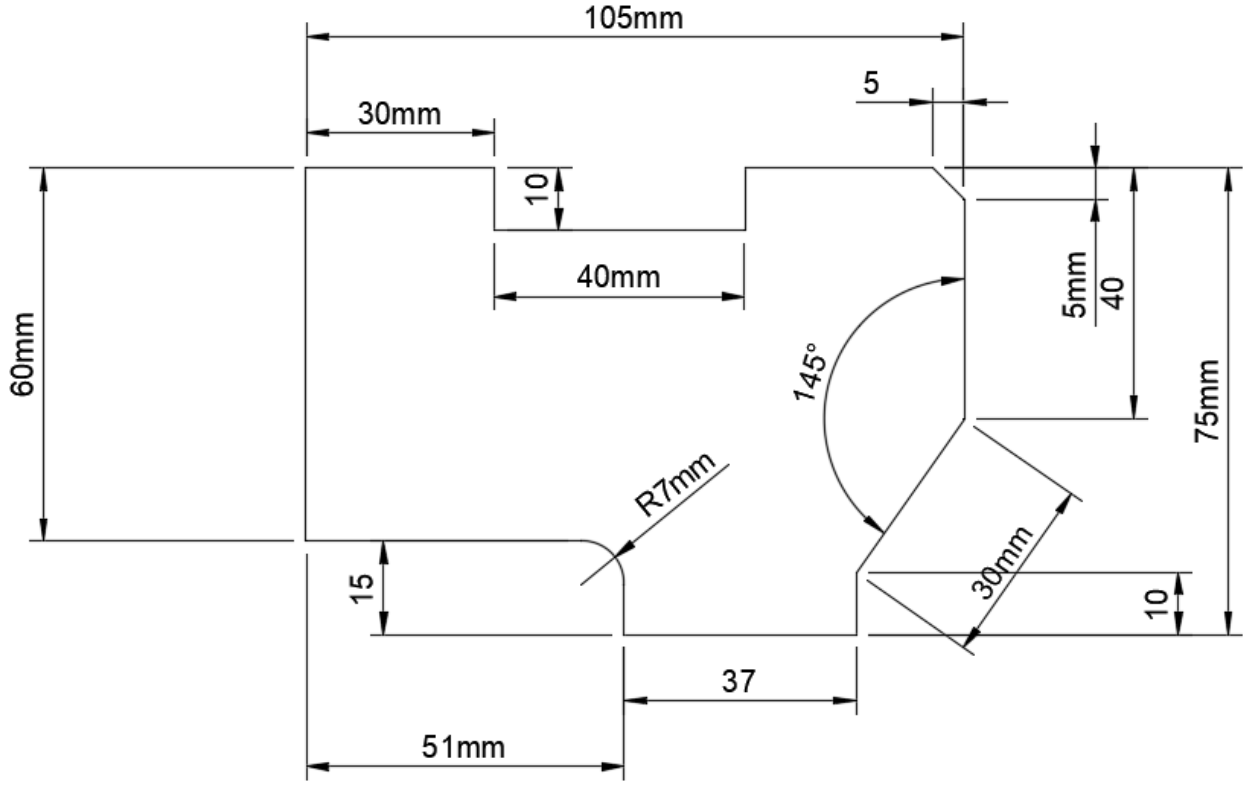


Exercise

Make a 2D CAD sketch in arbitrary plane according to dimensions mentioned in the Figure. After that, convert 2D design to the 3D object.

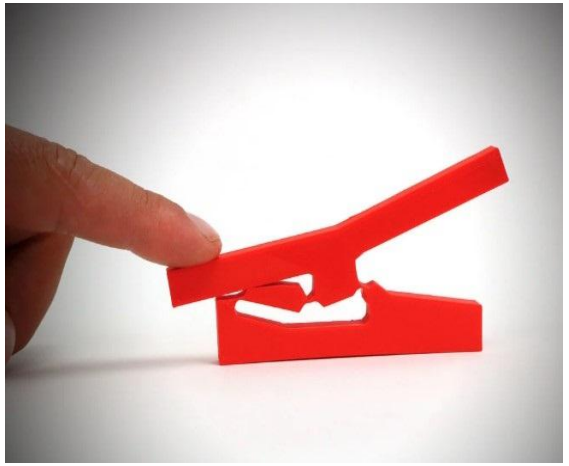
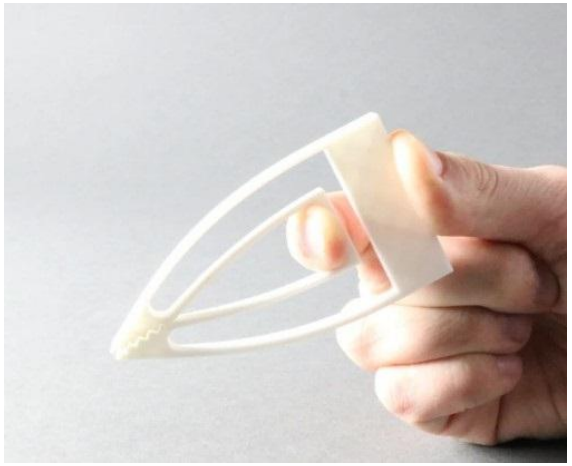


Figure

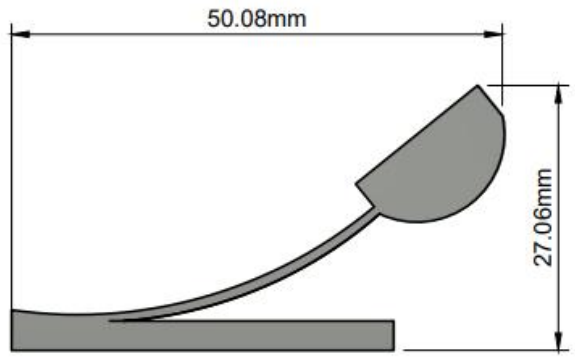


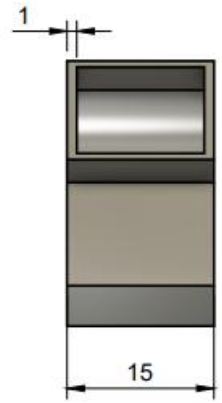
Dimensions

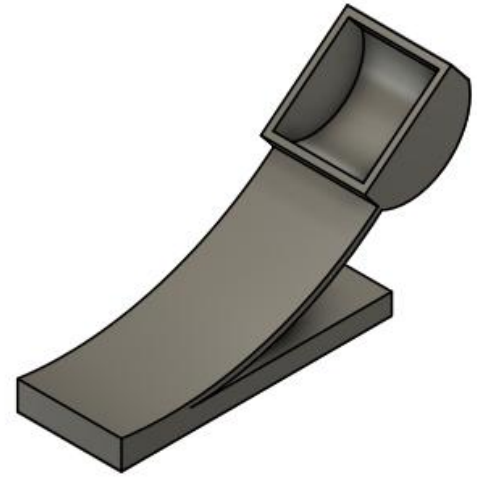
Task

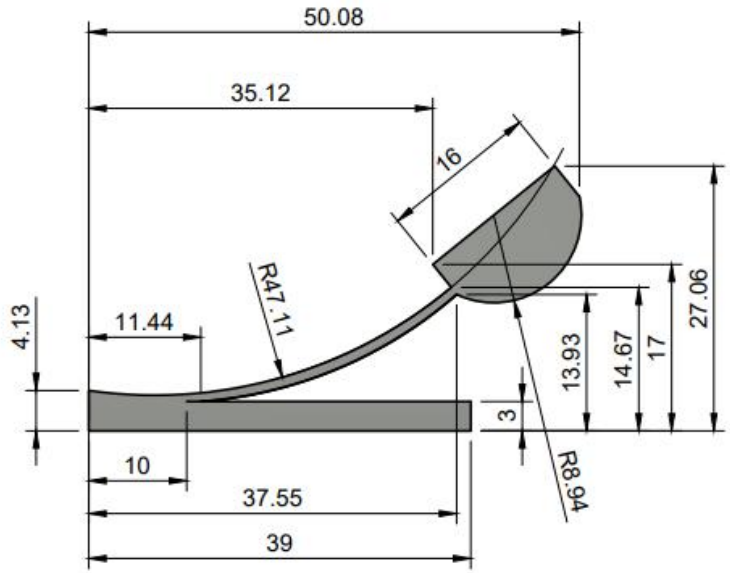
Group 1	Group 2	Group 3	Group 4
			
Euler catapult	Bistable switch	Surgical gripper	Plier

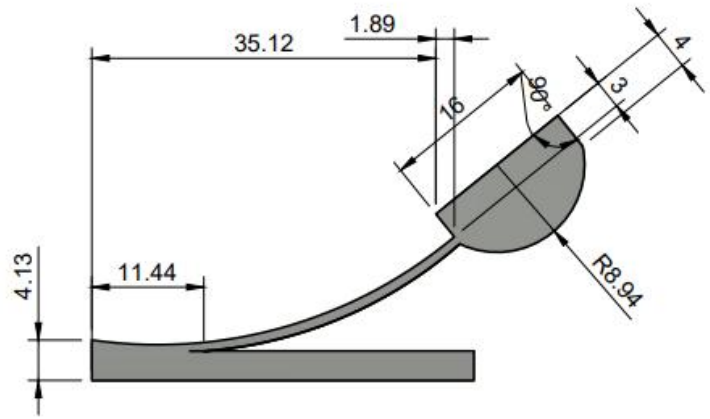
Euler catapult



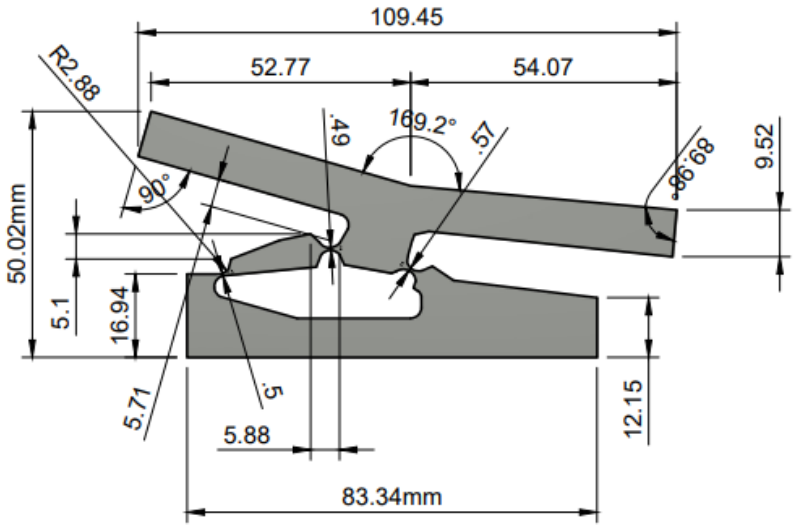


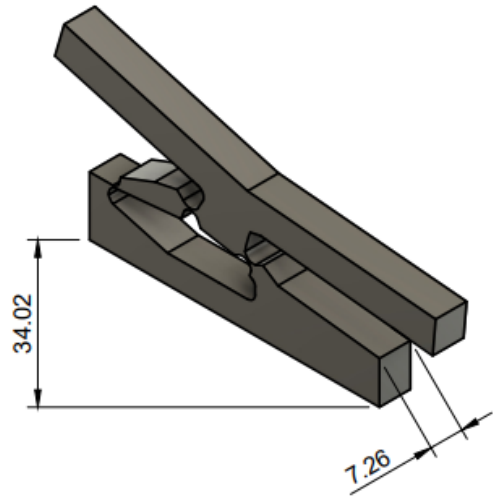


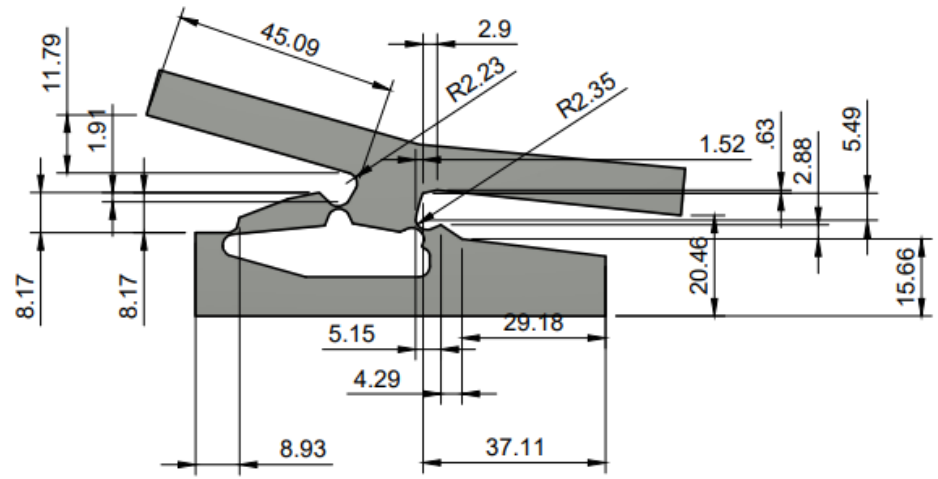


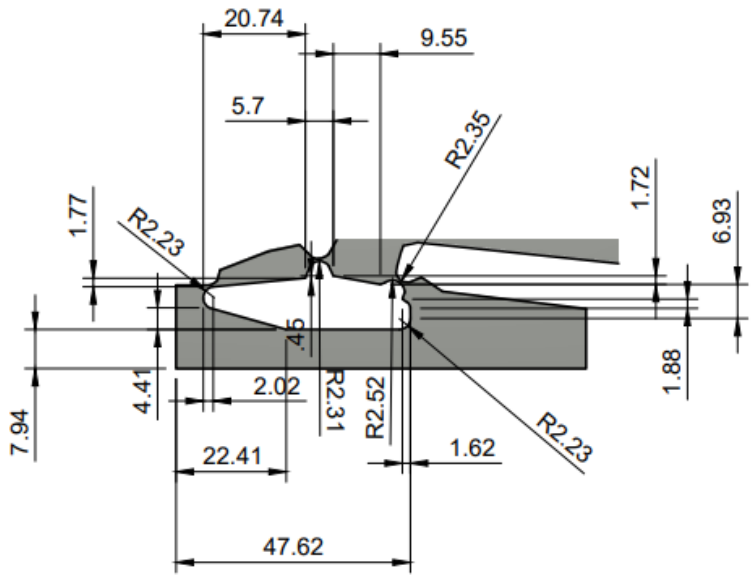


Bistable switch

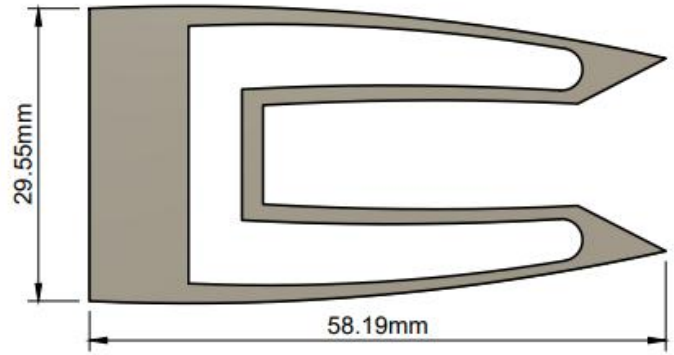


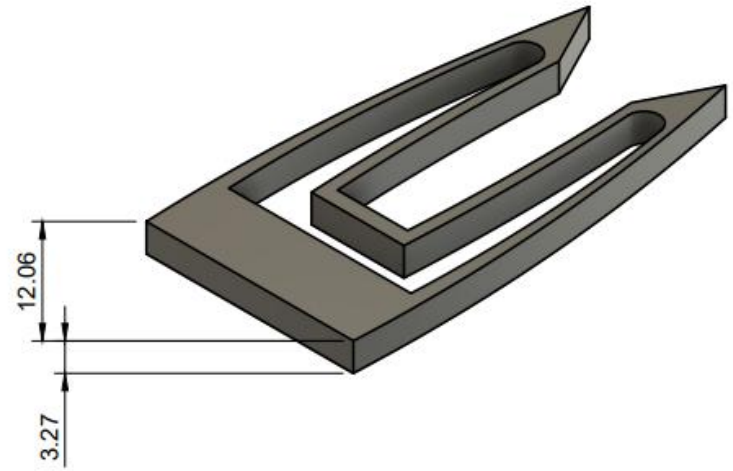


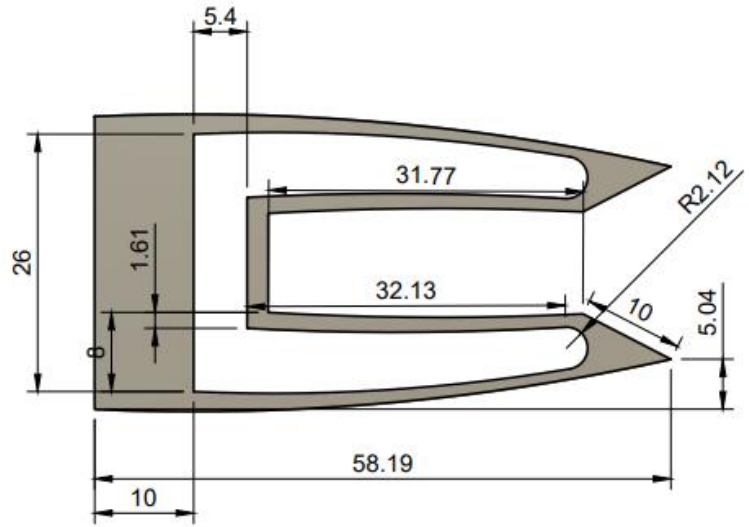


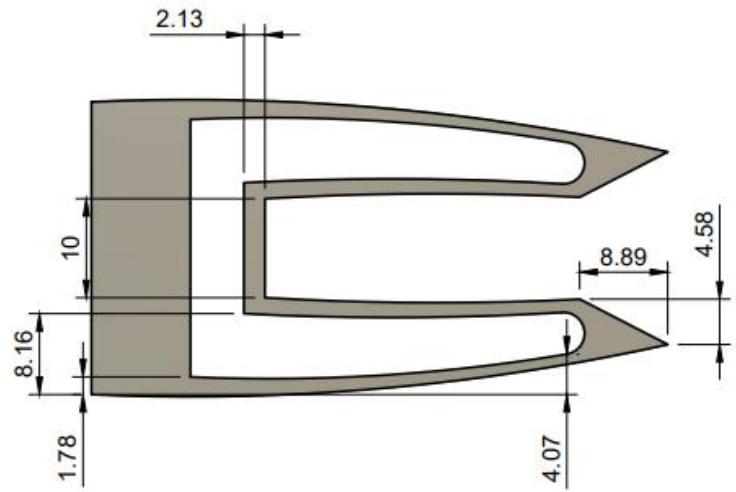


Surgical gripper

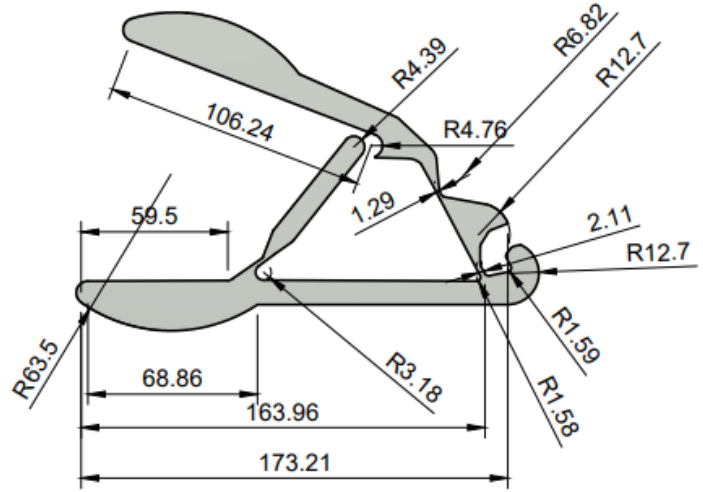


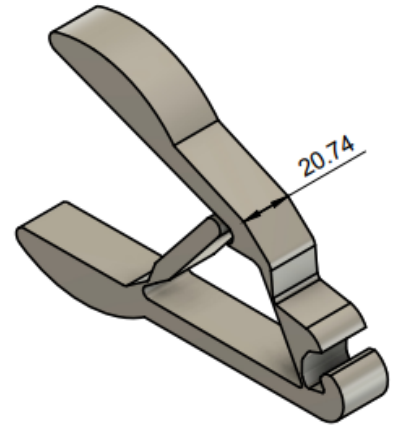


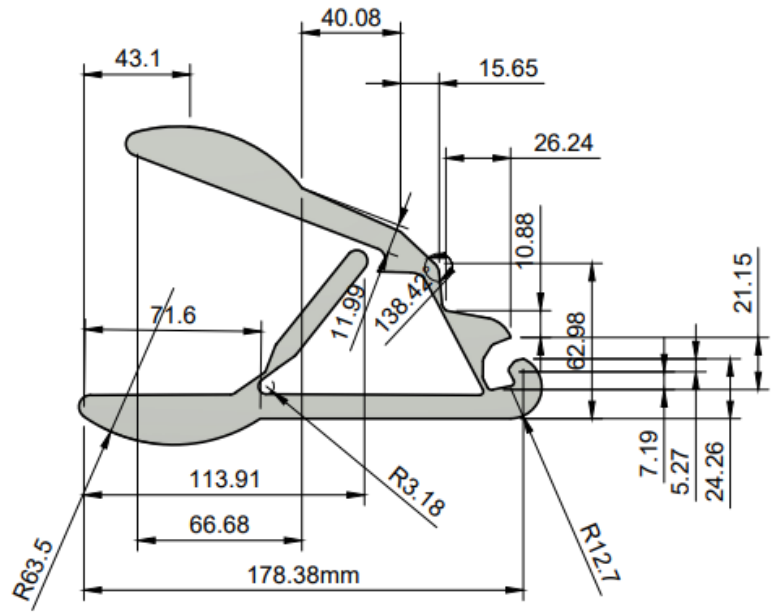


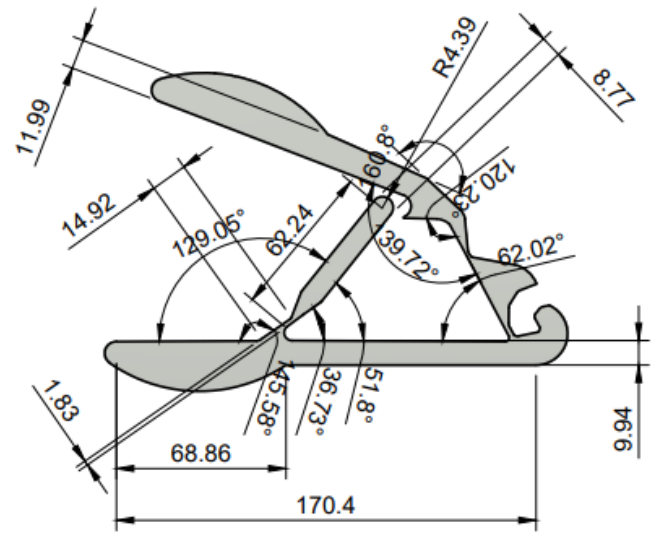


Plier


R4.39
R6.82
R12.7
106.24
R4.76
1.29
59.5
2.11
R12.7
R1.59
R63.5
R3.18
68.86
R1.58
163.96
173.21

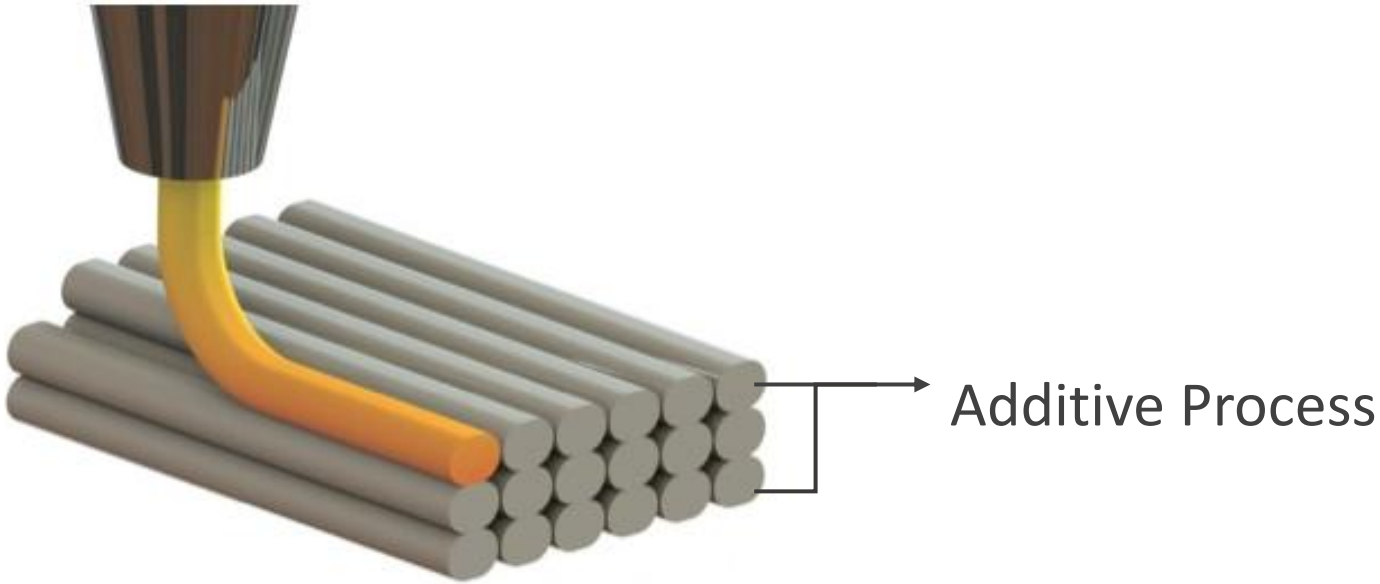

20.74


40.08
43.1
15.65
26.24
10.88
21.15
71.6
11.99
138.42°
62.98
7.19
5.27
24.26
113.91
R3.18
66.68
R63.5
178.38mm
R12.7


11.99
R4.39
8.77
160.8°
120.23°
14.92
129.05°
62.24
39.72°
62.02°
45.58°
36.73°
51.8°
1.83
9.94
68.86
170.4

3D Printing

3D printing or Additive Manufacturing is a manufacturing technique for the creation of three-dimensional objects by the additive process (layer upon layer) according to the computer-aided design (CAD) model.



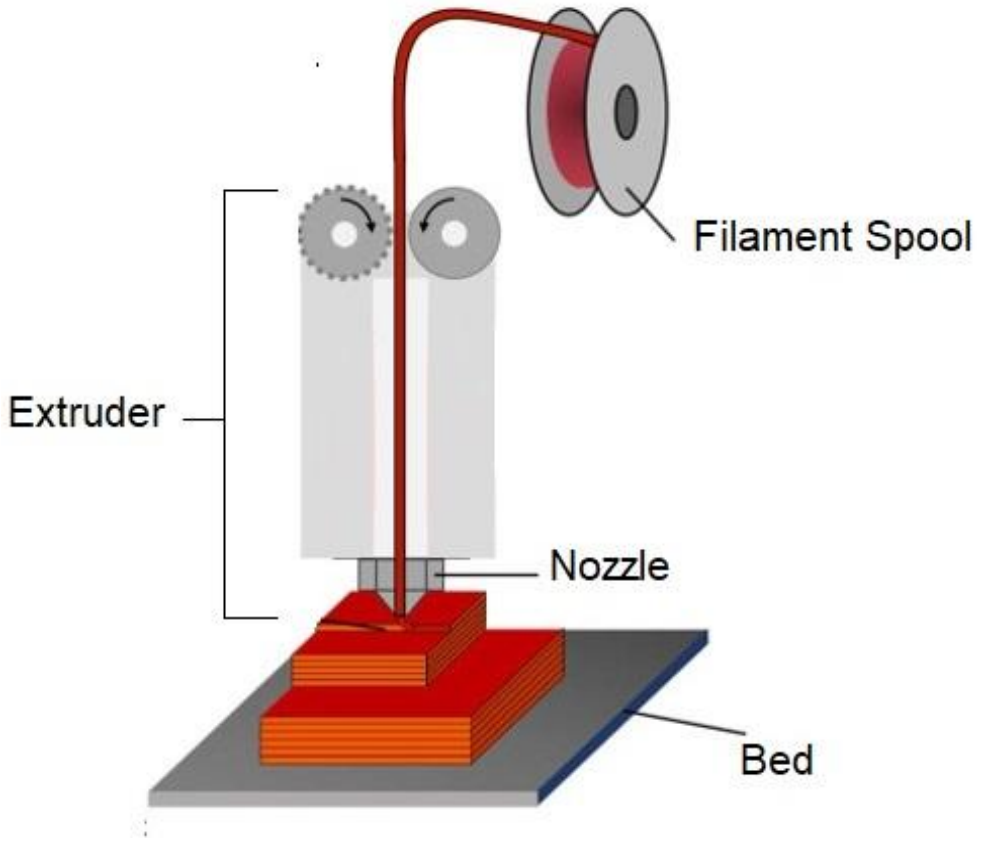
*The figure is taken from an open source repository.

Fused Deposition Modeling – One of the Additive Manufacturing Technology

Fused deposition modeling (FDM) is an additive manufacturing technology that creates 3D components using a continuous thermoplastic or composite material thread in filament form.

An extruder feeds the plastic filament through an extruding nozzle, which is melted and then selectively deposited layer by layer onto the build platform in a predetermined automated path.

The 3D Printers assigned to your tasks for 3D printing are FDM based printers.



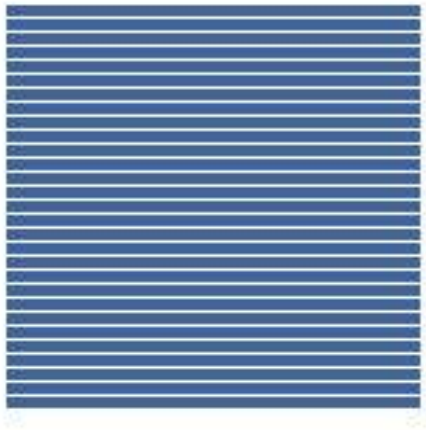
*The figure is taken from an open source repository.

Steps Involved in 3D Printing

CAD Model

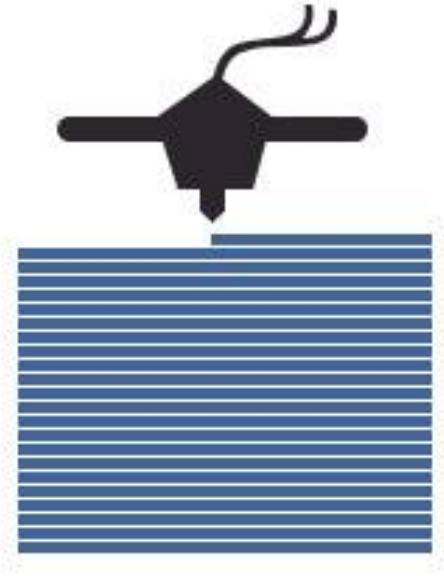
Export the file to STL Format

Slicing Software



Slices the Object and generates the G Code

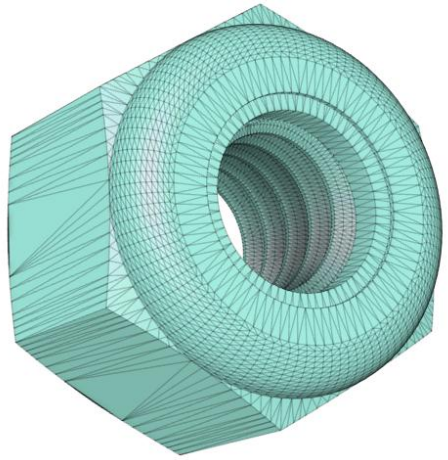
3D Printer



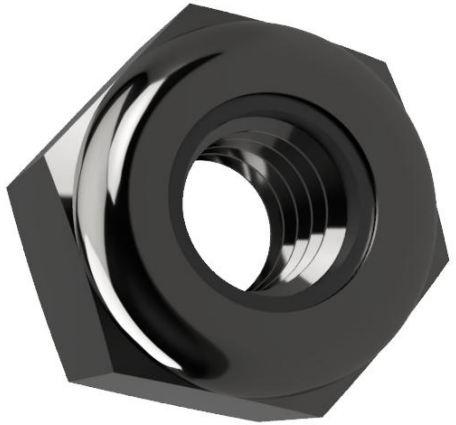
3D Printing

Why only the STL file is used for Slicing?

- Stereolithography (STL) file format represents the simplified geometry of an object as a collection of connected triangles (mesh) unlike *STEP* files that contain information about the model, material, surface, and other metadata.
- Slicing software can only interpret mesh information and therefore STL files can be directly used for 3D printing applications.



STL



STEP

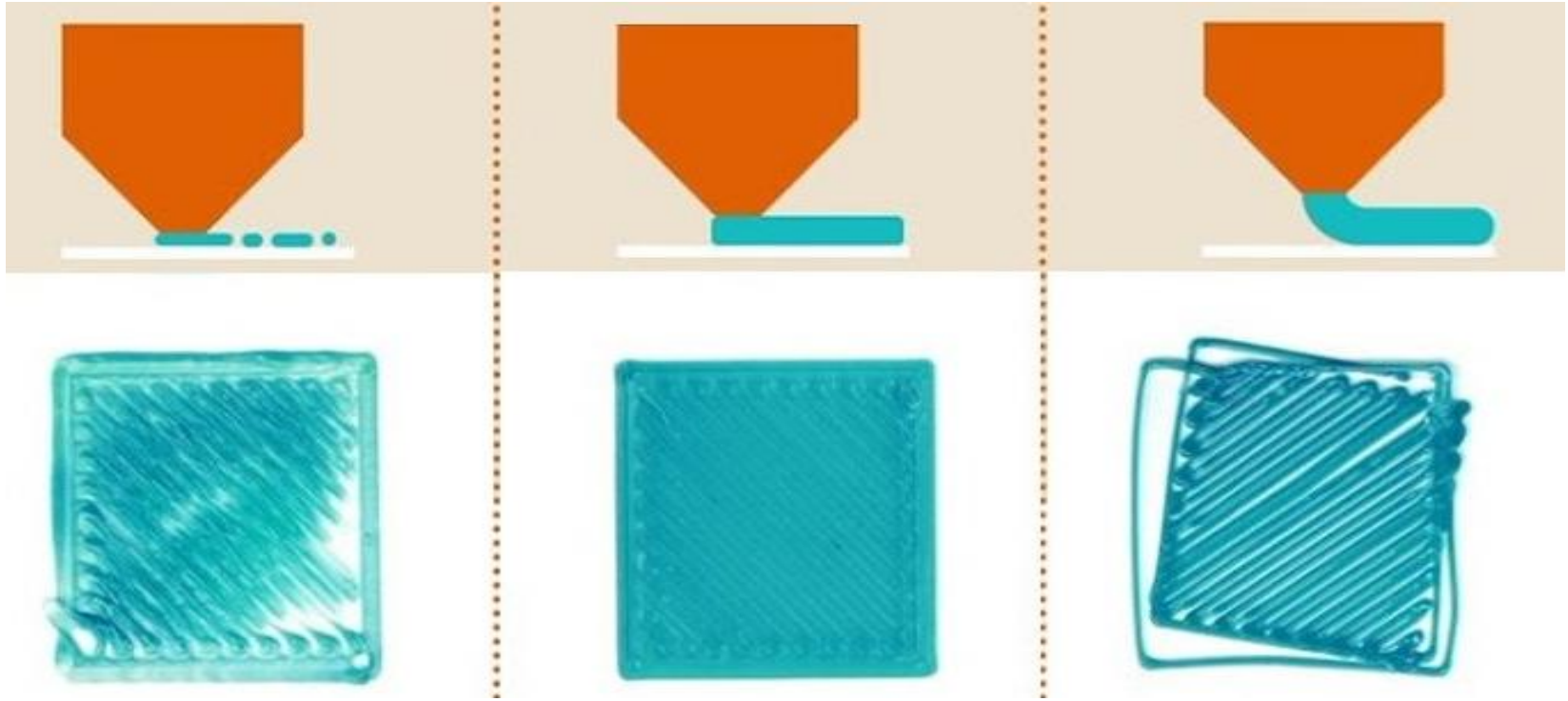
Slicer: Generating G-code for 3D designs

The slicer converts the 3D model into printable instructions (G-code) for the 3D printer.

The necessary Presets after loading the STL file includes

1. Update the 3D printer Information (Depth, Width, Height)
2. Nozzle Size
3. Filament Diameter
4. Layer Height and the Infill
5. Supports

Bed Leveling for 3D Printers



Nozzle Too Close to the Bed

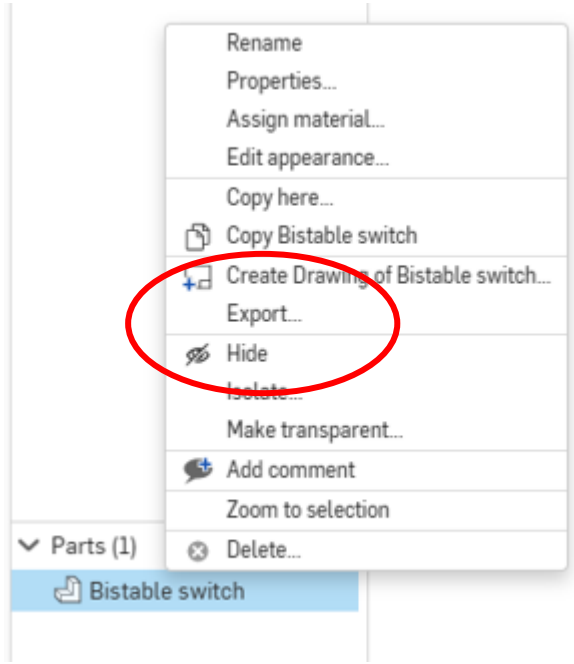
Desired

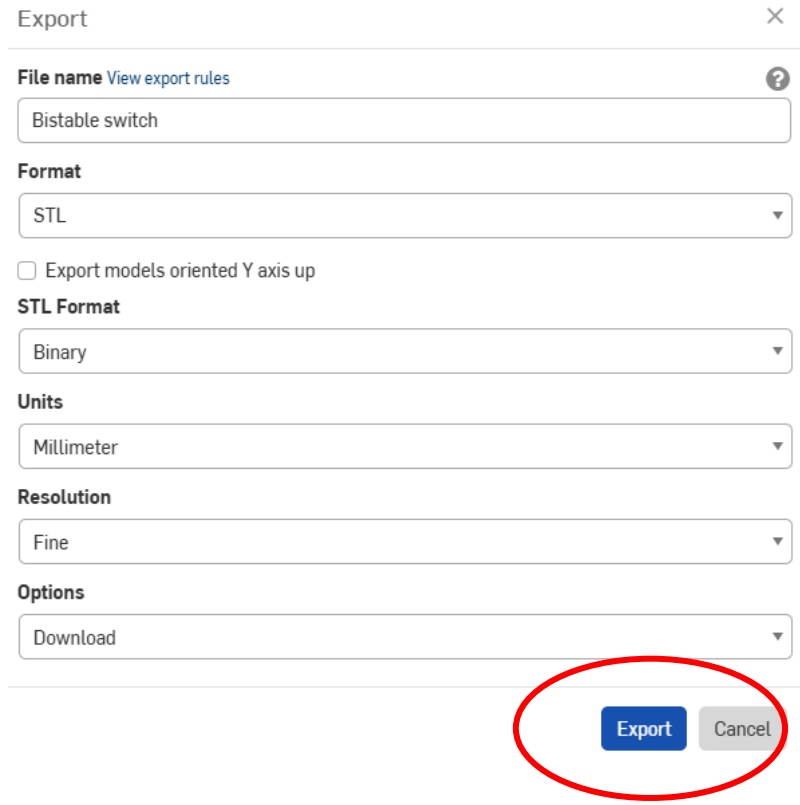
Nozzle far from the Bed

*The figure is taken from an open source repository.

Exporting CAD model in STL format

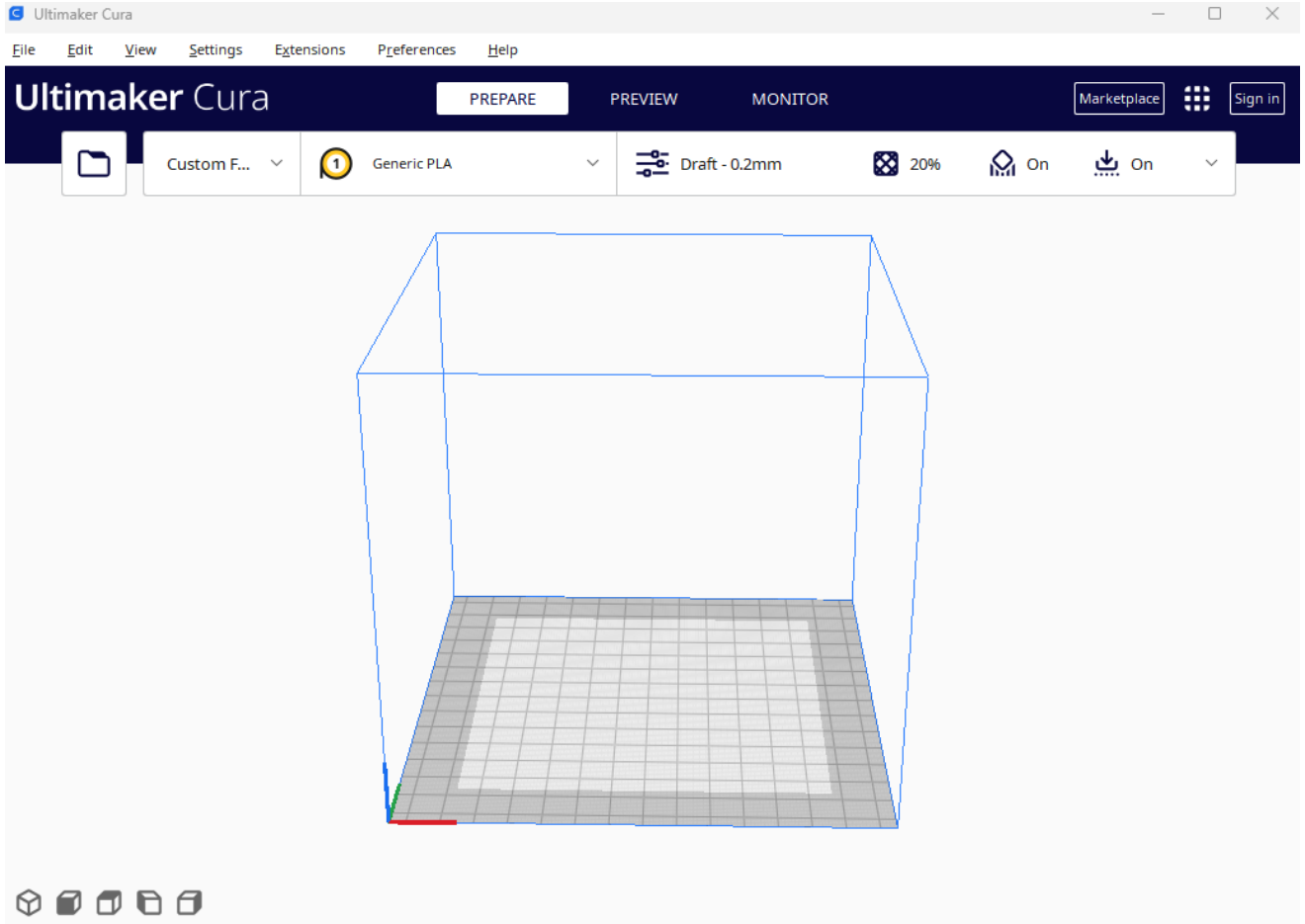
- Once the 3D CAD model is completed, save the file and go to the *File* in *Part studio.*
- Click on Export.
- Choose the file type *.stl* and Export .



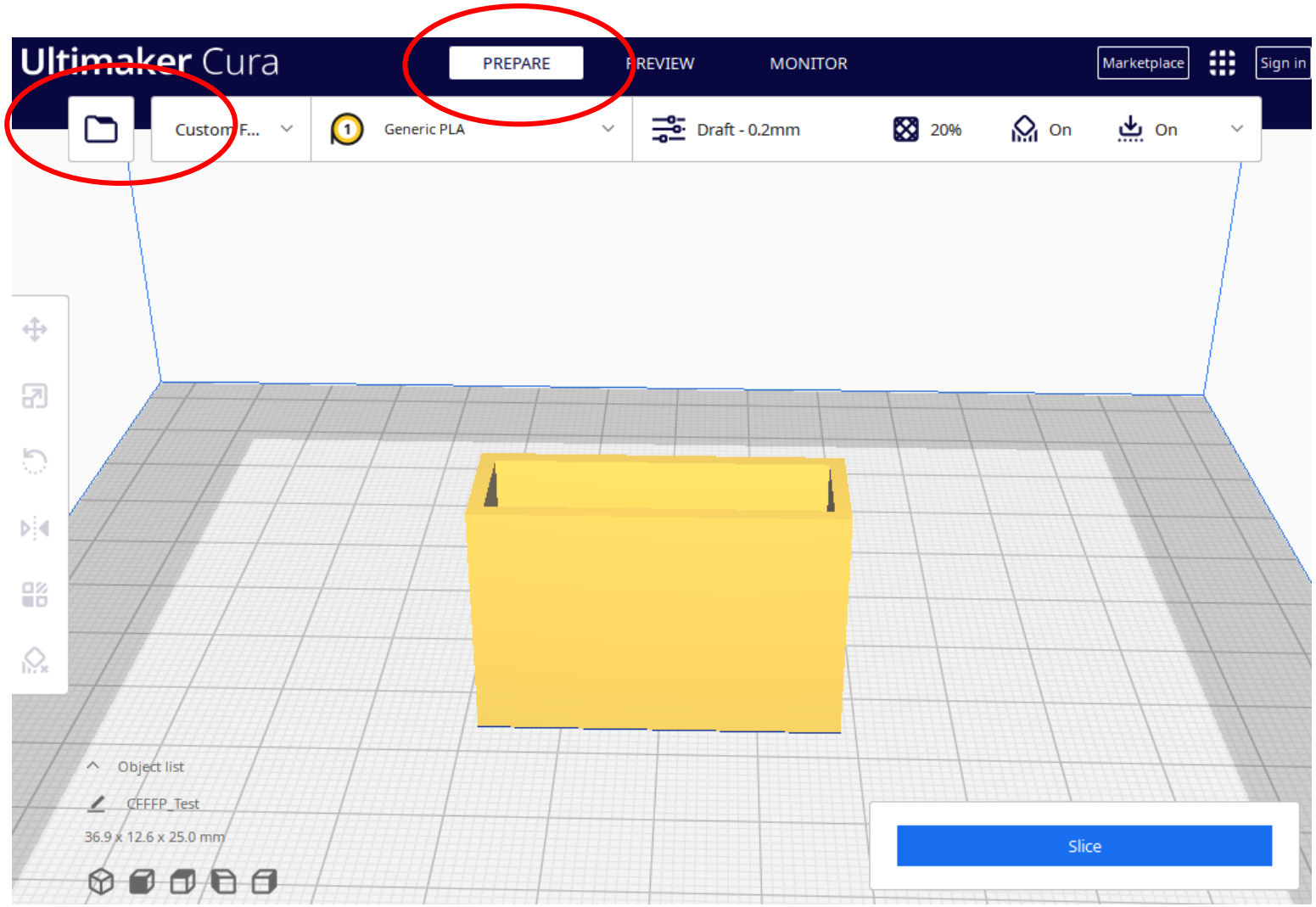


Loading STL file to the slicing software

Open the Ultimaker Cura software.

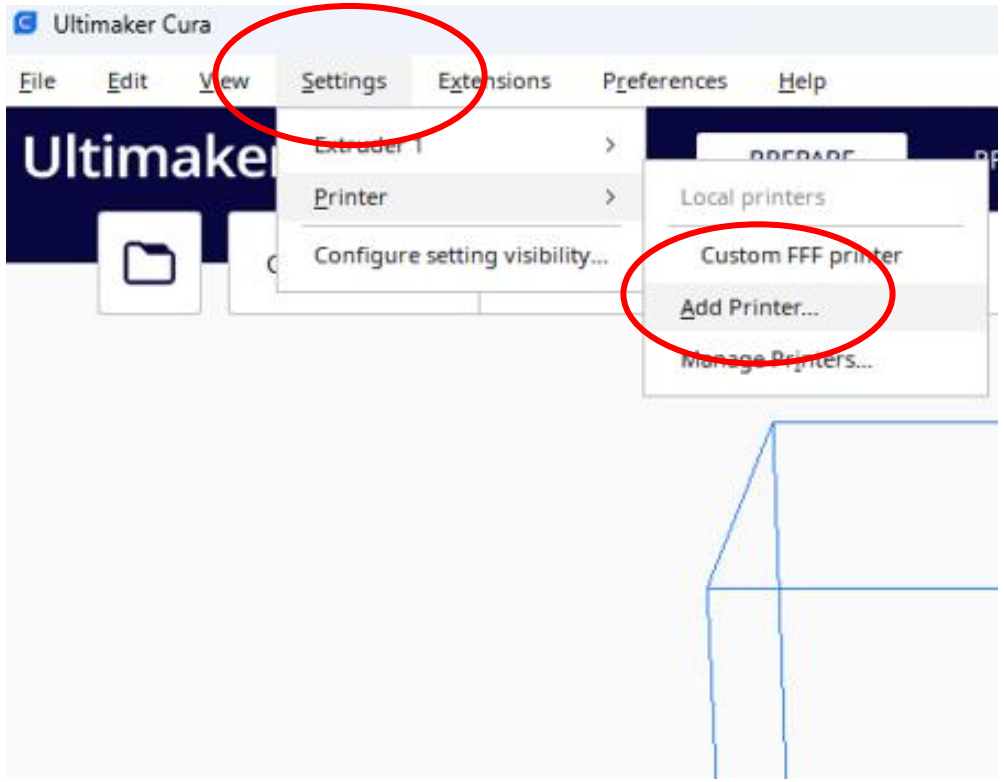


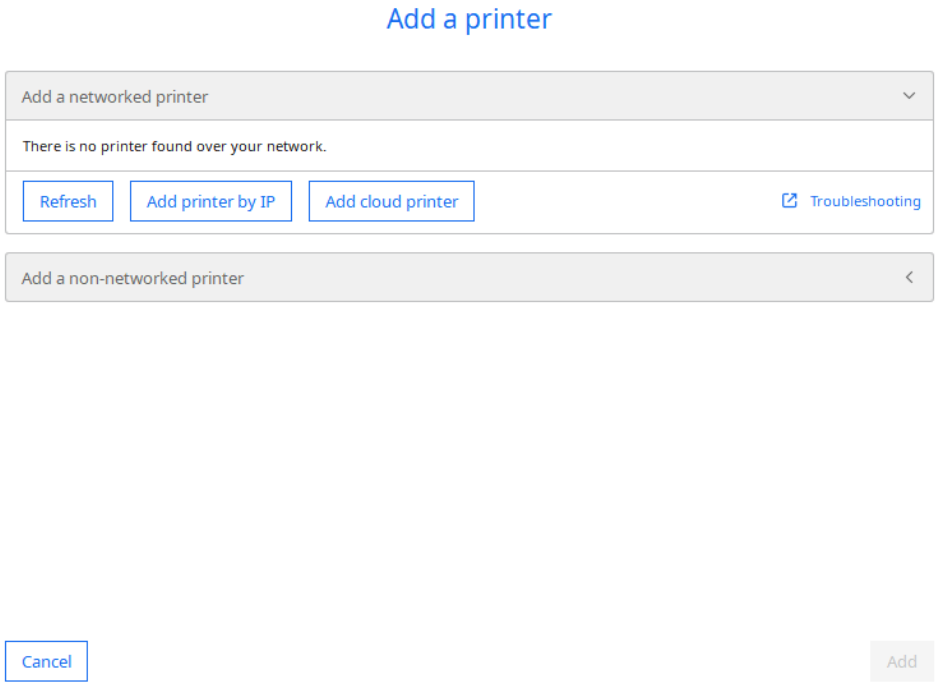
Click on  at PREPARE mode.



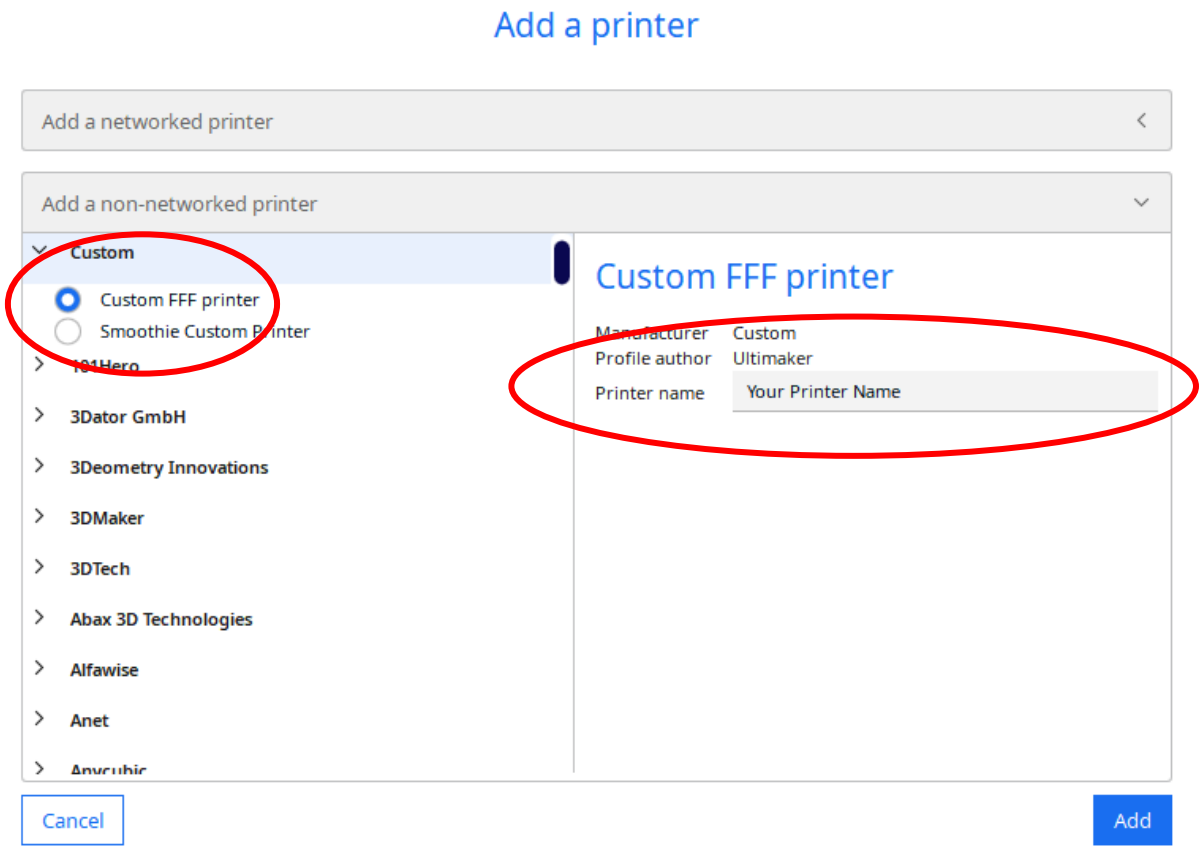
Updating the printer information

Go to Settings > Printer >Add Printer.

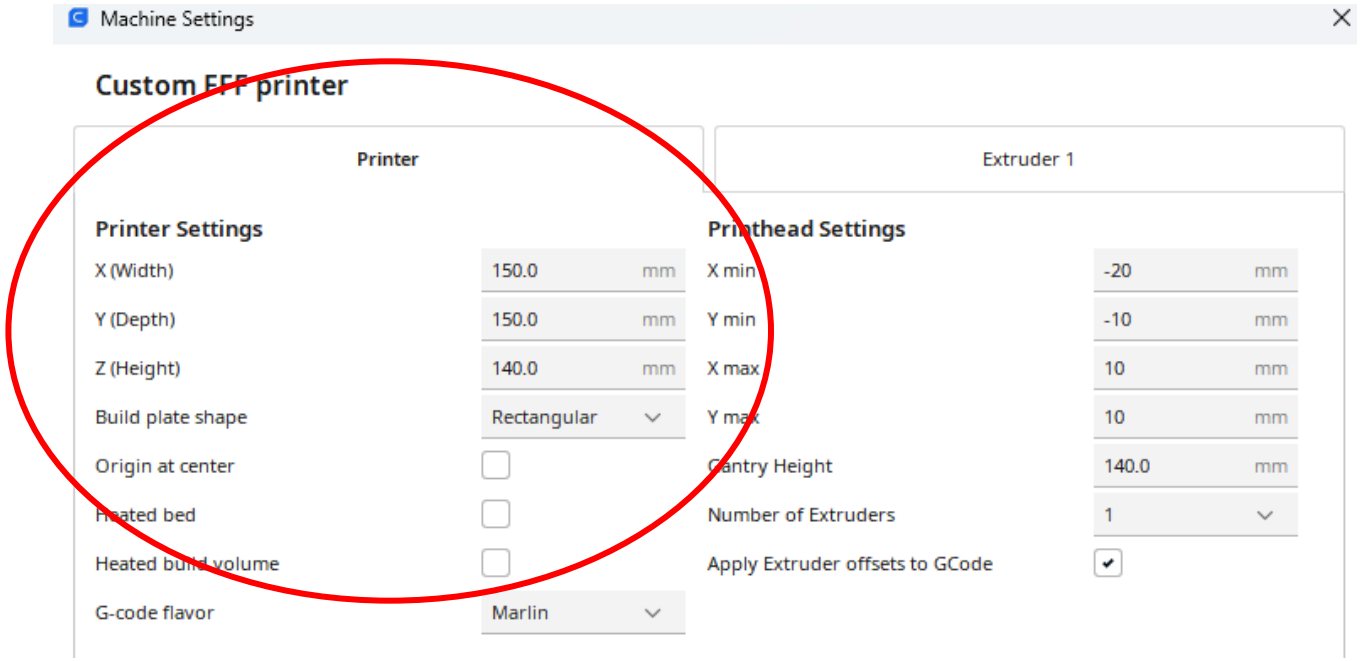




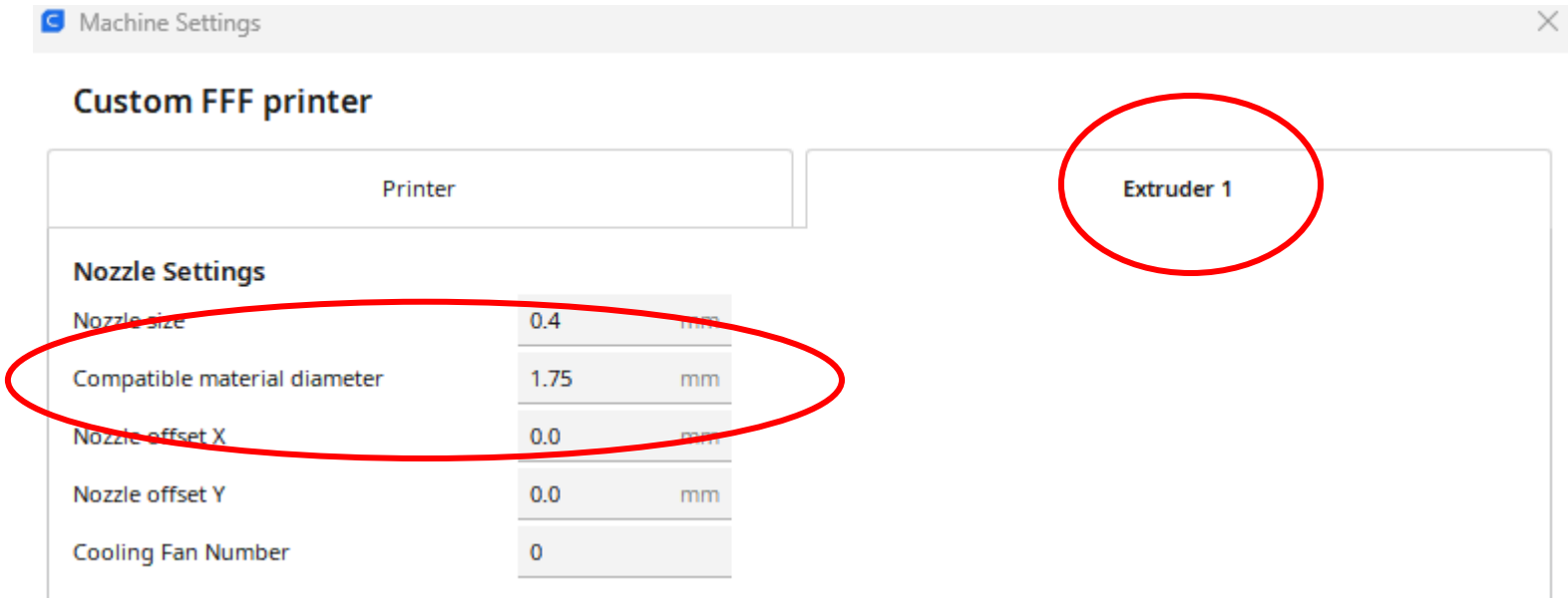
- Pop up the Add a non-network Printer.
- Select the Custom FFF Printer.
- Specify the 3D printer name to the Printer name.



- Update the Width (X), Y(Depth), Height (Z) according to your printer.
- For current 3D printer
  - Width= 150 mm,
  - Depth= 150 mm,
  - Height = 140 mm.

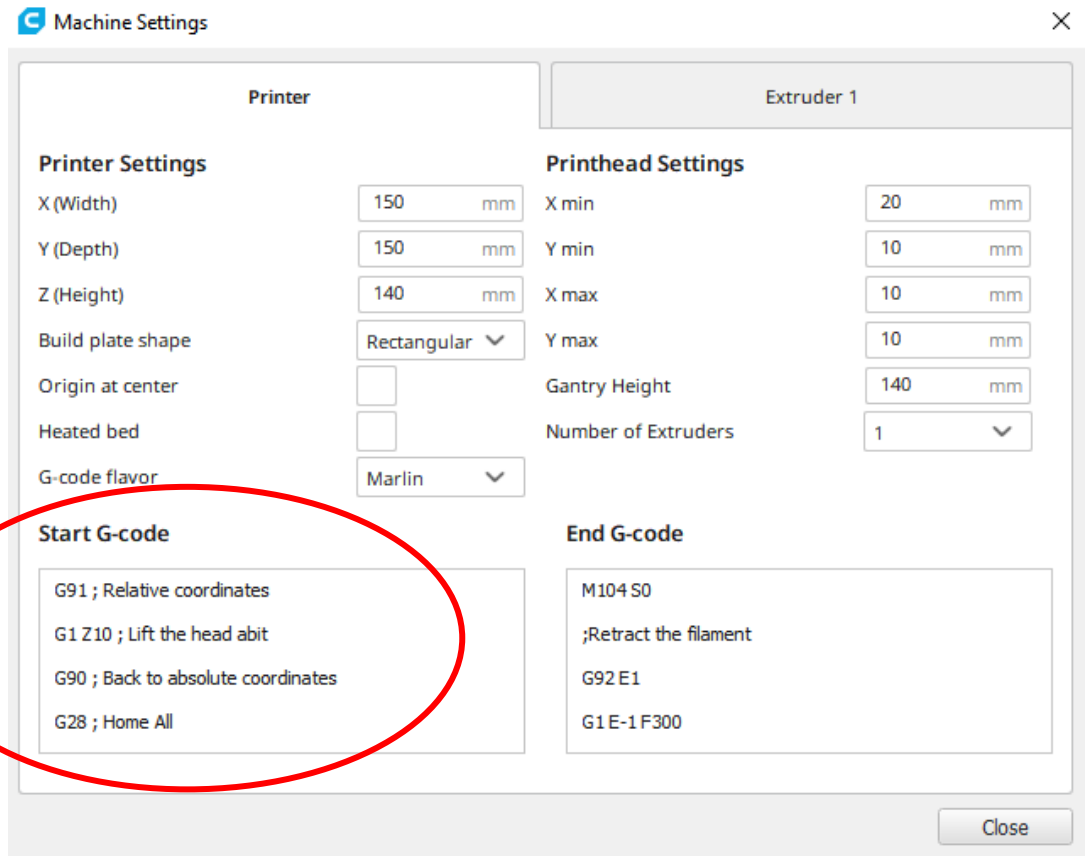


- Go to Extruder 1 under the same Machine settings.
- Update the Compatible material diameter according to the available filament.
- The lab-provided PLA filament diameter is 1.75 mm



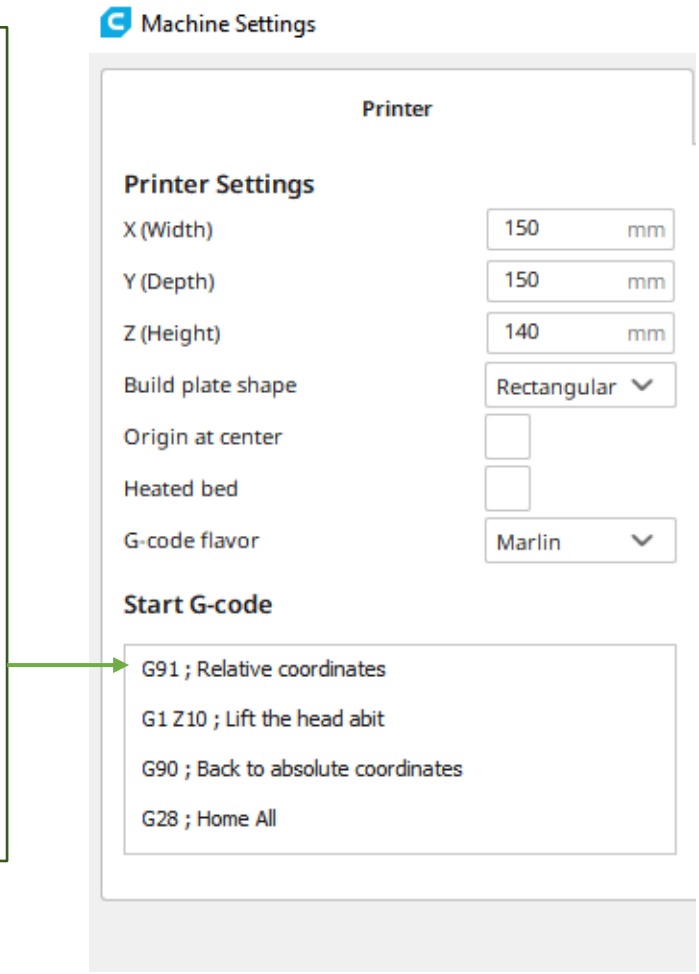
Updating Start G-Code (For Lab Provided 3D Printers)

Settings > Printers > Manage Printers > Your Printer Name > Machine Settings > Start G-Code



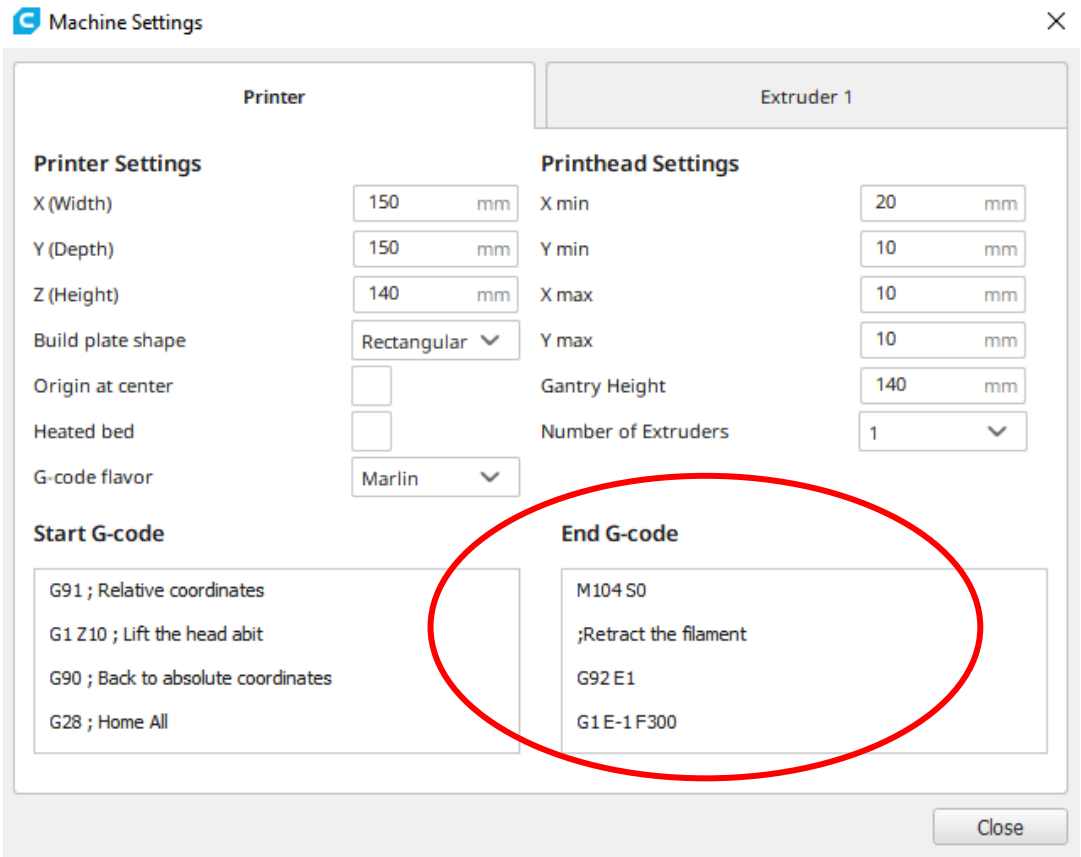
Copy the following G-Code & Paste it to the Start G-Code Section

```
M104 S{material_print_temperature_layer_0};; Begin Heating
G91 ; Relative coordinates
G1 Z10 ; Lift the head abit
G90 ; Back to absolute coordinates
G28 ; Home All
G1 Z10 F500; Dont want to burn the bed while heating
M109 S{material_print_temperature_layer_0}; Wait for pre-heat
; Prime the extruder
G92 E0; Set Extruder position to be 0
G1 E5 F100 ; Extrude a bit in the air
G1 X0 Y5 Z0.1 F500; Touch Down
G92 E0; Set Extruder position to be 0
G1 X100 Z0.3 E10 F500; Nozzle wipe
G92 E0; Set Extruder position to be 0
```



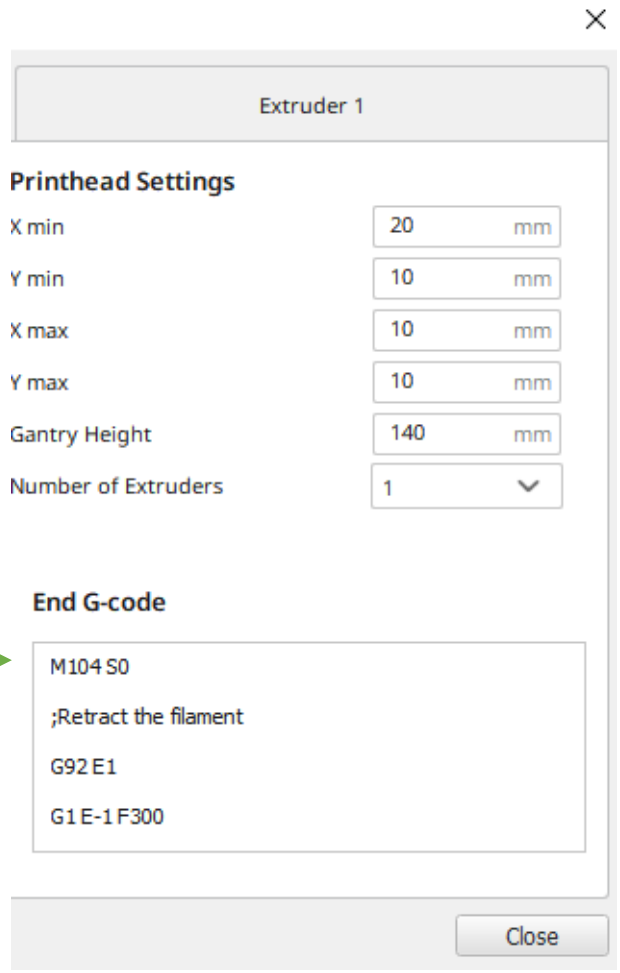
Updating End G-Code (For Lab Provided 3D Printers)

Settings > Printers > Manage Printers > Your Printer Name > Machine Settings > End G-Code



Copy the following G-Code & Paste it to the End G-Code Section

```
M104 S0
;Retract the filament
G92 E1
G1 E-1 F300
G1 X0 Y140
G1 Y150 F4000
M18 X Y E
M18 S36000
```



Click on Slice at the right bottom corner to finish the slicing and start 3D printing.