



Introduction

3D Printing

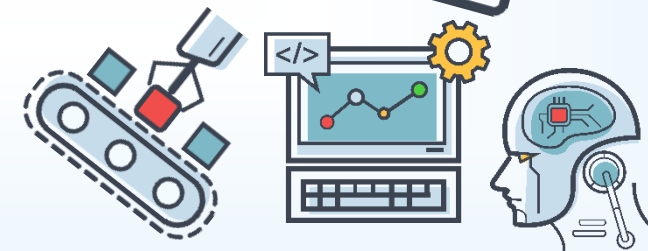
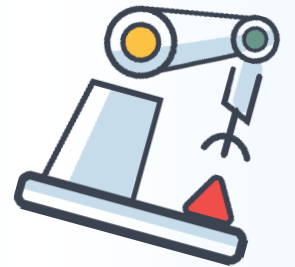
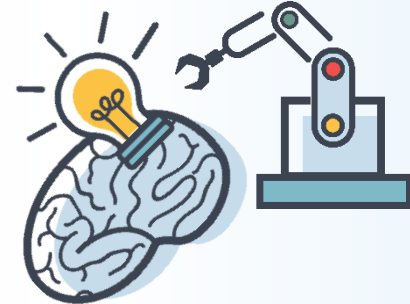


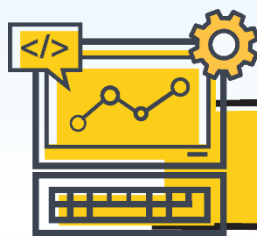


Activity

Let's make 3D object

- 1 Give blank Thermocol sheet to a group of participants
- 2 Ask them to cut the three sets of 2 different shapes from it. (Circle & Rectangle).
- 3 Ask them to paste the same shape on each other;
- 4 Inform them to write the Q1 in the Worksheet.
- 5 Ask them to color the objects (Optional)





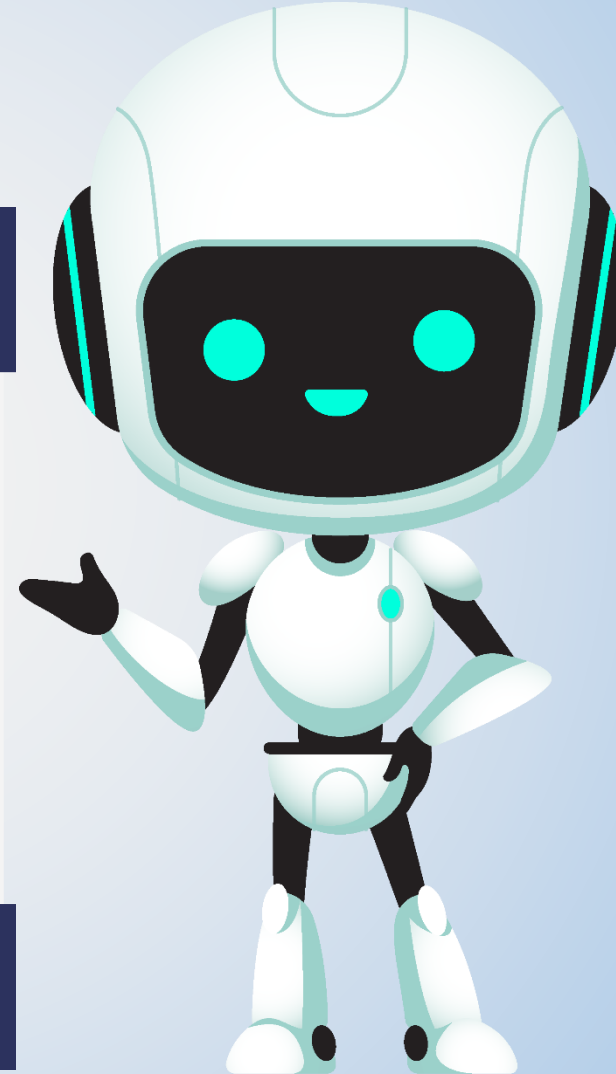
ANALYSIS

3D printer uses the same technique. When the circles were placed over one another it formed a Cylinder similarly when a rectangles were placed on top of one another it formed a cuboid. This process of adding layer by layer is known as “additive manufacturing process, which is used by all the 3D printers.



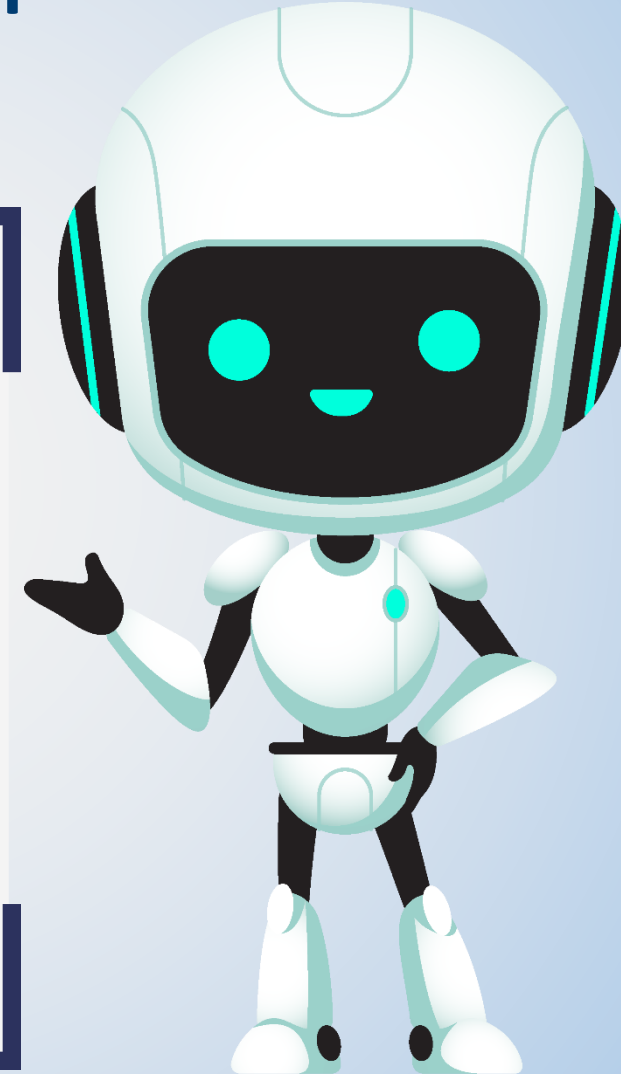


What is 3D Printing





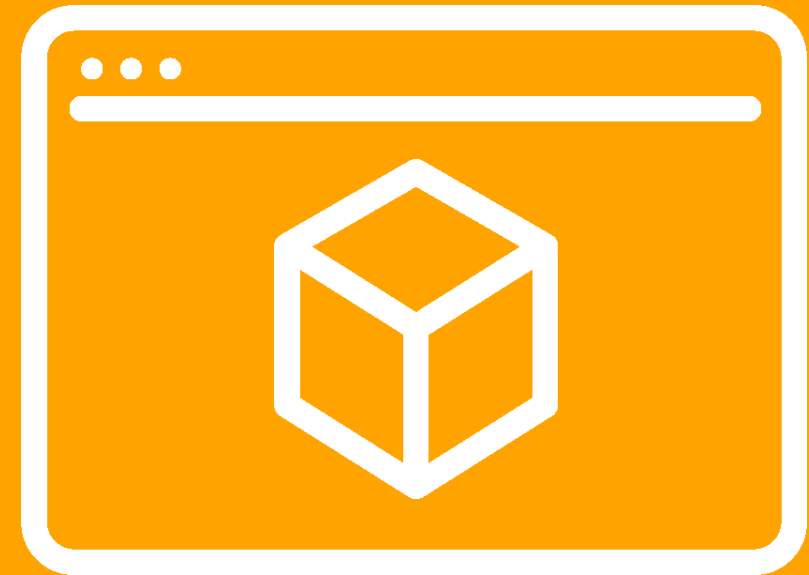
Process involves to develop a 3D model





3D PRINTING

The 3D printing process builds a three-dimensional object using a computer-aided design (CAD) model, usually by successively adding material layer by layer, which is why it is also called additive manufacturing.



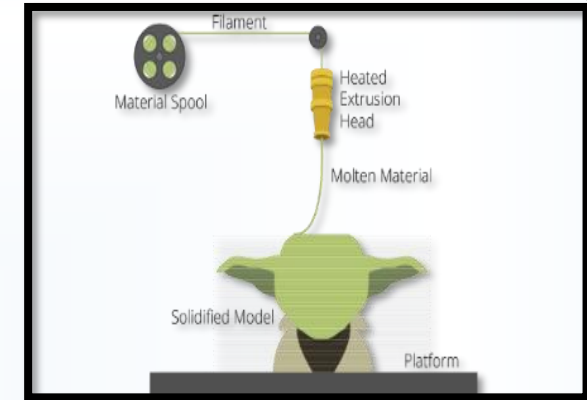
Father of 3D printing: Chuck Hull

While all 3D printers create objects using additive methods (the opposite of a CNC machine)

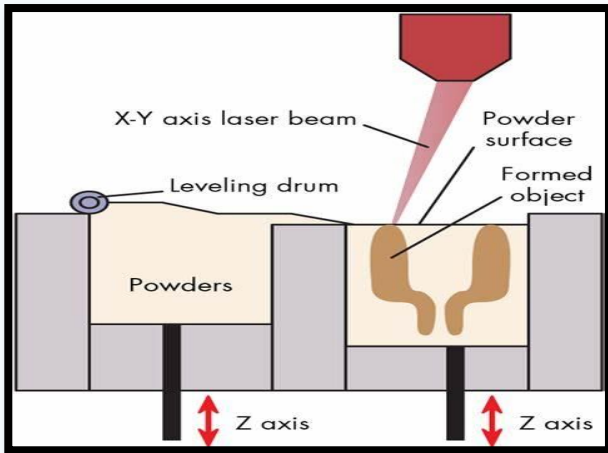


Method of 3D printing

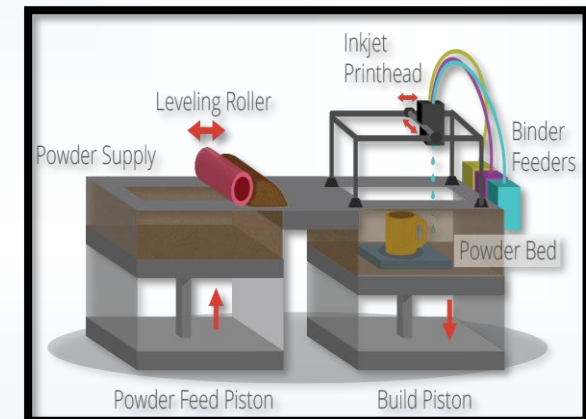
1. Fused Deposition Modelling- A common approach in which the part is printed by extruding molten stings of material that melt together to create the part.



2. Selective Laser Sintering -More common in industrial style prototyping settings, a laser melts together powdered plastic, ceramic, or other material, then spreads more powder on top, repeating the process to build the part layer by layer.

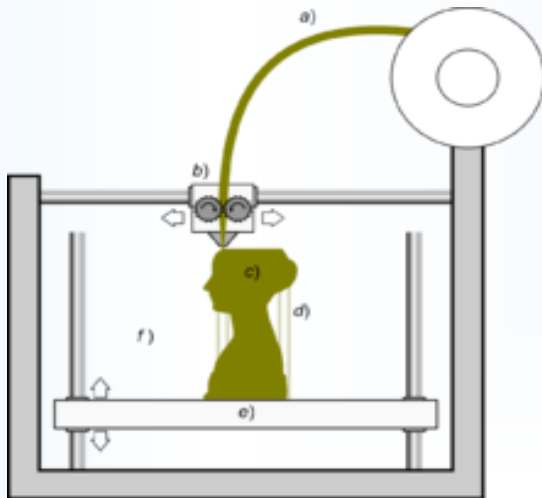
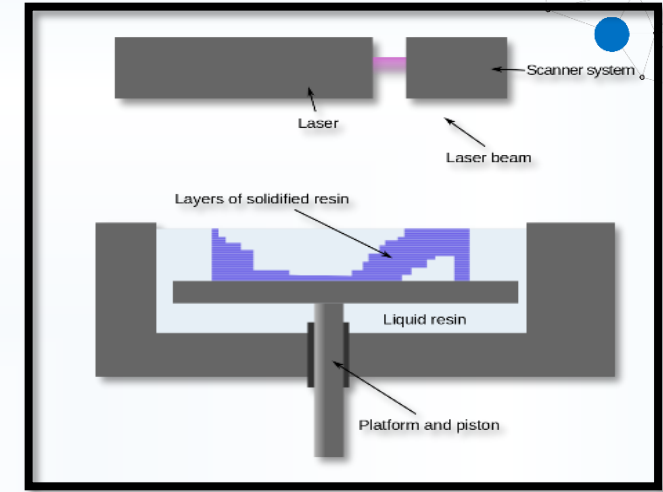


3. Powder Bed and Binder- Similar to Laser Sintering, an inkjet head distributes binder to the correct location on a bed of powder.



Method of 3D printing

4. Stereolithography- Here, an ultraviolet laser hardens resin in a vat layer by layer until the part is built. It can quickly create high definition parts that can be machined.



In our lab, we have an FDM (Fused Deposition Modelling) 3D Printer.

Let's Have a look at it

a) A filament of plastic material is fed through a heated moving head b) that melts and extrudes it depositing it, layer after layer, in the desired shape c). A moving platform e) lowers after each layer is deposited. For this kind of technology additional vertical support d), structures are needed to sustain overhanging parts.



Worksheet Time



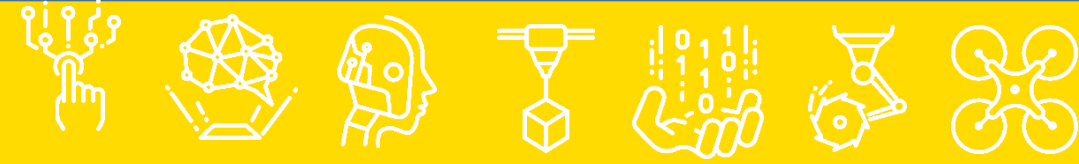


We can print five different colors from a single nozzle at once without changing the filament.



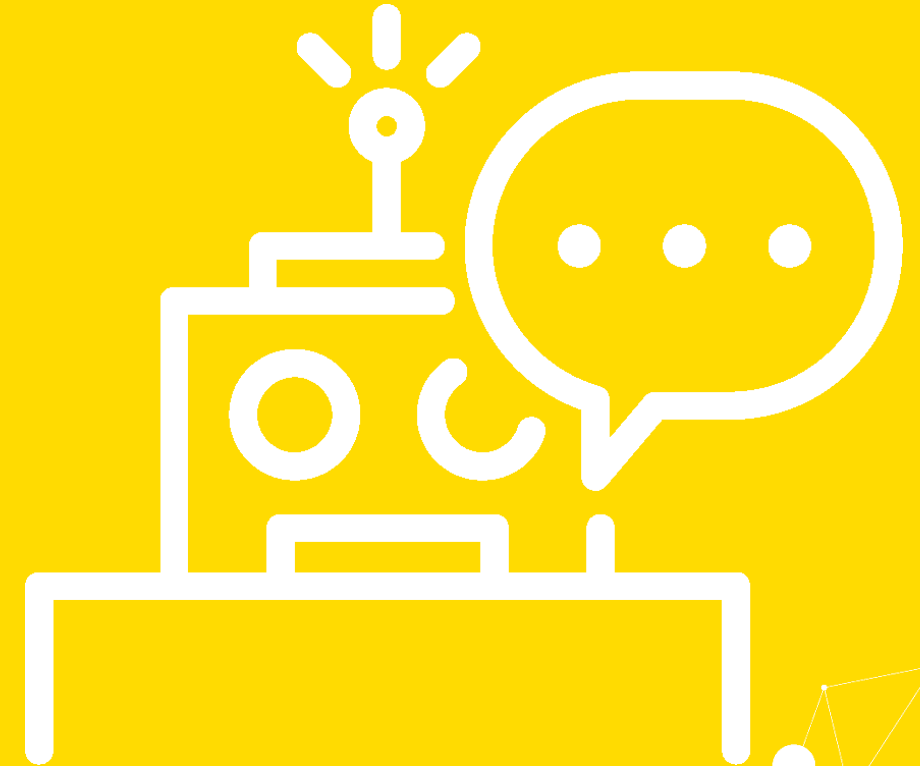
We can convert our 3D printer into a interactive printer using Octa-print.





3D Printing

3D Modelling



Open source CAD software

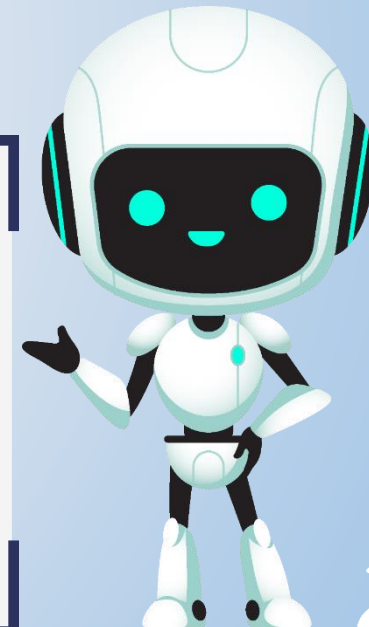
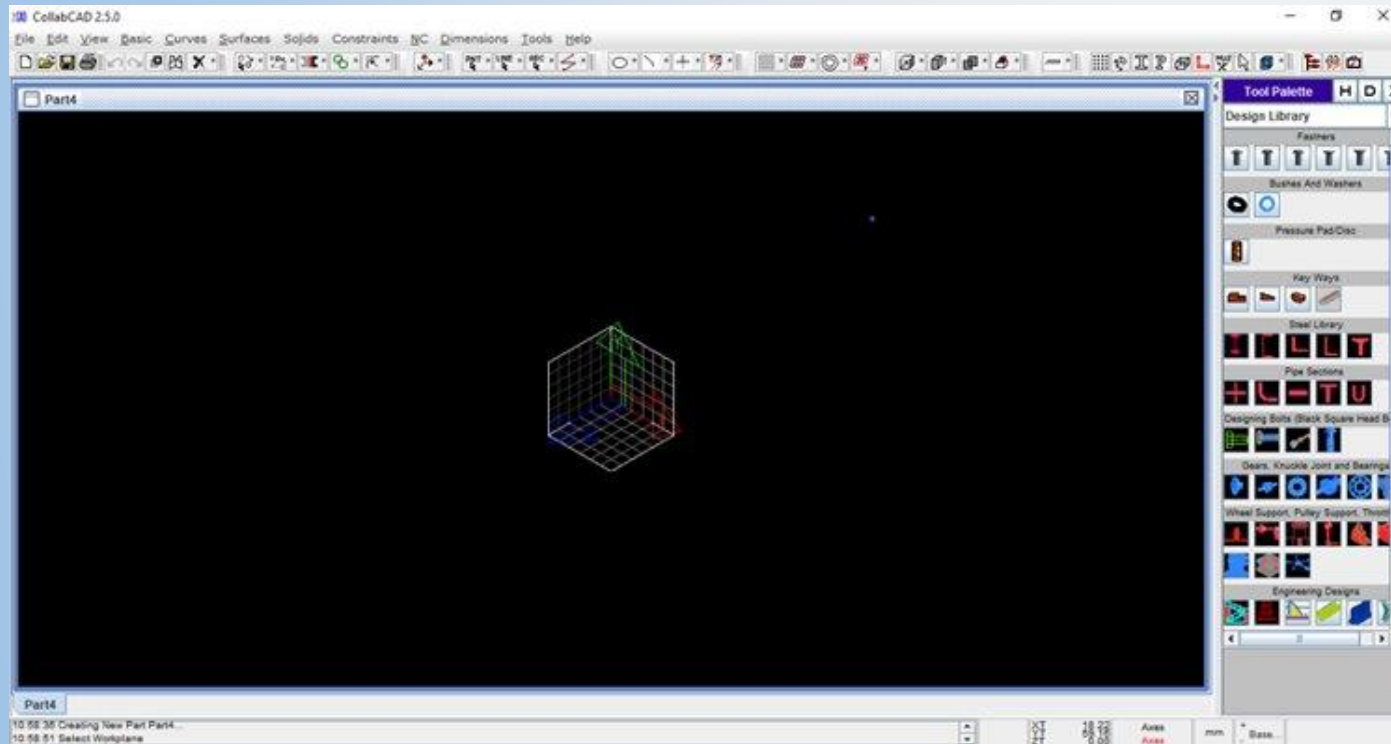
The 3D modelling process builds a three-dimensional object using a computer-aided design (CAD) Software.

- | | | | |
|---|------------|---|-----------|
| 1 | Tinker CAD | 6 | Sculptris |
| 2 | CollabCAD | 7 | FreeCAD |
| 3 | Blender | 8 | 123D |
| 4 | Sketch-UP | 9 | Makehuman |
| 5 | 3D Builder | | |

- It is a collaborative network, computer enabled software system, providing a total engineering solution from 2D drafting & detailing to 3D product design.
- The aim of this initiative is to provide a great platform to students of Atal Tinkering Labs (ATLs) across country to create and modify 3d designs with free flow of creativity and imagination.
- CollabCAD provides facilities to build and edit entities in virtual 3D space using basic geometry such as lines, arcs, and conic sections. 2D profile and free hand sketching is also available. In addition there is detailed dimensioning, tolerances, symbols etc.
- Besides the conventional CAD/CAM features, CollabCAD also provides a collaborative framework for CAD/CAM professionals to work across a network and concurrently access the same design for viewing and modification.

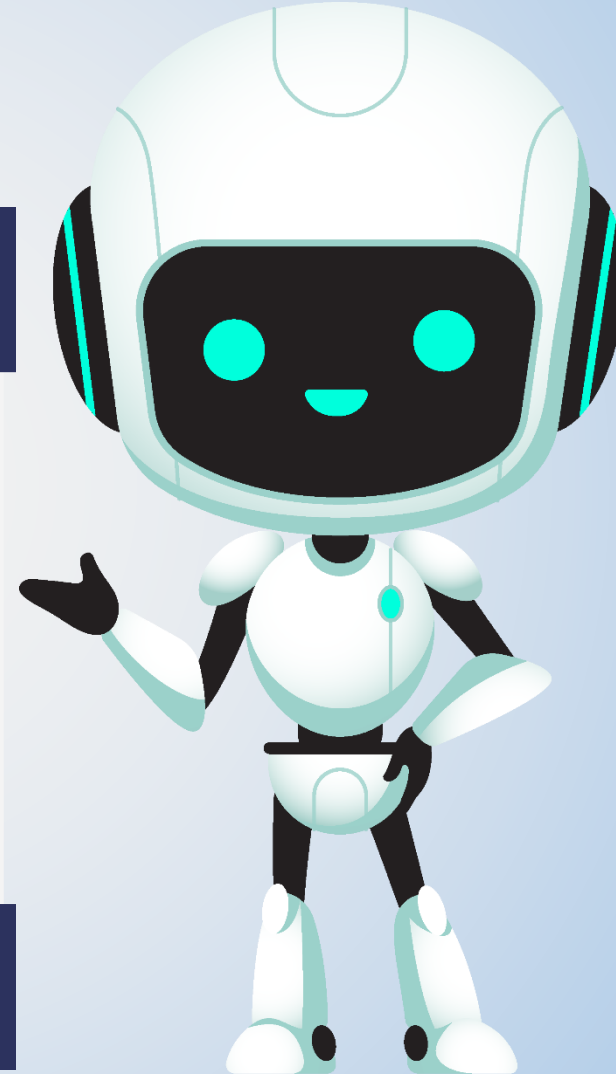


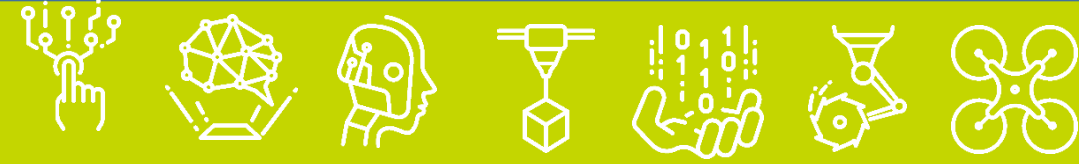
Download and Install CollabCAD





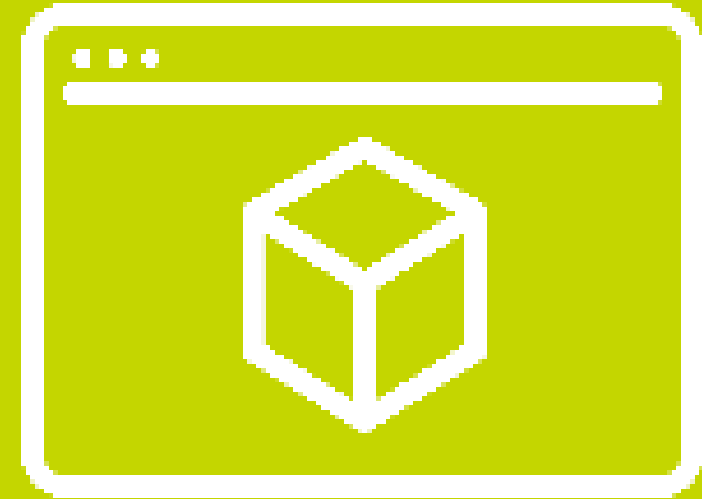
3D Modelling Software – TinkerCAD



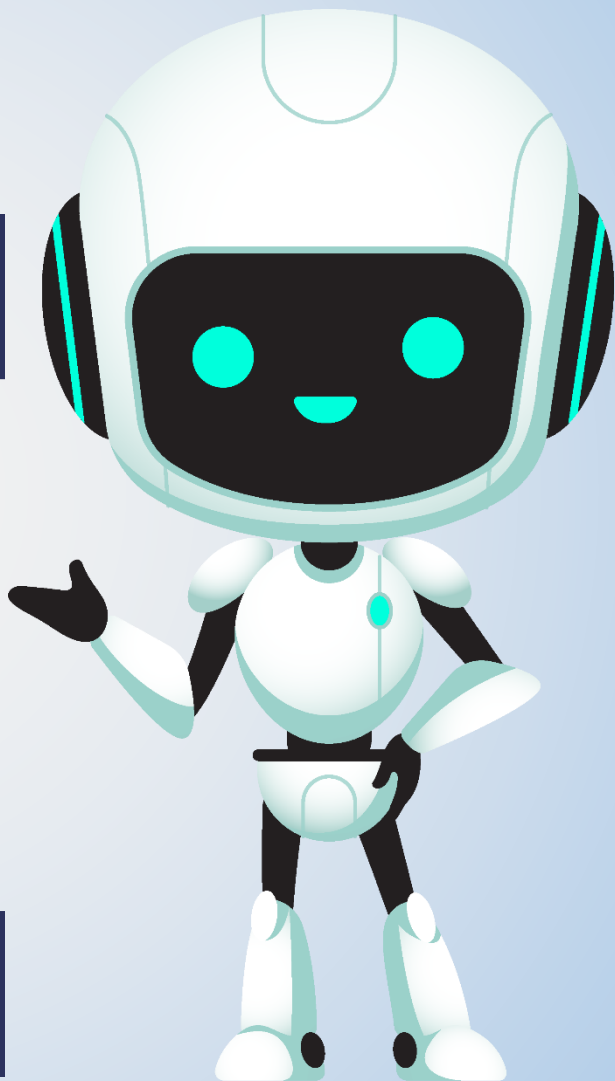


3D Printing

3D Slicing



3D Slicing Software



Definition

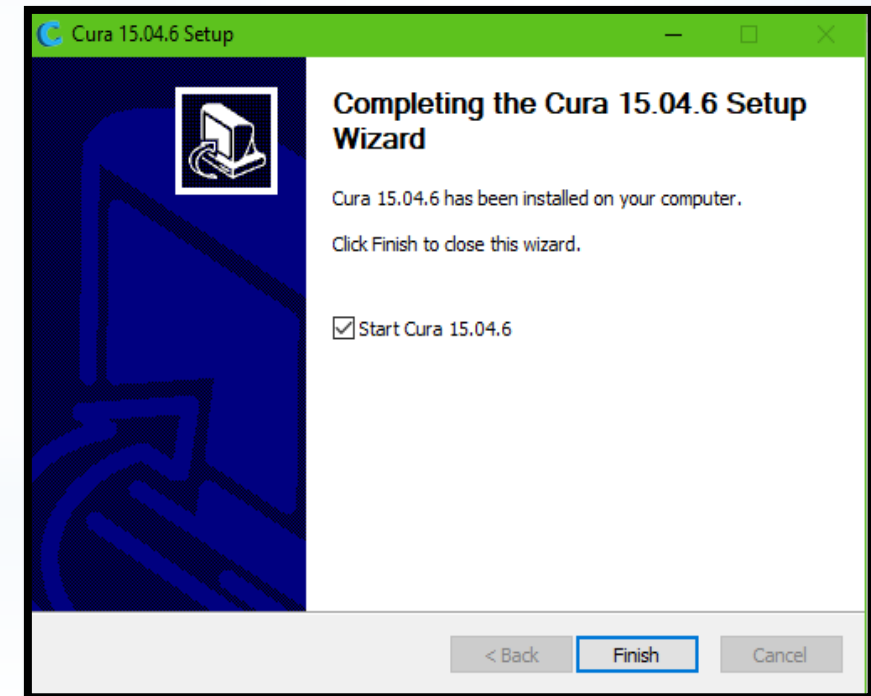
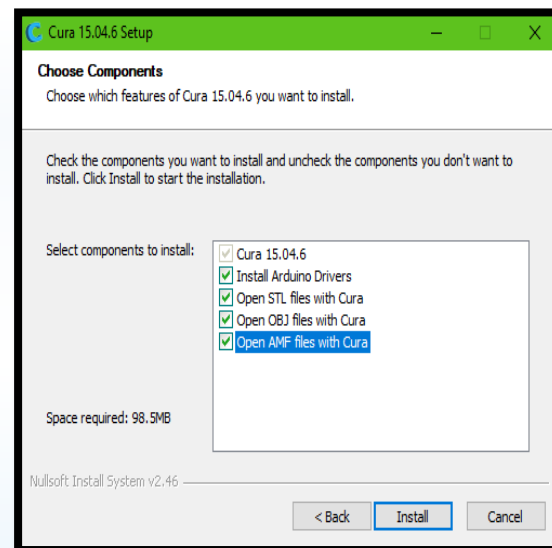
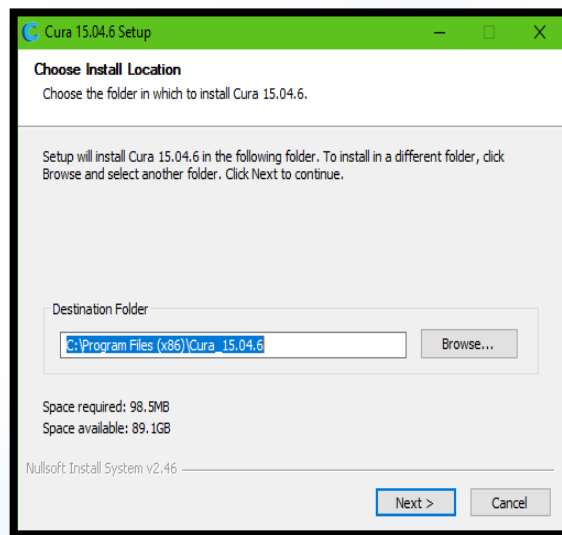
The slicer, also called **slicing software**, is computer software used in the majority of **3D printing processes** for the conversion of a 3D object model to specific instructions for the printer.

In previous lesson we have learnt that the model is saved in “.stl” format , CURA software converts it into G-code so that the printer can understand the file. We will use **Ultimaker CURA 15.04.6**

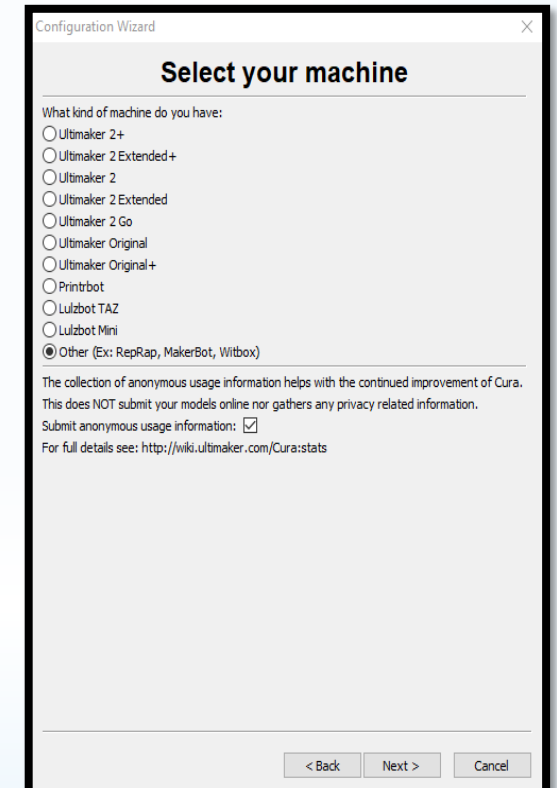
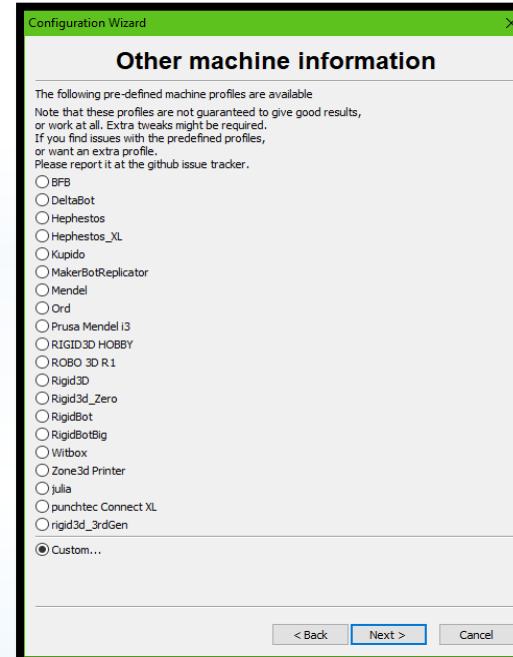
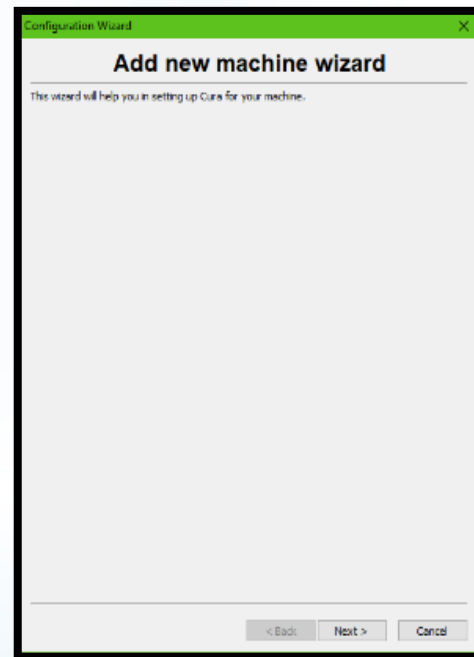
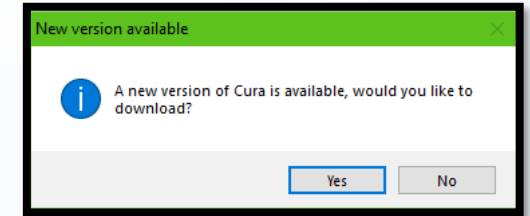
Installation Process:

Please follow these steps to Install CURA on your PC.

1. Install the CURA & you will see a screen as shown in figure Click on Next.
2. On the second Dialog, please select all the Components to install
3. Then Click on Next until you get a screen as shown in the figure. Now Click on **Finish**.



4. You will see a dialog box asking you to download a new version, Click on NO, otherwise Cura will be updated to the latest version.
5. Now you will see a wizard, labelled as the First time run Wizard or Add your Machine. Click on **Next**
6. Now it's an important step to select your machine or you can say to define your machine to adjust its settings according to it.
7. Now select your machine from the list if your machine is not listed then please Choose **Other** & click on Next.
8. Now again you will see a few more machines if still your machine is not listed then choose **Custom** & click Next now you will have a screen shown in the image.





9. This is a very Critical Step, please ensure you have proper information labelled in the User manual or provide by the manufacturer.

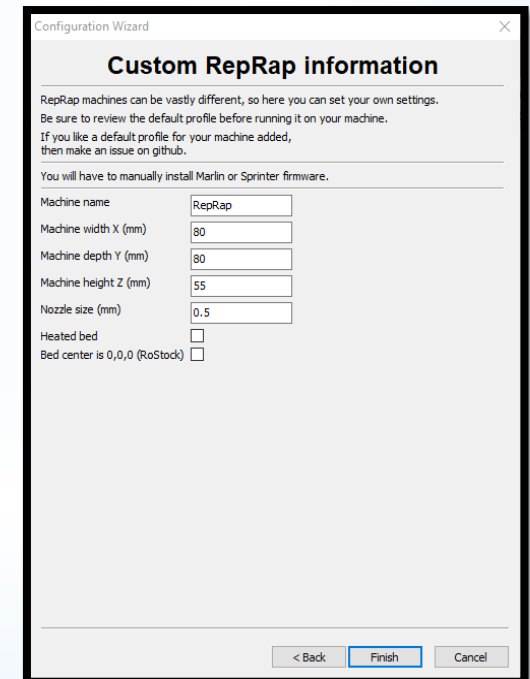
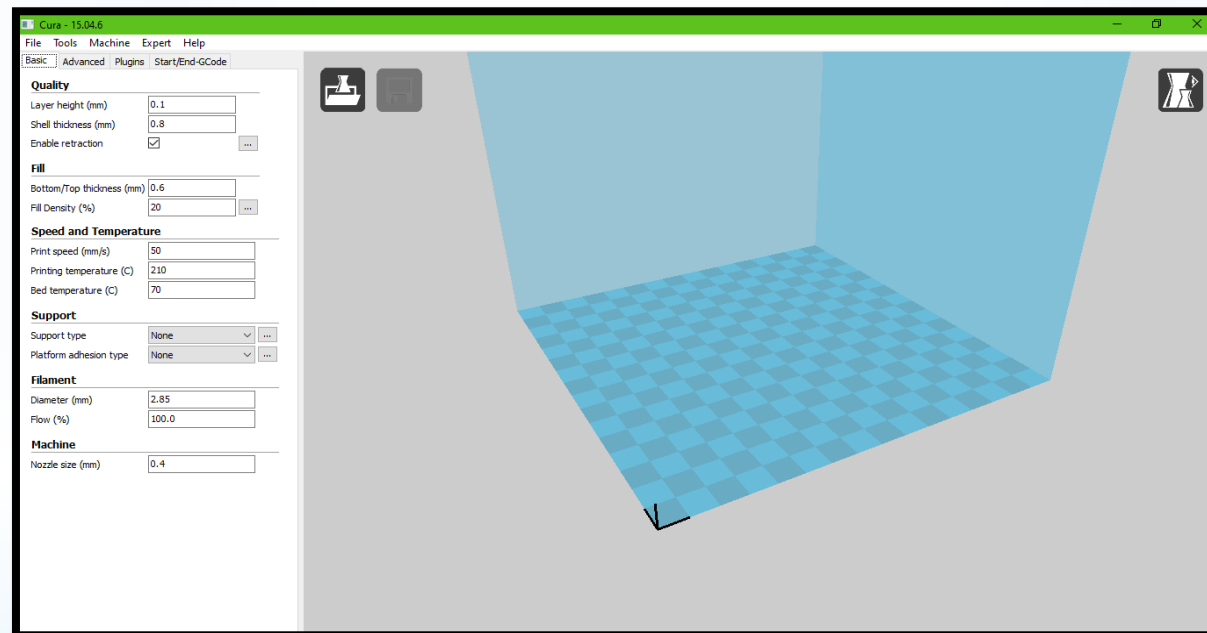
10. Write the Machine Name, then Machine dimensions provided by the manufacturer.

11. Write the nozzle size label on the user manual.

12. Check for heated bed if it's available then check the box. Don't check the Bed center option.

13. Click on Finish.

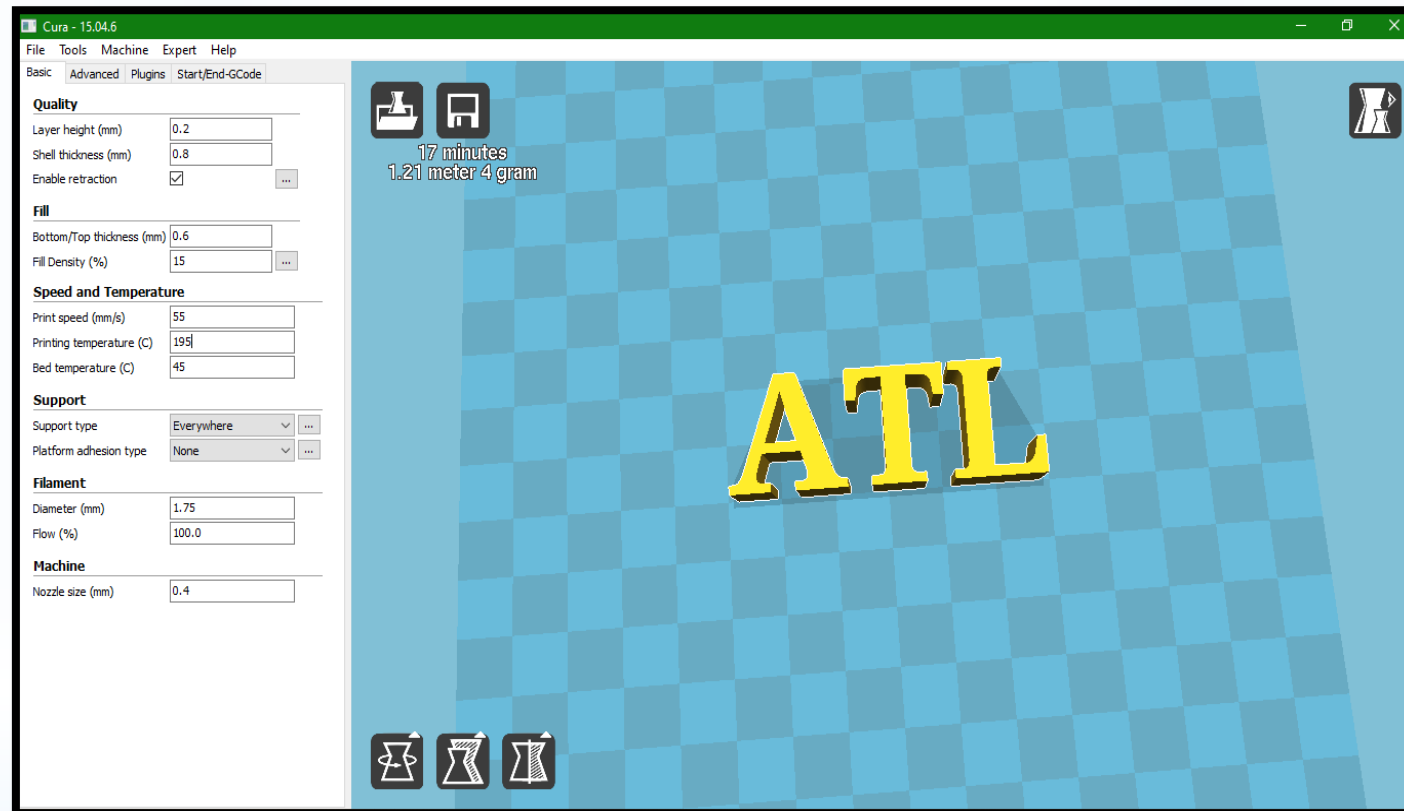
Now you will see a Screen with Some controls that will help in deciding the quality of the product. Before we start further look at Filament diameter change it to 1.75mm





Loading a model in CURA

1. You have already designed a model in tinker cad now we need to load that model in CURA.
2. Click on **file** then select **load model from file** then browse or locate your file that you saved in .stl file in your modelling software.



Basic Set up

01

Layer height: The height of each layer is the crucial parameter that determines the quality of the print and time.

- a. Setting range is 0.05-0.3, High quality setting is 0.1, and the medium quality setting is 0.2.

02

Shell Thickness: The thickness of the outer wall of the side of the model is set to an integral multiple of the diameter of the nozzle. If the nozzle is 0.4mm, the recommended wall thickness is 1.2.

03

Enable Retraction:
Recommended

04

Bottom /Top Thickness: This is generally an integral multiple of the layer thickness. It is recommended to be the same as the wall thickness and set to 1.2.

05

Fill Density: The filling density inside the model can be set to 10%~100% according to the actual situation. The larger the value, the denser the interior.

06

Print Speed: The moving speed of the print head during printing. Set it to low speed for complicated parts & high speed for simple elements. Generally set around 30-50, suggested PLA setting 50, ABS setting 30.

07

Printing Temperature: the temperature used by the nozzle to melt the consumables. The melting temperature is varied based on the material. It is recommended to set the PLA to 190 degrees and the ABS setting to 230.

08

Bed Temperature: Platform temperature, recommended PLA setting 50 °C, ABS setting 80-110 °C.

09

Support Type: support mode when printing a model with a suspended part,
1. "None" - no support,
2. "Touching buildplate" - creating support that can reach the workbench part,
3. "Everywhere" - create support for all parts which are suspended.
(Recommended)

Basic Set up

10

Platform Adhesion Type: None defaults to print one layer at a minimum, Brim refers to a skirt, and Raft refers to a bottom plate. It is recommended to select Raft, the PLA first layer Airgap is set to 0.15, and the ABS first layer Airgap is set to 0.1 or lower.

11

Diameter: The filament supplied is 1.75mm.

12

Flow: Default 100

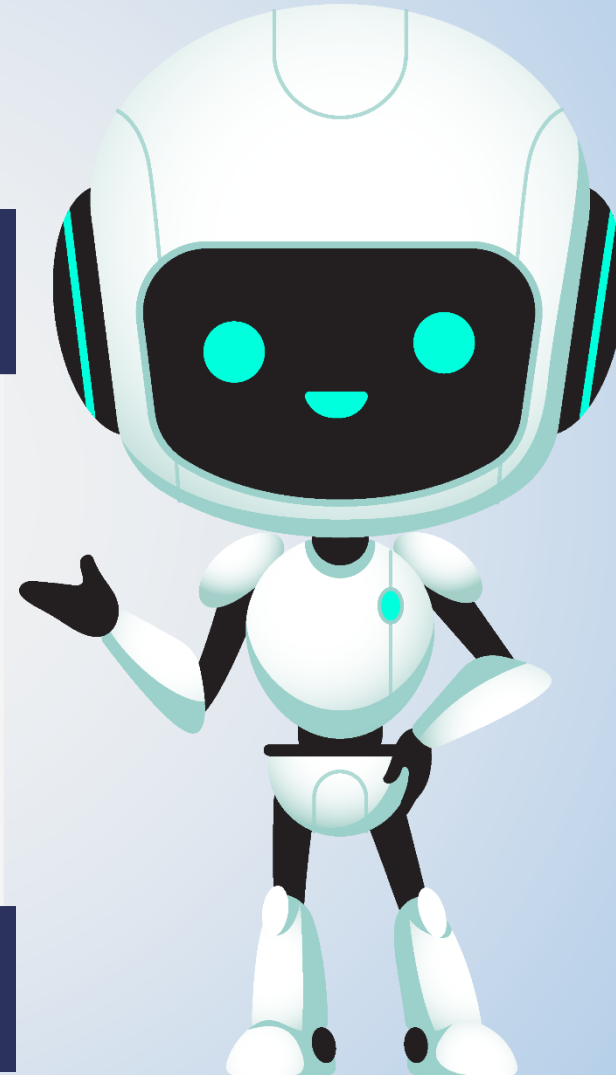
13

Nozzle Size: Nozzle diameter is generally 0.4mm. If you customize other nozzle sizes, fill in here.

- a. **Note** - The nozzle size does not affect the accuracy of the layer thickness and only affects the minimum print detail. The size of the nozzle and the printing speed are squared, and increasing the nozzle size dramatically increases the printing speed.

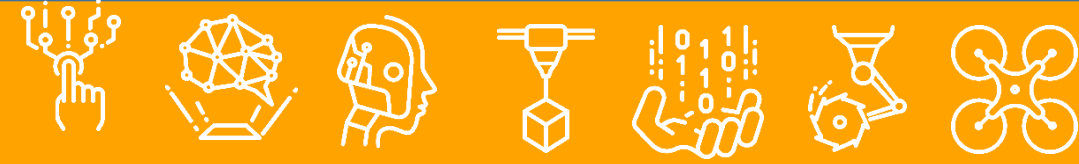


Now insert the SD Card & you will see SD card written on the screen,
now click on that and .gcode will be saved in your SD Card.



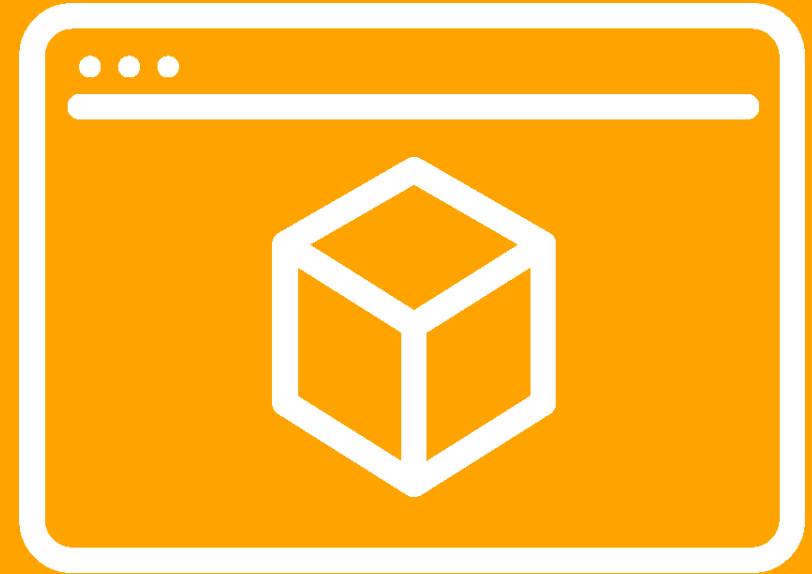
To know more about the function of CURA setting
please download the guide for the Given QR code





3D Printing

3D Troubleshooting





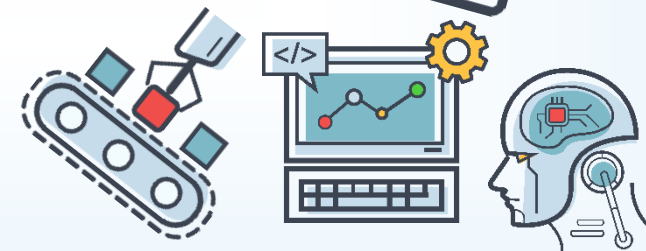
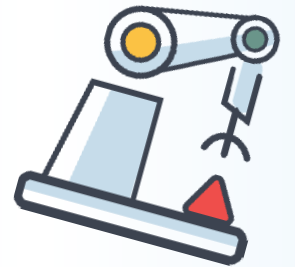
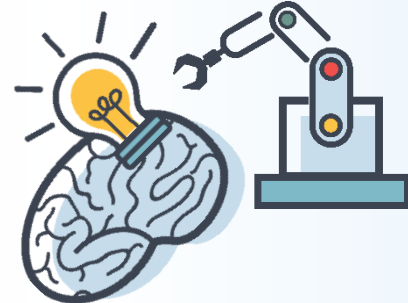
Activity

01

Try to level the gap between the nozzle & bed.

02

Look at automatic levelling and how to change levelling on one side.





ANALYSIS

The 3D printer is an important aspect in rapid prototyping, to Maintain the quality of the object & the long life of the machine we need to take some precaution while printing and some to service our machine.

01

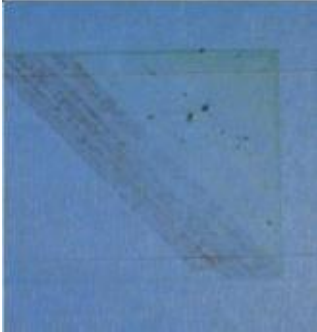


Bed Level – Every Printer has a different mechanism to level. The level word means the gap between the bed & nozzle at 5 Points. (4 Corner of bed & 1 center). The minimum difference between the nozzle and bed should not more or less than a 0.05mm (thickness of A4 size paper).

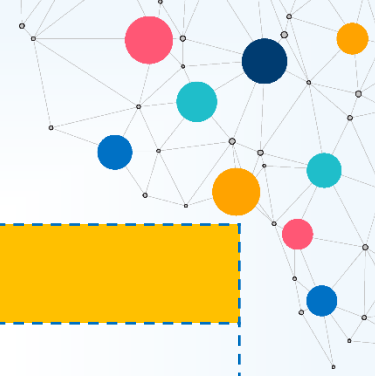
02

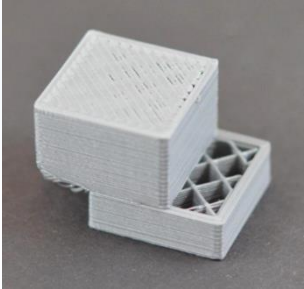
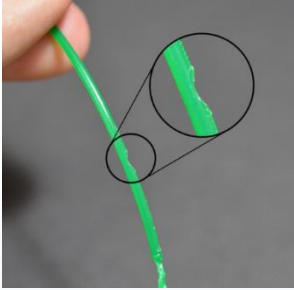

Nozzle Cleaning – This is a significant problem with many 3D printers of nozzle clogging. Go through your 3D Printer Manual for the process of nozzle cleaning for your respective printer.



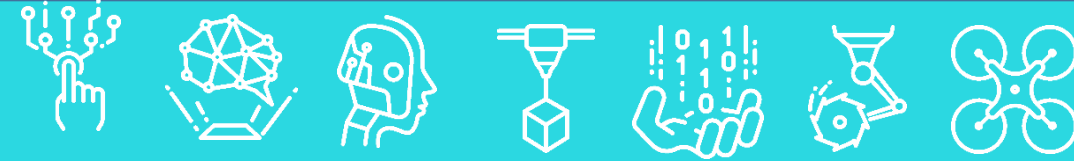
PROBLEM AND ITS CAUSES

S.No	Problem	Image	Major Cause
1	Not Extruding at start of print		<ul style="list-style-type: none"> <input type="checkbox"/> Filament was not loaded into extruder before printing <input type="checkbox"/> The distance between nozzle and bed is too close <input type="checkbox"/> The filament stripped against the drive gear <input type="checkbox"/> The extruder is clogged
2	Print Not Sticking to the Bed		<ul style="list-style-type: none"> <input type="checkbox"/> Build platform is not level <input type="checkbox"/> Nozzle starts too far away from the bed <input type="checkbox"/> First layer is printing too fast <input type="checkbox"/> Temperature or cooling settings <input type="checkbox"/> The build platform (tape, glues and materials) <input type="checkbox"/> When all else fails: Brims and Rafts
3	Snoozing or oozing		<ul style="list-style-type: none"> <input type="checkbox"/> Retraction distance <input type="checkbox"/> Retraction speed <input type="checkbox"/> Temperature is too high



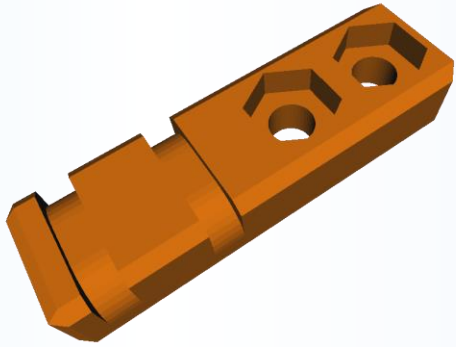
S.No	Problem	Image	Major Cause
4	Layer Shifting or Misalignment		<ul style="list-style-type: none"><input type="checkbox"/> Nozzle moves too fast<input type="checkbox"/> Mechanical or Electrical issues
5	Grinding Filament		<ul style="list-style-type: none"><input type="checkbox"/> Increase the extruder temperature<input type="checkbox"/> Print too fast<input type="checkbox"/> Check for a nozzle clog
6	Clogged Extruder		<ul style="list-style-type: none"><input type="checkbox"/> Manually push the filament to extruder<input type="checkbox"/> Reload the filament<input type="checkbox"/> Clean out the nozzle



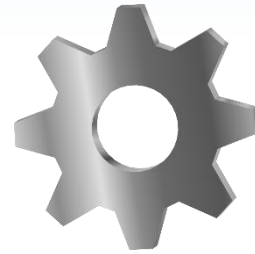


Worksheet Time





With the help of 3D Printing technology, people have already managed to create artificial organs.



There are 3D printers which uses Gold, Silver, Titanium, wood etc for 3D Modelling .



History

The history of 3D printing began in the 1980s.



