IIT HYDERABAD
<u>A BRIEF INTRODUCTION TO 3D PRINTERS</u>

Dhilli Venkata sai
Pilli Venkata Praneeth

Table of contents

1.)<u>Introduction To 3D Printing</u>

3D printing or additive manufacturing is a process of making three dimensional solid objects from a digital file.

The creation of a 3D printed object is achieved using additive processes. In an additive process an object is created by laying down successive layers of material until the object is created. Each of these layers can be seen as a thinly sliced cross-section of the object.

3D printing is the opposite of subtractive manufacturing which is cutting out / hollowing out a piece of metal or plastic with for instance a milling machine.

3D printing enables you to produce complex shapes using less material than traditional manufacturing methods.

Applications of 3D Printing

3D printing encompasses many forms of technologies and materials as 3D printing is being used in almost all industries you could think of. It’s important to see it as a cluster of diverse industries with a myriad of different applications.

A few Applications:

- – consumer products (eyewear, footwear, design, furniture)
- – industrial products (manufacturing tools, prototypes, functional end-use parts)
- – dental products
- – prosthetics
- – architectural scale models & maquettes
- – reconstructing fossils

- – replicating ancient artefacts
- – reconstructing evidence in forensic pathology
- – movie props

Why use 3D Printers for Rapid Prototyping?

In short: it's fast and relatively cheap. From idea, to 3D model to holding a prototype in your hands is a matter of days instead of weeks. Iterations are easier and cheaper to make and you don't need expensive molds or tools.

Besides rapid prototyping, 3D printing is also used for **rapid manufacturing**. Rapid manufacturing is a new method of manufacturing where businesses use 3D printers for short run / small batch custom manufacturing.

i.)Working Principle of 3D printers

All 3D printing techniques are based on the same principle: a 3D printer takes a digital model (as input) and turns it into a physical three-dimensional object by adding material layer by layer.

It is way different than traditional manufacturing processes such as injection molding and CNC machining that uses various cutting tools to construct the desired structure from a solid block. 3D Printing, however, requires no cutting tools: objects are manufactured directly onto the built platform.

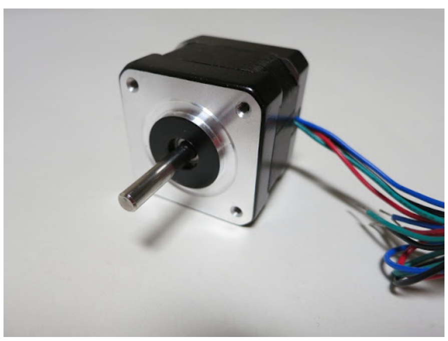
The printer we use in the lab is creality cr 10s pro ,this printer also holds the same working principle like other 3d printers available in the market .It has an extra feature compared to generic printers that is bed and extruder auto leveling using a proximity sensor

According to the gcode (it is the file we get after slicing the design using some softwares like slic3r,cura etc) the printer starts it movements i.e the extruder and bed starts getting heated up to the set temperature or we can manually give the desired temperature inputs using the on printer screen

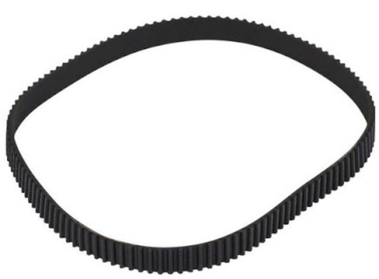
The Stepper motor starts movements according to the gcode, 2 Stepper motos control movement of Z-axis , 1 Stepper motor controls the movement of the bed that is Y-axis and 1 Stepper motor controls the movement of extruder that is X-axis.And the additional stepper motor is used to push the filament to the extruder.

ii.) Parts of the 3D printer(Creality CR 10s pro)

→Stepper Motors x 4



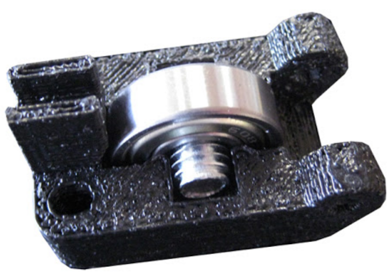
→chain gears x 1



→hobbed gears for stepper motors x 2

→long threaded rods for z-axis movement x2

→idler gear movement of extruder and bed x 6

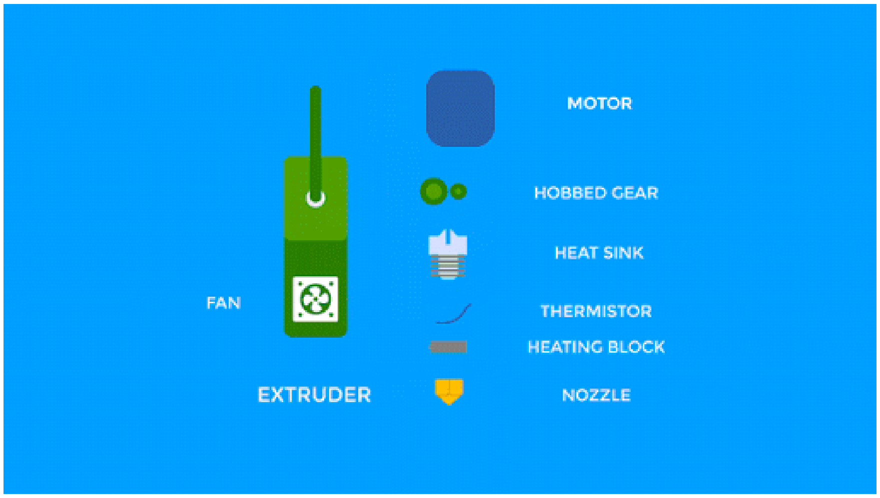


→Heating Bed (31x31cm) with aluminium sheet(3mm)

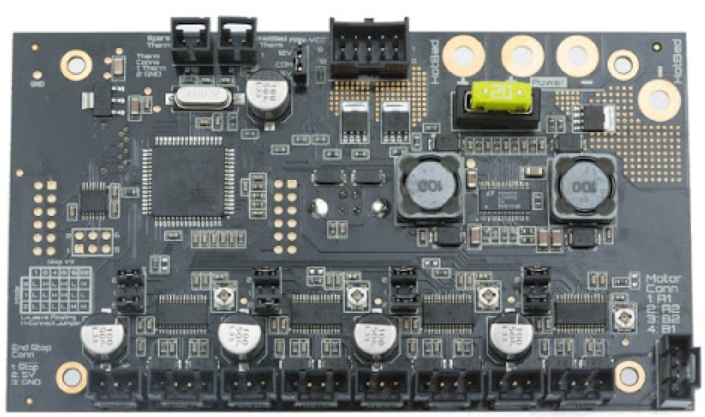
→filament detecting sensor
→lcd display

→ heat sink or heat end fan

→Extruder



→MotherBoard or Controller Board



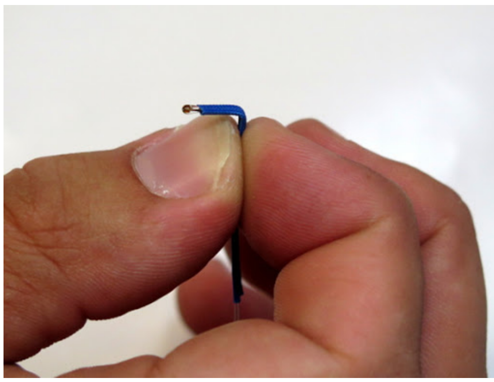
→Heater Cartridge

→SD card slot

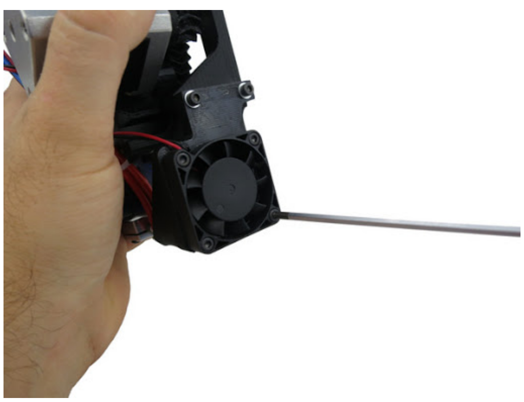
→power supply unit

→End Stops

→The Thermocouple

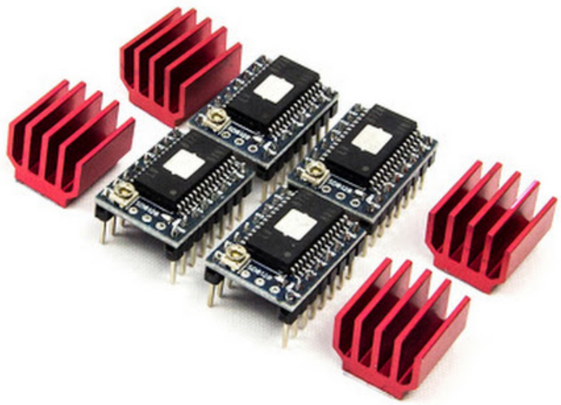


→The Cooling Fan



→Nozzle

→stepper drivers



iii.)Functionality of each part

Print Bed

The print bed is the surface that your objects are printed on to. Typically it will consist of a sheet of glass, a heating element, and some kind of surface on top to help the plastic stick.

Heated/Non-Heated

Most print beds are heated in order to prevent the object from warping while it is being printed. Due to thermal contraction, the plastic will shrink slightly as it cools. This causes the object to warp upwards around the edges and peel off the bed. Heated beds keep the bottom of the object warm, in order to prevent this.

Bed Surfaces

The bed surface helps the plastic stick to the bed during printing but also allows it to be removed easily when printing is done. There are many different kinds of bed surfaces. Most printers will come with some kind of all purpose surface, like BuildTak or PEI film. However, for best results you will want to use different surfaces depending on the material you are printing.

Filament

This is the plastic that's consumed by the printer. It comes on a spool. Printers use two different sizes of filament, 1.75 mm and 3 mm. There are a variety of different materials.

Extruder

The extruder is the core of the printer. It is where the plastic gets drawn in, melted, and pushed out. It is essentially a fancy hot glue gun. It is small, but it is where most of the printer’s technology is located. The extruder consists of two parts; the hot end and the cold end. The cold end has a motor that draws the filament in and pushes it through. The hot end is where the filament gets melted and squirted out.

Hot end - Heat Sink / Hot End Fan

This ensures that heat does not travel up the plastic and melt it prematurely before it reaches the nozzle. This phenomenon is called heat creep and it causes jams, especially with PLA. This fan should be running whenever the hot end is warm.

Thermistor/Thermocouple/RTD

These are all various types of sensors for determining the temperature of the hot end. They are essentially electronic thermometers. Thermistors are the most common type of sensor, but some printers will use thermocouples for extremely high temperature printing.

Heater Cartridge

The heater cartridge is pretty self explanatory. It heats the plastic. It is simply a high power resistor. Almost all modern printers use cartridge heaters, but many older printers used coils of nichrome wire (like the kind in a toaster). If you are replacing your heater cartidge, of even your entire hotend, make sure you know if your system is running 12v or 24v.

Nozzle

The nozzle is simply a piece with a small hole for the melted filament to come out of. Nozzles are interchangeable, and come in various sizes; 0.4 mm is normal, while you might use a smaller nozzle for finer detail or a larger nozzle to print faster. Nozzles can also sometimes get clogged. This is one of the most common issues with 3D printers.

End Stops (one for each axis)

The end stops are how the printer knows where it is. They are little switches that get pushed whenever an axis moves to the end. This is how the printer finds it’s starting point before printing. Most printers use mechanical switches, but some are known to use optical sensors.

Motherboard/Controller Board

The motherboard is the brain of the printer. It takes the commands given to it by your computer (in the form of G-Code) and orchestrates their execution. The motherboard contains a microcontroller (essentially a tiny, self contained computer) and all the circuitry needed for running the motors, reading the sensors, and talking to your computer.

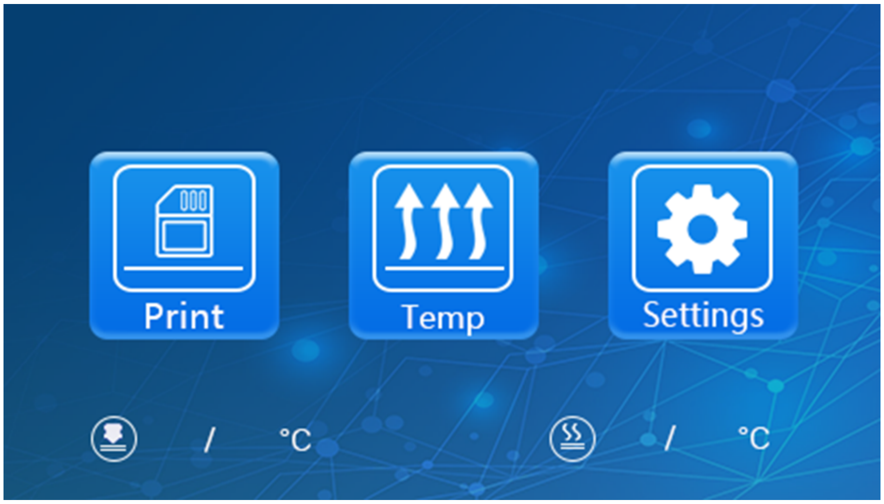
Stepper Drivers

These chips are responsible for running the stepper motors. They fire the coils of the motor in sequence, causing it to move in increments. Many motherboards have the stepper drivers built in, but some also have them in modules that can be unplugged. By balancing the power fed to each coil, the driver is also able to divide steps up into further increments. This is called microstepping, and allows more precise control over the motor than is normally possible. The stepper driver also controls how much electrical current is fed to the motor.

Screens and User Interfaces

Some printers have an LCD screen so they can be controlled directly without hooking them up to a computer. These can be basic black and white displays like the VIKI 2 or advanced enabled touchscreens like the one included on the new Ultimaker S5 3D printer.

This is the starting screen of the printer.



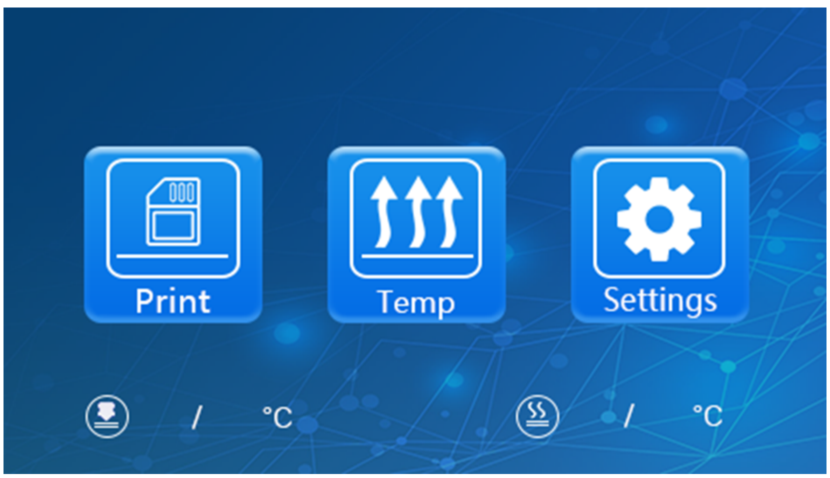
This is the home screen of the printer.

Then if we press the print option the above screen appears, in which we can see all the Gcodes that are in the SD card.

2.) How to operate the 3D printers

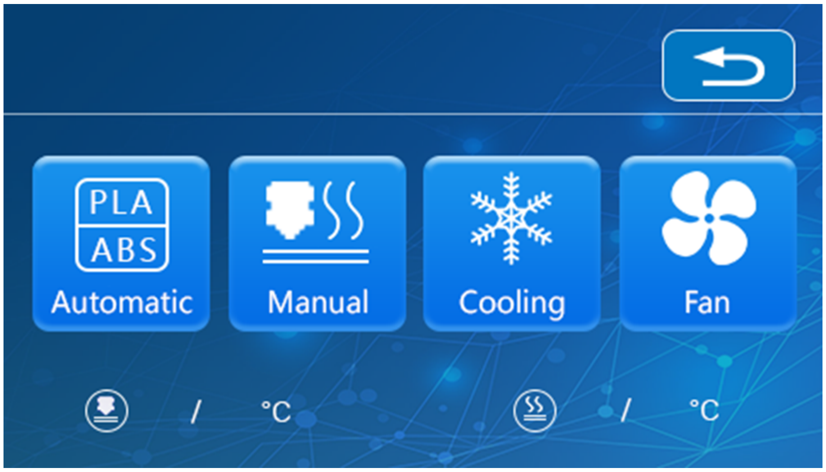
After we switch on the 3D Printer the below screen appears

And it takes a while for loading the Home screen

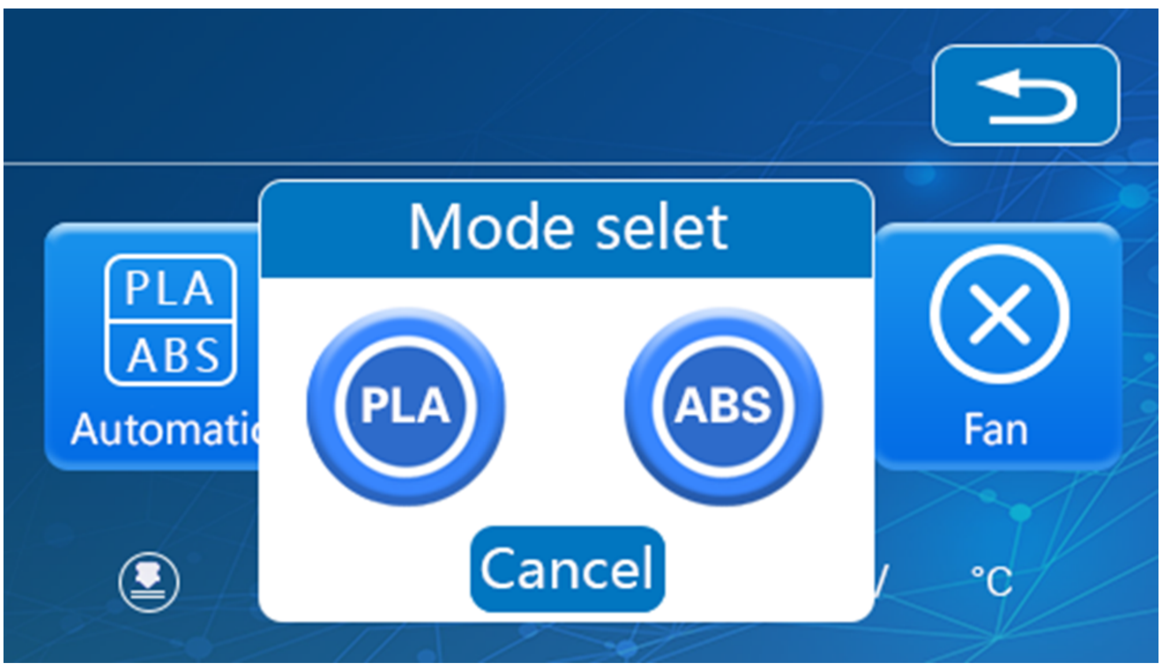


i.)How to setup 3D-printer for the print

- ***Temperature menu***



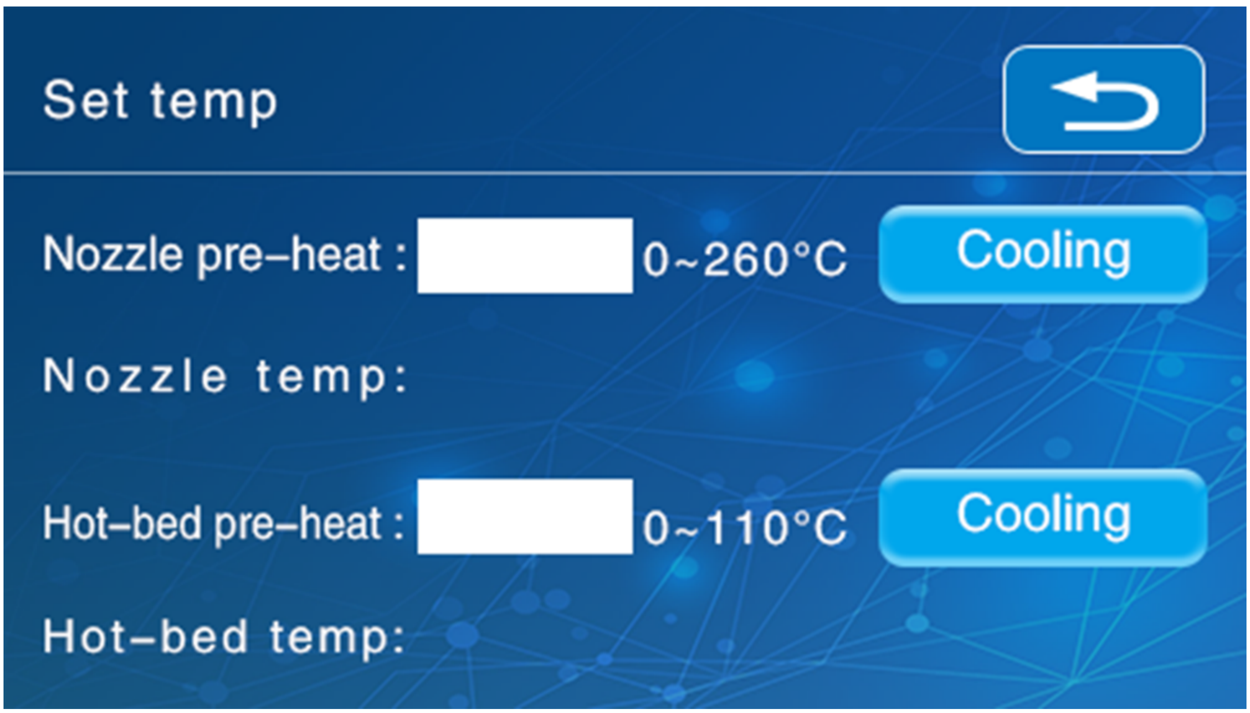
This is the temperature(temp) menu after selection it contains temperature modes for the material we are using to print which can be set accordingly or left as automatic



These are the mode and a brief explanation of ABS and PLA materials is given below for understanding.

ABS - In general it is more strong, flexible and rigid and hence preferred by a lot of engineers and professionals from other fields. It is resistant to the external factors which in some cases easily affect PLA. Finally, heated bed is an important requirements for printers printing with this filament and some printers cannot support ABS without an heated bed.

PLA - It is usually preferred by hobbyists and for other small scale innovations. The wide variety of the colours available gives the printed object a shiny finish. After the post processing, PLA is said to have sharp corners, low layer heights and increased printing speed.



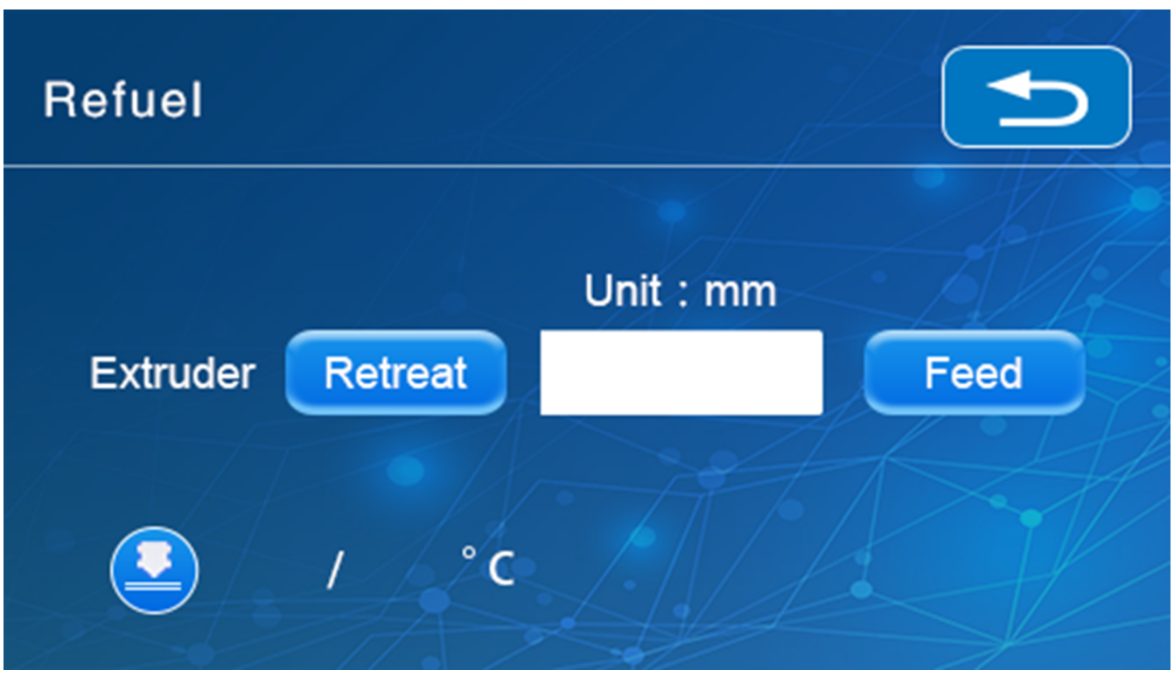
This is the manual temperature setting page in which we can set the nozzle and bed temperatures separately .

❖ ***Settings Menu***

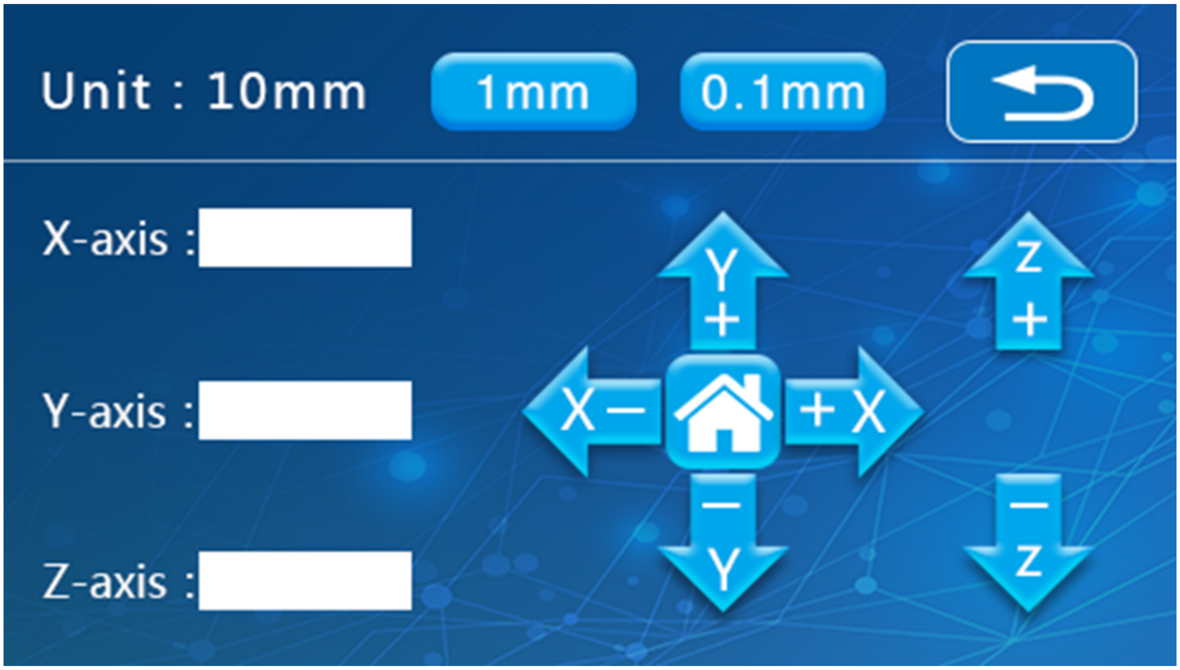
This is the settings menu which contains :

1. Leveling
2. Refuel
3. Movement of the bed and nozzle manually
4. Disabling the stepper motor
5. Language of the interface
6. Printer info

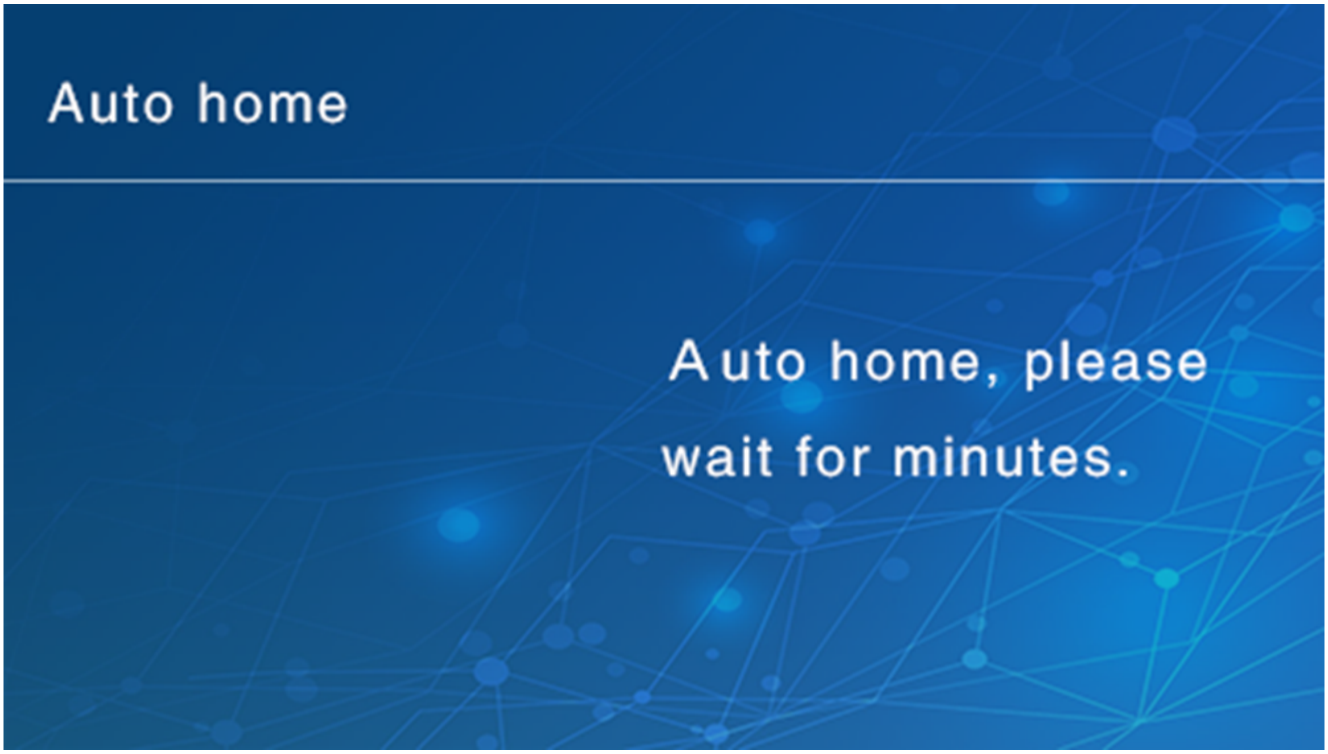
This is the leviling menu in which the proximity sensor near the nozzle measure the level of the bed and notes down the off set points of the bed as it will have some errors (like there may be a slight raise in one side of bed)



This is a refuel menu which is used to test the nozzle whether the material is extruding properly or to completely extract the previous material that is still there in the nozzle incase if we change the material after a print.



This is move menu in which we can move the X and Y axis of the bed or Z axis of the nozzle .

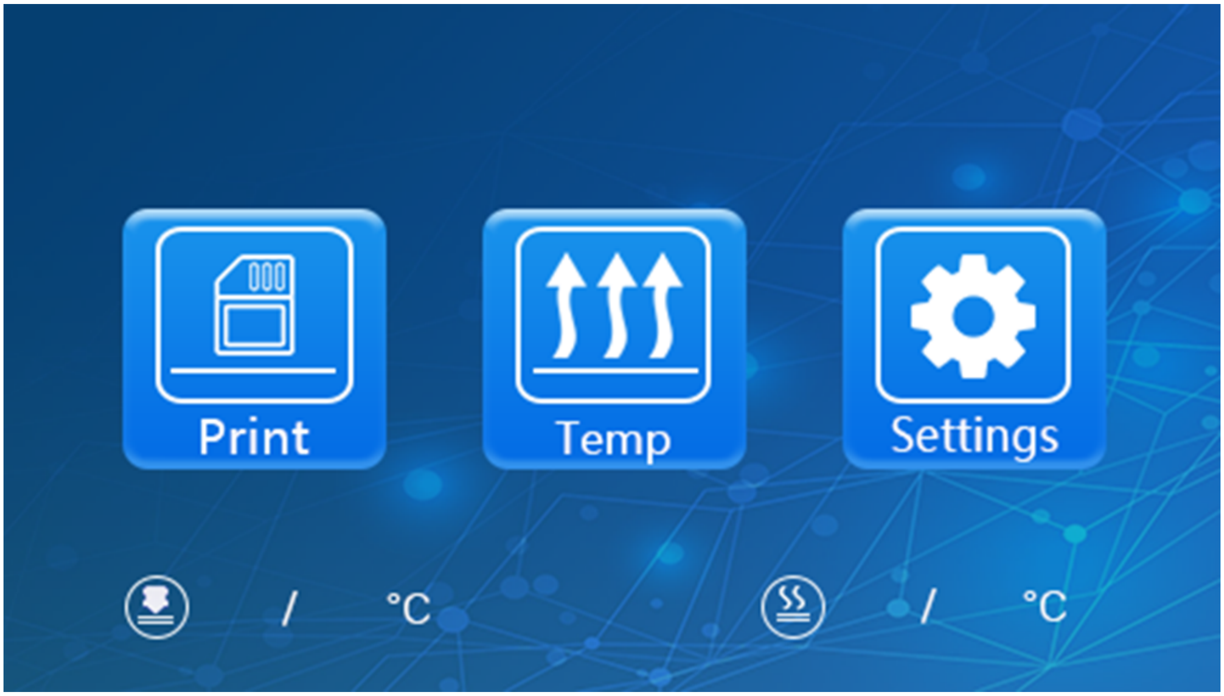


The home button in the center is called Auto-home. Auto home on a **Creality 10S-Pro**(and presumably all other 3D printers) moves the bed and the extrusion nozzle into a default “zero” position. In essence the bed and the extrusion nozzle are moved so they hit the end stop microswitches which signal to the controller board where the bed and nozzle are.

Most people assume that this is 0,0,0. In fact that is what it is set to in the factory defaults for the printer . But if you look at the bed of an Ender 3 after it has been auto homed you will find that the nozzle is actually off the bed by about 5mm. It is not in the bottom left hand corner of the plate as you would expect.

The consequence of this is that if you try to use the full surface area of the plate you will be off center. In the worst case you may actually extrude plastic into thin air. As most projects are small compared to the bed size a small offset in the center of the plate is irrelevant. But still, it is an annoyance.

ii.)How to print the design

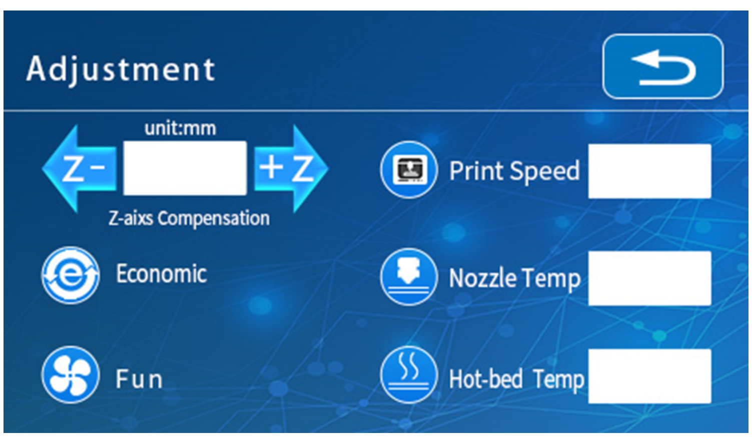


As we can see this is the printer home screen referred to in the above section. Here after we select the print option.

Here we can find the prints that we sliced selecting one will get us to the screen below.

Here we can see the progress of the print and temperatures of nozzle and bed while printing .

If we find any temperature drops or nozzle Z axis not properly touching the bed we can adjust those small changes in the *Adjust button* . Which takes us to the screen below.



In the Adjustment menu we can :

1. Change position of nozzle (i.e we move the nozzle up or downwards in 0.1mm based on the requirement).

2. Can change the speed of the printing to make the details look good incase of any distortion.
3. Finally can change the temperature of the nozzle and bed.

3.) Parameters for commonly used filaments

<u>ABS(Acrylonitrile Butadiene Styrene)</u>

This material was one of the first plastics to be used with industrial 3D printers.ABS is known for its toughness and impact resistance,which means the material can withstand much higher temperatures before it begins to deform. This makes ABS a great choice for outdoor or high temperature applications

Printing parameters

Before 3D printing with ABS make sure your 3D Printer meets the hardware requirements listed below to ensure the best print quality.

i.)Bed

Temperature: 95-110 °C
Heated Bed Require
Enclosure Recommended

ii.)**Build Surface**

Kapton tape

ABS Slurry

iii.)Extruder

Temperature: 220-250 °C
No special hot-end required

iv.)**Cooling**

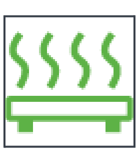
Part Cooling Fan Not Required.but can be used with 50% speed.

PLA(Polylactic Acid)

It is the default filament of choice for most extrusion-based 3D printers because it can be printed at a low temperature and does not require a heated bed. PLA is a great first material to use as you are learning about 3D printing because it is easy to print, very inexpensive, and creates parts that can be used for a wide variety of applications. It is also one of the most environmentally friendly filaments on the market today. Derived from crops such as corn and sugarcane, PLA is renewable and most importantly biodegradable

Printing parameters

i.)**Bed**



Temperature: 45-60 °C
Heated Bed Optional
Enclosure not required

ii.)**Build Surface**

Painter’s tape
PEI
Glass plate
Glue stick

iii.)**Extruder**

Temperature: 190-220 °C
No special hot-end required

iv.)**Cooling**

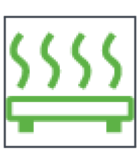
Part Cooling Fan Required
Fan Speed: 100%

PETG(Polyethylene Terephthalate Glycol**)**

PETG is a Glycol Modified version of Polyethylene Terephthalate (PET) PET . It is a semi-rigid material with good impact resistance PETG filaments are known for their ease of printability, smooth surface finish, and water resistance.The material also benefits from great thermal characteristics, allowing the plastic to cool efficiently with almost negligible warpage

Printing parameters

i.)**Bed**



Temperature: 75-90 °C
Heated Bed Recommended

Enclosure not required

ii.)**Build Surface**

Painter’s tape
Glue stick

iii.)**Extruder**

Temperature: 230-250 °C
No special hot-end required

iv.)**Cooling**

Part Cooling Fan Required
Fan Speed: depending on the material quality i.e from 40%-80%

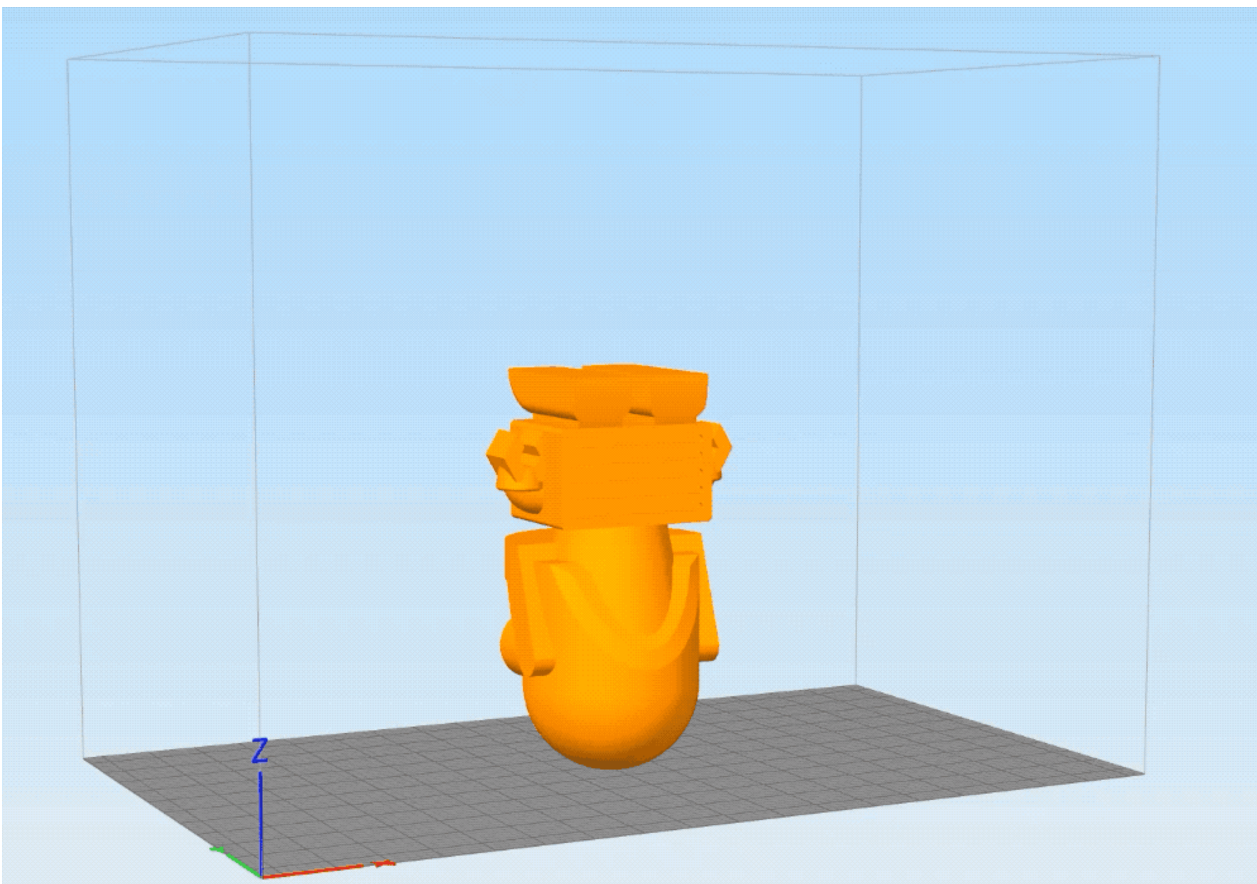
<u>4.) How to Slice the Designs</u>

_The 3D model file exists on your computer simply as digital information. In order to print it, we need to slice the model into directions that the 3D Printer can follow. These directions are known as G-code and are generated by a slicing software.
When slicing a 3D model, there are several factors you can control that will influence the performance of your 3D printed object. These include object orientation, object resolution, infill percentage, object shells, and support material.

STEP 1: POSITION THE DIGITAL OBJECT

First we will open our 3D model (STL File) in our slicing software. When the object appears in the software, we will need to orient it on the build platform.

The position of an object on the build platform can have a dramatic impact on the end result of the print. Orientation of your print impacts print time, success, accuracy, and strength of your print. Orient your model so a flat surface is facing down on the build platform.



Example of object being oriented on the build plate in the slicing software.

The Buildini™ prints in horizontal layers, building the object from the bottom up. As a rule of thumb: fewer vertical layers make for a stronger object that will also print faster.

STEP 2: OBJECT RESOLUTION AND LAYER HEIGHT

Just like a digital photograph, a 3D printed object has a resolution. High-resolution photos are made up of very densely packed pixels that create the illusion of a crisp and smooth picture. In contrast, low-resolution photos use far fewer pixels creating an image that looks pixelated.

In 3D printing, object resolution corresponds to the layer height, which is measured in millimeters or microns. Smaller layer heights result in higher resolution objects with smooth surfaces. Small layer heights will increase print time. High-resolution objects take significantly longer to 3D print than low resolution ones. This is because of all the added layers in a high resolution object.

Larger layer heights are considered low resolution, as the print lines are visible.

Example of low, medium, and high resolution objects.

Below is a chart with the different resolutions and the typical layer heights for these classifications.

Resolution	Layer Height	Relative Print Time
Low	0.3mm-0.4mm	Short
Medium	0.2mm-0.3mm	Medium
High	0.1mm-0.2mm	Long

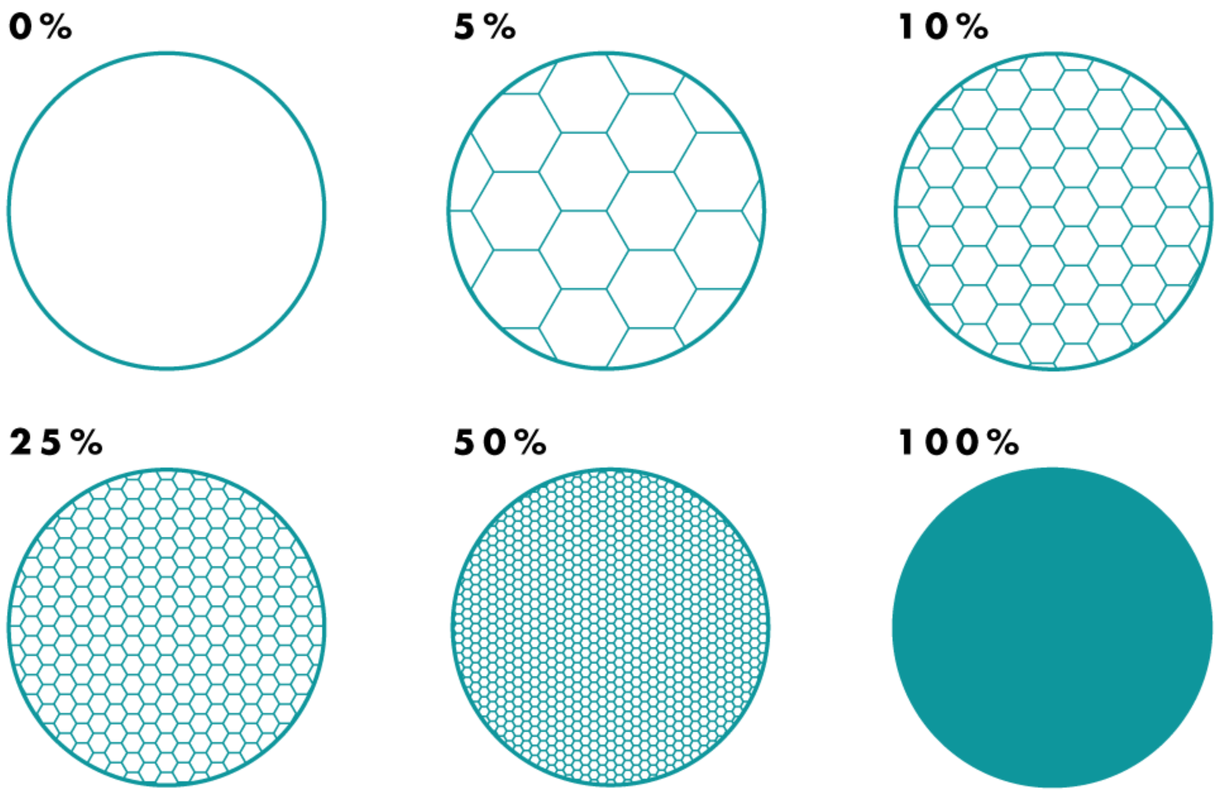
STEP 3: OBJECT SHELLS

Object shells, or perimeters, are the outer layers of the print that make up the walls of your object. The more shells, the stronger your object will be. Use fewer shells when creating display items, such as sculptures. Use more shells when your 3D printed object needs to withstand heavy pressures or stress, such as mechanical parts. Adding shells will also increase the print time and use more material.

STEP 4: INFILL

The object infill refers to the inside structure and density of a 3D printed object.

Infill is measured in percentage. An object printed at 100% infill will be 100% solid whereas an object printed at 20% infill will create a grid structure to fill 20% of the interior space of the object. The slicing software can generate infill in several patterns. We recommend the honeycomb pattern for speed and strength.



A diagram of different infill densities.

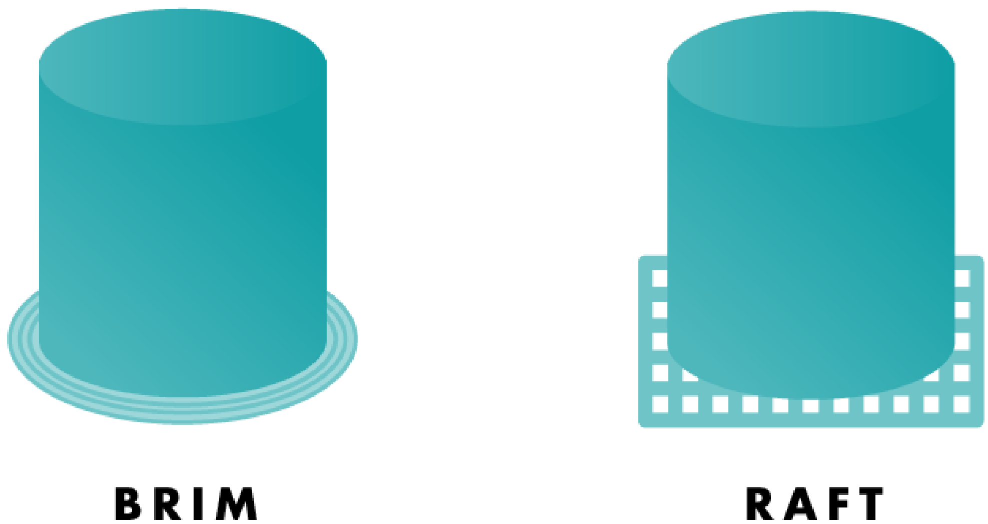
Light use and decorative objects can have a lower infill density, 5%-20%, to save time and money. Objects that need to stand up under heavy use should be printed with an infill density of 20% or higher. The denser the infill, the longer the print time.

STEP 5: SUPPORT MATERIAL AND RAFT

In the last module we mentioned that objects with overhangs or bridges that exceed 45° can be printed using support material. The slicing software includes an option for adding support material to these parts which is used as a scaffold to hold up these unsupported features. After printing, the support material will easily break away from your 3D printed object and can be discarded.

Example of an object printed with support material.

Some materials and 3D models can have a difficult time adhering to the build platform. In these scenarios it is helpful to use a raft. A raft is a removable latticework printed underneath an object to help with warping and bed adhesion. It creates a strong foundation for models with small features. Check the raft box within the slicing software and it will help to remedy your issue. After printing, the raft will easily break away from your 3D printed object.



Example of an object printed with a raft.

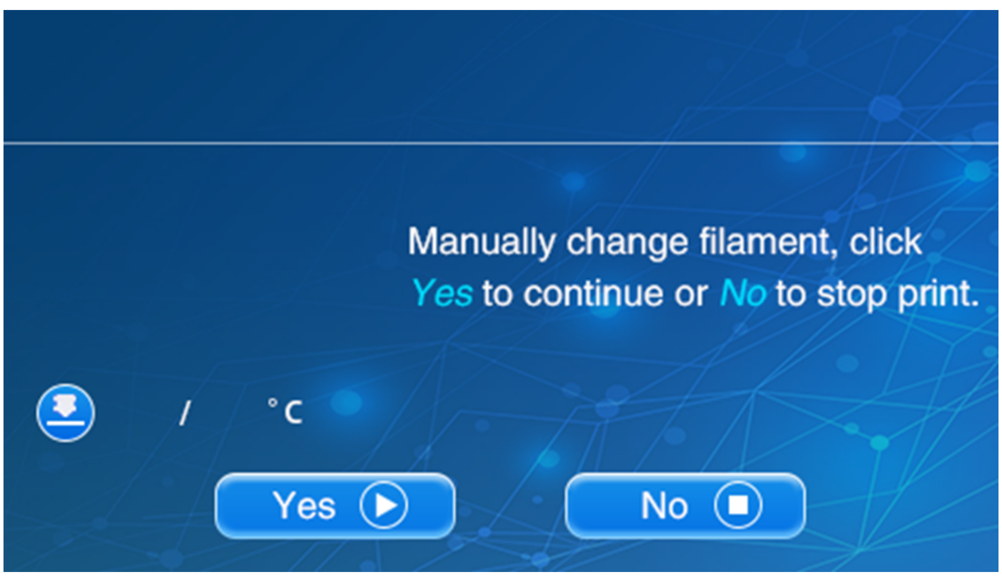
STEP 5: SLICE AND SAVE

After you have adjusted the settings to fit your printing needs, go ahead and slice your object. This will generate a G-code file. Save the G-code file to your computer or SD card. We are now ready to print with our 3D Printer.

5)Challenges faced during printing

The main challenges faced during printing are:

1. Make sure the temperature is set correctly in the gcode as well as in the printer correctly according to the materials we are using (most of the time the printer will take automatically, but should be monitored in case any change).
2. Then comes the adjustment of the proximity sensor, sometimes the sensor may take a different offset of the bed and this causes the nozzle to press hard on the heating bed or not at all touching it. In this case we adjust the screw on the proximity sensor to correctly identify the bed and make the nozzle print.
3. The filament should be place properly with no hindrance to its moment to the nozzle can take the filament smoothly without breaking the filament and if it is broken then the print will be stopped and kept in stand by until it is replaced.



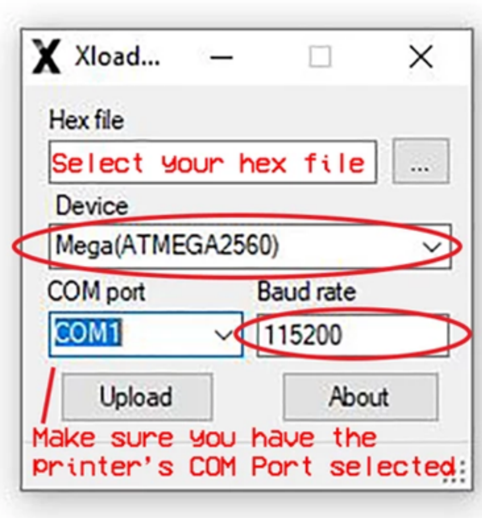
This is the prompt that asks to appear when the filament ran out.

6)How to update firmwares for the 3D printers

Download and use XLOADER to flash hex files. Prusa Slicer also works well for this. You can also visit our update guide outlining the process.

Your computer must recognize the printer and assign it a COM port number. If not, you may need a driver installed on your computer. Check out our resource page for drivers.

1. CR-10S Pro (capacitive or inductive)
2. CR-10S Pro with BL Touch (Z- port used)
3. Older Method, not suggested
4. CR-10S Pro V2 (Z+ port used)
5. Suggested Installation



Screen Files are located HERE

NOTE: You must update the screen files and mainboard for proper functionality!

Make sure the file name is **TM3D_DWINCombinedScreens_V5.7z**

7zip to extract these files. Mac/Linux Users See HERE

Flashing the screen is simple. You'll need an sd card just like the printer uses. In fact, you can back up your files, format the original card and load the appropriate DWIN_SET folder on the card. Here's a VIDEO of what it should look like.

For another resource to help you flash the screen, watch **Teaching Tech's VIDEO** on this.

For more information, see our Important Notes after installing our firmware.

For the most current firmware source, visit TM3D Source Code

Most currrent version of this document can be found at *https://docs.google.com/document/d/1cuYWpDw8aPuLEXus3uNsgDIEAKEze3Z9-hIDMWcqMK8/edit?usp=sharing*