STANDARD PRINT CO.

クリエト

An Introduction to 3D Printing

A beginner's guide to 3D Printing:
CREALITY 3D PRINTERS

STANDARD PRINT CO.

クリエト

An Introduction to 3D Printing

A beginner's guide to 3D Printing:
CREALITY 3D PRINTERS

How to use this book

Aim of this book is to prepare you so that they can get started 3D printing as quickly as possible. This book has also been structured in a way that mirrors the common workflow of a 3D printer operator and as such is intended to be read sequentially from beginning to end.

Part 1: The basics of how the technology works, how to operate your printer and covers a brief primer on designing models with 3D printing in mind.

Part 2: The predominantly digital portion of 3D printing and outlines the various steps and considerations that must be undertaken when preparing a 3D model for printing.

Part 3: Contains a number of checklists for each part of the 3D printing process as well as tips for troubleshooting your prints and cheat sheets containing important values

By the end of this book you should be able to:

- Design a 3D model for 3D printing
- Slice a 3D model into G code for your printer
- Load the model and filament into your printer
- Print your 3D model

A QUICK GUIDE TO YOUR FIRST PRINT

1. Read the safety instructions carefully **(Refer to Creality Manual)**
2. Download and Install the software **(Page 17)**
3. Calibrate your printer **(Refer to Creality Manual)**
4. Slice your first model and export to SD card **(Page 29)**
5. Insert the SD card into the printer and print your first model

---

Glossary of Terms

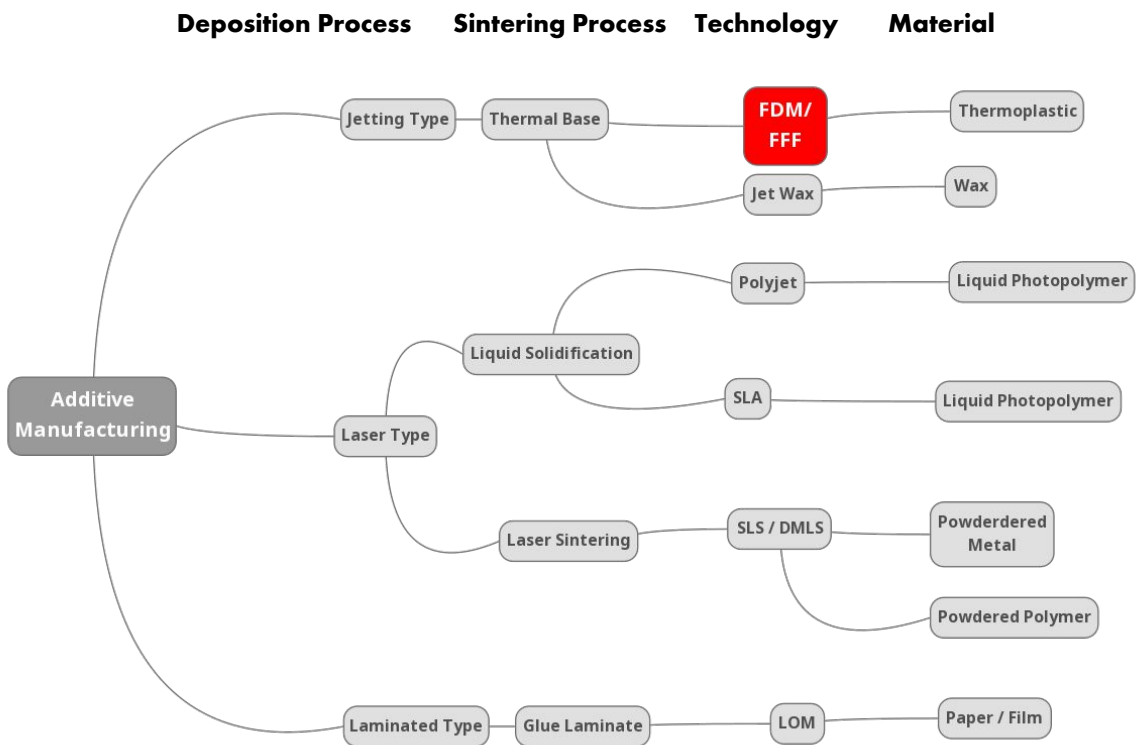|			
Additive Manufacturing	A 3D printing process where a 3D object is created from a digital model through the deposition of successive layers of material.	**Raft**	A layer of sacrificial material (see support) that is printed below an object to help it adhere to the print bed.
Build Plate	Also known as the bed of a printer. It is the plate on which 3D models are printed on. Commonly made of Aluminium, Steel, Glass or other composites.	**Shell**	The outer walls of a 3D printed object. Also known as perimeters or walls.
Computer Aided Design (CAD)	Software used to develop a 3D depiction of a 3D object.	**Slice**	A virtual 2D layer of a digital 3D object that is produced by a slicer program. This virtual slice provides a map and serves as a set of instructions for your 3D printer as it 3D prints a model. Each 3D model is composed of hundreds if not thousands of these layers stacked on top of each other.
Extruder	The hot end of a 3D printer. The part that filament is fed into at a certain temperature and deposits the molten filament onto the build plate.	**Slicer**	A specialised peice of 3D printing software that takes a 3D model and converts it into a set of printing instructions for your 3D Printer. through this software, one is able to set the parameters for a 3D print for their 3D printer.
FFF	Fused Filament Fabrication. A 3D printing process where molten thermoplastic is deposited in successive layers to produce a finished object using a digital model.	**Slicing**	The process of turning a 3D model into its individual slices. (see Slice)
Filament	The main type of printing material used by FFF 3D printers. Usually a thermoplastic that is fed into a hot extruder at the plastic's melting point and deposited onto the build plate. Filament comes in various materials, diameters and is usually sold in spools.	**Support**	Printing materials used to bolster overhangs on a 3D printed object. These materials are usually considered sacrificial and are discarded after the printing process.
G-Code	G-code is the language used to instruct your 3D printer to perform operations. G-codes control specific actions like motion, speed, rotation, depth, and other related switches and sensors used in the operation of a machine.	**X-Axis**	Part of the cartesian coordinate system used by FFF type 3D printers to navigate a 3D Space. This represents the left to right , horizontal movement.
Infill	The amount of material that will be printed inside the outer shell of a 3D printed object. In the interests of material conservation and expediency, models are rarely printed at 100% infill.	**Y-Axis**	Part of the cartesian coordinate system used by FFF type 3D printers to navigate a 3D Space. This represents the forward to back horizontal movement.
Overhang	Any part of a 3D model that lacks solid material to support it from below. Parts that protrude from the main body of the print at angles greater than 45° are generally considered overhangs.	**Z-Axis**	Part of the cartesian coordinate system used by FFF type 3D printers to navigate a 3D Space. This represents the up and down vertical movement.

Table of Contents

Part 1:
ALL ABOUT
3D PRINTING

Introduction into 3D Printing

There are several types of 3D printing technology currently available at the time of print. While these essentially aim to achieve the same thing - to give a solid form to a digital model - how this is achieved and the materials that are used to form the model are varied. These processes are collectively known as **Additive Manufacturing (AM)** but to keep things simple, we'll refer to it by it's practical name - 3D Printing.

Additive Manufacturing Visualised

 Below is a diagram depicting the various technologies currently available under the umbrella of additive manufacturing at the time of publication.

| Deposition Process | Sintering Process | Technology | Material |

Additive Manufacturing

- Jetting Type — Thermal Base — **FDM/ FFF** — Thermoplastic
- Thermal Base — Jet Wax — Wax
- Laser Type — Liquid Solidification — Polyjet — Liquid Photopolymer
- Liquid Solidification — SLA — Liquid Photopolymer
- Laser Sintering — SLS / DMLS — Powderdered Metal
- SLS / DMLS — Powdered Polymer
- Laminated Type — Glue Laminate — LOM — Paper / Film

Introduction into 3D Printing

. **How Does It Work?**

While there are several types of 3D printing technologies currently used by various industries today, the additive manufacturing technology covered in this guide is called **Fused Filament Fabrication (FFF)**.

1. FFF printing starts with a digital 3D model, most often generated from a **Computer Aided Design (CAD) program** or a **Digital Sculpting Program.**
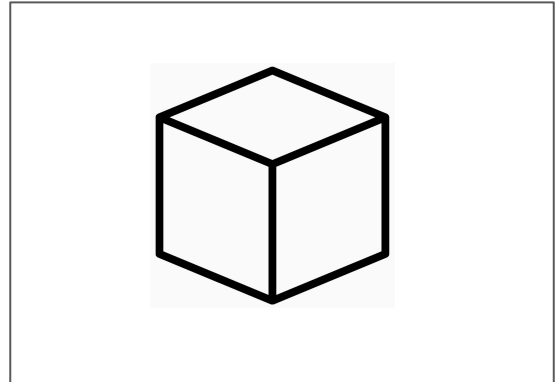
2. The 3D model is sliced in 2D layers using slicing software that is then sent to the printer as a set of instructions called **G code.**
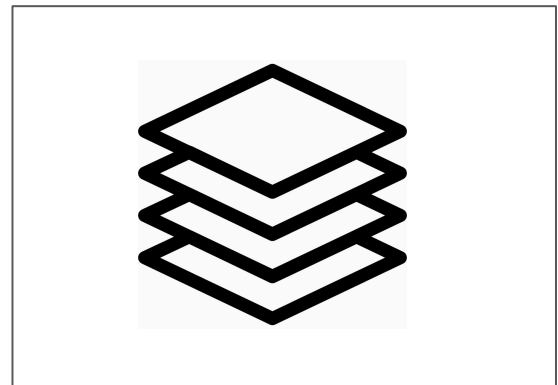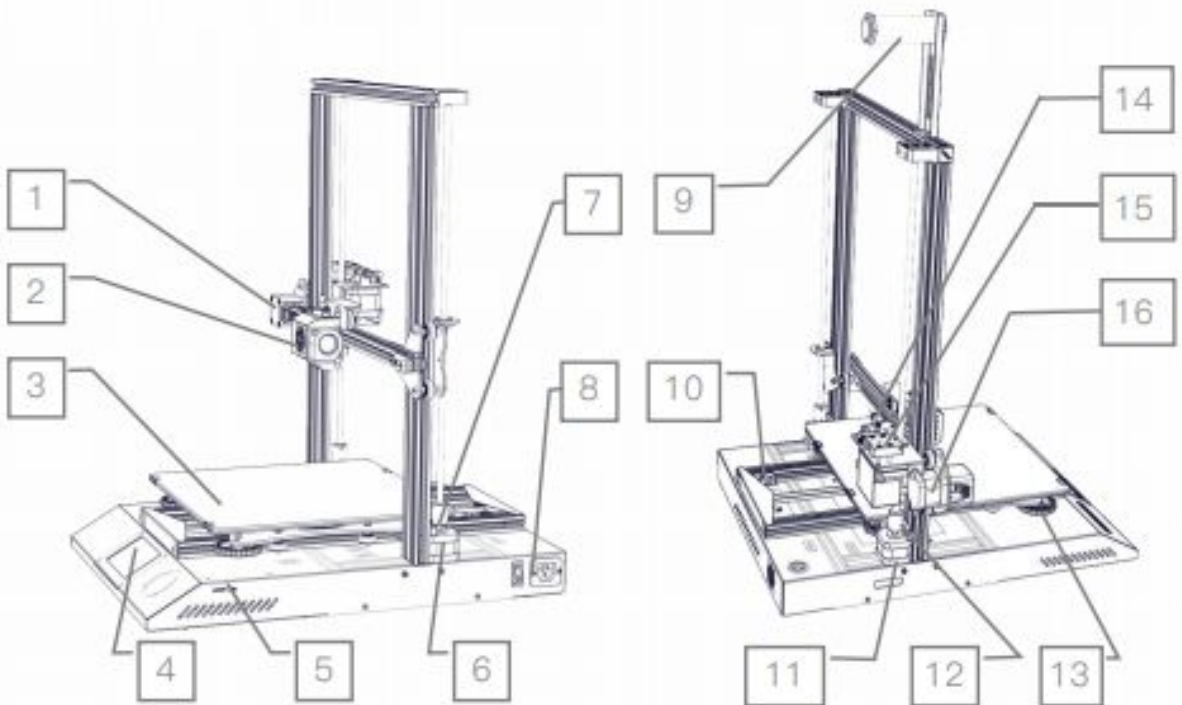
3. The printer follows the instruction set out by the **G code.** It does this by **extruding** molten plastic, drawing each 2D slice, layer by layer onto the **Build Plate.** Over time, these 2D layers are stacked on top of one another forming a 3D printed object.

Get to know your 3D printer

1

1. X Limit Switch
2. Nozzle Kit
3. Hot Bed
4. Touch Screen
5. TF slot & USB Port
6. Z-axis Motor (Z2)
7. Coupling (Z2)
8. Power Switch & Socket
9. Filament Holder
10. Y Limit Switch
11. Z-axis Motor (Z1)
12. Coupling (Z1)
13. XL Bed-leveling Nut
14. Extruder (E) Motor
15. Filament Detector
16. X-axis Motor

Get to know your 3D printer

2
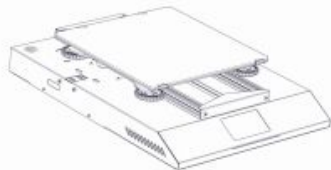
Gantry Frame
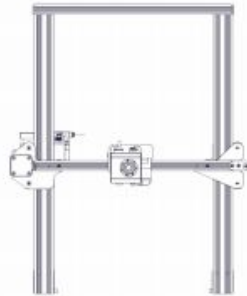
Base Frame

Tool Box

Filament(200g)

◆ Tool List

No.	Image	Name	Qty	No.	Image	Name	Qty	No.	Image	Name	Qty
1		Wrench & Screw driver	1set	6		Filament Holder Bracket	1	11		PTFE Tube	1
2		TF Card & Reader	1set	7		Filament Holder Tube	1	12		X–Axis Spacer	1
3		Spatula	1	8		Filament Holder Tube Nuts	1	13		Feeler	1
4		Pliers	1	9		Power Cable	1	14		Spare Parts	1set
5		0.4mm Nozzle Cleaner	1	10		USB Cable	1	15		M5X25 black Hex–Bolt and Lock Washer	4set
								16		M4X8 Screw and M4 T– nut	2set

Get to know your 3D printer

3

Screen Information

Choose to Print	Automatic
	Manual
	Cooling
	Fan
	Levelling
	Refuel
	Moving
	Motor
	Printer info
	Language

Current temperature of the Hot Bed
Set temperature of the Hot Bed
Current temperature of Nozzle
Set temperature of Nozzle
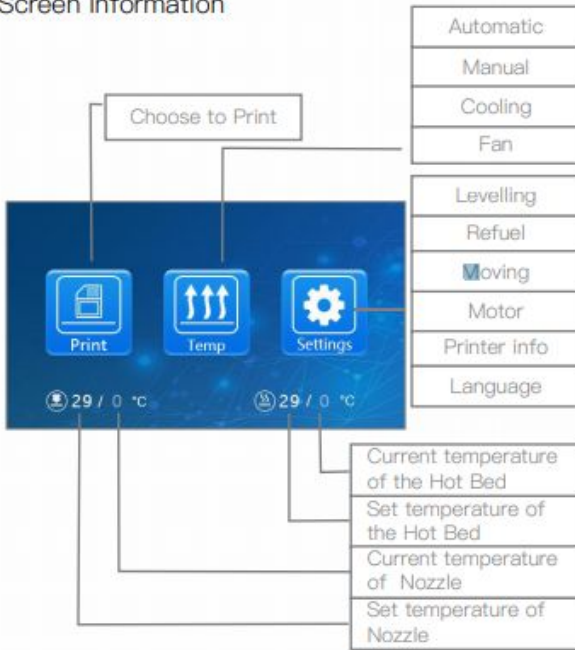
Screen Information			
Main Menu	Sub Menu	Explanation	
TF card	Selected file	Stop	
		Pause/Continue	
		Adjust	Print speed
			Nozzle temp.
			Hot-bed temp
			Z offset
			fan
Temp	Automatic	PLA\ABS (195°\240°)	
	Manual	Nozzle preheat	
		Hot-bed preheat	
	Cooling	Cool down? Yes\No	
	Fan	Turn on\Shut down	
Setting	Levelling	Z-axis: Z home, +0.1mm,-0.1mm	
		AUX leveling	Please click numbers to assist leveling (①-⑤)
		AUTO leveling	Auto leveling, please wait...
		Check level (Measurement parameters)	
	Refuel	Withdraw	
		Feed	
	Moving	X-axis	Y- axis	Z-axis	Z home
	Motor off	yes\no	
	Language	English\Chinese	
	Printer info	Machine type, Firmware version, Printing size, website	

Loading Filament

◆ Preheat

Method 1、

Method 2、

◆ Feeding

Press and hold the extruder lever and insert the 1.75mm filament through the filament detector and through the extruder motor. Continue feeding until you see filament extrude from the nozzle.

Replacing Filament During Printing:

1、Cutting the previous filament near the extruder and slowly feed new filament though the filament sensor and extruder, until the new filament is fed into the PTFE tube.
(Or)
2、Preheat the nozzle and withdraw the used filament. Feed the new filament as pictured above.
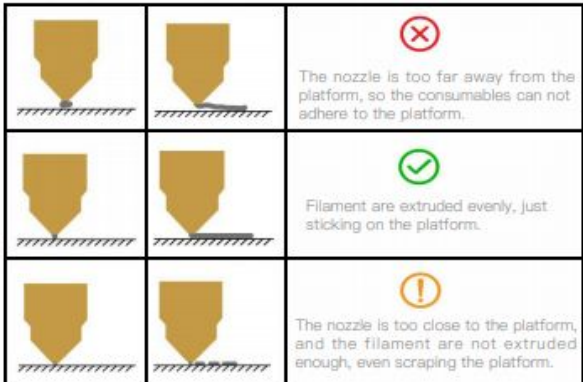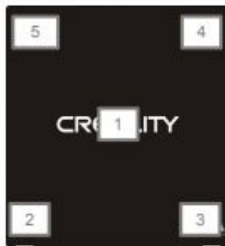
Get to know your 3D printer
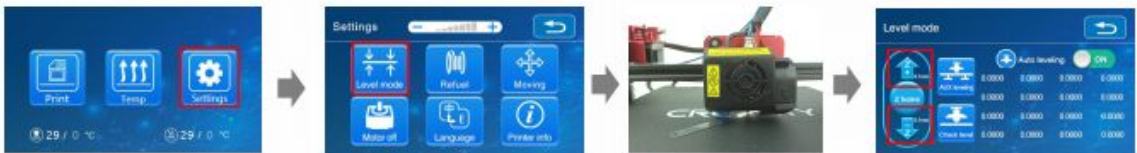
4

Bed Levelling 1

1. Slightly tighten the four levelling nuts at the bottom of the platform before initial levelling.
2. Select "Settings" → "Level mode", click on the number ②.

3. Adjust the platform height by turning the knob underneath. Use a piece of A4 paper (standard printer paper) to assist with the adjustment, making sure that the nozzle lightly presses on the paper.
4. Complete the adjustment on all 4 corners.
5. Click the number ① to test the platform height in the middle.
6. Repeat above steps one or two times if necessary.

		⊗ The nozzle is too far away from the platform, so the consumables can not adhere to the platform.
		✓ Filament are extruded evenly, just sticking on the platform.
		⚠ The nozzle is too close to the platform, and the filament are not extruded enough, even scraping the platform.

Bed Levelling 2

◆ Automatic leveling

1. In "Settings" → "Leveling mode" → Place the .2mm feeler gauge between the nozzle and the hot bed → If the gap is too thin or too large, click "Z+" or "Z−" on the interface to adjust the gap to slightly more than 0.2 mm(the distance between the nozzle and the hot bed should be about 0.2mm). At 0.2 mm the feeler can easily pass through the gap between the nozzle and the printing platform.

2. To adjust position of the auto−level sensor, turn the sensor clockwise to lower it and counter−clockwise to raise it.
3. If the LED is lit, raise the auto−level sensor by turning it counter clockwise.
4. The slowly turn the sensor clockwise until you see the LED light up again.

Restore hotend to original home position

Auto−level the printing surface

16 point precise bed height measurement

Printer will automatically compensate for variable bed height

*When using the auto level feature on the CR-10S Pro, we suggest adding a raft from the build plate adhesion section in software settings.

Designing for 3D Printing

Rule 1:
3D printers cannot print over air and must be supported either by the bed, a previous layer or a support layer.

In order to produce successful printed parts on the your 3D printer, it's critical to use the correct machine parameters, but even more important is to apply to correct design rules for your 3D printed parts. If the geometry to be printed does not comply with very specific design rules the part will not print correctly and there will be a risk of damaging the machine.

Your 3D printer print layers of molten polymer, in other words a viscous plastic. This polymer needs to adhere on the print bed or onto an existing polymer layer. In the pictures below those layers are visualized using **black horizontal lines**.

1

2

3

Overhang
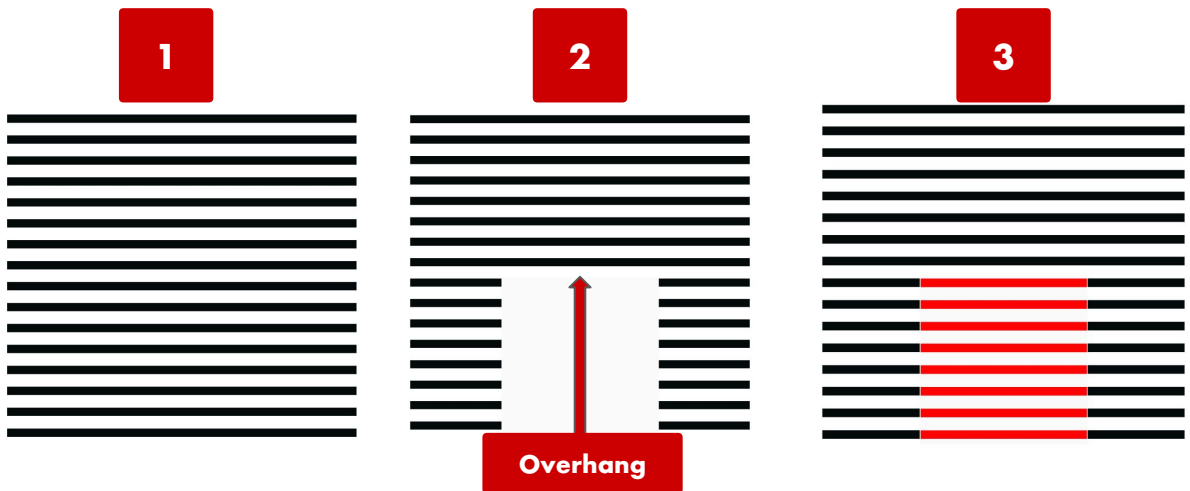
Looking at **model 1)**, one can see that the model is solid with each layer being perfectly preceded by another layer. When printing this model, **support material is not required.**

When viewing **model 2)** one can see that there is a section of unsupported material. In 3D printed, this is called an **Overhang.** When 3D printing this model, special attention is required when selecting the correct machine parameters as print failures or machine damage could occur without proper attention.

In order to print model 2) properly, **support material** needs to be generated in a way that the overhanging layers are supported. This can be seen depicted in red in **model 3).** More on support materials and support material generation will be covered in chapter 7.

Part 2:

PREPARING TO 3D PRINT

Installing Software

To begin 3D printing, you'll need access to the prerequisite software.

3D Model Design

In order to 3D print, you'll need a 3D model to print. Here are a few ways to acquire your 3D model.

Designing a 3D model is the best way of getting the exact model you're after. When designing a 3D model you have 2 main ways of going about it:

> **Parametric Modelling** with a Computer Aided Design (CAD) software such as Fusion 360, Rhino or Solidworks.

> **Digital Sculpting** with Digital Sculpting software such as Meshmixer.

Scanning an existing object with a 3D Scanner.

Download a model from a website such as Thingiverse or Grabcad

3D Model Slicing

Cura is the dedicated slicing software for your 3D printer and is required for proper operation of the printer. Within Cura you can view your 3D models, manipulate them, set your desired print parameters and most importantly, convert your **.stl** 3D model into a printer readable **.gcode** file to be sent to your printer and printed. Cura is free and available here: https://ultimaker.com/software/ultimaker-cura
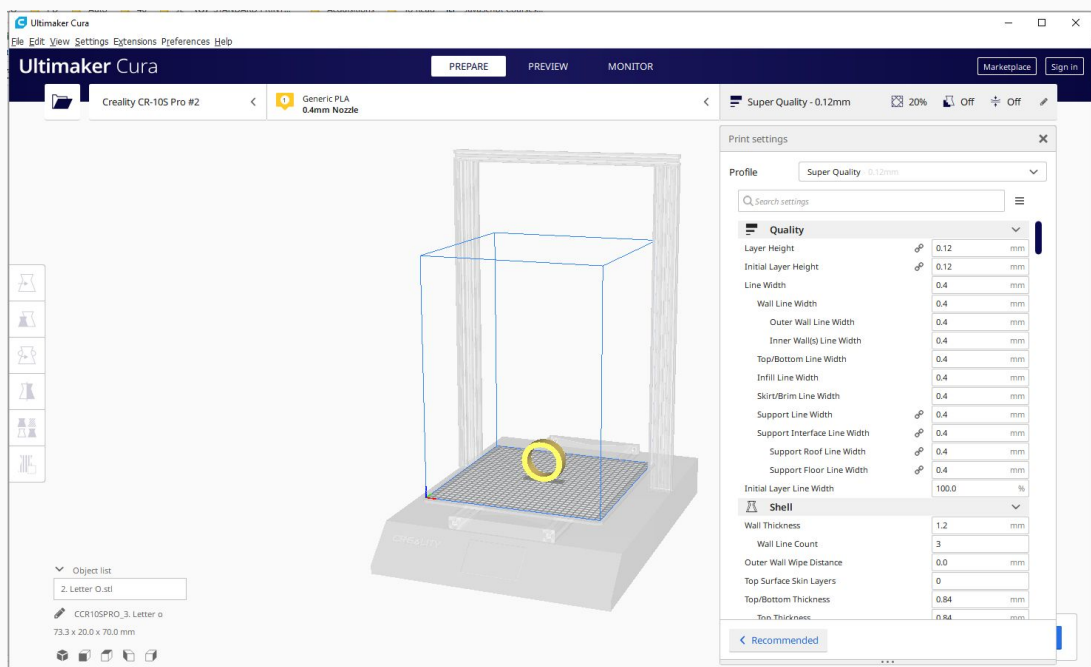
Take a moment to download Ultimaker Cura if you haven't already done so. It's required before proceeding further in in this publication.

https://ultimaker.com/software/ultimaker-cura
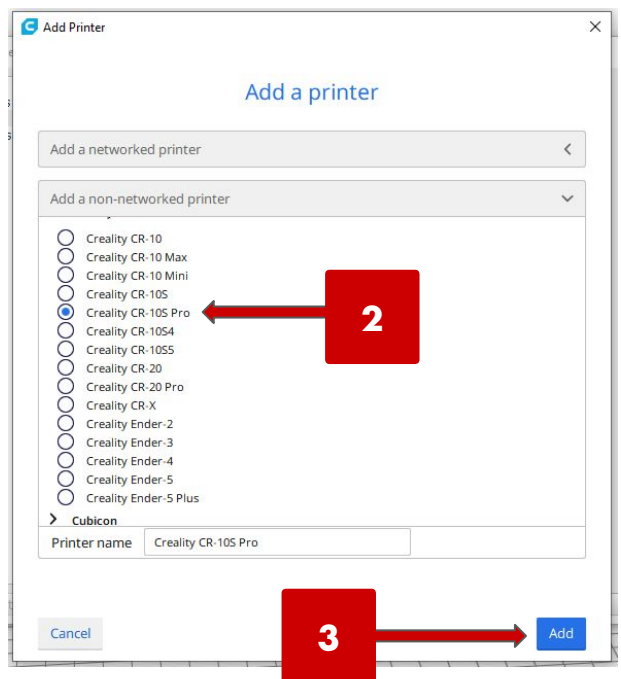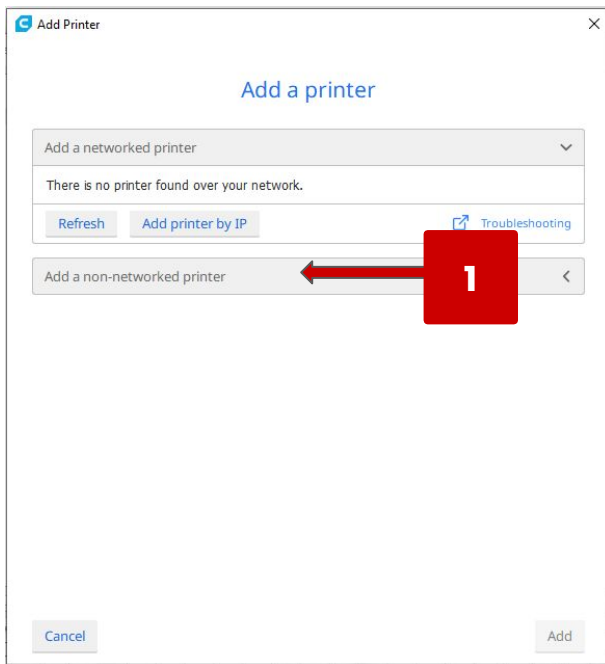
Installing Software

Setting Up Your Printer

Upon opening and installing Cura, you will be prompted to setup a new printer.

Step 1: Select **Add a non-networked printer.**
Step 2: Scroll down to **Creality > Creality CR-10S Pro.**
Step 3: Click **Add.**

Material Choice

Rule 2:
Know the material properties of the different types of filament, their behavior and match them with the correct application.

Why so many materials?

Different materials have different inherent material properties. Looking to print something dimensionally accurate for prototyping and fit testing? Try PLA. Looking to print a durable and chemically resistant part? PETG may be what you're looking for. It is important therefore to choose the correct material for your chosen application.

Material Settings

With different material properties come different printing settings. These range from the most basic of settings such as **nozzle and heat bed temperatures** to more advanced settings like **retraction speed** and **coasting.** Settings for each material will differ slightly and can be adjusted in your slicer's **material profile** section. Don't worry, you won't have to remember all the settings for each material type, these settings can be stored and edited in **material profiles** that are usually provided by the manufacturer of each material.

Filament Care

Thermoplastics are almost all universally **hygroscopic -** they absorb moisture from the ambient air. While its effect varies across materials, in general this absorbed material results in a weaker 3D print in and may present problems during the printing process. As such, it's best keep your filament in an airtight environment with a humidity level of less than 20%. This can be achieved by placing the filament in close proximity to substances such as desiccants which actively absorb moisture from the air inside the container. While some filaments are resistant to UV degradation, it's also advised to keep your filament out of direct sunlight. Refer to the filament properties chart on the following page for more details.

Material Choice

Filament Properties Table

|	ABS	ASA	Carbon Fiber Filled	Flexible	HIPS	Metal Filled	Nylon	PLA	PETG	PC	PP	Wood Filled	PVA
Strength	40MPa	55MPa	45 - 48MPa	26 - 43MPa	32MPa	20 - 30MPa	40 - 85MPa	65MPa	53MPa	72MPa	32MPa	46MPa	78MPa
Stiffness	5/10	5/10	10/10	1/10	10/10	10/10	5/10	7.5/10	5/10	6/10	4/10	8/10	3/10
Durability	8/10	10/10	3/10	9/10	7/10	4/10	10/10	4/10	8/10	10/10	9/10	3/10	7/10
Maximum Service Temperature	98°C	95°C	52°C	60 - 74°C	100°C	52°C	80 - 95°C	52°C	73°C	121°C	100°C	52°C	75°C
Printability	8/10	7/10	8/10	6/10	6/10	7/10	8/10	9/10	9/10	6/10	4/10	8/10	5/10
Extruder Temperature	220 - 250°C	235 - 255°C	200 - 230°C	225 - 245°C	230 - 245°C	190 - 220°C	220 - 270°C	190 - 220°C	230 - 250°C	260 - 310°C	220 - 250°C	190 - 220°C	185 - 200°C
Bed temperature	95 - 110°C	90 - 110°C	45 - 60°C	45 - 60°C	100 - 115°C	45 - 60°C	70 - 90°C	45 - 60°C	75 - 90°C	80 - 120°C	85 - 100°C	45 - 60°C	45 - 60°C
Flexible				✓		✓	✓						✓
Elastic				✓									
Impact Resistant	✓	✓			✓		✓				✓		
Soft				✓		✓							✓
Composite			✓									✓	
UV Resistant		✓											
Water Resistant						✓	✓		✓				
Dissolvable					✓								✓
Heat Resistant	✓	✓			✓		✓						
Chemically Resistant						✓			✓		✓		
Fatigue Resistant				✓		✓	✓		✓		✓		✓

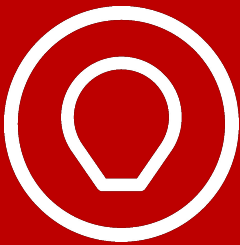# Material Choice

HOW TO LOAD A MATERIAL PROFILE

As each material is unique, each requires a matching **Material Profile** to be correctly printed. These material profiles communicate to the printer the print settings for each material. A material profile can be acquired in the following ways.

1. A Generic Profile can be selected form the list. This will provide the least optimised printing experience.

2. Create Your Own profile if there is no preexisting setting that is suitable.

3. Import a Profile if you have pre-prepared profiles. Pre-optimised, material profiles for Standard Print Co. Filaments can be found in your "Standard Print Co. - Getting Started with 3D Printing" file pack or downloaded here.
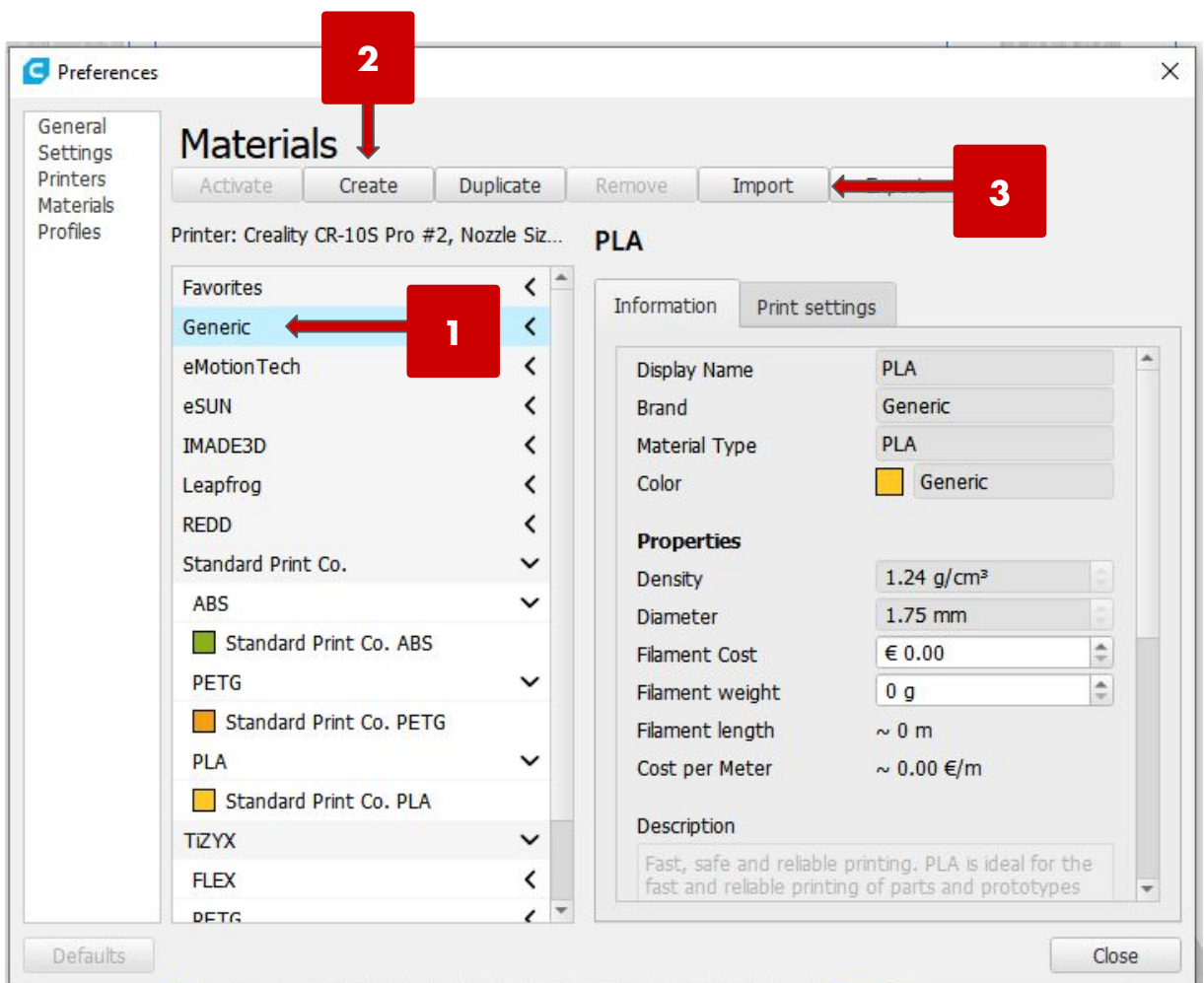
Take a moment to import the Standard Print Co. Material Profiles included in **4. Material and Print Settings** of your welcome pack before proceeding.

Material Choice

DO'S

- Store filament in a cool, dry location, preferably in an airtight and dehumidified container out of direct sunlight.

- Choose the most optimal material for your desired application.

- Know the properties of the materials that you are printing in.
-
- Apply the correct material profile for the material you are printing in.

- Ensure that printing occurs in an area with proper ventilation

- Ensure the ambient temperature stays constant with the printing area free of any strong draughts.

- Unload filament from printer when not in use to keep it in optimal printing condition.

- Know the safety guidelines associated with the materials you intend to print with prior to printing. (see chapter 13)
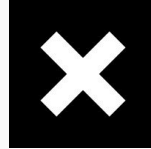
DON'Ts

- Leave filaments out in the open air and expose to sunlight.

- Place your printer in high traffic areas or near air conditioning.

- Use an incompatible **Material Profile** for your chosen material.

- Print with materials without first consulting it's **Material Safety Data Sheet (MSDS)** and **Material Specification Sheet** available upon request.

- Keep filament "loaded" in the printer for long periods of time without printing.

Orientation

Importing A Model

Once you've configured your material settings, it's time to import a model. This can be done by dragging and dropping a model into the window or clicking the **Open A File** button on the top left corner.

Manipulating your model

After importing your model, it may not appear in an orientation that is optimal for printing. If this occurs, you may need to manipulate the model using the following controls:

(T) Move - Allows for manipulation your 3D model by eye using the arrows or more accurately by entering integers in the X, Y and Z windows shown. It is important to always check the Z value is 0 to ensure the part is situated on the bed unless you generate **Manual Supports** or activate **Generate Supports** on the right pane. (see chapter 7)

(S) Scale - Allows accurate scaling of your model along the X, Y and Z dimension.

(R) Rotate - Lets you rotate your model. Tick **Snap Rotation** to rotate your model in 15° increments or **Lay Flat** to auto orientate your model on the bed.
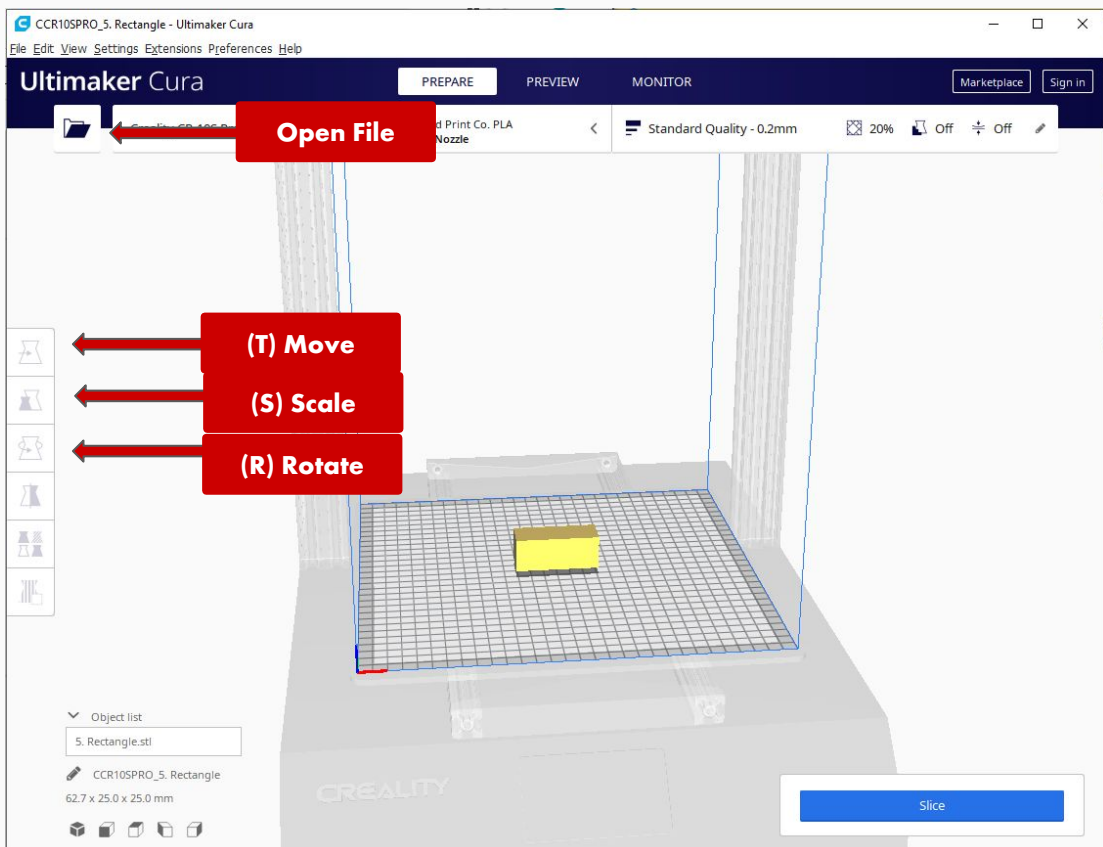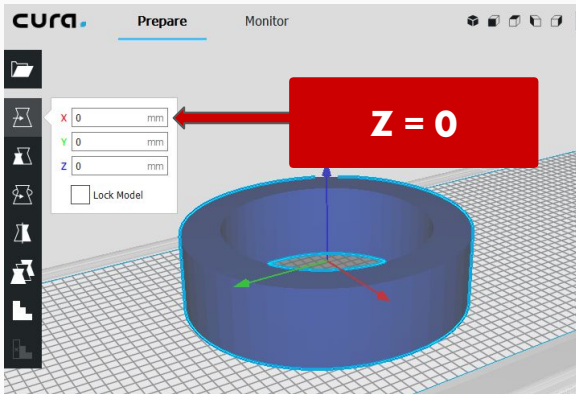
Orientation

Orientation Tips

TIP 1 - Bed Orientation

It is important to always check the Z value is 0 to ensure the part is situated on the bed unless **Manual Supports** are generated or **Generate Supports** on the right pane is activated. (see chapter 7) Any parts of your models below the bed (z=0) will not be printed. As standard, CURA has the setting **Automatically Drop Models To Build Plate** activated in **Preferences > Configure Cura.**

TIP 2 - Support Minimisation

Supports can be minimised in most cases by simply re-orienting the model. To visualise the supports generated by the slicing software, select the **Layer View** pane on the top right hand corner of the window to preview the auto-generated supports.


Support   Generation
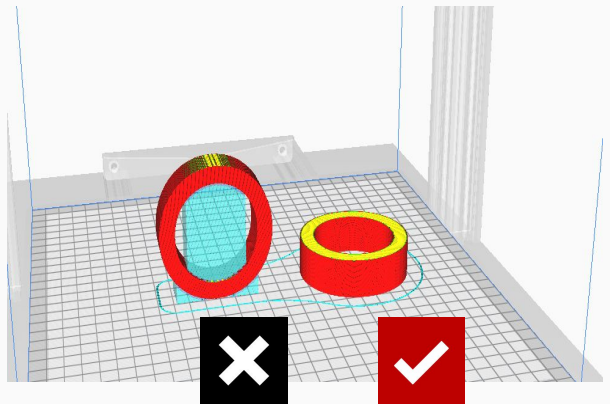
Remember [rule 1](#)? It's not always possible to print solid models or models that are **overhang free**. An overhang is any part of a **print** that extends outward, beyond the previous layer, without any direct support. To print models such as these, we'll need to generate support material.
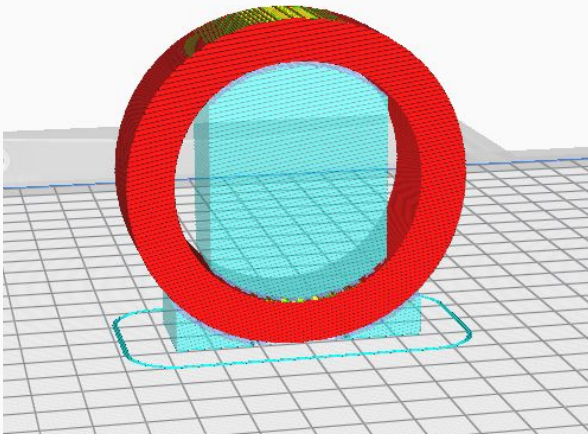
Automatic Support Generation

For models that can't be oriented to eliminate the need for support material, you'll need to generate some. This can be done by activating **Generate Support** under the **support** tab on the right hand side of the window. This will drop supports for all overhangs over a certain angle.

(0° for all supports and 90° for none) In **Solid View** the supported surfaces will highlighted in red. If in doubt set support angle to 90°.
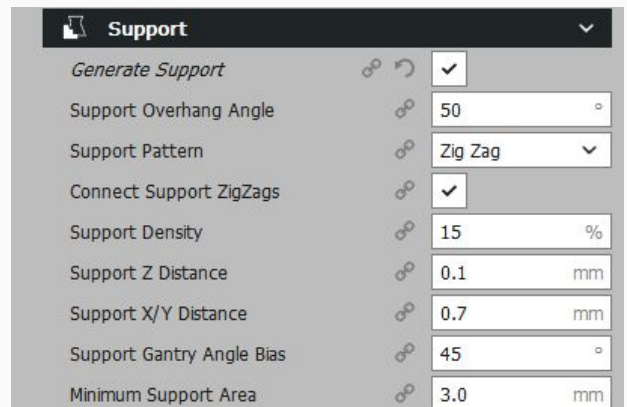
Support and Rafts

Support Blocker

Sometimes the strategic placement of small sections of support material is called for instead of supporting the entire print. Printing the minimum amount of support material is beneficial as it saves both material used and printing time.

Automatic Support Generation - While we could generate support material under all unsupported overhangs using the **automatic support generation** as pictured in the image above (2) this would result in a print time of **4hr 15min using 140g of filament in total.**
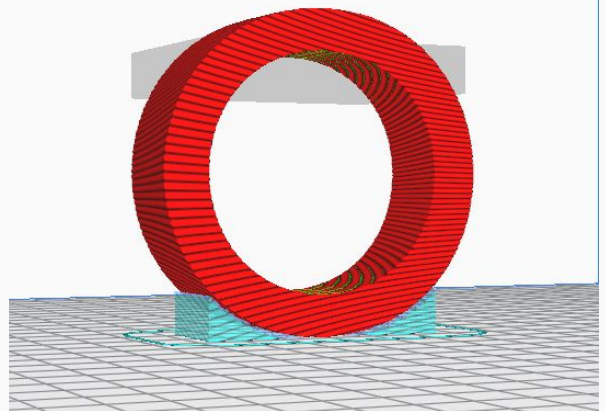
Support Blocker - Instead, we could choose to generate two small blocks of support material (3), manip Bulating their size and position to support only the overhangs that require them. In the configuration above, this would result in a print time of **3hr 31min using 131g of filament in total.**

Support and Rafts

Support Blocking

Step 1 - Begin by selecting the model, hitting the (**E) key** to activate **Support Blocking.**

Step 2 - Click anywhere on the model to automatically attach a negative support block. This will generate a digital block that blocks out support material. Don't worry if your support block isn't exactly in the right place when you first generate it as this can be adjusted later.

Step 3 - Manipulate the size and position of your support blocks using the **Move (T)** and **Scale (S)** tools, making sure to situate the support structures where existing support material is situated. You can check on the progress of your support blocking by switching to the **preview pane** and **slicing** yout model to see what support material is generated and what is blocked.
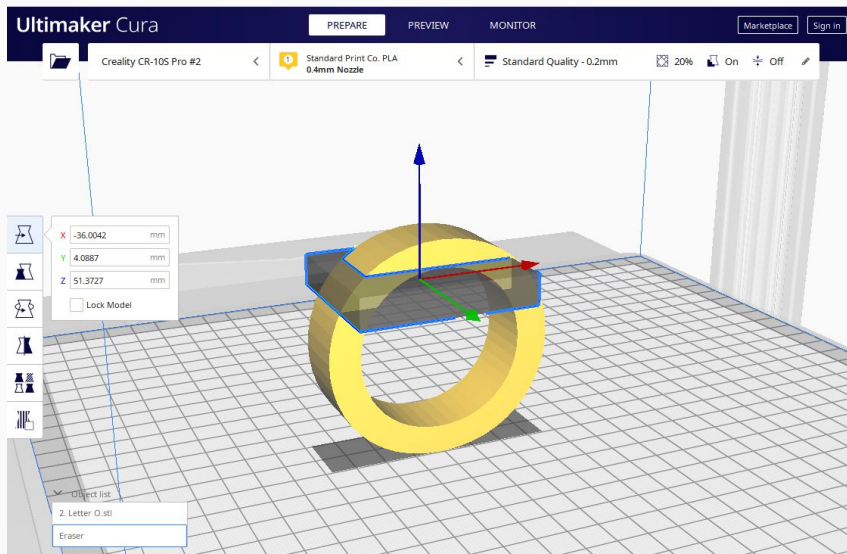
Note: When unsure of size and placement of support material is necessary, leave more support structures in place than necessary, keeping them under all overhangs that you're unsure of. Automatically generated support blocks will conform to the shape of your model and are optimised for each model according to the **Support Overhang Angle** setting listed under the **Support Dropdown Menu** in the **Print Settings Menu**.

Don't Know Where To Begin?

While there are many support geometries available in Creality CURA, try starting off by choosing **Zig Zag** with a **15% support density** for a robust support layer.

Walls And Infill

Walls

Determines the number of outer layers (and thickness) of a model that your printer prints. The greater the number of walls printed, the stronger the resulting model will be - at the cost of longer print times and increased material usage. As a general rule, models used for visual purposes can be printed with **2-3 walls**, while functional end-use models are often best printed with **4+** walls.

Infill

The infill density defines the amount of plastic used on the inside of the print. A higher infill density means that there is more plastic on the inside of your print, leading to a stronger object. A selection of infill densities can be visualised below.
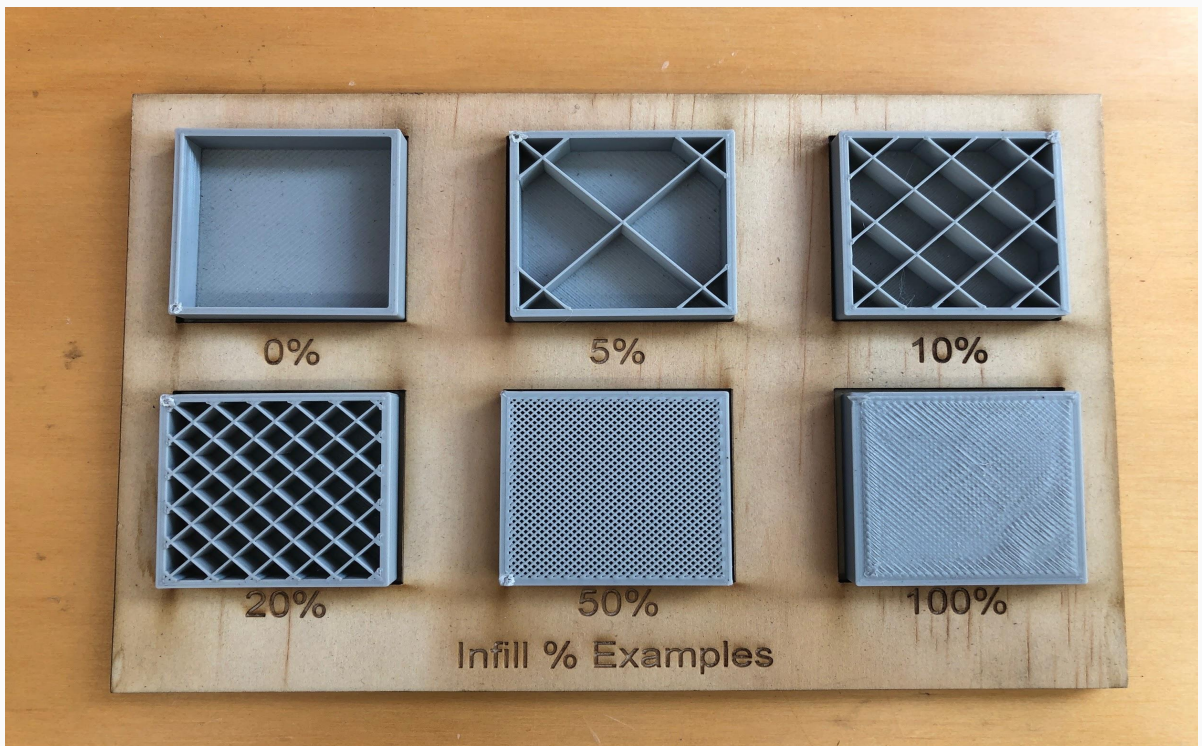
Infill   Pattern

The pattern of infill can be altered in Cura and can be accessed via the dropdown menu under the **Support > Support Pattern** tab.

Don't Know Where To Begin?
While there are many support and wall settings available, for a print with a great balance of durability and print economy, we suggest setting **Wall line count** of **3** and an **Infill Volume** of **30%.**

Final Checks & Exporting

Slicing

Once you are happy with your selected print settings, you will want to **Slice** your model by clicking **Slice** on the bottom right hand side of the window. This will tell Cura to generate a set of instructions for your printer called a **G-code**. Within this g-code contain all your choices and preferences that you have made for this specific print.
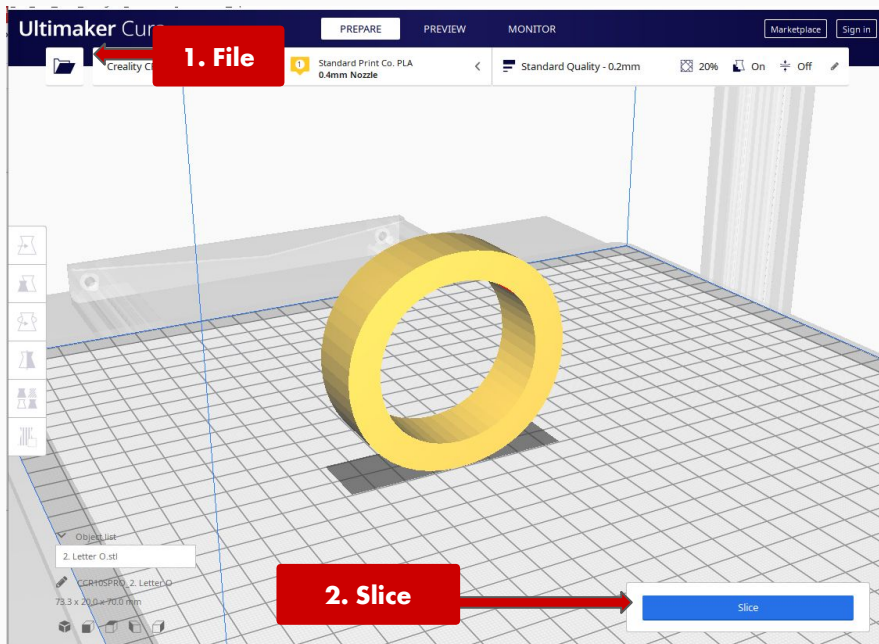
Tip: Saving a Project File (.3mf)

Once your have "Sliced" your model and exported your g-code, you will <u>not</u> be able to alter any of your preferences contained in your g-code file. Changes can only be made within Cura <u>before</u> the G-code is generated. When exporting a g-code file, it is recommend that you also save the project you are working on by **creating a project file.** This can be done by clicking **File > Save...**

We recommend you save the project file under the same name as your g-code file. By saving an accompanying project (.3mf) file along with your g-code (.gcode) files, you'll be able to quickly and easily make changes and generate another g-code file should you need to make any adjustments to the print settings.

Final Checks & Exporting

Layer View

After preparing your g-code you'll be able to exit **Solid View** and enter **Layer View** via the tab on the top left hand side of the viewing window. When in layer view, the entirety of your print job can be viewed, layer by layer using the **slider.**

Selecting **Colour Scheme** > **Line Type** via the drop down menu will allow you to visualise the entirety of your 3D print colourised by line type.
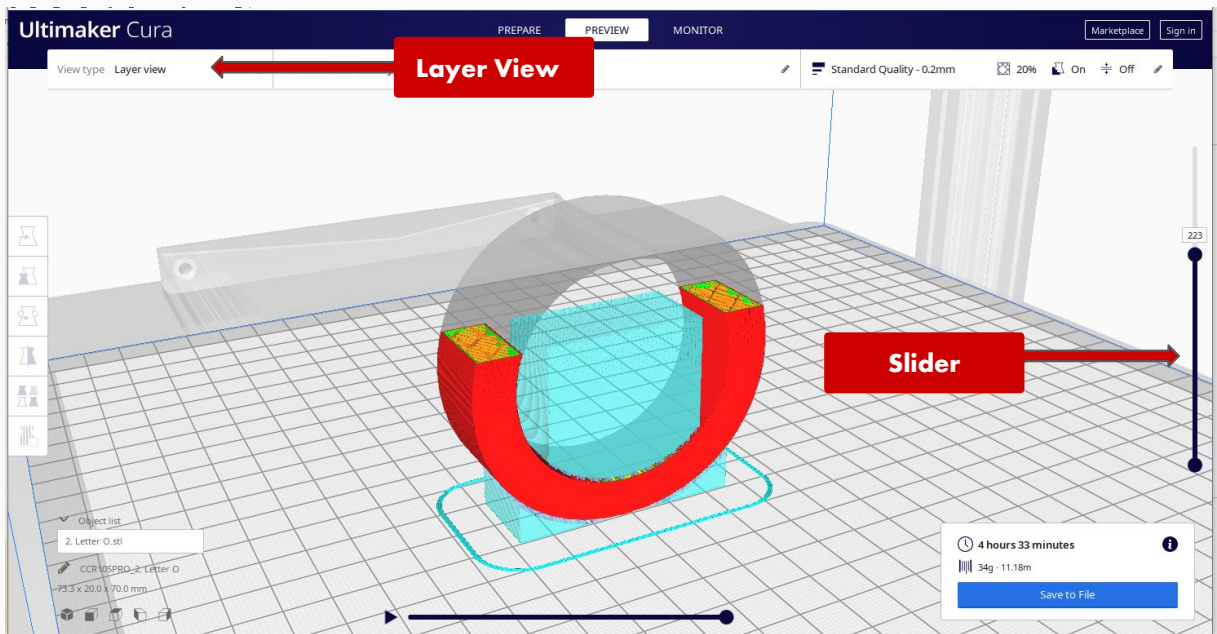
Final Checks

It is at this point, it's important to conduct a final check over your print job to check for any **unsupported overhangs** or other areas of the print that may cause issues printing. While the Creality 3D printer is able to print some unsupported overhangs (see Reorientation Of Your Print p.26) and does have an auto-support generation feature, it is still important for the operator to identify any problem areas that may arise. This is particularly important if custom supports have been used or if the **Support Overhang Angle** setting has been adjusted.

To conduct your final check, briefly slide the slider from 0 to the final layer, paying special attention to areas that are particularly steep and may need supporting while printing. If you do notice an unsupported spot, you may consider either manually generating a support block in **Layer View** or by decreasing the **Support Overhang Angle** number to increase the amount of support being generated.

Final Checks & Exporting

(Optional) Naming Convention

Aim to keep the names of your projects in an orderly manner before exporting your files. Using a version system (V1, V1.1, etc) or similar can be helpful in keeping g-code files organised should you subsequently need to adjust your print settings and to re-slice a g-code file.

It is also advised that you keep file names under 20 characters as long file names may cause display issues. Should a file you've saved to the Creality's memory card not be showing up, try shortening the file name and see if it shows up.

Exporting

Once your final checks have been completed, and you're satisfied that your print job is problem free, you can move on to saving your g-code to the Creality's SD Card. Click Save to file to export your g-code
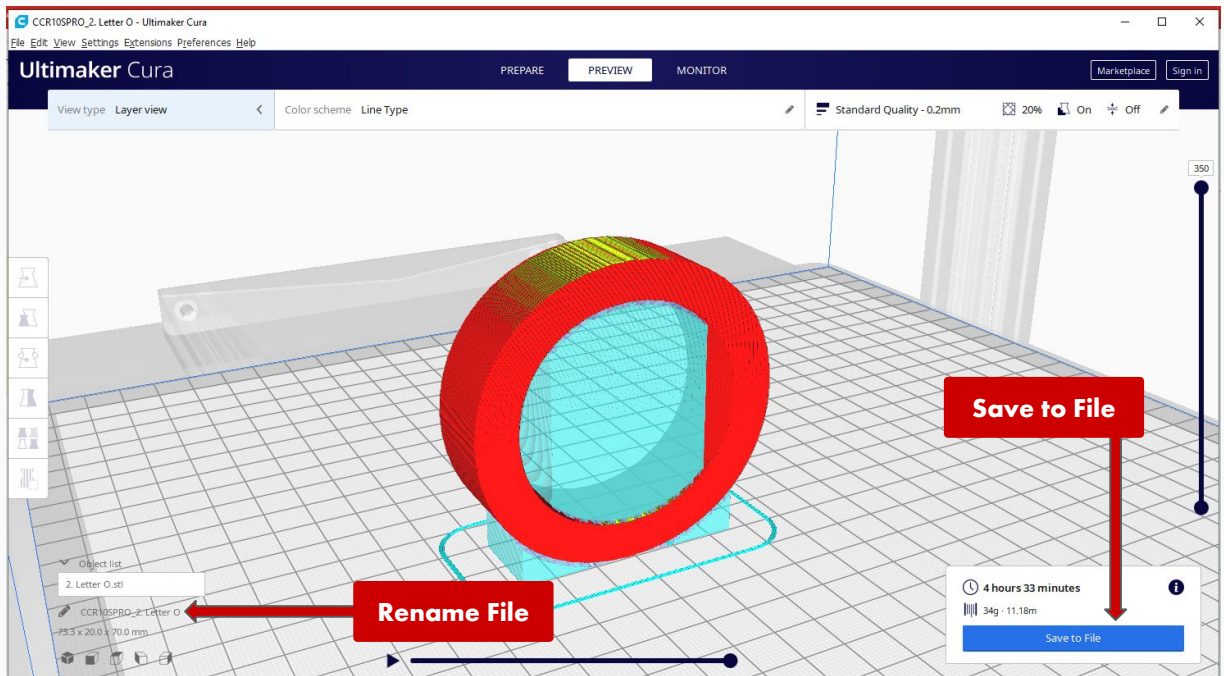
Part 3:
PRINTING AND TROUBLESHOOTING

Troubleshooting
Looking for the most up to date printing tips and guides we've prepared? Head over to our website and download your very own cheat sheets and print guides.

https://www.standardprintco.com/read

Assignments

Slicing 3D Models

Your colleague has tasked you with preparing the models found in **3. Test 3D Models** for 3D printing using the print profiles found in **4. Material and Print Settings**

1. Import the following files into your slicer. (one model per project file)
2. Manipulate the model into the best orientation for printing.
3. Generate the correct support structure for each model. (Either using auto, manual or no supports)
4. Apply the correct material and print profiles for each of the 3D models.
5. Slice the 3D model.
6. Conduct a final visual check of each model in Layer View.
7. Export a **g-code file (.gcode)** and a **Project file (.3mf)** for each model.
8. Repeat for all models.

Model	Material	Nozzle
Crankshaft	PLA	0.4mm
Letter O	ABS	0.6mm
Letter T	PETG	0.8mm
Extrusion Profile	PLA	0.4mm

When you have sliced and exported all g-code and project files (you should have 4 x g-code and 4 x project files in total), send them to our 3D printing team for review. Due to the large file sizes involved, you may wish to send them via the following options:

- Dropbox
- Wetransfer
- Manual Upload Via Our Website

If you are having trouble sending your completed assignment files, get in contact with us here and a member of our team would be happy to assist you.

Checklists
File | Materials | Slicing | Printer

File:

1. ▢ The file in .stl format
2. ▢ The file is "watertight" when designing

Material:

1. ▢ The correct material has been chosen for the project
2. ▢ The filament kept to filament in a dark and dry place and is in printable condition
3. ▢ The filament is not kinked, damaged or snapped.
4. ▢ The end of the filament is trimmed to a 45° Angle
5. ▢ There is no dust or debris on the spool or filament

Slicing:

1. ▢ The model is in contact with the print bed or auto support generation is enabled.
2. ▢ The correct **Nozzle Setting** have been selected
3. ▢ The correct **Material settings** have been selected
4. ▢ The correct **Print Profile** has been selected.
5. ▢ The all overhangs requiring support material have been generated
6. ▢ Final visual check has been completed in **Layer View**
7. ▢ Project labeled correctly
8. ▢ Saved Project file
9. ▢ Saved and exported G-Code

Printer:

1. ▢ The nozzle is in contact with the bed
2. ▢ The bed is clean and free of dust and debris
3. ▢ There is filament loaded into the printer and is extruding

Acknowledgements

Trademarks

Autodesk, Autodesk Inventor, Fusion 360, Autodesk 123D, Maya, Meshmixer, Mudbox, Tinkercad, and 3Ds MAX, are trademarks or registered trademarks of Autodesk, Inc.

Blender is a registered trademark of the Blender Foundation.

FFF and grabcad is a registered trademark of Stratasys, Inc.

Rhino is a registered trademark of Robert McNeel & Associates.

Solidworks is a registered trademark of Dassault Systèmes SolidWorks Corporation.

The Standard Print Co. "Extruder" logo, Standard Print Co. are trademarks of Standard Print Company Pty Ltd all rights reserved.

Thingiverse, and Thingiverse Education are trademarks or registered trademarks of MakerBot Industries, LLC. All rights reserved.

Cura is a registered trademark of Ultimaker BV. All rights reserved.

Screenshots

Blackbelt CURA software screenshots were used with permission from Blackbelt BV.

The Standard Print Co. An Introduction into 3D Printing is an official product of Standard Print Company Pty Ltd, and is not authorized, sponsored, associated with, or otherwise associated with any of the other parties listed above in this legal section or otherwise mentioned in this book.