Creatable D3 User Manual

VER. 1.0

Safety Warnings

(Hot Surface Hazard)

(For indoor use only)

(Caution)

- To avoid being burned, do not put your hands inside of the printer while it is in operation.
- It is recommended to use the printer indoors with adequate ventilation during and after operation.
- To avoid injury to finger or hands, keep your hands away from the interior of the printer during operation.
- The warnings contained herein are for your own physical safety, so please read all warnings carefully.
- We are not liable for injuries caused due to willful disregard of the warnings contained herein.
- We are not liable for damage caused to the printer by using filaments not officially approved for use on the Creatable Labs Marketplace.

Table of Contents

Specifications	**3**
Setup/Installation	**4**
Bed Leveling Instructions	**10**
Filament Loading / Unloading	**15**
Using your D3 for the first time	**17**
Filament Unloading	**19**
Filament Loading	**19**
Make a print with Cura	**20**
General rule	**45**
First layer	**45**
How to fix errors of modeling data	**46**
Maintaining your printer	**49**
Troubleshooting guide	**49**
Additional information	**50**

1. Specifications

a. Physical Dimensions
   i. Print Technology : Fused Filament Fabrication
   ii. Printer Size : 363L x 381W x 620H mm
   iii. Printer Weight : 10kg (22.0 lbs.)
   iv. Build plate : Soda-lime glass
   v. Build Volume : 250 (dia.) x 200H mm

b. Software
   i. Software Package : Cura CREATABLE Edition
   ii. Supported File Types : .stl, .obj, .amf, .dae
   iii. OS Support : Windows or MacOSX 10.6 and above
   iv. Connectivity : SD card(included), USB port

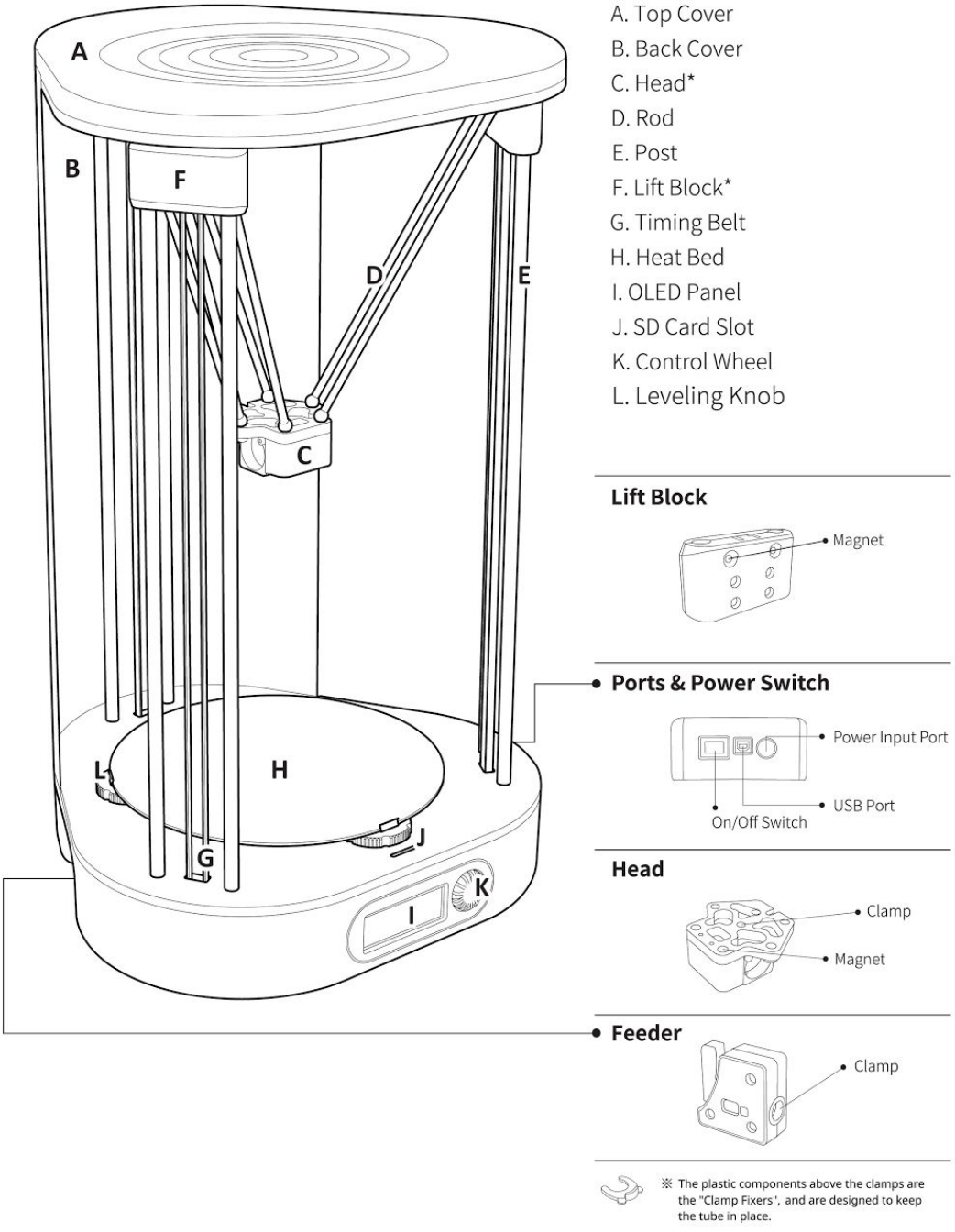
c. Temperature
   i. Ambient Operating Temperature : 15~32 °C (60~90 °F)
   ii. Storage Temperature : 0~32 °C (32~90 °F)
   iii. Heatbed Temperature : 20~125 °C
   iv. Nozzle Temperature : 20~260 °C

d. Head and Nozzle
   i. Layer Resolution
      - 0.25 mm Nozzle : 150 to 70 micron
      - 0.4 mm Nozzle : 300 to 70 micron
      - 0.6 mm Nozzle : 400 to 70 micron
      - 0.8 mm Nozzle : 600 to 70 micron
   ii. Nozzle Diameter
      - 0.4 mm (basic)
      - 0.25 / 0.6 / 0.8 mm (optional)
   iii. Filament Diameter : 1.75 mm

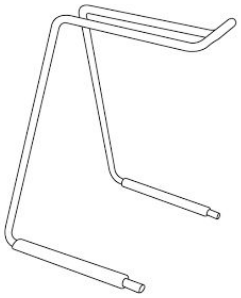
e. Electrical
   i. AC Input : 100-240 V, ~3 A, 50-60 Hz
   ii. Power Requirements : 24 V DC @ 9.16 A

2. Setup/Installation

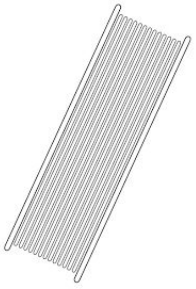
a. Parts Information & Components information



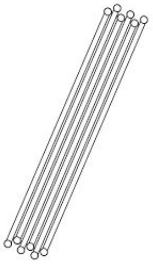
※ Please confirm that you have received all components listed in this section.



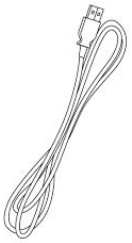
Filament Hanger



PLA Filament

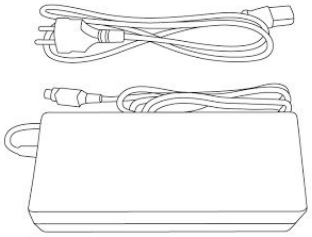


Rods



USB Cable

Quick Manual



Power Adapter / Power Cable

SD Card

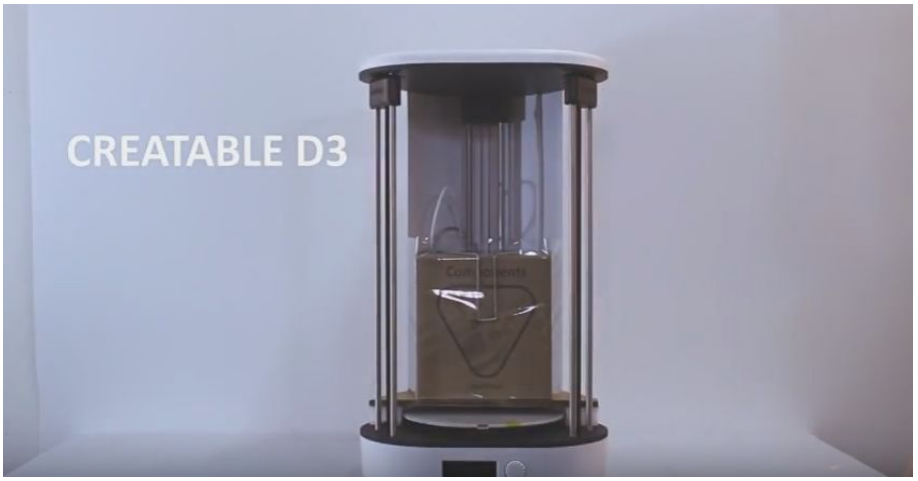
Hex Wrench

Glue Stick

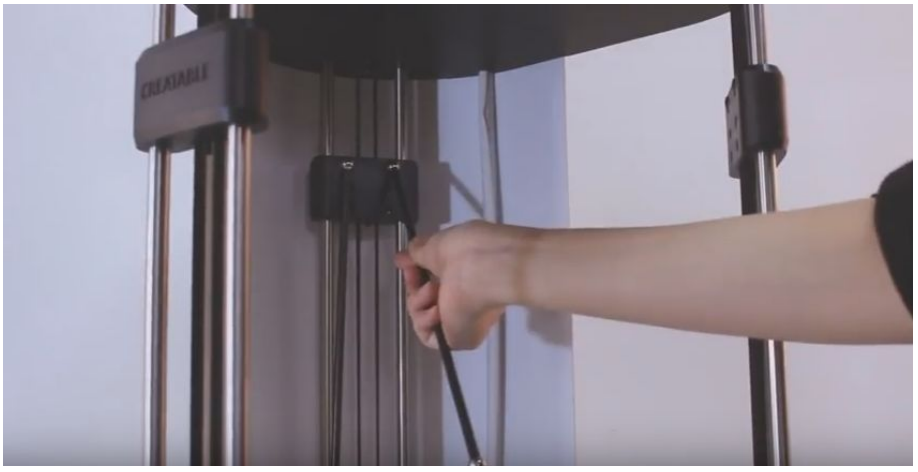
Bed Level Guide

b. Setup Instructions

i. Unboxing Creatable D3 and setup



1. Connect all 6 rods to each of the 6 magnetic joints on the lift blocks (both ends of the rods are identical)



2. After insertion of the filament tube into the print head, put the tube clamp in place to lock the tube in place. Pull on the tube lightly to ensure that the tube/clamp installation has been performed correctly.

3. On inspection of the power cable, you will notice that there is a rounded and flat portion to the cable. Ensure that the flat portion is facing down before attempting to connect the power cable to the printer, otherwise you will be unable to turn the printer on.

4. On the right side of the printer, ensure that the power button is in the 'OFF' position (the white stripe on the button should be raised).

5. Connect the power cable in the orientation mentioned in step 3 into the power port on the far right of the power button.

.

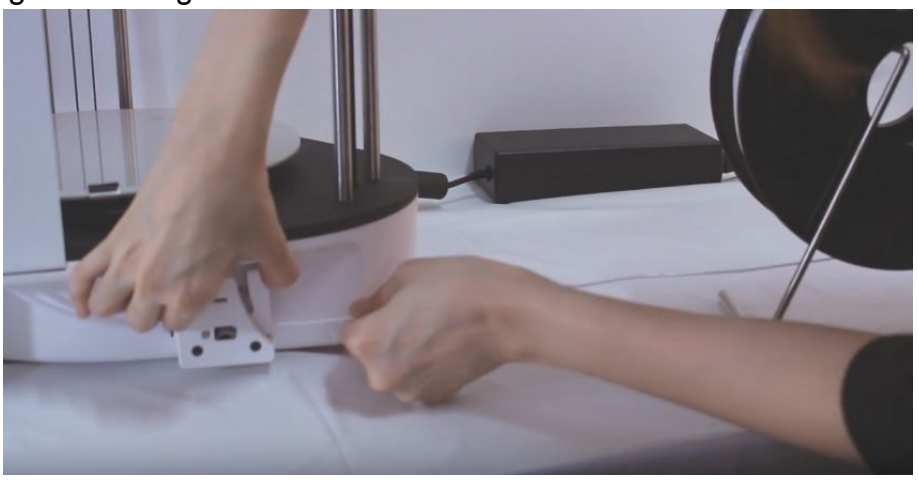
6. Once the power cable has been inserted, press the power power button to turn your printer on (after pressing, the white stripe on the button should be depressed)

7. Once the printer is turned on, you should see the printer’s current status, as well as the message, "CREATABLE D3 READY", on the OLED display.

ii. Power Cord and Filament Holder Installation

1. Suspend your filament spool on the filament holder, draw out a short length of filament, and cut the end of the filament so that the tip of the filament has a sharp point

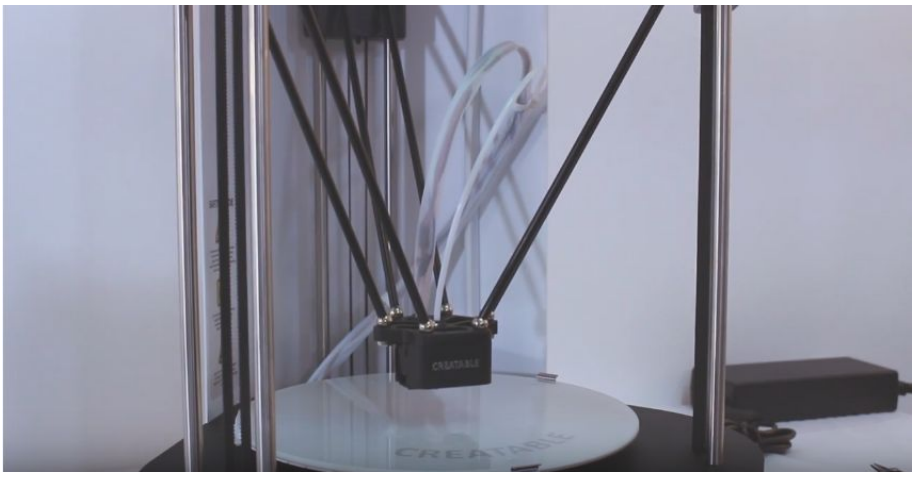
2. Push the lever fully to load the filament into the port. Make sure that the filament is positioned between the gear and bearing, and pushed far enough to be seen through the tubing on the other side of the feeder.



3. Bed Leveling Instructions

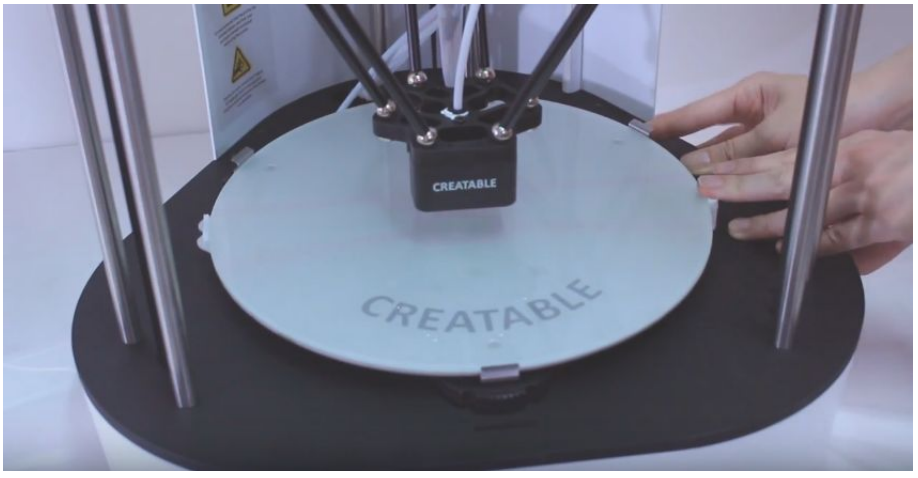
a. Once the OLED screen displays the startup message, press the rotating button twice to proceed to bed leveling.

b. The print head should move down towards the build plate.



c. Insert the leveling guide in the space between the build plate and the printer base, and turn the front leveling knob until leveling guide fits snuggly within that gap.

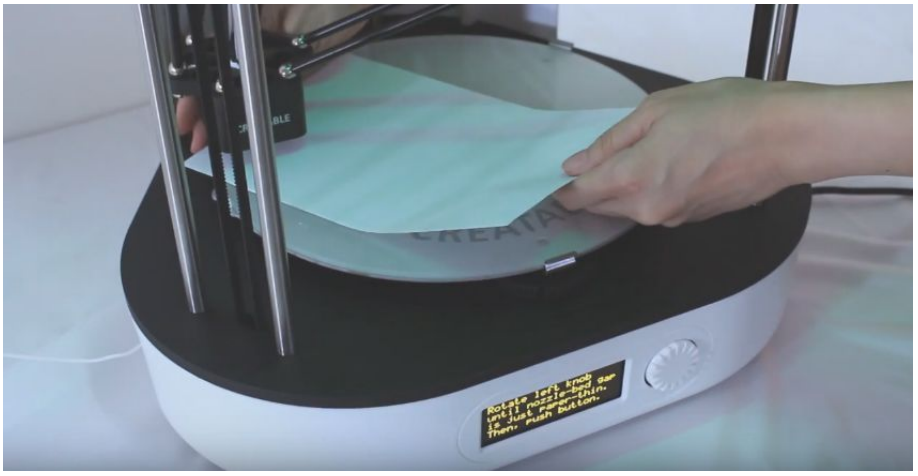
d. Repeat step (c) for the remaining two leveling knobs (ensuring that the leveling guide is inserted in a space close to each respective knob)



e. Upon pressing the button again, the print head should move into the 6 o’clock position.

f. Once the print head has stopped moving, place a sheet of A4 paper between the nozzle and the build plate. Adjust the leveling knob until there is a small amount of resistance when moving the sheet of paper back and forth between the nozzle and the build plate.

g. Repeat step (f) for the other two leveling knobs (pressing the button a second time will move the print head into the 10 o'clock position, and a third time will move the print head into the 2 o'clock position)



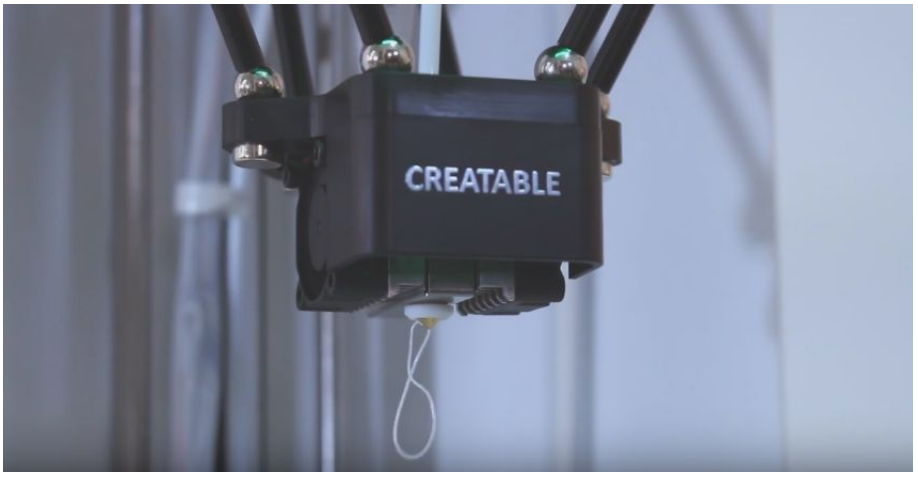
h. After the print head has come to rest at the 2 o'clock position, should you press the button once more, the print head will rise to the 'HOME' position and the leveling process will be complete.

4. Filament Loading / Unloading

a. After bed leveling is complete, you will see the message above on the OLED display. Press the button again to begin heating the nozzle of filament loading. During the heating process, you will see a message similar to the one shown in the image below.

b. Once the nozzle reaches the target temperature, the printer will automatically draw the filament in a certain length.

c. Once the feeder has stopped moving, rotate the wheel clockwise until you see filament being extruded from the nozzle (be careful not to click the button too early!)



d. Now the initial setup is finished! Enjoy printing!

5. Using your D3 for the first time

a. Locate the SD card included with the D3, remove it from it's case, and insert it into the SD card slot above the OLED display. If the SD card has been correctly inserted, you should see the message 'CARD INSERTED' on the OLED display. You are now ready to start printing!

b. Click the button, select ‘PRINT FROM SD’, then select your file from the options. When the nozzle and heatbed reach their designated temperatures, your print job will begin.

*When printing is complete, even though the printer may not be moving, the nozzle and heatbed may still be hot. Be cautious of burn-related injuries at all times.

6. Filament Unloading

a. Menu Selection Sequence

i. Press the main printer button

ii. From the main menu, select `Filament Change` then `Unload Filament`

iii. Once the nozzle has reached the pre-selected temperature, the feeder will begin unloading the filament.

iv. Push the lever fully with one hand and manually pull the filament out through the feeder port with your other hand to finish unloading the filament.

7. Filament Loading

*During the loading and unloading of filament, the nozzle becomes very hot (up to 350 °C). Be cautious of burn-related injuries at all times.

a. Loading Filament into the Feeder

i. Push the lever fully to load the filament into the port with one hand.

ii. With the other hand, carefully insert the filament into the feeder opening.

iii. Make sure that the filament is positioned between the gear and bearing, and pushed far enough to be seen through the tubing on the other side of the feeder.

b. Menu Selection Sequence

i. Press the main printer button.

ii. From the main menu, select `Filament Change` then `Load Filament`

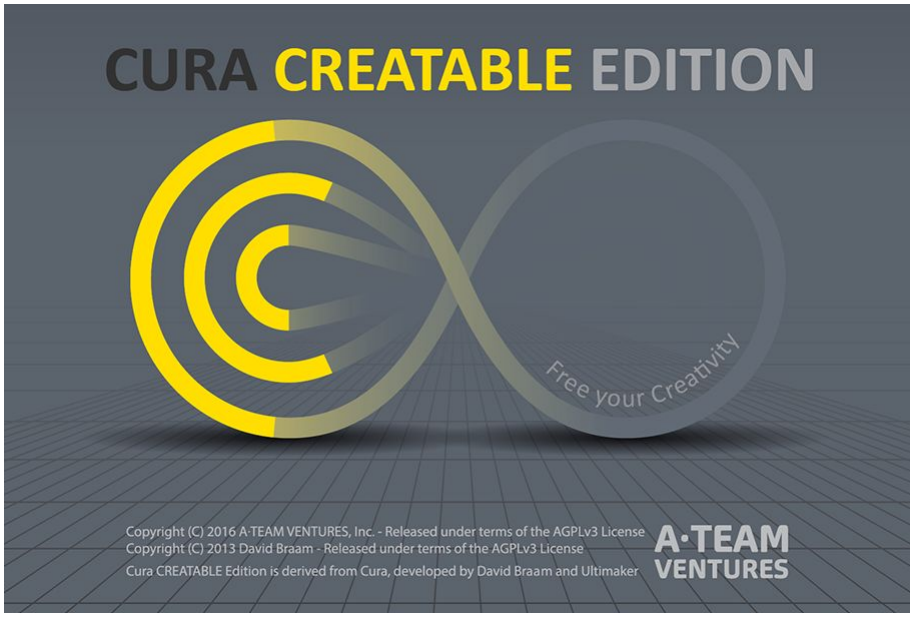
iii. Once the nozzle has reached the pre-selected temperature, the feeder will draw the filament close to the nozzle.

c. By rotating the control wheel, drive the filament towards the nozzle until you can see filament extruded out the other end. At this point, filament loading is complete. Press the main printer button to finalize the process and start cooling down.

8. Make a print with Cura

a. Installation

i. What is Cura CREATABLE Edition?



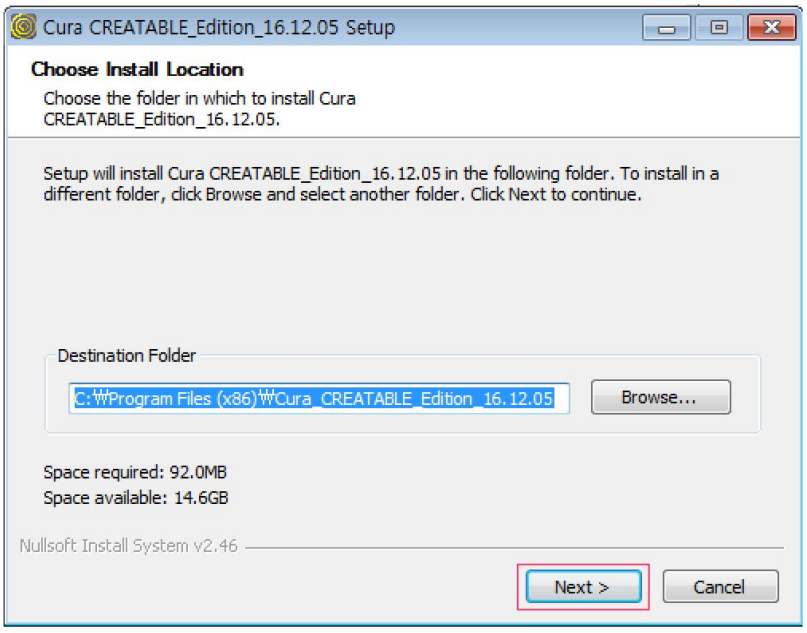
- Cura CREATABLE Edition is the dedicated 3D model slicing software to be used for all printers of the CREATABLE model.
- The role that slicing software performs is to convert 3D model data into instructions that your 3D printer can understand and execute.

ii. Specification

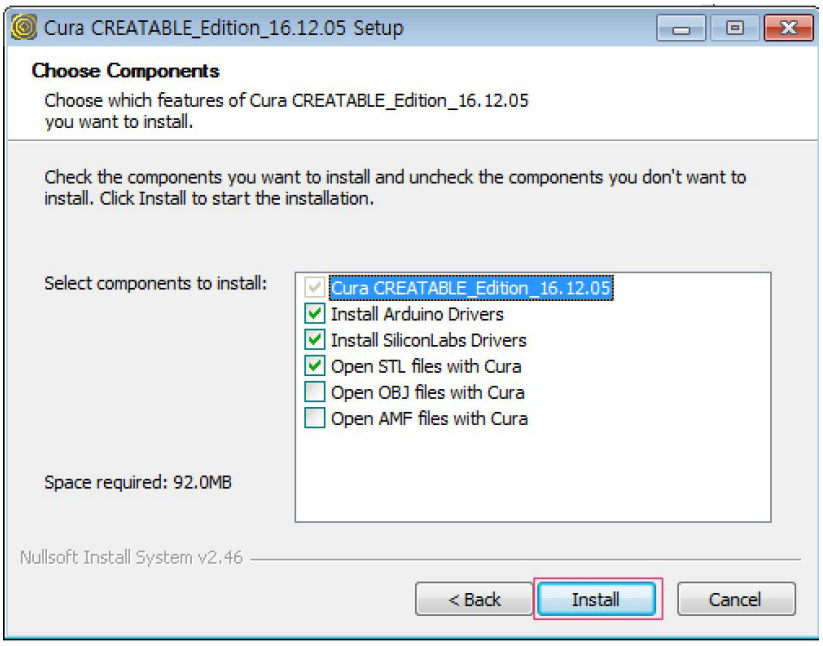
Cura CREATABLE Edition is compatible with Windows XP and above, and Mac OSX 10.6 and above

iii. Windows Installation

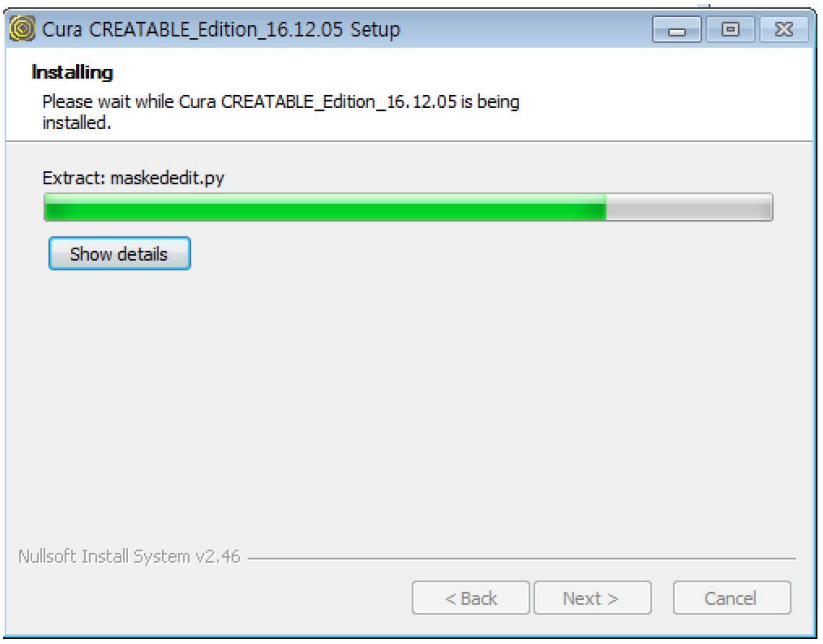
- Open the executable file once it has been downloaded.



- Click `Next` to move on to the next step.

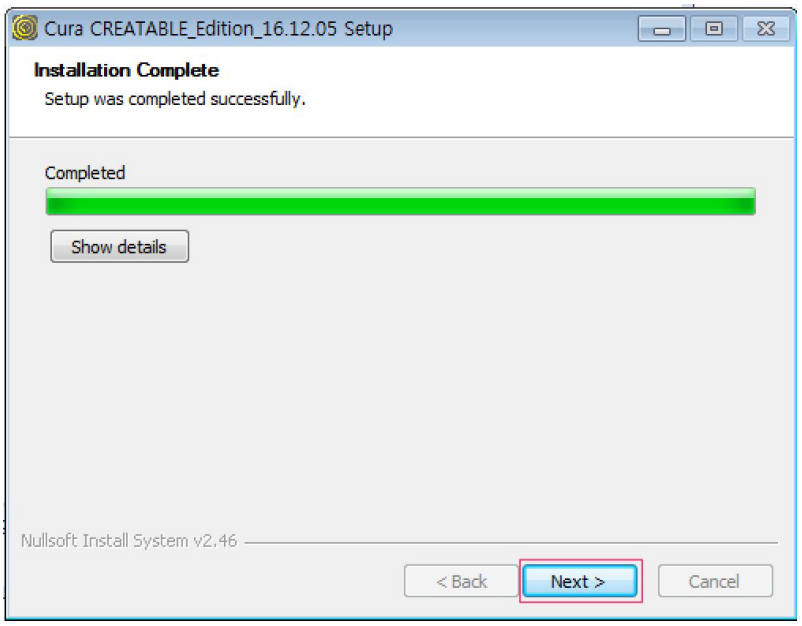


- The default settings should appear as in the image above. Press `Install` to continue to the next step.

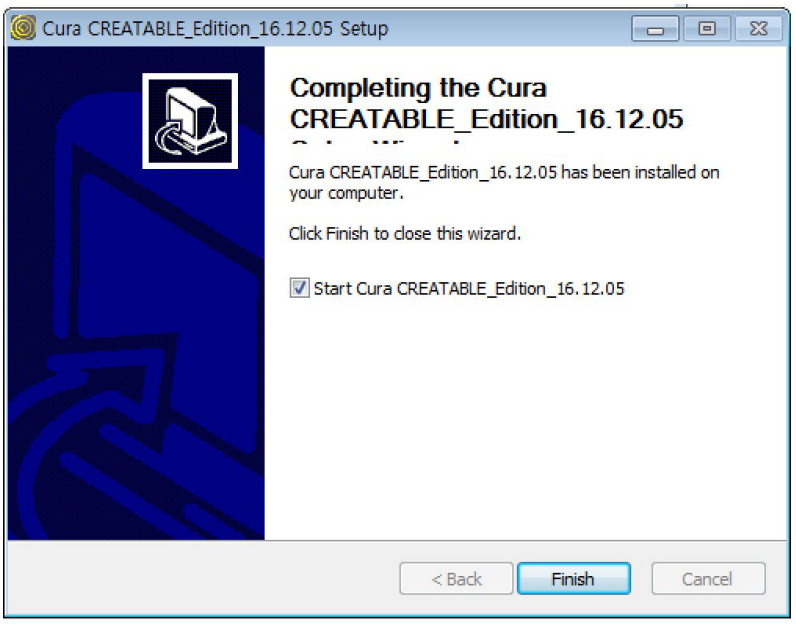


- Please wait until the installation has finished.
- After the software installation is complete, a window will appear concerning drivers you may need to install. Click `Next` to proceed to the next step.
- The installation of this driver is now complete. Press `Finish` to proceed to the next step.

- In order to start the installation of one more required driver, press `Next`.
- Select that you agree to the terms and conditions for this driver and press `Next` to proceed to the next step.
- At this point, installation of all required drivers should be complete, press `Finish` to complete the process.



- Press the `Next` button to proceed to the next step.

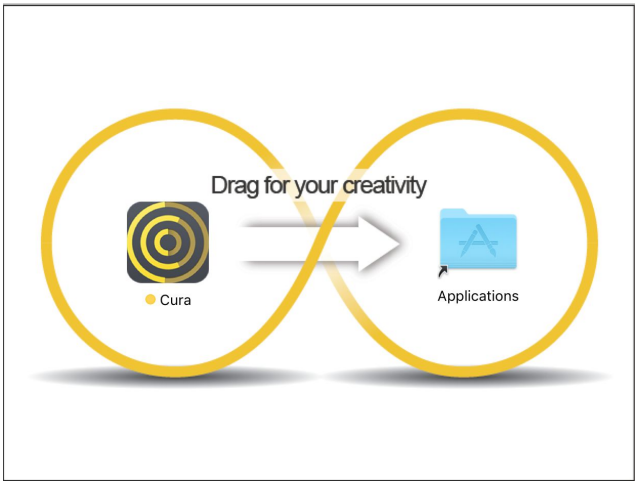


- Cura CREATABLE Edition installation has been completed. Press `Finish` to finalize the process.

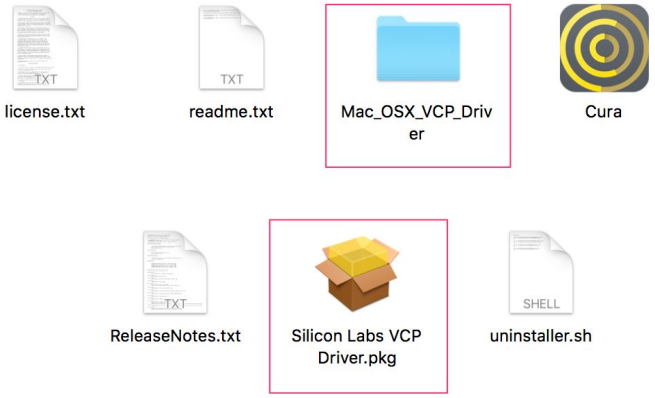
iv. Cura installation on Mac OSX

Cura_CREATABLE_Edition.dmg

- Double-clicking the downloaded Cura dmg file will mount the software for installation.



- Click and drag the Cura icon into your Applications folder.



- Navigate to the Cura folder. Inside that folder you should find another folder named `Mac_OSX_VCP_Driver`. Inside the `Mac_OSX_VCP_Driver` folder, double-click the Silicon Labs VCP Driver.pkg in order to initiate installation.

b. How to use the Cura Graphical Interface

i. Navigating and controlling the main window

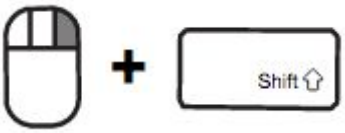
- Window Perspective Rotation

  - Utilize the arrow keys on your keyboard to rotate the perspective, or
  - Right-click, hold, and drag to utilize the mouse to rotate your perspective around the build plate rendering

- Zooming in and out on a particular point in your window

  - Scroll down and up on the mouse scroll wheel to zoom in and out

- Moving around in 3D space



  - While holding the shift key down, right-click and drag to move around in space with your perspective held constant

- Moving your 3D content

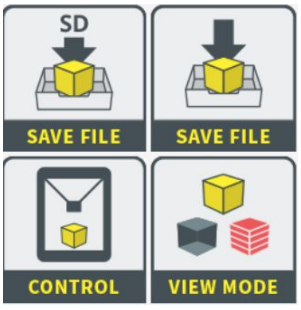
  - Right-click and hold to select your 3D content, and drag to move it to where you want it to be

ii. Working with 3D content in Cura

- Loading 3D content into Cura

- Click the `Load File` button and then select a file. Cura will automatically find an empty space on your build plate and render your 3D content there.

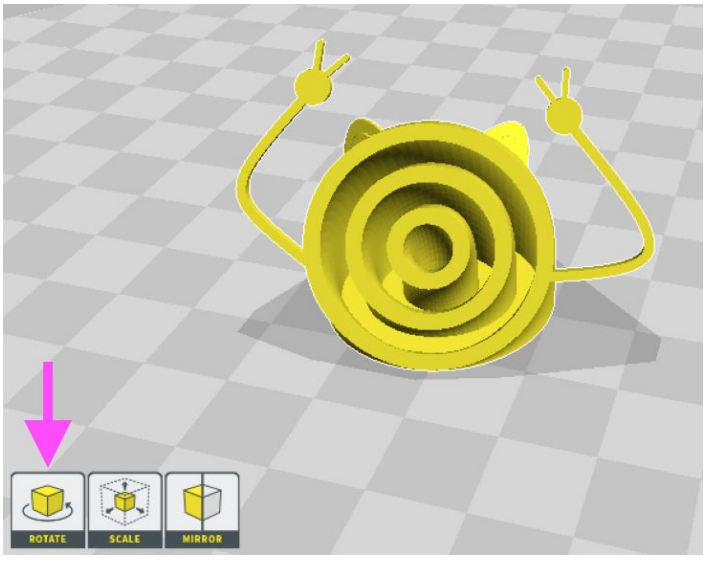
- Other useful buttons in the Cura graphical interface



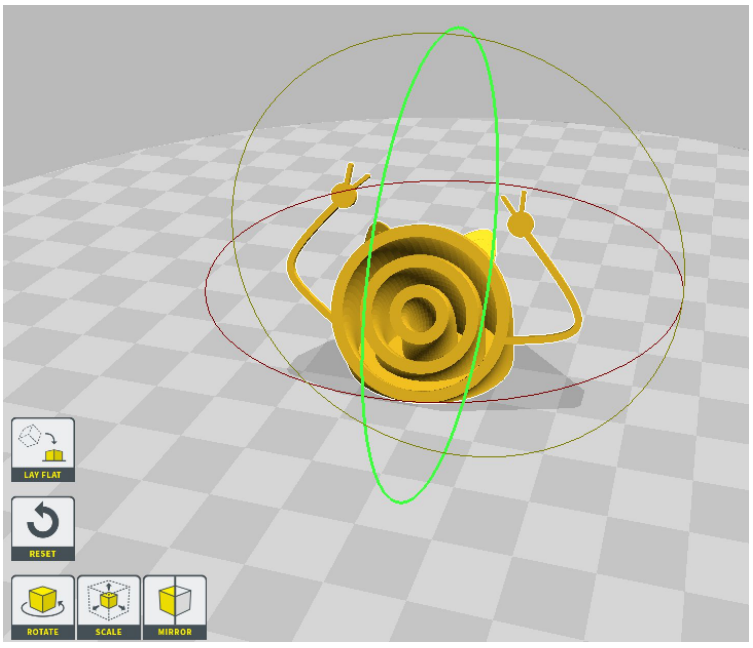
  - Once your 3D model file has been loaded into Cura, the following buttons will be available for you to use (from the top-left corner in clockwise fashion):
    - `SAVE FILE` to an SD card (if an SD card has been inserted)
    - `SAVE FILE` to a place on your computer
    - `CONTROL` your printer directly through the graphical interface
    - `VIEW MODE` for different ways to visualize your 3D content

iii. Rotating your 3D content

- Once your 3D content has been loaded into Cura, several new buttons will be available for you to use, one of which is the `ROTATE` button



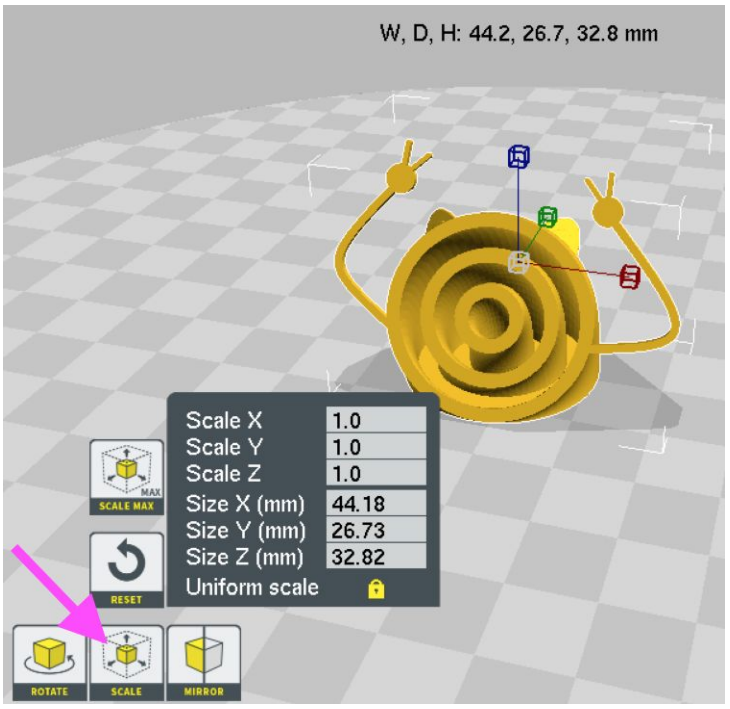
- If you click the `ROTATE` button, you will see three circles of different colors surrounding your 3D content. Clicking and dragging these circles will allow you to rotate your 3D content in either the x, y, or z-axis. For more precise, rotational control, hold the shift key down as you click and drag on these circles.



    - The `LAY FLAT` button adjusts the position and rotation of your 3D content in order to maximize the surface area contact between your 3D content and the build plate. This should should in theory make your 3D content easier to print!
    - The `RESET` button, as you might suspect, removes changes you've made to the 3D content, and restores its default settings when you first loaded it into Cura

iv. Adjusting the size and dimensions of your 3D content

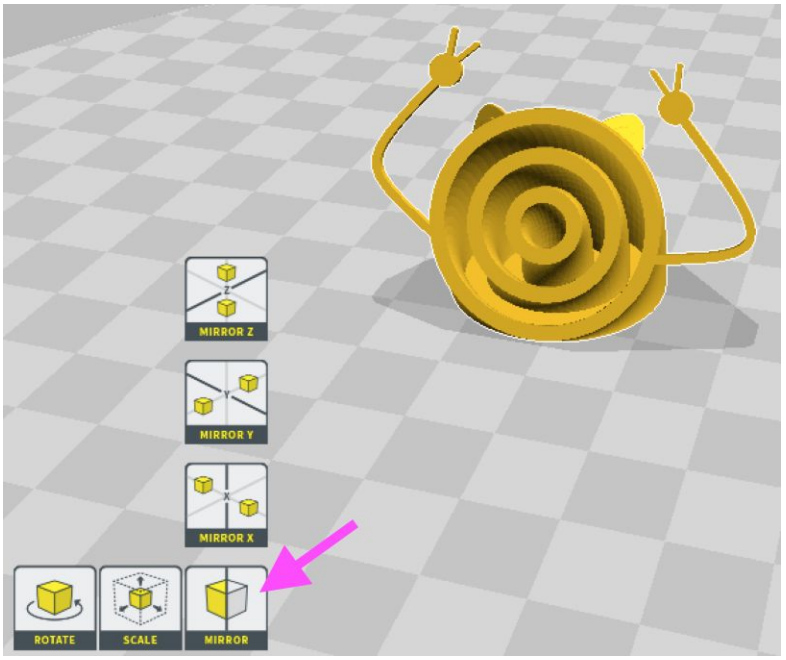
- After clicking on your 3D content, among the buttons that appear will be the `SCALE` button. If you click the `SCALE` button, handles should appear around your 3D content by which you can click and drag to resize that content.



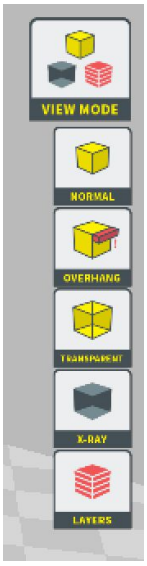
- As mentioned above, clicking and dragging any of the handles that appear around your 3D content will allow you to resize that content. Holding down the shift key while you click and drag will give you more precise control.
- You also have the option have editing the scale and the size through the numerical inputs in the popup window.
    a. When the *Uniform scale* lock icon is in the locked position, any change you make in the size or scale of the model will trigger changes in all the other dimensions to maintain the same, overall ratios of the model
    b. When the *Uniform scale* lock icon is in the open position, you can edit any scale or size in any one dimension without triggering a change in the other dimensions
- Clicking the `SCALE MAX` button will scale up the size of your 3D content to the maximum size without exceeding the build boundaries of your printer
- The `RESET` button, as you might suspect, removes changes you've made to the 3D content, and restores its default settings when you first loaded it into Cura

v. Performing symmetrical operations on your 3D content

- After you have clicked on your 3D content, among the buttons that appear will be the `MIRROR` button. Clicking on the `MIRROR` button will cause three additional mirror-operation buttons to appear, one for each 3D axis. As you might suspect, click on any of these specific mirror buttons will cause your 3D content to be mirrored on the other side of the plane that runs through that axis. Although a bit tricky to explain, a simple trial of each button will give you a clear idea of how the process works!



v. Utilizing the `VIEW MODE` button

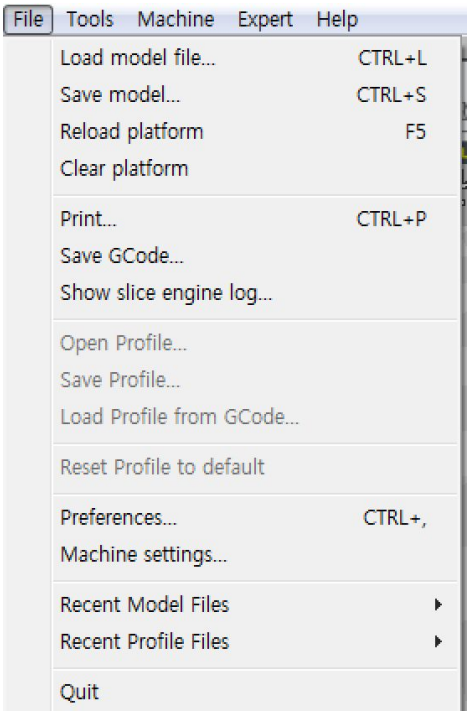


- Upon loading any 3D content into Cura, the `VIEW MODE` button will be made available for you to visualize your content in a variety of useful ways

- NORMAL: As the name would suggest, `NORMAL` mode renders your 3D content in way that imitates how you would see the finished, 3D printed object in real life
- OVERHANG: When `OVERHANG` mode is selected, Cura highlights parts of your 3D content that may require 3D printed “support material” in order for the overall object to be printed successfully
- TRANSPARENT: As the name would suggest, `TRANSPARENT` mode allows you to see somewhat through your 3D content
- X-RAY: When `X-RAY` mode is selected, from whatever perspective you are looking at your 3D content, the internals of that content, and the parts of that content directly opposite your point of view will be completely transparent
- LAYERS: When `LAYERS` mode is selected, each individual layer of the sliced 3D content is displayed. You are even allowed to simulate the printing process by rendering different stages of the layering process

c. The Cura Menu

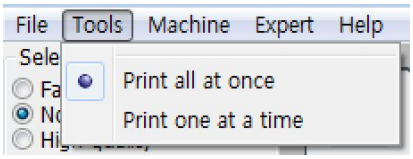
- Menu Tabs
  a. `File` tab



  - `Load model file…`: Select and load a 3D model file into Cura
  - `Save model…`: If you’ve made any changes to your 3D model, you have the option to save the changed version

- `Reload platform`: This option clears your build space and reloads all of the 3D content that was there previously
- `Clear platform`: This option clears your build space
- `Print…`: If your computer is connected to your 3D printer via a USB cable, this option allows you to start the print process once Cura is finished slicing your 3D content
- `Save GCode…`: Save the GCode Cura has generated to a location of your choice
- `Show slice engine log…`: This selection allows you to see what Cura is doing under the covers, a direct look at the output of CuraEngine which powers Cura
- `Open profile…`: This selection allows you load in an external slicer profile, and usually requires the file to be of the .ini extension
- `Save profile…`: This selection allows you to export your current slicer settings into an external file (usually of the .ini extension)
- `Load profile from GCode…`: This selection allows you to load the same slicer configurations from a previously generated GCode file
- `Reset profile to default`: As the name might suggest, this selection removes any changes you've made to the slicer configuration and restores all of the default settings
- `Preferences…`: This selection presents to you more options for customizing your Cura environment
- `Machine settings…`: This selection allows you to edit Cura settings related to your 3D printer
- `Recent model files`: Cura keeps track of the 3D model files you've loaded into Cura, and you can browse through the most recent ones through this selection
- `Recent profile files`: Cura keeps track of the slicer profile files you've loaded into Cura, and you can browse through the most recent ones through this selection
- `Quit`: This selection closes Cura CREATABLE Edition

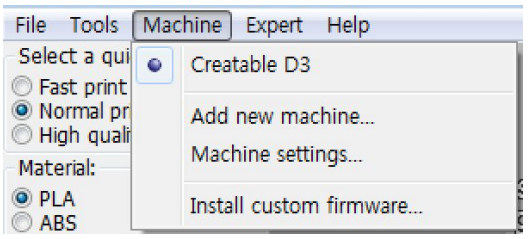
b. `Tools` tab



- `Print all at once`: Should you have multiple 3D model files loaded into Cura, and your printer is connected to your computer via USB cable, this selection allows you to print all of the loaded 3D models at once (in the same print job)

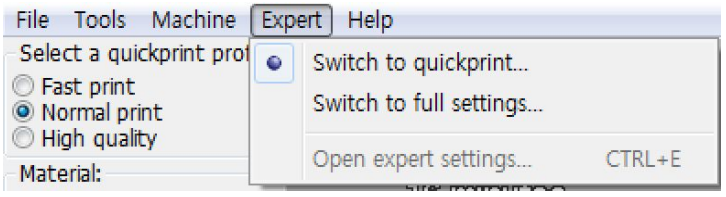
- `Print one at a time`: Should you have multiple 3D model files loaded into Cura, and your printer is connected to your computer via USB cable, this selection allows you to print each of the loaded 3D models one at a time (in separate print job)

c. `Machine` tab



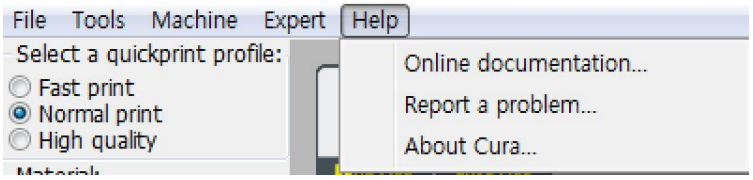
- Selecting the `Machine` tab will should you the list of 3D printer profiles you've already registered with Cura. Selecting any of these profiles will load the default slicer settings for that printer and adjust the Cura interface accordingly
- `Add new machine…`: This selection allows you to register a new 3D printer with Cura, and will guide you step by step through the process
- `Machine settings…`: This selection allows you to edit the Cura settings in detail for the printer currently selected
- `Install custom firmware…`: If your 3D printer is connected to your computer via USB cable, you can update that printer's firmware with this selection

d. `Expert` tab



- `Switch to quickprint…`: Quickprint mode shows you the bare minimum settings to get a print job started
- `Switch to full settings…`: Full settings mode allows you to fine tune your slicer configuration to the very last detail

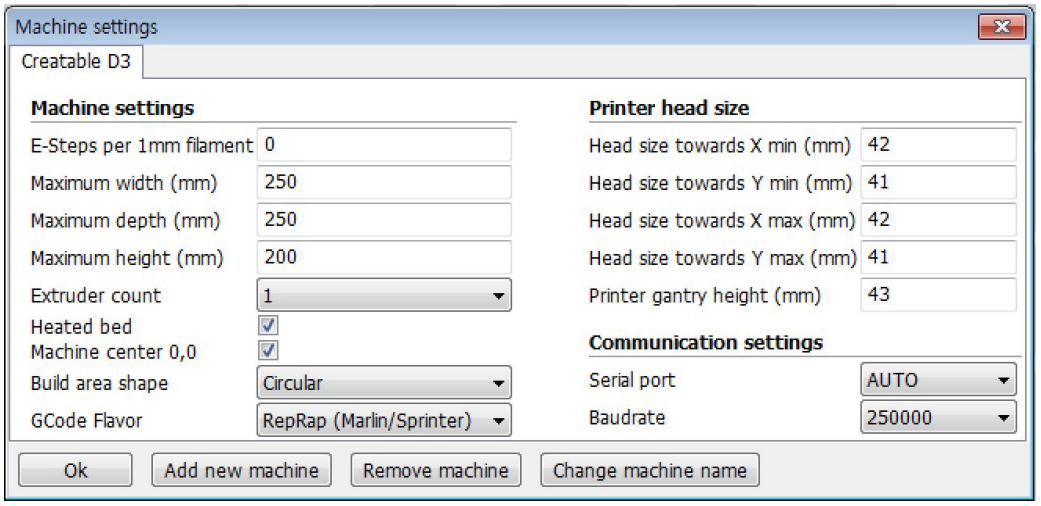
e. `Help` tab



- `Online documentation…`: This selection will forward you to the CREATABLE LABS website
- `Report a problem…`: This selection will forward you forward you to the CREATABLE LABS *contact us* page

- `About Cura…`: This selection will give you information about the Cura project

ii. Machine Settings Window

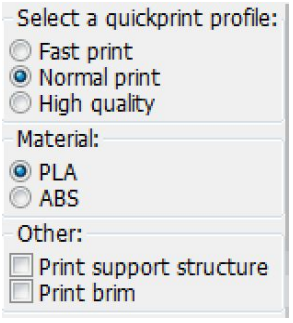


- Machine settings: This selection allows you to customize the Cura settings for the 3D printer model currently selected
  - `E-steps per 1mm filament`: This setting governs how many ‘steps’ the feeder motor needs to draw in 1 mm of filament (if a value of 0 is selected, the firmware default value is used instead)
  - `Maximum width` / `Maximum depth` / `Maximum height`: These values represent the maximum printable dimensions in the x, y, and z-axis
  - `Extruder count`: This value represent the maximum number of feeder motors available on the currently selected 3D printer model
  - `Heated bed`: This value indicates whether or not your printer has a heated build plate
  - `Machine center 0,0`: The value indicates whether or not the X/Y coordinates of the zero position of the printer is at the center of the printable area
  - `Build area shape`: This value indicates the shape of the build plate without taking unprintable areas into account (delta-based 3D printers have a circular build area shape)
  - `GCode Flavor`: This value indicates the type of gcode to be generated (the CREATABLE 3D printer model requires the RepRap (Marlin/Sprinter) flavor of GCode)
- Printer head size: If you only print one model at a time, the values to consider interference with other output.
  - Head size towards X/Y min/max : The minimum and maximum value between the printer head and the nozzle in the xy-plane

    - Printer gantry height : The height difference between the tip of the nozzle and the gantry system (X and Y axes)
- Communication settings : These settings are relevant should you connect to your printer directly via USB cable
    - `Serial port`: The port number used to connect to your printer
    - `Baudrate`: The connection speed between your computer and your printer
- `OK` : Click this button to save your changes
- `Add new machine`: Register a new printer model with Cura
- `Remove machine`: Remove a registered printer model with Cura (there must be at least one registered printer for this option to work)
- `Change machine name`: Rename a registered printer model

iii. Slicer Settings

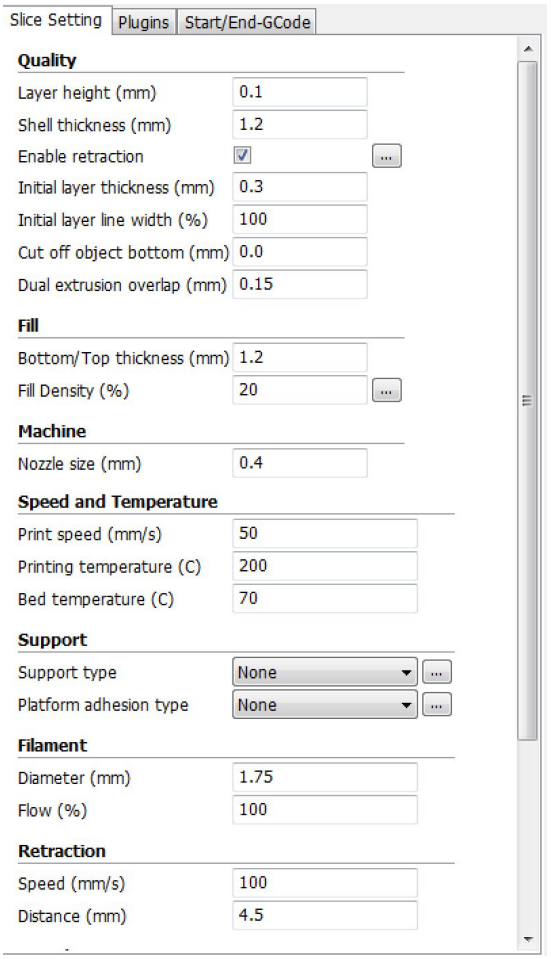
- When `Switch to quickprint…` is selected, Cura will show you the bare minimum information required to successfully print your 3D content



- Select a quick print profile: Pick among three quickprint profiles of varying quality levels and print speeds
    - `Fast print`: With this selection, the speed of the print job is maximized at the expense of printing detail (each printed layer is thicker than normal, making the layers a bit more noticeable)
    - `Normal print`: With this selection, Cura engine will select slicer settings that will correspond to what it judges to be a good compromise between print speed and quality
    - `High qualtiy`: With this selection, the quality of the print is maximized at the expense of printing speed (each printed layer is thinner than normal, making the layers even less noticeable)
- Material : Pick among two 3D printer filament materials with varying print temperatures and material properties
    - `PLA`: A biodegradable and bioactive thermoplastic derived from renewable resources, such as corn starch, tapioca roots, or sugarcane, typically used for packaging material, plastic wrap, plastic cups and plastic water bottles

    - `ABS`: An oil-based plastic, often used to create robust plastic objects for everyday use, for example in cars, electrical equipment or even in Lego bricks
- Other : Two more options are made available to you concerning support material and adhesion type
    - `Print support structure`: Selecting this option ensures that potentially problematic sections of your 3D print (with large amounts of overhang) have 3D printed support to ensure a successful print
    - `Print brim`: Selecting this option helps your print stick to the print bed by increasing surface area contact between your printed object and the print bed in the form of a paper-thin aura of additional printed material around your content

- Full settings: This mode allows you to fine tune your slicer configuration to the very last detail

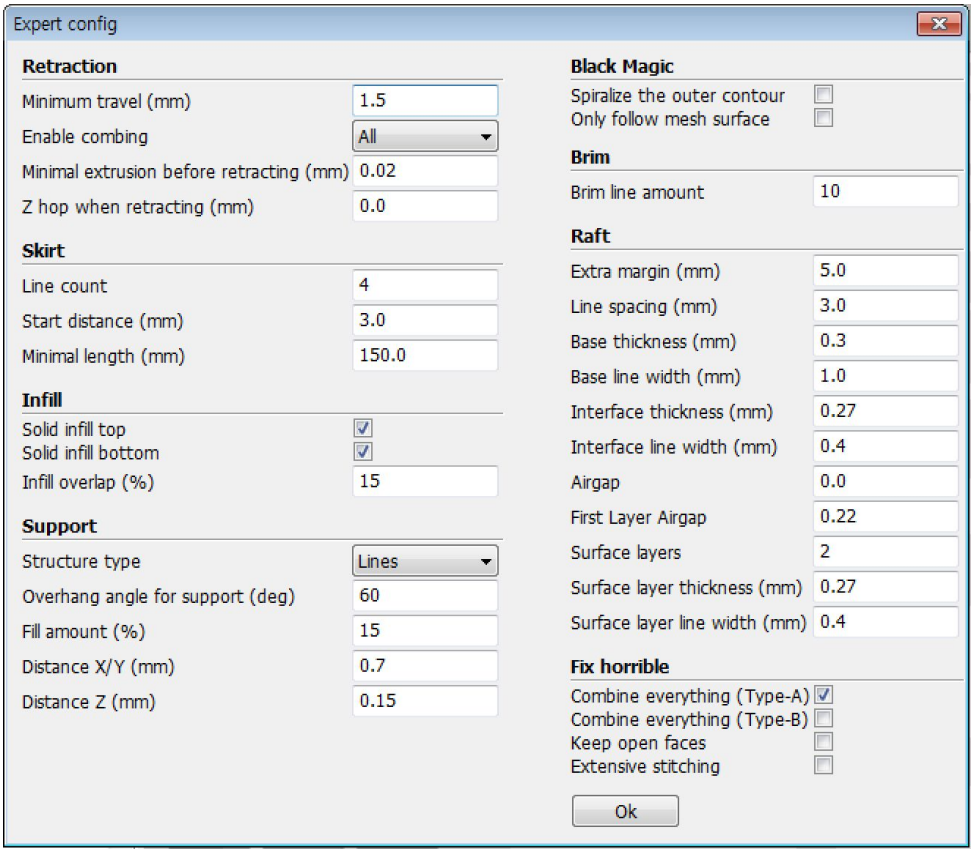


    - Quality
        - Layer height : The height of each layer in mm. Higher values produce faster prints in lower resolution, lower values produce slower prints in higher resolution

        - Shell thickness : The thickness of the walls in the horizontal direction. This value divided by the wall line width defines the number of walls
        - Enable retraction : Retract the filament when the nozzle is moving over a non-printed area
        - Initial layer thickness : The height of the initial layer in mm. A thicker initial layer makes adhesion to the build plate easier
        - Initial layer line width : Width of a single line in the initial printed layer. Generally, the width of each line should correspond to the width of the nozzle. However, slightly reducing this value could produce better prints
    - Fill
        - Bottom/Top thickness : The thickness of the top/bottom layers in the print. This value divided by the layer height defines the number of top/bottom layers
        - Fill Density : Adjusts the density of infill of the print (an infill of 0 generates a print of only the outer layer)
    - Machine
        - Nozzle size : The inner diameter of the nozzle. Change this setting when using a non-standard nozzle size
    - Speed and Temperature
        - Print speed : The speed at which printing happens
        - Printing temperature : The temperature used for printing
        - Bed temperature : The temperature used for the heated build plate. If this is 0, the bed will not heat up for this print
    - Support
        - Support type: Adjusts the placement of the support structures. The placement can be set to touching build plate or everywhere. When set to everywhere the support structures will also be printed on the model
            - None: No support is printed at all
            - Touching buildplate : Cura makes sure only to print support that has direct contact with the buildplate on one end
            - Everywhere : Cura prints support for all potentially problematic areas regardless of whether or not there is direct contact with the buildplate on one end

        - Platform adhesion type : Different options that help to improve both priming your extrusion and adhesion to the build plate.
            - None : No additional adhesion methods are used
            - Brim : Brim adds a single layer flat area around the base of your model to prevent warping.
            - Raft : Raft adds a thick grid with a roof below the model.
            - Skirt: Skirt is a line printed around the model, but not connected to the model
    - Filament
        - Diameter : Adjusts the diameter of the filament used. Match this value with the diameter of the used filament
        - Flow : Flow compensation, the amount of material extruded is multiplied by this value
    - Retraction
        - Speed : The speed at which the filament is retracted and primed during a retraction move
        - Distance : The length of material retracted during a retraction move
    - Speed
        - Travel speed : The speed at which travel moves are made
        - Bottom layer speed : The speed at which bottom layers are printed
        - Infill speed : The speed at which infill is printed
        - Top/bottom speed : The speed at which top/bottom layers are printed
        - Outer shell speed : The speed at which the outermost walls are printed. Printing the outer wall at a lower speed improves the final skin quality. However, having a large difference between the inner wall speed and the outer wall speed will affect quality in a negative way
        - Inner shell speed : The speed at which all inner walls are printed. Printing the inner wall faster than the outer wall will reduce printing time. It works well to set this in between the outer wall speed and the infill speed

iv. Expert Settings



- Retraction : These settings govern filament retraction when the nozzle is moving over a non-printed area
    - Minimum travel : The minimum distance of travel needed for a retraction to happen at all. This helps to get fewer retractions in a small area
    - Enable combing : Combing keeps the nozzle within already printed areas when traveling. This results in slightly longer travel moves but reduces the need for retractions. If combing is off, the material will retract and the nozzle moves in a straight line to the next point. It is also possible to avoid combing over top/bottom skin areas by combing within the infill only
    - Z hop when retracting : Whenever a retraction is done, the build plate is lowered to create clearance between the nozzle and the print. It prevents the nozzle from hitting the print during travel moves, reducing the chance to knock the print from the build plate
- Skirt : Skirt is a line printed around the model, but not connected to the model
    - Line count : Multiple skirt lines help to prime your extrusion better for small models. Setting this to 0 will disable the skirt

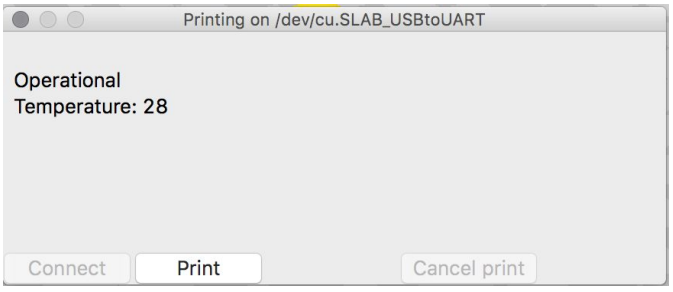
    - Start distance : The horizontal distance between the skirt and the first layer of the print. This is the minimum distance, multiple skirt lines will extend outwards from this distance
    - Minimal length : The minimum length of the skirt. If this length is not reached by all skirt lines together, more skirt lines will be added until the minimum length is reached. Note: If the line count is set to 0 this is ignored
- Infill : This setting adjusts the density of infill of the print
    - Infill overlap : The amount of overlap between the infill and the walls. A slight overlap allows the walls to connect firmly to the infill
- Support : Support material ensures that the risk potentially problematic overhangs jeopardizing your prints is mitigated
    - Structure type : Adjusts the placement of the support structures. The placement can be set to touching build plate or everywhere. When set to everywhere the support structures will also be printed on the model
        - Grid : This selection utilizes a grid pattern for the support structures of the print. The different options available result in sturdy or easy to remove support
        - Lines : This selection utilizes a line pattern for the support structures of the print. The different options available result in sturdy or easy to remove support
    - Overhang angle for support : The minimum angle of overhangs for which support is added. At a value of 0° all overhangs are supported, 90° will not provide any support
    - Fill amount : This setting governs how dense your support structures are
    - Distance X/Y : Distance of the support structure from the print in the X/Y directions
    - Distance Z : Distance from the top/bottom of the support structure to the print. This gap provides clearance to remove the supports after the model is printed. This value is rounded up to a multiple of the layer height
- Black Magic
    - Spiralize the outer contour : Spiralize smooths out the Z move of the outer edge. This will create a steady Z increase over the whole print. This feature turns a solid model into a single walled print with a solid bottom. This feature should only be enabled when each layer only contains a single part. Smooth the spiralized contours to reduce the visibility of the Z seam (the Z-seam should be barely visible on the print but will still be

visible in the layer view). Note that smoothing will tend to blur fine surface details
    - Only follow mesh surface : Treat the model as a surface only, a volume, or volumes with loose surfaces. The normal print mode only prints enclosed volumes. "Surface" prints a single wall tracing the mesh surface with no infill and no top/bottom skin. "Both" prints enclosed volumes like normal and any remaining polygons as surfaces
- Brim : Brim adds a single layer flat area around the base of your model to prevent warping
    - Brim line amount : The number of lines used for a brim. More brim lines enhance adhesion to the build plate, but also reduces the effective print area
- Raft : Raft adds a thick grid with a roof below the model
    - Extra margin: If the raft is enabled, this is the extra raft area around the model which is also given a raft. Increasing this margin will create a stronger raft while using more material and leaving less area for your print
    - Line spacing : Width of the lines in the top surface of the raft. These can be thin lines so that the top of the raft becomes smooth
    - Base thickness : Layer thickness of the base raft layer. This should be a thick layer which sticks firmly to the printer build plate
    - Base line width : Width of the lines in the base raft layer. These should be thick lines to assist in build plate adhesion
    - Interface thickness : Layer thickness of the middle raft layer
    - Interface line width : Width of the lines in the middle raft layer. Making the second layer extrude more causes the lines to stick to the build plate
    - Airgap : The gap between the final raft layer and the first layer of the model. Only the first layer is raised by this amount to lower the bonding between the raft layer and the model. Makes it easier to peel off the raft
    - Surface layers : The number of top layers on top of the 2nd raft layer. These are fully filled layers that the model sits on. 2 layers result in a smoother top surface than 1
- Fix Horrible
    - Combine everything : Ignore the internal geometry arising from overlapping volumes within a mesh and print the volumes as one. This may cause unintended internal cavities to disappear

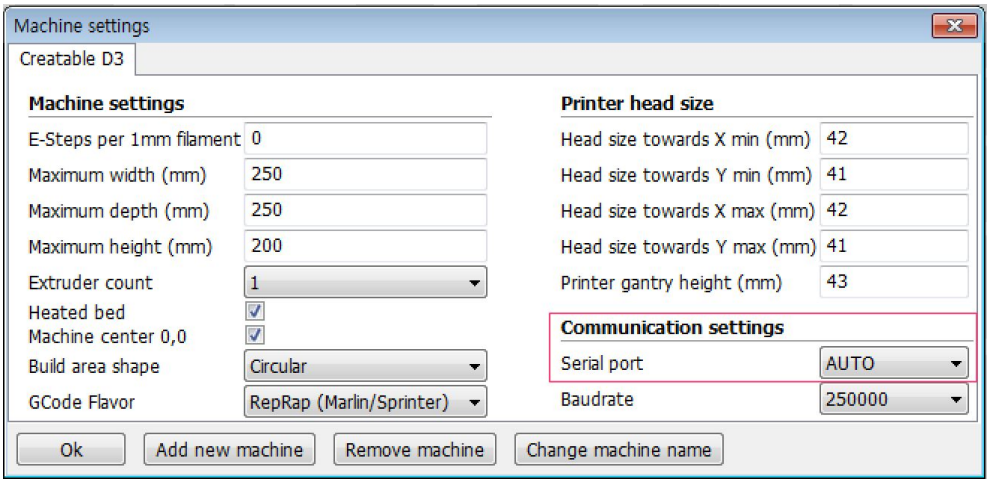
- Keep open faces : Normally Cura tries to stitch up small holes in the mesh and remove parts of a layer with big holes. Enabling this option keeps those parts which cannot be stitched. This option should be used as a last resort option when everything else fails to produce proper GCode
- Extensive stitching : Extensive stitching tries to stitch up open holes in the mesh by closing the hole with touching polygons. This option can introduce a lot of processing time

Starting a print through your computer

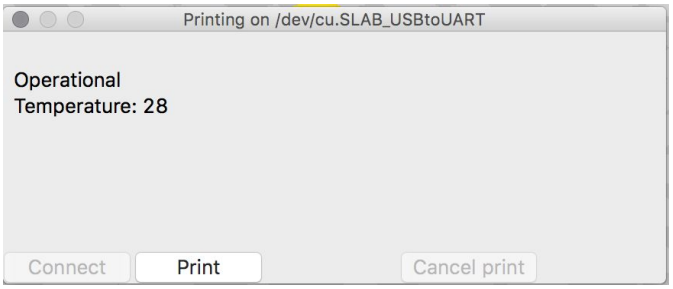
- Connect your computer to your printer via USB cable



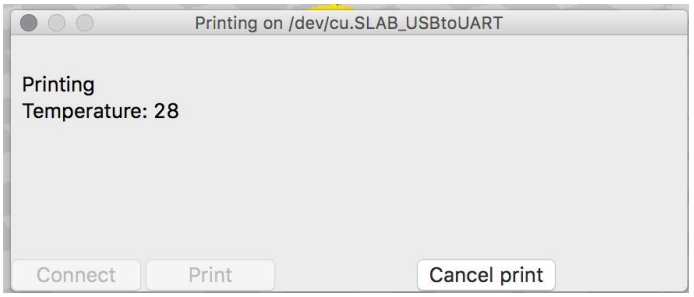
- Pressing the `Connect` button will begin communication with your printer (the window that you see may be different from the one above)



- In the case that automatic connection does not work, from the menu bar select `Machine`, then `Machine settings`, and select your `Serial port` manually



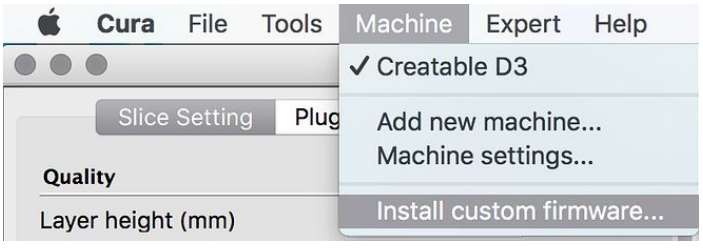
- Once your printer is successfully connected, pressing the `Print` button will initiate the printing process (the window that you see may be different from the one above)



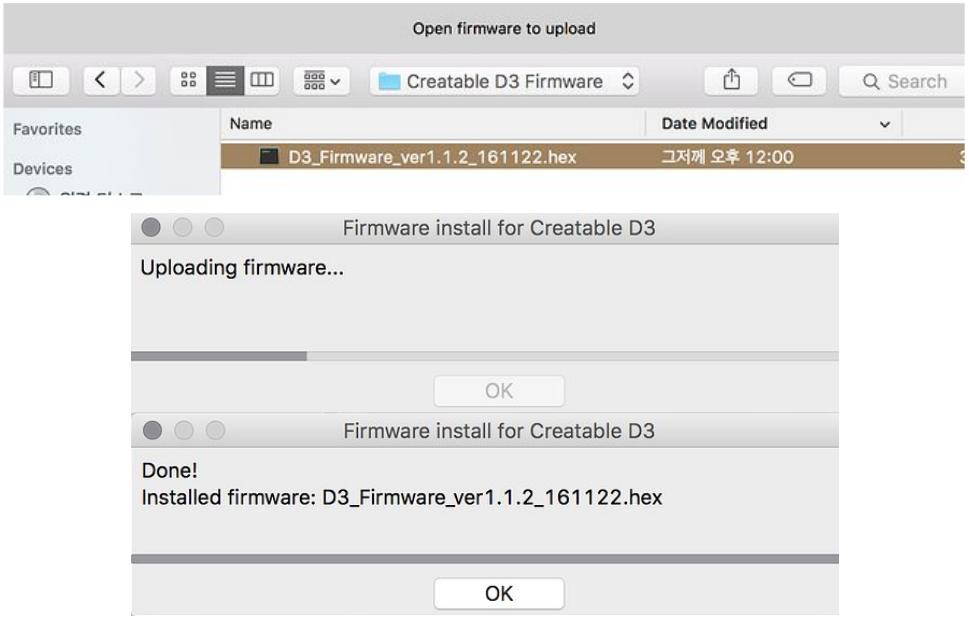
- Should there be connection problems between your printer and your computer during the printing process, the print job may be interrupted, and thus caution is advised when printing this way (the window that you see may be different from the one above)

e. Firmware Update Procedure

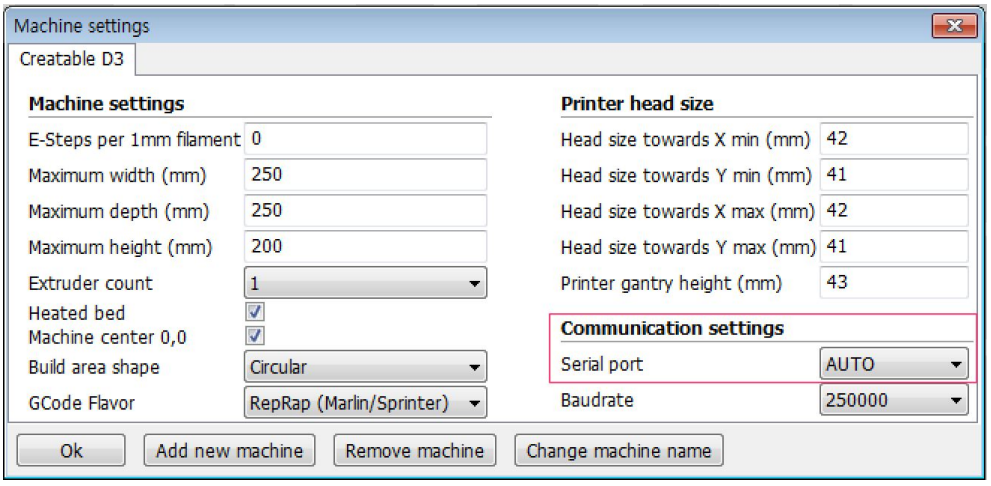
Connect your computer to your printer via USB cable



From the menu bar select `Machine`, then `Install custom firmware`



Upon selecting the firmware file you desire for the update, firmware upload should begin (the window that you see may be different from the one above)



- In the case that automatic connection does not work, from the menu bar select `Machine`, then `Machine settings`, and select your `Serial port` manually

Once the firmware update is complete, go to the printer main menu on the OLED display, then select `INFO/SETTINGS`, then `RESTORE DEFAULT` to finalize the update

A Guide to Successful 3D Printing

Unlike typical 2D paper printers, 3D printers oftentimes require more attention and care to ensure a successful print. Although the causes behind a failed print are varied and unique to each printer, the following list includes several of the most common categories of issues you can look out for.

- Mechanical Parts: issues with the nozzle, extruder, feeder, fan and/or heat bed
- Slicer: incorrect 3D model slicing parameters selected in the software
- Filament: material-specific issues when using a new 3D printing filament
- Modeling files: issues caused by error or defects in the source 3D model file

General rule

Here are a few rule of thumb to consider when attempting to make your first 3D print:

1. Try printing a simple 3D model : Rather than trying to print a large, complicated model, print out a model that is small, and has a simple shape. If the 3D print is successful, continue to practice printing by changing up some of the slicing options
2. Make careful observations: FDM 3D printers melt filament and build up your 3D model one layer at a time. Therefore, if the first layer is not properly printed and does not stick well to the bed, the next layer will most likely face the same problems. It is essential to make sure that the first layer is printed successfully

First layer

Use this checklist to maximize your chances of a successful first layer:

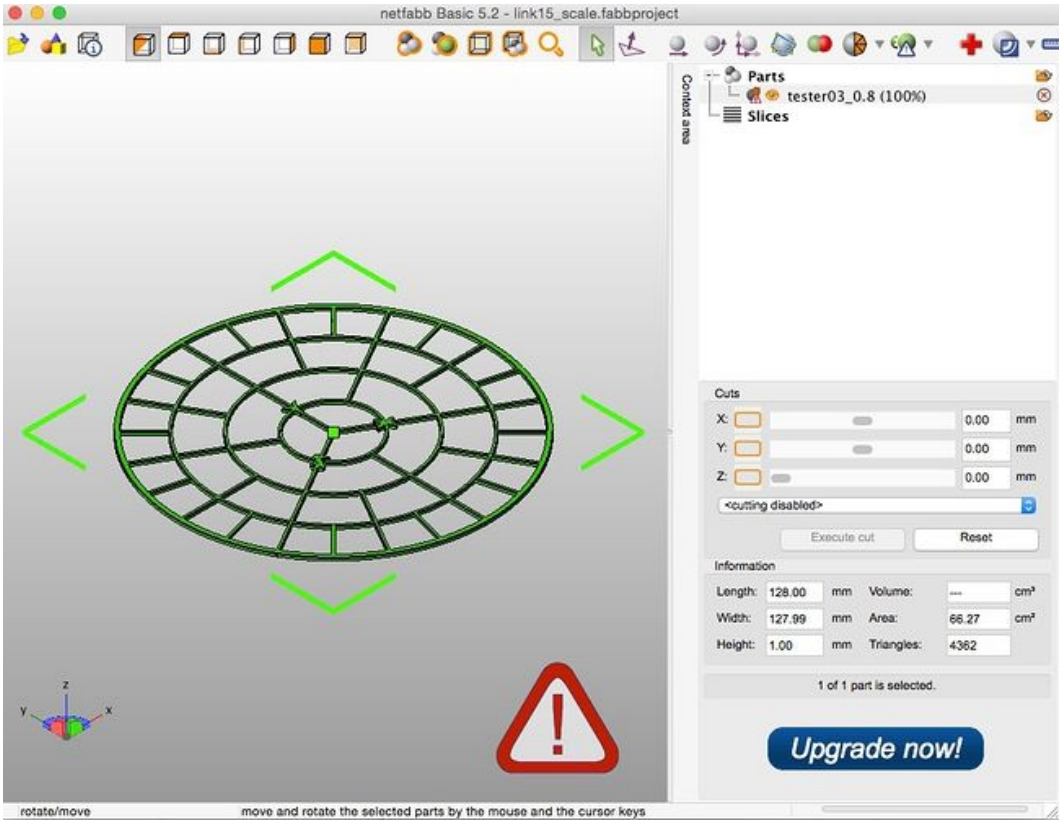
1. Bed leveling: adjust the build plate until it is perfectly horizontal and the gap between the nozzle and the build plate is approximately the thickness of one sheet of A4 paper. If the spacing is less than the thickness of one sheet of A4, the filament will not be extruded properly. If the spacing is too large, the filament will not stick properly to the build plate
2. Using a heat bed: the purpose of a heated bed is to help the filament stick to the build plate. For PLA, set the bed temperature to 50 ~ 70 ℃, and for ABS, set it to about 90 ℃
3. Adhesive bed materials: Kapton tape, PET, and sometimes plain stick glue are often used to improve adhesion to the build plate
4. Set custom first layer slicer settings: Raise the print temperature of the first layer by about 5 ℃ and reduce the output speed to about 30-50% of the default printing speed
5. If the slicing software allows it, apply ‘brim’ or ‘raft’ to increase the surface area of the first layer

How to fix errors of modeling data

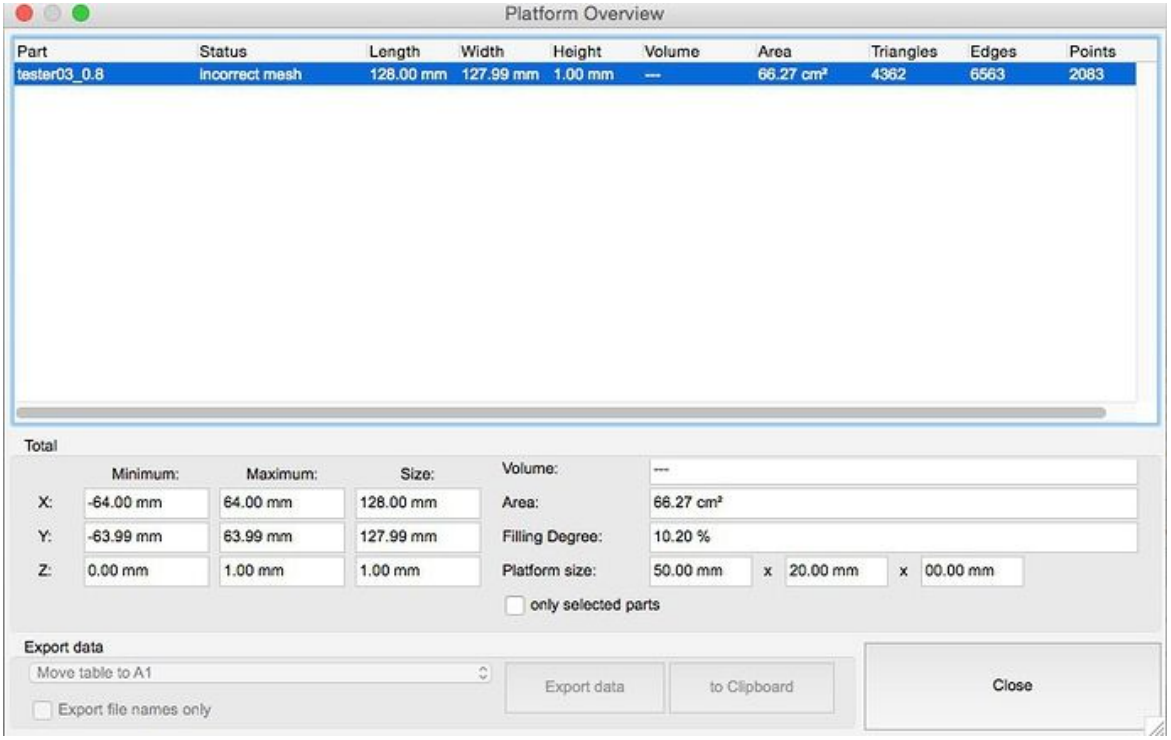
Printing will fail if there is a problem or defect within the 3D model file itself (e.g. if the 3D mesh has unintentional holes or flipped surface geometry). Tools like Netfabb should be used to fix errors in the 3D model.

Netfabb can be downloaded from the link below.
https://www.autodesk.com/products/netfabb/free-trial

First, when you import the problematic model, a red exclamation point appears as shown below:



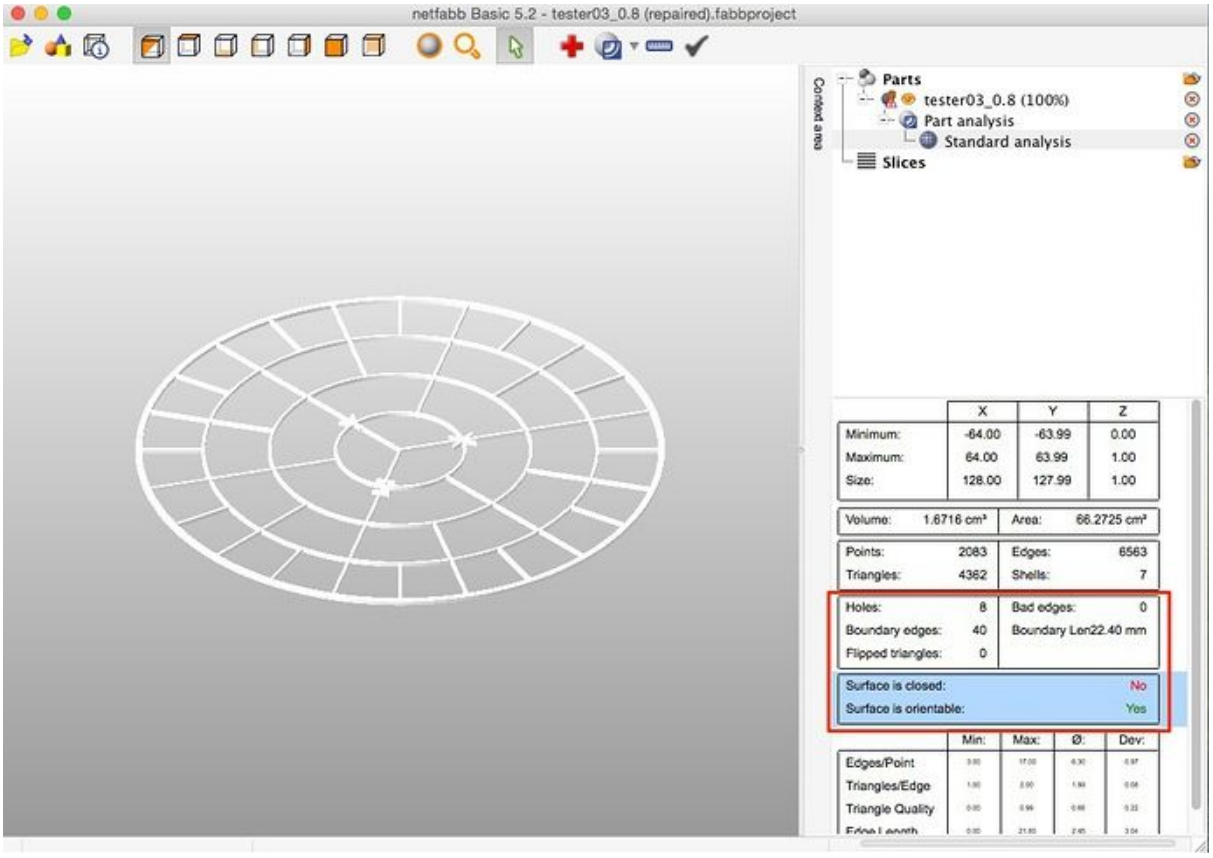
If you select **Edit> Platform** Overview from the menu, the problematic mesh is displayed.


Platform Overview
Part
Status
Length
Width
Height
Volume
Area
Triangles
Edges
Points
tester03_0.8
incorrect mesh
128.00 mm
127.99 mm
1.00 mm

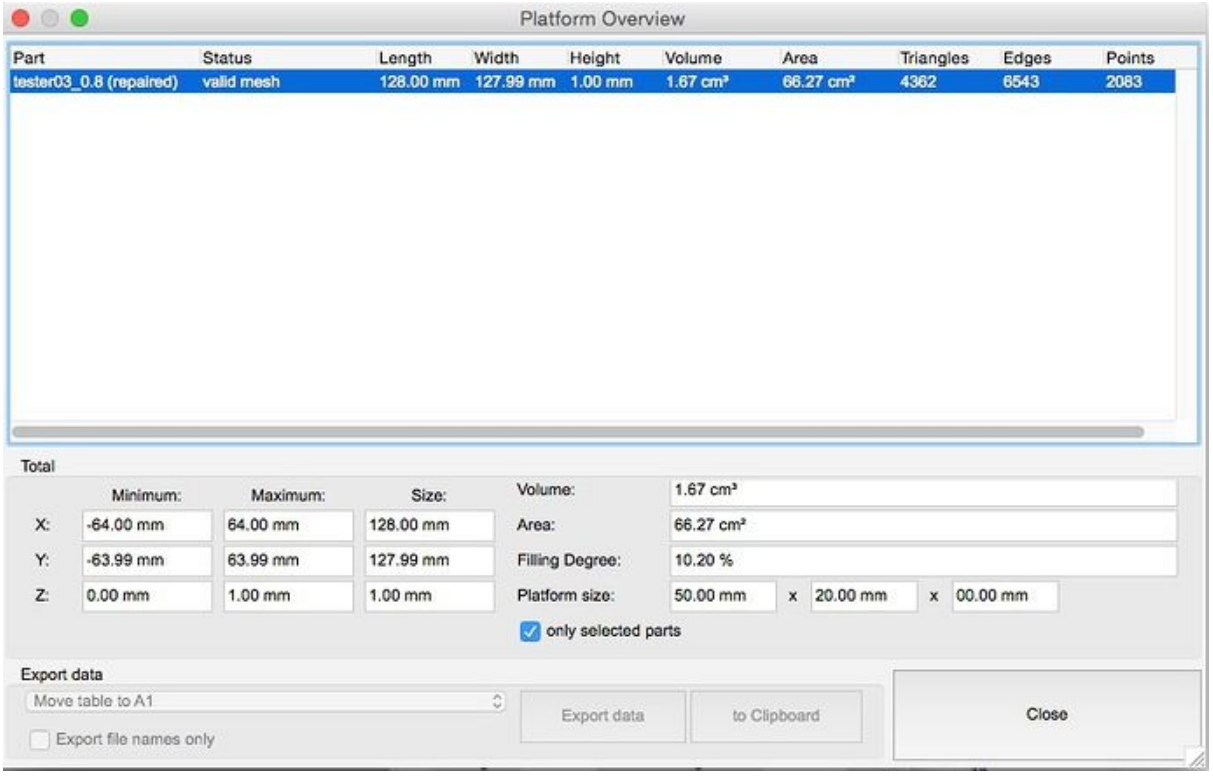
66.27 cm²
4362
6563
2083
Total
Minimum:
Maximum:
Size:
X:
-64.00 mm
64.00 mm
128.00 mm
Y:
-63.99 mm
63.99 mm
127.99 mm
Z:
0.00 mm
1.00 mm
1.00 mm
Volume:

Area:
66.27 cm²
Filling Degree:
10.20 %
Platform size:
50.00 mm
x
20.00 mm
x
00.00 mm
only selected parts
Export data
Move table to A1
Export file names only
Export data
to Clipboard
Close

To find out more, select **Extras> New** analysis from the menu.



Select **Extras> Repair**>Automatic repair from the menu, to apply Netfabb's auto-repair algorithm to the modeling file. Select **Edit> Platform Overview** from the menu again to confirm that the mesh has been fixed.



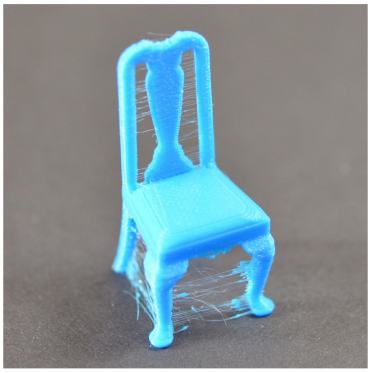
Note that not all problematic 3D models can be fixed with netfabb. If you are unable to repair your model with Netfabb, you may need to manually correct errors in the model file itself.

Maintaining your printer

1. Apply a single drop of machine oil (e.g. sewing machine oil) to all moving parts to ensure smooth movement
2. Remove dust or impurities from the magnetic ball at the ends of each rod supporting the print head
3. Remove dust or oil from the heat bed. If any adhesive materials (e.g. glue stick) were applied to the heat bed, please take care to clean the heat bed thoroughly.
4. Tighten the timing belt if it is loose.
5. If filament is stuck on the end of the nozzle, heat the nozzle and clean off any excess filament using a metal brush if available, or a piece of paper.

Troubleshooting guide

1. Clogged Nozzle
    a. Cut the filament from the feeder and remove the filament tube
    b. For PLA, raise the temperature of the nozzle to 220 ℃ and pull the filament out by hand
    c. Apply a needle to the end of the nozzle to puncture the clogged area.
    d. Filaments (ABS or nylon) that melt at higher temperatures can be pushed into the nozzle to remove melted filament from the nozzle.
    e. Allow the nozzle to cool before attempting to reassemble
2. Stringing or Oozing



This is caused by leakage of filament from the nozzle as it travels to a new position. To minimize this, it is helpful to increase the retraction speed and distance, and slightly lower the print temperature.

3. Gaps in Thin Walls

Problems can occur when printing very thin walls that have a thickness only a few times the nozzle diameter (e.g. nozzle diameter: 0.4 mm, wall thickness: 1.0 mm). To minimize this problem, setting the infill to 100% in your slicer settings is recommended.

Additional information

If you have any other problems, please leave a message in the CREATABLE LABS forum.
https://forum.creatablelabs.com/

Contact support

* Company Information
www.ateamventures.com

* For all maintenance and support issues, please contact us at
support@creatablelabs.com or +82-743-6322(ext.2)