

# BCN3D Cura Manual

# INDEX

Introduction	<b>3</b>
Get Sarted	<b>5</b>
First Use	<b>7</b>
Explore BCN3D Cura	<b>11</b>
Settings	<b>25</b>
Tutorials-How To	<b>36</b>

# INTRODUCTION

BCN3D Cura is a free and easy-to-use 3D printing software that prepares your model for 3D printing. It provides an intuitive user interface and an improved workflow, both for newcomers and expert users. It is based on the renowned Open Source slicer, developed by Ultimaker together with the community. A special thanks, therefore, is sent to Ultimaker for taking the lead on the software development and the explanations this manual is based on.

Even though the BCN3D printers are compatible with all the different Gcode generator softwares, it is recommended the usage of the BCN3D Cura version. Just as other Gcode generator softwares, BCN3D Cura sections the 3D model in layers and establish a difference between the perimeter and the infill of the object. In order to provide a better printing experience, the version developed by BCN3D contains a number of changes both in the interface and in Cura functionalities.

**GET STARTED**

## Technical requirements

### Operating systems:

Windows: Windows Vista or higher, 64 bit.

Mac OS: Mac OS 10.11 or higher, 64 bit.

Linux: Ubuntu 14.04 or higher, 64 bit.

### System Requirements:

OpenGL 2 compatible graphics chip, OpenGL 4.1 for 3D Layer view.

Intel Core 2 or AMD Athlon 64 or newer.

205 MB available hard disk space.

4GB RAM memory.

**Note:** Although these are the minimum requirements, we recommend using better computers for a better use experience.

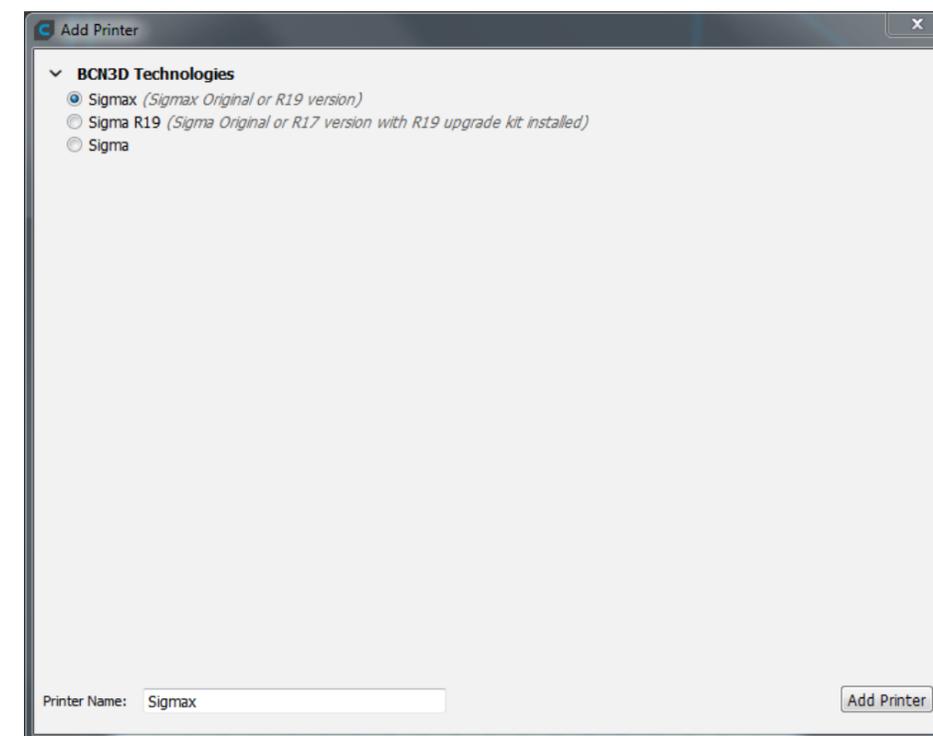
## Downloading and installing BCN3D Cura

Visit <https://www.bcn3dtechnologies.com/en/3d-printer/bcn3d-cura/> and download the suitable version of BCN3D Cura for your Operative System. Once downloaded, follow the installation wizard.

After installation, open BCN3D Cura for the first time.

## Select printer

Once you open BCN3D Cura for the first time, you will be asked to select the 3D printer that you have. Optionally, you can give the printer a custom name.

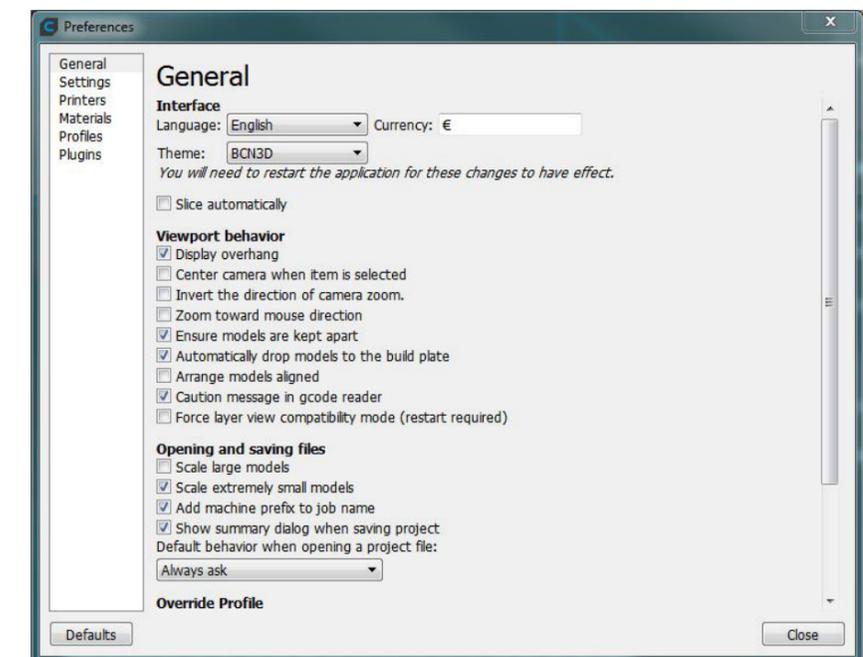


## Change language settings

By default, BCN3D Cura is configured in English. However, it is possible to set it up in a variety of languages to ensure a better accessibility.

To change the language settings, go to *Preferences --> Configure Cura*.

A new window will prompt. Under the tab General, in the section Interface, select the preferred language at the list.

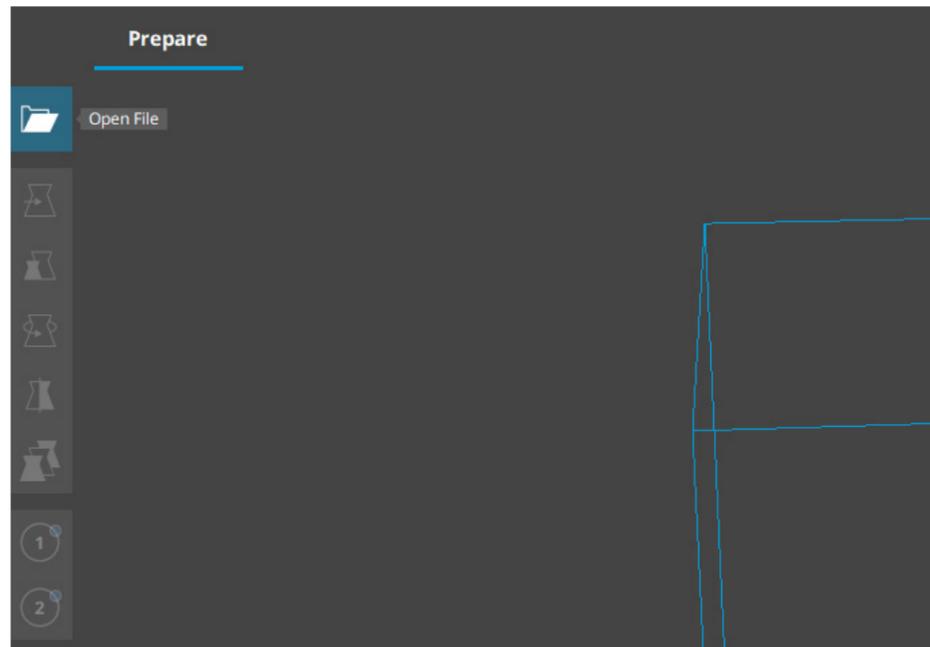


Close the window and restart BCN3D Cura to apply changes.

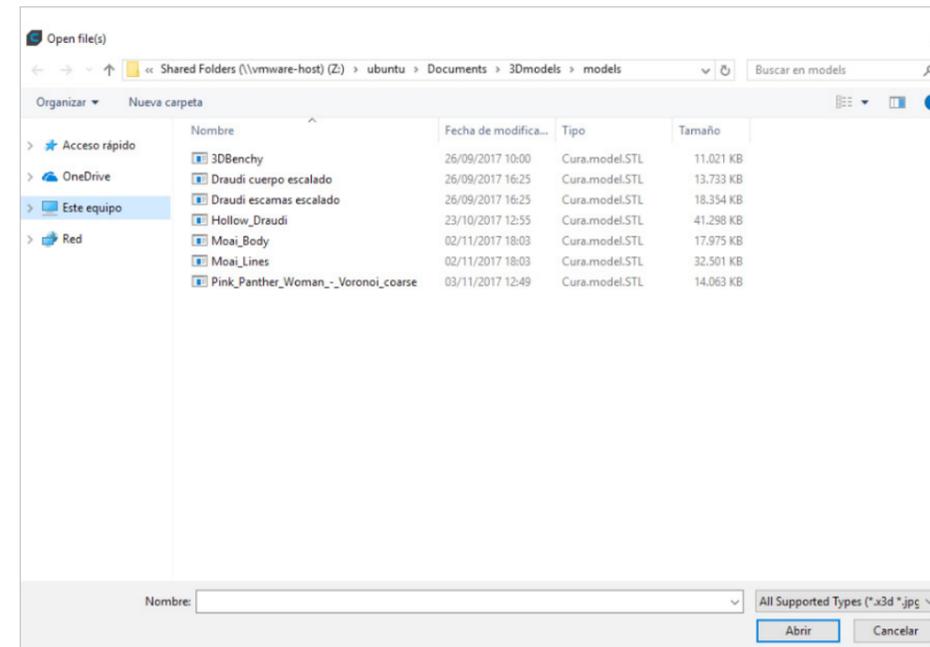
**FIRST USE**

## Load a 3d model

Click on the Open File button in the top left corner of the screen.



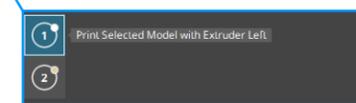
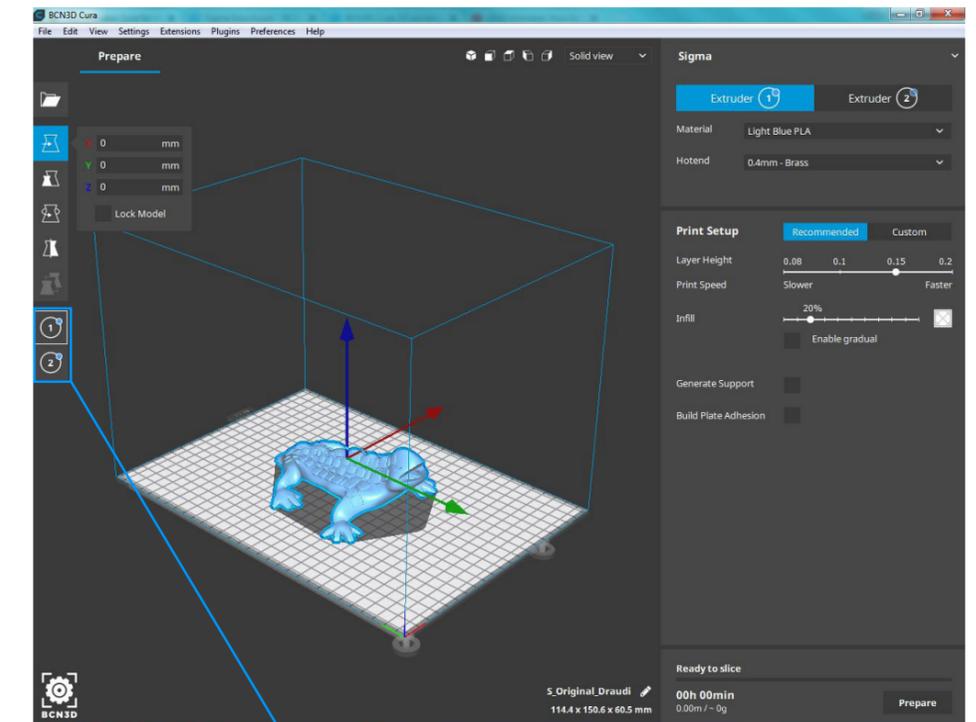
Browse the model you want to prepare for printing and open it.



## Extruder assignment

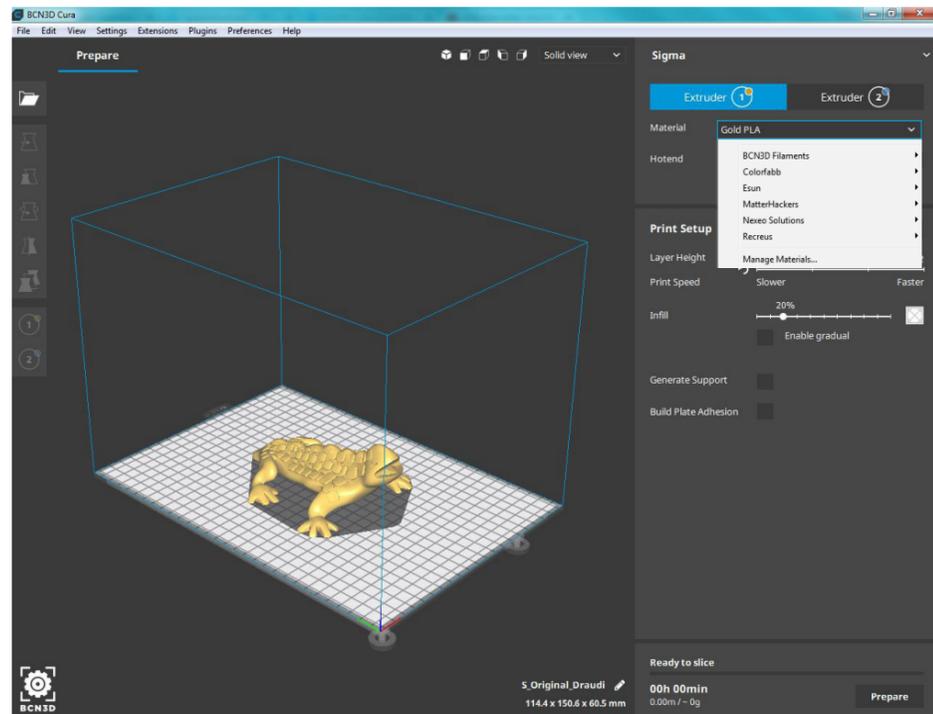
Both the Sigma and Sigmax count with 2 extruders. By default, any 3D model is assigned to the Left Extruder (1). However, it is possible to reassign the extruders.

Click on the model and it will be highlighted with a blue outline. Then, at the left of the screen, choose the extruder to print that model.



## Material assignment

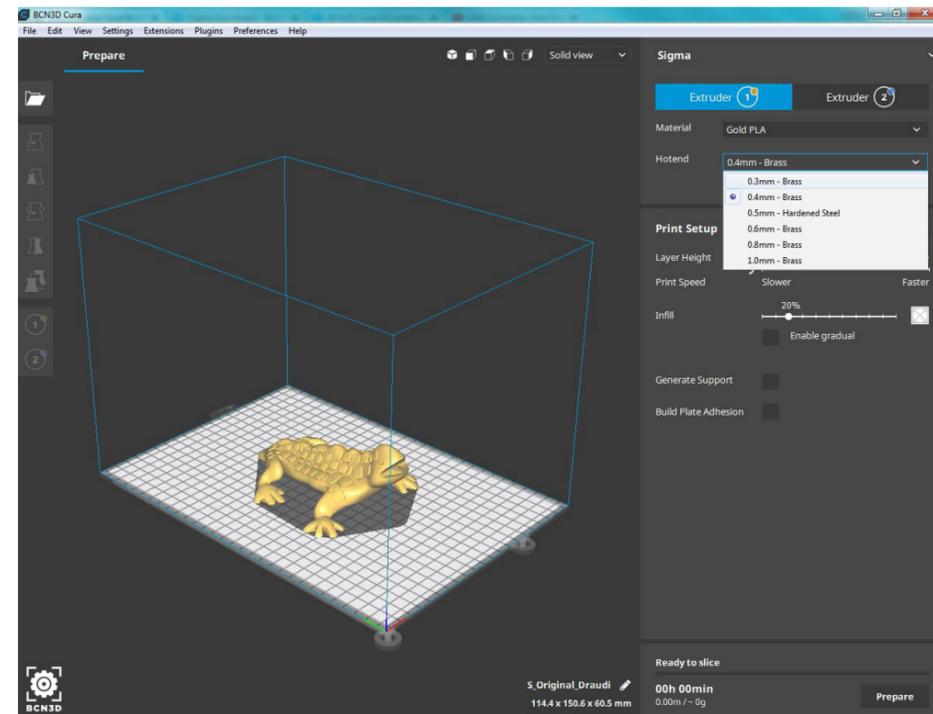
At the top of the right side bar, click the extruder to set up, the same you have assigned to print the model. To select the material that will be used to print the 3D model, just click on the top drop-down list and browse. Materials are organized by manufacturer and type.



## Select hotend

The BCN3D printers allow printing with different hotend sizes: 0.3, 0.4, 0.5 (steel nozzle), 0.6, 0.8 and 1.0 mm. Small nozzles are ideal for detailed models. Instead, big nozzles allow to fabricate more resistant parts or for rapid prints. Select on the drop-down list the size you have installed at your printer.

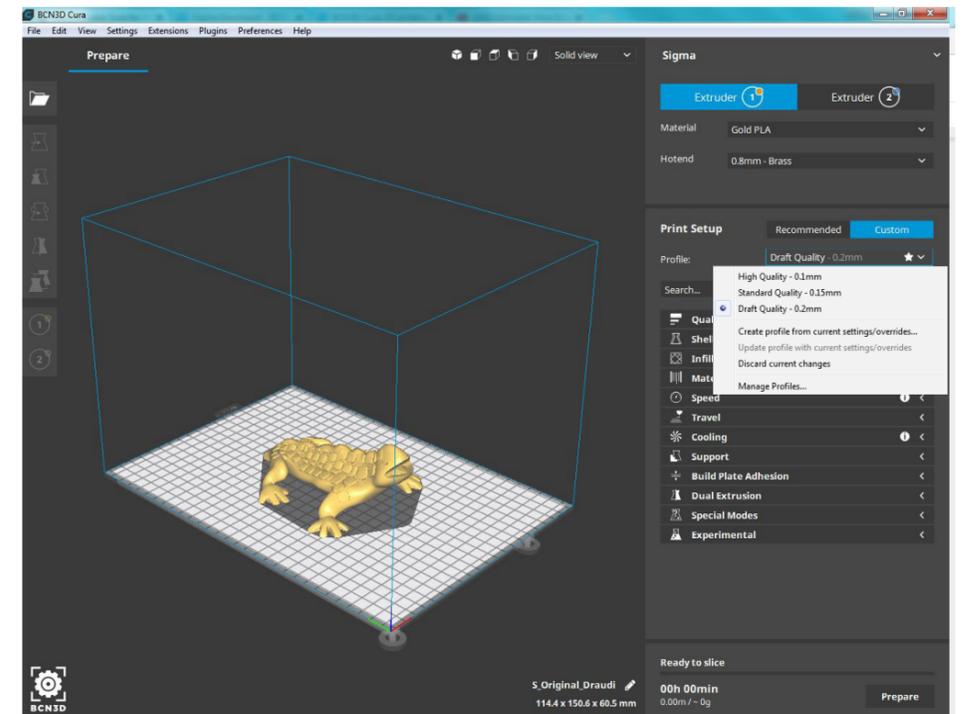
Note: By default, the Sigma is equipped with 0.4mm hotends and the Sigmax with 0.6mm hotends.



## Select printing profile

In the Print Setup submenu, at the right sidebar, it is possible to choose between Recommended and Custom views.

While the Recommended view is ideal for inexperienced users, the Custom tab allows to finely set up the print job. In the Profile drop-down list, select the Quality Profile to print with.



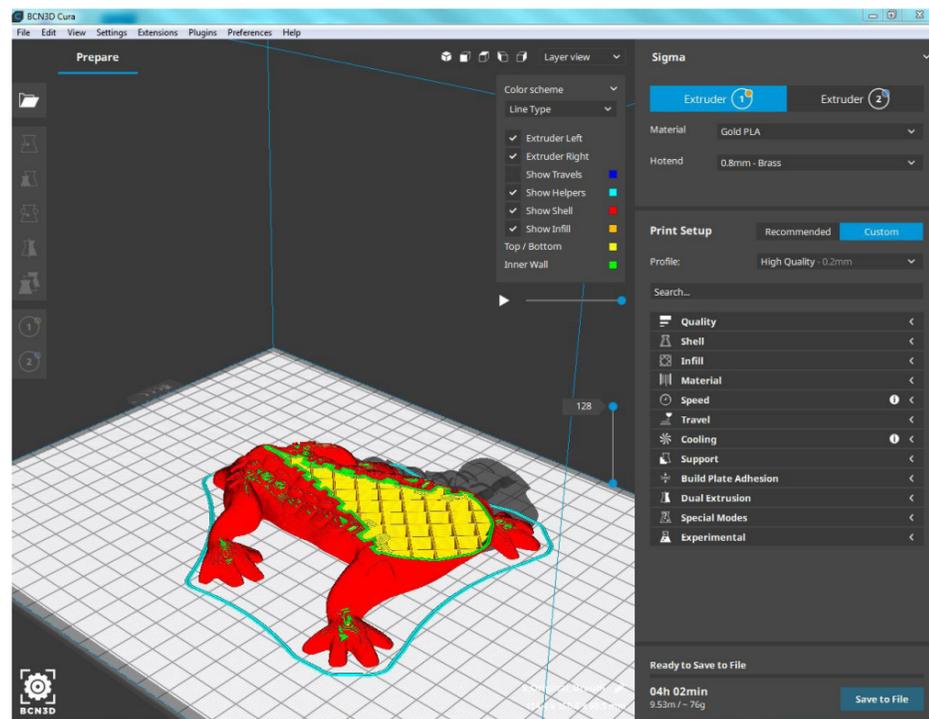
## Slicing And Preview

Once the printer and the models have been set up, at the bottom of the right sidebar click the Prepare button. At that point, BCN3D Cura will slice the model and generate the file the printer will use to print the desired model. When the slicing process finishes a time and material consumption estimation will be given.

To preview the slicing result, click on the 'Solid view' button in the top right corner of the 3D viewer and change it to 'Layer view'. This view is ideal for seeing what the inside of your print will look like and how it will be printed. Use the slider to check important parts of your 3D slice. Change the printing strategy until you are satisfied.

## Save the print file

To finish the preparation process, just click the Save to File button, at the bottom right corner, once the slicing process has finished. You can save the file to your computer or to a removable disk like the SD card provided with the printer. Safely remove the SD card from the computer and insert it in the printer.



**EXPLORE BCN3D CURA**

## Interface overview

1. Open file
2. Adjustment tools
3. 3D model
4. Camera views
5. View mode
6. Extruder, material and hotend configuration
7. Print setup
8. Model information
9. Print job information
10. Prepare / Save to file

## Adjustment tools

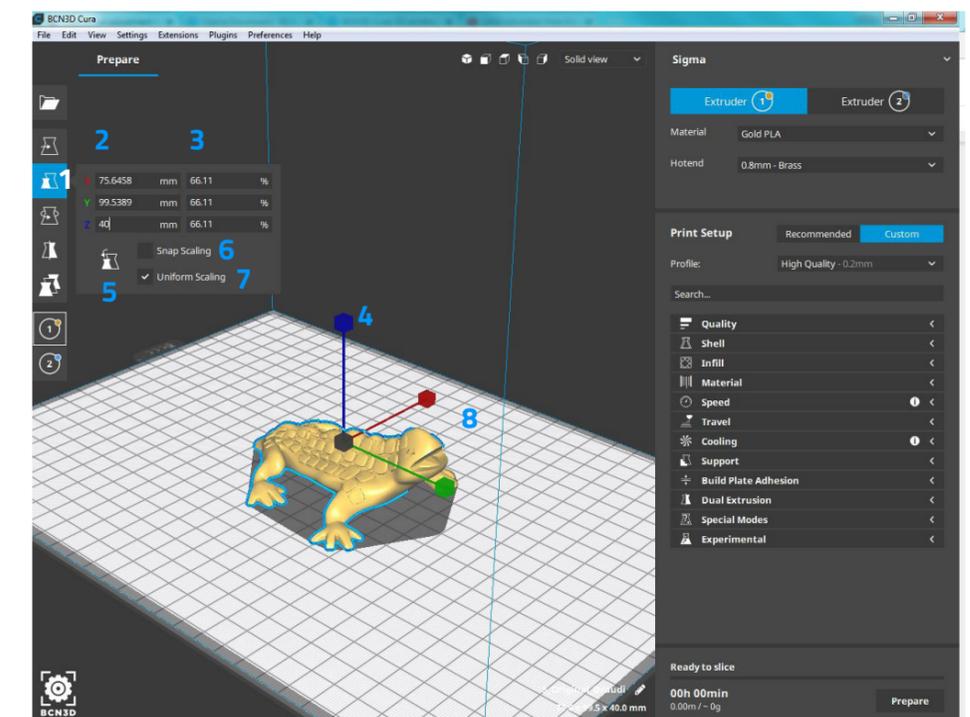
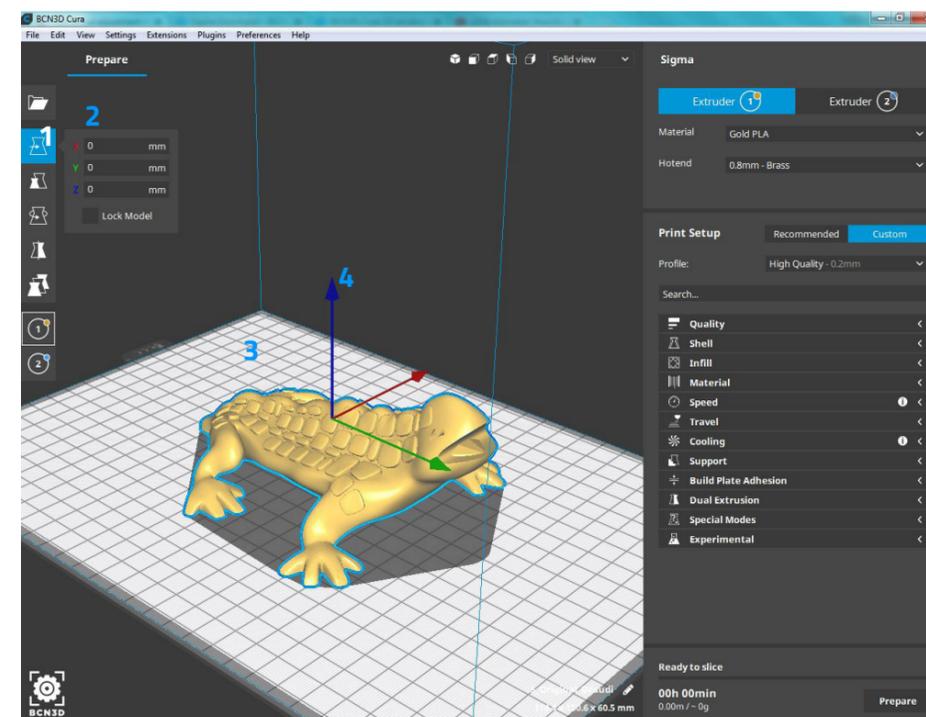
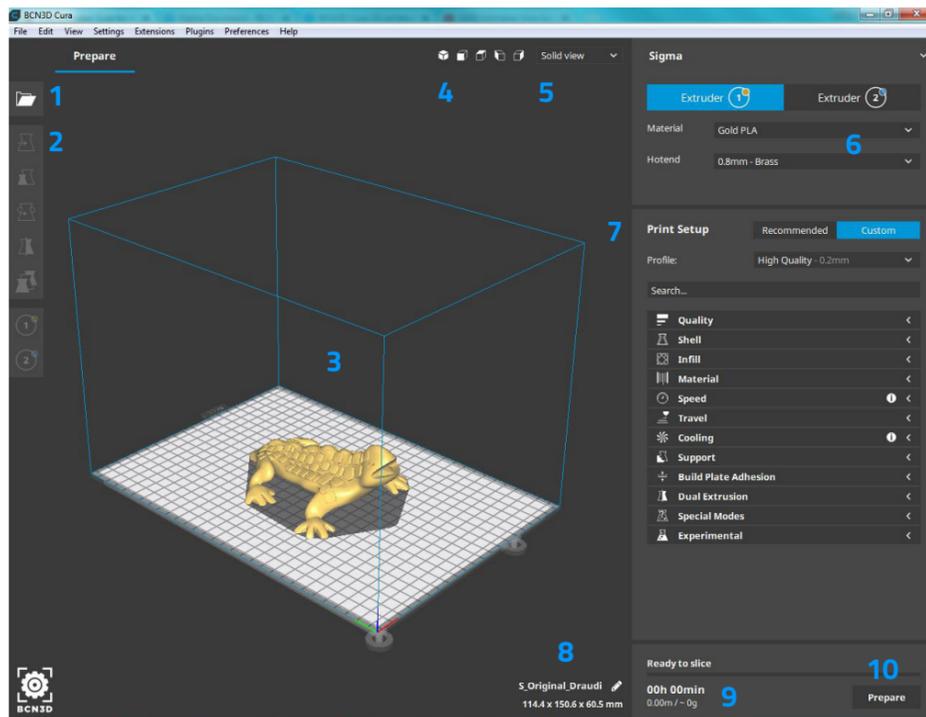
BCN3D Cura offers different tools to adjust and transform the model before printing:

**Move tool:** Position the model in the desired location:

- Select the model. It will be highlighted with a blue outline (1).
- The move tool opens by default (1).
- Use the input field to set a specific coordinate (2).
- Or drag the model (3) to move it freely.
- Or drag the arrows to move it locked on one axis (4).

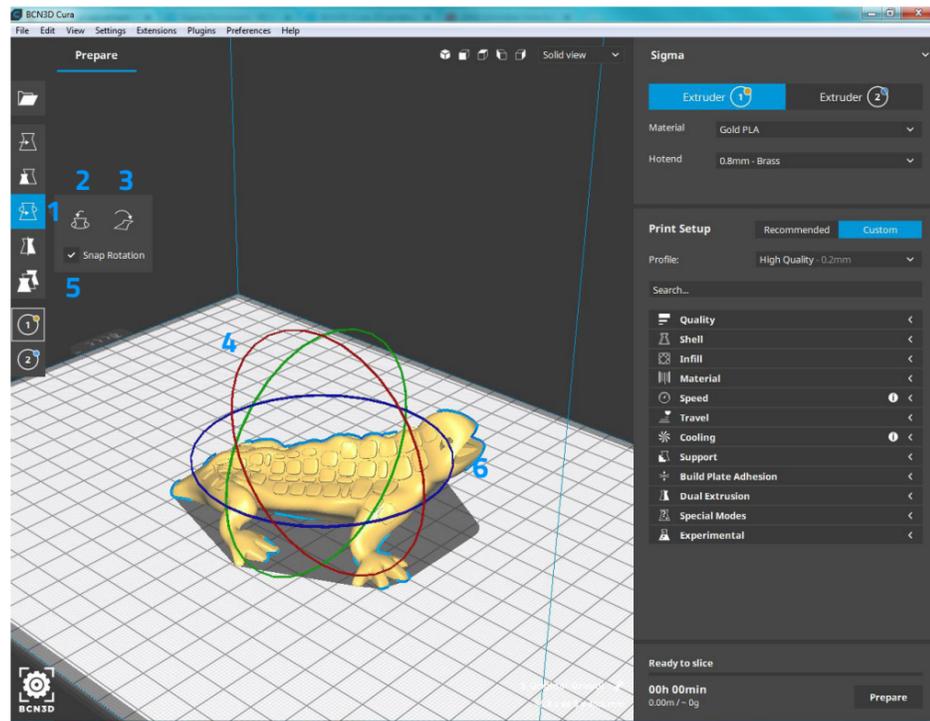
**Scale tool:** Scale the model, both uniform and non-uniform:

- Select the model. It will be highlighted with a blue outline (8).
- Select the scale tool (1).
- Set a specific size in millimeters (2).
- Scale the model to a percentage (3).
- Drag the handlebar (4) to freely scale the model.
- Reset (5) the model to its original scale.
- Enable/disable snap scaling (6). Snap scaling allows to scale the model in increments of +/- 10% when dragging the handlebar.
- Enable/disable uniform scaling (7).



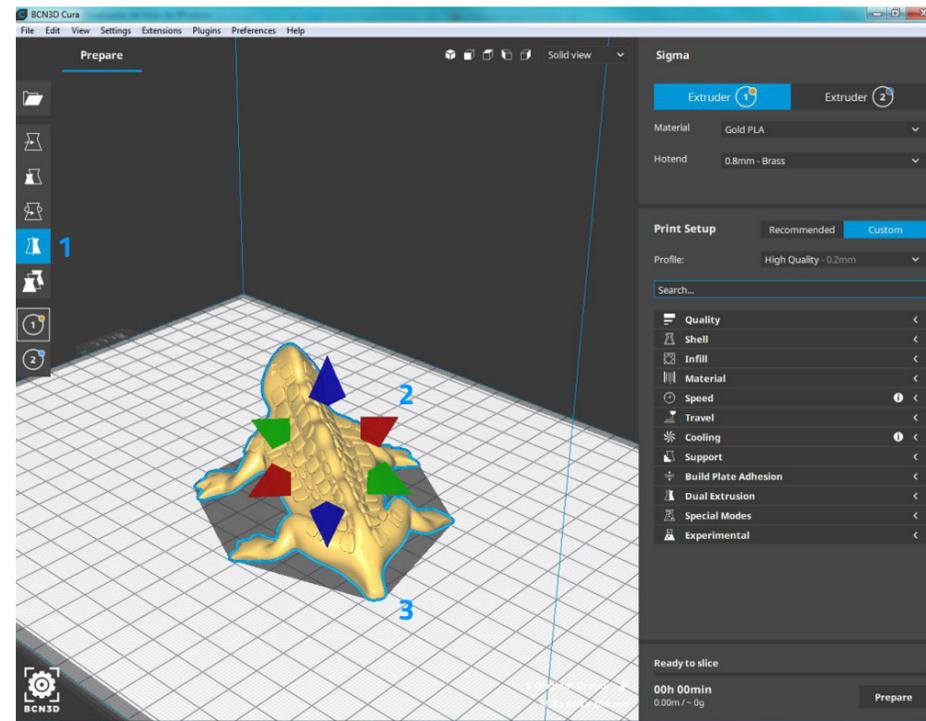
**Rotate tool:** It allows to rotate the model on all 3 axes. This is particularly useful to reduce the amount of support needed and to increase the model's footprint for build plate adhesion.

- Select the model. It will be highlighted with a blue outline (6).
- Select the rotate tool (1).
- Reset model to its original orientation (2).
- Lay the model flat on the build plate (3). BCN3D Cura will automatically determine the best face to put the model flat.
- Drag the hoops (4) to rotate the model in 15° intervals.
- Snap rotation (5) is enabled by default. Disable it here or hold shift while dragging the hoops to have a more precise rotation.



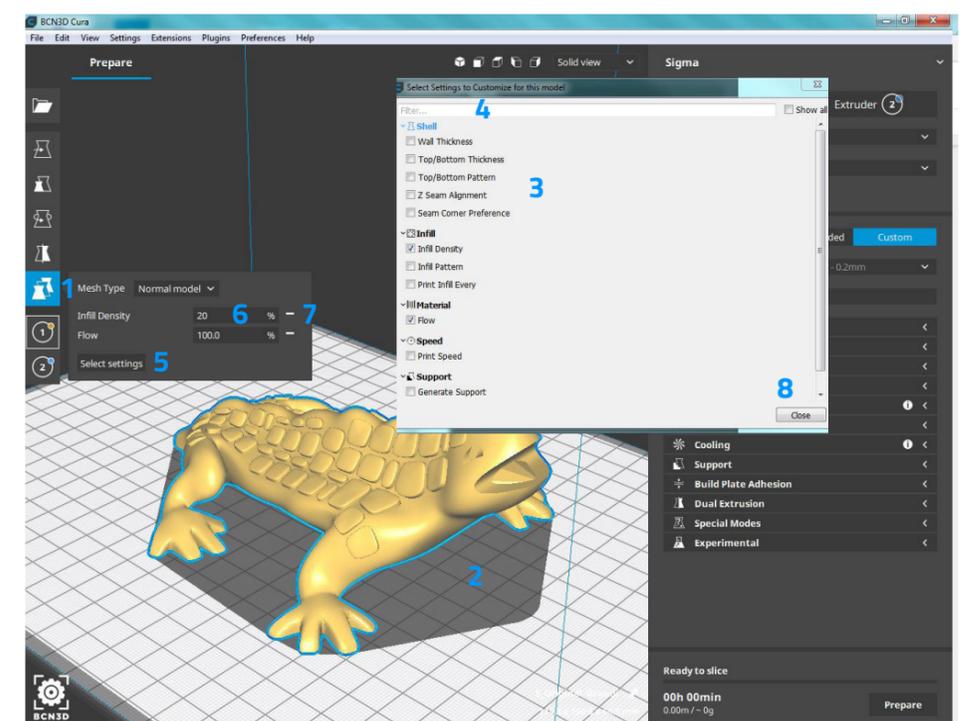
**Mirror tool:** It allows to mirror the model in one axis.

- Select the model. It will be highlighted with a blue outline (3).
- Select the mirror tool (1).
- Click the arrowheads (2) indicated by the axes colors to mirror the model on that axis.



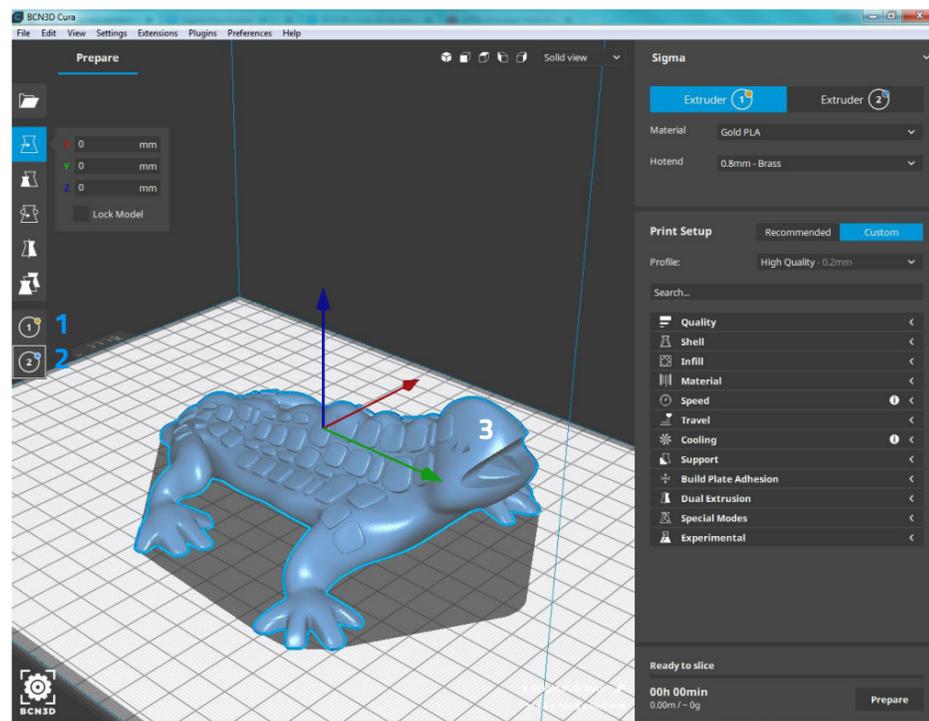
**Per model settings:** it allows to set different print settings for multiple models. Therefore, it is possible to have different printing strategies in the same print job.

- Select the model (2) you wish to have modified settings.
- Select per model settings (1).
- 'Select settings' (5) opens a pop-up list with available settings.
- Locate the setting and enable the checkbox (3).
- Use the filter (4) to quickly find your settings.
- Close (8) the window when you have added all settings you want to change.
- Use the input field (6) to change the setting value.
- Use the '-' (7) button to remove the custom setting from the list.



**Select Extruder:** It is possible to assign an extruder to specific models on the build plate:

- Select the desired model on the build plate (3).
- Select the extruder (2) you want to print the model with. By default, loaded models are assigned to the left extruder (Extruder 1).
- The extruder selector has a colored circle (1) to indicate what material this model will be printed with. It corresponds with the loaded material in the printer settings and the model on the build plate.



## Print Modes

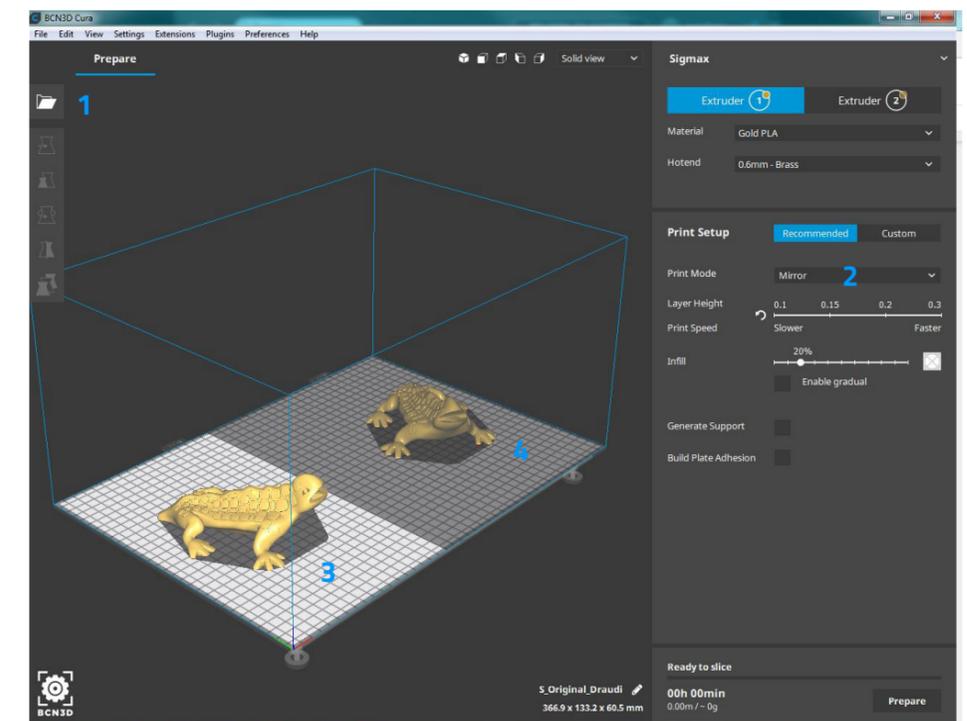
Thanks to the IDEX technology, the BCN3D printers are able to work in multiple printing modes.

Note: All printing modes are enabled by default at the Sigma. To enjoy the highly productive Mirror and Duplication modes with a Sigma, update its firmware to version 1.3.0 or newer. [Check out page 20 to know how to update the firmware.](#)

**Regular:** The most common printing mode, selected by default. It only uses one toolhead. It's ideal to print large models requiring more than half of the printing surface, multicolor or multimaterial parts.

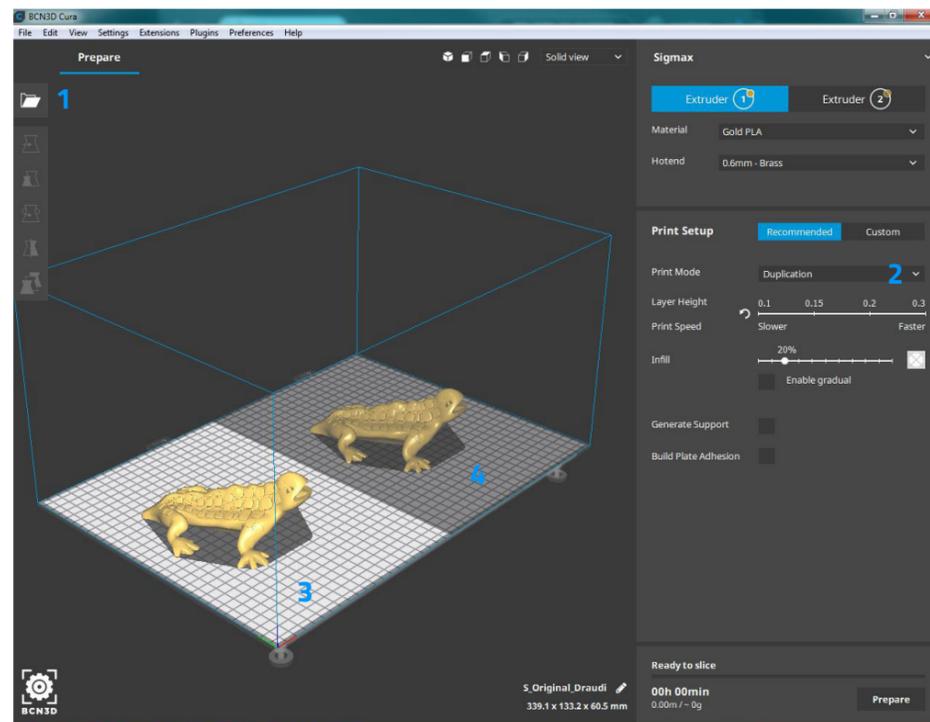
**Mirror:** Print the model and its symmetrical part (along the X axis) at the same time. Using this printing mode, the printing capacity is doubled, but the available printing surface is reduced to less than half the original size.

- Load the model (1).
- Select Mirror printing mode (2). The left model at the build plate is editable (3) and the right model (4) is just a preview of the mirrored model.
- Adjust the model at the left side, and the right mirrored model will be automatically adjusted.



**Duplication:** Print the same model with both toolheads simultaneously and double the printing capacity. This mode is ideal for short runs of production. Bear in mind the available printing surface will be reduced by half.

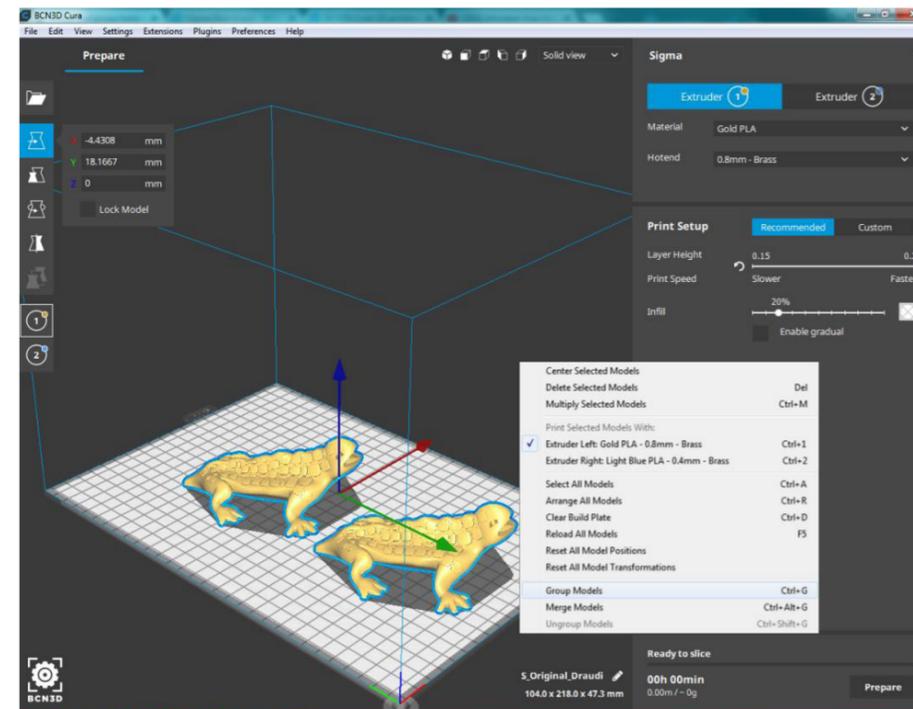
- Load the model (1).
- Select Duplication printing mode (2). The left model at the build plate is editable (3) and the right model (4) is just a preview of the copied model.
- Adjust the model at the left side, and the right copied model will be automatically adjusted.



## Group models

Grouping models allow the models to be adjusted simultaneously to reduce the number of actions needed and to reduce the number of possible errors made during adjustments.

Select two or more models on the build plate, by Shift + Left Clicking every model. Selected models get a blue outline. Right click one of the selected models, and select Group Models. Or hit Ctrl / Cmd + G.



Ungrouping is done with the same workflow, but in step 3. choose Ungroup models.

Grouping models has the following effect:

- The models keep the same position on the build plate.
- The models keep the same orientation and position to each other.
- Models can be scaled, rotated and mirrored simultaneously.

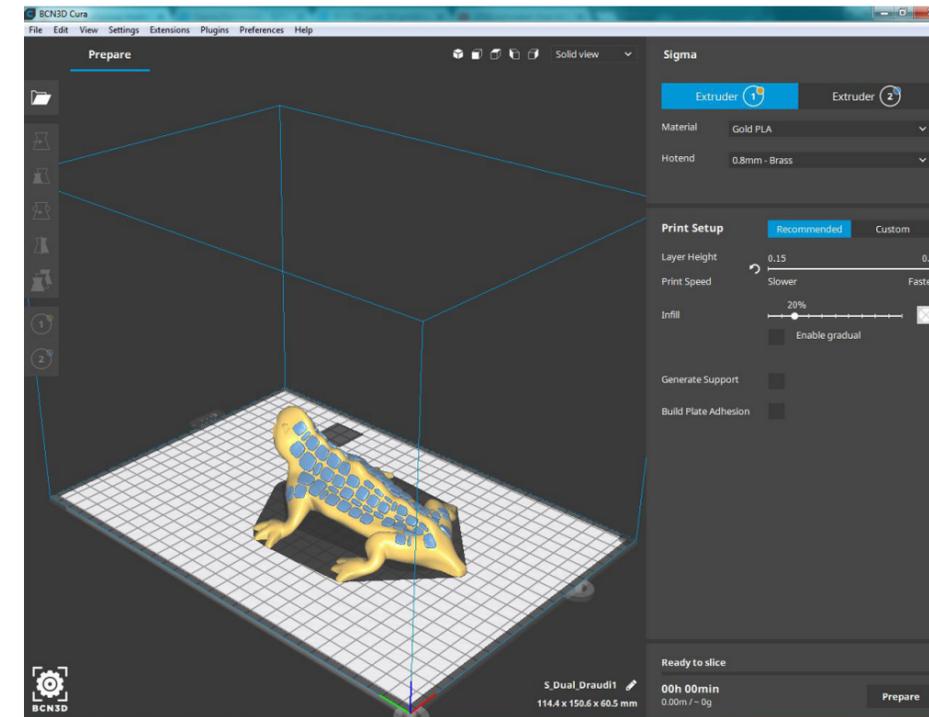
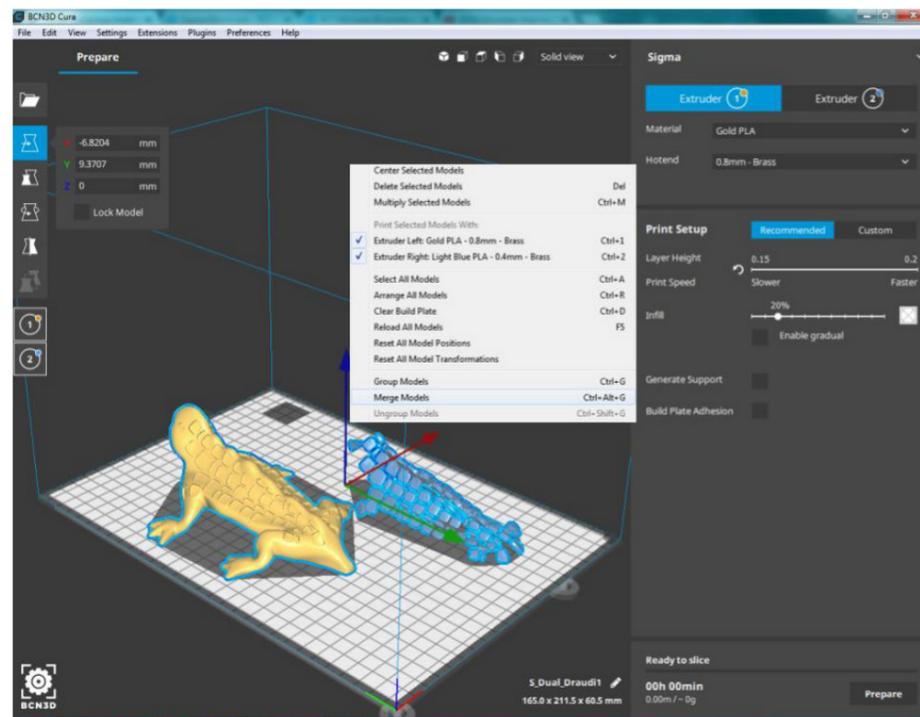
## Merge models for Dual Extrusion

This feature is useful when working with dual material/color models. Merging allows the models to be combined in the easiest way possible. Follow these steps to Merge Models:

- First, assign the extruder for each model.
- Then, select two or more models on the build plate, by Shift + Left Clicking every model.
- Selected models get a blue outline.
- Right click one of the selected models, and select Merge Models. Or hit Ctrl / Cmd + Alt + G.

Merging models has the following effect:

- The models are aligned to the position of both models' origin points.
- Models can be scaled, rotated and mirrored simultaneously.

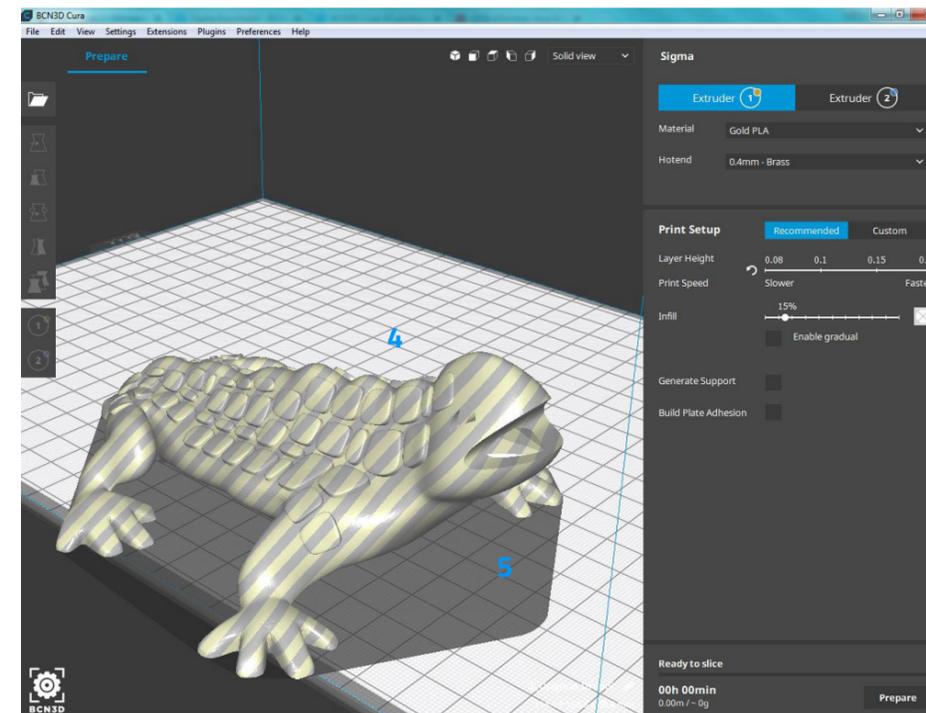
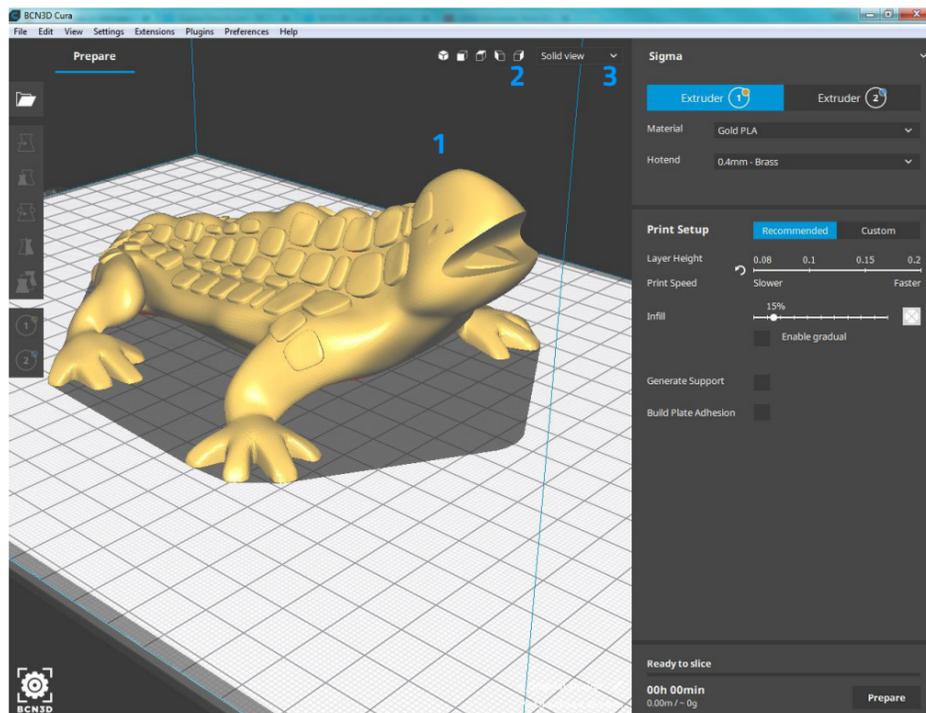


## Model View Modes

BCN3D Cura offers three different ways to view the model, each one providing different information.

**Solid view:** The default view mode, represents the model as it was digitally created. This view lets you to check if the desired model is properly loaded and placed on the build plate.

- The model (1) is shown corresponding to the view mode selected. Its color will depend on the material selected.
- The view mode button (3) is located on the top right corner of the 3D viewer.
- Click it and ensure 'Solid view' (2) is selected.
- When a model is located in a non-printable area (5), it is displayed with stripes (4). Use the Move tool to place the model in a printable area.



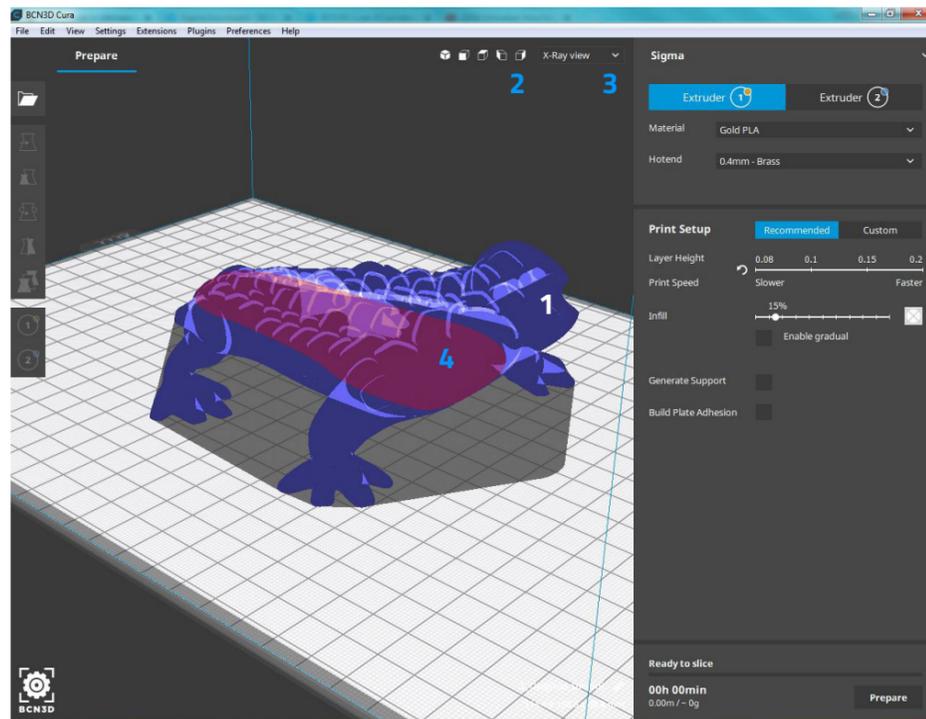
# Explore BCN3D Cura



**X-Ray view:** It allows looking through the model, making it easy to detect missing and unwanted internal geometry. If there were any errors in the model they would be displayed in red. Not all types of errors are displayed, so this is not a guarantee that the model is correct.

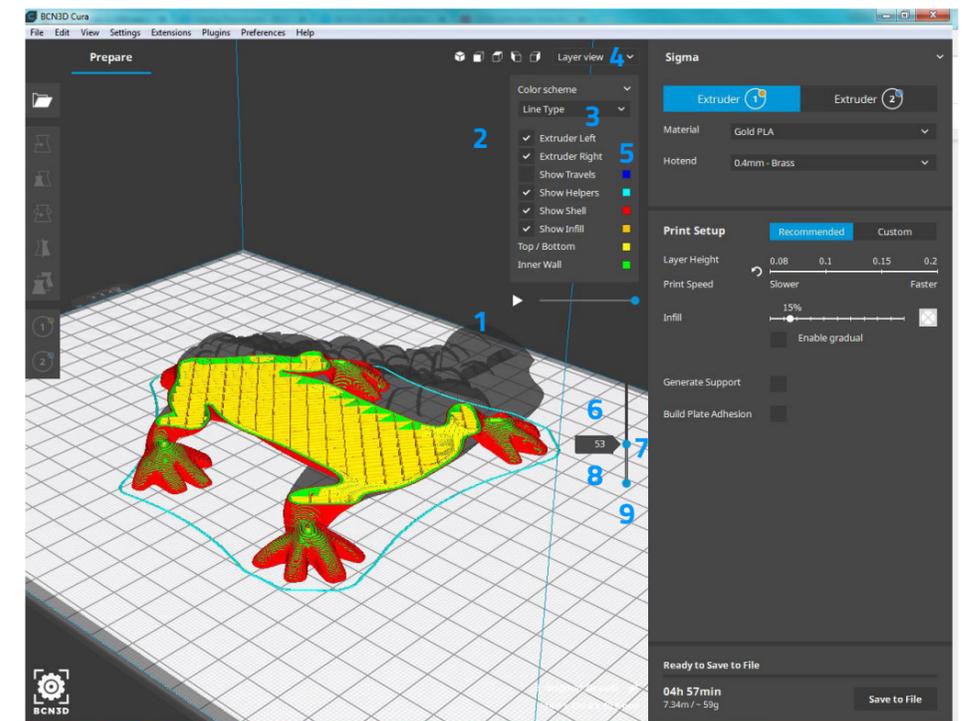
Note: Although some models are 'broken' when loaded, BCN3D Cura's mesh fixes allow the model to be printed (seemingly) correct.

- The model (1) is shown corresponding to the view mode selected.
- The view mode button (3) is located on the top right corner of the 3D viewer.
- Click it and ensure 'X-ray view' (2) is selected.
- Broken model parts are indicated by red areas (4).



**Layer View:** The layer view represents the print moves of the nozzle(s). This way, the result of all input from the settings panel is visualized, ideal for determining your printing strategy.

- The model (1) is shown corresponding to the view mode selected.
- The view mode button (4) is located on the top right corner of the 3D viewer. Click it and ensure 'Layer view' is selected.
- Visualization of the extruders (2) can be toggled to ease viewing of individual parts.
- The colors of the lines can be set to line type (3), material color, feedrate or layer thickness.
- Different parts of the line types (5) can be disabled to get a closer view of certain model parts.
- The layer box (6) shows the current layer viewed. It can be used to enter values to directly jump to.
- The layer slider has three parts. A top (7), mid (8) and bottom (9) part. These parts can be dragged separately to examine the layer view.



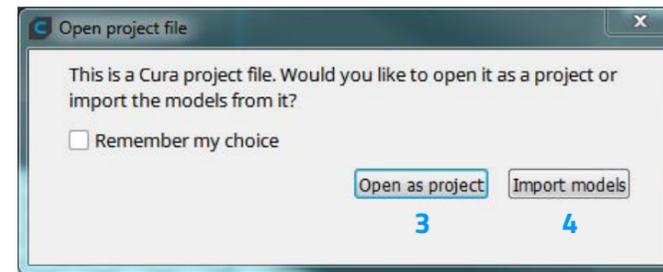
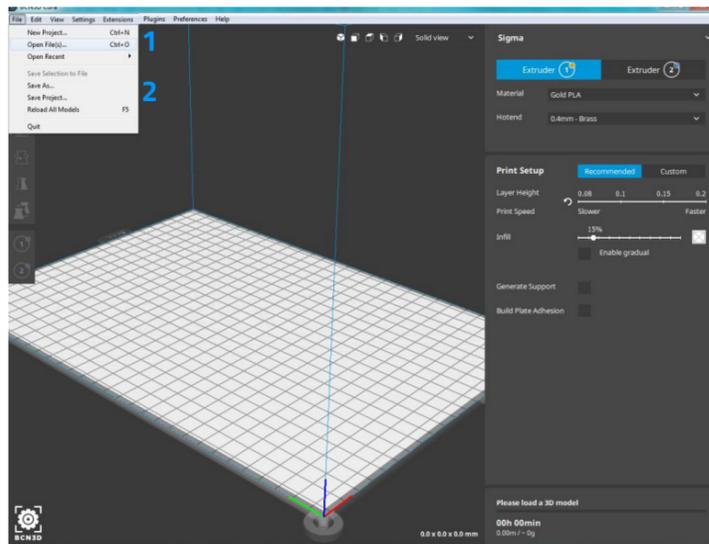
## Project saving

Preparing a print job is a thorough work, since there's a lot of information involved: from printer and models to profiles and materials. Sometimes it is convenient to store all that information to resume the work later or to share it with another user. This is how Project Saving can help you.

### Open and Save projects

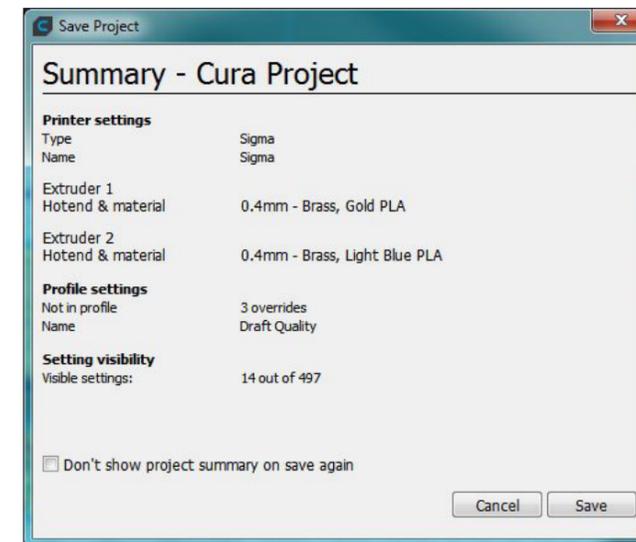
Just like with any other file, go to the File menu to open a project or to save the current one.

- Select 'Open file(s)...' (1) to open a project file.
- Select 'Save project' (2) to save a project file.
- Open the project by clicking 'Open as project' (3) in the pop-up window.
- If you wish to only load the embedded model files, choose 'Import models' (4).



### Save project: Summary

When saving a project BCN3D Cura shows a summary of all settings that are going to be saved. The models on the build plate are saved as well, complete with their transformations. That means that this project will load exactly as it was saved at the time.

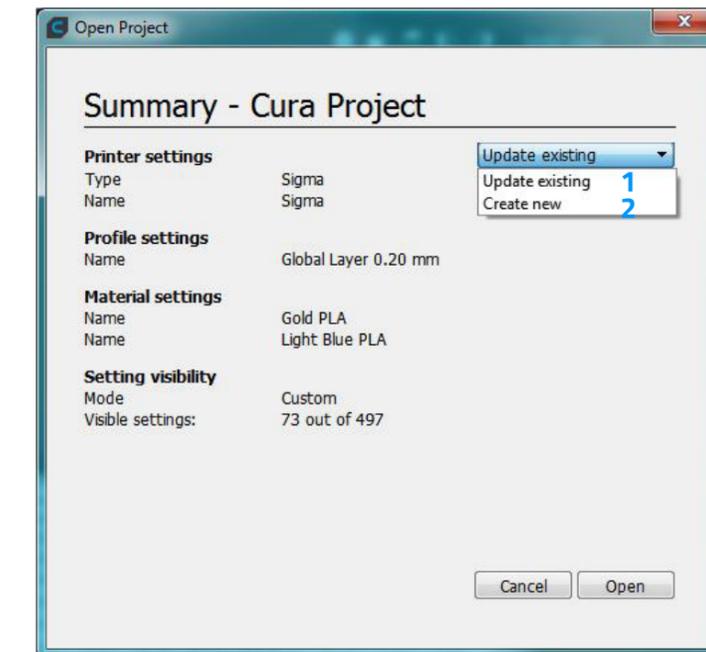


### Open Project

When opening a project file which contains equally named settings, BCN3D Cura cannot know which information is most recent. Therefore, when loading a project you might get the option whether to update the existing (1) setting or create a new (2) one to keep your local settings saved as well.

Update existing: Will update your local settings and therefore not create new setting entries.

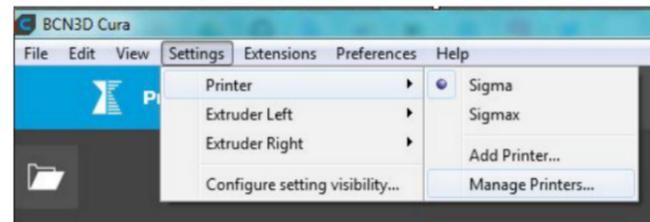
Create new: Will create the setting(s) as new to prevent your local settings from being adjusted.



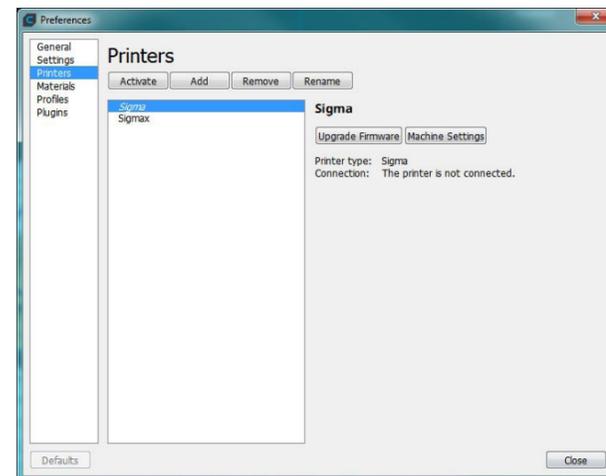
## Update firmware

At BCN3D Technologies we want you to have a great printing experience, and to achieve this we are continually working to create new firmware upgrades for our printers. Follow this process through BCN3D Cura for checking if the latest firmware version is installed on your printer. If not, it will get the latest firmware and it will be installed.

1. Use the USB cable to connect the Sigma/Sigmax to the computer.
2. Open BCN3D Cura. Go to *Settings* --> *Printer* --> *Manage Printers*.



3. Select *Upgrade Firmware*.



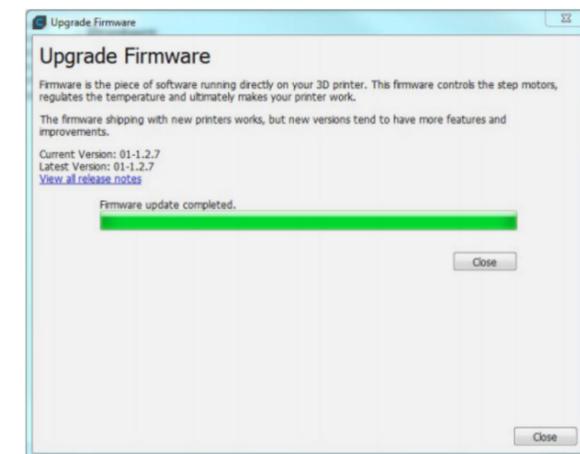
4. Click "Automatically upgrade Firmware" to download and install the latest version available.



5. Wait until the firmware is installed. The process may take a few minutes to complete.



6. Once the process is completed, close the window.



Depending on the firmware you have, it will ask you to update the micro SD files. Follow the steps that will appear in the BCN3D Cura.

## Custom mode

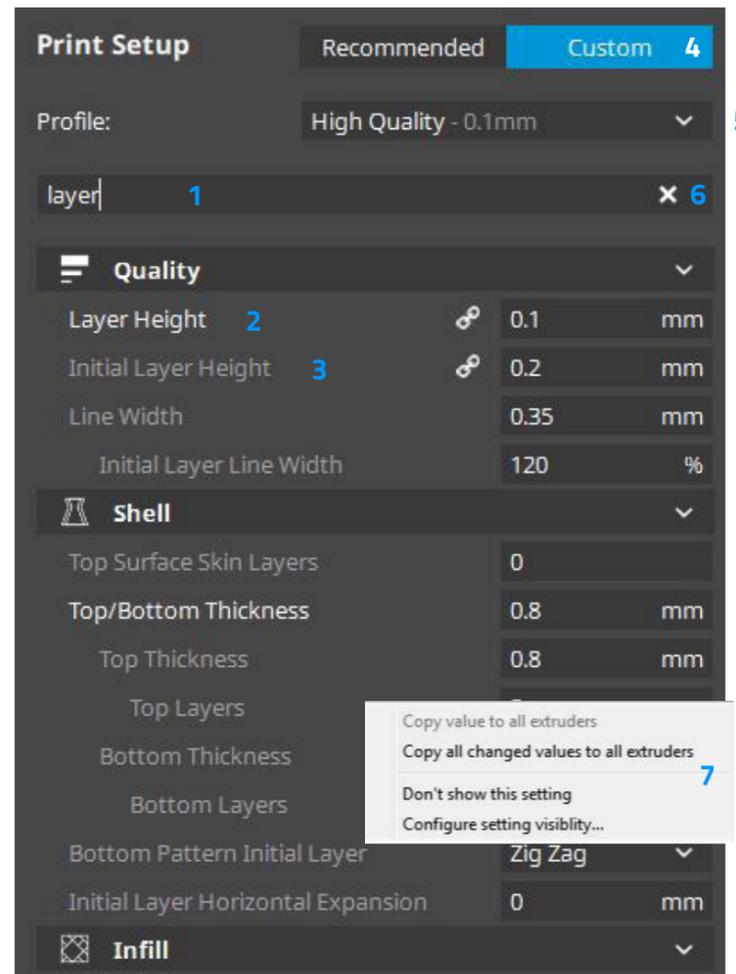
In BCN3D Cura there are two modes available in the settings panel: recommended and custom. The recommended mode is perfect for beginners, while the custom mode offers more settings for experienced users.

**Profile:** Depending on your machine, hotend and material setup, a variety of profiles are available to print out of the box. These profiles are a perfect start point to base your own custom profiles on.

Based on the selected printer, nozzle size and material BCN3D Cura will automatically update each profile with the right settings. Please note that depending on the nozzle size and material selected some profiles are may not available.

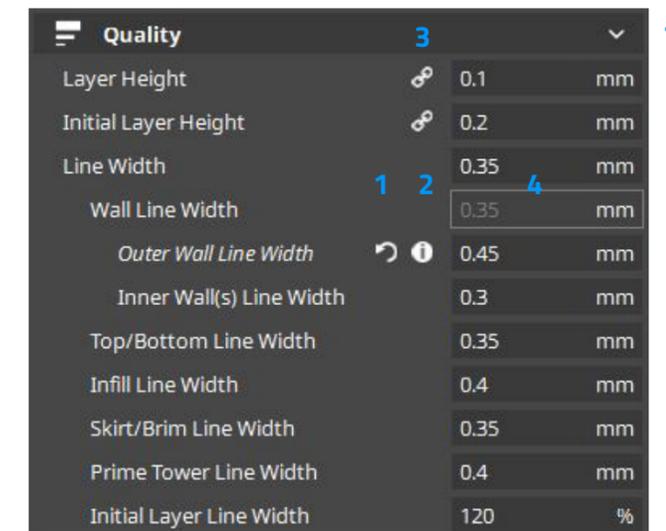
**Search bar:** The custom mode features a search bar to easily access new settings or to filter the currently visible settings to your needs:

- This input field (1) can be used at all times to filter and search for settings.
- White settings (2) are visible, even without the filter.
- Grey settings (3) are visible now, but only because it is currently filtered on.
- Ensure the 'custom' mode (4) is active.
- Choose a base profile (5) to customize here.
- Click the 'cross' (6) to clear the filter.
- 'Right click' a setting (7) to have four options; Copy value to all extruders, Copy all changed values to all extruders, Don't show this setting, Configure setting visibility...



**Adjusted, hidden and calculated settings:** The settings in BCN3D Cura sometimes show indicator icons. The meaning of these icons are explained here:

- The reset icon (1) indicates that this setting has been adjusted manually, and does no longer correspond to the profile setting. Click the icon to set the value back to the profile setting.
- The calculated setting icon (2) indicates that the value was calculated by BCN3D Cura, but is now overwritten by an absolute value. Click the icon to change the value back to the calculated value.
- The chain icon (3) indicates that the value set here is synchronized for all extruders. It is only visible on dual extruder machines.
- If an input is greyed out (4), it means that the value is overwritten by values in its child settings.
- The settings icon (5) can be used to configure setting visibility settings.
- The info icon (6) indicates a user-customized setting is hidden. Click the icon to unhide it.
- The category arrow (7) can be used to view or hide all category settings.



**Settings:** The settings in the side panel are divided in categories, so you can easily navigate through it. These are the available categories:

- Quality: Settings that define the (visual) quality of the print.
- Shell: Settings related to the outside of the print.
- Infill: All settings that have to do with the inside of the print.
- Material: Material related settings.
- Speed: Speeds at which the print head moves while printing.
- Travel: How the print head behaves during travel moves.
- Cooling: Settings that define how the plastic is cooled.
- Support: Options for printing with support structures.
- Build Plate Adhesion: Defines the adhesion of your model to the build plate.
- Dual Extrusion: Settings to ensure a reliable dual print.
- Mesh fixes: Options for fixing faults in models.
- Special modes: Features that influence the way of printing.
- Experimental: Experimental new features.

All categories can be (un)folded by clicking on the arrow icon at the right side, which reveals the available settings. BCN3D Cura shows a selection of most common settings by default, but it's possible to adjust the visibility of settings to your own preferences. To change the settings visibility take the following steps:

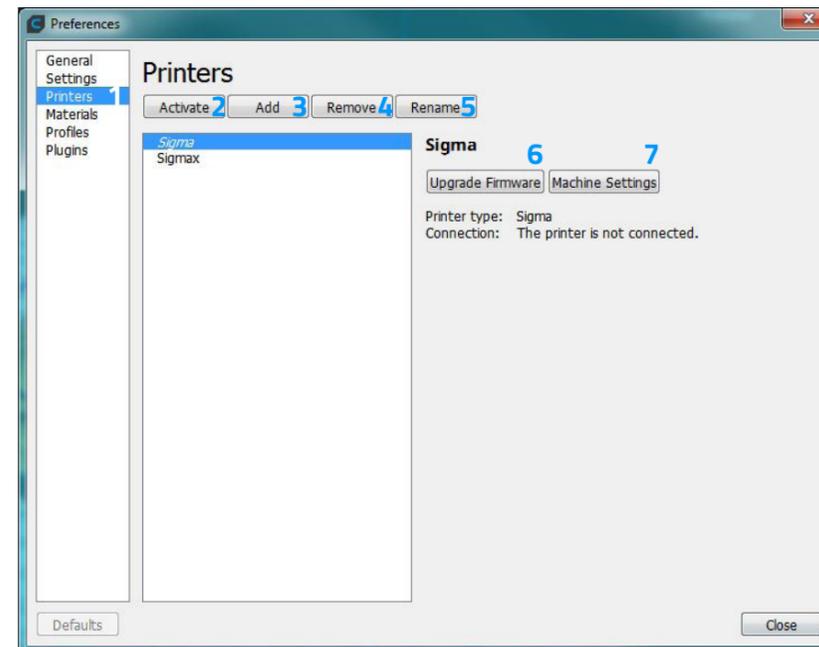
1. Go to *Settings* --> *Configure Setting Visibility*. Alternatively, click on the gear icon of one of the categories.
2. Select or deselect the settings you want to have displayed in the sidebar.

Furthermore, all settings have tooltips, providing a short description of each setting's functionality.

## Manage Printers

When you run BCN3D Cura for the first time, the "add printer wizard" will pop up letting you to choose your BCN3D printer. To add new printers anytime, go to *Settings* --> *Printer* --> *Manage printers*. The "printer manager" will open:

- This (1) is the printers section in the preferences panel.
- Click activate (2) to activate the selected printer in the list below.
- Click add (3) to add a new printer to your BCN3D Cura.
- Click remove (4) to remove the currently selected printer in the list.
- Click rename (5) to give the current printer a custom name.
- This button (6) can be used to update the firmware of the active machine.
- Machine settings (7) can be changed depending on the active machine.

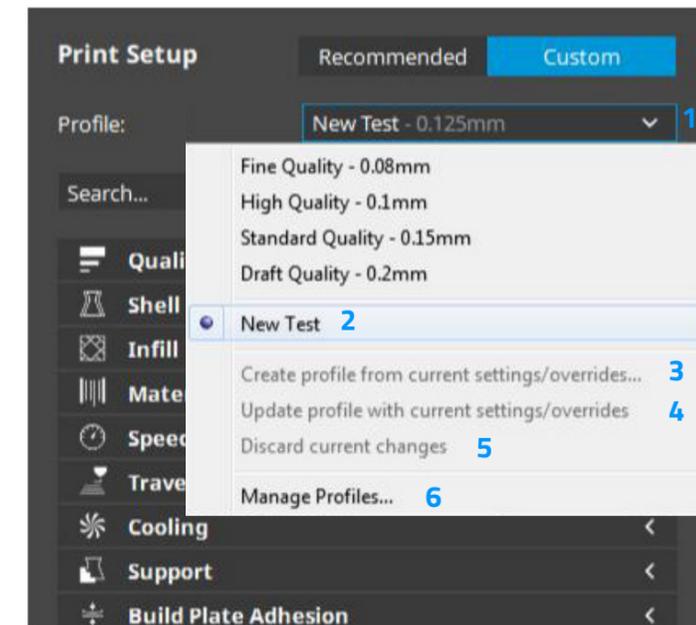


## Manage Profiles

BCN3D Cura provides the ability to save, share and manage your printing profiles, the best way to work efficiently.

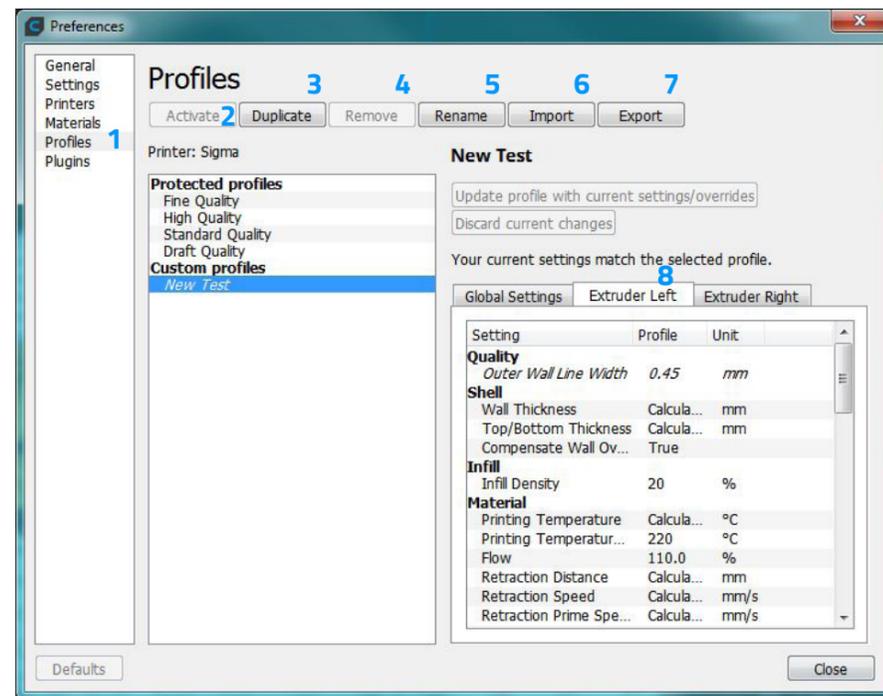
**Quick menu:** The easiest way to quickly create, update and save a profile is through the quick menu. It is also the easiest way to access the profile manager.

- Access the quick menu by clicking the profile (1).
- The currently active profile is indicated (2).
- Create (3) a new profile with the current settings.
- Update (4) an already saved profile with the current adjustments.
- Discard (5) the current adjustments and revert to the saved profile.
- Open the profile manager (6).



**Profile Manager:** When you click on Manage Profiles it will open in a new screen, with all options available as displayed below.

- The *Profiles* tab (1) is located on the left side of the preferences panel.
- Select a profile and click *Activate* (2) to enable it.
- Select a profile and click *Duplicate* (3) to make a copy.
- Select a profile and click *Remove* (4) to delete the profile.
- Select a profile and click *Rename* (5) to change its name.
- *Import* (6) and *Export* (7) profiles to share it with others.
- The setting list (8) contains all custom settings of the profile.



**Add a new profile:** In BCN3D Cura you can add new print profiles based on existing profiles. To add a profile, take the following steps:

- Click on Profile in the menu bar or in the settings panel and select Add profile.
- A new screen will show up; select Add.
- BCN3D Cura has now automatically added a new profile, which is visible in the list of profiles as a customized version of the profile that was selected.

**Change settings of a profile:** It is also possible to change existing profiles in BCN3D Cura. Therefore you simply select the profile and exit the profile manager. Use the setting sidebar in the main window to adjust your settings. When you now switch to another profile BCN3D Cura will ask you whether you want to update the profile or not, or if you want to create a completely new profile.

**Import or export profiles:** BCN3D Cura allows you to share profiles with the import and export feature.

Take the following steps to import a profile:

- Click on Profile in the menu bar or in the settings panel and select Manage profiles.
- Select Import and choose the profile or G-code you want to load.
- Gcode files have the BCN3D Cura print profile embedded as well, therefore 'curaprofile' and 'gcode' files can be loaded.

Exporting a profile can also be done in a few simple steps:

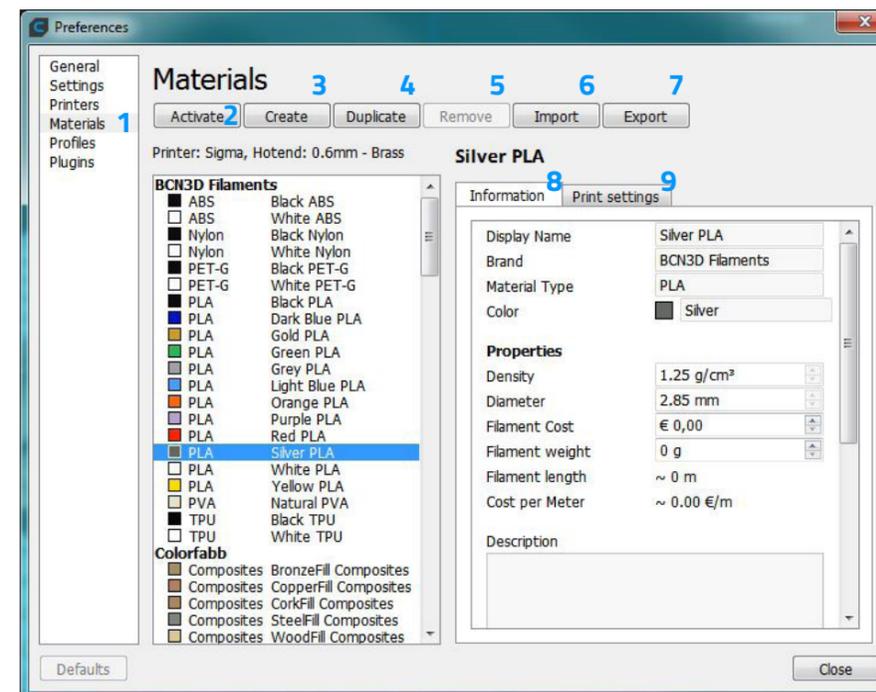
- Click on Profile in the menu bar or in the settings panel and select Manage profiles.
- Select the profile you want to export and click on Export.
- Enter a name and save the profile on your computer.

## Manage Materials

BCN3D Cura comes preloaded with BCN3D materials to make it easy for you to start printing. It also includes some other brands that have been distributed by BCN3D in a recent past.

However, BCN3D printers don't use proprietary filament, so it's possible to use 3rd party filaments. Therefore, it is possible to add them yourself at BCN3D Cura. This way the material settings are conveniently stored, so you can print with your favorite materials every time. To go to the material settings go to *Preferences* --> *Configure Cura*.

- Click the Materials tab (1) to open the material management window.
- To use the selected material, click Activate (2).
- To create an editable material from the current settings, click Create (3).
- To duplicate an existing profile, click Duplicate (4).
- To remove a custom material, select it and click remove (5).
- Import (6) custom materials from others.
- Export (7) your custom material to share.
- The information tab (8) shows generic material information. This information is not accessible in this case since the profile is protected.
- The Print settings tab (9) contains material settings that will be used during printing.



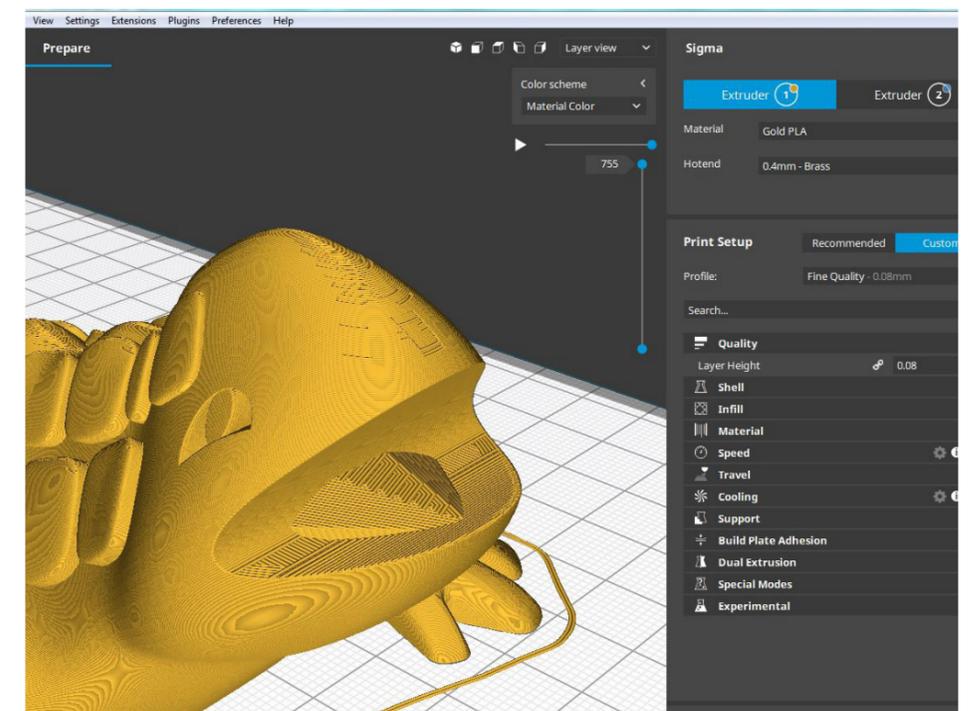
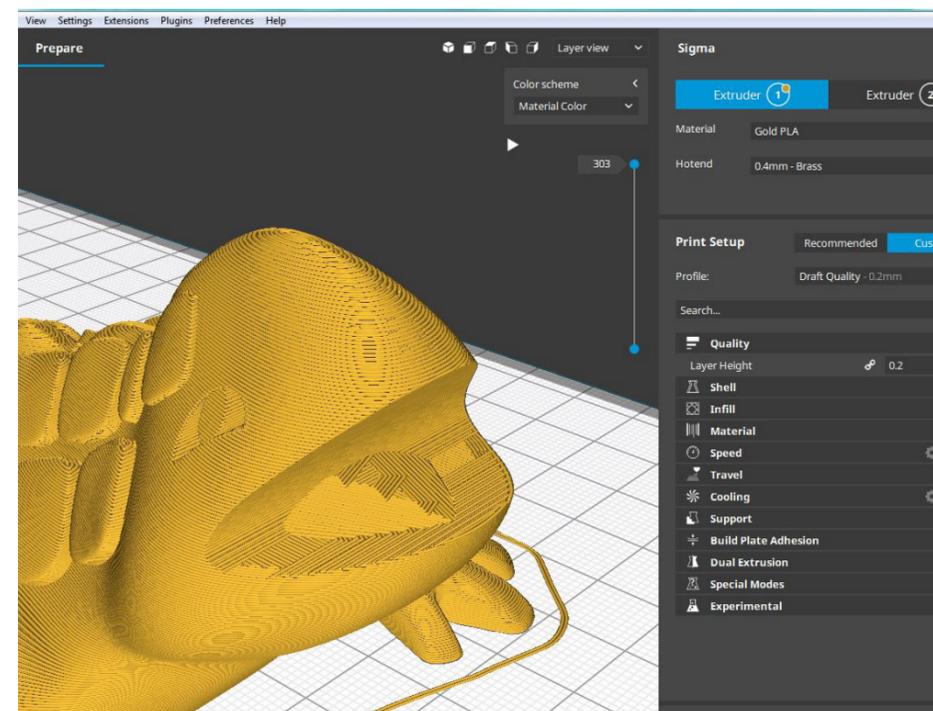
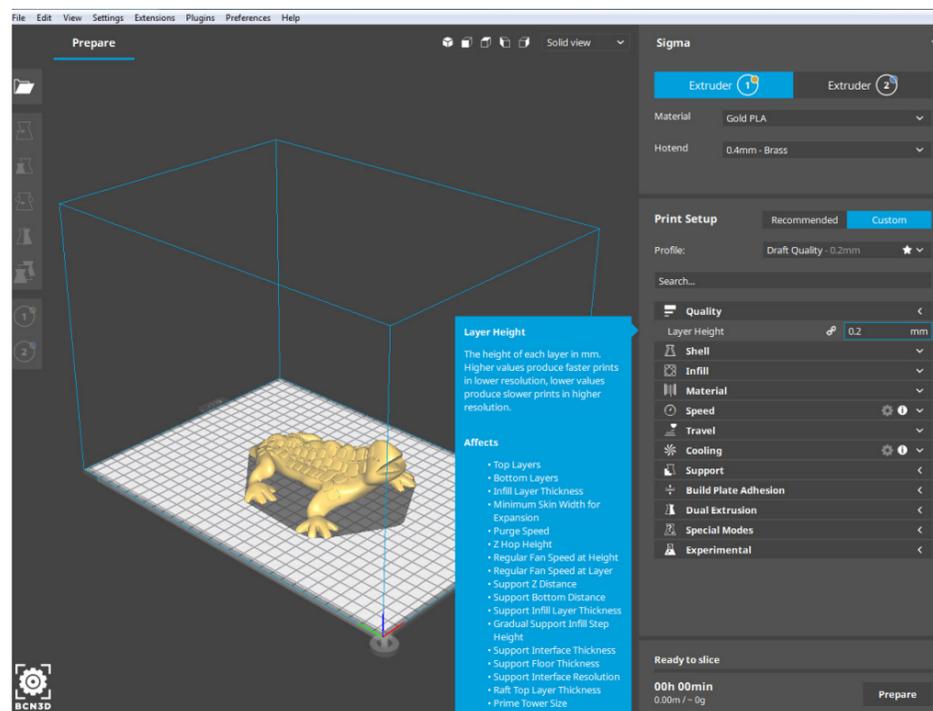
**SETTINGS**

3D Printing is such a complex process and there are hundreds of parameters involved to reach a fine outcome. BCN3D Cura provides total control of the printing process, allowing the user to set the proper printing profile for each job. By default, the Custom Print Setup tab lets you modify the key parameters. Hover over the parameter to see a brief description of the parameter and the implications it may have to modify it.

**For new users, use only the Recommended Tab due it includes Integrated Printing Profiles for each material developed by BCN3D. Follow the process explained in First Use chapter, page 7. Custom Tab is only for experienced users.**

## Quality

**Layer height:** One of the most frequently changed settings. It is the thickness of one printed layer in millimeters. With a thinner layer height you can increase the quality of the print, leading to a smoother surface and more detail visible in the Z-direction (height) of the model. On the other hand, by using thicker layers you can decrease the print time substantially.

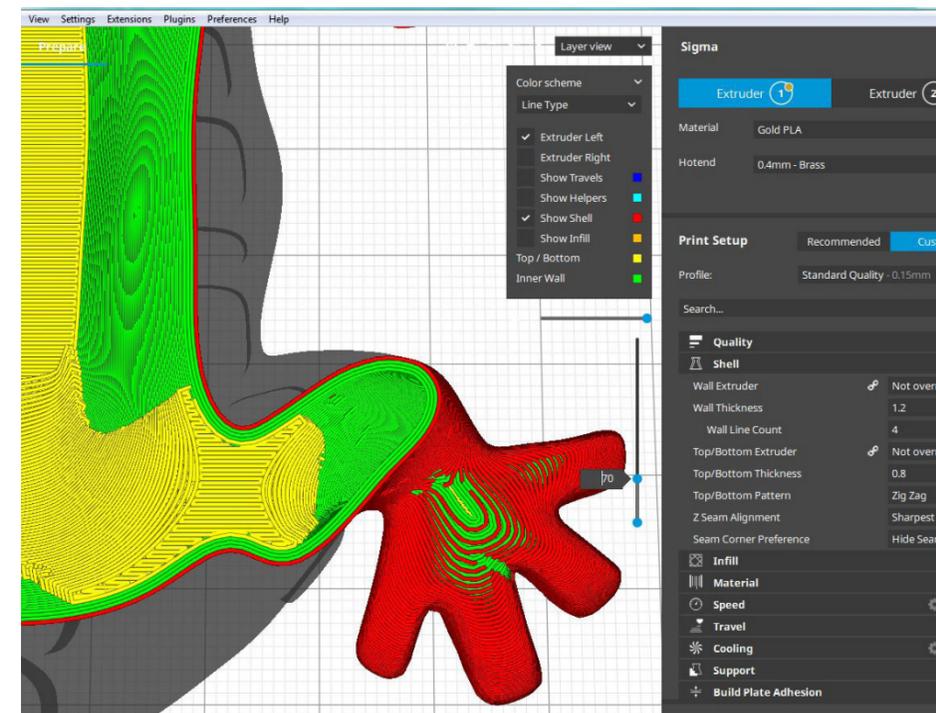
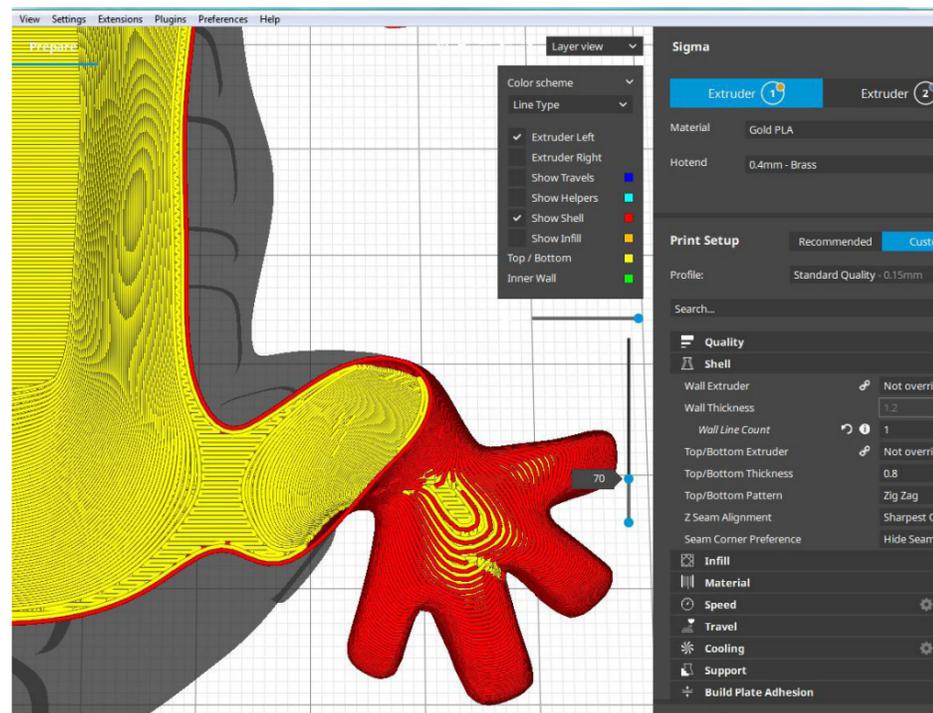


In the following sections the most important parameters in the 3D Printing process will be explained, dividing them into the same categories present at the Custom Print Setup tab.

## Shell

**Wall Thickness:** Number of perimeters compounding the wall of the model. BCN3D Cura rounds the wall thickness to a multiplication of the line width. In general, a wall thickness of two or three times the line width is sufficient. A higher value will create a sturdier model and decreases the chance of leaks, while a lower value can significantly decrease the print time and filament costs.

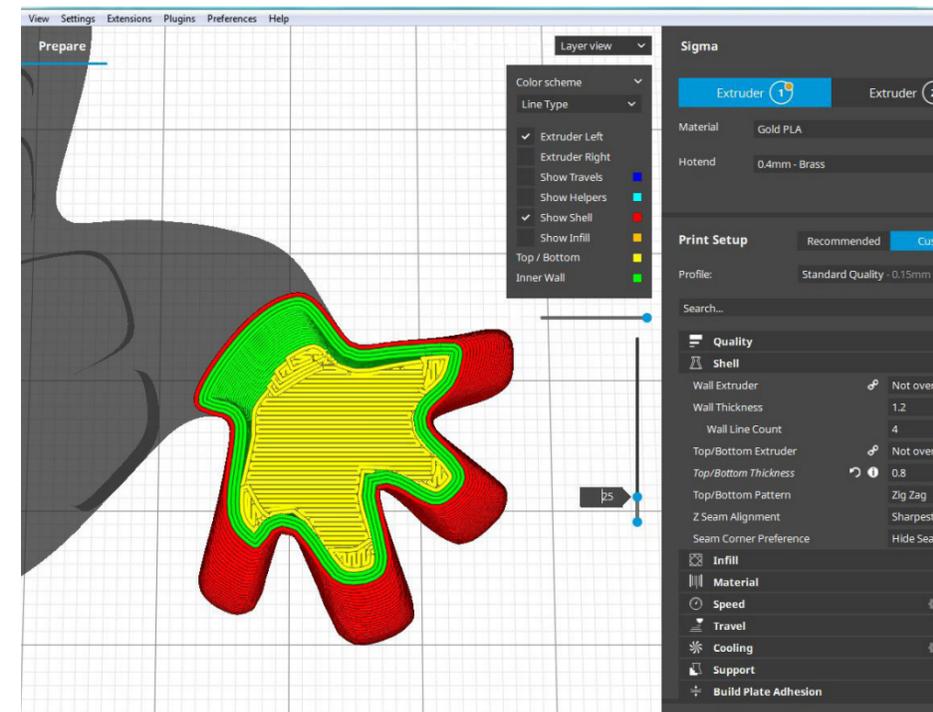
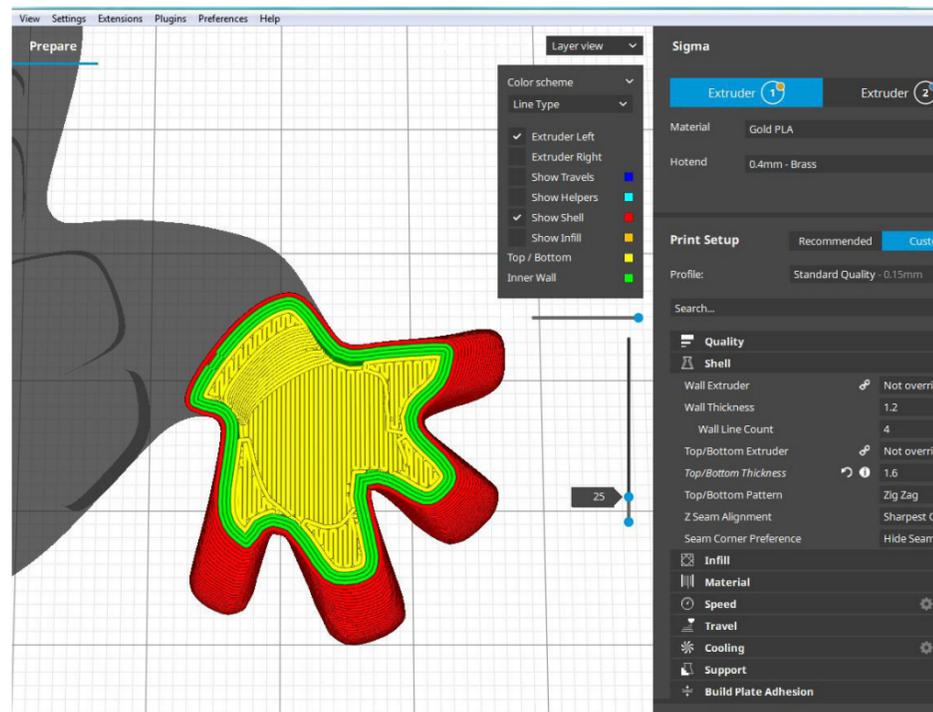
Instead of setting a thickness in millimeter of the walls, you can also set a number of walls. When you set the wall line count, the wall thickness is calculated and will grey out.



## Shell

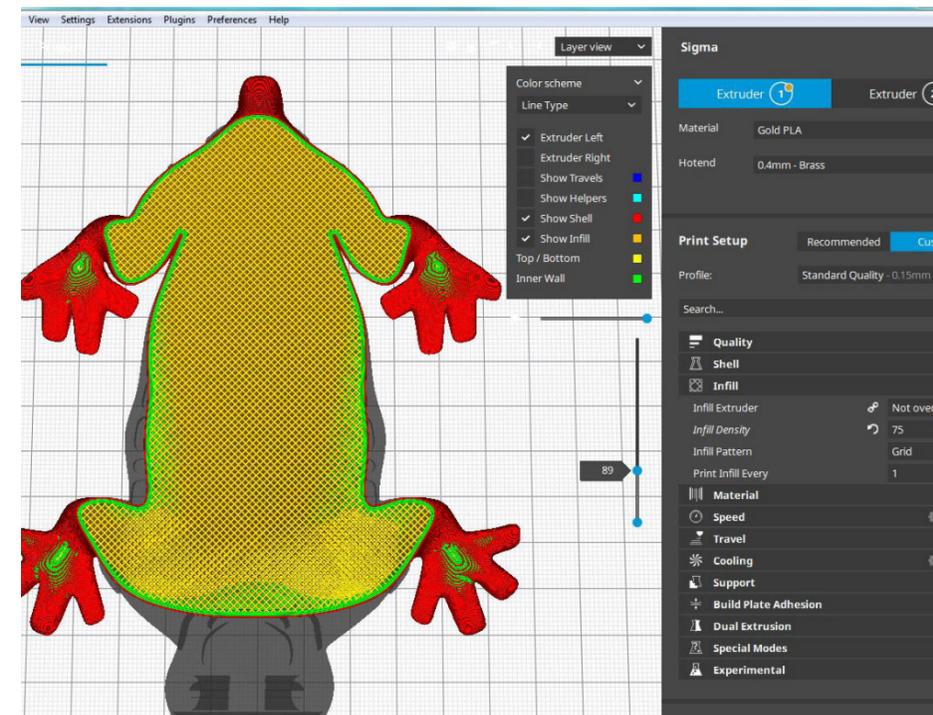
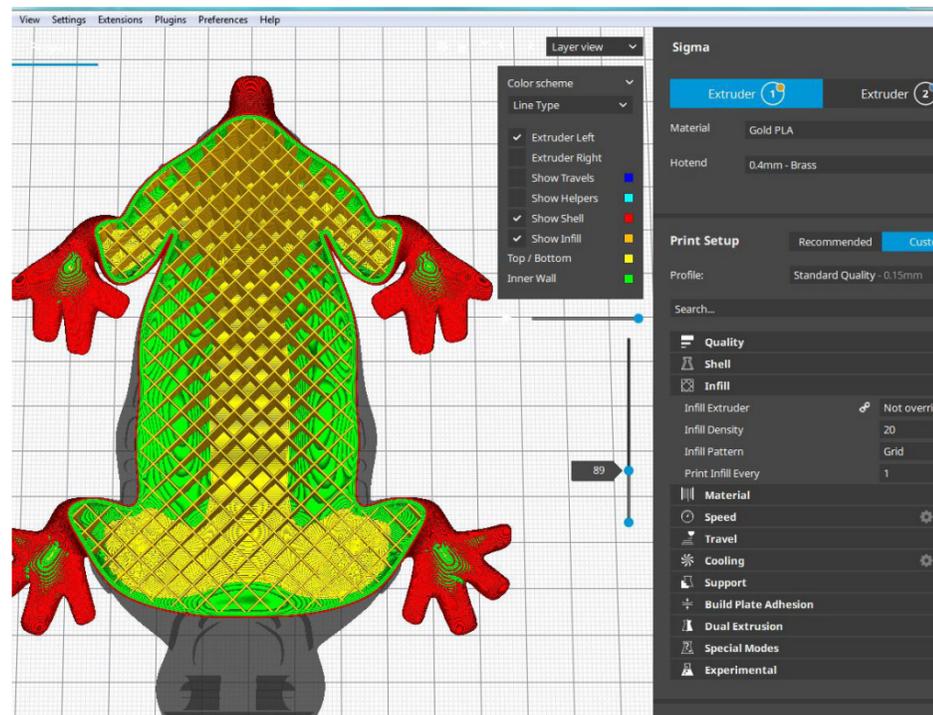
**Top/Bottom Thickness:** With the top/bottom thickness you can set the thickness of the solidly printed top and bottom layers of the print. A higher value ensures all gaps on the top and bottom layers are closed completely. However, this can also increase the print time and amount of filament used.

It is advised to always use a multiple of the layer height for the thickness of the top and bottom.



## Infill

**Infill Density:** 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. An infill density around 20% is used for models with a visual purpose, higher densities can be used for end-use parts.



## Infill

Infill Pattern: BCN3D Cura allows you to change the pattern of the printed infill structure, which is beneficial in some use cases. For example:

Strong 2D infills are used for everyday prints.

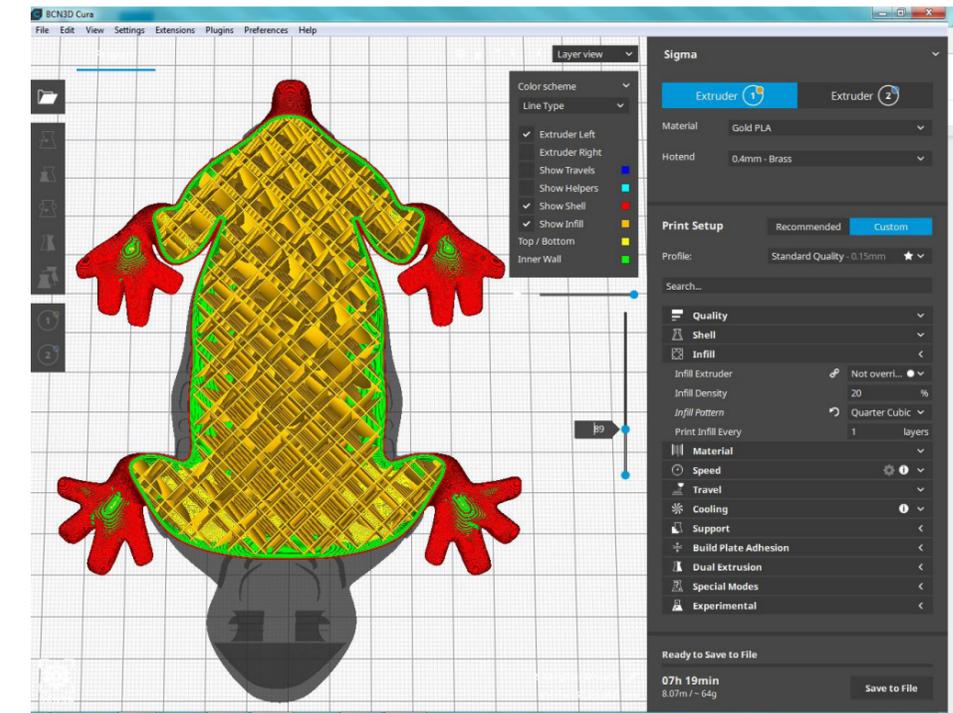
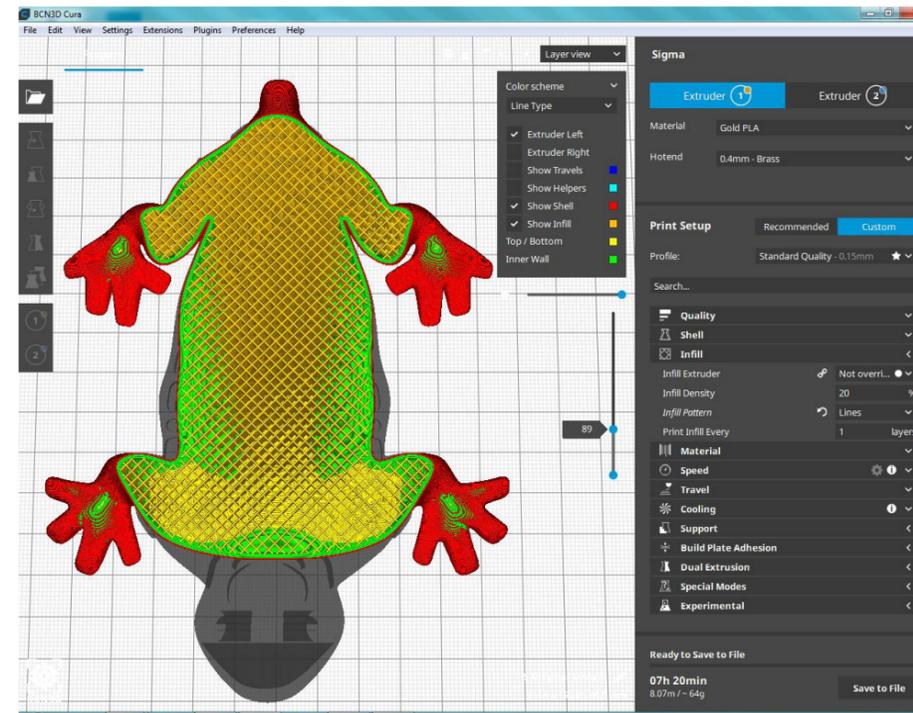
Quick 2D infills are used for quick, but weak models.

3D infills are used to make the object equally strong in all directions.

3D concentric infills are used for flexible materials.

The following pattern options are available:

- Grid: Strong 2D infill
- Lines: Quick 2D infill
- Triangles: Strong 2D infill
- Tri-hexagon: Strong 2D infill
- Cubic: Strong 3D infill
- Cubic (subdivision): Strong 3D infill (this saves material compared to Cubic)
- Octet: Strong 3D infill
- Quarter cubic: Strong 3D infill
- Concentric: Flexible 3D infill
- Concentric 3D : Flexible 3D infill
- Zig-zag: A grid shaped infill, printing continuously in one diagonal direction
- Cross: Flexible 3D infill
- Cross 3D: Flexible 3D infill



## Material

**Printing Temperature:** This refers to the temperature of the nozzle while printing, including the adapted extrusion rate. Each printing profile has a slightly different printing temperature to create the best print result.

**Build plate Temperature:** This setting defines the heated bed temperature during the printing process. Each material has an ideal build plate temperature, which is set here.

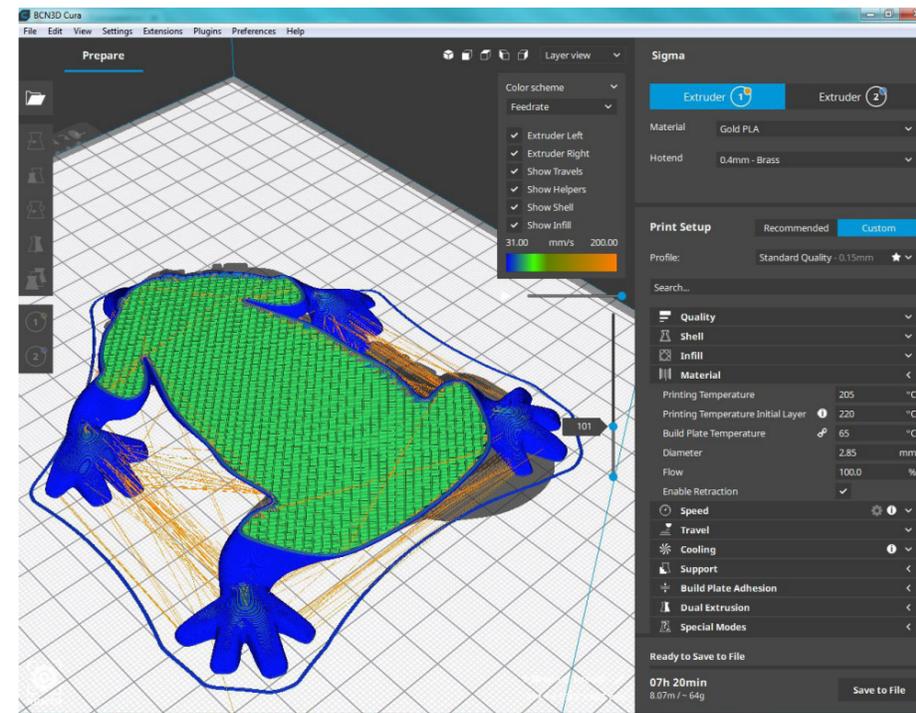
**Flow:** It refers to the volume of plastic extruded. The automatically calculated value (100%) represents the theoretical amount of extruder plastic based on the filament diameter and feeder of the printer.

## Speed

**Print speed:** The print speed defines the speed (in mm/s) at which the tool head moves while printing. Based on this setting, BCN3D Cura calculates the extrusion flow. The print speed can be visualized per feature in the *Layer view* --> *Feedrate*.

A higher print speed will lead to a shorter print time. Keep in mind that increasing the print speed means that you may have to increase the temperature as well to ensure the filament is properly melted.

Although you can choose one overall print speed for the complete print, it is also possible to use different print speeds for specific parts of the print, such as infill, walls or supports.



**Travel speed:** This is the speed at which the print head moves when it's not extruding. A higher travel speed decreases the chance of filament oozing from the nozzle, resulting in a cleaner object. However, higher speed could also cause the nozzle to hit a previously printed part, which may damage the print due to the heated nozzle. This can be prevented by using Z-hop when retracting.

The travel speed for the initial layer differs from the rest of the print to ensure proper adhesion with the build plate.

## Cooling

**Enable Print Cooling:** With this setting you can enable or disable the print head fans during printing. The print head fans ensure that the material is cooled properly before the next layer is printed. For layers with a short layer time, and those with bridges/overhangs, cooling will increase the print quality.

However, cooling may affect the layer adhesion of certain technical materials, such as ABS, causing layer cracks or mechanical defects. For this reason, some profiles set the cooling fans off.

BCN3D Cura will automatically regulate the cooling fan speed depending on the layer time, geometry or height of the model.

## Support

**Generate Support:** Some models have overhanging parts, which means that parts of the model float mid-air when you would print the model. In this case, you must print a support structure under the model to prevent the plastic from falling down. This can be achieved by enabling this setting.

**Support Extruder:** Thanks to the IDEX architecture, BCN3D printers are able to print support structures with a water-soluble material called PVA, instead of using the exact same material of the model. Choose the extruder responsible to print the supports.

It is also possible to specifically choose the extruder to print certain parts of the supports: specific parts of the supports:

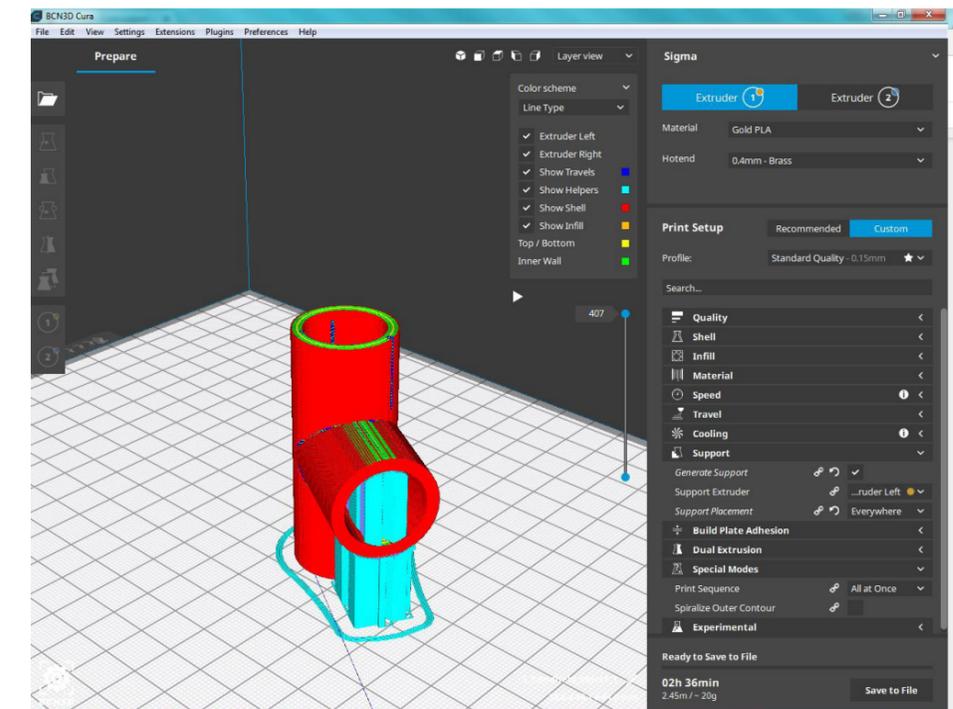
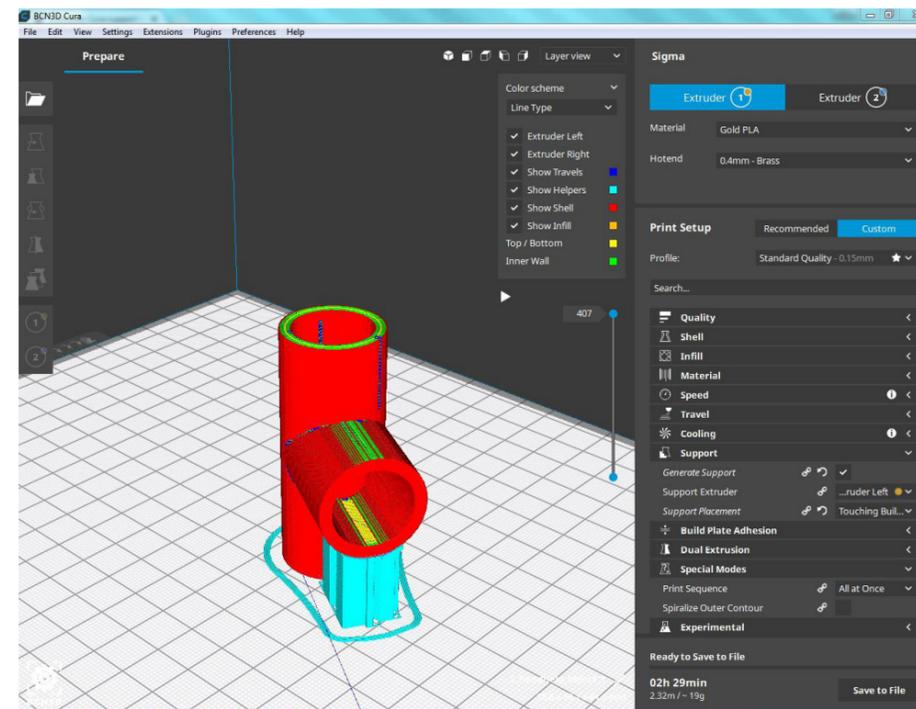
- Support infill
- First layer support
- Support interface
- Support roof
- Support floor

This can be useful to save material and money, since PVA is more expensive than most of the filaments.

## Support Placement

This setting defines where the support structure is printed. It contains the following options:

- Touching build plate: Support material is only printed from the build plate up.
- Everywhere: Support material is printed below every part that needs support, which means that it can also be placed on or inside a model.



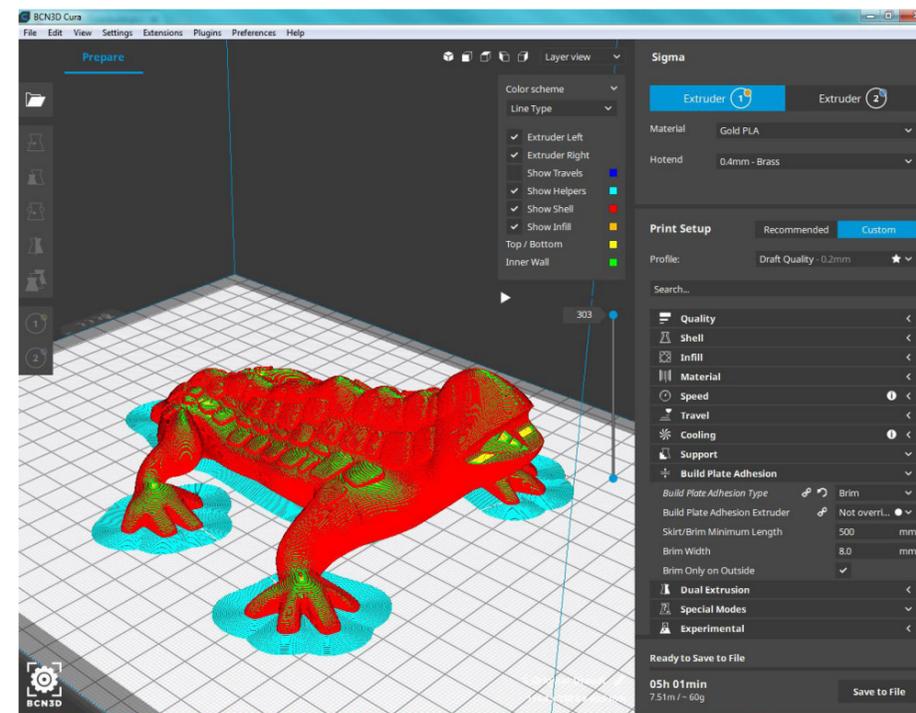
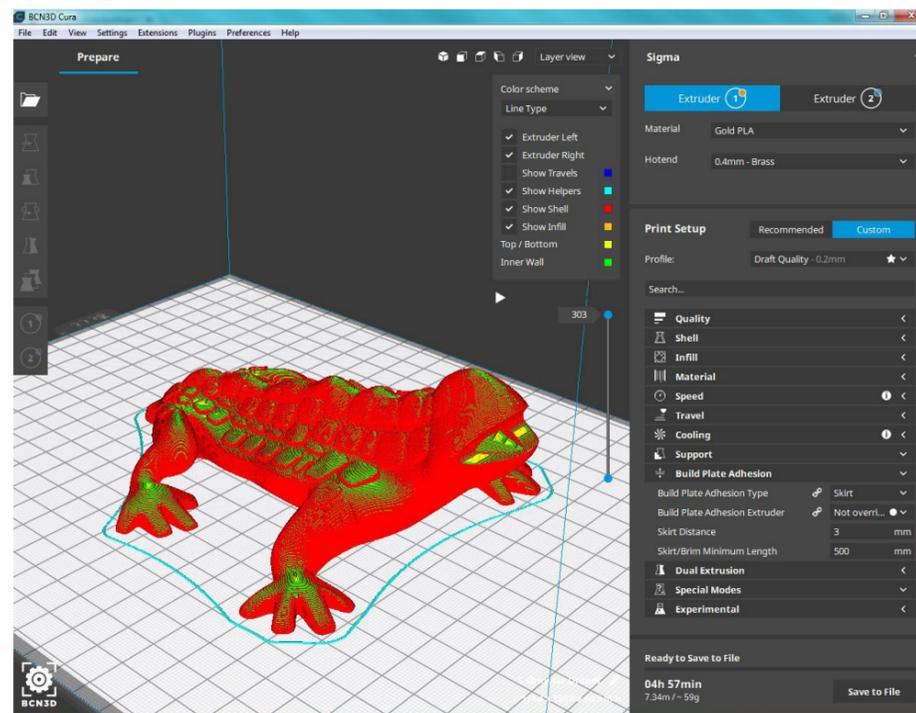
## Build Plate Adhesion

**Build Plate adhesion type:** First layer is the most important moment of the 3D Printing process. A good first layer decreases significantly the chance of failure. For this reasons, there are different strategies to ensure a proper adhesion of the deposited plastic onto the build plate:

**Skirt:** A skirt is a line printed around the object on the first layer, but not connected to the object. This helps prime the extrusion nozzle and can be an additional check for bed leveling before the print begins.

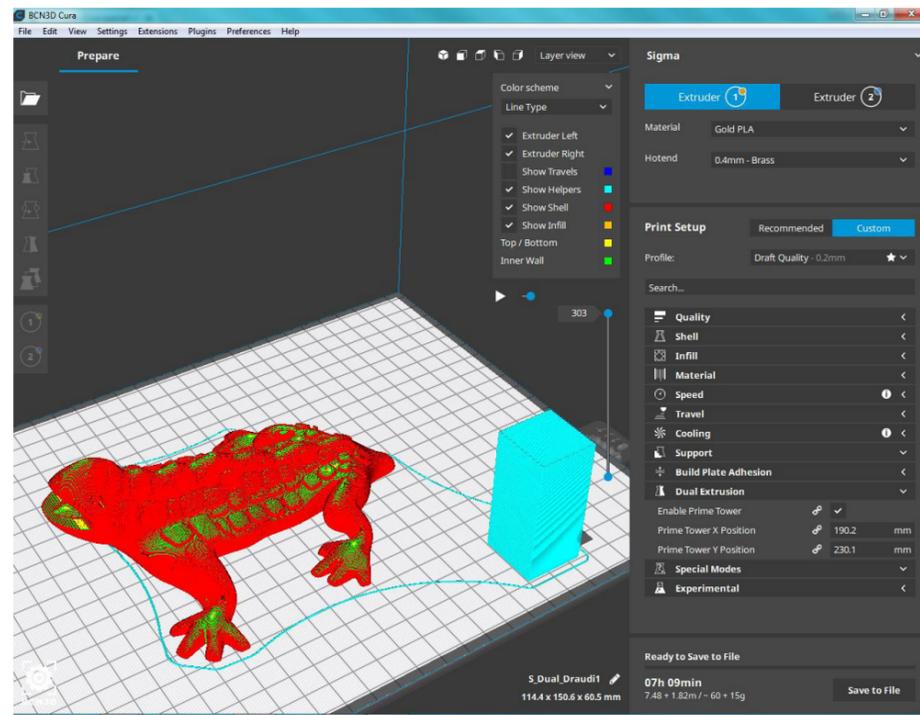
**Brim:** Brim adds a single layer flat area around the base of the model to prevent warping. The brim is connected to the model and makes the bottom surface area bigger. This increases the adhesion to the build plate and, in case of warping, the corners of the model are less likely to curl up because of the brim attached to it.

**Raft:** A raft adds a thick grid with a roof between the model and the build plate. This can be useful when the bottom surface of a model is not completely flat or has little adhesion to the build plate. A raft ensures that the model will stick better to the build plate.



## Dual Extrusion

**Enable Prime Tower:** A prime tower is an additional print that's created on the build plate to help prepare the nozzle before printing the next layer. It reduces oozing, under extrusion and enhances overall printing quality. Its use is highly recommended when a dual job is printed.



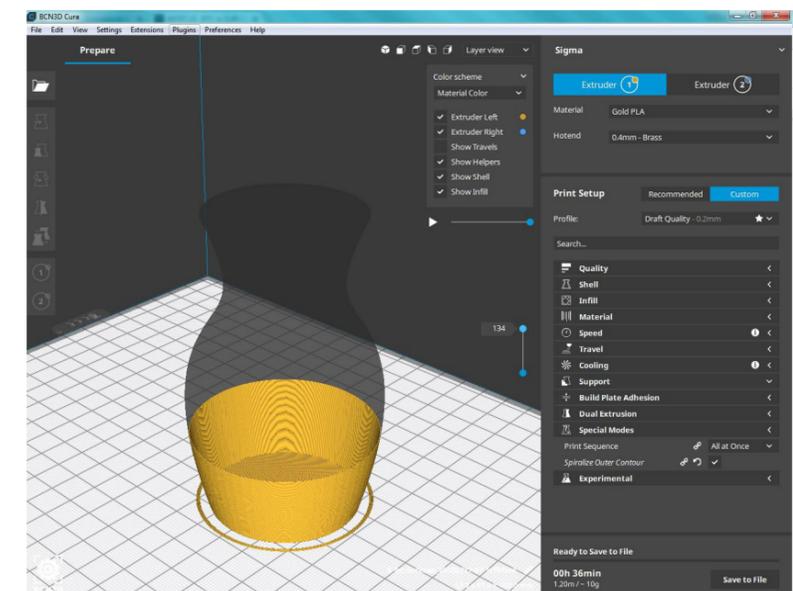
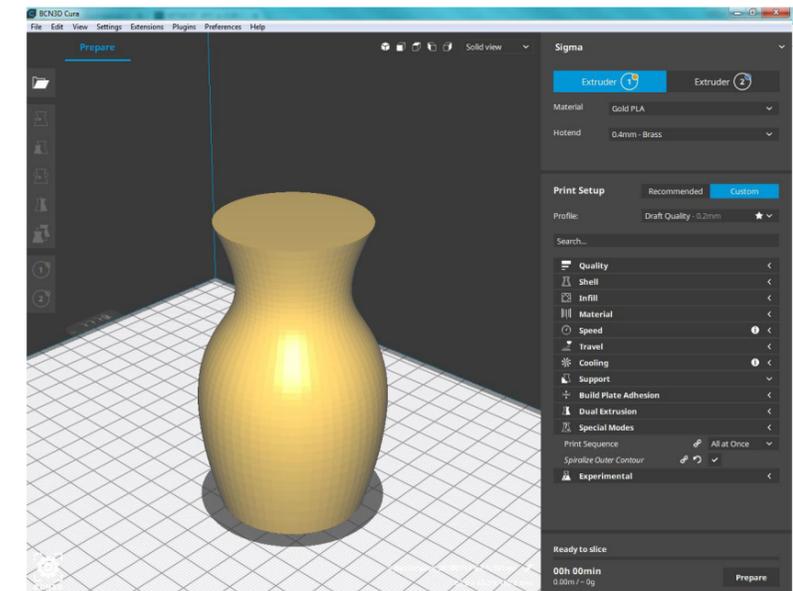
## Special Modes

**Print Sequence:** When multiple models are placed on the build plate, you can define the sequence in which they should be printed. This option is only available on single extrusion print jobs. There are two options:

*All at once:* All objects will be printed at the same time, which means that it will print one layer of each object before moving onto the next layer.

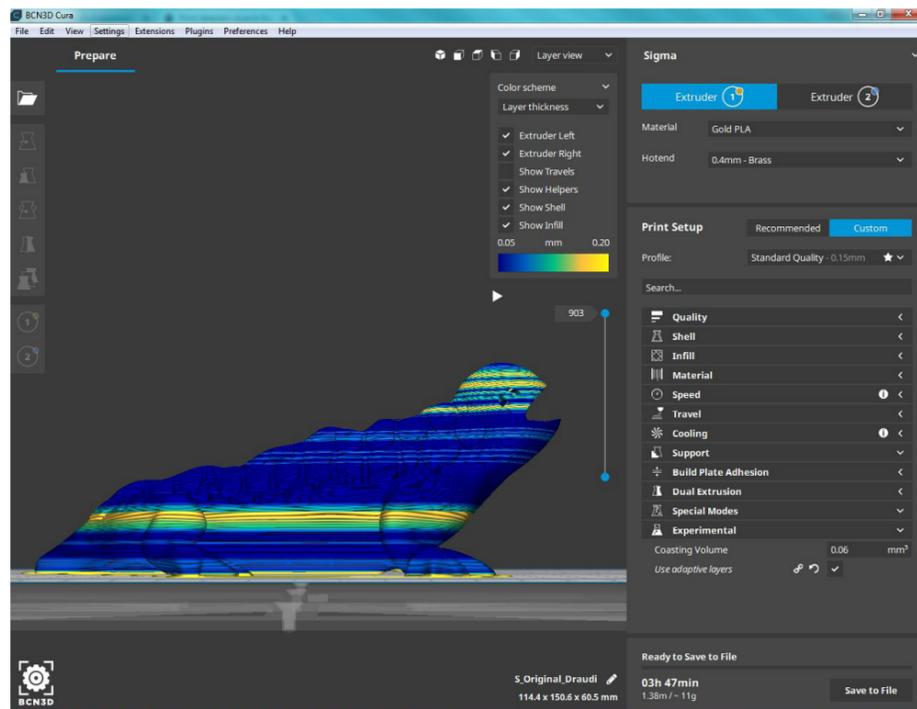
*One at a time:* Objects will be printed one by one. This mode can only be used if the models are smaller than the gantry height, which is the distance between the build plate and lowest print head shaft.

**Spiralize Outer Contour:** Typically, the printing platform moves down by the layer height after printing a complete layer. Spiralize mode smooths out the Z move over the edge. This means that the Z Axis will gradually move down over the whole print, like a spiral. Solid objects are turned into single-walled prints with a solid bottom by using this option, which is perfect for printing items like vases, for example.



## Experimental

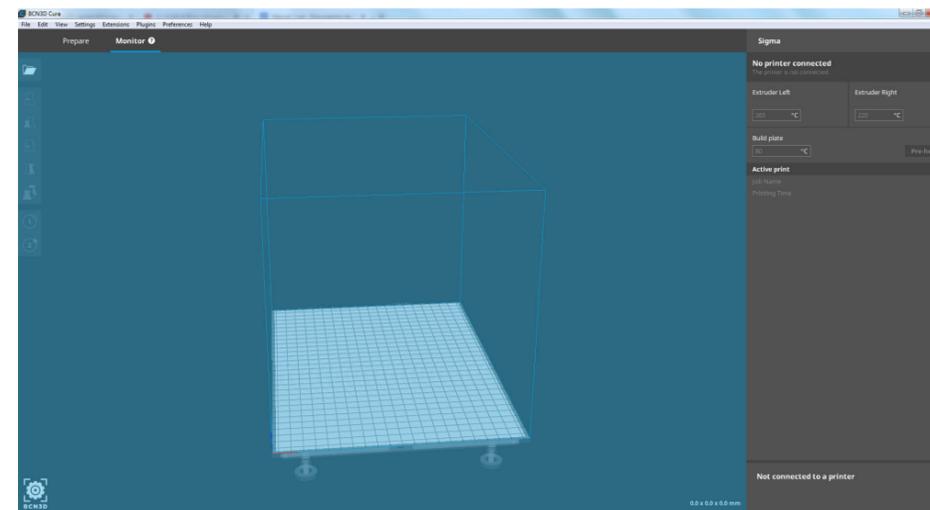
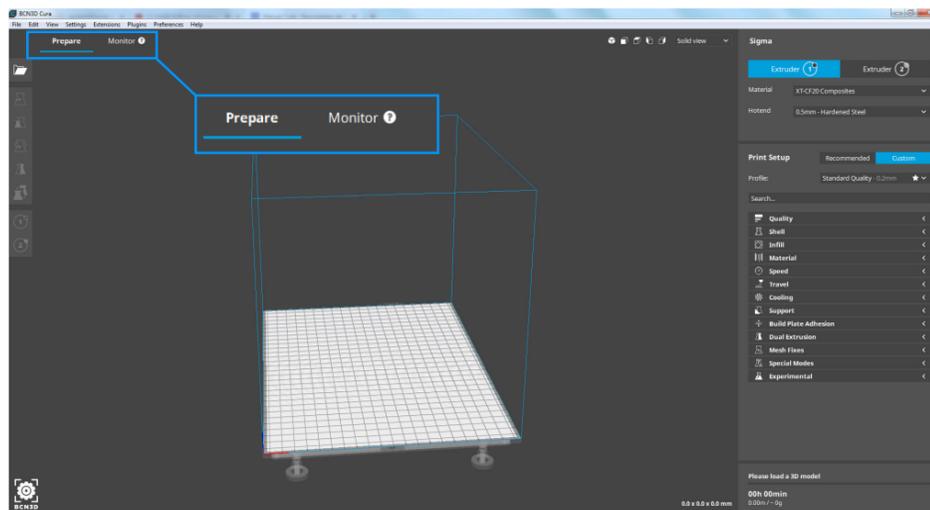
**Use Adaptive layers:** This feature analyzes the slope and angle of the outer wall of the model. The slicing engine will then optimize the layer height for each section. Curves are printed with a finer layer resolution, while vertical, flat areas are printed with thicker layers, often reducing the print duration.



**TUTORIALS-HOW TO**

## Monitor Printing

BCN3D printers are compatible with the printing via USB. On the top side-bar, click the Monitor Option to enter the USB printing menu.



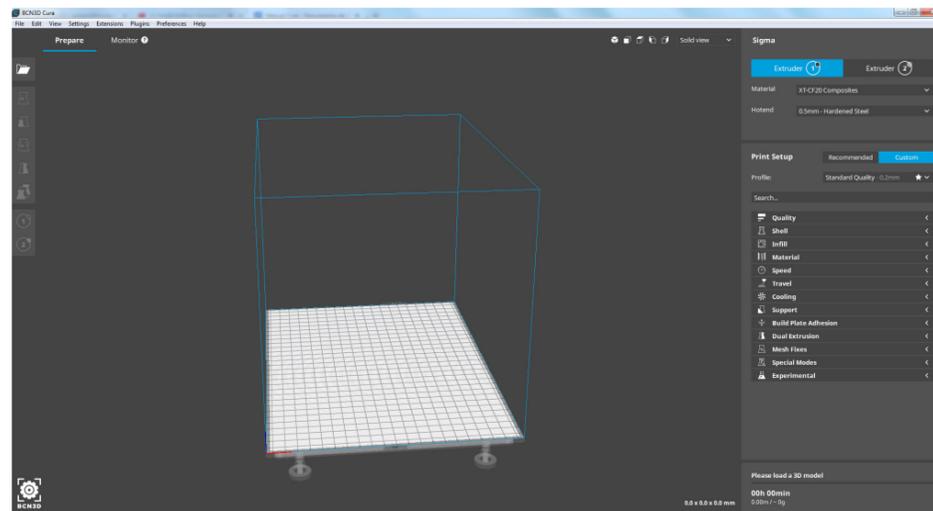
Once there, plug in the BCN3D printer and you will be able to:

- (1) Heat Extruder Left
- (2) Heat Extruder Right
- (3) Heat Build Plate
- (4) Prepare your job
- (5) Print via USB

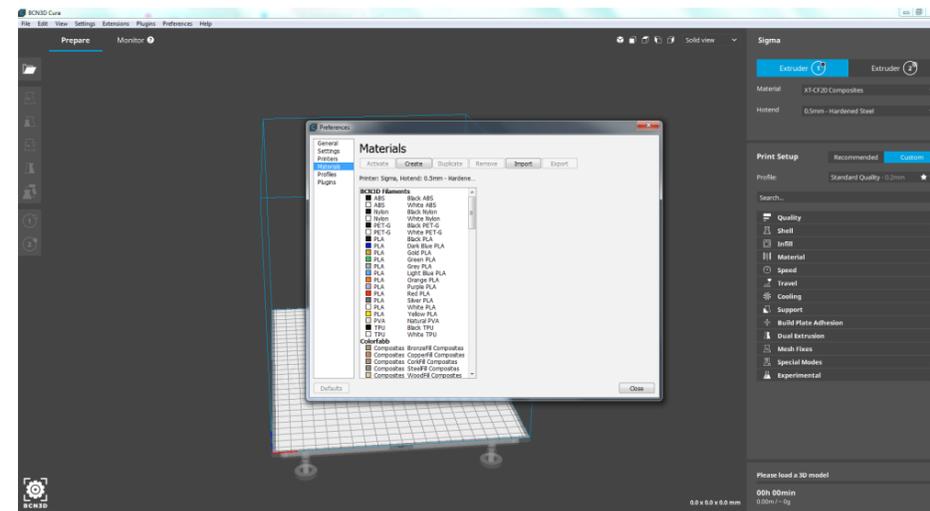
## Calculate the price of your part

When using the printers, most of the times an expense control is required. BCN3D Cura allows you to calculate an estimation of the printed parts price.

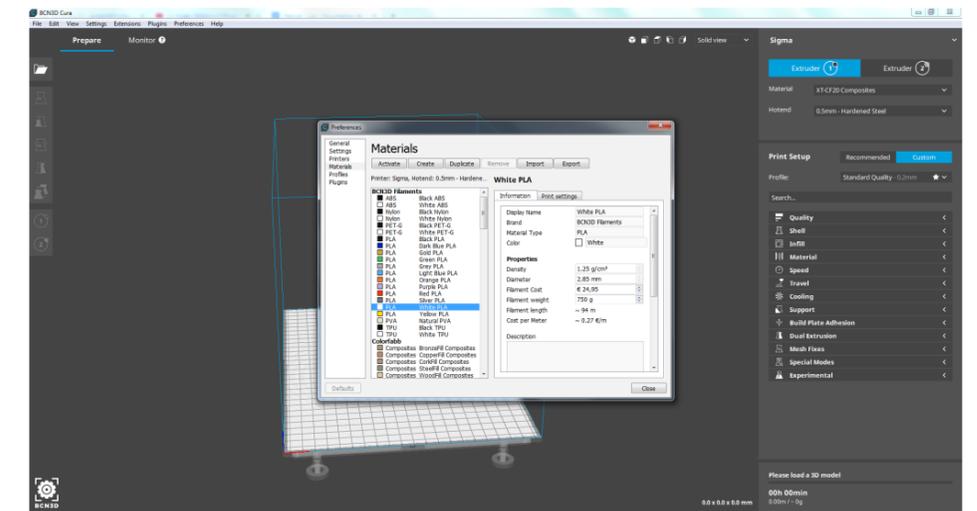
1. Open BCN3D Cura. Go to *Preferences* --> *Configure Cura*.



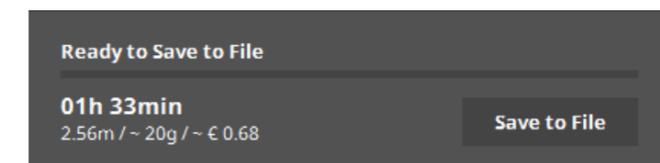
2. Choose Materials on the left side-bar.



3. Select the material to print with and change the Filament Cost and the Filament Weight, depending on the price you bought it (Remember that you can check the prices of BCN3D Filaments at our website).



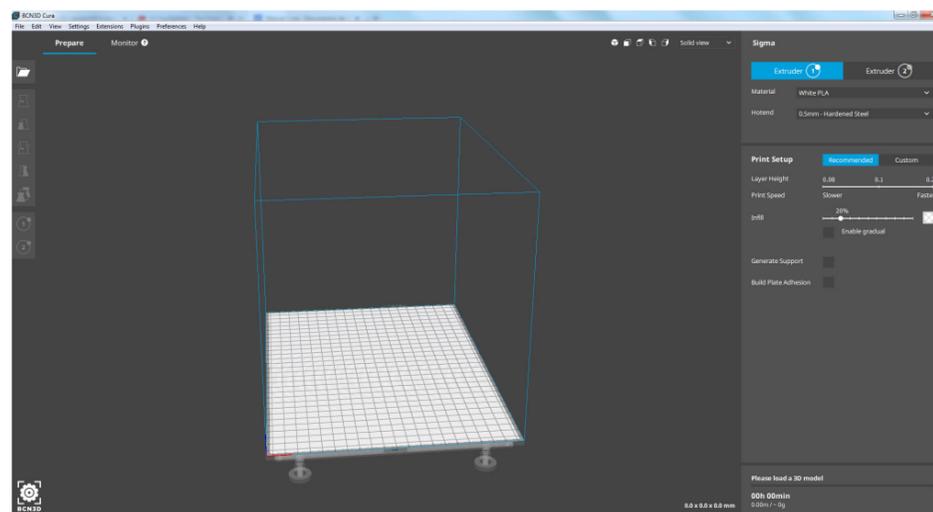
4. Once the job prepared, it will appear an estimation of the time, material used and price.



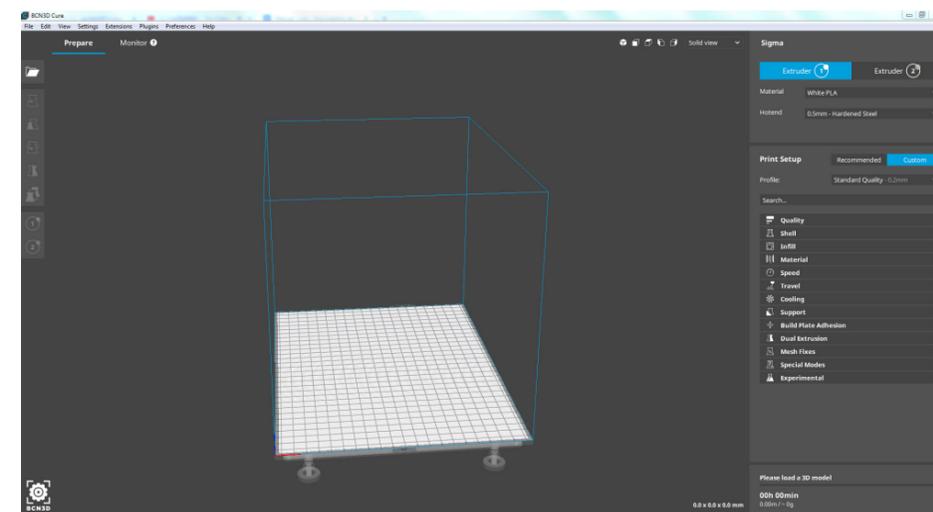
## Visualize all printing parameters on BCN3D Cura

For experts users who want to fully optimize their printing experience changing some parameters, this is a really useful option. With this process, you can activate the visualization of all the printing parameters that the software has, and not only the recommended ones.

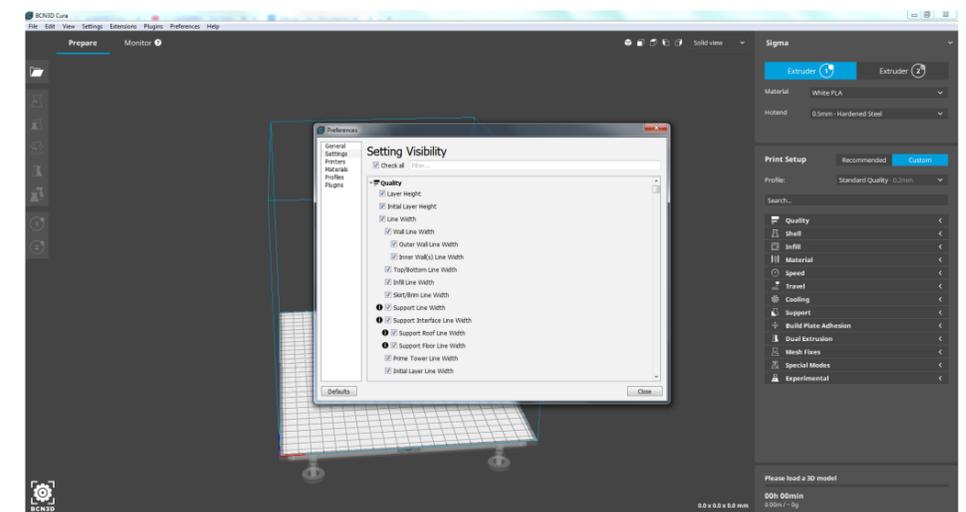
1. Open BCN3D Cura, and click on Custom on the right side-bar.



2. When moving the mouse on the printing parameters tags on the right-side bar, it will appear a gear. Click on it.



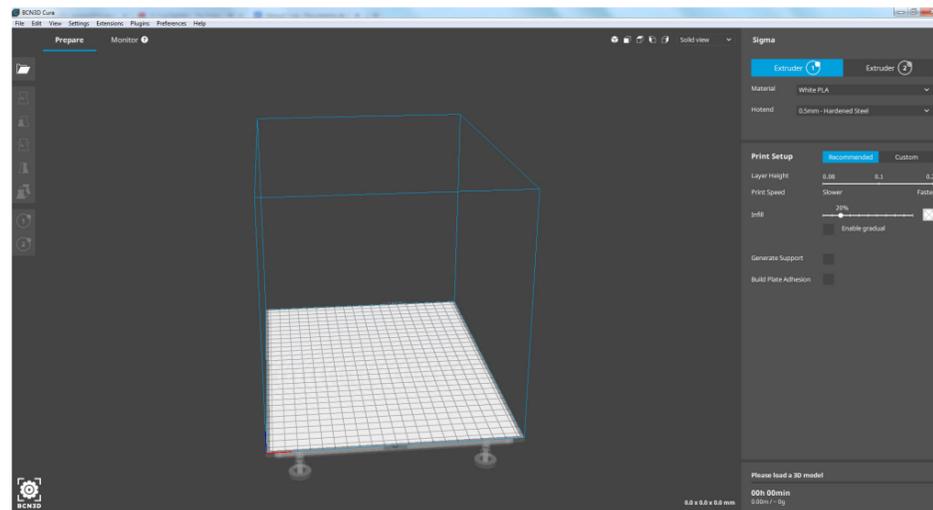
3. On the Setting Visibility menu, click on Check all to activate all the parameters and then Close.



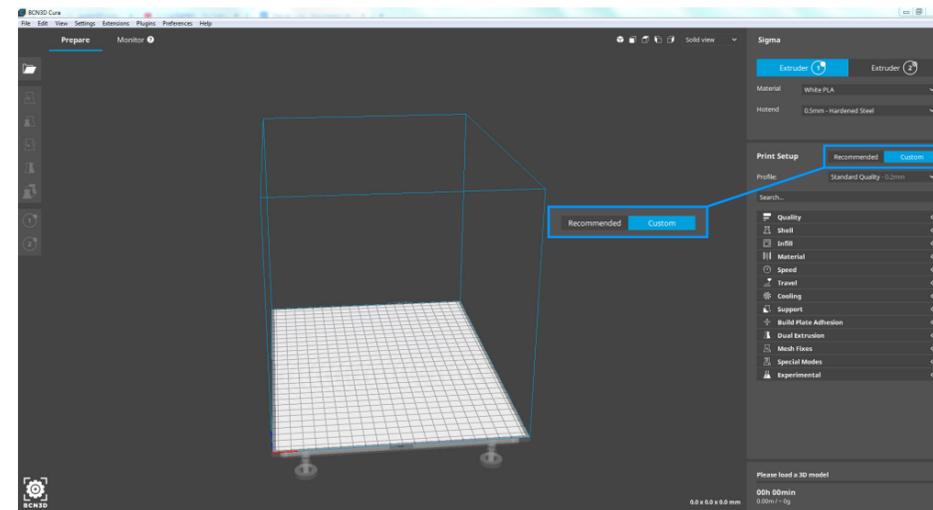
## Z seam Alignment

Z Seam Alignment allows you to choose the point where the printer will move from one layer to the next one. This parameter is useful when the part needs to have a good visual aspect and it can be changed with BCN3D Cura.

1. Open BCN3D Cura.



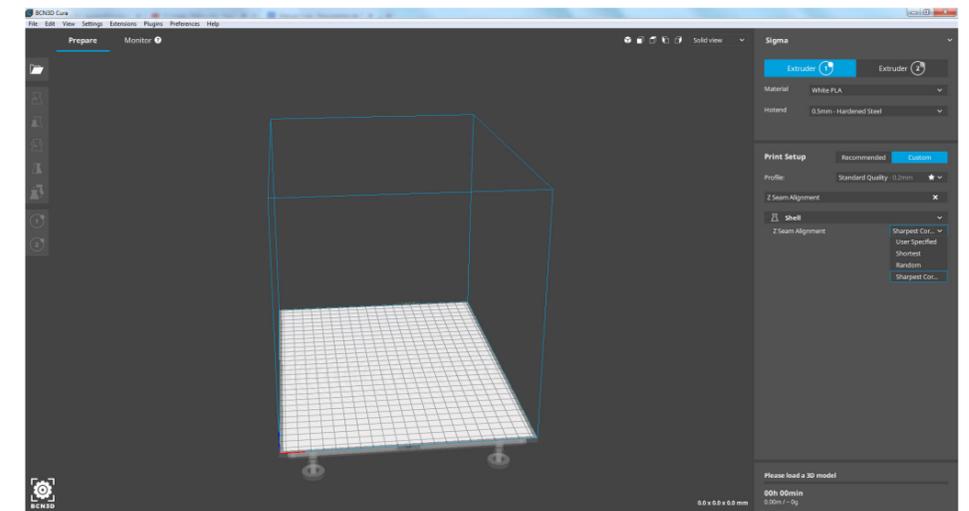
2. Click on Custom on the right-side bar. On the search bar, write Z Seam Alignment.



3. When choosing:

1. User specified: it will be easier to remove, as the user can choose where the layer change will take place.
2. Random: the layer change can be less noticeable as it is randomly distributed.
3. Shortest: the path of the layer will be the shortest one. It may appear a vertical line, corresponding to the all layer changes of the part.
4. Sharpest Corner: it will find the sharpest corner and use it as reference to change the layer.

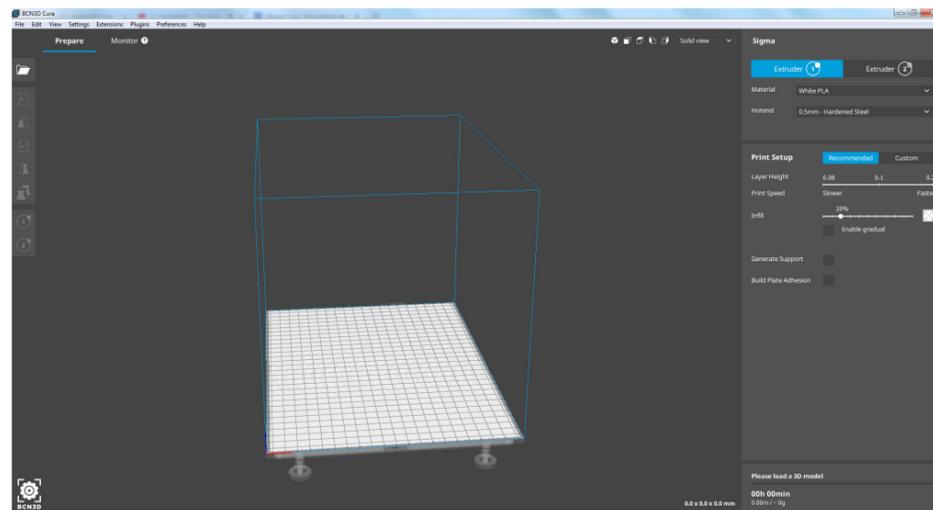
By default BCN3D recommends Sharpest Corner, in order to hide the point where it changes the layer.



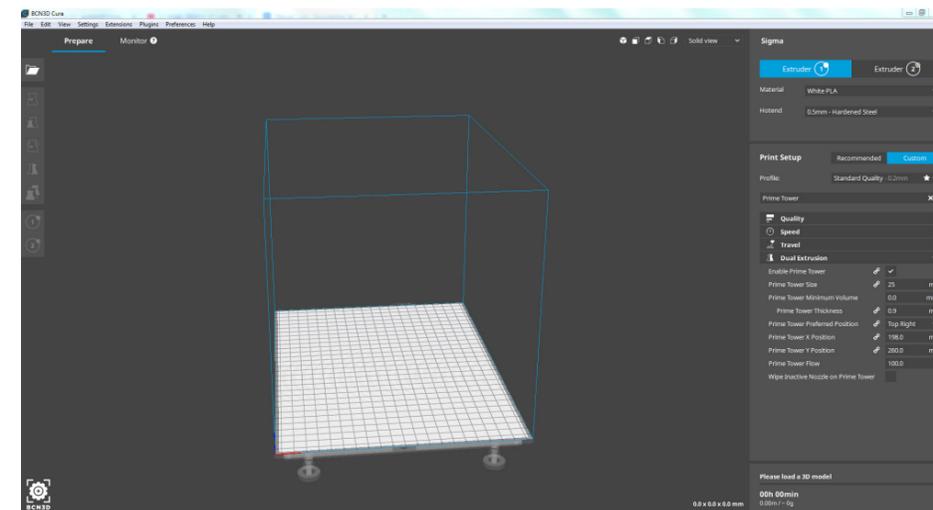
## Prime Tower

The Prime Tower is a purge zone that creates BCN3D Cura to put the nozzle in the working position before printing the part, mostly when using both extruders. It can be enabled/deactivated, or its position can be changed.

1. Open BCN3D Cura.

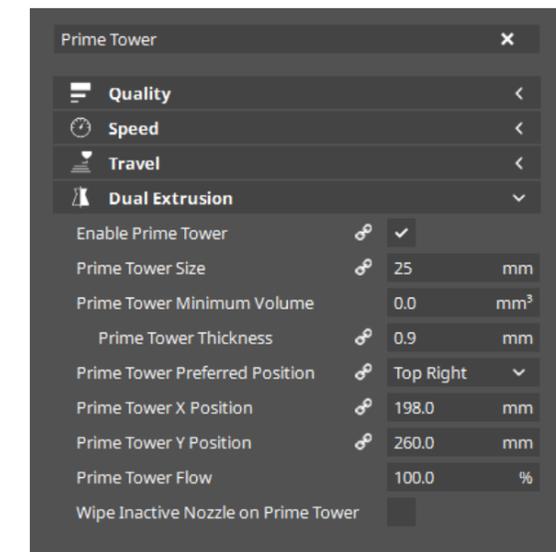


2. Click on Custom on the right-side bar. On the search bar, write Prime Tower. Then find the Dual Extrusion tag on the results.



3. There you can:

- (1) Uncheck the Enable Prime Tower. This option will deactivate the purge tower and it is recommended when having tower sticking problems or to reduce printing time. In case you want to try this option, remember to activate the option Purge at tool change for both hotends.
- (2) Change the thickness of the tower. Also recommended for sticking problems or when some parts of the printed piece have burnt material.
- (3) Move the tower on the XY plane. The tower is automatically placed on the Build Plate, but is an option especially recommended for big prints.

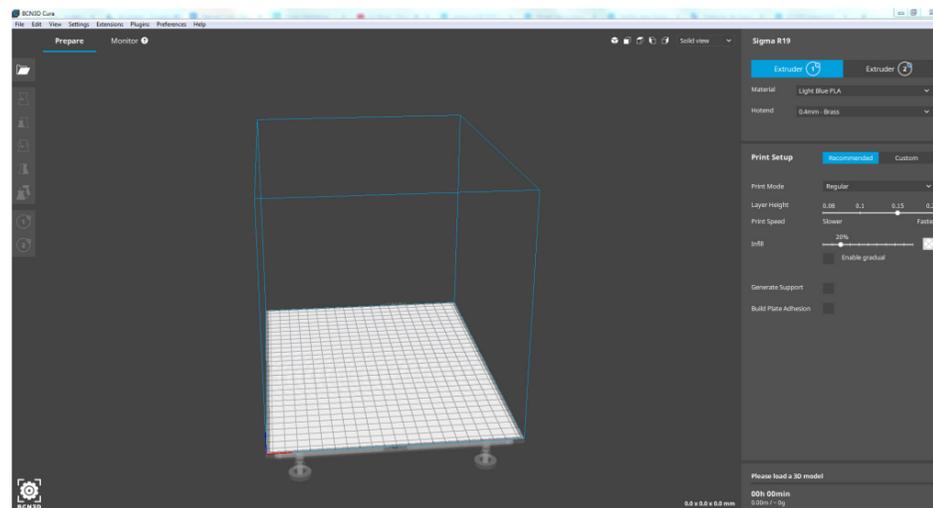


## Print with PVA supports

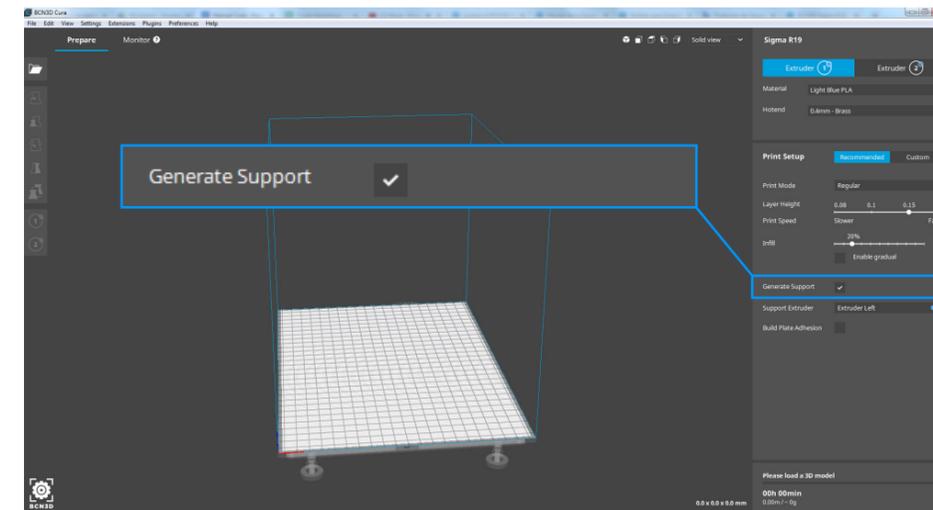
BCN3D Cura is especially prepared to optimize multi-material prints, due to the IDEX system used in BCN3D printers. The main application of this technology is printing water-soluble supports with PVA without having cross-contamination between materials.

Using this option is recommended if you want the finest surface quality when using supports.

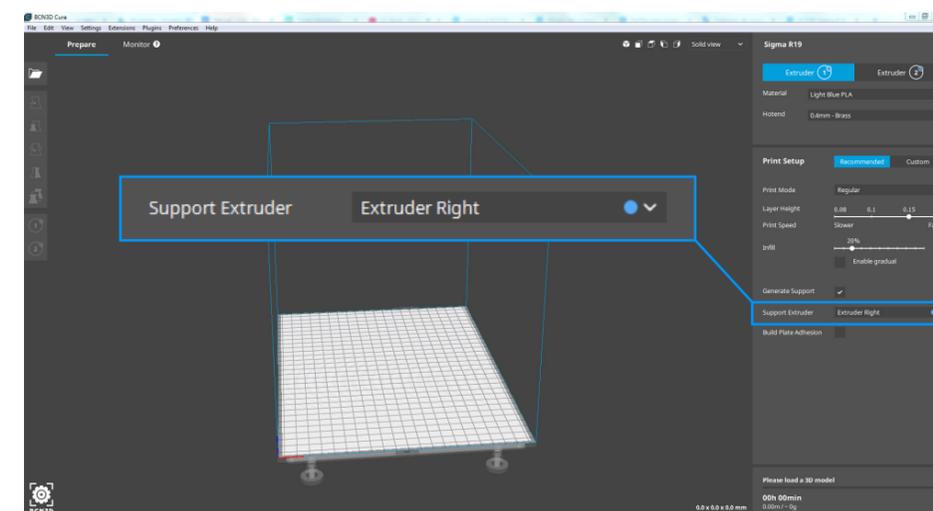
1. Open BCN3D Cura.



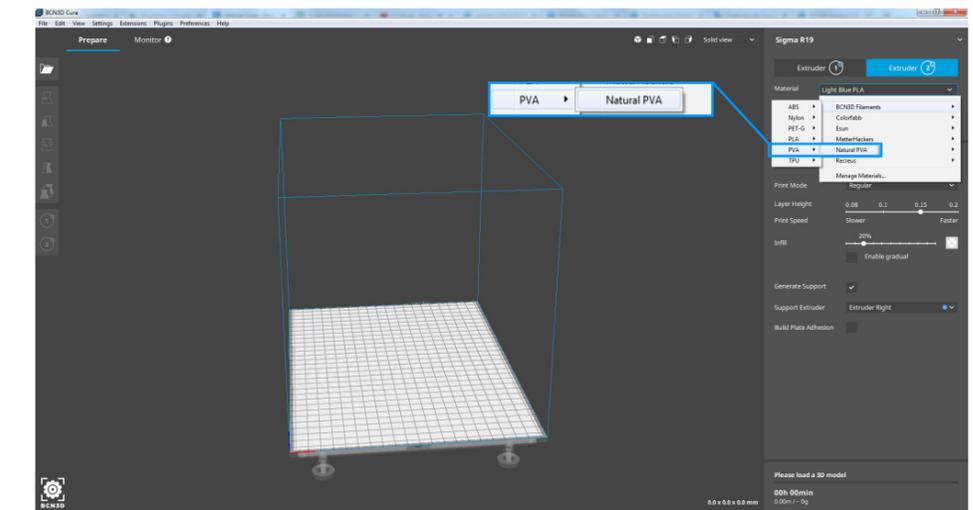
2. Click on Recommended, then check the box of Generate Support.



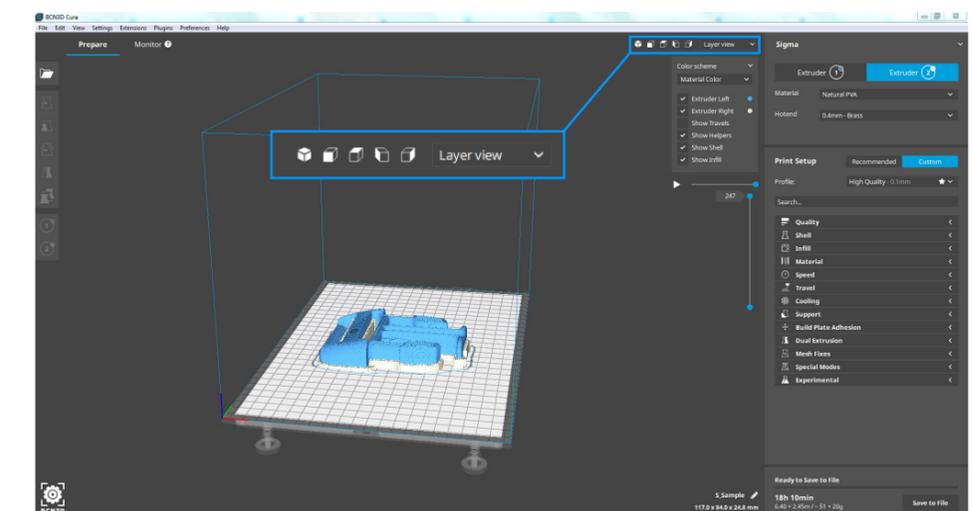
3. Choose Extruder Right on Support Extruder.



4. Move to Extruder Right options by clicking Extruder 2 at the right-side bar. Then choose Natural PVA from BCN3D Filaments as printing material.



5. Once all the configuration is prepared, activate the Layer View. This option lets you see how the printer will produce your part. In the following picture there is an example of a PLA part with a structure of PVA supports.



# Tutorials - How To

## Your first dual print

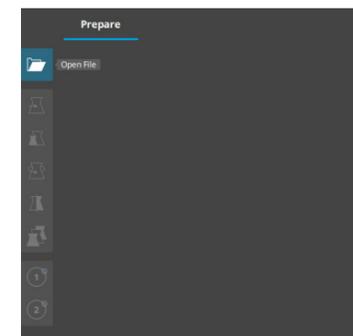
1. Open BCN3D Cura previously downloaded and installed.



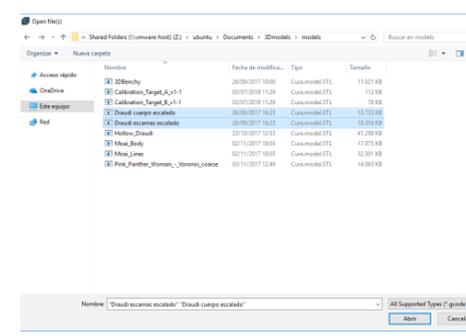
2. Select your printer in the menu Add printer.



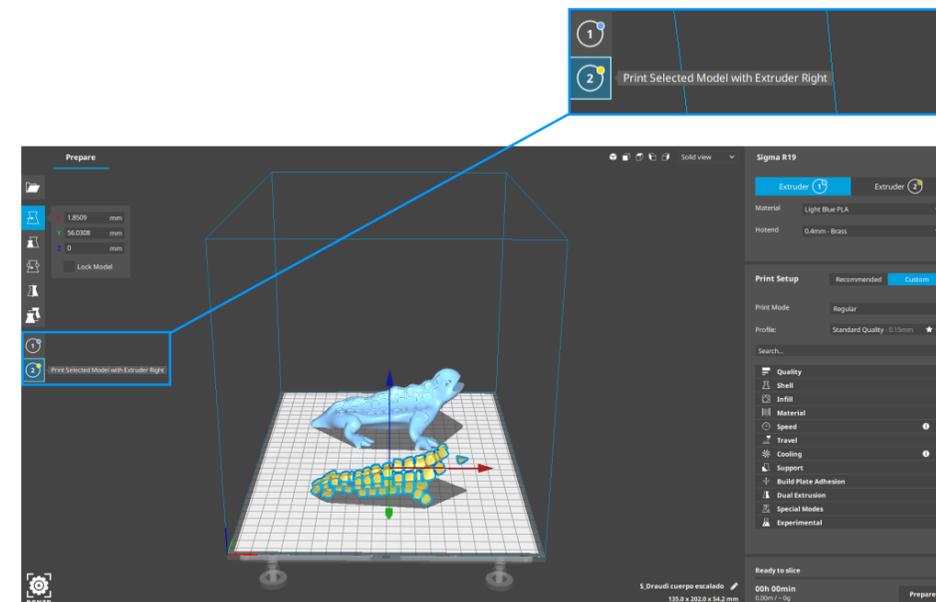
3. Click *Open File*.



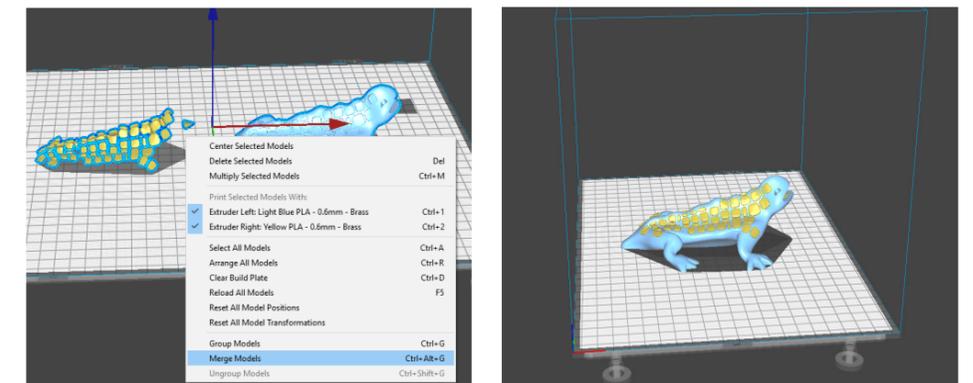
4. Select files to print.



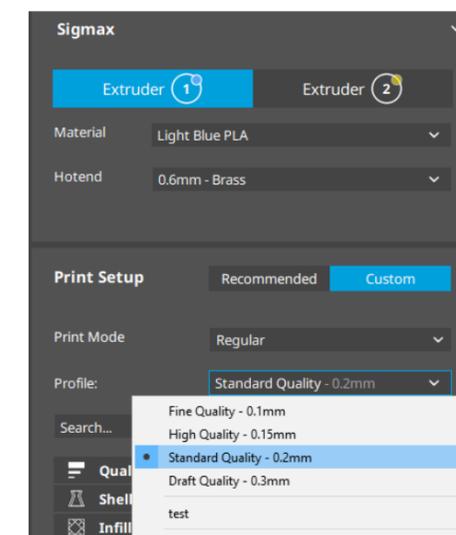
5. Click the model and assign the extruder to print with.



6. Select the models by Shift+Left Click. Selected models will be highlighted with a blue outline. Right click on any of the selected models, and select Merge Models.



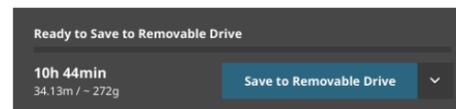
7. Select profile.



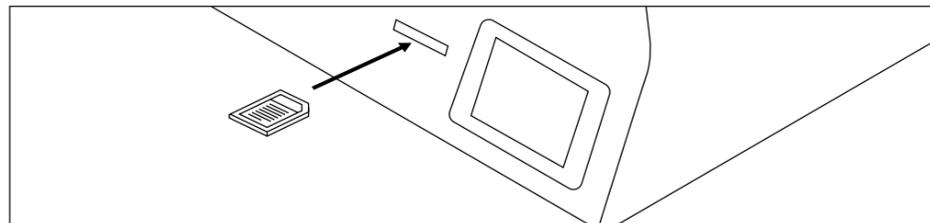
8. Click Prepare to generate the printing file.



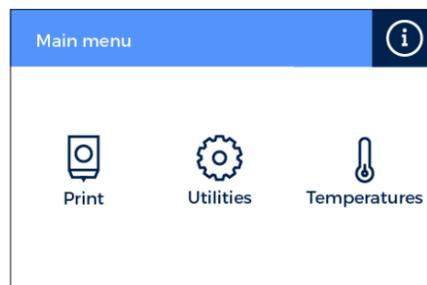
9. Insert the SD card in your computer and save the file.



10. Safely remove the SD card from the computer and insert it in the printer. After that, turn it on.



11. Select *Print* in the main menu of Sigmax's touchscreen.



12. Select the file to print.

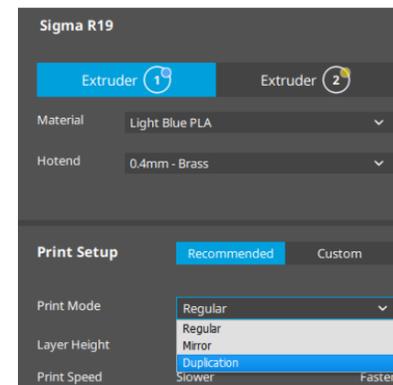


## Your first Mirror/Duplication print

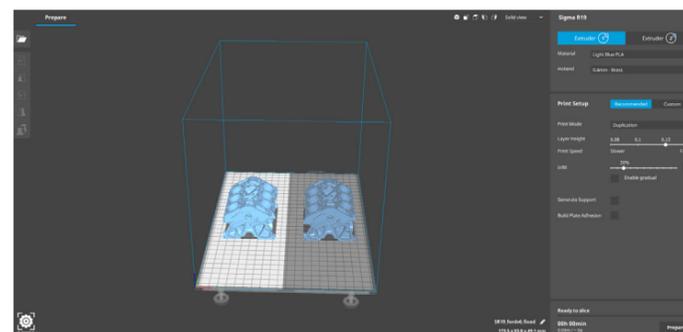
1. Load files to print and move, scale or rotate the models as needed.

2. Select the printing profile for the Left Extruder. Make sure the hotend and material loaded at Right Extruder are the same than in the Left Extruder.

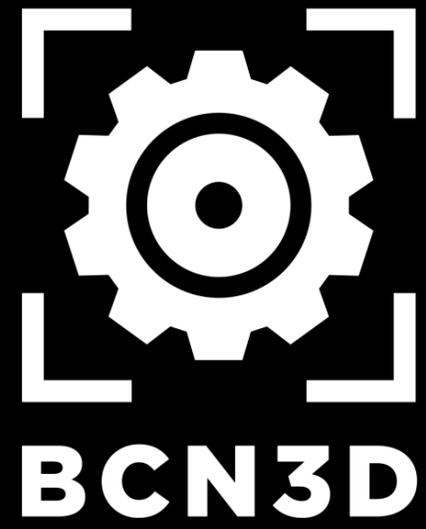
3. Select the desired printing mode in the drop-down list.



4. Note how the available printing volume is reduced and the loaded models automatically move to the left. These models can still be modified. A preview of the duplicated or mirrored parts is available at the right.



5. Click Prepare, save the file and print it.



Contact us if you have questions about any process:

[support@bcn3dtechnologies.com](mailto:support@bcn3dtechnologies.com)