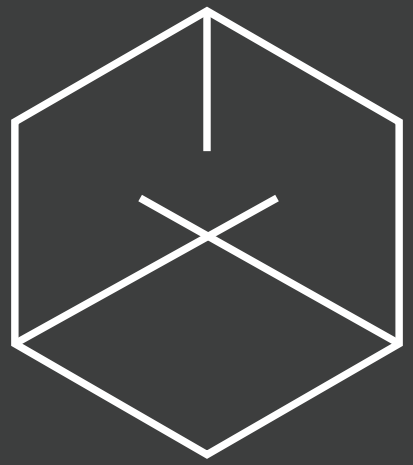


LUGO Cura

Getting Started

- Getting Started
- Compatible Operating Systems
- Incompatible Operating Systems
- Minimum System Requirements
- Recommended System Requirements
- Installing LUGO Cura
- Printer Setup
- Language Settings

Getting to Know LUGO Cura

- Interface of LUGO Cura
- Move Tool
- Recommended Mode
- Custom Mode
- Managing Printers
- Managing Printing Profiles
- Preview

Printing

- Workflow Explained
- Loading Files
- Single Printing
- Dual Printing

Automatic Support Generation

- Creating Automatic Supports
- Using Support Blocker
- Using Modeled Supports (Merge Feature)
- Using Dual Extruders for Soluble Supports

Frequently Used Features

- Per Model Setting
- Material Profiles / Adding
- LUGOLABS Support Page

Get start

compatible operating systems

- Windows Vista or higher, 64 bit.

Although Windows 11 is the latest version, it is not yet fully supported and we cannot actively react to any issues found on the platform.

- Mac OSX 11 Big Sur or higher, 64-bit

Ultimaker Cura is 64-bit and therefore not available for the older 32-bit variant systems.

Incompatible operating systems

- Chrome OS
- Virtual desktop applications / operating systems (VDI, Citrix, etc)

Minimum system requirements

- OpenGL 2 compatible graphics card, OpenGL 4.1 for 3D layer view
- Display resolution 1024 x 768
- Intel Core 2 or AMD Athlon 64
- 550 MB available hard disk space
- 4GB RAM memory

Recommended system requirements

- OpenGL 4.1 compatible graphics card for 3D layer view
- Display resolution 1920 x 1080
- Intel Core i3 or AMD Athlon 64
- 600 MB available hard disk space
- 8GB RAM memory

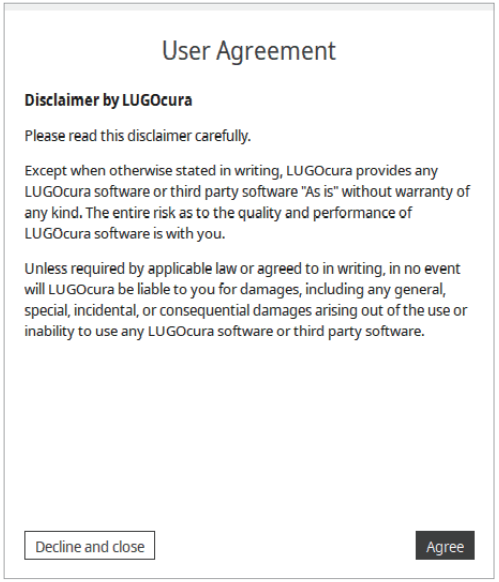
Get start

Installation

Visit **https://www.lugolabs.xyz/lugocura** to download the appropriate version of LUGO Cura for your operating system. When the download is complete, follow the installation wizard. Open LUGO Cura for the first time after installation.

User Agreement

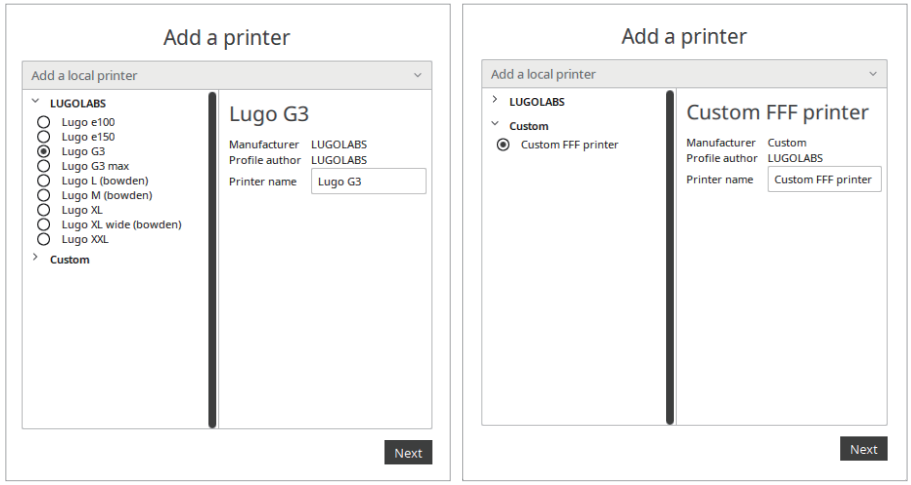
The first time you open LUGO Cura, the user consent form opens. Please read the consent form and click 'Agree'.



Get start

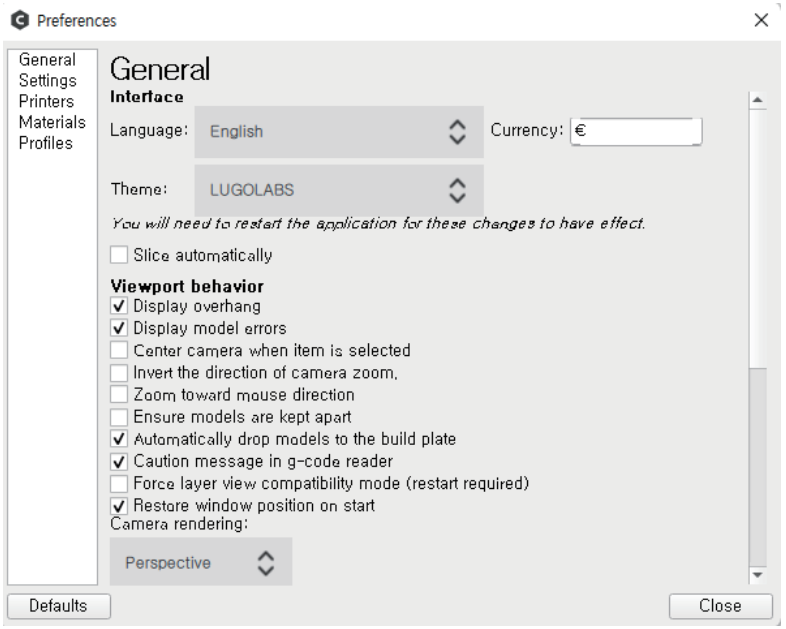
Printer setting

You are prompted to select the following 3D printers.
For LUGOLABS printers, you can select the appropriate model or specify a custom name in Custom.



Language Settings

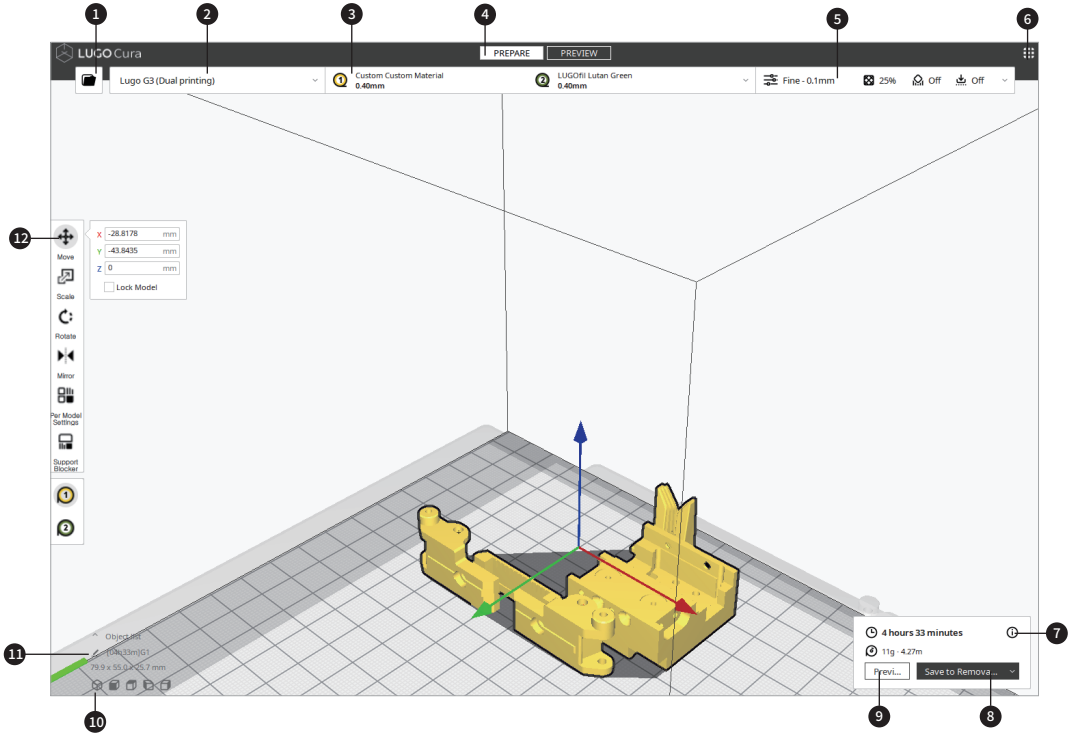
By default, LUGO Cura consists of English.
However, it can be set up in a variety of languages to ensure better accessibility. Choose [Preferences] -> [Configure] to change language settings. On the General tab, in the Interfaces section, select the desired language from the list.



Note: The newly set language will be set after the LUGO Cura reboot.

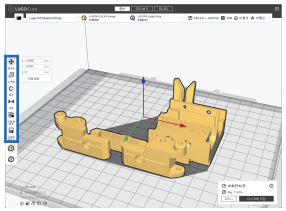
Overview

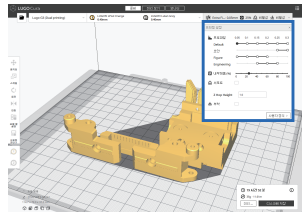
Interface overview of LUGO Cura



1. **Open file**: Opens a 3D file.
2. **Printer selection panel**: Displays the selected printer.
3. **Configuration panel**: Contains the material and print core setup.
4. **Stages**: The prepare, preview. Each stage is arranged to efficiently go through each 3D printing step.
5. **Print settings panel**: This is the set value input panel for successful output.
6. **LUGOLABS Support page**: You can purchase printer maintenance materials, output tips, and materials by visiting the LUGOLABS online shop.
7. **information panel**: There is detailed information about the slicing result.
8. **Save**: Save to removable disk or save to file.
9. **Preview**: Proceeds to the next stage, the preview stage.
10. **Camera position tool**: Easily positions the camera to default show default viewing angles.
11. **Model information**: Contains the 3D model name and dimensions of all printable models on the build plate.
12. **Move Tool:** move, rotate, mirror, scale, and more options.

Overview

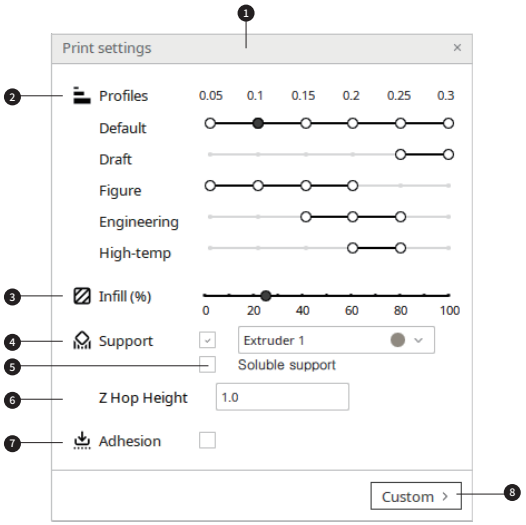




Move Tool

명칭	설명
Move	You can position your model in any way you like using the move tool, which allows the model to be moved along its X, Y, and Z axes.
Scale	You can scale your model in any way you like using the scale tool, which allows the model to be scaled along its X, Y, and Z axes. ☑ **Snap Scailng** : This option will scale the the model by 10% at a time, when dragging the scale handles. ☑ **Uniform Scailng** : This will ensure the model's proportions are consistent while scaling.
Rotate	You can rotate your model in any way you like using the rotate tool, which allows the model to be rotated along its X, Y, and Z axes. – Reset all rotations of the selected models. – This will lay the model flat on the closest flat surface of the model. – Select this button to make the model active. Click on any face of your 3D model to automatically align this face to the build plate.
Mirror	You can mirror your model along its X, Y, and Z axes.
Per model Settings	You can change the print settings differently for each model loaded on the build plate. This feature is available only when it is set to custom mode in the Print Settings panel. – 모델을 보통으로 인쇄하려면 이 옵션을 선택해야 합니다. – 선택한 모델을 서포터로 프린팅 – 선택한 모델의 오버랩 설정 수정. 채우기/자르기 중 선택 가능 – 선택한 모델 일부분의 오버랩을 차단
Support Blocker	You can block automatic support generation.

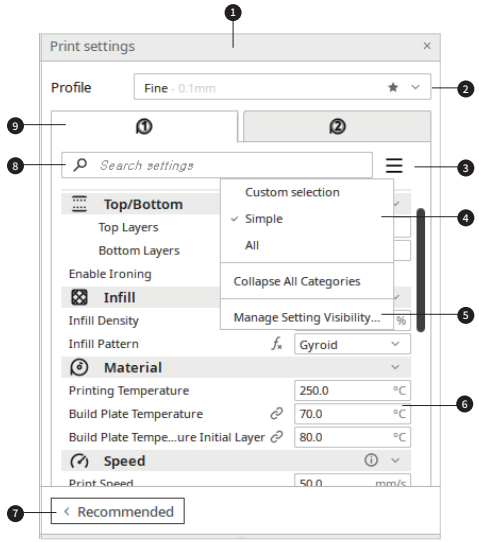
Recommended mode



1. This panel shows the current printing strategy at a glance, click it to view the entire panel
2. The available profiles for the current configuration are visible.
3. The infill slider can be used to easily adjust the overall model strength.
4. Enable or disable automatically generated support structures, with either of the available extruders, to get reliable and successful prints when necessary.
5. Check when using soluble support.
6. When dual printing, the head is lifted slightly when passing over the part.
7. This is enabled when using materials that have significant shrinkage.
8. This will open the custom mode to manually finetune printing profiles.

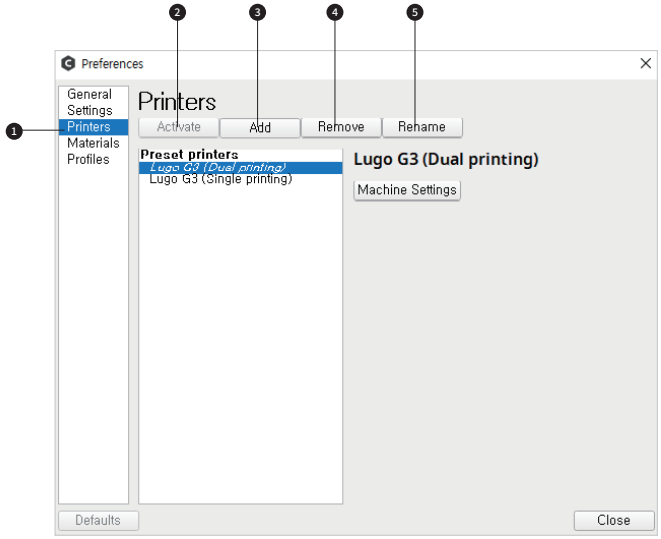
Overview

Custom mode



1. This panel shows the current printing strategy at a glance, click it to view the entire panel.
2. This contains a list of available profiles suitable for the current configuration.
3. This menu has the ability to quickly view or hide a preset of print settings.
4. Simple, Custom selection views, increasing number of available settings with every step.
5. Opens the preferences panel.
6. These parameters define the setting values, each is indicated with the right unit type.
7. Toggles between custom and recommended mode.
8. Start typing to immediately filter settings by name.
9. This panel is used for nozzle selection.

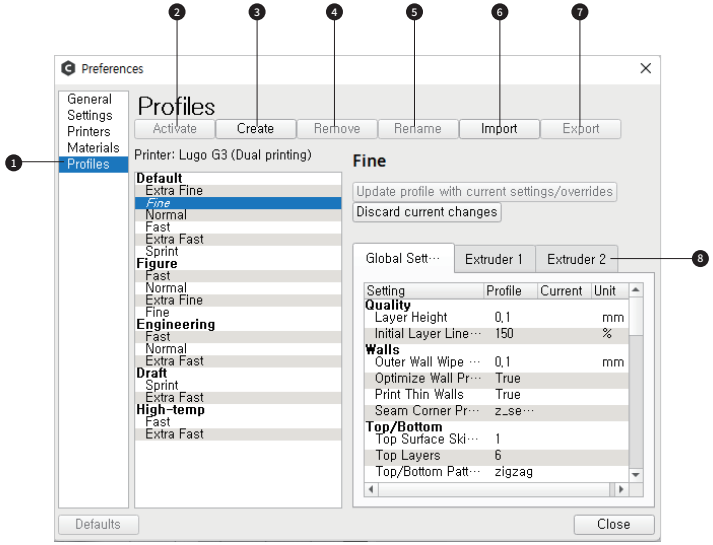
Manage Printers



1. Printer preferences.
2. Activate the selected printer.
3. Add a new printer to Ultimaker Cura.
4. Remove the selected printer.
5. This option is greyed out for networked printers. Change the name of networked printers through Cura Connect.

Overview

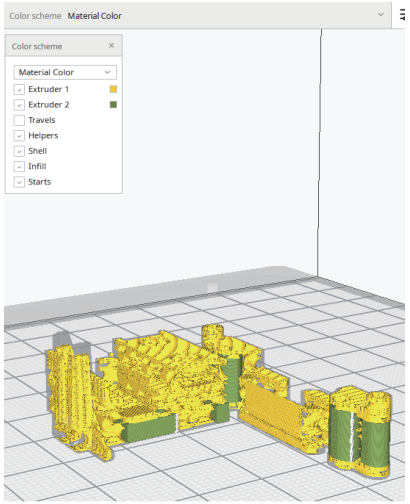
Manage printing profiles



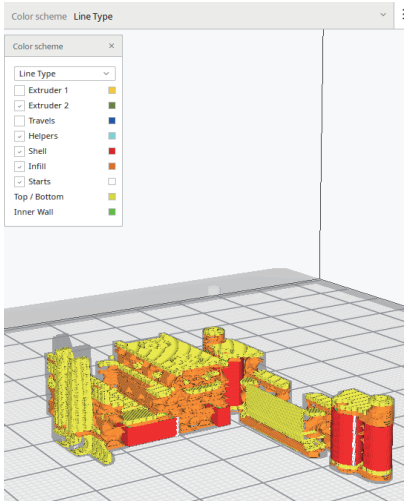
1. The profile tab in the LUGO Cura preferences panel.
2. Select a profile and click Activate to change to the selected profile.
3. The selected profile will be duplicated.
4. The selected profile will be removed from Ultimaker Cura.
5. Give the selected profile a new name.
6. Allows importing previously exported print profiles into Ultimaker Cura.
7. The selected profile will be exported and can be shared or saved as a backup.
8. Print profile information is stored in the global settings tab that counts for both extruders. The separate extruder tabs contain extruder unique parameters

Preview

After the slicing is complete, you can press the preview button to view the output simulation screen. Here, in the "Color Configuration Table", the colors are categorized according to the material and line type.
It is easier to see the simulation before output.



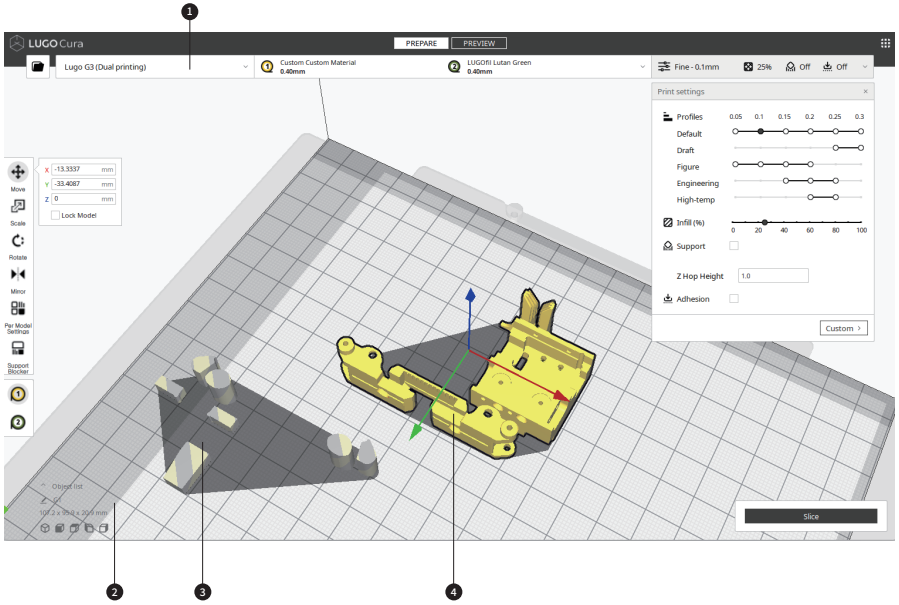
< Material color >



< Line type >

Start printing

Workflow explained

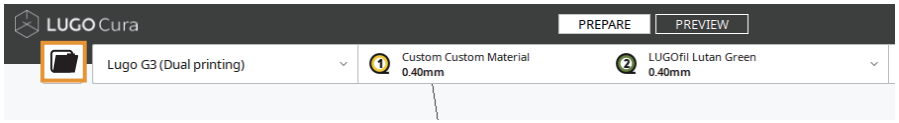


1. **Stage menu**: Contains the printer selection, configuration and print settings panel.
2. **Non-printable area**: This area is indicated by a gray shadow on the build plate.It is controlled by the slicing setting value.
3. **Non-printable model**: Due to parts being in non-printable areas.
4. **Printable model**: Displayed in the color corresponding to the material selected to print with

How to open a file

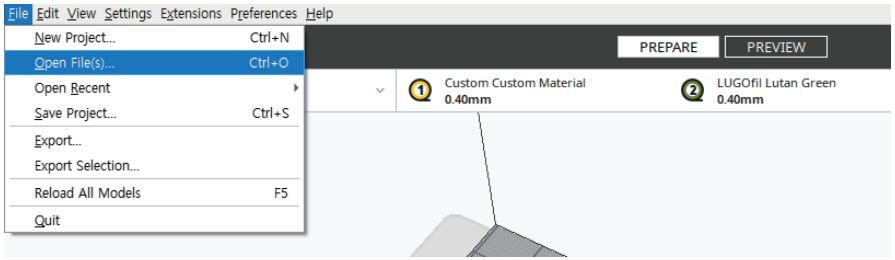
- **Open file icon**

  Click the 'Open File' button in the upper left corner of the Stage Menu.



- **File menu**

  Open a file by going to 'File > Open File(s)...' to locate the file on your computer.

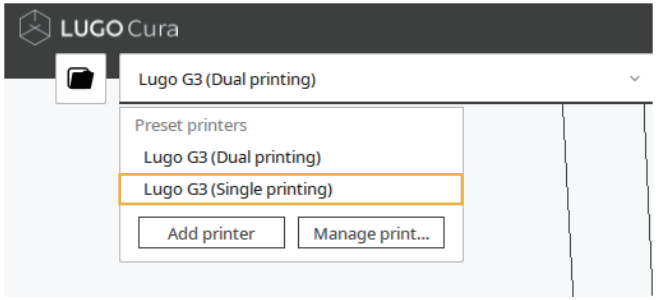


Start printing

Single Print

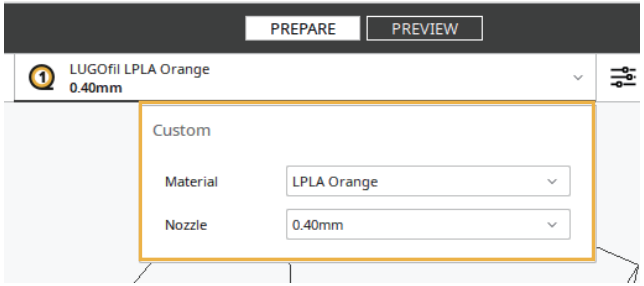
❶ Select a printer

The printer selection panel is located on the second button of the Stage menu.
Select the printer that corresponds to single printing.



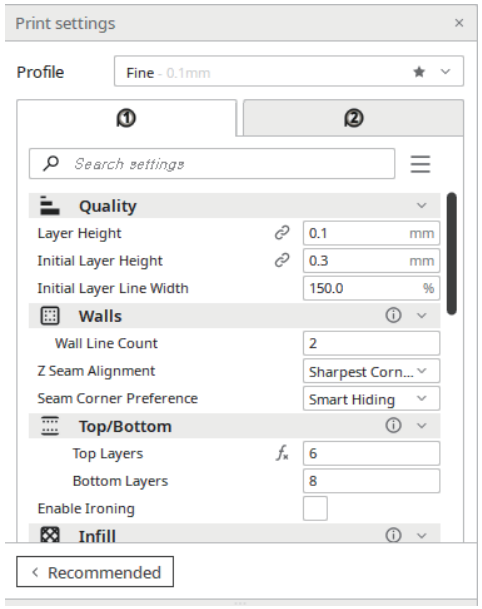
❷ Nozzle and material settings

The third button on the Stage menu has the Configuration panel.
Click to display the current nozzle holes and material settings in the panel.



❸ Print setting

The rightmost button of the Stage menu contains a print settings panel that contains all the settings that define how to print. Opens in Recommended mode by default. This mode is ideal for fast printing with an optimized print profile.

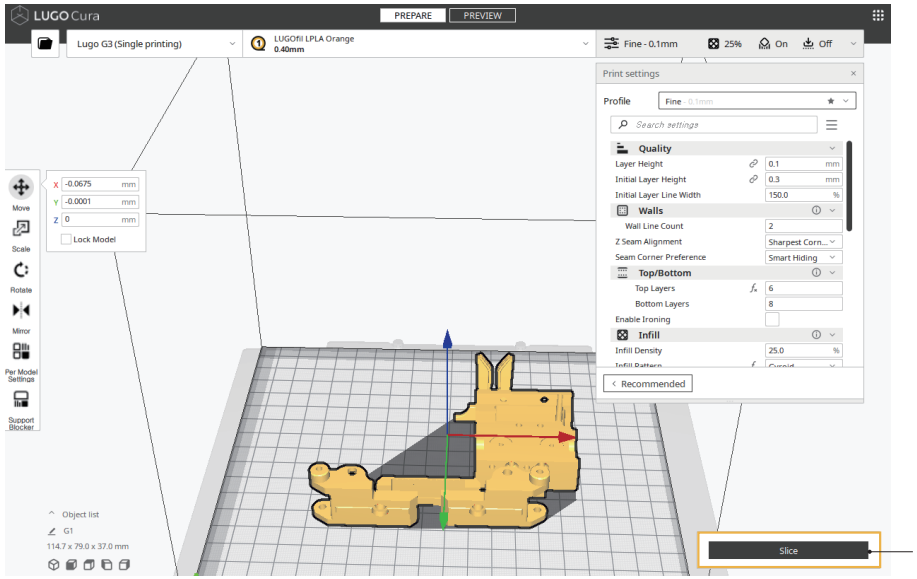


Start printing

Single Print

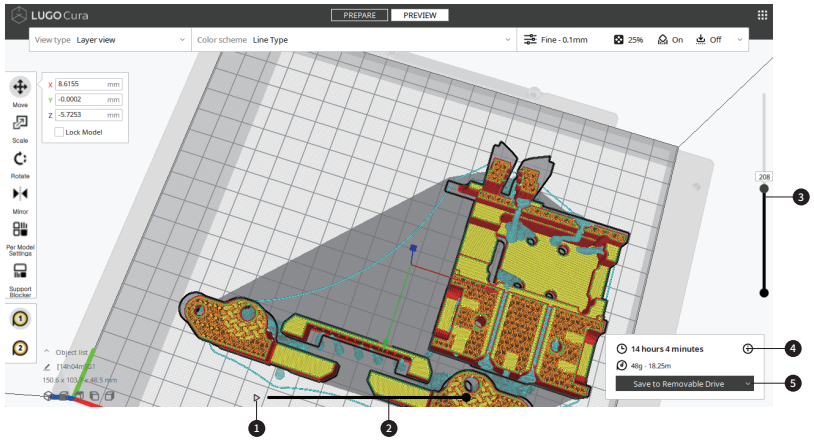
❹ Slice

Press the 'Slice' button in the lower right corner when printing setup is complete. When the process is complete, a 'Preview Button' will be displayed immediately, when clicked, to go to the Preview step. Previewing is a very important process for determining what the inside of a print looks like and how it will be printed. Use the Layer slider and the Simulation view to identify important parts of the 3D slice.



❺ Preview and save

When the slicing process is complete, click the Save to File button in the lower right corner. You can save files to a removable disk, such as an SD card that came with your computer or printer. Safely remove the SD card from your computer and insert it into the printer.



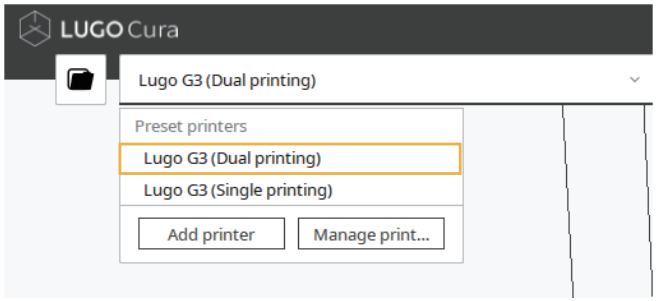
1. **Start simulation**: This starts the simulation of the currently active layer to provide visual feedback on the printing process.
2. **Path slider**: This indicates the current position of the simulation view. This handle can be dragged or moved with the ‘arrow keys’ or ‘shift + arrow keys’.
3. **The layer slider**: This can be dragged to drag a layer range up and down.
4. **Print information**: Hover the mouse over the icon to view detailed print time information per printed feature.
5. **Print over network**: This will send the print job directly to the printer over the network. Non-networked printers have options to save to removable disk or save to file instead.

Start printing

Dual Print

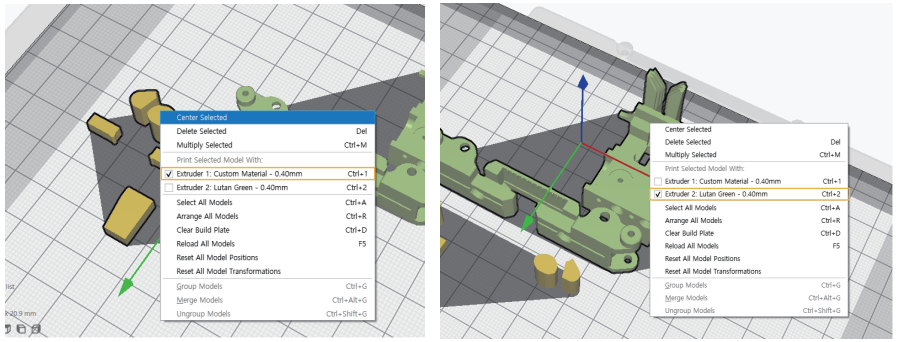
❶ Select a printer

The printer selection panel is located on the second button of the Stage menu.
Select the printer that corresponds to dual printing.



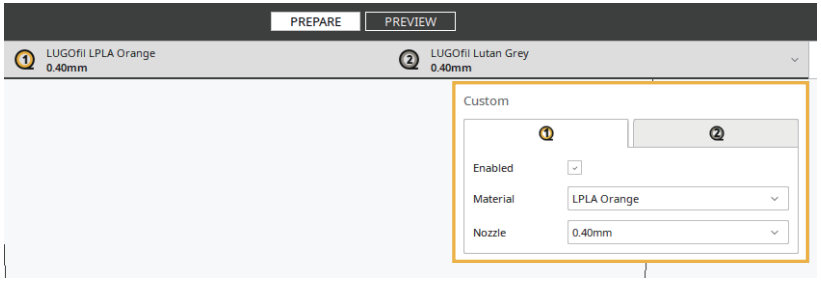
❷ To set the output nozzle by object.

Right-click each model to set the output nozzle.



❸ Nozzle and material settings

The third button in the Stage menu contains the Configuration panel.
Click to display the current nozzle aperture and material settings on the panel.
Set each nozzle.

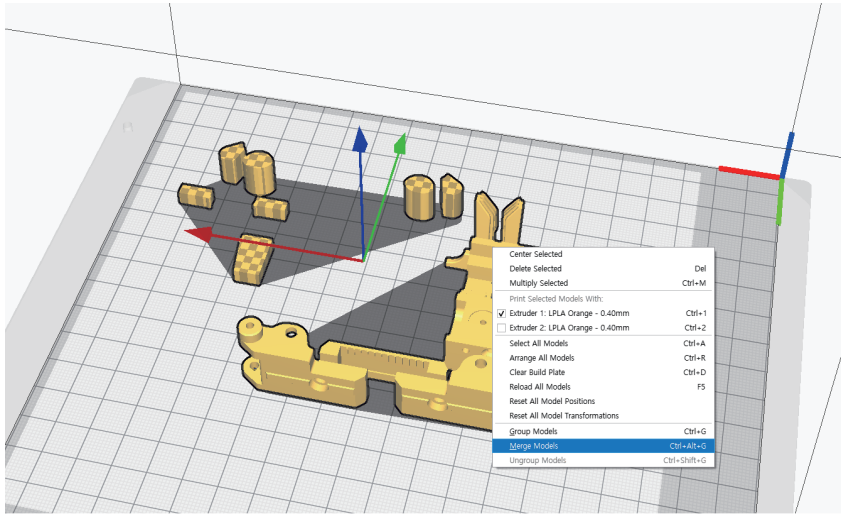


Start printing

Dual Print

❹ Modeling Support (Merge)

You have to go through the process of combining the separated models into one.
This feature automatically aligns the model, which is useful for dual print



<Procedure>

① Use the model 'Shift + Left Click' to select more than one model from the build plate.

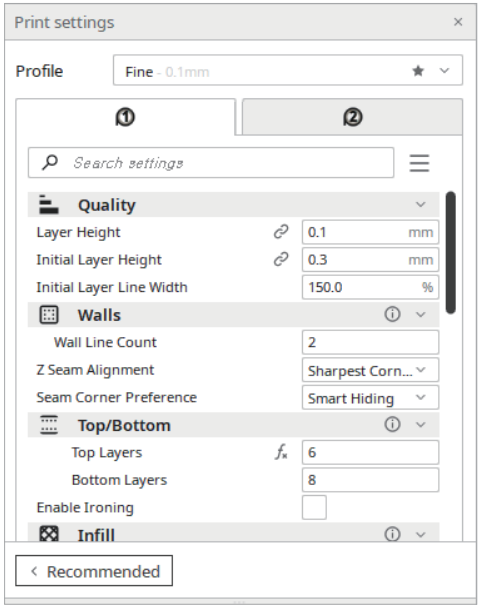
② The selected model is indicated by a black outline.

③ 'Right-click' one of the selected models and select 'Joint Models'.
The shortcut for this command is 'ctrl / cmd + alt + G'.

④ The merged model has a gray boundary box.
The merged model is aligned to the origin of the two models.

❺ Print setting

The rightmost button of the Stage menu contains a print settings panel that contains all the settings that define how to print. Opens in Recommended mode by default. This mode is ideal for fast printing with an optimized print profile.

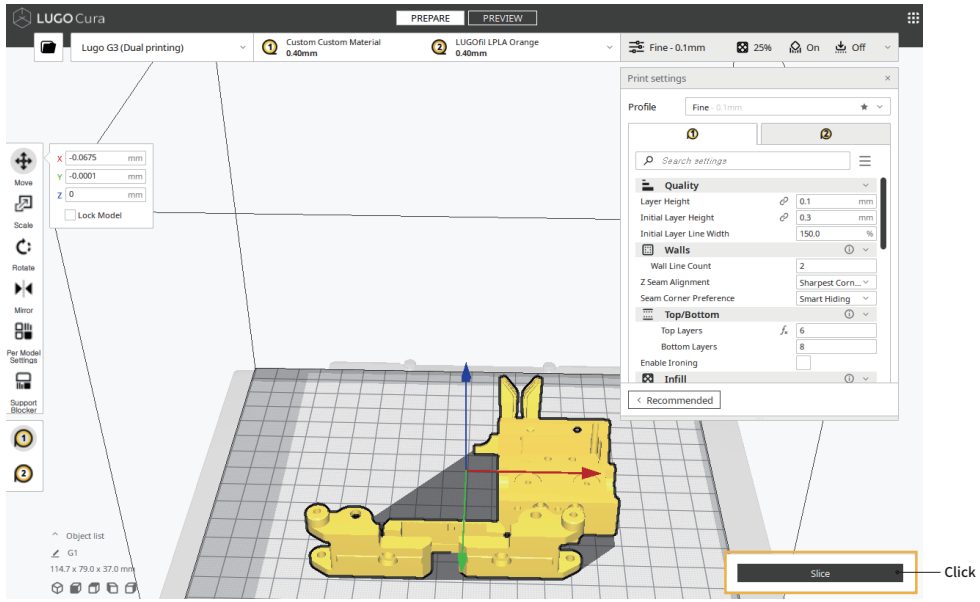


Start printing

Dual Print

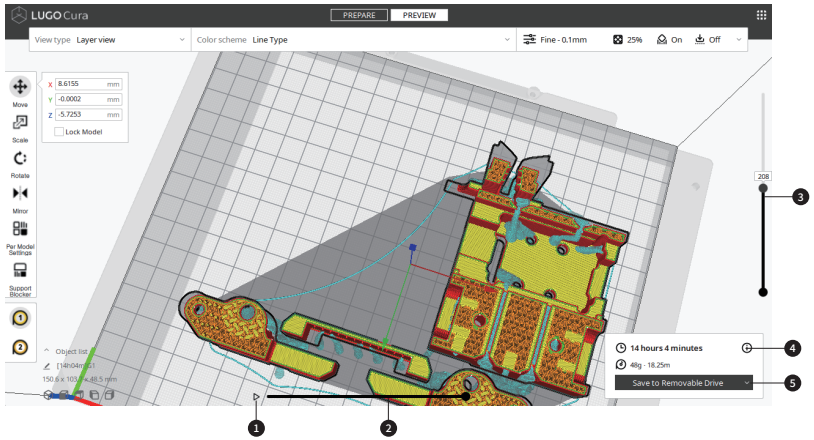
❻ Slice

Press the 'Slice' button in the lower right corner when printing setup is complete. When the process is complete, a 'Preview Button' will be displayed immediately, when clicked, to go to the Preview step. Previewing is a very important process for determining what the inside of a print looks like and how it will be printed. Use the Layer slider and the Simulation view to identify important parts of the 3D slice.



❼ Preview and save

When the slicing process is complete, click the Save to File button in the lower right corner. You can save files to a removable disk, such as an SD card that came with your computer or printer. Safely remove the SD card from your computer and insert it into the printer.

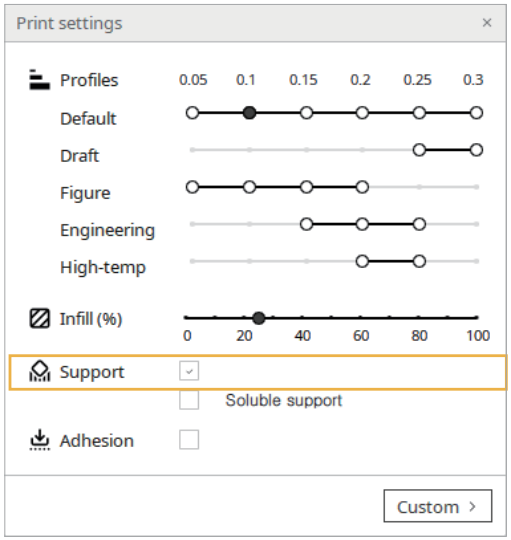


1. **Start simulation**: This starts the simulation of the currently active layer to provide visual feedback on the printing process.
2. **Path slider**: This indicates the current position of the simulation view. This handle can be dragged or moved with the ‘arrow keys’ or ‘shift + arrow keys’.
3. **The layer slider**: This can be dragged to drag a layer range up and down.
4. **Print information**: Hover the mouse over the icon to view detailed print time information per printed feature.
5. **Print over network**: This will send the print job directly to the printer over the network. Non-networked printers have options to save to removable disk or save to file instead.

다양한 서포트 생성 방법

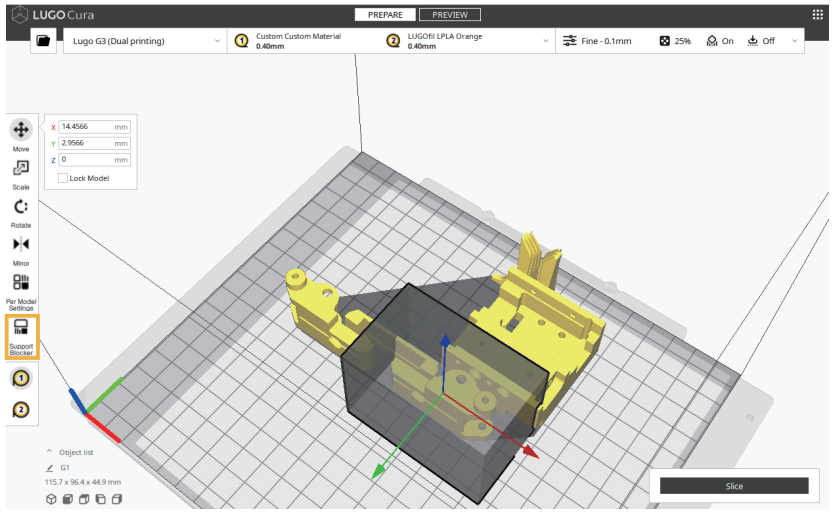
❶ Creating Automatic Supports

By clicking on the print settings panel on the right side of the screen and selecting "Support" in the Recommended mode, you can enable or disable automatic support settings.



❷ How to block support

You can block support using the 'Support Blocker' tool in the Move Tool menu.



<Procedure>

① Select the model.

② Select the move tool to activate it.

③ Click the location of the model to generate a 1cm by 1cm by 1cm support blocker.

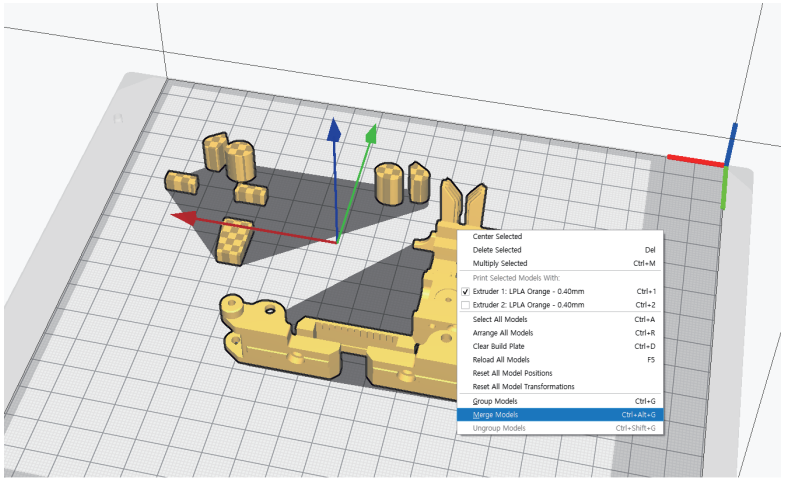
④ Move, scale and rotate the support blocker to position it to your likings.

⑤ Support generation will be blocked where the support blocker completely overlaps the 3D model.

다양한 서포트 생성 방법

❸ Merge models for Dual Extrusion

Support modeling enables automatic alignment of models.
This feature is useful when working with dual material/color models.



<Procedure>

① Select the model you want to support and click the ‘Per model setting’ in the Move Tool. ' Press the 'Print as support' button to set it.

② The selected model is marked with a check pattern.

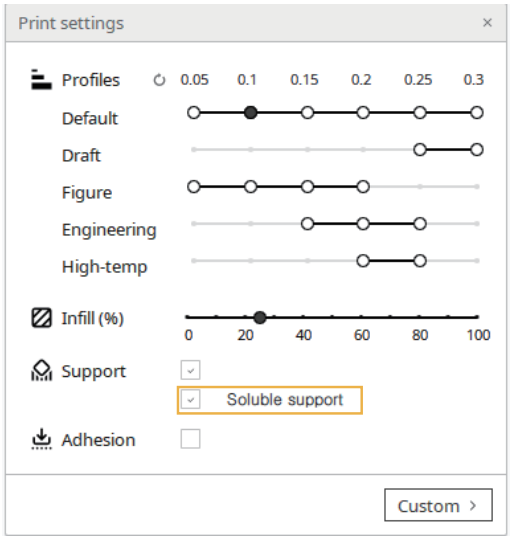
③ Then, select two or more models on the build plate, by Shift + Left Clicking every model.

④ Right click one of the selected models, and select Merge Models.
Or hit Ctrl /Cmd + Alt + G.

⑤ The merged model has a gray boundary box.
The merged model is aligned to the origin of the two models.

❹ Soluble support

Click the print settings panel on the right side of the screen, check and select 'Support' in recommended mode, and select 'Soluble support' to use it.



Features

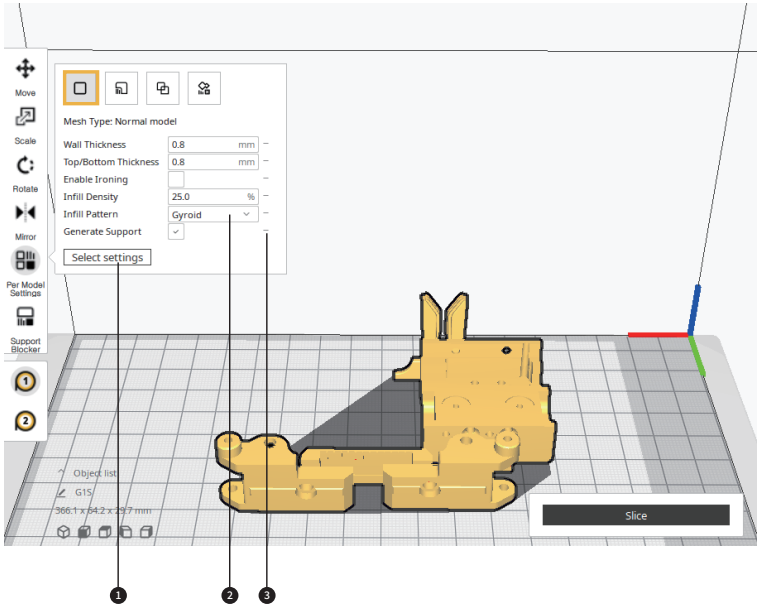
Per Model Setting

You can change the print settings for each model loaded on the build plate using the per-model settings.
This feature is available only when the Print Settings panel is set to Custom mode.

This feature provides the ability to have multiple models using different printing strategies on the same build plate. This makes it easy to print model placement for a variety of purposes.

❶ Normal model

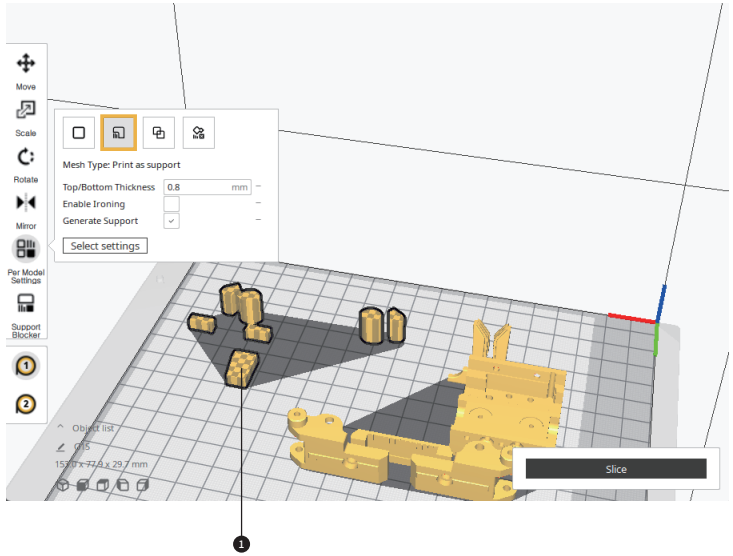
Ensure to keep this option selected to print the model 'as normal'



❶ Brings up the list of available settings to add to the printing strategy.
❷ Change these to the desired values.
❸ Click the '-' icon to remove the per model settings from the list.

❷ Print as support

Select this mesh type to convert a 3D model to a support structure.



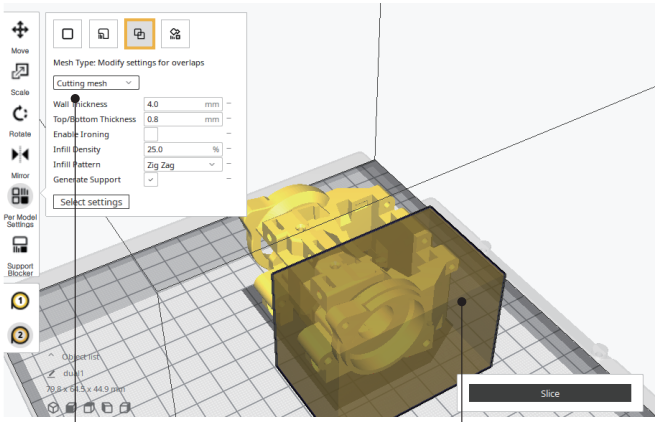
❶ The selected model is marked with a check pattern.

Features

Per Model Setting

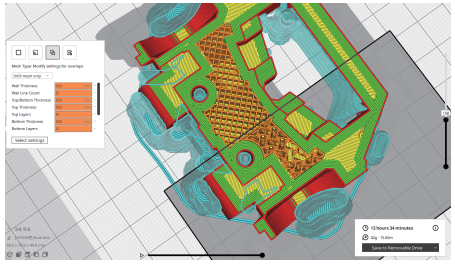
❸ Modify overlap settings overview

It is possible to set partial overlap settings with two options: "Mesh Type/Inner Wall" or "Mesh Type/Cutting Mesh." Load another file to overlap, set it as "Extruder 2," and then set the position to fill or cut the area, selecting the "Inner Wall" or "Cutting Mesh" option, respectively, and proceed to slice.



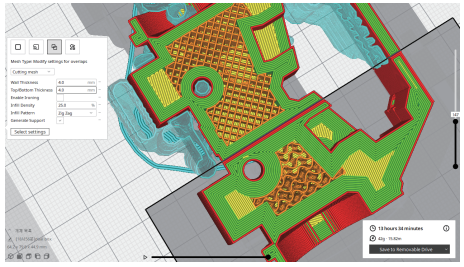
Select by Options

The translucent area is the overlapping area.



< Infill mesh only >

Only areas that are translucent through previews You can see that the infill settings are different.

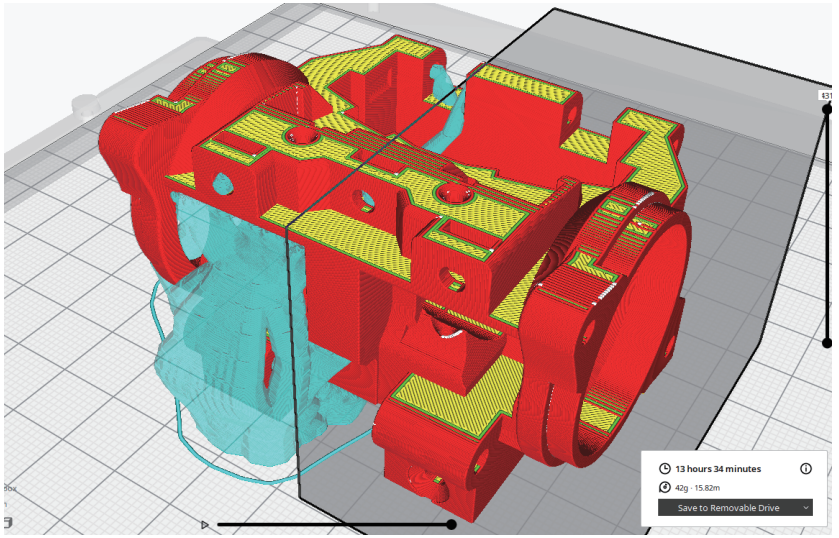


< Cutting mesh >

You can use the preview to ensure that only the areas marked translucent are not affected by the printing options settings.

❹ Block support overview

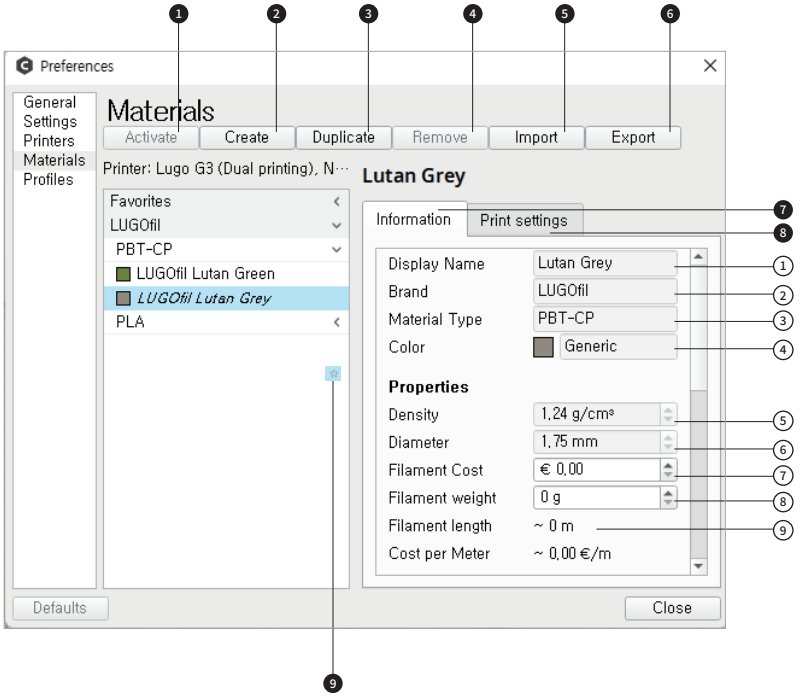
Models blocking support are colored translucent grey, in order to keep your other 3D models visible on the build plate.



Features

Manage Materials

Profiles for materials are preloaded so you can easily start printing immediately without having to perform a test print. 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 ‘file > Configure’
(Windows - 'CTRL + K' / Mac OS - 'CMD + K')



1. To use the selected material, click Activate.
2. Create new materials.
3. This duplicates the selected material.
4. Removes the selected material.
5. Allows to import downloaded and shared material profiles.
6. Export material profiles.
7. Displays information of the selected material. Default material profiles, like tough PLA, cannot be altered.
8. Shows the print settings used while printing this material.
9. Click the icon to add the material to your material favorites.

① Choose an easily recognizable name that represents your custom material.

② Enter the brand of the filament manufacturer.

③ Enter the material name. If the material is not registered in Lugo Cura, you must set it to ‘Custom’ to use the recommended setting.

④ Select the square color icon to change it. Loaded models with this material will be displayed in this color on the build plate.

⑤ The weight of the filament per square centimeter.

⑥ Lugofil only supports 1,75mm filament.

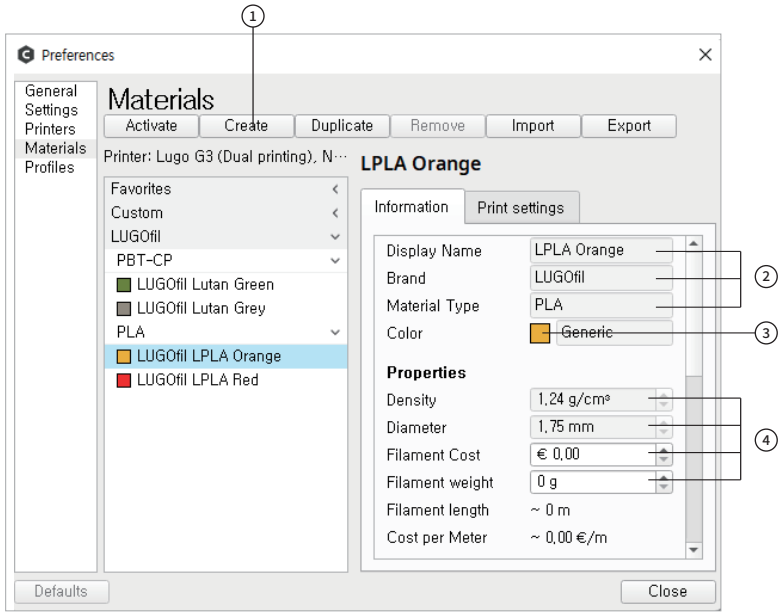
⑦ Enter the cost of your filament spool.

⑧ Enter the weight of a single spool of filament.

⑨ These are automatically calculated by Ultimaker Cura based on the previous settings.

Features

Create Material Profile



<Procedure>

① Select the Create button to add a new material.

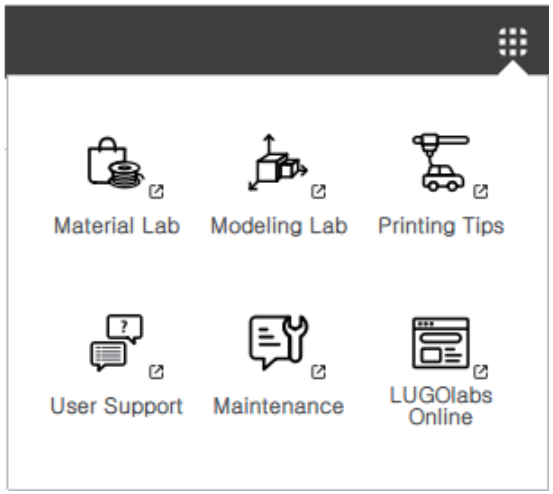
② When you enter a display name, trademark, and material type, the corresponding category name is created.

③ Click the color scheme to specify the desired color.

④ If you enter Density, Diameter, Cost, and Weight, the filament length and cost per meter are automatically calculated.

LUGOLABS Support Page

Click the icon in the upper right corner of the interface to access information and maintenance or material purchases that are useful for output by visiting the LUGOLABS online shop.



❶ **Material Lab:** Provide information about materials.

❷ **Modeling Lab:** Provide modelling materials for output.

❸ **Printing Tips:** Useful Materials for Output.

❹ **User Support:** Provides download materials for output.

❺ **Maintenance:** Provides information about maintenance.

❻ **LUGOLabs Online:** Lugolabs Oline shop.