

UltiMaker 3D Printer Guide

☐☐ Important Reminders (Read First)

- **Please name your file using your Slack name.** Students are responsible for the machine they are using, improper setup can cause damage or fire, so if we can't identify you, your print will be paused or canceled.
 - Always **wear gloves** when touching the build platform or removing prints, please.
 - **Double-check** that the build platform is correctly reinstalled — all four alignment holes must **match precisely**. Misalignment can **damage the printer**.
 - Not confident or unsure? **Please ask a Technician before printing.**
-

3D Printing Workflow Manual

From Modeling to USB Printing with UltiMaker Cura. Applies to: UltiMaker Cura users and USB-compatible 3D printers (e.g., UltiMaker series).

☐☐ Workflow Overview

1. **CAD Export** – Save your model as `.STL` or `.OBJ`
 2. **Install Cura** – Download and install **Ultimaker Cura**
 3. **Slice in Cura** – Import, adjust slicing settings
 4. **Export G-code** – Save file **named after your Slack ID**
 5. **Prepare USB** – Format to **FAT32**, save `.gcode`
 6. **Pre-Print Self Check** – Inspect setup and hardware
 7. **Monitor First Layers** – Watch first layers carefully
 8. **Post-Print** – Cooldown, safe removal, clean up
-

☐☐ Step-by-Step Instructions

1. CAD Software Recommendation

Use CAD tools **optimized for 3D printing**:

- **Fusion 360** – Great for parametric design and functional parts. We have [Student Fusion Access](https://www.autodesk.com/uk/education/education-software/overview#FSN), you could access via <https://www.autodesk.com/uk/education/education-software/overview#FSN>
- **Rhino** – Ideal for complex, freeform modeling with great STL control

- **Tinkercad** – Browser-based and beginner-friendly

☐ Avoid **Maya** and **Blender**:

- They're built for animation/VFX, not solid modeling, so objects might not be closed properly, or may not be watertight.
- Blender lacks parametric accuracy and often produces **non-manifold** meshes.

“ Tip: Choose tools built for precision modeling to reduce printing errors.

2. CAD Export

- Export as `.STL`, or `.OBJ`
- Recommended format: `.STL`
- Ensure **correct unit scale** (preferably millimeters)

3. Install Ultimaker Cura

- Download from: ultimaker.com/software/ultimaker-cura
- Add your printer model (e.g., **Ultimaker S7**)
- Set **Material 1** to: `White PLA AA0.8` (for a fast prototyping) or `White PLA AA0.4` (for the final projects), check the available machines' nozzle sizes before exporting your print.
- Need different materials? Please speak with a Technician

4. Slice in Cura

- Import your model into Cura.
- Default settings may not work for most models. Adjust them by clicking on the settings bar in the top-right corner.



- Choose **Fast** for PLA prototyping and **Normal** for finer printing.
- Consider whether your model needs support. If it does, enable it using the support toggle.



- Click on **Show Custom** to access and adjust advanced settings such as layer height, supports, and speed.

- **If using PLA**, set the printing temperature to **215 °C** and speed to **80-110 mm/s**.
 - **If using PVA**, set the printing temperature to **225 °C** and speed to **35-50 mm/s**.
 - For more guidance, use the **“Settings Guide” plugin** ([Main page](#) > [Top right corner](#) > [Marketplace](#) > [Search](#)) or speak to a technician.
 - Preview the slice before exporting ([PREVIEW](#) page - top center).
 - **Check in X-ray view** ([PREVIEW](#) page - top center > [View type](#) - top left corner > [X-Ray view](#)):
 - If everything is blue, it's likely safe to print.
 - If there are red areas, return to your 3D modeling software to fix any issues.
-

5. Export as **yourslackname.ufp**

- Save your file as [.3mf](#) for further editing, [.ufp](#) for printing
 - **Filename format:** [yourSlackSame_modelMame.gcode](#)
 - Unnamed or unclear files will be **paused or cancelled**
-

6. Prepare USB

- Save [.ufp](#) file to the **root directory**
 - Bring it to the Lab
-

7. Pre-Print Self-Check

- Confirm **bed leveling** — all **4 holes must be in their positions**
- Ensure **the nozzles are clean** and not clogged. If there is something wrong, please let the technician know and choose another machine to use
- Load the filament; verify material matches
- Carefully install the build platform — all **4 holes must be in their positions**

“ ⚠ If unsure, **ask a Technician** – improper setup can cause damage or fire, and the user would be responsible for that.

8. Monitor the First 5 Layers

- Insert the USB stick into the printer, choose your model, and start to print.
- Watch the print until a few layers are complete, users are responsible for not damaging the machine
- Check:
 - Adhesion to bed
 - No warping or dragging

- If issues occur, **pause immediately** and contact a Technician
-

9. Post-Print Safety & Cleanup

- Wait for the bed to **cool down** for few minutes
 - Always **wear gloves** if you need to touch the platform
 - Gently remove the model and/or build plate
 - **Return tools and platform** exactly as found
-

“ Blender for 3D Printing would be harder but not banned, here are some **Key Considerations**

1. **Use Manifold Geometry.** Ensure the model is watertight—no holes, loose edges, or non-manifold geometry.
2. **Set Correct Units.** Set the unit system to Metric and choose millimeters to match most slicers and printers. (Scene Properties > Units)
3. **Apply Scale & Transforms.** Use Ctrl+A to apply Scale, Rotation, and Location before exporting.
4. **Model with Thickness.** Avoid zero-thickness surfaces. All parts of your model must have physical volume.
5. **Normals Facing Outward** Recalculate normals so all faces point outward. (Edit Mode > Select All > Shift+N)
6. **Check Wall Thickness.** Thin parts may not print well. Minimum recommended wall thickness is usually 0.8mm to 1.2mm, depending on the printer.
7. **Avoid Too Much Detail.** Overly fine details may not appear well on small prints.
8. **Use 3D Print Toolbox Add-on.** Enable this built-in add-on: (Edit > Preferences > Add-ons (or Get Extensions) > Search: “3D Print Toolbox”) It helps detect non-manifold edges, intersecting faces, and other print issues. If you couldn't get it installed, please update your blender to a newer version (4.2+).<https://extensions.blender.org/add-ons/print3d-toolbox/>

9. **Export as STL**, and name it use your slack name Use 3D Print Add-ons

> Export > STL

The screenshot shows a dark-themed software interface. At the top right, there is an 'Options' dropdown menu. Below it, a vertical sidebar contains buttons for 'Item', 'Tool', 'View', and '3D Print'. The main area displays a menu with the following items: 'Analyze', 'Clean Up', 'Edit', 'Export', and 'Options'. The 'Export' item is expanded, showing a file name field containing '//' and a 'Format' dropdown menu set to 'STL'. Below the 'Export' menu is a large grey button with an upward arrow icon and the text 'Export'. At the bottom of the screen, a dark notification bar displays the text 'Export selected objects using sett'.

10. **Check “Selection Only”** if you only want to export the active object.
11. **Test in Slicer Software** Import the STL into Cura to preview printability and generate G-code

Revision #9

Created 1 May 2025 11:23:26 by Yuwei Chen

Updated 21 May 2025 12:38:19 by Yuwei Chen