

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
- **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`
- Need different materials? Please speak with a Technician

4. Slice in Cura

- Import your model into Cura
- Adjust settings (layer height, infill, supports, speed)
- Use **“Settings Guide” plugin** for help (`Main page` > `Top right corner` > `Market Place` > `Search`)
- Preview the slice before export

5. Export G-code

- Click Save to Disk button
- `Filename format`: `yourSlackSame_modelName.gcode`
- Unnamed or unclear files will be **paused or cancelled**

6. Prepare USB

- Save `.gcode` 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

Revision #4

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

Updated 1 May 2025 15:42:18 by Yuwei Chen