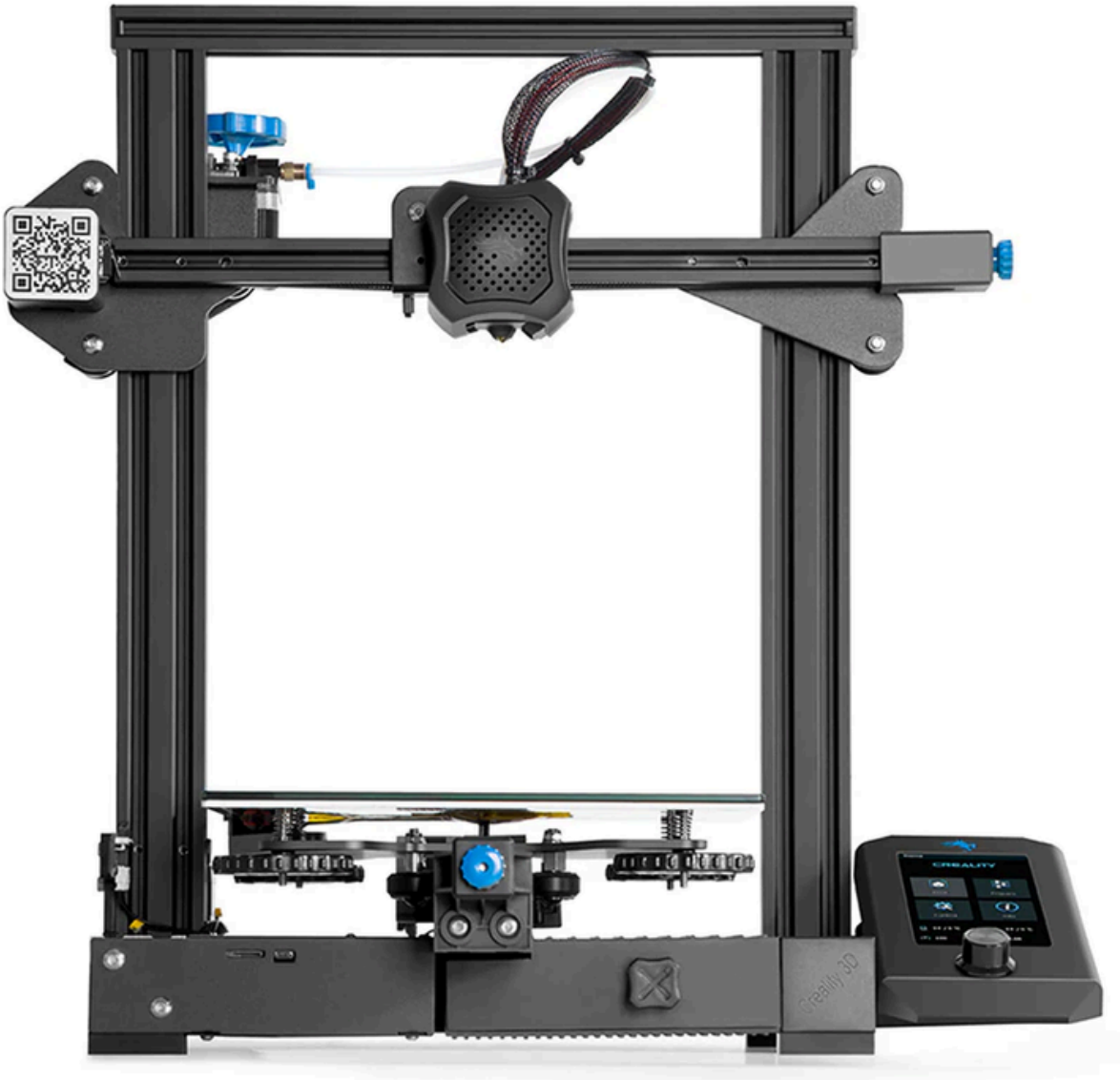
Ender 3 V2

Quick Guide



CREALITY

Contents:

1. Screen overview
2. Starting and managing a print
3. Bed leveling
4. Preheating
5. Loading filament
6. Common errors & Signs of failure

Home Screen

Print:
This allows you to select a file from the SD card to print. Once selected, the file is loaded and the print starts while displaying print details and progress.

Prepare:
This menu allows you a number of options to get your printer ready for printing, such as manually moving the axes, auto homing the printer (returning the Nozzle to the starting position for any print), preheating the bed and nozzle to preset values, putting the printer in cooldown mode and selecting the display language.

Control:
This allows manual selection of the fan speed, nozzle and bed temperatures and the changing of some of the default values for motion and temperatures.

Info:
Shows some information such as the current firmware version

Selecting a file

This menu displays a list of printable files in the root directory of the SD card. If there is no SD card inserted or no files are found the list will be empty.

Find the file with this format:

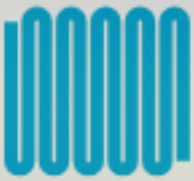
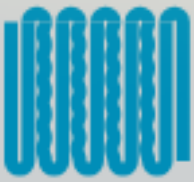
fnameLname_projectName

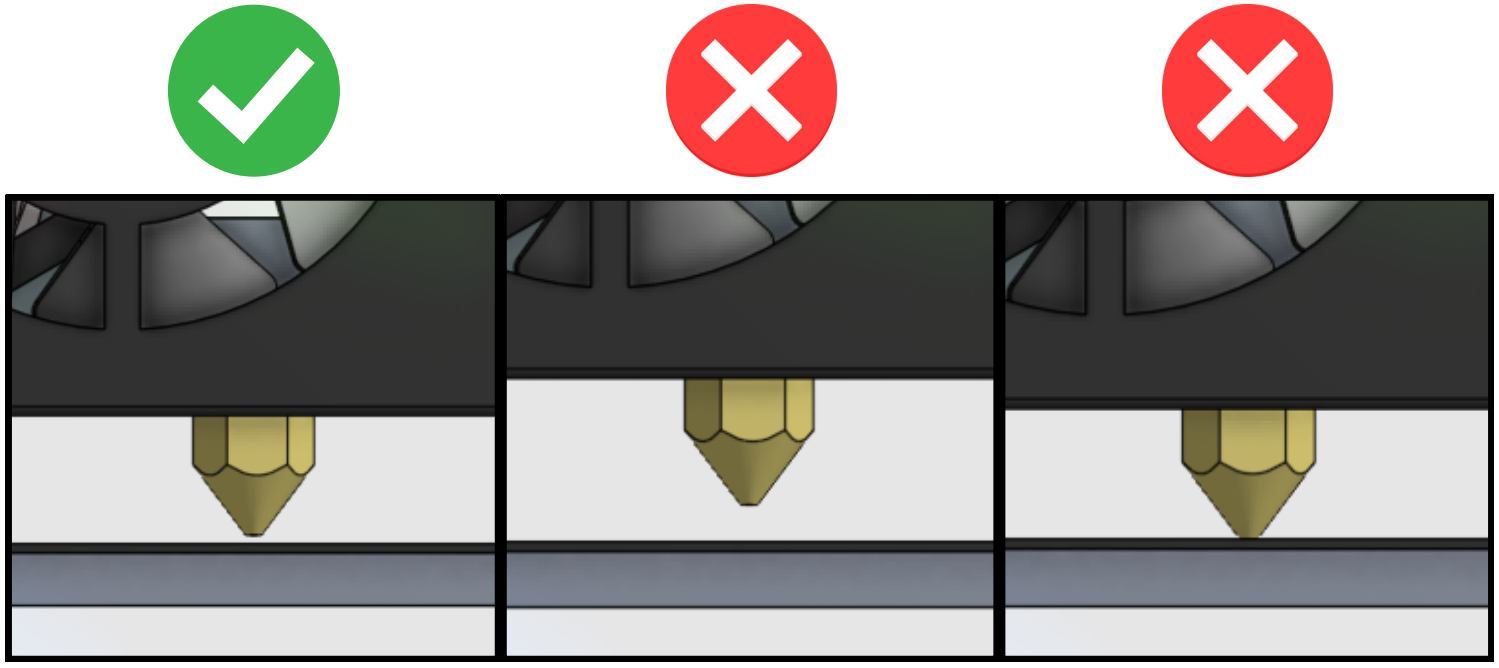
Printing Screen

Tune:
Tune allows you to manually modify the current print speed, nozzle and bed temperature and the fan speed. There is also a Z offset control allowing the position of the Z axis to be modified.
Pause:
Pauses the print allowing it to be resumed.
Stop:
After confirmation the print is stopped and cannot be resumed

3. Bed leveling

- Home the nozzle (In the **“Prepare”** screen)
- Move the tool head to one of the four corners of the build plate and check the height
- Refer to the graphic below:
  - **Too Close**: Turn the leveling wheel clockwise
  - **Too Far**: Turn the leveling wheel counter clockwise
- Repeat for all corners

		The nozzle is at the correct height above the platform.	✓
		The nozzle is too far from the platform. This may cause the extruded material to not stick to the build plate.	X
		The nozzle is too close to the build plate. This may damage the nozzle and the build plate.	X

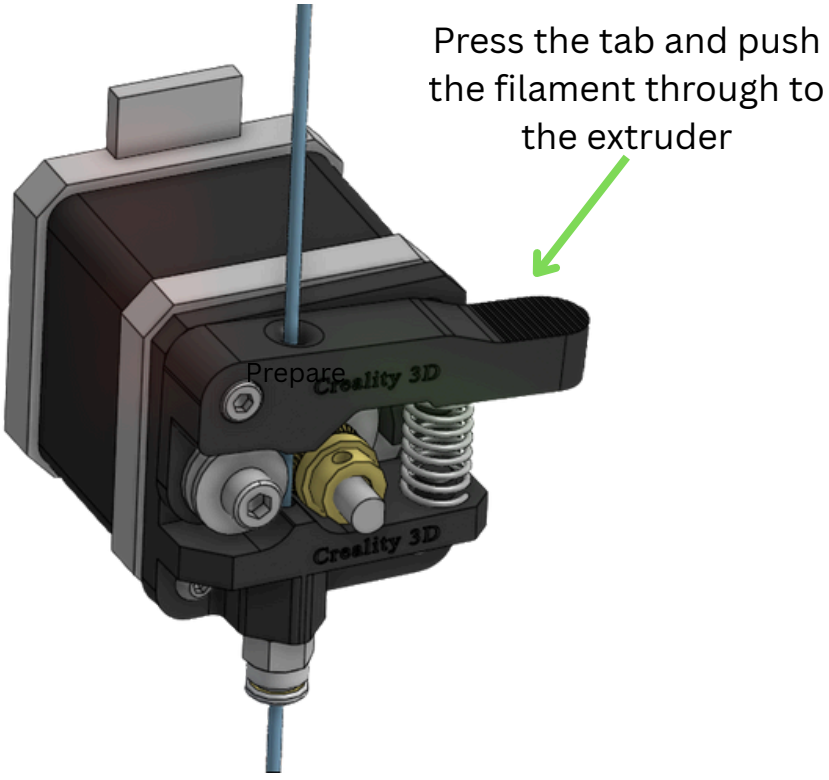


Correct **Too Far** **Too Close**

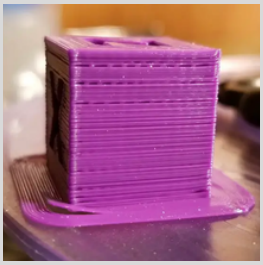
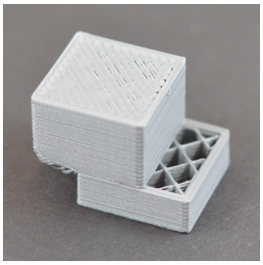
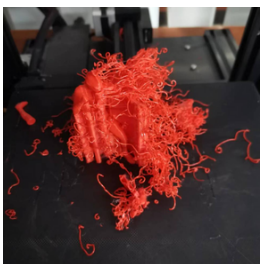
5. Loading Filament

- The nozzle **must** be hot to load or remove filament.
- Preheat the nozzle on the "**Prepare**" screen to the required temperature for the filament you are loading/removing.
  - If switching to a new material, make sure the nozzle is hot enough for the old material as well!
- Push filament through to the extruder and push until the old color has purged.

Material	(Nozzle) Extrusion Temperature
PLA	180 to 230 °C
ABS	210 to 250 °C
PETG	80 to 110 °C
Nylon	240 to 260 °C
TPU	210 to 230 °C



6. Common Errors & Signs of failure

Issue	Cause	Solution
Filament is not extruding (Often linked with the extruder motor clicking or ticking)	Nozzle is too close to the bed	Re-level the bed. Follow steps on **page 5**
	Nozzle is clogged	
	Filament is out, tangled or snapped	Reload filament and ensure that the quality is apt for printing.
Filament is under-extruding 	Nozzle temperature is too low for the loaded filament	Adjust the nozzle temperature in the slicer software and restart the print.
		OR Click the tune button and manually adjust the current printing temperature
	Nozzle has a minor clog	
Layer shifts 	Poorly tensioned belts & pulleys	Tension belts and pulleys **with assistance**
Warping (Item begins to peel off the bed during printing)	Moist filament	Load a new roll of filament
	Incorrect print temperatures	Check the material specifications listed on the spool
	Excessive sharp corners in your design	Add rounding to sharp outer corners of your design (Tool is often called **Fillet**)
Spaghetti Failure 	Bed is poorly leveled	Re-level the bed. Follow steps on **page 5**
	Bed temperature is incorrect	Adjust the bed temperature in the slicer software and restart the print.
		OR Click the tune button and manually adjust the current printing temperature
	Surface contact area is too low for your design	Try adding a **"Brim/Skirt"** in the slicer