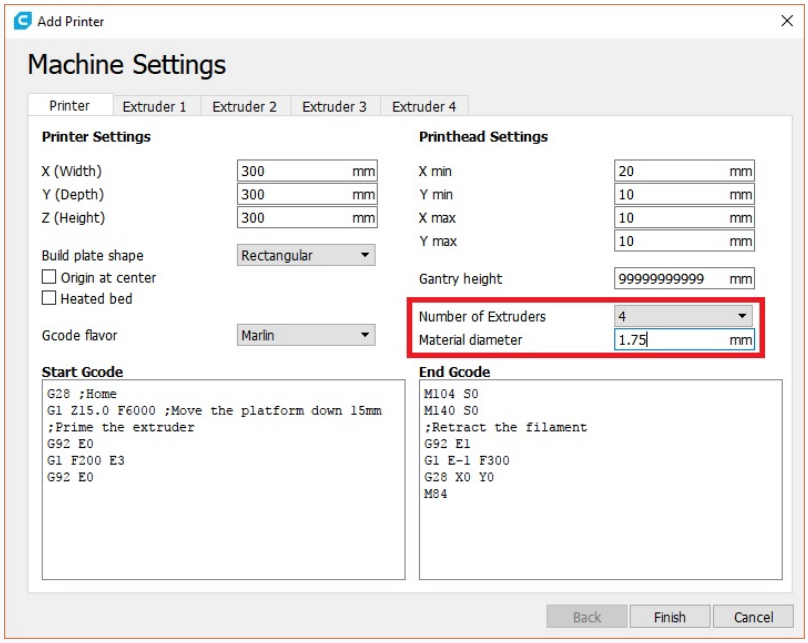
Adding Additional Extruders to Cura (v2.0 onwards)

Jonny Yeu

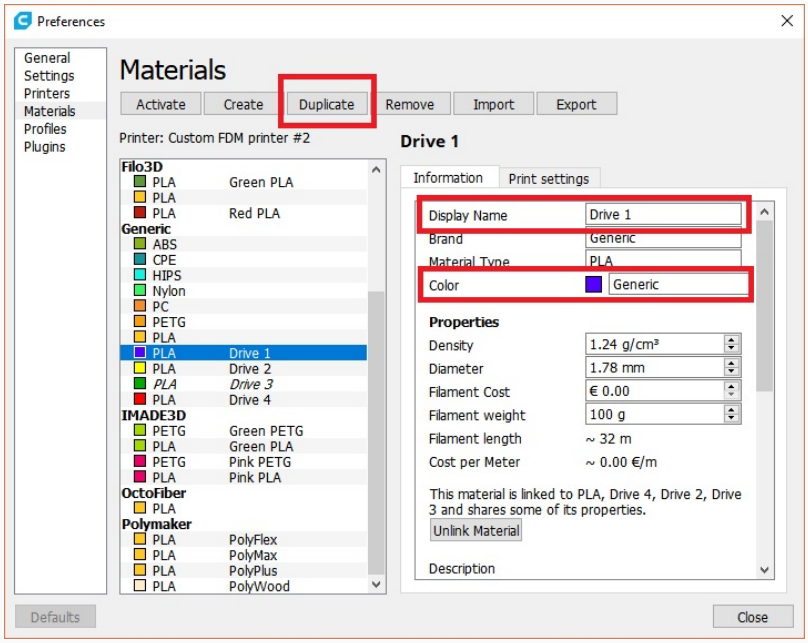
Updated 3 months ago

This article will help users add additional extruders to Cura (v2 & 3) in order to slice for Palette+. *To learn how to set up Cura V15.x.x for Palette+, please click here.*

1. If you already have a printer profile in Cura, open this profile's machine settings *(Settings > Printer > Manage Printers > Machine Settings*). Copy your Start and End GCode to a document for later use if you see any in your settings.
2. Add a new printer (*Settings > Printer > Add Printer*).
3. Add a Custom FDM printer even if your printer is on the list. Even if you have an existing printer profile, there is currently no way to copy a machine and add extruders in Cura.
4. Fill in your printer's information, including dimensions and settings. Set the number of extruders to 4 and the diameter to 1.75 mm.

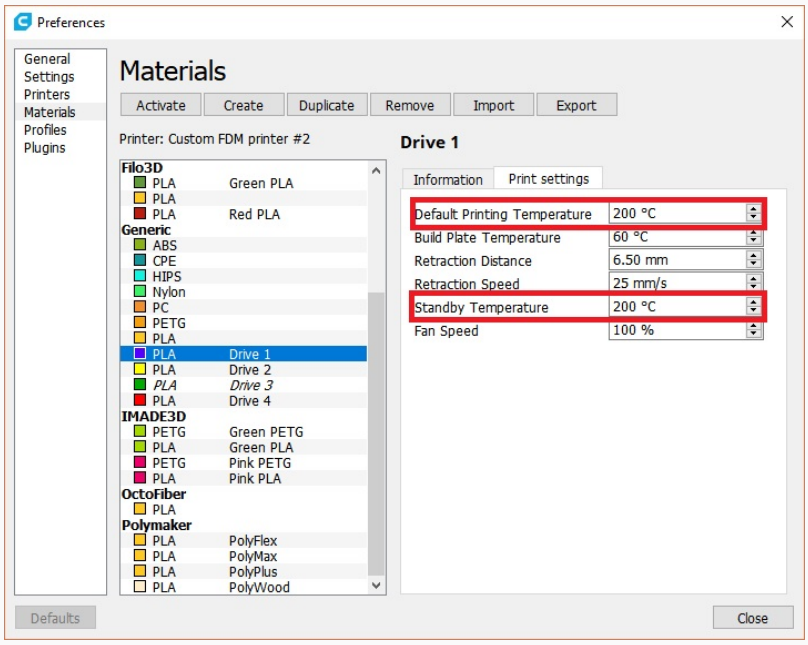


5. Under the *Extruder tab* for each extruder, enter your nozzle diameter and ensure that the other fields are empty or 0 (including start and end GCode). Click *Finish* to return to the home screen.
6. On Cura's main menu, go to *Settings* > *Extruder 1* > *Materials* > *Manage Materials.* On this screen, select the material you would like to print with (ex. PLA) and create a Duplicate. Under the *Information* tab, update the name and assign different colors. This will help distinguish between extruders and profiles.

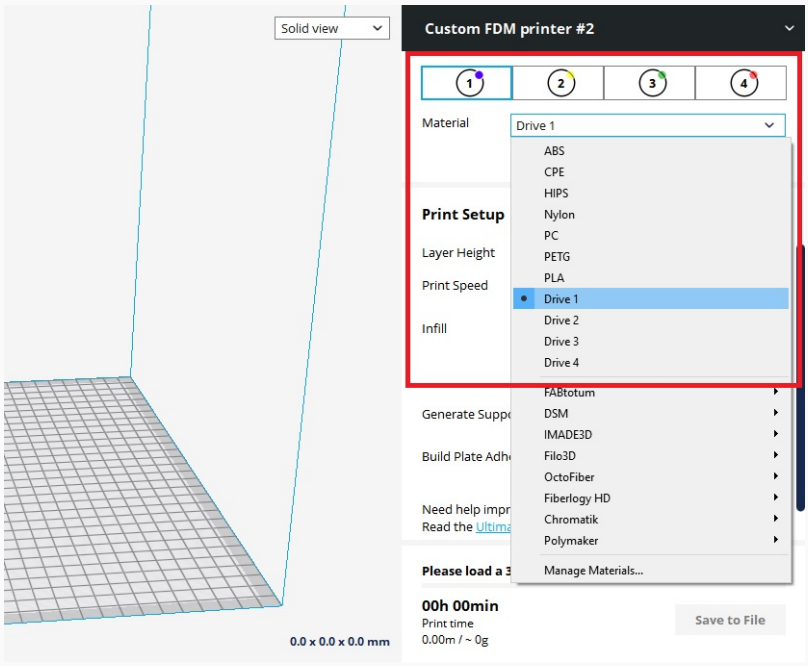


Under the *Print Settings* tab, set the *Standby Temperature* to be the same as the *Default Printing Temperature*. Repeat this steps for 4 different profiles (one for each extruder). Once

updated, click *Close* to return to the home screen.

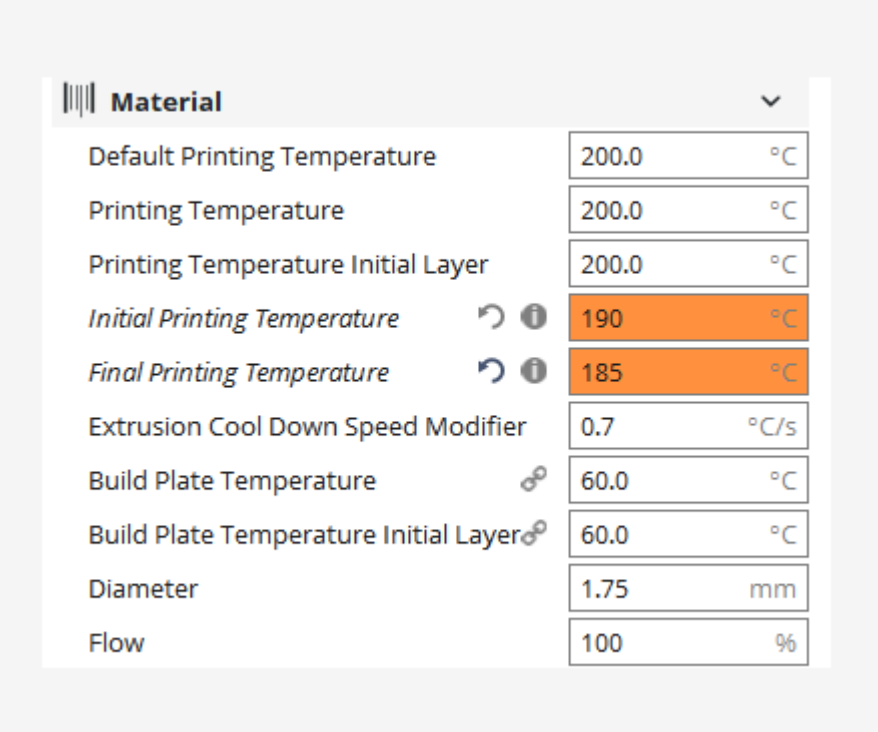


7. At the top of the right banner, you can assign each material profile to an extruder. Please ensure that all 4 profiles are the same for each extruder.

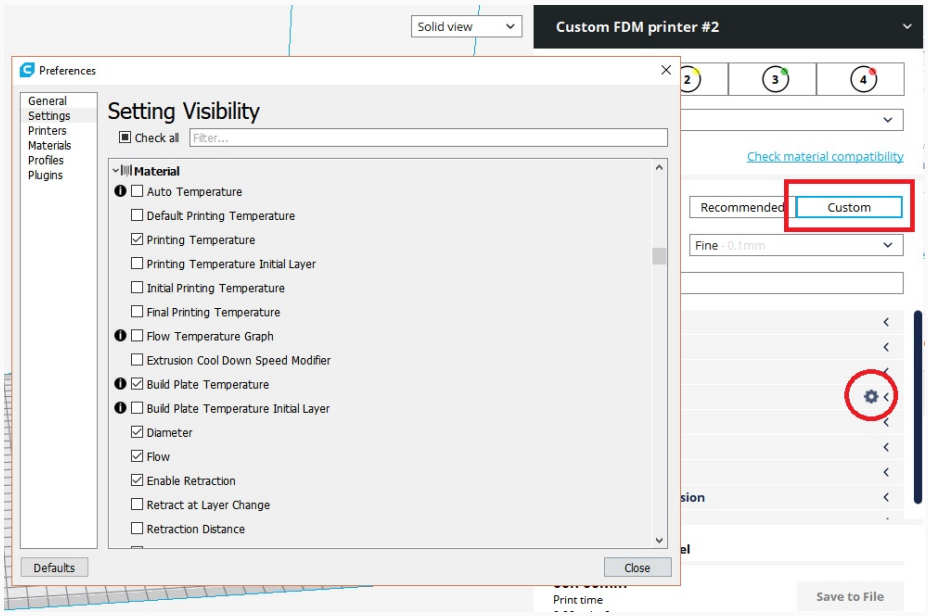


8. You'll also need to set the *Initial Printing Temperature* and *Final Printing Temperature* to be the same as your printing temperature. This is found on the right side of the home screen under *Printer Setup*, select *Custom,* which will expand a menu allowing you to change

the profile settings. Hover over the *Material* tab on the home screen and you'll see a gear symbol appear. Clicking this will further expand the menu of customizable settings.

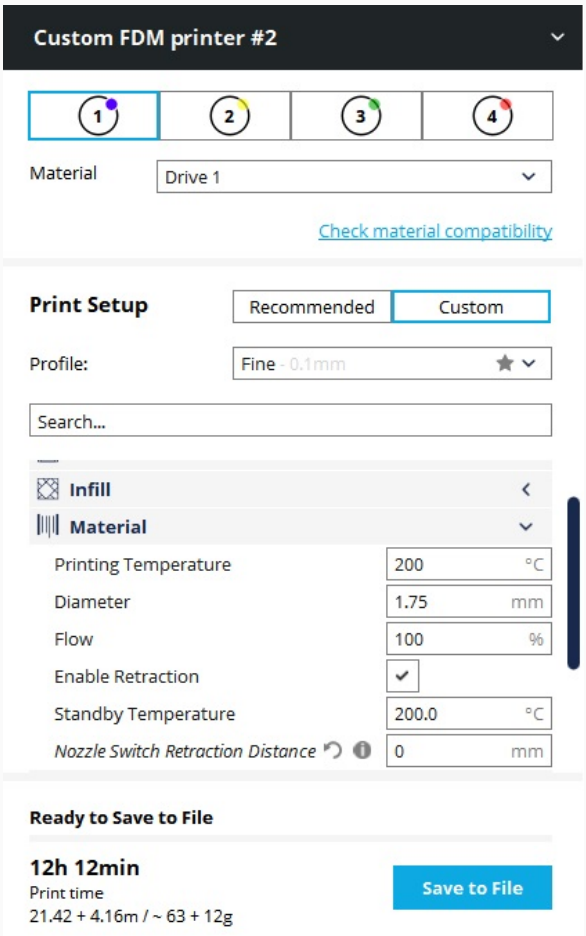


9. Ensure that *Standby Temperature* and *Nozzle Switch Retraction Distance* are also selected.

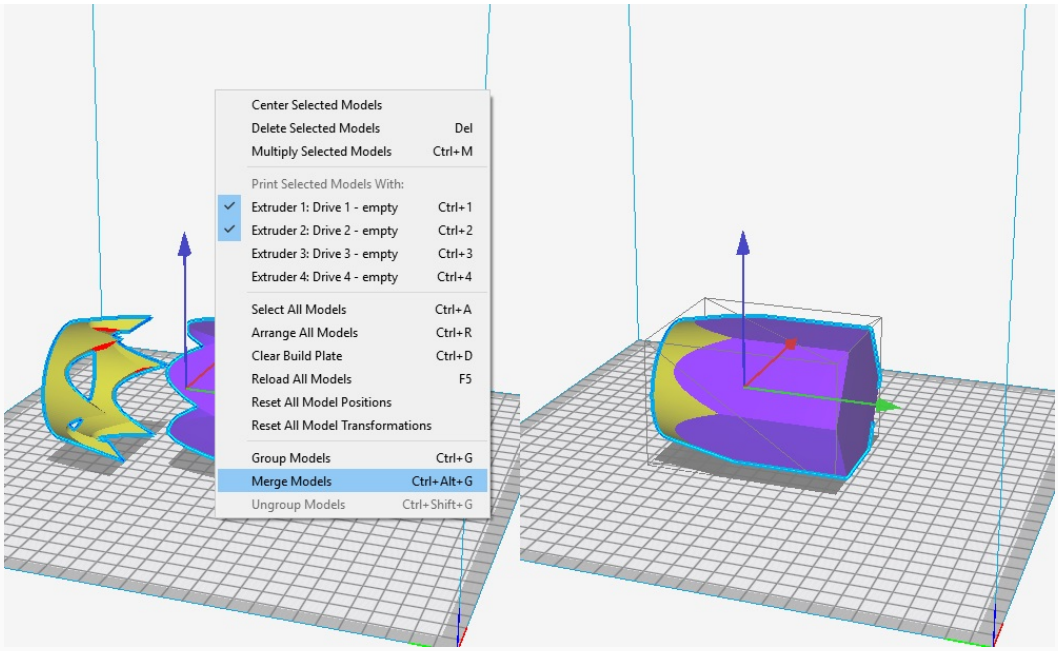


10. Back under the Material category, set the *Printing Temperature* and the *Standby Temperature* to be equal and set the *Nozzle Switch Retraction Distance* to 0. Repeat this for all 4 material

profiles.



11. Now you're set to slice! When you load multiple STL files, you can assign an extruder for each file which will color-coordinate with the material profile. Do this by right-clicking an STL and selecting an extruder for it. Once all files are loaded and assigned, select all parts by holding the "Shift" key and clicking each part of the model, then right-click and select "Merge Models". Please ensure that all pieces are loaded and assigned to different extruders before merging, as you are unable to assign extruders once the model is merged.



For older versions of Cura, follow this **video tutorial** to learn how slice and prepare models for Chroma.

If you run into any issues or questions, please send us a message at support@mosaicmanufacturing.com.

8

Was this article helpful?

✓ Yes × No

5 out of 5 found this helpful

Have more questions? Submit a request

Return to top

Related articles