

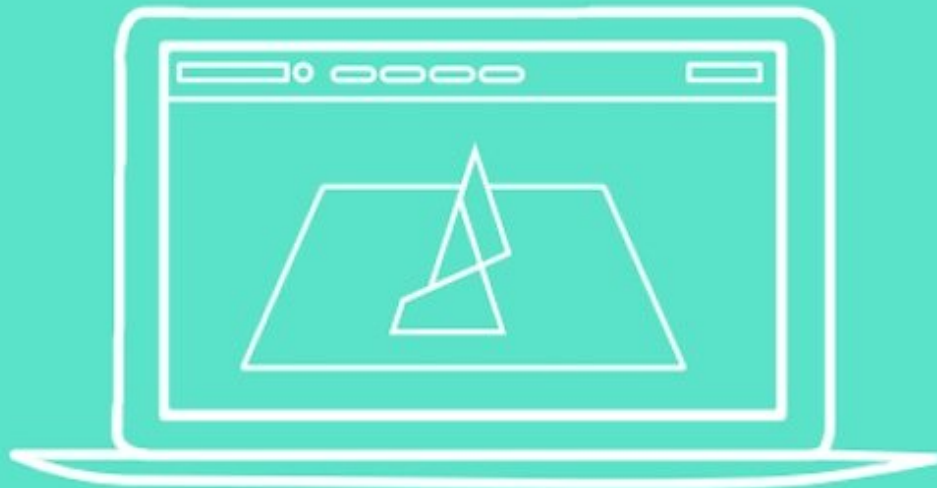


# Slic3r/PrusaSlicer and Chroma Setup

This guide will take you through setting up Slic3r/PrusaSlicer to work with Chroma and Palette.

Written By: Mosaic Support

## Chroma



## INTRODUCTION

This guide will take you through setting up Slic3r/PrusaSlicer to work with Chroma and Palette.

*Note: This guide assumes that PrusaSlicer/Slic3r has already been successfully configured for your printer.*

## Step 1 — Configure Settings Part 1



- *Application Preferences*: Ensure **Mode** is set to **Expert**
- *Printer Settings > General*
  - Set **Extruders** to 4 (settings from Extruder 1 will be copied to Extruders 2–4)
  - Change **Retraction when tool is disabled** settings for Extruders 1–4
  - Set **Length** to 0
  - Set **Extra length on restart** to 0
  - Dual-extruders: ensure that Extruder 2 settings match Extruder 1 settings (particularly **Extruder offset** being 0 and 0)

## Step 2 — Configure Settings Part 2



- *Printer Settings > Custom GCode*
  - **Dual-extruders:** ensure your Start GCode and end GCode don't include any nozzle priming or temperature settings for extruders other than T0
- *Print Settings > Skirt and Brim*
  - Set **Loops (minimum)** to 0
  - Set **Minimum extrusion length** to 0
- *Print Settings > Output Options*
  - Ensure **Complete individual objects** is unchecked
- Save this print configuration for future re-use

## Step 3 — Slice Multi-Tool G-Code



- **Plater**

- Select the print and printer configurations that were just saved
- Select the same filament configuration in all four extruder drop-downs
- Load in ONE of the STLs you would like to print, rather than all of them
- Double-click on the STL preview on the bed
- Click **Load part...** and select the rest of the STLs
- For each STL, select the extruder to be used for that STL
- Click **OK** when all STLs have been mapped to an extruder. That's it! Click **Export Gcode...** and you're ready to load the file into Chroma.

## Step 4 — Slic3r Prusa Edition



- For Slic3r Prusa Edition, there is a **G4 ;wait** configured in the END GCODE as a default. Chroma will provide an error as it is expecting a value after the G4 command (Chroma does helpfully tell you a recommend value). To avoid this, you'll need to either delete this line from your end sequence or modify it to take a numeric value.
- Please ensure **Print setting/Multiple Extruders/Wipe tower** is not checked off as Chroma will produce a wipe tower for you that is compatible.
- We would also suggest reviewing the default start sequence as there is a flow rate modifier that may cause issues with pings:
  - M221 S{if layer\_height==0.05}100{else}95{endif}
  - This command will get processed at slice-time and will result in the command 'M221 S95'. Chroma will not take this command into account and can cause issues with low pings.
- To fix this issue, please remove this line from the starting GCode and lower the extrusion multiplier to compensate.

## Step 5 — Troubleshooting - "Transition Tower is Too Large to Fit on the Bed"



- If you've sliced your project and the transition tower is unreasonably large, please check that the following settings are adjusted:
  - Please ensure that all the **min/max layer heights** are the same across all 4 extruders.
  - Check that your **Contact Z distance** is the same as your layer height.
  - Check that you've disabled Slic3r's **wipe tower**.
  - Lastly, please turn off **Variable Layer Heights**.

## Step 6 — P2PP



- For users who would like to use Slic3r's features without having to process the file through Chroma, there's a user-made plugin called P2PP that creates the necessary files directly in Slic3r. For more information, please see [here](#).

If you have any questions, please contact us at [support@mosaicmfg.com](mailto:support@mosaicmfg.com).

This document was generated on 2021-12-29 04:19:40 PM (MST).

