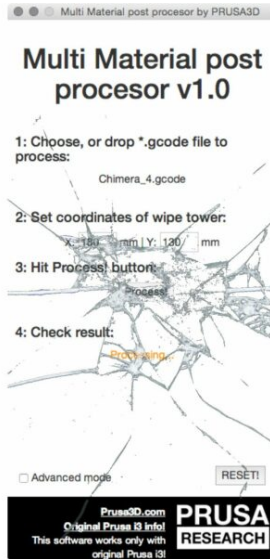


Table of Contents

Multi Material Post Processor Troubleshooting

- 3
- Step 1 - Wrong slicer 4
- Step 2 - Already processed file 5
- Step 3 - Wrong settings 6
- Step 4 - Unsupported layer height 7
- Step 5 - Unsupported first layer height 8
- Step 6 - Variable layer height 9
- Step 7 - Processing... 10
- Step 8 - Loading file... 11

Multi Material Post Processor Troubleshooting

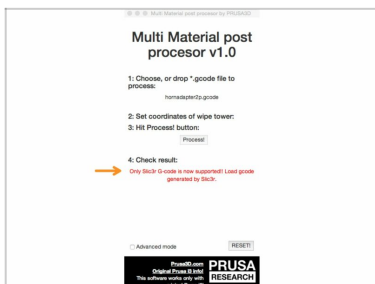


help.prusa3d.com/g108688


Scan the QR code to
display the latest
version of this
chapter.




STEP 1 Wrong slicer

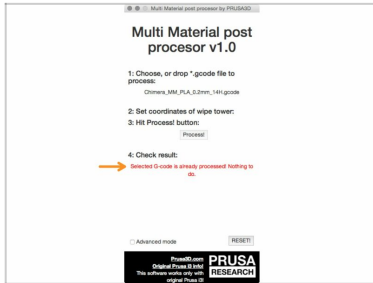


Only Slic3r G-code is now supported!! Load gcode generated by Slic3r.


 This error pops up if you'll use unsupported slicing software.


 Make sure that you used Slic3r to generate the G-code

STEP 2 Already processed file

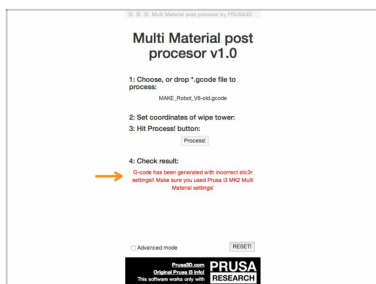


Selected G-code is already processed! Nothing to do.

 This error pops up if you loaded already processed g-code by the Multi Material post processor

 Post processor has nothing to do, everything is fine and the g-code you tried to load is already prepared. Just print it.

STEP 3 Wrong settings

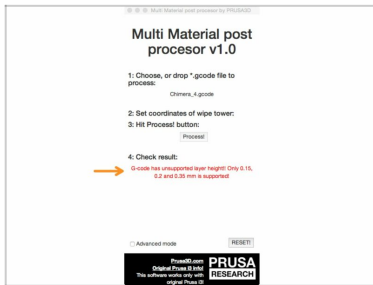


🟡 G-code has been generated with incorrect slic3r settings!! Make sure you used Prusa i3 MK2 Multi Material settings!

📘 This error pops up if you loaded g-code generated with incorrect settings. For example with profile for "Prusa i3 MK2"

📌 Regenerate g-code with "Prusa i3 MK2 Multi Material" profile and try it again.

STEP 4 Unsupported layer height

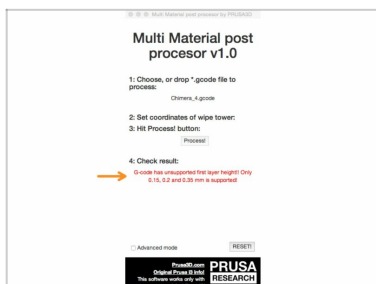


🟡 G-code has unsupported layer height!! Only 0.15, 0.2 and 0.35 mm is supported!

📘 This error pops up if you sliced the model with unsupported layer height

📌 At this time, only 0.15, 0.2 and 0.35 mm layer heights are supported, make sure you have supported layer height set in your slicer profile and try again.

STEP 5 Unsupported first layer height

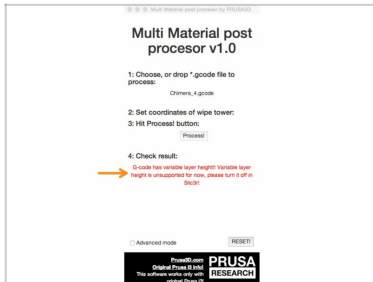


🟡 G-code has unsupported first layer height!! Only 0.15, 0.2 and 0.35 mm is supported!

📘 This error pops up if you sliced the model with unsupported first layer height

📌 At this time, only 0.15, 0.2 and 0.35 mm first layer heights are supported. Make sure you have supported first layer height set in your slicer profile and try again.

STEP 6 Variable layer height



🟡 G-code has variable layer height!! Variable layer height is unsupported for now, please turn it off in Slic3r!

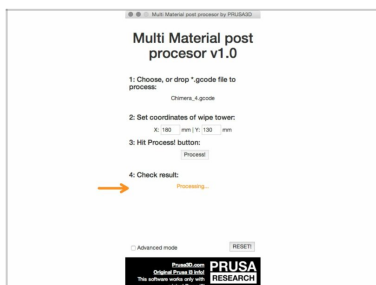
📘 This error pops up if you sliced the model with variable layer height.



At this time, variable layer height is not supported, make sure you used right slicer profile and variable layer height is turned off.

📘 Variable layer height can be turned off by unchecking the box in "Printer Settings" -> "General" -> "Enable variable layer height feature:"

STEP 7 Processing...



Processing...



This informs you that work is in progress.

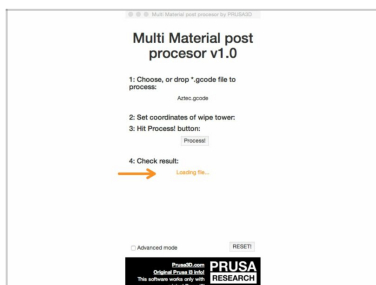


Post processing time depends on your hardware specs, g-code size and number of extruders.





If the post processing time exceeds 10 minutes and you don't have hardware from stone age, try restarting the post processor and repeat process.


STEP 8 Loading file...



Loading file...

 This status tells you that loading selected file is in progress.

 Loading time depends on your hardware specs and g-code size.

 If the post processing time exceeds 10 minutes and you don't have the g-code on super slow drive, try restarting the post processor and repeat process.
