

Restarting a stopped print tutorial using .Gcode

If your print failed before it was completed, you might be able to save it!

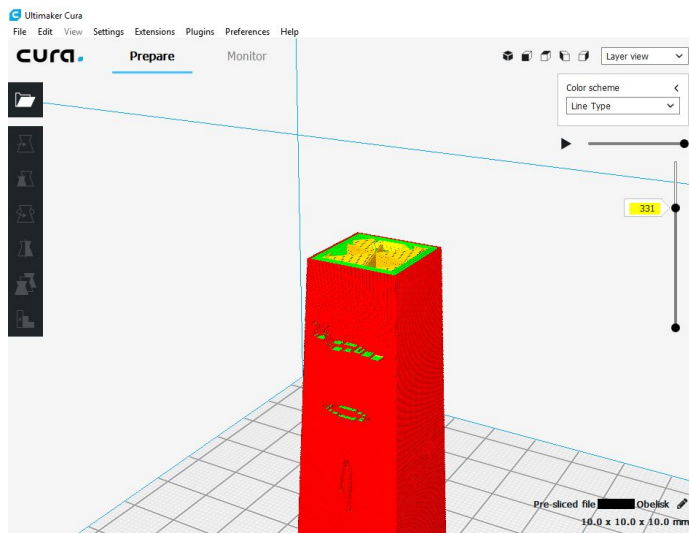
This tutorial will work if your printer jammed, ran out of filament, or stopped because of a power interruption.

Prerequisites:

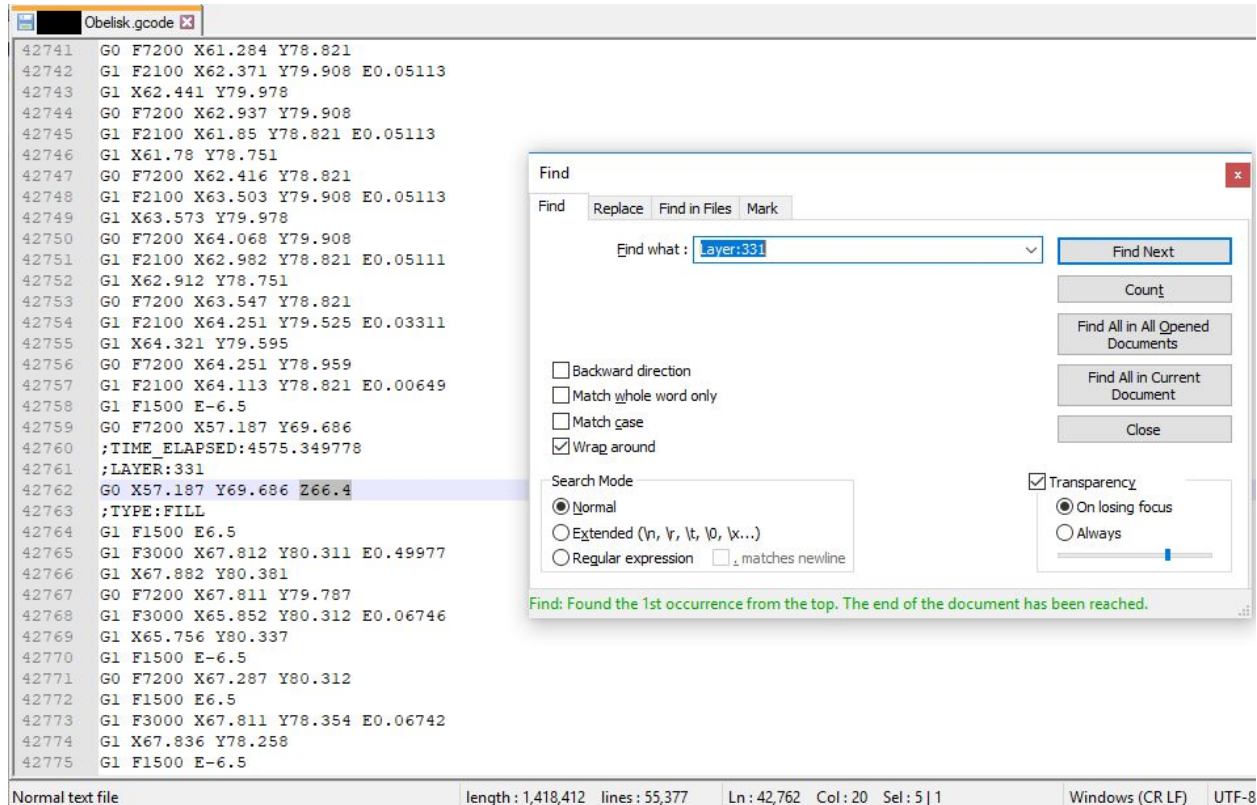
- Make sure there is enough space on the build plate for the printer to return to the home position without getting caught on the existing print..
- The model must be in its original position and **not** have moved or shifted. If the model has been moved (such as the build plate was bumped while the model was attached), refer to Step 18.

Process:

1. Remove the microSD card containing the .gcode file from the 3D printer.
2. Boot Cura, or another 3D slicer, and insert the microSD into your computer.
3. Create a copy of the failed .gcode on the computer and load the copied .gcode into your 3D slicer. This should generate a layer view with a slider to move through print layers.
4. Move the slider as close as possible to the failed layer by comparing the Slicer image on your screen to your failed 3D print. Image Example: Looks like our print stopped around the 331st layer.



- Open the .gcode file with a text editor program (such as Notepad++ or TextEdit). Search for the approximated layer using the CTRL+F command. Example: We searched and found "Layer:331."



- Take note of the Z value of the target layer. The Z value is highlighted in the image above.
- Home the 3D printer on all axes. Once homed, disable motors/steppers.*
* A5 and A31 Commands: *Setup > Auto home* and *Setup > Disable motors*
- Using a jog feature on your 3D printer, find the actual Z-height of the last printed layer. Start by moving the Z to the value found in Step 6.*
* A5 and A31 Commands: *Controls > Move Axis > Move 1 mm > Move Z*
- Gauge the distance from the nozzle to the top of the last layer. The distance should be just enough to move the extruder above the model, typically 0.1mm. The distance can be narrowed by using the move command with the smallest increment (0.1 mm).*
* A5 and A31 Commands: *Controls > Move Axis > Move 0.1 mm > Move Z*

10. After finding the Z-height with the nozzle, record the value displayed on the screen. Ex: Z66.8mm.*

* Should the screen time-out, the value can be found on the home screen coordinates or by repeating the commands to Move the Z-Axis.

11. Using the text editor, find the exact Z-height inside the .gcode. Use the CTRL+F command and search for the Z height. Example: Our Z-height was 66.8, so we searched "Z66.8" and found layer 333!

```

42969 G1 X60.993 Y81.665 E0.05987
42970 G1 X55.835 Y81.665 E0.17156
42971 G1 X55.834 Y68.534 E0.43674
42972 G1 X55.834 Y68.334 E0.00665
42973 G1 X56.034 Y68.334 E0.00431
42974 G1 X69.165 Y68.335 E0.43674
42975 G1 X69.166 Y81.466 E0.43674
42976 G1 X69.166 Y81.666 E0.00665
42977 G1 X68.966 Y81.666 E0.00431
42978 G1 X63.832 Y81.666 E0.17076
42979 G1 X63.832 Y80.064 E0.05328
42980 G1 X63.832 Y79.864 E0.00431
42981 G0 F7200 X63.632 Y79.864
42982 G1 F1500 E-6.5
42983 G0 F7200 X57.201 Y69.7
42984 ;TIME_ELAPSED:4595.435347
42985 ;LAYER:333
42986 G0 X57.201 Y69.7 Z66.8
42987 ;TYPE:FILL
42988 G1 F1500 E6.5
42989 G1 F3000 X67.799 Y80.298 E0.4985
42990 G1 X67.869 Y80.368
42991 G0 F7200 X67.797 Y79.791
42992 G1 F3000 X65.905 Y80.298 E0.06515
42993 G1 X65.809 Y80.323
42994 G1 F1500 E-6.5
42995 G0 F7200 X67.29 Y80.299
42996 G1 F1500 E6.5
42997 G1 F3000 X67.798 Y78.406 E0.06519
42998 G1 X67.823 Y78.31
42999 G1 F1500 E-6.5
43000 G0 F7200 X67.797 Y73.58
43001 G1 F1500 E6.5
43002 G1 F3000 X57.201 Y76.418 E0.36485
43003 G1 X57.105 Y76.443
  
```

Find what: Find Next

☐ Backward direction
☐ Match whole word only
☐ Match case
☒ Wrap around

Search Mode
☒ Normal
☐ Extended (\n, \r, \t, \0, \x...)
☐ Regular expression ☐ matches newline

☒ Transparency
☒ On losing focus
☐ Always

Normal text file length: 1,418,412 lines: 55,377 Ln: 42,986 Col: 23 Sel: 5 | 1 Windows (CR LF) UTF-8

12. In order to pick up where the print stopped, we have to make space for the new layer. To do this, we'll need to delete the layer we found and start on the next layer. In this case, we'll delete all the .gcode from layer 332 all the way back to include the first layer of the print. Be sure to leave the starting .gcode that tells the printer to auto home and other important commands. For Cura, this is typically the first 17 lines of code. Example: Highlighted lines include the commands that should be left and first three lines of Layer 333.

```

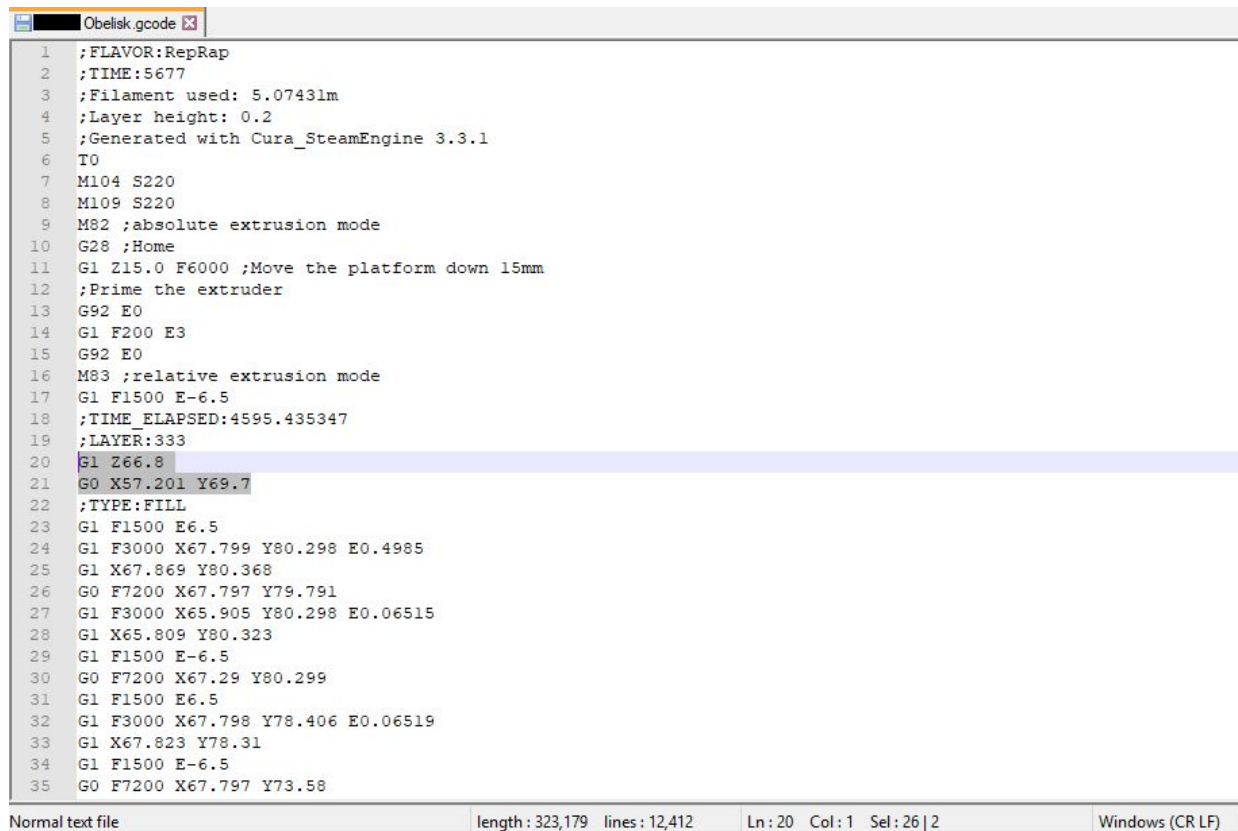
1 ;FLAVOR:RepRap
2 ;TIME:5677
3 ;Filament used: 5.07431m
4 ;Layer height: 0.2
5 ;Generated with Cura_SteamEngine 3.3.1
6 T0
7 M104 S220
8 M109 S220
9 M82 ;absolute extrusion mode
10 G28 ;Home
11 G1 Z15.0 F6000 ;Move the platform down 15mm
12 ;Prime the extruder
13 G92 E0
14 G1 F200 E3
15 G92 E0
16 M83 ;relative extrusion mode
17 G1 F1500 E-6.5
18 ;TIME_ELAPSED:4595.435347
19 ;LAYER:333
20 G0 X57.201 Y69.7 Z66.8
21 ;TYPE:FILL
22 G1 F1500 E6.5
23 G1 F3000 X67.799 Y80.298 E0.4985
24 G1 X67.869 Y80.368
25 G0 F7200 X67.797 Y79.791
26 G1 F3000 X65.905 Y80.298 E0.06515
27 G1 X65.809 Y80.323
28 G1 F1500 E-6.5
29 G0 F7200 X67.29 Y80.299
30 G1 F1500 E6.5
31 G1 F3000 X67.798 Y78.406 E0.06519
32 G1 X67.823 Y78.31
33 G1 F1500 E-6.5
34 G0 F7200 X67.797 Y73.58
35 G1 F1500 E6.5
  
```

Normal text file length : 323,175 lines : 12,411 Ln : 1 Col : 1 Sel : 382 | 20 Windows (CR LF)

13. Next, we need to make sure the nozzle or gantry don't collide with the print as they move up to resume the print. We do this by modifying the .gcode file so the Z movement occurs before the X and Y movements.

- Find the "G0" command after the "LAYER" line. (See Line 20 in the example above.)
- Rearrange the one-line G0 command into two lines of code. The first line uses the G1 command and lists the Z movement value. The second line uses the G0 command and lists the X and Y movement values. Example: We cut Z66.8 and pasted it after the "G1" on line 20 and left the X and Y values after "G0" on line 21.*

*Sometimes it's also helpful to add a little to the Z value so the nozzle will start higher than the print. This reduces the chance of it accidentally hitting the model. In this case we could add 5mm and change Z66.8 to Z71.8.



```

1 ;FLAVOR:RepRap
2 ;TIME:5677
3 ;Filament used: 5.07431m
4 ;Layer height: 0.2
5 ;Generated with Cura_SteamEngine 3.3.1
6 T0
7 M104 S220
8 M109 S220
9 M82 ;absolute extrusion mode
10 G28 ;Home
11 G1 Z15.0 F6000 ;Move the platform down 15mm
12 ;Prime the extruder
13 G92 E0
14 G1 F200 E3
15 G92 E0
16 M83 ;relative extrusion mode
17 G1 F1500 E-6.5
18 ;TIME_ELAPSED:4595.435347
19 ;LAYER:333
20 G1 Z66.8
21 G0 X57.201 Y69.7
22 ;TYPE:FILL
23 G1 F1500 E6.5
24 G1 F3000 X67.799 Y80.298 E0.4985
25 G1 X67.869 Y80.368
26 G0 F7200 X67.797 Y79.791
27 G1 F3000 X65.905 Y80.298 E0.06515
28 G1 X65.809 Y80.323
29 G1 F1500 E-6.5
30 G0 F7200 X67.29 Y80.299
31 G1 F1500 E6.5
32 G1 F3000 X67.798 Y78.406 E0.06519
33 G1 X67.823 Y78.31
34 G1 F1500 E-6.5
35 G0 F7200 X67.797 Y73.58
  
```

14. Save the edited file, under a different name than the original, and put the new .gcode onto the microSD card.

15. Insert the microSD card into the printer and print the renamed file.

16. Observe the printer as it moves into place. If the printer is going to collide with the model, refer to step 18.
17. Watch the printer as it prints the first few new layers! If the nozzle digs into the model, or it is too high above it, refer to steps 7-16.
18. **If your nozzle or gantry collided with your print while moving to the Z-height, or the build plate was moved, it will be hard to restart your print.** Here's where it gets weird! You'll have to try and "eyeball" the correct placement of your model while it's printing, so the layers will fuse together properly. If the nozzle/gantry can't home, it won't be able to restart.
 - a. Insert your microSD card and print your renamed file.
 - b. While it's heating up, take off the clips and move your build plate (with the model attached), so that the nozzle/gantry is able to home, when it has finished heating.
 - c. After it homes and while the nozzle is moving to the Z-height to start printing at the failed layer, reattach the build plate.
 - d. When it starts printing, move the build plate around slightly until the nozzle is lined up with the last layer path on your model. It won't look perfect, but it can still work. Good luck!