Maker's Tool Works

Dual Extruder Supplement

Instruction to install and setup the basics of the second extruder for the MM3 3D printer.

Written By: Micro

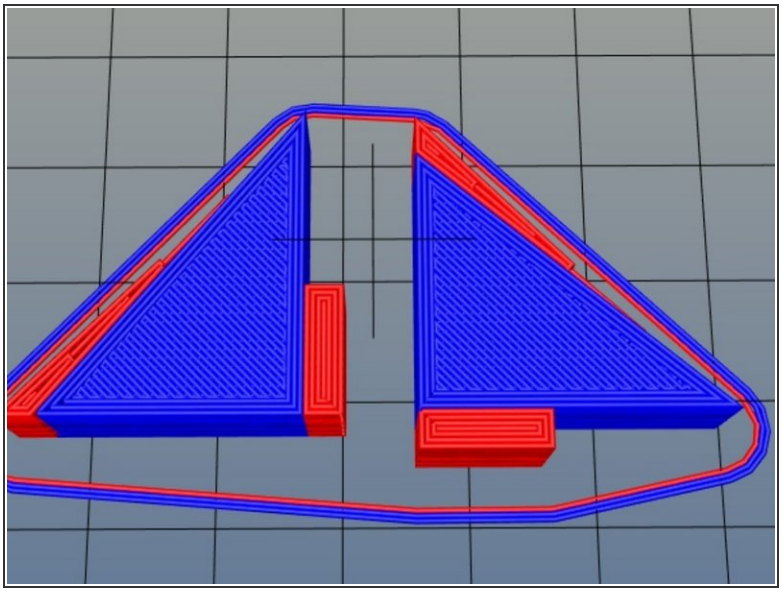
INTRODUCTION

The Dual Extruder option should not even be attempted until you are printing "very" well with a single extruder. If you are having problems printing with a single extruder enabling the dual will increase the problems by X3!

TOOLS:

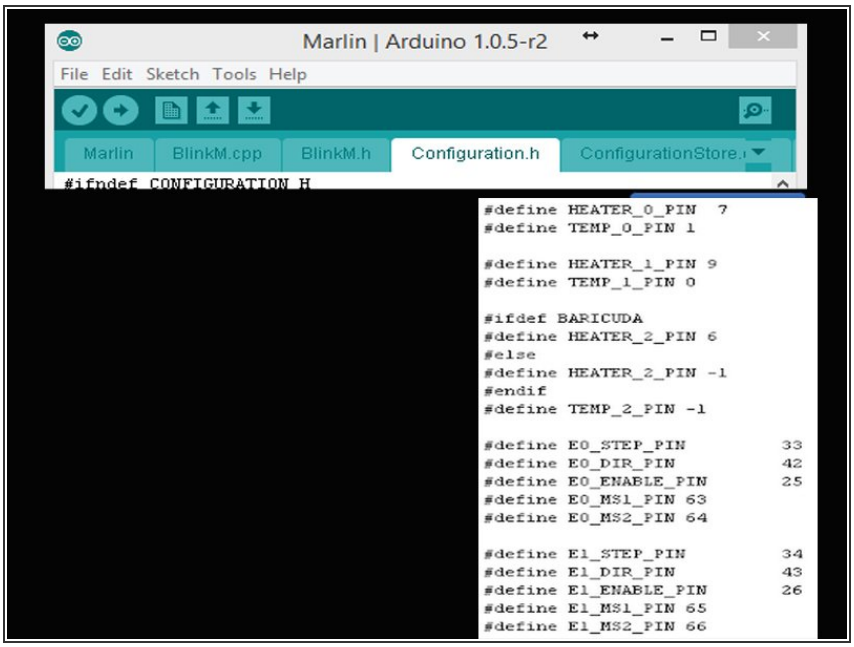
- Alen wrench for Hotend Set Screws (1)

Step 1 — Getting Started



- Hardware (todo) -level to each other -Firmware -Repetier -Slic3r -Test print -settings adjustments-
- Hardware Changes First hardware change is moving the X endstop as far left as possible By moving the X endstop to the far left end the right extruder nozzle should be at the front left starting position on the bed, while the left extruder is left of the bed in open air.
- Next will be installing the 2nd hotend and extruder assembly. This will be installed exactly the same as the first except for the plugs (E1 motor, T1 Thermistor, E1 heater) The extruder cooling fan (the one mounted to blue shroud) will need to be connected into the same plug as the first extruder.
- The motor connector should be installed in the opposite orientation of the first extruder. The red wire should now be closest to the USB connector.
- This way the fans will come on no matter which hotend is heated up.

Step 2



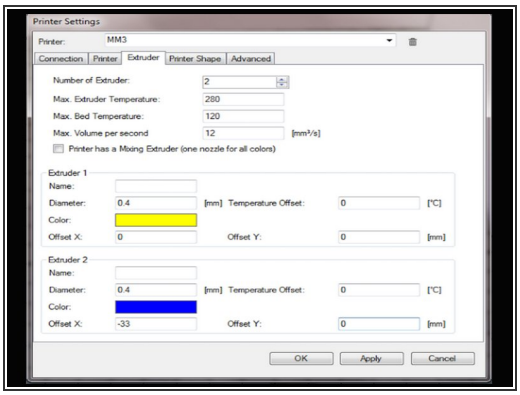
- Firmware Changes NOTICE: this dual firmware will reverse the left and right extruder making your right extruder your primary, this is to ensure full use of the bed. by uploading the single extruder firmware the left will be primary again Open up Marlin.ino and goto the Configuration.h tab .
- Search for and change #define EXTRUDERS 1. Change it to read: #define EXTRUDERS 2
- This enables firmware to know it has two extruders wired up. Next we will enable the thermistor type for the second extruder Search for and change #define TEMP_SENSOR_1 0 to say #define TEMP_SENSOR_1 5
- Both Temp sensor 0 and 1 should have 5 as the value. Next for the e3d we will up the default max temp to 300C Search for and change #define HEATER_1_MAXTEMP 275 to say #define HEATER_1_MAXTEMP 300

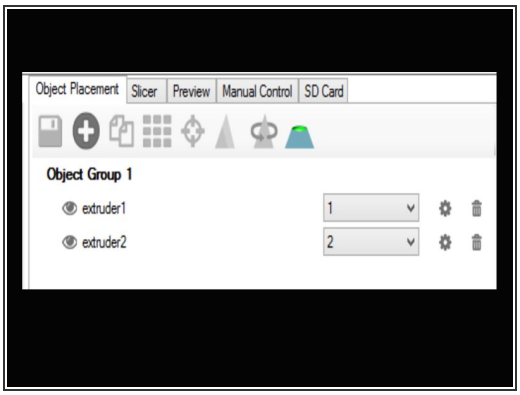
- Next we need to goto the Configuration_adv.h First we need to enable the e3d cooling fans to come on automatically if either one is hot Search for and change #define

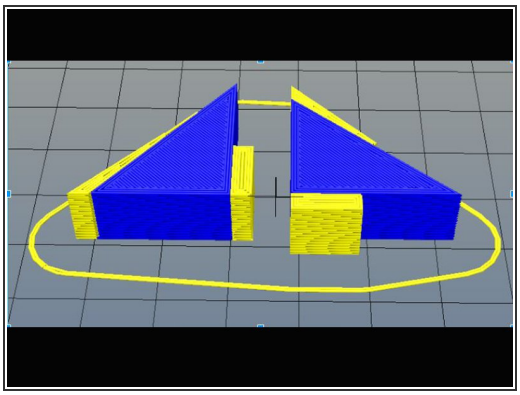
EXTRUDER_1_AUTO_FAN_PIN -1 to say #define EXTRUDER_1_AUTO_FAN_PIN 6

- LAst open pins.h tab search for motherboard “301” Switch all the numbers for E0 and E1 to reverse the extruders Switch heater and therm pins also See image on below for new pins

Step 3 — Repetier Host Setup

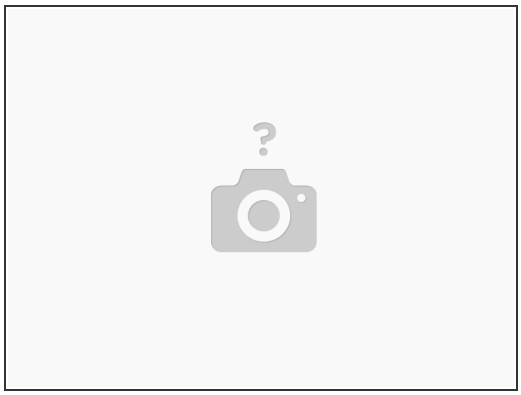






- Click Printer Settings Choose MendelMax 3.0 from dropdown list Click Extruder Tab Change Number of extruders to 2 for extruder 2 choose a color of choice different from the first Add in offset x and Y Starting value should be x=-33 y=0 due to variance in assembly and hotends you will fine tune this number later.
- Slic3r setup In repetier hit configuration to access slic3r settings. Under Printer Settings > General change Extruders to 2 Click on Extruder 2(is should have copied settings from Extruder 1) and set offset to -33 for x and 0 for Y http://i.imgur.com/BphSEzi.png Save
- Test Print Use http://share.makerstoolworks.com/physibl... Load both files into repetier host and move them into the same object group by clicking and dragging the 2nd item into the object group of the top item
- BUG NOTE For some reason Repetier has a extruder selecting bug, the first item in the group is always slic3r to be printed withthe right extruder no matter what the plating view has shown. you can click the item in the list to switch position
- You will have to tell it which extruder goes to which part of the print. do this by selecting the extruder number in the dropdown list next to each item Tell it to slice and your gcode should come out with both parts properly centered and colored according to each extruder.
- Once your gcode looks like this you can try a test print with your filament settings. If your print is way off there is no need to finish the print to start making adjustments, you can cancel after a couple layers to try a small adjustment

Step 4



- If your print is way off there is no need to finish the print to start making adjustments, you can cancel after a couple layers to try a small adjustment Any adjustments to distance have to be saved in both slic3r settings and repetier settings unfortunatly.
- left triangle results If the left large triangle is not touching the small rectangle you need to increase your X distance to fit (from -33 to -33.2 or more as needed) https://lh3.googleusercontent.com/-Nkiq7...
- If the left large triangle is overlapping the small rectangle you need to decrease your X to fit (from -33 to -32.8 or less as needed) https://lh5.googleusercontent.com/-3Iu4_...
- right triangle Results If right large triangle is not touching the small rectangle you need to increase your Y spacing (from 0 to .2 or more as needed)
- If right large triangle is overlapping the bottom rectangle you need to decrease the Y spacing (from 0 to -.2 or lower as needed) https://lh5.googleusercontent.com/-rcezj...
- Once both triangles are matched up to their small pieces correctly then your xy offset is correctly set. https://lh5.googleusercontent.com/-tqjmn...
- For a more challenging Dual color print try the dual color logo http://share.makerstoolworks.com/physibl...

