Configuration guide for the Tevo Tarantula 3D printer

This guide would like to help on basic configuration of Tevo Tarantula 3d printer.

ATTENTION: The use of this printer require some basic knowledge and a little insight so if you don't have these skills make sure that you have a good friend as neighbor! :D

The author don't take any kind of responsability about wrong usage. Use at your own risk!

HARDWARE CONFIGURATION - BUILDING

1. For building the printer I suggest you to follow the manual and the videos that you find below, very well done:

   - `https://www.youtube.com/watch?v=-84HitN0c2E`

   - `http://ruijc.webnode.com/news/tevo-tarantula-prusa-i3/`

   IMPORTANT, don't care about the versus where you have mounted the motor, you'll be able to change this feature later. If there are few redundant parts, keep them as spare!

2. First of all make sure that the PTFE tube, where the filament will pass through, that goes from the motor extrude to the extrude it's well pushed (maximum reachable). This steps even it seems ridiculous will prevent many problems related to a phenomena called 'overextrusion'.

   It's possible to insert the filament with care.

FIRMWARE CONFIGURATION

1. For the first step it's neccessary to download the Arduino IDE envoirment. This program allows you to upload the latest version of the firmware into the main board of the printer.

   The software is available on the follow:

   `https://www.arduino.cc/en/Main/Software`

   Once you have installed the software, run it and edit the following setting by the menù *Tools-Board-**Arduino Mega ADK***. Now you can plug in the usb
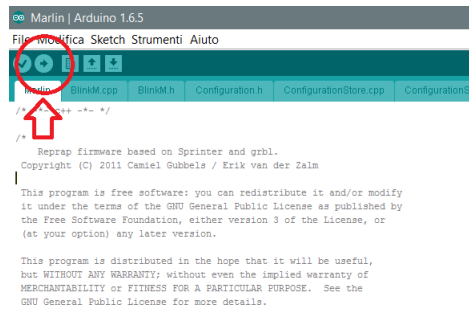
Figure 1: Upload button

cable of the printer and discover what name of the COM port the pc assigne to the device. Put this string into *Tools-Port-**COMx*** where x it's your com number.

2. Now it's time to download the firmware that will guide your printer.

   In the *File* section on the group of Facebook you can find all the versions of the firmware. There are many of this versions beacause each one it's modified for satisfy specific requirement. If you are at the beggining I suggest you to install the basic one and take confidance with the machine. This is named:

   tarantula-marlin.community-v.0.1d

   Download it and extract it.

3. Extracted the firmware move into the directory named *Marlin*. Here you will find the file ***Marlin.ino***. Open it with the Arduino IDE installed at step 1 and (after setting the right com port and board, and plugged in the usb) you are able to upload the program by the button in Fig.1.

   If all goes right you have to see the follow sentence *Upload complete*. Now your firmware is updated.

CONTROL SOFTWARE CONFIGURATION - GENERATE THE GCODE

1. For take control of the printer by pc's interface I recomend you the software at the link below, called Printrun. It's easily donwloadable toghether with a slicing software, called Slic3r.

   `http://koti.kapsi.fi/~kliment/printrun/`

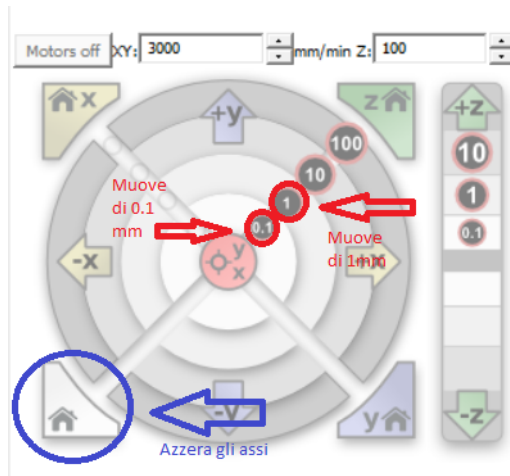   These two software are the heart of the printer.

2

Figure 2: Control pannel

Downloaded and extracted the package above you can run the Pronterface application (red icon). At the top-left of the window it's important to set the proper COM port, the same used in step 1, and click on *Connect*.

At the moment you are linked to the printer and you can control manually the movements.

2. As second step I invite you to verify what's the direction where your Y axis is moving.

The right setting is the following: for an increment of Y(+) the bed have to move forward (opposite side with respect to the motor position).

For move the axes you can use the interface pannel show in Fig.2.

If it's not your case you must change a value of one variable in the file *Configuration.h* inside the firmware directory. The variable is:

INVERT_Y_DIR (row 326)

this variable assume two values *true* or *false*, if the Y axis is moving in the wrong way you have to change this value.

Also you can check if the other axes are moving in the right way too.

Finally make sure that the *home* button, blu circle in figure, move all the axes on their zero position where the three endstop are mounted. In negative case make sure that the endstops are well positioned (check the figure in the building sites) and right connected to the main board.

3. Another foundamental step for the configuration of this machine it's the calibration of the print bedplate.

   This guide (for now) will explain only the manual calibration process.

   The procedure is very easy, what we have to do is make a very levelled surface where the nozzle will deposit the filament. To make sure that the material will be unfiromly deposited in all the surface of the plate it's necessary that this surface is plain. When the Z axis is in its 0 position between the head of the nozzle and the surface of the bedplate one piece of paper have to pass through.

   With a lot of patience repeat the following steps:

   (a) Move to zero all the axes (homing)

   (b) Check if it's possible to bring near the head of the nozzle with the plate by turning the wheels under the springs. If it's not, it's necessary to move a bit down the Z axis endstop. Repeat this analysis.

   (c) If you think it's possible to bring near the parts as required, move a bit the Z+, take position with Y+ and X+ over each corner of the plate (not to far from the center), move again Z- until you touch the plate and then calibrate the position with the wheel.

   For a better explaination of the procedure I invite you to watch the ArcaED's video below:

   `https://www.youtube.com/watch?v=y2Sj9c1Mqlg`

4. Let's move on to the control of the extrusion and to the temperature setting.

   From Pronterface windows show in Fig.3, once you have switched on and plugged in the printer, write in the textbox (A) in figure the value 210 (if you are using the PLA filament equipped with the printer) and click on **Set**, same thing for the textbox (B) writing 60. This procedure will start the heating process of the bed and the extruder.

   Look at the heating process in the graphics near the buttons on the interface and on the LCD screen.

   If the bad reach the temperature all is gone right and you can cook an eggs over it (I'm jocking :D), you can click on **Off**.

   While when the extruder reach the temperature, insert the filament if you didn't do this before, and with care (unblocking the step mechanism) starting to push it until you can see a bit of material coming out from the nozzle. Done it release the step mechanism and try to click a couple of time the **Extrude** (green arrow
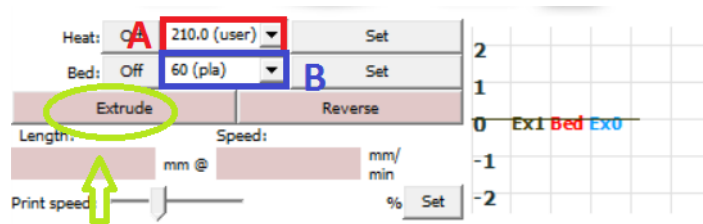
Figure 3: Temperature settings

in figure) button. In this way you tell the printer to move forward the filament, by step of $5mm$ each click.

If during this operation you don't hear strange sound from the extruder or you don't see blockage of the filament this step is finished, otherwise check again if the PTFE tube is well insert.

5. Now (after have fun moving up and down the axes :D) we can move on to generate the gcode.

   For do this come back to the folder created at step 1 and inside the subfolder Slic3r run the *slic3r.exe* application.

   To be sure that the commands generate will be right it's necessary to set some parameters inside the three sheds at the top named: *Print Settings*, *Filament Settings*, *Printer Settings*.

   Below I'll show you the most important parameters to set for a beginner use. You can find more informations about all the parameters and about the software on:

   `http://www.ivanbortolin.it/wp-content/uploads/downloads/2012/08/Guida_`
   `a_Slic3r.pdf`

   Print Settings

   - Layer height = 0.2, allow you to define how much has to be the distance between each layer, of course the more is little this value the more will be the number of the layers and the total print time.
   - Perimeters = 1, this parameter define how many perimeters must have each layer or in other terms how many time the countour will be printed before the filling.
   - Fill density = 50, set how much will be filled the piece, 100 it will be solid, 20 it will be more empty.

- Fill pattern = Honeycomb, define the infill method. Honeycomb it's a strong structure with less material respect to others method.
- $_{\text{speed}}$ Perimeters = 20, speed for the perimeters
- $_{\text{speed}}$ Infill = 40, speed for the layers
- $_{\text{speed}}$ Travel = 130, speed for the non-print movements

Filament Settings

- Diameter = 1.75, diameter of the filament that you are using.
- Extruder: First layer = 235, Other layers = 230, extruder temperature during first layer print (first) and the other (other). It depends on the material that you are using!
- Bed: First layer = 65, Other layers = 60, bedplate temperature during first layer print (first) and the other (other). Again it depends on the material that you are using!

Printer Settings

- GCode flavor = RepRap(Marlin/Sprinter/Repetetier), set which gcode it will be generate based on the firmware that your printer is running.
- Nozzle diameter = 0.4, it's the diameter of the nozzle that the extruder mount.

Set all these parameters and then come back to the main window.

6. If you are at this point, my personal congratulations, you are ready to find the first piece to print.

   I reccomend you a test cube findable on Thingiverse:

   `http://www.thingiverse.com/thing:38108`

   or another piece quite small in a way that if somethings goes wrong you don't lose material and time

   To upload the piece in Slic3r you have to click on **Add** button. Moving the piece as you wish in the plate and then export the gcode with the button *Export G-code* in a folder.

7. Final step it's come back on Pronterface and with the **Load file** button load the .gcode file just create. Connect the printer and click on **Print** button to start printing.

   You can monitoring the status of the printing by the graphics animation. If something goes wrong you can switch off the printer clicking on **Off** button.

Some important feautures of the printer, like auto-levelling, double extruder or 'advanced' settings are not discussed in this guide because I don't have it yet and because I would like to write an introductive guide (I leave some fun for you of course).

If you like the guide or you want to contribute and/or find some mistakes, you can contact me on Facebook group or by sending me an email at this address richimenga@hotmail.it.

Have fun!

Riccardo Mengacci.