

# To flash new firmware or optiboot Bootloader to Creality v1.1.4 board and then Calibrate the printer:

Before you do anything get your current EEPROM settings before you flash a bootloader or firmware to the Creality board by doing the following:

Backup Firmware settings?

Issue a M503 command in a g-code terminal to report your current settings, write them down, flash the firmware then do a M501 command to load the saved settings from EEPROM, then a M503 so you can see the settings and compare them to what they were before you flashed the firmware, if all settings are correct do an M500 to save them (just in case you made changes).

URL on EEPROM settings:

<https://github.com/MarlinFirmware/Marlin/wiki/EEPROM>

**\*\*NOTE:\*\*** PLEASE, PLEASE do a M503 in a terminal window (via Octoprint or pronterface) so you can document your EEPROM settings (all you calibrations you have done up to this point). Sometimes when you flash new firmware you will get an EEPROM version mismatch error and to clear it you have to do "M502 and M500 then M501 is normally enough to clear the eeprom and write a new version of everything including the version number" {quote from <https://github.com/MarlinFirmware/Marlin/issues/12860>, which sets all your EEPROM values back to the Marlin default settings and you lose all your calibrations.

Now flash the bootloader or firmware:

To flash new firmware to the Creality board you must first place optiboot bootloader onto the CrealityV1.1.3 or V1.1.4 motherboard. If you think you have flashed the bootloader already then go to the next step. The bootloader only needs to be flashed one time. If you try to send a new firmware file to the motherboard and you get the error: " avrdude :stk500\_getsync(): not in sync: resp=0x00" then you thought you flashed the bootloader but the Creality motherboard at this moment DOES NOT have a bootloader flashed and saved correctly. This happened to me.

To flash the bootloader I did the following:

I bought this kit from Amazon

(<https://www.amazon.com/Creality-3D-Upgraded-Leveling-Ender-3s/dp/B07R3RRR13>) so I could one day install a BLtouch but this kit also comes with a USB ISP programmer that you can

use to flash a bootloader or flash firmware onto the Creality motherboard. A bootloader allows a user to flash firmware onto the motherboard without the need of the programmer once the bootloader has been flashed properly.

You need to get to the motherboard so unscrew the bolt on the top of the box that is in the front right side.

Turn off the printer and unplug it, turn over the printer and unscrew the three screws to detach the box cover.

===== power off main power on printer, motherboard will be powered by USB port

To flash the bootloader on to the Enders 3 pro follow these directions:

Remove the LCD cable and find the USB ISP programmer that came with the BLTouch kit from Creality.

Plug in the cable to the programming board (only one way to connect due to cable notch) the other end will plug into the USB port on your computer BUT do not plug into computer yet! Now find the VCC and GRD on the programming board (this side will go against the side of the metal case when you plug in the programmer on the six pins next to the LCD port connector. Yes, the LCD cable will not be plugged in while the programmer is plug into the motherboard.

URL to help use the programmer that came with the BLTouch kit:

[https://www.reddit.com/r/ender3/comments/cfmbdy/howto\\_installing\\_a\\_bootloader\\_to\\_an\\_ender\\_3\\_pro/](https://www.reddit.com/r/ender3/comments/cfmbdy/howto_installing_a_bootloader_to_an_ender_3_pro/)

The “bootloader “ link does not work so go to the source to find the optiboot\_atmega1284p.hex boot loader file.

URL:

Optiboot bootloader file found at this URL:

<https://github.com/Lauszus/Sanguino/tree/master/bootloaders/optiboot>

Get the file labeled optiboot\_atmega1284p.hex. You will have to download the whole .zip file from GitHub to get to this file. Use this URL:<https://github.com/Lauszus/Sanguino>, then hit “clone or download” and select “Download ZIP”.

Follow the reddit article until you get to inputting the fuse values. Use these fuse values instead (the reddit article gives you the fuse values for flashing the firmware NOT for flashing the optiboot boot loader).

Found these values at URL:

<https://github.com/Lauszus/Sanguino/blob/master/boards.txt>

Use “...” button to find the screen to enter these values, enter below values and hit “write”.

====flashing optiboot bootloader to motherboard ====

Fuse values for flashing the opti bootloader file optiboot\_atmega1284p.hex:

LowValue: FF  
HighValue: DE  
ExtValue: FD

Use “Load Flash” button to select the optiboot\_atmega1284p.hex file.  
Open.

Ensure the following are checked before you hit “Auto”:  
Chip Erase, Program Flash, Program Fuse.

Hit “Auto” Button.

If successful a green progress bar begins.

Will show 1: Erase,Write Flash, Write Flash , Successfully done.

Now do it again, just to ensure it has taken. Others have found that it might take two flashes before the board actual takes the flash.

LowValue: FF  
HighValue: DE  
ExtValue: FD

Use “Load Flash” button to select the optiboot\_atmega1284p.hex file.  
Open.

Ensure the following are checked before you hit “Auto”:  
Chip Erase, Program Flash, Program Fuse.

Hit "Auto" Button.

If successful a green progress bar begins.

Will show 1: Erase,Write Flash, Write Flash , Successfully done

I have now been told that the above flash value may not work for others. But, doing it does not damage the motherboard. I would then recommend to go ahead and use the following fuse values ( the same ones to flash the firmware to the motherboard) if the above fuse values do not flash the bootloader for you:

Flash the bootloader with:

LowValue: DC

HighValue: D6

ExtValue: FD

Use "Load Flash" button to select the optiboot\_atmega1284p.hex file.  
Open.

Ensure the following are checked before you hit "Auto":  
Chip Erase, Program Flash, Program Fuse.

Hit "Auto" Button.

If successful a green progress bar begins.

Will show 1: Erase,Write Flash, Write Flash , Successfully done

And do it again for good measure.

Flash the bootloader for the last time using the following values:

LowValue: DC

HighValue: D6

ExtValue: FD

Use "Load Flash" button to select the optiboot\_atmega1284p.hex file.  
Open.

Ensure the following are checked before you hit "Auto":  
Chip Erase, Program Flash, Program Fuse.

Hit "Auto" Button.

If successful a green progress bar begins.

Will show 1: Erase,Write Flash, Write Flash , Successfully done

Once done flashing twice or four times, exit progisp.exe and disconnect the USB cable from your computer, and disconnect the programmer from the Creality motherboard, plug your printer's LCD screen cable back in and close up the motherboard box. You have now properly flashed the optiboot bootloader to the Creality motherboard.

====flashing the FIRMWARE file of your choice=====

Fuse values for flashing the your preferred firmware file (firmware.hex):

Fuse values for flashing your firmware MarlinV2-bugfix or firmware.hex file or the firmware file of you preference!:

LowValue: DC

HighValue: D6

ExtValue: FD

Use "Load Flash" button to select the firmware.hex file.  
Open.

Ensure the following are checked before you hit "Auto":  
Chip Erase, Program Flash, Program Fuse.

Hit "Auto" Button.

If successful a green progress bar begins.

Will show 1: Erase,Write Flash, Write Flash , Successfully done.

Once finished , exit progisp.exe and disconnect the USB cable from your computer, and disconnect the programmer from the Creality motherboard, plug your printer's LCD screen cable back in and close up the motherboard box. You have now properly flashed the optiboot bootloader to the Creality motherboard.

Note: one user pointed out that they had to do the flash twice before it finally worked. So if it did not work the first time try it again.

====combing the firmware and bootloader in one file:

See this URL:

<https://forum.arduino.cc/index.php?topic=193431.0>

=====end of flashing boot loader to motherboard and Firmware to motherboard =

Use VScode or ATOM to make changes to the Marlin V2.0 firmware or the firmware preference of your choice.

Here is a great video on how to setup VScode so you can use Marlin (or your preferred firmware) on your Ender 3:

Link to YouTube videos to install VScode for Marlin firmware:

<https://youtu.be/W6zYvRgGr3Q>

<https://youtu.be/7Xr9Eb7o0rw>

Follow this YouTube Video to setup Atom Editor and how to configure vanilla Marlin V2.0 bugfix for Ender 3, now take that folder and rename xxx-OLD, copy that folder and call it xxx and place it in c:\Users\your\_user\_name\Documents\Arduino\Marlin-bugfix-2.0-Ender 3-CrealityV1.1.4 . As long as it's in the same subdirectory as where you did the Arduino projects. On windows 10 machine, VScode will not be able to find the libraries correctly unless you create your MarlinV2.0 in the same directory as your Arduino projects.

URL:

[https://youtu.be/U8\\_IdMckGDE](https://youtu.be/U8_IdMckGDE)

And

Starting at 4:16 <https://youtu.be/GNGN2iSQ5j4>

In this video he is setting up MarlinV2 for SKRV1.3 but you use MarlinV2 on CrealityV1.1.3 or Creality V1.1.4 boards ... just name you folder in Arduino "Marlin-bugfix-2.0-Ender 3-CrealityV1.1.4" instead of his name "Marlin-bugfix-2.0-Ender 3-SKRV1.3".....still copy the correct Ender-3 header files like he does at 6:47,

at 7:48 you do NOT want the environment set for SKRV1.3 instead ( for the Creality motherboard V1.1.3 and V1.1.4 ) set default\_envs in platformio.ini in ATOM or VScode editor to the following:

default\_envs = sanguino\_atmega1284p

USE VSCode to make changes to firmware and compile to get your firmware.hex file.

Since you flashed the opti bootloader you have a choice on how to flash the firmware file to the motherboard: You can use VSCode to upload to flash the firmware automatically by attaching a USB cable to the front of your 3D printer or You can use OctoPrint "Firmware Updater" to flash the firmware.hex file.

VScode will store the new .hex file for the Creality V1.1.4 board in the following directory:

"C:\Users\your\_user\_name\Documents\Arduino\Marlin-bugfix-2.0-Ender 3-CrealityV1.1.4\.pio\build\sanguino\_atmega1284p\firmware.hex"

++++ OctoPrint Firmware Updater+++

Firmware Updater Plugin:

On the raspberry pi running OctoPrint, you must install the following:

\$sudo apt-get install avrdude

\$whereis avrdude

URL for OctoPrint Firmware Updater:

<https://github.com/OctoPrint/OctoPrint-FirmwareUpdater>

To use the OctoPrint plugin set the following in "Firmware Updater" plugin:

Flash Method: avrdude

AVR MCU: ATmega1284p

Path to avrdude: /usr/bin/avrdude

AVR Programmer Type: arduino

On my OctoPrint avrdude is in "/usr/bin/avrdude"

Ensure OctoPrint is connected to the Ender 3 Pro with the USB cable (ensure that the USB cable is powering the Creality board by removing the USB adapter, bought from TH3D( <https://www.th3dstudio.com/product/power-blough-r-pi-usb-power-blocker/> ), that stops the power going to the board) without the adapter board, have the Enders 3 pro powered off, now hit the "Flash from File" button.

"C:\Users\your\_user\_name\Documents\Arduino\Marlin-bugfix-2.0-Ender  
3-CrealityV1.1.4\pio\build\sanguino\_atmega1284p\firmware.hex"

\*\*\*\*Note: if you turn on the printer and get a blank blue screen it means no firmware has been loaded.

\*\*\*\*\*Notes: if VSCode has problems, try disconnecting the USB cable , exit VScode, install USB cable ,open VScode, wait for VScode to load , select Platformio from sideMenu, select Project tasks, select Rebuild Intellisense Index" then try to recompile or upload again

#####%%#####

=====3D Printing calibrations:

1. Calibrate your extruder: only needs to be done if something changes or you reload the firmware , follow [Teaching Tech](https://youtu.be/VyDv0CnLWU0) video: start at 4:38 <https://youtu.be/VyDv0CnLWU0> or
  - a. M503, write down M92 command's E value, my E is 98.86
  - b. M83
  - c. G1 150
  - d. G1 E100
  - e.
2. Calibrate your X,Y,Z axis: only needs to be done if something changes or you reload the firmware, use the following URL to calibrate X,Y,Z axis or follow my steps below <https://all3dp.com/2/how-to-calibrate-a-3d-printer-simply-explained/>
  - a. So, print a 20mmx20mm cube XYZ (<https://www.thingiverse.com/thing:1278865>) cube. Take your calipers and measure the X-Axis length (8 different times), take the average of the X-Axis length then  $20\text{mm} \times (\text{X-Axis Steps/mm from M92} / \text{average of X-Axis length that you measured}) = \text{new X-Axis Steps/mm value}$ . Now do an M92 Xii.ii in terminal window, followed by an M500 to save new value to EEPROM. Repeat until happy.
  - b. So, from the print of the XYZ cube, take Y-Axis measurement 8 times. Take the average of the Y-Axis length. Now take  $20\text{mm} \times (\text{Y-Axis Steps/mm from M92} / \text{measured average of Y-Axis length}) = \text{new Y-Axis Steps/mm value}$ . Now do an M92 Yii.ii in terminal window, followed by an M500 to save new value to EEPROM. Repeat until happy.
  - c. So, from the print of the XYZ cube, take Z-Axis measurement 8 times. Take the average of the Z-Axis length. Now take  $20\text{mm} \times (\text{Z-Axis Steps/mm from M92} / \text{measured average of Z-Axis length}) = \text{new Z-Axis Steps/mm value}$ . Now do an M92 Zii.ii in terminal window, followed by an M500 to save new value to EEPROM. Repeat until happy.
  - d. So, now that you have done all the axes do an M503 to see if you new values have been saved to EEPROM.



3. Calibrate your filament, do this every print. Use a caliper, measure your filament diameter at several locations. Average out the measurements (at least 3) and enter that into your slicer under filament diameter.
4. Calibrate your z-height and first layer. Do this whenever something changes in your printer. First write down the First Layer Height percentage and First Layer Width percentage you're currently using in your slicer settings (so you can change them back after you finish this calibration), also do a M503 in terminal window (if you are using an ABL) so you can write down your M851 values for X, Y, and Z (this is your Z probe offset value when using ABL), if you do not use an ABL then you will have to manually move your Z end stop:
  - a. Print a single layer (say 20\*20mm cube) with your first layer at 100% height and width.
  - b. Using a caliper measure the print in several places (at least 8) and adjust your bed (no ABL) or gcode z offset (if you have Automatic Bed Leveling or ABL) by using M851 Z-xx. In the M851 the Z number should be a negative number. Use M500 to save and use M503 to see if the change occurred.
    - i. You measure the height of the print in 8 different places in order to find out if your z height is set correctly. Take your print height (for me 0.2mm) - the average print height measured = adjustment for the Z offset or amount to manually move up the build bed. A negative z height means the nozzle is too far away from the build surface. If you get a negative z height move the nozzle closer to the bed by adjusting Probe Z offset if using auto bed leveling (use M851) If not using ABL just level your bed manually and bring the nozzle closer to the bed before setting your z end stop.
  - c. Repeat until you are happy.
5. Calibrate your extrusion multiplier! Do this every print, to be a little lazy every roll, to be really lazy every brand of material.
  - a. Print out a cube (20\*20mm)
  - b. In vase mode (single outline, no infill, no top or bottom layers)
  - c. Set your extrusion multiplier to 1
  - d. Set your extrusion width to be equal to: In Simplify3D use "auto" which defaults to 0.48mm extrusion width; In other slicers use 0.5mm as your extrusion width.
    - i. You do not want to use 0.4mm extrusion width for 0.4mm nozzle because the 3D printer slicer does not like that thin of a wall. Here is a quote from "[Maker's Muse](https://gumroad.com/l/QWAH) 50 3D Printing Tips 2017" (<https://gumroad.com/l/QWAH>) I quote "The thinnest you can realistically reproduce on a single section in a hobby FDM 3D print is the thickness of a single extrusion width, which with a 0.4mm diameter nozzle will be about 0.48mm wide. However, in practice, these thin details will be VERY weak, as 3D Printing does not carry the same level of strength as injection or blow molded plastic."
      - In practice, I would recommend the thinnest you go in your model is around 1.5mm, and any stand alone detail such as vertical columns

should be at least 2.5mm thick to avoid breaking and bending during the print.

- If you want to test your 3D Printer's capability in reproducing thin wall details, check out our video covering how using the Maker's Muse thin wall test file <https://youtu.be/uUcK2QiX2NE> “.
  - e. Print the cube, Measure the walls with a caliper in several places (at least 8) and get an average. If you used Simplify3D then you are looking for measurements close to 0.48mm for other slicers you want 0.5mm..
  - f. Change your extrusion multiplier: new multiplier = old multiplier x (extrusion width/ average measurement). For Simplify3D: new extrusion multiplier =  $1.0 \times (0.48 / \text{measured average wall width})$ . For other slicers: new extrusion multiplier =  $1.0 \times (0.5 / \text{measured average wall width})$ .
  - g. Repeat until you are happy.
6. PID tune your Hotend and your heated Bed (do this when... just like #4 item). Before and after you calibrate temperatures and any time you change a fan or move something or season changes. PID tune your hotend and bed to keep your temperature fluctuations to a minimum. To use M304 PID tune for Heated Bed you need to enable it in Marlin firmware (uncomment PIDTEMPBED):
- a. Note current PID values M503.
  - b. M303 E(Extruder 0 for hotend, -1 for bed)C(#of cycles 3-8)S(Desired Temperature).
  - c. M303 E0 C8 S210= PLA Hotend tune for 8 cycles.
  - d. Enter new HOTEND values in terminal M301 Pxx.xx lxx.xx Dxx.xx (Kp is new P, Ki is new I, Kd is new D. Use M301 for Hotend ...USE M304 for Heated Bed!!
  - e. M303 E-1 C8 S60 = PLA heated BED.
  - f. For HEATED bed new values enter in terminal M304 Px lx Dx
  - g. M500 save to EEPROM.
  - h. M503 check values are saved.
7. Calibrate your temperatures: do this for every different filament (color, brand, material, etc).
- a. Grab a temperature calibration tower off Thingiverse.  
<https://www.thingiverse.com/thing:915435>
  - b. 2. Set the temperature range to the range listed on your filament or by manufacturer.
  - c. 3. Print the calibration tower and choose the best temp.
8. Calibrate your fan speed. (I have a Delta with three powerful layer fans. If they all run at 100% the hotend loses temp)
- a. Print your calibration tower again but change the fan speed vs the extruder temp.
  - b. Choose the best fan speed for the finish you desire.
9. PID tune your hotend again with the layer fan set to the ideal speed. As noted above if your fans in any way change the tempting your hotend (all layer fans do) you should PID tune the hotend with the fan set at the most common speed to keep temperature fluctuations to a minimum.

10. More calibration tests that can be run to tune your slicer settings: "Ender 3 Pro: Initial Setup and Recommended Prints": URL: <https://www.instructables.com/id/Ender-3-Pro-Initial-Setup-and-Recommended-Prints/> and [Teaching Tech's](https://www.teaching-tech.com/speed-test/) (speed test) <https://www.thingiverse.com/thing:3071464> and <https://youtu.be/3ylebnVjADM>.
11. Calibrating for retraction speed. I started with this profile from Angus' Ender 3 profile. URL: [https://www.youtube.com/watch?v=\\_QRb54zVPfQ&t=24s](https://www.youtube.com/watch?v=_QRb54zVPfQ&t=24s). From Angus' above video on YouTube channel [Maker'sMuse](#):
- Angus' new Simplify3D and ideaMaker profiles for the Ender 3 (will be updated as they are tweaked and improved - enter '0' to download for free):
  - [https://www.youtube.com/redirect?v=\\_QRb54zVPfQ&redir\\_token=M3FGqLpxhsXYbVoJyIEN30LvQHp8MTU3NTQwMDE3NUAxNTc1MzEzNzc1&event=video\\_description&q=https%3A%2F%2Fgum.co%2FZrffZ](https://www.youtube.com/redirect?v=_QRb54zVPfQ&redir_token=M3FGqLpxhsXYbVoJyIEN30LvQHp8MTU3NTQwMDE3NUAxNTc1MzEzNzc1&event=video_description&q=https%3A%2F%2Fgum.co%2FZrffZ)
  - I started with this profile and changed it's Extrusion Multiplier (you calculated thus in STEP 5). I used his default print speed because my printer can print at 90mm/sec from step 10 ([Teaching Tech](#). Speed test). So I ONLY changed my extrusion multiplier all other setting I used and to my pleasant surprise I ended up with NO STRINGS!!
  - I reran the XYZ calibration cube just to ensure everything was correct.
  - After this I considered my profile calibrated. I hope this helps others.

=====