

Guide to building your FDM Printer - PART 1

(Creality Ender3v2 also useful for other Enders/FDM Printers)

17 May 2024

Posted on Reddit

[Ender3v2 Hardware Build Guide for newbies : ender3v2 \(reddit.com\)](#)

Prepared by reddit user Kwakers2001 (copyright 2022)

Preface	6
Make a note of everything – Take Photos	7
Examples	7
Useful Videos	7
Useful Links	8
Check your Display Screen	8
Board Codes for Stepper Motors	8
Build your Ender	9
Build Videos	10
Tomb of 3D printed horrors	10
My Experience	11
Just Vlad	11
Ricky Impey	12
Recommended - Alignment Problems and Solutions Video	12
Alex Kenis	12
Printables	12
Possible Build Issues	12
Bed Hitting the Y motor	12
Z axis bracket issue	12
Eccentric Nuts Video	13
Ender3v2 User Manual	13
You have built it, yes. Don't switch on yet....	14
Basic Tests	14

Cabling issues	15
Fire Risk	15
Connector Issues	15
Earthing	16
House Earthing	16
Printer Earthing	16
Check Thermal Runaway software protection	17
Creality Carborundum Glass Plate	18
Cleaning The Glass	18
How to get rid of the print “image” on the bed.	18
Prints are just not sticking	18
How to make prints stick video	19
Acetone	19
I can’t get my print off the glass.	19
If you have bought a probe	20
Ready to Bed level (really you mean “Tramming”)	21
Filament Loading	21
How to load your filament the easy way	21
Filament Unloading	22
Temperatures (for PLA)	22
Manual - Bed Tramming/Levelling using the wheels	22
How to properly level your bed.	22
Worth watching to understand what happens when you adjust your bed.	22
Cheps Bed Levelling Video	22
Marlin Bed Levelling Guide	22
Paper Thickness and Wheel Movement	23
Wheel Turns	23
Probe - Bed Tramming/Levelling with a Probe	24
Set your initial Z-offset	24
Bed Levelling Prints (Manual & Probe)	24
My Recommendation	24
Initial Bed Level	24

Time to find the “Squish”	24
The Good and the Bad - Bed levelling diagram	25
The Diamond Print	25
Teaching Tech	25
Bed Levelling Issues	27
Change stock springs and wheels	29
Why Change the Springs and the Wheels	29
Bed Spring Options	29
My Setup	29
Spring Guides	29
Spinning Bed Screws	29
Clicky Wheels (0.01mm per click)	30
Filament Wheel	31
The Plastic Extruder Arm	32
Wet Filament	33
Wet Filament out of the bag	33
Drying Wet Filament	33
Firmware	34
If you have a probe	34
BLtouch, CRTouch or a 3Dtouch (clone)	34
Other Probes	34
Which Firmware	34
Marlin Based Firmware	35
Non-Marlin Based Firmware	35
Creality Firmware (Marlin Based)	35
Firmware Upgrade	35
Check Hardware	36
Save EEPROM settings (optional)	36
Your Original Firmware	36
Email a Copy	36
Installing Firmware using the SD Card	37
SD Card Requirements	37

Filename Requirements	37
Not firmware but...	37
What I do	37
Firmware flash official video	38
Screen Firmware Install & Reverting To Stock	38
Which Screen Do I Have?	38
DACAI Screen Firmware	38
DWIN Screen Firmware	39
VIEWE Screen Firmware	39
Updating the Screen firmware	39
Possibly add a probe	40
Which Probe - BL/CR touch/3d Touch (Clone) or BFPTouch	41
Homemade Probes	41
Probe Installation	42
Firmware Installation	42
Physical Installation	42
Useful Videos	42
Useful Links	42
Probe Motherboard Connector	42
Nozzle to Probe XYZ offset	43
Physically Installed Your Probe, Don't do anything yet.....	43
Before trying to bed level, create a mesh and print, Test your probe	44
Probe Test	44
Good Probe	44
Example Problem Probe	45
Next Tram/Level your bed (using the wheels)	46
Wheel Turns	46
Bed Tramming Wizard	46
Now you can create a Mesh	46
Finally - Bed Level Prints	47
Probe Issues	47
BLTouch Blinking Red	47

Cabling issues	48
Bltouch mounting	48
Probe Interference	48
Loose grub screw and nozzle crashes	48
Range and Standard Deviation Issues	48
Probe tip dropping	49
Remagnetise Your Probe	49
Silicone Spray (Don't Do It !!!!)	50
Cleaning the pin and hole	50
Probe not dropping at the correct angle	50
Marlin's BLtouch FAQ	51
USB Cable Connection/Wifi connection	51
My Setup	51
WARNING: USB Connection - 5v Power Issue	51
Top Tips	52
Basic Calibration	53
Printables for the XYZ Steps Calibration	53
Slicer Flow Calibration	53
Printing Issues	53
First things to print after calibration	54
That's it for now - no.... There's a Part 2	54
Link to Part 2 of The Guide	54
Glossary	55
Link to 3D printing glossary	55
Document Changes	56
Required Updates	56
Requested Updates	56

1. Preface

“Sorry if all this seems really obvious. We make a lot of assumptions when we give advice. I often try to go back to basics.” (a reddit user wrote this and I liked it, as back to basics and KISS are always good!!).....this guide is not simple!

After a couple of months of headaches, I thought I would write a guide for everyone to try to help them build their hardware in less time than it has taken me. Hopefully by the end of this you should be able to print a good bed levelling print and be able to calibrate your printer.

PLEASE NOTE: This is a guide and you do everything at your own risk. If you are not comfortable or competent to do any particular procedure then DO NOT follow this guide. All the information was found on the internet and I have just compiled it into an order I thought would be useful to people. Please read up and research the points I make yourself and check all information for yourself before proceeding with any changes to your own machines. You follow this guide at your own risk.

1st August 2002

The guide has now been split into two parts as it was beginning to get quite long.

Part 1 : I have tried to keep part 1 as a get you up and printing guide.

Part 2 : Is more of a list of hints, tips, and possible useful links.

19th April 2024

If you are actually reading this and you want a speedier solution to all your worries with bed levelling this is my advice 2 years on. (although manual bed levelling is a good learning curve).

Build as per this guide (square, true, electrics etc.)

Don't start manual levelling, instead:-

Install silicone spacers (or look at solid spacers, but these are a bit more advanced as you cannot bed tram and must rely on a mesh and the spacers being pretty accurate and your frame being pretty square).

Invest in a “good” (cheapest use triangle labs as they are good) probe.

Install mrisco firmware. Then....

Bed tram

Mesh

If you need to use glue stick, use it (but you should not need to and it will leave a layer on your print, there is a solution with pva glue spray mixture of some sort that I need to test). Also , use brims if needed.

That should sort you out quickly. Honestly don't bother with manual bed levelling or really really cheap probes, and don't use creality firmware.

Once you can print a decent print, do Chapter "Basic Calibration" and then....

Print all the spare parts you may need in the future, especially the "Plastic Extruder Arm".

Then enjoy your printing future.....

2. Make a note of everything – Take Photos

Before you start to build anything:-

1. Open your delivery box and check everything is there according to the instructions.
2. Take off the cover for the mainboard (the box with the SD slot and USB port) as per the [Service tutorial Ende 3 V2 the mainboard replacement](#)
3. Make a note of the motherboard version number, the processor number (you may need a magnifying glass or macro lens) and also any markings on the SD card slot or processor (e.g. O on the processor or letters/markings on SD Card slot – this may give you stepper motor information if you need it later).

4. Take good photos that you can zoom in on. You may want the information a month down the line and you do not want to have to turn your printer over just to get a version number.
5. Open up the screen display and take a photo. See [Check your Display Screen](#) below (you have to open up the display, it's easy just some screws on the back – take a photo as you may need it later).

Examples

Here, for example, is my information.

Motherboard Version : v4.2.2 (or you may have a v4.3.1 or v4.2.7)

Processor : GD32F303 RET6 (512K) (or you may have a STM32F103 RCT6/RET6 or another)

Steppers : TMC2208 see [Board Codes for Stepper Motors](#) below (get this from the code on the SD card metal cover, or on the processor. You don't need the "steppers" info at the moment but in future you might (don't worry if you cannot find out which stepper motors you have).

Screen : DACAI (or you may have a DACAI or VIEWE), see [Check your Display Screen](#) below.

Useful Videos

 Service tutorial Ende 3 V2 the mainboard replacement

 Service tutorial Ender 3 V2 power supply replacement

Useful Links

Check your Display Screen

[Identifying Ender 3 V2 screen versions - Frequently Asked Questions - Marlin Firmware Service - Forum \(crc.id.au\)](#) - There are now 3 types of screen according to this site.

<https://www.th3dstudio.com/hc/guides/troubleshooting/creality-printers-dacai-and-dwin-color-lcds/> - Which screen do you have, DWIN or DACAI?

Board Codes for Stepper Motors

[Creality V4.2.X Board Driver Codes - TH3D Studio LLC](#) - Letters on the SD Card Slot

My Letter O on GD processor

My GD processor has an O on it and I emailed creality and they said I had TMC2208 drivers, so I assume it would be the same for anyone else with an O (I doubt they actually have a record on my machine), however the only real way to check is to remove the heatsink and replace afterwards with the correct heatsink compound/glue.

3. Build your Ender

Don't rush into building the Ender3v2, in fact if you are reading this whilst waiting for delivery it is worth watching these three videos at least, I know they are long but you may save yourself hours of fault finding later by building the frame square and tightening nuts etc.

IMPORTANT : If you have purchased a probe, put it to one side and ignore it for now. You will fit it later, once you know that your printer is working OK.

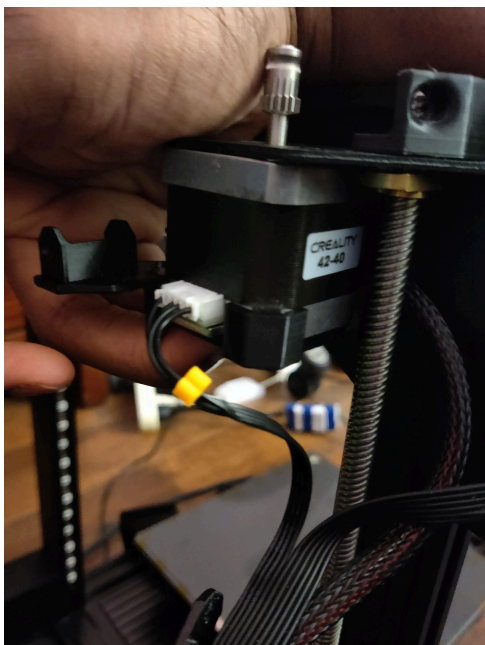
IMPORTANT : Make sure **everything is square**. If you do not have an engineering square, go to the pound shop or dollar shop and buy the kids plastic maths set, two triangles and a protractor, make sure you get the solid plastic type one, not the flexible TPU type ones. That should do the trick.

WARNING: Don't over tighten nuts/screws and strip the thread, some are only aluminium.

WARNING: (reddit user) Make sure you don't tighten down the screw holding the thermistor in place. You can crush the wire coating and cause a short.

TIP: You could use aluminium foil as fillers to adjust the uprights if you are struggling to adjust them.

TIP: During your build, it may be worth rotating the Extruder Motor 90 degrees whilst you are building your ender (see picture below). I found that my connector was just catching the Z screw when the cable was plugged in (I saw the wear on the connector after a couple of months) and rotating it 90 degrees stopped this from happening. Yours may not, so it's up to you.



Build Videos

Tomb of 3D printed horrors

I think is one of the best video build guides I have found :-

 Creality Ender-3 V2 assembly and pro build tips - UPDATED 2021! (build starts at 8:05ish)

00:00 Intro and other bits that may be useful, but not required right now, maybe later.

8:05 Base wobble check. **IMPORTANT TIP:** You need to check that the Y axis mounting bolts are tight (see the next video - Just Vlad at 8:40) and then come back to this video.

9:37 Remove glass plate and clips (more clips available on aliexpress)

10:25 Bowden Tube Couplers replacement (**you do not have to change them right now, but may be worth buying a couple, I'm still using the stock ones after 5 months**)

10:50 with Fan Shroud Removal (for people who need to know how to remove the shroud)

14:02 **re: TIP cable tie.** Your printer should have come with blue C clips instead of using the cable tie. If not see Ender3v2 Printable Parts section [Other](#) Ender3v2 Bowden Clip, when you are up and running you can swap out the cable ties if you want to.

14:48 Eccentric Nuts

16:57 Vertical Extrusions

24:30 X Axis Build

27:48 X Gantry Alignment

29:30 Z Axis Motor Install

31:10 X Axis Timing Belt and tensioner

35:03 Fitting the X Axis Assembly to the Verticals

35:55 Fitting the Z Limit Switch. **IMPORTANT TIP:** For now, make sure that the limit switch is above the bed, so that when you switch on the nozzle does not crash into your bed. It can be adjusted later to where it needs to be for printing.

37:29 Put the glass back on.....**IMPORTANT TIP:** turn the glass over so the carborundum side is down and the glass is up. That way, if you make any mistakes, you do not damage your nice new carborundum glass.

37:48 Fit LCD screen

38:11 Fit top extrusion arm

38:30 Fit the Filament Holder

38:43 Connect the wiring to the motors and switches

39:56 Power selection 115v or 230v

40:30 Switch on. **IMPORTANT - DON'T Switch ON**, you will switch on in a minute. First read on to the end of the next chapter.

My Experience

I have not done the bowden tube bit and have so far not had any problems. However, I would recommend you doing it or at least keeping it in mind if you have problems. I was thinking of printing ABS at some time in the future, so I may buy a Capricorn tube and do the "Chop" fix later (youtube video). At the moment I am running stock bowden, no adjustments, and have not had any problems.

Just Vlad

This video is useful as it mentions some useful things at the beginning regarding tightening the Y axis frame bolts.

 [Creality Ender 3 V2 - 3D Printer - Unbox & Setup](#)

Note: The problem I have with this video is, he does not actually square anything. He is, however, following the Creality build manual, if that helps you to build yours.

7:55 Chassis level check

8:40 Y axis mounting bolts **IMPORTANT**: This is missing from the 1st video (Tomb of 3D printer horrors)

9:15 Bed check

10:45 Y belt check

11:48 Following the Manual - Step 1

14:46 Step 2

16:06 Step 3

18:30 Install the X axis Belt

19:25 X Axis Tensioner

20:21 Install the X Axis Assembly

22:00 Install Top Extrusion and tighten everything up

23:45 Adjust Eccentric Nuts

24:50 Install the Display

25:40 Install the Spool Holder

27:08 Connect Bowden Tube

28:15 Connect the cables

29:30 Power Select 110v/230v

If you are following this video. **STOP watching HERE and follow the guide.**

Ricky Impey

This video is useful to show you how to adjust your Y axis eccentric nuts if the bed is not moving freely. (watch from 1min 52 secs)

 Ender 3 v2 Full Build, Assembly & Setup - All The Basics For Beginners

Recommended - Alignment Problems and Solutions Video

I thoroughly recommend you watch the whole of this video to help you with your build. Hopefully you have read through and watched the videos before you started building, but even if you have already built your printer, they are worth a watch and spending time to correct any issues you may have.

Alex Kenis

 Ender 3 alignment: problems and solutions - ensure things are aligned (I recommend starting from 3:18). This video also relates to the Ender3v2.

Printables

Recently I found this, but you need to have your printer up and running (obviously) but they may be useful if you find you have the problems again later (because, for example, you did point J in the Chapter: [Bed Levelling Issues](#)). Or you can just use a couple of CD cases.

<https://www.thingiverse.com/thing:5183028> - X gantry Level tool for Ender 3 V2

Possible Build Issues

Bed Hitting the Y motor

IMPORTANT: Make sure that your bed is high enough and not hitting the Y axis motor when you push the bed back over the motor. There is not much of a gap there when you lower the bed as low as it can go. If you need to, turn each of the four wheels by a couple of turns to make sure the bed does not hit the motor.

Z axis bracket issue

Some people have found their bracket is not at a 90 degrees right angle. If you are having a problem take a look at this. It was a problem with Ender3's production quality at one point. A Reddit User has confirmed that they have had the same problem on the Ender3v2, so it is worth keeping in mind (probably very rare).

 Creality Ender-3 Z-Axis Alignment Correction

Eccentric Nuts

With the power off, all your axes should run easily, freely and smoothly, but with no wobble. SLOWLY move by hand the bed , head and X axis gantry, forwards, backwards, up, down , side to side. If you can feel lumps and bumps as they move then your eccentric nuts are too tight and need to be adjusted.

You are looking for JUST tight enough, but with no wobble.

 [HOW TO - Fix Wobble & Adjust the Eccentric Nuts \(3DPrinter\)](#)

 [Investigating wear of POM \(Delrin\) 3d printer wheels, and if PC \(polycarbonate\) are be...](#)

Ender3v2 User Manual

 [Ender3_v2 1 June 2020.pdf](#) - PDF of the Ender3v2 User Manual 1st June 2020.

4. You have built it, yes. Don't switch on yet....

Set your Z axis limit/stop switch above the bed so that your nozzle does not crash into the bed. You can adjust it later.

Do not put any filament in the printer yet, as you will be turning the printer over again in a few minutes.

IMPORTANT TIP: When you are first setting up your printer and have not levelled your bed, here is a handy hint from a reddit user. To **save your Carborundum glass from damage**. **TURN YOUR GLASS BED OVER** and level your bed using the flat glass side. Then you will not damage the carborundum whilst learning to bed level/tram. When you have levelled/trammed your bed and are happy that you know what you are doing, just flip the glass over and do a bed level print on the carborundum side. So, **RIGHT NOW TURN YOU GLASS BED OVER SO THE GLASS IS ON THE TOP**, you can turn it back later when you are happy you can level your bed. Just make sure the bed screws are not sticking up and could scratch the carborundum glass. Note: Make sure the glass is clean and degreased.

IMPORTANT : Have your finger on the on/off switch just in case you need to switch off for any reason like, nozzle hitting the bed, funny noises, smells , smoke (hope not!!) etc. After about a week my nozzle cooling fan made a right noise when I switched it on, I think this is normal for most people as the fans are a bit poor. A tap on the fan, or switch off/on usually fixes it or it's fine when it has warmed up. So don't worry if it happens to you just look at investing in a new "quiet" fan later. Update: I took the fan shroud off and squirted a tiny bit of silicone spray into the "bearing" (I don't think they have one, just plastic), after a few prints the noise was gone. I have not had the noise for over 5 months.

BEFORE YOU TRY TO BED LEVEL OR INSTALL FILAMENT OR PRINT!!

Now you can check that the printer powers up and run through some basic tests, before you do anything else.

Basic Tests

IMPORTANT: Keep your finger on the on/off button whilst you run through the basic tests.

BASIC TESTS: Using the front display check that you can home your printer, that the X Y Z and E axis move using the display (under prepare/move). Check that the Nozzle and the Bed heat up to the correct temperatures (Control/Temperature) nozzle 210C bed 60C and then use the display to "Cooldown" (Prepare/Cooldown). Then when the temps are below 30 degrees C switch off the machine and do the following.....

5. Cabling issues

During transportation and over time, cable connectors and screw terminals can become loose due to vibration.

Once you know your machine works, I recommend switching off and unplugging your machine from the mains and unplugging any USB connector you may have plugged in. Put the machine on its side and open up the mainboard ([Service tutorial Ende 3 V2 the mainboard replacement](#)).

Fire Risk

Fit ferrules, or cut off the solder on the soldered terminals (24V in, Cables to Bed and Nozzle), see the youtube video below on the **Fire Risk** of the Ender3v2, as a solder connector in a screw terminal is not good practice.

The surface area in contact is just the top and bottom of the solder and with high current will cause heat. Either cut off the solder and twist the bare wire, making sure there are no loose strands (that could cause a short circuit) when you insert it into the screw terminal, or preferably use crimp connectors that are designed for screw terminals (ferrules).

 [Install Ferrules on Ender-3 V2 Wiring](#) - **Fire Risk** Install Ferrules on Ender-3 V2

Connector Issues

There are so many people that have random problems due to a terminal connector or screw terminal being not quite snugly fitted.

One at a time, disconnect/pull out each of your cables on the mainboard and reconnect them yourself **one at a time**. This includes any screw terminals you have not done above.

You will have to carefully cut out the hot glue using your snips/cutters. Once unplugged, plug it back in and give each individual cable going into the terminals a little tug/pull to make sure they are connected properly, carefully use a prong to push individual connectors in via the metal tab if you think they are loose. 30-60mins pre-work doing this will save you hours wondering why a print has a slight issue and you find out weeks later it was a dodgy cable problem.

A reddit user's screen stopped working after a few weeks, it was the screen connection to the mainboard, they removed the glue and plugged it back in and it now all works fine.

TIP: (Reddit User re:glue removal): Just use isopropyl alcohol on the hot glue next time. Lift one edge of the glue and capillary action will take care of the rest. Doesn't dissolve but helps.

While you have your machine on its side and the cover off, also check your earthing/grounding.

6. Earthing

House Earthing

It is important that your earthing in your house is correct. You can buy power outlet testers on Amazon/Ebay etc. that plug into your wall socket and tell you if your earth is cabled correctly.

<https://www.amazon.co.uk/s?k=power+outlet+tester> - example power tester

Printer Earthing

Ensure that you have earth continuity from the earth on the 3 pin mains input to the rest of the chassis.

You will need a multimeter or continuity tester.

Disconnect the kettle (power) lead so there is no power and remove any USB connection if you have one. There should be no power or other earths in any form to the mainboard & PSU.

Now check the continuity of the earthing by connecting between the main earth on the 3 pin mains input socket (Top pin in the uk, Round pin in the US) and a number of screws around the chassis. They should read 0 ohms.

If you do not have these readings, there are videos on youtube to help you test and correct this issue, see below.

▶ Grounding Your 3D Printer: Test and Fix It! - Video on how to measure continuity

▶ Creality Ender (and others!) safety issue - chassis ground connection - How to Fix

▶ This is why I was shocked by my 3D Printer IMPORTANT FOLLOWUP - Why you need to check chassis earthing/grounding.

Note: if you decide to remove the black case around the PSU and print the PSU cover (<https://www.thingiverse.com/thing:4774312>) then you may have an earth problem that needs correcting. My two screws that hold the new plastic PSU cover on had spring washers that bit into the metal case to earth the mainboard. With a plastic cover the mainboard was no longer earthed. The easy fix for me was to move the washers to the other two screws at the top of the power supply so that they cut into the chassis and provided the earth again. I did each screw separately one at a time (only one in, each time I checked) so that I knew both the screws earthed the mainboard correctly..

Note / WARNING : If you check the continuity to the SD card metal plate cover on the mainboard using a multimeter this will read around 0.3ohms. This is because the PSUs 0v line does not connect directly to earth i.e. 0V is a floating earth. Do not try to earth the mainboard / 0v line.

7. Check Thermal Runaway software protection

These videos will show you how to test your printer to ensure the firmware is running thermal runaway protection and that it is working correctly.

 [How to make sure your 3D printer won't catch fire!](#)

 [How To Test Thermal Runaway Protection...Safely!](#)

8. Creality Carborundum Glass Plate

Cleaning The Glass

From new, and before you try to print, remove the plastic cover and just clean with dish soap and hot water using a washing up sponge/plastic scourer, give it a good scrub with the plastic scourer side, lots of bubbles and water, rinse with clean hot water. This will remove the “sticky film” left over from the plastic cover that you removed, this film can cause poor adhesion. I dry the glass with a lint free tea towel. If any yellow residue is seen on the drying towel, rewash the bed until no residue is noted on the drying towel.

Once clean, try not to touch the glass face with your fingers

If you get your level correct this is all you should need to do every other day or so. I spent ages trying to level my bed because I thought it was out and it just needed a good clean. Squirt with water and wipe with a cloth after each print.

The Ender 3v2 Instruction manual (v1.5) (Page 1 point 9):

Use glass cleaner or isopropyl alcohol to clean the print surface before every print for consistent results.

Reddit User: Wiping the bed with Isopropyl Alcohol (IPA), using a microfiber cloth, can aid in adhesion as the wiping will remove any oils for the hands or fingers inadvertently left on the bed.

WARNING: Some people on the internet have said that IPA has ruined their carborundum glass, how true this is I don't know. It could be people trying to sell you a different product, it could be a defective batch of glass beds who knows. Just be aware that Creality will not replace your glass under warranty.

I've used IPA a couple of times, but it made no real difference to me than using water and cleaning my bed properly. You may want to try it, you may not.

How to get rid of the print “image” on the bed.

You can't.

You will still see the image of the print after you take the print off the bed, but your prints should stick. Don't expect to be able to get a clean looking glass like it was new or near new, I always see the image of the print and have no problems printing. My OCD for a clean glass disappeared after a few days of not being able to clean it as I was able to print OK.

Prints are just not sticking

If your print will just not stick and you get spaghetti and you want to test that it is not the glass causing an issue you could use a glue stick (“Pritt Stick”) and put some glue on the bed where you are going to print. Just a light amount. If the print sticks then it is likely you have not cleaned your carborundum glass properly. You could then stick with using a glue stick or clean your glass properly if you don't want the hassle of using a glue stick or hair spray. Glue

stick is water soluble so it will wash off, just soak the glass for an hour in hot water if you want.

How to make prints stick video

 PSA: How to make prints stick on a new Creality glass bed. Fix adhesion on a new Ende...

IMPORTANT: (info from a reddit user) Poor adhesion is a symptom of poor leveling, offset, or both. Sometimes temperature as well. Gluestick, hairspray, magicgoo etc may mask the symptoms, but they're not a fix. You don't need them with PLA on an e3v2 with carborundum glass, they're messy and unnecessary.

Acetone

WARNING: Some people have stated that they have removed the textured surface of their glass beds using Acetone. Do not try this unless you're prepared to have no texture, or to replace your glass.

If you really have a problem you could try acetone (proper nail varnish remover that says acetone, you should really use pure acetone but if it is just a small section...) to clean the bit of the bed that has too much PLA (the acetone sort of just spreads it out across the plate I find) and then fully wash the glass with soap and hot water as above. Then you should be able to print over it.

Some people have said that they have soaked their glass in industrial acetone for 20-30mins. I have not had to try this myself.

I have two or three pages on official creality web sites (I saved the pages as pdf's for reference) that sell the carborundum glass plate that state that it can be cleaned with acetone.

So there is lots of confusion around if you can, or cannot, use acetone. I would use it as a last resort only, and be prepared to either buy a new plate or use glue stick if need be.

Maybe you're lucky, maybe you're not. At least now you've been warned!

I can't get my print off the glass.

Put the glass in the freezer if the print does not come off. Do not try to force it or use tools to remove it. Leave it 10-20 mins and it should pull off. (Update : 12th May 2024) I just went to remove the glass for the freeze and I picked it up with my thumb and finger. It exploded in my hand, I have glass all over the freezer. I think I left it in there for about 40 minutes, but it is also 2 years old and is a "consumable". Just be warned this can happen (thermal shock I guess). Scarily, I bought my G10 sheet only last week.....

If that does not work put the glass back on the printer and heat to around 60-70C and you should be able to carefully pry it off. The print may now however, have a warped/uneven bottom.

9. G10 possibly a better option in 2024

 G10 - The best print surface you've never heard of.

G10 PLA not sticking (especially if new)

Reddit User : *I finally got mine to work by cleaning it with hot soapy water and scrubbing it with a new green scrubbing pad. I used new because I didn't want any chance of oil contamination and they are really 'sharp' when they are just out of the wrapper.*

Reddit User: *This right here, can't upvote enough. I just did this with a new sponge with a green scrub bit and I put some elbow grease into the scrubbing. I can't see any noticeable difference really besides some light scuffing that doesn't show on the prints but all of a sudden prints just started sticking even at 55c. Thanks for the advice!*

Other advice is to up your bed temperatures by 5 to 10 degrees C.

10. If you have bought a probe

If you have bought a probe with your printer, now is the time to head over to the following two chapters

Chapter : [Probe Installation](#)

Chapter : [Physically Installed Your Probe, Don't do anything yet.....](#)

And then return here to continue your build.

11. Ready to Bed level (really you mean “Tramming”)

IMPORTANT TIP (Deliberate Repeated TIP): When you are first setting up your printer and have not levelled your bed, here is a handy hint from a reddit user. To **save your Carborundum glass from damage**. **TURN YOUR GLASS BED OVER** and level your bed using the flat glass side. Then you will not damage the carborundum whilst learning to bed level/tram. When you have levelled/trammed your bed and are happy that you know what you are doing, just flip the glass over and do a bed level print on the carborundum side. So, **RIGHT NOW TURN YOU GLASS BED OVER SO THE GLASS IS ON THE TOP**, you can turn it back later when you are happy you can level your bed. Note: Make sure the glass is clean and degreased. Just make sure the bed screws are not sticking up and could scratch the carborundum glass. Note: Make sure the glass is clean and degreased.

IMPORTANT: It is important that you learn to level, and set your z-offset.

IMPORTANT: Before you start. You have moved your printer, you have turned it over, you need to recheck your X axis is level with your base. Get your CD's back out and re-check the X-axis before you start trying to level, if it is no longer level, you may want to re-tighten the screws that you adjusted to level the X axis in Chapter [“Build your Ender”](#). Whenever you move your printer you may need to re-level, if you are lucky and have squared and tightened everything correctly, you won't.

IMPORTANT: Make sure your printer is on a level surface.

IMPORTANT: Re-run the [Basic Tests](#) as you have unplugged and plugged everything back in.

IMPORTANT: If you do not have a probe, sSet your Z axis limit switch as per the Creality instructions - follow some videos if need be. BUT also.....

IMPORTANT: Make sure that your bed is high enough and not hitting the Y axis motor when you push the bed back over the motor. There is not much of a gap there.

IMPORTANT: Keep your finger on the on/off button whilst you press the home button in case the nozzle crashes into the bed.

Filament Loading

WARNING : When you handle filament, never lose control of the free end of the filament, otherwise you will end up with a tangled spool. The free end should be in your hand, in the extruder (filament feeder), or clipped to the side of the spool

WARNING : If I don't use my printer for a day or so I remove the filament as I have learnt that the extruder squashes the filament out of shape over time sitting in the same place and then it gets stuck in the bowden tube when you try to print.

Cut the end off of your filament at a 45degree angle to help with loading the filament. Then watch the video.

How to load your filament the easy way

 Why you probably hate the Ender's extruder, and how you can learn to load it in 4 sec...

Filament Unloading

I just heat the nozzle to around 210C (for PLA) using the display screen. When it's up to temperature, I just squeeze the extruder arm a little, but enough, to pull the filament out by hand, rolling the filament roll so that it rolls back onto the roll. I then slip the end through the holes on the side of my filament roll for storage (I used to cut the end to 45degrees with the cutters, but as the holes in the reel damage the filament, I no longer see the point). Then press "cooldown" on the display, wait for the nozzle to reach about 80C and then you can switch off the machine.

Temperatures (for PLA)

For PLA you should start with a bed temperature of 60C and a hotend temperature of 200C. The hotend temperature can be tuned later (Chapter : [Basic Calibration](#)). Also you may want to change the Bed temperature later if you are having problems, but leave it at 60C for now.

Manual - Bed Tramming/Levelling using the wheels

WARNING: DO NOT under any circumstances expect to push your nozzle around by hand and end up with a 'level' bed. Use a prepared gcode file (cheps bed levelling) or some other firmware that has manual levelling tools (e.g. mriscoc Bed Tramming Menu) that move the nozzle around the bed for you.

Read through this chapter and then follow some videos on bed levelling using the piece of paper method.

How to properly level your bed.

 How to PROPERLY level your 3D printer bed - with explanations, diagrams and evidence.

Worth watching to understand what happens when you adjust your bed.

 How to level your bed on ender 3 v2 / other 3d printers - easy process-manual on the ...

Cheps Bed Levelling Video

 Creality Ender 3- Easy Way To Level Your Bed

<https://www.thingiverse.com/thing:3235018>

Marlin Bed Levelling Guide

I've not watched this but here's a link I found that a reddit user posted.

 Complete Marlin Leveling Guide

I have added some more notes further down the guide in "[More info on how to bed level](#)". At some point I will try to tidy this section up a little.

Note: Some gcode that moves your print head automatically to the probe points may, or maynot, compensate for the thickness of paper. Before upping the bed by the 0.1ish thickness of your paper, start your test print and keep your finger on the on/off button in case the nozzle hits the bed.

Recommendation : Install mriscoc MM (manual mesh) firmware (see chapter : [Firmware](#)) as it has the "bed tramming" wizard, which will help with your bed levelling.

 [Manual Mesh Bed Levelling - Free warped bed solution](#)

Paper Thickness and Wheel Movement

To clarify, as there are questions all over the internet about this. When you use the paper method, your paper is usually 0.08-0.1mm thick. If it is just touching the nozzle you will still need to lift the bed by 0.08-0.1 so that the nozzle touches the bed when you print. When the Slicer software slices your stl file and the printer goes to print, the software will tell the printer to move to $z=0.2$ (if you have chosen to use 0.2 as your first layer height). So if you do not set the bed so that the nozzle is at 0 but leave it at the thickness of the paper then the nozzle will be at $Z=0.02+0.08\text{ to }0.1$ (the thickness of the paper) e.g. 0.3mm ish. However this is something to bear in mind when your print is not sticking to the bed. It is probably best for your first attempt at bed levelling to go with the thickness of the paper and adjust the bed up very slowly as you print, otherwise you run the risk of the nozzle hitting the bed and scratching it quite badly if you have tried to put the nozzle at $Z=0$.

Wheel Turns

How big a turn of the wheel to turn 0.08-0.1.

I read that the screw pitch of the bed screws is 0.7mm, so one full turn should be 0.7mm. So expect to turn the wheels of the bed in the "bed up" direction by about a 10th of a turn per wheel (e.g. this would be 0.07mm) if you have used the paper method, so not much really. I think this may be where people go drastically wrong, not knowing how much the wheels should be turned.

You should be able to level your bed and print with stock springs. The issue with the stock springs is that they are weak and may potentially move over time and you have to re-level the bed.

Recommendation : As soon as you are able to print, print the clicky wheels in chapter : [Change stock springs and wheels](#). One click is 0.01mm approximately, great when tramming your bed.

Probe - Bed Tramming/Levelling with a Probe

See Chapters : “Probe Installation” onwards and then come back here for Bed Levelling Prints.

Set your initial Z-offset

Home your printer and find the z-offset menu.

Using a piece of paper, place the paper between the nozzle and the bed. Slowly, change the z-offset (going in the negative direction) and move the piece of paper back and forth, until the piece of paper just starts to “catch” the nozzle. It should be tight, but movable, it should not start to fold up. Write the number down on a piece of paper and save this z-offset in your EEPROM.

You are now ready to find a more precise z-offset. You now need to print some “Bed Levelling Prints” and adjust your z-offset again using the display screen.

Bed Levelling Prints (Manual & Probe)

My Recommendation

Once you have done the paper method (manual tramming) or the “bed tramming” wizard (probe tramming) then you need to do some bed level prints to get the correct “squish”.

I find it easiest to do the bed levelling using three different prints that get closer to the correct bed level.

Initial Bed Level

Start with Cheps Bed Level Print (the concentric squares).

Probe : Watch the video, then start the print. Click the tune button and adjust your z-offset (in the negative direction) instead of turning the wheels, until the print sticks using your finger as in the video. **REMEMBER** : A little at a time -0.01mm at a time.....Check, then another -0.01mm. You don't want to scratch your glass.

Once you have your z-offset where you want it, write it down and also store it in the EEPROM using the display.

Manual : Just follow the video.

 [Creality Ender 3- Easy Way To Level Your Bed](#) - Files are downloadable in the summary.

 [Perfect Z Offset On Your 3D Printer - How to get it right!](#) - This may be worth watching too.

Once you have the concentric squares sticking we can move on to getting the correct “squish”

Time to find the “Squish”

The Good and the Bad - Bed levelling diagram

As a reddit user said to someone, absorb the information in this link:

[tbvOIB9.jpeg \(4032x6815\) \(imgur.com\)](#)

The Diamond Print

I personally use the below print after “cheps bed level print” to adjust the z height. It is a 100x100 centre square, which I have rotated 45degrees in my slicer so that it is a centre diamond. I use this because I find I cannot see the print (e.g. the corner square prints such as teaching tech below) when they are printing because I find the nozzle is in the way and I can’t see the print properly to know which way to adjust the wheels/z-offset or by how much.

<https://www.thingiverse.com/thing:2789093> – **SUGGESTION:** Rotate in your slicer to make a diamond shape.

Probe : If you are using a probe, you should then be able to change your z-offset height by 0.01 at a time and you can mark on the print using a sharpie as you do it (if white pla, or a pencil if black). You can easily get about 10-12 changes onto the print e.g. 0.00 to 0.12. You should find that your bed levelling window (between way too high and way too low) will be around 0.14 on a 0.2mm layer height print.

Manual : If you are manually using the wheels, I suggest adjusting all four wheels by the same amount when running this print. The next print (Teaching Tech) will allow you to correct each corner one at a time.

Teaching Tech

[Teaching Tech 3D Printer Calibration \(teachingtechyt.github.io\)](#)

After the diamond print, I then use teaching tech first layer print and do final small corrections on the corners.

Manual : There are other bed level prints on the internet that do corners etc. to fine tune each corner wheel if required when manually bed levelling.

Probe : If you are using a probe, the corners should now be pretty much correct. If they are not, then it will be worth you looking at creating a mesh using your display, saving it to EEPROM and selecting “restore saved ABL” in the Auto Bed Levelling section of the form (or

adding the code to your slicer to load the mesh, if not using the teaching tech website print). Then run the print again and it should be good. If not, check out chapter: [Probe Issues](#).

12. Bed Levelling Issues

If your first layer is still not sticking try these things:

a) Use PLA to level the bed. I have read that it is the easiest to print. No good starting with the hard stuff e.g. ABS.

b) Clean the glass, give it a really good scrub.

c) Up the bed temperature to 70 (for PLA), if you measure using a temperature probe you will find the glass will be around 60 now (with the bed at 60 the glass is around 50).

d) Up the nozzle temperature by 5 then possibly 10 degrees depending on the filament.

All filament prints better at different temperatures, it is not one fits all. Even different reels of the same make/model can have different optimum print temperatures. That's why there are calibration test prints for filaments (but you can look at those later, after you have your printer working).

e) Can you hear high pitched pops and clicks at the nozzle. You may have wet filament (see Wet Filament below)

f) See Chapter: [Spinning Bed Screws](#)

g) Check your eccentric nuts. [HOW TO - Fix Wobble & Adjust the Eccentric Nuts \(3DPrinter\) - YouTube](#). They should be not loose and not tight. Switch off the printer and move everything by hand slowly (nozzle and bed), it should run smooth, it should not feel lumpy or bumpy and should not be wobbling about.

h) Check your belt tension - not too tight and not too loose.

i) You can check with a ruler (or a good flat edge) and a torch to see if the bed/glass is warped.

j) Oh... sorry forgot to say (or have I already said it...remember). **Do not move the X axis by hand**. If you have to move it by hand, hold the X axis motor and move it. The X axis is a cantilever arm. If you push up/down on the right hand side (when looking at it) and you have not tightened the nuts holding the cantilever enough (or even if you have) you will change the bed levelling you have done and will need to a) correct the X axis level with the base of the printer again and b) re-level the bed again. I know it's difficult when you are right handed not to just hit disable motors and then grab the cantilever arm on the right. **BUT DON'T DO IT !!!** you will be constantly bed levelling..

k) If you think your Z stop switch is faulty e.g. you do not get the same homing point every time you home, you can change it with the X axis switch test if that is the problem.

l) Is your printer on a flat, level, sturdy surface, i.e. not on a desk with wheels. Make sure the table does not shake as this can cause issues.

If your prints are sticking great..... read on, to save the headaches later.

Stress Saver!! : When printing a normal print (not a bed levelling print), if the first layer looks like it is sticking but not sticking well, let the printer try to print 2 or 3 layers before you cancel the print (unless you have spaghetti, in which case cancel the print) as quite often, even though the first layer looks a mess, the print will "fix" itself enough that you can live with it.

13. Change stock springs and wheels

Why Change the Springs and the Wheels

You will probably find once you have levelled your bed and are able to print, after a while you have to level it again (if you are on stock springs) or catch the wheels (it only needs to move a fraction), or you move the printer. Unless you change your springs and use clicky wheels (or similar) you will find that you will be having to bed level quite often.

Bed Spring Options

- 1) Thicker Yellow (other colours available) springs
- 2) Silicone Spacers (multiple colours available)
- 3) Solid Aluminium/Steel Spacers (you will need to load Manual Mesh or Probe Mesh firmware) as you will not be able to use the wheels anymore to tram/level the bed.

Note: 18-8 Stainless Steel Unthreaded Spacer 10 mm OD, 16 mm Long, for M4 Screw Size.

Note: Make sure you get the correct length springs/spacers for your printer.

My Setup

I am, at the moment, using silicone spacers (3x18mm, 1x16mm), but I did print and fit the Spring Guides whilst I was waiting for my delivery (it can take months from the other side of the world).

Spring Guides

I used this one, there are others.

[Ender 3 Bed Spring Stabilizer by supchaka - Thingiverse](#)

It lasted around 4 months printed in PLA. It may have lasted longer if PETG. The left two guides eventually “stuck, sort of” to the bolt, probably due to the heat from the bed. I probably should have drilled the whole a little bigger too to allow more movement.

I had to “unscrew” the guide from the bolt to get it off, not a biggy but it stopped me from being able to adjust the left side bed wheels correctly. They are good for a temporary fix whilst you are waiting for your replacement springs or silicone spacers. I have now installed my silicone spacers.

Spinning Bed Screws

Whilst you have your bed screws off, make sure they don’t spin.

I put a spring washer and a nut on the bed screw to stop the screws turning. e.g. take off everything, put the screw in bed, then the washer, then the nut, then springs/spacers, attach the bed to the chassis, then fit the wheels.

How to video here: [Fix Your Creality Ender 3 Bed Level](#)

Clicky Wheels (0.01mm per click)

[Ultimate Ender 3 Bed Leveling Knob](#)

Given the thread pitch of 0.7mm for the bed adjustment screws the 70 teeth give 0.01mm per 'click' according to the comments. This is really handy for manual levelling and is a great companion to using the mriscoc "bed Trimming" menu (see Chapter: [Firmware](#)) with 0.01mm per click up or down. They also "lock" your wheels in place, so that you don't accidentally adjust your wheels when you didn't want to..

Also I found the remixed double clicker better for me.

Double Clicker - [more consistent and stronger click for Ultimate Ender 3 Bed Leveling Knob by jaimebpg - Thingiverse](#)

I printed one of each clicker mount (a single clicker and a double clicker) and checked them on one wheel first to see which I preferred, then printed the other three wheels and clickers. You may need to file down the holes with a mini file a little (nothing new in FDM though) especially in the clicker bit so that the bed screw moves up and down correctly.

This may help with bed levelling and keeping a level bed over time e.g. weeks and not hours. I personally love them for the 0.01mm click when bed trimming.

14. Filament Wheel

Personally I have removed my "Blue" filament wheel and used a sharpie "a red one" to mark the flat spot on the pin, to see when it is moving. Why you may ask. Well, I left one print running and I heard this noise and it turned out to be the wheel spinning backwards and forwards, clicking from the motion (The wheel has a brass insert which is slightly loose when fitted to the extruder). My thought was, why do I want the weight of a wheel, spinning backwards and forwards during a print making noises that could possibly be causing any sort of marks/pops etc. on the print. So I removed it.

I usually just push my filament by hand all the way to the hot end with the temp at 210C when I load it and check if I get ooze and to unload it I pull at 210C. I have never used the wheel and just think it is a weight on the extruder that is not required. If I need to move the E by a certain amount I use the front screen. I am not saying that you should remove yours, I am just giving you my opinion.

Please let me know your thoughts on this on Reddit if you think differently and why.

15. The Plastic Extruder Arm

The plastic extruder arm will break at some point, suddenly giving you pulsing under extrusion.

📺 [Fix Under Extrusion from Cracked Extruder Arm on Creality Ender 3](#) - Cheps video of the problem



Either print a better one, or upgrade to metal or direct drive BEFORE it breaks preferably. Here is one you can print. [The Extra Strong Extruder Arm](#)

Video on how to remove the arm (around the 7min mark)

📺 [How to replace extruder arm & extruder feeder | how to upgrade hardware on 3D pri...](#)

Here another interesting video.

📺 [Creality Ender 3 V2 - Will A Plastic Extruder Assembly Last? & What To Do If It Doesn't](#)

📺 [Everything about Extruder Pre-Tension!](#) CNC Extruder Pre-Tension Video

TIP : If it is broken and you need to print a new one, you can possibly glue it back together (CA, Epoxy), whilst you print a new one. Other people have used elastic bands or even held it whilst they print one.

16. Wet Filament

If your print is not sticking and you can hear pops when you are printing you may have wet filament (difficult with the noise of the fans).

Wet Filament out of the bag

People have said that their new filament in a sealed bag with silica has been shown to be wet (has soaked up moisture from the air). i.e. they have had wet filament delivered to them. Even though this is probably rare, bear it in mind if you are having problems printing.

▶ 3d Printer Filament - The Truth of How to Actually Keep it Protected And Dry

Drying Wet Filament

Before you dry your wet filament it is worth weighing the reel with the filament on, using some kitchen scales. Then why you have dried it, weigh it again and compare the weight.

TOP TIP: You can put it in your airing cupboard (a UK thing, not sure if other countries have them) overnight to dry it if your airing cupboard gets hot (30-40 degrees C). Your towels dry in there and so does your filament, no need for expensive drying equipment. Don't put wet towels in with your filament though.

Search the web/youtube for how to fix wet filament. People have made an oven with their printer bed at 40C and a cardboard box over it. Others pop it in a low oven (around 40C, my oven will not go that low) or use dehumidifiers and printed boxes on Thingiverse.

▶ Dry your PLA and Nylon 3d printer filaments the easy and cheap way. Build a chamber...

▶ Amazingly Simple Free Filament Dryer -You already have one!

WARNING !!: Check the web for drying temperatures etc. of your filament before doing this as you could end up with a big ball of plastic if you get it too hot. Material

Info below form : <https://www.3devo.com/blog/3-ways-to-dry-filament>

	Temperature	Time in Oven
PC	80°C-90°C	7-10 hours
ASA	80°C-85°C	4-6 hours
Nylon	75°C-90°C	4-6 hours
ABS	65°C-75°C	4-6 hours
PETG	60°C-65°C	4-6 hours
PLA	40°C-50°C	4-6 hours
TPU	40°C-45°C	4-5 hours

Dry box

▶ Ultimate Filament Storage Dry Box - Print while keeping your filament dry

17. Firmware

The firmware from Creality is, to say the least, very poor and full of bugs. It is recommended to change your firmware as soon as you can when you are happy your printer is working mechanically well.

IMPORTANT: You need to ensure that you have the correct firmware for your device. You need to check the version number of your mainboard as describe in chapter: [Make a note of everything – Take Photos](#) e.g. V.4.2.2 and also your processor type, probably either a STM32F103 RCT6/RET6 or a GD32F303 RCT6/RET6.

If you have a probe

BLtouch, CRTouch or a 3Dtouch (clone)

When looking for firmware, if you have a BLtouch, CRTouch or a 3Dtouch (clone), you will usually need the version of firmware for the BLtouch. (it is worth checking the firmware site's wiki/readme to confirm that the BLtouch version will work with the CRtouch).

The firmware filename will usually have “blt” in it.

For example:

If you have a V4.2.2 mainboard and a probe you would need at the time of writing this version of mriscoc professional firmware:

[Ender3V2-422-BLT-20220716.bin](#)

Machine name - Mainboard Version - BLTouch Version - Release Date . bin

Other Probes

If you have a different probe, then you may have to compile marlin yourself. Sorry, but this is not covered in the guide.

Which Firmware

As of writing all the firmware below is FREE.

I recommend Mriscoc Professional Software. It has a “bed Tramming” wizard in the menus that will help with bed tramming/levelling.

There is other firmware available such as Jyers, Marlin, TH3D, Klipper.

Note: As of 1st August 2022, Jyers release has not been updated since Nov 2021 (marlin 2.0.1) , I believe a fork of Jyers is in pre-production.

Mriscoc is regularly updated at the moment (July 2022 update).

Please NOTE: mriscoc runs in High Speed (HS) mode, which makes the initial homing noisy. You can turn off high speed mode if you require, see mriscoc github for details (search google for “mriscoc ADAPTIVE_STEP_SMOOTHING”)

Marlin Based Firmware

Links tested July 2022.

<https://github.com/mriscoc/Ender3V2S1/releases> - For Ender3v2 or S1 (**RECOMMENDED**)

<https://github.com/Jyers/Marlin/releases> - For the Ender3v2 (last updated Aug 2021)

<https://marlinfw.org/> - Non-compiled firmware. You will need to compile this yourself.

[Marlin Firmware Builds – 3dprintscape.com](#) - Prebuilt Marlin for Creality Printers

<https://www.th3dstudio.com/2020/09/10/unified-2-firmware-is-live-ender-3-v2-v4-2-2-board-support/> - Also firmware for other Creality Printers on this site

[Ender 3 Pro Firmware Install Guide - Update Firmware with MicroSD Card – Shiny Upgrades](#) - For Ender 3 / Ender 3 Pro

[Marlin Firmware Service \(crc.id.au\)](#) - pre built firmware for Creality Printers. You need to register. Also from a Reddit User: You need to activate your membership through the donation page. A donation amount of 0, while not ideal, is acceptable.

(reddit user: for the Ender 3 Pro - use Ender 3 as there are no real impacting firmware differences to Ender 3, another reddit user: Correct. The only difference in the firmware is the logo on the status screen.)

Note: There is a known bug in the bed levelling procedure in Jyers and possibly all marlin based firmware - It seems to roughly accumulate the probe deviation each time it probes in the same position. So repeated probes on the same point will eventually put the nozzle in the bed. Make sure you move your probe position each time you probe the bed when tramming. E.g. front left then front right, back left then back right. Not front right, front right, front right.

Non-Marlin Based Firmware

<https://www.klipper3d.org/>

Creality Firmware (Marlin Based)

Creality Firmware - Not recommended for the Ender3v2, lots of known bugs and Mriscoc and Jyers have much better interfaces.

[3D Printer Software& Firmware Download - Creality 3D](#) - New Creality Download Web Site

[Creality Cloud - All-in-One 3D Printing Platform](#) - New Creality Cloud Download Site

[Download | Creality 3D](#) - Older Creality Firmware Website

Firmware Upgrade

VERY IMPORTANT : You should not need to do a firmware update of the LCD display for the Mriscoc Professional Software or Jyers unless you want fancy fonts. I would look at that later when you can actually use the printer and get good results if you want it.

Check Hardware

You need to ensure that you have the correct firmware for your device. You need to check the version number on the mainboard see chapter : [Make a note of everything – Take Photos](#) e.g. V.4.2.2 and also your processor type, probably either a STM32F103 RCT6/RET6 or a GD32F303 RCT6/RET6.

If you have the GD processor then Creality has different software for this processor and you will need the GD software version. Also if you have a probe you will need a Bltouch version (blt). There is sometimes a version for a filament runout switch if you fit one too.

Save EEPROM settings (optional)

If your printer is new and you have not really saved any EEPROM settings you don't need to do this.

Before changing any firmware, if you can, use a terminal (Prontepoint, Octoprint) and do a M503 and save the information in a .txt file on your computer. This has your printers saved EEPROM details in it and you can reference any changes you saved to the EEPROM data later if required. E.g. Steps per minute, Feedrates. Especially if you have changed settings and saved them to EEPROM. If you have calibrated your printer with callipers etc. you really don't want to have to do it again because you did not save the information anywhere.

Your Original Firmware

Before updating any firmware, press the info button on the display and take a photo of the software you have. You need to check your board version number and processor (see chapter : [Make a note of everything – Take Photos](#)) to get the correct software for your machine.

Download a copy of the firmware you are running now, if you can. If you can't, don't worry about it.

See here for links to the Creality firmware [Creality Firmware \(Marlin Based\)](#)

Email a Copy

Always have a copy of the original firmware/working firmware you used so that if you have issues you can go back to it to check if it is a firmware issue and not your hardware. Email yourself a copy of the firmware so that you can find it a year later if needed. Also download both the non-bltouch and the bltouch versions, at least you will have them if and when you upgrade to a probe. You never know when a company will go out of business and their website disappears.

Installing Firmware using the SD Card

SD Card Requirements

IMPORTANT: For firmware installs, the SD card must be a maximum 8Gb, formatted to Windows 10, FAT 32 standard allocation size 4096. It is best to use the supplied 8Gb card that came with the machine.

Whilst the Ender 3 supports cards up to 64Gb (with 8Gb partition) then 3v2 may only support 32Gb. Most people have problems with large cards. If you are having problems with a larger than 8Gb card I would try a smaller one first. See link below for further details.

<https://3dprintingwiz.com/ender-3-not-reading-sd-card/>

Reddit user: You forgot the vital MBR step, and max partition size of 8Gb for these machines.

If you are having issues, Windows formatting could be to blame. You can always try this software that was linked by a user on reddit.

<https://www.sdcard.org/downloads/formatter/> - Windows/Mac Software to format correctly

NOTE: You may wish to do a full long format (instead of a quick format) on the SD card as this has helped users who have had issues with installation in the past.

Filename Requirements

IMPORTANT: For the **Ender3v2**, the filename for the .bin file needs to be unique. You could use the time firmware080856.bin i.e. firmwareHHMMSS.bin. For the **Ender3**, I have read that the filename must be firmware.bin and be the only file on the SD card (I don't know if this is correct or not). *I have a txt document with the names I used for the files and the version of the firmware to keep track.*

IMPORTANT: For the **Ender3v2**, if you try to update the printer with the same named firmware e.g. reinstall over itself, it will do nothing. For the **Ender3**, I don't know, but if someone could let me know I will update the guide.

Not firmware but...

SD Card issue: (reddit user) The name of the gcode file needs to be less than 20 characters or else weird things start to happen with the print speed (maybe other things including the display).

What I do

Personally I use the supplied card as it must be 8GB only. I have backed up the card and then formatted it using Windows 10, FAT 32 standard allocation size 4096. The only file on the card when updating the firmware should be the .bin file you want to install (although on my ender3v2 I have successfully installed a .bin file with a .oldbin on the card too useful for swapping between two firmwares if needed and if it works for you).

Firmware flash official video

WARNING: Stop watching at 2mins 35secs. **DO NOT** try to update the screen.

 Service tutorial Ender - 3 V2 flash firmware - Creality firmware install

 Ender 3 v2 Firmware update - step by step guide - another video showing how to install the firmware. **DO NOT** install the screen firmware (see VERY IMPORTANT at the beginning of this chapter).

NOTE: The printer can sometimes look like it has hung when you install firmware (depends on the firmware). Give it a 30 seconds and then switch off the printer, pull out the card and see what boots up. If it is not the new firmware, just try again (format the card). The printer upgrade can be temperamental. If you are not sure if it has been updated, install a different firmware, check the version on the screen and then install the firmware you want and check the version on the screen.

Screen Firmware Install & Reverting To Stock

VERY IMPORTANT: You should not need to update the screen display for the Mriscoc Professional Software, Jyers or Marlin unless you want fancy fonts etc. But I would look at that later when you can actually use the printer and get good results if you really want it.

Which Screen Do I Have?

The Ender3v2 can come with one of two different screens, a DWIN or a DACAI. To check which you have, follow this link.

<https://www.th3dstudio.com/hc/guides/troubleshooting/creality-printers-dacai-and-dwin-color-lcds/> - Which screen do you have, DWIN or DACAI?

[Identifying Ender 3 V2 screen versions - Frequently Asked Questions - Marlin Firmware Service - Forum \(crc.id.au\)](#) - There are now 3 types of screen according to this site.

[How to update the display · mriscoc/Ender3V2S1 Wiki · GitHub](#)

DACAI Screen Firmware

If you have the DACAI Screen then you can get the stock screen firmware here

Update 21/05/2022 and tested by a reddit user (a big thank you to you know who you are) : [DACAI screen stock firmware : ender3v2 \(reddit.com\)](#)

You need to put the private folder on your SD card and put it in the screen USB slot and reboot.

DWIN Screen Firmware

If you have a DWIN screen then you can get the stock screen firmware from the creality website.

https://www.creality.com/pages/download-ender-3-v2?spm=..page_1934481.products_display_1.1&spm_prev=..index.header_1.1 (web site may change, just find the link to support, downloads and look for the zip file that looks like this)

download this zip file : Ender-3 v2 4.2.2mainboard (32bit) 28 Jan. 2021 (date may change)

open zip and find folder : Marlin2.0.1 original firmware (mainboard+screen)

You need to put the DWIN_SET folder on your SD card and put it in the screen USB slot and reboot.

SYNWIT/ VIEWS Screen Firmware

Sorry, but I do not have a link to this firmware yet. If someone gets it could they post the info on the Guide's reddit page please - Thanks.

Update 15/04/2024 also see this page.

<https://github.com/mriscoc/Ender3V2S1/tree/Ender3V2S1-Released/display%20assets>

Updating the Screen firmware

To update the screen firmware you have to remove the back cover of the Display unit and insert the SD card with the screen firmware on it into the USB slot that is in the back of the

screen. e.g. the private directory for DACAI (or if a DWIN screen the DWIN_SET directory), and reboot.

DWIN Screen needs the DWIN_SET directory on the root of your SD Card.

DACAI Screen needs the private directory on the root of your SD Card.

[Firmware flash official video](#) - second half shows how to update the screen firmware

18. Possibly add a probe

A probe will not “Automatically Level” your bed. The probe helps with your print sheet not being flat. You can check to see if your print sheet is flat by putting a rule or flat bar across the bed and using a flashlight to see if you can see any light coming through.

You may not need a probe. Don't rush to get one if you are having levelling/tramming issues, work out what is wrong, it could just be a dirty glass sheet and try new firmware e.g. mriscoc or jyers, to help with tramming.

(Comment from another reddit user)

BLTouch / CRTouch do NOT auto level your bed. They can't physically move the screws - you still need to do that. They can compensate for a degree of non-level, but that's really not what they're for, and the result is a non-level, skewed bottom face on your print. (which you may not care about..)

19. Which Probe - BL/CR touch/3d Touch (Clone) or BFPTouch

There are a number of probes available, here are some of them:-

CR Touch, BL Touch, 3D Touch, Geeetech 3D Touch (claims 0.005 range), BFP Touch, Allen Key Probe.

The clones are not as precise, usually approximately 0.02 standard deviation. The BLtouch I think has a deviation of 0.0012 or something like that. As noted in "[Further info regarding bed levelling/tramming](#)" trying to get to the 0.01 level is just not really required for general home 3D printing and with a levelling window of around 0.14 for 0.2mm (check your first layer test diamond) , so you should be able to get by with a clone.

I have a clone (3Dtouch) and it was not working for me, after a long fault finding mission I have found a solution to my issue that I don't think has been documented before (at least I did not find it on the internet), I will add the information to this document later. The problem was, if I did a 3x bed mesh in a row then the results are completely different, sometimes 0.05-0.09 or more out. The standard deviation came back around 0.002-0.003 when I tested using M48, but the range was quite high. This has caused me issues with prints sticking at 0.16mm and falling off 10mins into a print, so I have reverted back to manual bed levelling with no mesh at the moment (I have a carborundum glass sheet). But there are people who say clone probes are great. If you are thinking of printing at 0.12 or with smaller nozzles, then I would guess a BLtouch would definitely be a preference as they state they are more accurate.

When I find the issue I will let you know, I did get it to go to R=0.004 and SD=0.001179 but only for a short while, so I am still investigating.

IMPORTANT: If your probe works, great. If not send it back and get a refund straight away, some are good, some are bad. Also make sure you get one with 1.5 to 2 Metres of cable as the cable run on printers is further than you think (or use a piece of string to check the length you need before you purchase).

Homemade Probes

If you want to try making a probe yourself, here is a link to some information on the BFPTouch (I have not read this myself) and the Allen Key Probe.

BFPTouch on Ender 3V2

[Guide for BFPTouch on Ender 3 v2 : ender3v2 \(reddit.com\)](#)

Allen Key Probe

For the Ender3v2 - even though I did the remix for this, I have not tested it as I picked up a cheap 3Dtouch (possibly a mistake!!) before I got around to trying the allen key probe out. Here it is if you want to have a go yourself.

<https://www.thingiverse.com/thing:5371656>

20. Probe Installation

Firmware Installation

You will need to change your firmware when you install a probe.

See Chapter : [Firmware](#)

And then return to this page.

Physical Installation

WARNING : Make sure the order/colour codes of the wires of your probe are correct before doing anything else, see wiring diagram in [3dTouch Setup](#) as an example. You need to check the documentation that came with your probe. If the wiring is wrong you could blow your mainboard.

Useful Videos

 [How to add BL Touch to Creality Ender 3 v2](#)

 [How to Install a CR Touch on a Ender 3v2 3D Printer \(Step by Step for Beginners\)](#)

Useful Links

Here is a link on how to install a probe on the Ender3v2. I have not used this page myself.

<https://www.crosslink.io/2021/04/26/ender-3-v2-bltouch-upgrade-creality-4-2-2-or-4-2-7-mainboard/>

WARNING: When you first press home on your printer, after installing the probe, make sure you have your finger on the on/off button just in case you have issues. Like the latest creality bltouch firmware that crashes the head into the right hand gantry and does not stop trying to do so, so you have to switch off and install mriscoc (or some other firmware) instead.

Probe Motherboard Connector

If you have a probe that does not have a good connection to the motherboard and wish to change the 2 connectors for a 5 pin connector then you require a **JST-XH Male Connector 5 pin**.

These are printable if you can print ok. Then use a prong to carefully take the pins out of the two connectors and plug them into the printed jst-xh plug, make sure you get them in the correct order. See below for link.

JST-XH Connectors

see PART 2 of the Guide Chapter : JST-XH Connectors & Tools for printable connectors and tools for jst-xh connectors.

Nozzle to Probe XY offset

(Reddit User) *You'll want to measure it yourself. There's slight variations between every machine, it may also depend precisely how you mounted it and how tight your screws are. That said, I've got mine at -46 and -6 (If I remember correctly). I've seen other people use -45 and -5...so, it's probably best to use your own measurements.*

There is usually a Probe Setting menu in the firmware that allows you to input your XYZ probe offsets.

How to Measure your X and Y Probe Offsets the easy way

- 1) First either take a piece of Blu Tack and flatten it into a flat coin size disc (Blu Tack Coin) OR tape a piece of paper to the centre of your print bed.
- 2) Using the control panel (go to Prepare->Z Probe Wizard->Probe Z Offset) ensure that the Probe Z Offset is set to 0.
- 3) Now autohome, but be ready to switch the machine off if the hot end goes into the bed, if this happens you have not mounted your probe correctly and you need to investigate the issue.
- 4) If all is good and the printer has homed correctly, make a note of the current X and Y positions on the display. Mine were X=156.3 and Y=123.8
- 5) Blu Tack Method - Using the control panel move Z up so you can place the flattened piece of Blu Tack on the bed under the hot end. Now move the Z axis down and gently lower the nozzle into the blu tack to create a depression
- 6) Paper Method Using the control panel, move the Z axis up so that you can put a dot (using a pen/pencil) where the nozzle is.
- 7) Now again using the display, move the X and Y axis so that the probe pin is directly over the depression/dot. If you have a terminal connection, you could use M401 and M402 to deploy/stow the probe if it helps BUT make sure that your Z axis is high enough that you don't drag the probe pin across the bed/blu tack as it will damage the probe. My probe lights up and I used the light beam.
- 8) Make a note of the new X and Y positions on the display (Mine were X=198.2 Y=132.6). Subtract these from the original positions noted in point 4. This will give you your X and Y probe offsets that then need to be configured in the printer. It is most likely that both of these figures will have a negative value for a standard probe on the Ender3v2 (probe being on the left and of the nozzle and slightly forward of the nozzle).
- 9) Using the Control Panel go to Advance->Probe Settings and set the "Probe X offset" and "Probe Y offset" to your values in point 8. Mine were Probe offset X= -41.9 Y= -8.8.

Nozzle to Probe Z offset

- 1) Auto Home and set the printer to preheat both the bed and the nozzle.
- 2) Using the control panel go to Prepare->Z Probe Wizard.
- 3) Ensure that the Probe Z offset is set to 0.
- 4) Using the control panel go to Prepare->Move Axis.
- 5) First move the nozzle to where the home point (the probe) was. To do this you need to subtract the probe offsets from the homing XY. e.g. So in my setup, my homing X=156.3 and Y=123.8 I need to take off my Probe offsets so $X=156.9-41.9 = 115$ $Y=123.8-8.8 = 115$. So I "Move X" to 115 and "Move Y" to 115 on my printer. Do the same for your printer with your settings/calculation. This puts the nozzle where the probe homed the printer. Note: It should be X=115 Y=115 as that is half the printer bed size 230x230. If yours is not 115 and 115 then you may not have measured your Probe XY Offsets correctly - may be worth a recheck.
- 6) Once the X and Y have been set, slowly "Move Z" down to 0. You should not be able to go below 0 and the hot-end should still be in the air and not touching the bed.
- 7) Using the control panel go back to Prepare->Z Probe Wizard.
- 8) Get yourself a piece of paper e.g. a sticky note.
- 9) Place the paper under the nozzle, you are going to slide the paper back and forth whilst you slowly lower the nozzle towards the bed using the "Probe Z offset" setting.
- 10) With the paper under the nozzle go to "Probe Z offset" and start to lower the nozzle, the number will become a minus figure. You are aiming for the paper to be able to move under the nozzle with a slight scratching type sound as the nozzle scrapes the paper. If you cannot move the paper then you have lowered to nozzle too far and need to move it back up slightly. My "Probe Z offset" was -2.25. Yours will be different depending on your bed wheel adjustments.
- 11) Using the Control Panel go to Advanced-> Store Settings and store your settings.

 [ABL offset guide including new probe Z offset wizard](#) - video on XYZ probe offsets, a bit in depth but worth a watch to understand why you are doing a XYZ offset.

XYZ Offset Table Example - My Printer (DO NOT USE Mine, calculate your own)

	Homed	Blu Tack depression point	Probe Offset	Probe Point When Homed
X	156.9	198.2	-41.9	115
Y	123.8	132.6	-8.8	115
Z	7.0		-2.25	

After this you will be using the Prepare->Bed Trimming Wizard to adjust the bed wheels to ensure the bed is trammed. Then you can go ahead and create a mesh. However, before doing this it will be worth continuing through this manual.

21. Physically Installed Your Probe, Don't do anything yet.....

PLEASE READ THIS: A probe does not level your bed, it is not an automatic bed leveller. If you have fitted a probe hoping for this, forget the idea. Go back to basics, do not try to create a mesh yet. Feel free to use the probe to home the printer, but you need to learn to Tram/Level your bed, ensure everything is square etc. Then once your printer is trammed, you can then look at creating a mesh. A probe is used to help with unevenness of the print surface e.g. warped glass or undulating metal sheet. It is not used to level your bed. YOU have to do this using the four wheels.

Before trying to bed level, create a mesh and print, Test your probe

The first thing you should do when you install your probe and before you try to do a Mesh and print something out. You need to test your probe.

IMPORTANT: Keep your finger on the on/off switch when you first try to home your printer using the display, just in case you have cabled it up wrong or the firmware goes nuts.

Probe Test

First make sure you can home all axis on your printer.

Now, if you can, connect to your printer via USB using a program such as Pronterface or Octoprint and go to the terminal interface screen (info on various web sites). If you cannot do this, you may find a "Probe Test" menu in your firmware screen display menus, you may not.

Note: You can run the M48 "Probe Test" from the front screen on some firmware, but you will only get the Standard Deviation result and using a terminal allows you to copy the results to a file for future reference.

From the terminal window run the following commands:-

G28

M48 v4

The G28 will home the printer and the M48 V4 will run the probe self test in verbose level 4 debug mode. If you want to get fancy and have the head move around you can do M48 V4 S1.

The probe will repeatedly probe the bed (in the middle) multiple times until it has done 10 interactions of the test (the v4 will output a result every iteration).

Good Probe

When finished, it will then give you a final summary that will look something like this (you will have different numbers).

Recv: Mean: 0.008875 Min: 0.002 Max: 0.017 Range: 0.015

Recv: Standard Deviation: 0.004125

You are specifically looking at the “Range” number and the “Standard Deviation”.

You want your numbers to be as follows:

Range : below 0.03

Standard Deviation : below 0.01

If you would like to read further information on these numbers you can read it here:

<https://www.th3dstudio.com/hc/product-information/ezabl/troubleshooting-ezabl/is-your-ezabl-working-well-m48-test/>

You want to run this test a number of times over an hour or so to make sure that you have consistent readings. It is also worth doing a quick check every week to make sure nothing on your machine has changed and the results are still within spec. Also run the tests anytime you think you may have a problem with the probe, or you move your printer.

If your results are consistently within range, then your probe is working properly and you can move on to Tramming (levelling) your bed to within 0.05mm and then creating a Mesh and printing etc.

Example Problem Probe

If you have numbers like this, then you have a problem (these were some of mine when I had a problem):-

Recv: Mean: -0.007411 Min: -0.032 Max: 0.022 Range: 0.054

Recv: Standard Deviation: 0.017147

Recv: Mean: -0.027659 Min: -0.059 Max: 0.015 Range: 0.074

Recv: Standard Deviation: 0.023881

The problem with the range above being 0.074 is that the print head could be 0.074 above where it should be. Let's say that the bed is smack on and the probe should read 0.0 but is reading 0.074, if you are printing at 0.2mm the printer would print the 1st layer at $z=0.2 + \text{the probes value}$, which is 0.074mm e.g. 0.274 instead of 0.2. This means that the print is unlikely to stick to the bed. The same if it was -0.074 it would be $z=0.126$ and would likely be far too close and again not stick and you would possibly get the extruder skipping. This gets worse if you try to print lower e.g. 0.16 or 0.12.

These were some of my numbers when I fixed my problem:-

Recv: Mean: -0.005300 Min: -0.009 Max: -0.002 Range: 0.007

Recv: Standard Deviation: 0.002112

Recv: Mean: 0.008875 Min: 0.002 Max: 0.017 Range: 0.015

Recv: Standard Deviation: 0.004125

If you have a problem probe, check out the chapter : [Probe Issues](#). If you cannot solve your issue, send the probe back and get a new one as soon as you can.

Do not try to live with a problem probe, it will drive you insane and you will waste hours of your life with prints not sticking, bed meshes being incorrect etc. You will be better off going back to using the z limit switch and manually levelling your bed.

22. Next Tram/Level your bed (using the wheels)

Wheel Turns

How big a turn of the wheel.

I read that the screw pitch of the bed screws is 0.7mm, so one full turn should be 0.7mm.

A 10th of a turn per wheel would equate to 0.07mm, so not much really.

I think this may be where people go drastically wrong, not knowing how much the wheels should be turned.

You should be able to tram/level your bed and print with stock springs. The issue with the stock springs is that they are weak and potentially move over time and you have to re-level the bed.

Recommendation : As soon as you are able to print, print the clicky wheels in chapter : [Change stock springs and wheels](#). One click is 0.01mm approximately, great when using mriscoc's bed tramming wizard.

Bed Tramming Wizard

If you have installed Mriscoc (possibly Marlin, not sure about Jyers), you will find that there should be "Bed Tramming/Levelling", or something similar somewhere in their menus.

Find the Bed Tramming option on your display and run that.

Use these to level your bed using the wheels. Your probe will be used to home the printer and then to measure the four corners of the bed. When measured, you then need to adjust the wheels in each of the corners that the firmware tells you to, until it tells you the bed is level "within its tolerances' ".

23. Now you can create a Mesh

Once you have trammed/levelled the bed, you can then go ahead and create a MESH and store it.

Best to watch some videos on the web for your firmware.

For Mriscoc you can do this from the Advanced screen menu. Set your desired mesh size, create the mesh and then don;t forget to store it.

NOTE: Please refer to the Firmware's own website on how to use the firmware to do the above and create/store a mesh and call a mesh from your slicer software. (I will try to update the guide later with more detailed information on this).

<https://github.com/mriscoc/Ender3V2S1/wiki>

Gcode (reddit user request to document)

I think you should add that using a probe and mesh settings means you need to enable it in GCode. G28 homing turns off mesh, and I have had to add M420 S1 (stored mesh settings) after G28. Others have used G29 J to turn mesh on and test a few points to validate mesh.

If using UBL Unified Bed Levelling

(another reddit user) : Just for reference, G29 L0 loads a mesh saved in slot 0.

G29 has several sub commands, there are more options than just the basic 'make a mesh and use it' that you get with pure G29.

G29 in the context of **UBL** is far superior to M420.

[Bed Leveling \(Unified\) | Marlin Firmware \(marlinfw.org\)](#)

BIG NOTE: *Since I have updated to the mriscoc with UBL (latest version) I have now replaced the M420 S1 gcode with G29 L0 J. The J will do a 3 point bed probe to check the tilt of the table and will load the mesh from memory slot 0 and modify it for the tilt. I then just use the*

front panel display to do a mesh every now rather than at every print. (see part 2 of the guide for my prusa slicer gcode).

Finally - Bed Level Prints

Now you need to return the Chapter: [Ready to Bed level \(really you mean “Tramming”\)](#) to print out some bed levelling prints and correct your z-offset.

24. Probe Issues

Is everything square and tight?

Is your X axis level with your printer base and are the screws holding it tight?

Are your eccentric nuts set correctly X and Y?

Did you check the bolts in your bed plate, Y axis runners, Y mountings etc.

Are your bed springs tight, the bed not wobbling?

Check the Z axis, tighten the lead screw at the joint/motor very tight to make sure there is no play and that the lead screw is not touching the motor arm (see the first video at around 30 mins in Chapter : [Build your Ender](#)).

Check the screws in the top of the Z screw to extruder. Make sure your Z axis is running correctly.

Is your print head mounted firmly, not wobbling?

Is your probe mounted firmly, no wiggle?

Checked all cable connections, checked earthing?

BLTouch Blinking Red

<https://www.3dprintbeast.com/bltouch-blinking-red/>

Cabling issues

Check your connections to the motherboard and the probe itself.

If the motherboard connections are loose you could use hot glue to glue them in, some people have tried Blu Tack but it does work loose over time.

The best thing to do if you have a probe that does not have a good connection to the motherboard is to change the 2 connectors for a 5 pin connector. You require a **JST-XH Male**

Connector 5 pin. This is printable if you can print (See above Physical Installation - Useful Links).

Bltouch mounting

Some people have reported the bltouch being too high when they have mounted it. There is supposed to be a simple solution and that is to flip the mount over, it should then be at the correct height. I don't have one so I don't know.

Probe Interference

There have been mentions of radio interference from microwaves and other electrical devices. You may have to buy a twisted pair/shielded cable for it to stop interference (there is info out there on the web about this).

Loose grub screw and nozzle crashes

I have found that my screw in the top of the probe (used to remove the probe/adjust the probe height) works itself loose when probing. I found this out when my nozzle crashed into the bed and the probe started flashing. The probe itself had disappeared into the probe case. I had to re-level the X axis because of the crashed head and thus the bed, I lost a couple of hours in the process.

Top Tip : Use teflon tape (thread seal tape - plumbing tape) on the screw. It will be easy to fine tune the pin and it will stay in place.

Range and Standard Deviation Issues

To test these see Chapter: [Physically Installed Your Probe. Don't do anything yet.....](#)

BL touch deviation is ,I believe, around 0.0012

Clones (3D touch) are, I believe, around 0.02 but can be lower depending on the clone.

If you do a M48 probe test, you will get back some figures. You are looking for the values of Range, and Standard Deviation. You really want them below these figures:-

Range below 0.03

Standard Deviation below 0.01

If you do a M48 test and find yours is miles off e.g. 0.03 or more, then you could try remagnetising the probe and grub screw. Chapter: [Remagnetise Your Probe](#)

https://www.reddit.com/r/3Dprinting/comments/bhlvuv/improving_bltouch_accuracy/

Probe tip dropping

Try remagnetising - Chapter: [Remagnetise Your Probe](#)

Remagnetise Your Probe

https://www.reddit.com/r/3Dprinting/comments/bhlvuv/improving_bltouch_accuracy/

The below information is copied from the link above in case the link disappears.

Instructions:

- 1) Run the BLtouch repeatability test a few times and copy the results to a notepad as above (M48 V4).*
- 2) Running this test only once is not sufficient.*
- 3) Run multiple tests and copy the output to a notepad.*
- 4) You really want to compare multiple tests before and after, to eliminate a measurement of dumb luck*
- 5) power-down your printer.*
- 6) Measure the length of the pin outside the BLtouch housing while it is in idle (high) position*
- 7) Open up the BLtouch hole that's covered by a screw on the top. This screw is magnetic, which holds the pin in its idle place.*
- 8) Get the pin out of the BLtouch. This can easily be done with a combination of a magnet and a nail that fits the hole.*
- 9) Re-magnetize the magnetic screw, and the pin of the BLtouch.*
- 10) Use the strongest magnet you can find: rare-earth magnets, speaker, harddisk magnets, toys.*
- 11) I would advise to polarize both screw and pin both in the same direction as the screw holds the pin in idle position.*
- 12) Let the screw and pin stick on a strong magnet for 5 minutes and put it back in.*
- 13) Adjust the screw so the pin is again on the same length outside the BLtouch housing.*
- 14) Run the BLtouch repeatability test a few times again and copy those results also to a notepad like the first test above (M48 V4)*
- 15) Reply back to this thread with one average before and after measurements.*
- 16) Hint: A double space at the end of a line creates a newline*

Silicone Spray (Don't Do It !!!!)

Please do not take your probe pin out and use silicone spray or oil on your probe pin to make it run smoother/quicker, it does not help. In my tests when I added silicone spray, my range increased and so did my standard deviation far higher than the 0.02 and 0.01 recommendation. I had to clean my pin and the hole out.

Cleaning the pin and hole

Because I had an issue with my probe and I was doing lots of testing, I had to clean my probe. I used a homemade spray mixture of IPA and water. I sprayed the pin and cleaned it with tissue paper. For the hole I used a pipe cleaner and sprayed the pipe cleaner at one end, then put the wet end in first, ran it back and forth a little and then ran the dry end through to dry the hole.

Interesting post here about cleaning your probe pin.

https://www.reddit.com/r/3Dprinting/comments/6kch74/the_bltouch_pin_no_touchie/

Probe not dropping at the correct angle

You may have an issue with your probe pin dropping at the wrong angle. This can increase the range and standard deviation. This is really a fault with the probe. If (like me) you ended up with either a free probe or a very cheap probe and you want to give it a go, see this. My probe is now happily well in spec.

Fault Finding my 3D Touch probe for my Ender3v2

<https://docs.google.com/document/d/1ownedFyXR9yV2ZnWJldXQM3l8jxuGVyjDspWiTuy6yw/edit?usp=sharing>

Z Adjustable Probe Mount for the Ender3v2

Link Here :

[Z Adjustable Probe Mount for the Ender3v2 - 3DTouchBLTouch by Kwakers2001 - Thingiverse](#)

Note: The Probe offset is very large and may not work very well. I will look at modifying this later, but at the moment I have changed my probe offset to -42 instead of -50 (because mriscoc does not like it lower than -43. My glass bed is pretty much flat so it is not a big issue for me that I am 8mm off the nozzle and I am only doing a 3x3 probe on my bed.

NOTE: I eventually purchased a new probe (Triangle Labs) and it is far superior to the cheaper 3DTouch I purchased, so my advice is Creality Probe or if you want to go cheaper stick with a Trianglelabs.

Marlin's BLtouch FAQ

[BLTouch - Frequently Asked Questions - Frequently Asked Questions - Marlin Firmware Service - Forum \(cnc.id.au\)](#)

25. USB Cable Connection/Wifi connection

When you are happy that you are able to print a good first layer and you are moving onto further prints, you may wish to look at connecting your printer remotely. This will save you having to remove your SD card for every print and can also give you additional features.

Here are some programs you can use to connect to your printer.

Octoprint - <https://octoprint.org/> - can run on Raspberry Pi (Octopi), Android Phone (Octo4a), Windows, Linux and Mac.

Pronterface - <https://www.pronterface.com/>

Creality software that came on the SD card as well as CURA Slicer itself.

Klipper - <https://www.klipper3d.org/>

You can also get Wifi SD cards that work with the Ender3v2.

My Setup

I had an old laptop and put the windows version of octoprint on it and connected it to the printer. The terminal interface comes in really useful to see what is happening on the printer and to run probe tests and copy EEPROM configs to .txt files for backup and to see what I have configured. If you do add octoprint and also change your firmware e.g. to Mriscoc, make sure you read the octoprint integration pages on the firmware web site so that you get all the features of octoprint and mriscoc e.g. remaining print time showing on the display when printing from octoprint.

WARNING: USB Connection - 5v Power Issue

Please see the link below regarding USB port connections such as Octoprint (and other connections to the USB port) and the issue with the 5v rail on the USB cable.

<https://github.com/mriscoc/Ender3V2S1/wiki/Octoprint>

If you don't want to use tape, someone on reddit found this (I have not tried it myself).

[Thing files for USB-A male plug pin isolator \(back-power blocker\) by magnurz - Thingiverse](#)

Reddit User about the thingiverse print above: It usually falls out of the usb connector when I unplug it but it can just be pushed back in again. I have not broken the usb power blocker yet, but it might get annoying if you unplug it a lot.

26. Top Tips

- If you are missing bowden clamps, the little C type plastic clamps (mine were blue), you can use a small cable tie whilst you print new C type clamps (see Chapter : [Other](#)).
- You may want to look at upgrading your plastic extruder arm to an all metal extruder, or at least printing out an additional arm (so you have an emergency spare), as people say that the stock plastic extruder arms often crack underneath where you cannot see it. You are then chasing a problem that you cannot see without removing the arm. (see chapter : [The Plastic Extruder Arm](#))
- If you need to remove your bowden tube from your hotend because you have issues, before you remove it, read up on the hot end fix, capricorn tubes, all metal hotends/bi-metal heat break replacements etc. Here is a link for the hotend fix (it's free just print it out and follow the video) [Simple HOTEND FIX for Creality Ender 3, CR-10 - YouTube](#). Also this link shows the same thing using a metal washer <https://www.youtube.com/watch?v=a8yzmBa9Al4>
- I also advise to never move the x axis up by hand. Instead, use the screen or turn the joint on the lead screw. If you do lift it by hand, DO NOT lift it by the right hand side as you will most likely adjust the cantilever angle on the x-axis. If you must move X by hand, hold the X axis motor (the bit with the QR code) and NOT the right hand side (easily done as most people are right handed).
- If the printer crashes your nozzle into the bed then there is a good chance the X axis cantilever level has changed by a tiny amount e.g. < 1mm on the right hand side. You will probably have to re-level the X-axis and re-level your bed, so keep this in mind.
- SD Card issue: (reddit user) The name of the gcode file needs to be less than 20 characters or else weird things start to happen with the print speed.
- If you are not going to be printing for a day or two, heat up your nozzle and remove your filament. Otherwise there is a good chance that the extruder will squash the filament over time and then it will get stuck at the bottom of the bowden tube and you will have extrusion issues when you start to print again.
- Reddit User: *With an E3V2 there aren't many upgrades that you HAVE to do. You could upgrade the springs/spacer and the extruder. Start with a good quality PLA and just get good at printing it. Avoid the urge to get fancy filament or mods until you are confident about the core principles of slicing and your machine. Every change in filament or machine mod is an opportunity for problems. I personally see it as a challenge, but not everyone does. If you're going to mod, do one at a time and redo all the calibrations. Print a lot with the new mod before doing the next one. It's hard to isolate problems when there are too many simultaneous changes to your set up.*

27. Basic Calibration

Note: The Teaching Tech calibration site does not use your slicer. You download the gcode from the website. This is useful, as once calibrated, if you have further issues with sliced gcode then the issue is more likely to be your slicer settings than your hardware/firmware EEPROM setup.

<https://teachingtechyt.github.io/calibration.html> - For Marlin based firmware (no point reinventing the wheel - unless you have improvements)

If you are struggling with your extrusion check out this link.

[Volumetric Extrusion PSA : ender3 \(reddit.com\)](#)

Klipper Firmware : If you are running Klipper then this site has been recommended.

[GitHub - AndrewEllis93/Print-Tuning-Guide](#) - For Klipper Firmware

New Filament Calibration

Big note here..... Yes EVERY new filament you buy, and then configure the values into your slicer, create a new filament setting for every filament and use that filament setting for that filament when slicing. You will get better, more consistent prints by calibrating.

You really do need to do the “Temperature Tuning” and “Retraction Tuning” for every new filament if you want good prints (<https://teachingtechyt.github.io/calibration.htm>). Especially if you are trying to print lower than 0.2 as you may find you start getting more clogs in your nozzle.

NOTE: I seem to always have problems with these two prints, I don’t know if it is the way it is sliced or what. I have had to lower the bed temp to 50 and add glue stick, but sometimes the temp tower still comes off the bed half way through the print. I will try setting the first layer to 210/215 next time and see if that is the issue. I know that my slicer sets the bed temp to 60 for the first layer and then drops to 50 and I don’t normally have any problems. I have noticed that the print as it prints clips the legs of the tower as it prints the bridges which causes the whole print to wobble and therefore pull the print off the bed. So as I say, not sure if it is the way it is sliced or if it is supposed to do that, but it gives me major headaches.

Printables for the XYZ Steps Calibration

If you have digital callipers try these Thingiverse prints:-

2020 Caliper Axis Calibration Mount - Tevo Tornado, Creality Cr-10, Ender 3, Flash

<https://www.thingiverse.com/thing:3044827>

(and the remix) 2020 Caliper Axis Calibration Mount - extra Calipers clip

<https://www.thingiverse.com/thing:5195905>

The calibration "should" help you get correct size prints.

Slicer Flow Calibration

Don't forget to put the "slicer flow calibration" into your slicer. I also changed the extrusion width flow rates as a percentage of my result e.g. I have 0.4mm, Prusa default was 0.45mm, first layer changed to 0.38, default 0.42 etc.. I may have done it wrong and if I have can someone tell me please, but I think I have done it right.

Printing Issues

This also may help with printing issues:-

[Print Quality Guide \(simplify3d.com\)](#)

[Get help to diagnose and fix 3D print problems \(reddit.com\)](#)

28. First things to print after calibration

So that you have a spare when it breaks - See Chapter: [The Plastic Extruder Arm](#)

[The Extra Strong Extruder Arm](#) - Reddit User recommendation

And have 4 of these. At some point your belts will break Chapter: X and Y belt replacement.

<https://www.thingiverse.com/thing:2349093> - Simply GT2 6mm Belt Clamp. I recommend it.

<https://www.thingiverse.com/thing:5154404> - Ender3v2 Bowden Clip

<https://www.thingiverse.com/thing:5256466> - Bowden Clip 2mm

[NEMA 17 Improved Pulley Puller Gear Extractor \(Ender-3\) by Pastew - Thingiverse](#) - Pulley Gear (I have not tested this yet).

<https://www.thingiverse.com/thing:3236093> - Cutter for the Hot End Fix (I have not tested this - research required). Don't forget to print the fix too (link in the summary).

Also see [Part 2 Ender3v2 Hardware Build Guide](#) of the guide and the Chapter: Ender3v2 Printable Parts, near the end of the guide, for other things you may want to print before they break.

29. That's it for now - no.... There's a Part 2

Hopefully at this point you have a level bed and have been able to print some bed levelling stl's such as "Chep's Bed Level Print" or "teaching techs" bed level prints. If you have gotten this far and have calibrated your printer, then your next step is to start learning the slicer software and design software. I hope you have found some of "The Guide" useful.

Oh, forgot to say, if you decide to upgrade things, do one and only one upgrade at a time. Run the printer for a few weeks and make sure you're happy, then do the next and run it for a few weeks etc. It will save you a lot of hassle.

Good luck with your future designing, slicing and printing .

And watch this:-  Have your 3D Prints started to suck? Watch this!

30. Link to Part 2 of The Guide

Part 2 of the guide has lots of hints, tips and links to various things that you may or may not find useful. If you fancy another good read then here you go.

[Part 2 Ender3v2 Hardware Build Guide](#)

31. Glossary

Link to 3D printing glossary

A reddit user requested a 3D printing glossary, but google is great for these things, so try this link:-

<https://3dinsider.com/3d-printing-glossary/>

Document Changes

1 August 2022 - Document Created. Re-write of the original [Ender3v2 Hardware Build Guide](#)

6 August 2022 - Updated Chapter: Eccentric Nuts

2 April 2024 - Update Chapters : Nozzle to Probe XY and Z offsets and why change bed spring
added Note: 18-8 Stainless Steel Unthreaded Spacer 10 mm OD, 16 mm Long, for M4 Screw Size.

18 April 2024 - Updated Chapter : New Filament Calibration with note about printing issues.

19 April 2024 - Update Preface.

2nd May 2024 - Add link to G10 video.

3rd May 2024 - Added Dry box Video.

3rd May 2024 - Added Video - The Truth of How to Actually Keep it Protected And Dry.

12th May 2024 - Added My Glass Bed exploded paragraph,

15th May 2024 - Added G10 PLA not sticking (especially if new). All printing good now.

17th May 2024 - Added Windows SD card formatting software link.

Required Updates

Possible Fan shroud removal/clean - link etc..

Possible Fix I have tested for Carborundum Glass - still testing but looks possible (at least for me, or was it the probe issue I had????). Need users to test for me.

SD Card issue: (reddit user) For the first point, the name of the gcode file needs to be less than 20 characters or else weird things start to happen with the print speed.

Add pictures of filament coming out of the nozzle onto the bed to show what you are looking for when bed levelling.

PSU Fuses, checking testing drawings/pics.

Requested Updates

None at present.

