THE 3D PRINTER BEE

(https://the3dprinterbee.com/)

Ender 3 Extruder Calibration – Step by Step Instructions

Author | Recent Posts

Martin (Https://the3dprinterbee.com/author/martin_l/)

Besides many test devices, Martin now has his third own 3D printer running and prints as a hobby for friends, family and himself. He is happy to share his experience with each new article.

(https://the3dprinterbee.com/author/martin_l/)

*Disclosure: Links marked with * are Affiliate Links. I earn from qualifying purchases if you decide to make a purchase through these links – at no additional cost for you!*

The Ender 3* (https://amzn.to/36kv6KZ) can only do a good job if you calibrate it correctly. This includes especially the calibration of the extruder. Extruder calibration is critical to your 3D printer for many reasons. For example, it ensures that your printer extrudes exactly the right amount of plastic through the hot end during printing.

If too little filament (https://the3dprinterbee.com/3d-printing-filament-guide/) is used, your 3D printed model will have gaps between the layers, or the layers will be too thin or non-existent. Also, the printed object does not have good adhesion and the layers can delaminate and warp.

If, on the other hand, too much filament is forced through the nozzle, this leads to overextrusion problems, which in turn can result in blobs (https://the3dprinterbee.com/how-to-avoid-blobs-and-zits/) or stringing (https://the3dprinterbee.com/ender-3-stringing-retraction-settings/) within the print model. Extreme overextrusion can even lead to blockages or complete filament jams at the hot end, which can only be removed with great effort.

To avoid these problems in the first place, you should make sure that your extruder is always correctly calibrated.

To calibrate the extruder of your Ender 3 you have to follow these steps:

1. **Load and mark filament**
2. **Extrude and measure the length**
3. **Calculate and enter steps per millimeter**

How exactly you have to proceed, you will find out in the following article. You'll also learn why correct calibration of your 3D printer's mechanical components is critical to producing successful 3D printed models.

Contents [hide]

Necessary Tools and Materials for Extruder Calibration

In order to be properly prepared for the calibration of your extruder, you will need a computer with an installed slicer software in addition to your Ender 3. You also need a filament. There are many possibilities here, it should only be a nonflexible filament. Furthermore, you need a permanent marker.

6 Steps for Optimal Calibration of your Extruder

Step 1: Loading the filament

To calibrate your extruder, the first step is to heat the nozzle of your Ender 3 to the temperature required for the filament you are using. In case you have already loaded a non flexible filament, heat it up to the appropriate temperature.

If this is not the case, load the filament as usual when preheating your 3D printer, making sure that any previously used material has been carefully removed from the machine.

Step 2: Connect the 3D Printer to your computer

Now you connect your 3D printer to your computer via either USB or Wi-Fi. Then open a slicer software that is capable of sending single-line G-code commands to the printer.

For example, you can use Simplify3D (https://www.simplify3d.com/), Pronterface (https://www.pronterface.com/), Repetier Host (https://www.repetier.com/) or OctoPrint (https://octoprint.org/). Then you activate the relative mode on the extruder by sending the command M83 to your Ender 3.

Step 3: Marking and extruding the filament

To calibrate the extruder, send 100 millimeters of filament through the hot end of your Ender 3. Before doing so, you should use the permanent marker to mark the filament 120 millimeters before it enters the extruder.

Now send the command "G1 E100 F100" to your 3D printer. This causes the device to pick up the 100-millimeter filament it accepts and send it through the extruder. This process takes about one minute, and you avoid problems with both filament tension and the hot end of the printer itself.

Step 4: Repeat measurement

After the third step, your 3D printer should have extruded exactly 100 millimeters of filament. To verify this, you can measure the distance between the extruder and the original mark on the filament. If the measurement is 20 millimeters, your extruder is properly calibrated and you do not need to take any further steps to calibrate it.

However, if the dimension is more than 20 millimeters, your 3D printer is most likely under-extruded and you need to increase the setting for steps per millimeter.

If the dimension is less than 20 millimeters, it is an overextrusion. This means that you must reduce the setting steps per millimeter.

Step 5: Calculation of the correct steps per millimeter value

To accurately calibrate your extruder, you must first determine the current, incorrect steps per millimeter value of your 3D printer and then calculate the correct physical value.

To do this, send the command M503 to your ender 3, which will result in a string being returned to your monitor. Find the line that starts with "echo M92" and then find the E-value, which is usually at the end of this line. This is the current value for steps per millimeter.

To determine the physical step per millimeter value, you must first find out how much filament has actually been extruded by your printer. To do this, you measure the distance between the extruder and the mark on the filament and subtract this value from 120, so the formal is: 120 – length from the extruder to mark = actual extruded length

In the next step, you look at how many steps the extruder has taken to extrude this amount of filament. You can calculate this by multiplying the value for the steps per millimeter by the length that the printer should have extruded. In this case, this is 100 millimeters. Accordingly, the formula is as follows: Steps per millimeter * 100 = steps

This way you can calculate the physical, correct steps per millimeter value by dividing by the extruded length and using the following formula: steps / actual extruded length = exact steps per millimeter value

Finally, set this value as steps per millimeter of your 3D printer and you're ready to go.

Step 6: Define a new step per millimeter value

To set a new step per millimeter value, first send the command M92 E xxx to your 3D printer. Replace the placeholders with the exact value for steps per millimeter that you calculated before. You can save this setting in your printer using the command M500.

Afterwards it is recommended to turn the printer off and on again. Then send the command M503 to the device again and make sure that the E-value matches the new value for steps per millimeter.

If this is not the case, repeat the first part of this procedure.

To check whether your extruder is now correctly calibrated, you can repeat the first four steps. However, this time you should get a value of exactly 20 millimeters between the extruder and the mark you have made. If not, recalculate the value and save it again.

After completing these six steps, your 3D printer should now have a perfectly calibrated extruder. As a general rule, a new 3D printer always needs to be calibrated before the first print. Especially for low-cost devices, you can't assume that they will be tested and correctly adjusted before shipping.

This means that you must do this as a user. However, especially if you're new to 3D printing, this isn't always obvious. In the long run, it may make sense to simply replace the stock extruder with an all-metal version. While the plastic lever and brass gears of the existing part will work fine for a while, even a 40T stainless steel gearbox at an affordable price* (https://amzn.to/3kGVrYL) can make a big difference to your future print models and 3D printer operation.

Other Ender 3 Calibrations

With the Ender 3* (https://amzn.to/36kv6KZ), Creality has succeeded in developing a great successor for the CR-10* (https://amzn.to/2GNgk6P). The Ender 3 is very good value for money and has made a lasting impression on the 3D printing community.

And because it is unusually powerful for its low price, it is especially popular with beginners to 3D printing.

However, it is necessary to make some calibrations on Ender 3 to ensure that the mechanical components of the device are carefully matched to each other even after many hours of work. Some of these calibration steps will be explained here:

Tightening the screws

While Creality 3D printers offer exceptional value for money, there is a lack of quality control. This is evidenced by the fact that the screws on many printers are not fully tightened.

So it is your responsibility as a user to ensure that all screws are tightened. This sounds simple but can make a big difference in the end result. By tightening the screws you not only prevent problems like ghosting, but you can also reduce the number of faulty prints significantly.

The correct leveling of the printing bed

Another very simple yet important step in calibrating your 3D printer is to correctly align the printing bed. This is one of the most important adjustments but is often forgotten. Ender 3's print bed adjustment is quick and easy, and immediately improves the surface quality of your 3D printed model.

To level the printing bed accordingly, you first reset the position of all axes to the default value. Then activate the stepper and move the print head to a corner. Make sure that there is a sufficient distance between the nozzle and the building platform.

While the nozzle is moved, it should not be able to damage the building platform in any way. Now unscrew the head under a corner of your Ender 3 while slowly moving a paper back and forth. Continue doing this until you notice a slight friction. Repeat this procedure in all four corners and you will get a flat printing bed at the end.

The belt tension

The belt tension of your Ender 3 also has a big influence on how the final result of your 3D prints looks like. Loose belts can cause ghosting, layer shifts or dimensional accuracy problems.

You can tighten the belts very easily by unscrewing the brackets of the bearings and then moving them so that sufficient tension is created between the stepper motor and the bearings.

The right sensitivity is crucial here, as the tension must be neither too strong nor too weak.

Adjusting the eccentric nuts

The next important calibration for Ender 3 concerns the eccentric nuts. You will find these nuts under the bed and on the Z-axis. You can turn them with the supplied wrench until they are tightened.

The eccentric nuts should be so tight that you can neither turn nor tilt the bed. However, they must also not hinder the movement. It is best to first loosen all the nuts and then tighten them again until the bed is stable.

The X-axis should also be adjusted in this way. It should be in a horizontal position and should not offer too much resistance when moving up and down.

Related Questions

How can an overextrusion on Ender 3 be avoided?

To correct overextrusion at Ender 3, it is first important to check the extrusion multiplier. This determines the speed at which the 3D printer extrudes plastic.

If unusually large layers or filament jams occur in the nozzle, the extrusion multiplier is usually switched off. The default value of this multiplier is usually 1 or 100 percent. If the printer extrudes too much material, this setting should be reduced by 2.5 percent in stages. If this does not lead to the desired success, the printing temperature should be reduced in the next step. If it is set too high, the filament can melt over and flow out of the 3D printer nozzle in an uncontrolled manner.

The printing temperature is reduced in steps of five degrees Celsius until the perfect temperature is reached for both the device and the selected filament. If there is still the problem of overextrusion, the set filament diameter should be checked. The three standard values are 1.75, 2.85 and 3 millimeters. If the slicer software uses a thinner filament diameter than is actually used, the filament will be discharged through the extruder at a higher speed, which will result in overextrusion and faulty 3D printed models.

What causes strained or weeping 3D printed models?

So-called cords or weeping prints always occur when small plastic chains are left behind on a 3D printed object. Normally, this phenomenon is caused by the plastic flowing out of the nozzle while the extruder is still moving.

However, there are different settings in the usual slicer programs that can solve this problem. The most common setting, which can prevent strained or weeping 3D printed models, is called retraction.

If this function is activated, the filament is pulled back into the nozzle after printing a section of the 3D print model. This is an effective countermeasure if the filament tends to seep out of the nozzle afterwards. Once printing is to resume, the filament is pushed back into the nozzle so that the molten plastic can begin to flow out of the tip again.

You can make sure that the retract function is enabled by going to the “Extruder” tab of the “Edit Process Settings” menu. There you can enable the retract option for the extruder on the 3D printer.

What happens if the 3D printer overheats?

The temperature of the filament exiting the extruder can be between 190 and 310 degrees Celsius. As long as the liquid plastic is still hot, it is flexible and can easily be formed into various shapes.

As soon as it cools down, however, it becomes solid within a short time and retains its shape. In order for the plastic to flow freely through the nozzle, it is therefore necessary to achieve the optimum balance between temperature and cooling. If this delicate balance is not achieved, some problems with print quality may be observed.

These become noticeable when the exterior of the 3D printed object is not as precise and defined as desired. Overheating of the 3D printer can cause the filament to liquefy too much, which cannot cool down in the time available, resulting in warped 3D printed parts. In the worst case, the liquid filament will flow uncontrolled out of the overheated nozzle.

Search