INSTRUCTION GUIDE

Vector 3D Printer Calibration and Testing

Version 1.1

VECTOR3D

Contents

Introduction

Congratulations on getting started with the Vector 3D printer calibration and testing files. This document exists to help you with getting the most out of the files, help you understand the test results, and tune your printer to improve performance.

Filament Usage

As these parts have very little functional use after the testing is completed, I've done my best to minimise the amount of plastic used for these tests in an effort to reduce waste. This has the added benefit that they are also quite quick to print, but I would avoid trying to print multiple tests on the same print bed; mainly because this can affect the print results.

Orientation

All the STL files are orientated correctly upon importing to the slicer so there is no need to spin them about. They are also always marked with the X and Y directions on the parts so that you know once the file is removed from the printer, which way it was facing when it was printed. This is important and often overlooked as knowing the direction can be important to diagnosing specific issues like cooling or dimensional calibration.

Getting Help

You can ask on the Vector 3D discord in the #calibration channel, we're here to help so come and join us using this link: https://discord.gg/xXmuUpJhxc

Discord

Overhang

Test file

The overhang test is designed to determine the ability of you printer, settings, and filament to print steep overhanging angles in the four principal directions; X+, X-, Y+ and Y-.

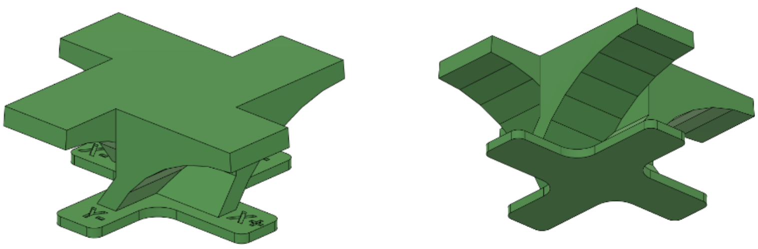

Figure 1: Vector 3D Directional Overhang Test; "Overhang Directional Test".

The angles measured range between 35 and 85 degrees from the vertical so 0 degrees would be vertical, 90 degrees would be horizontal. The increments are 10 degrees so there are 6 individual test angles. The overhangs are repeated four times, once for each direction with no overhanging feature in between so that the results of one direction do not impact another. The directions relative to the printer axis are labelled on the feet.

Settings

When slicing this file its best to go with your normal slicer settings as that will allow you to understand what to expect from a typical print. PLA typically performs best in this test as its normal to print with high amounts of cooling as it doesn't warp very much.

If you want to find the maximum, increase cooling as much as possible and reduce layer height to the lowest you can.

Reading Results

Its difficult to measure this test objectively. Instead, you just want to look for the overhang lines to be as straight and uniform as possible. Where strands of filament droop down off the print, or the steepness is no longer maintained due to softness, the overhang limit has been reached.

45 degrees is what is normally expected so getting this in all directions is the minimum needed to PASS. 85 degrees is virtually impossible using standard FDM setting with current popular consumer technologies so do not expect to achieve this.

For best understanding of your machine, record the overhang angle achieved in all four directions for a selection of filaments that you use. You will typically see that some directions repeatedly perform better than other due to the design of the part cooling system installed. You can use this to your advantage when arranging prints on a bed angling the overhangs into the direction which performs the best.

Making Adjustments

Increasing cooling will improve overhang results but be aware that some filaments such as ASA and ABS are significantly affected by this as it reduces the layer strength.

Purchasing a higher power fan, more fans, increasing RPM, or optimising flow characteristics of the fan ducts are all other ways to improve overhang capability.

Bridging

Test File

The bridging test is designed to determine the ability of you printer, settings, and filament to print unsupported bridges in the four principal directions: X+, X-, Y+ and Y-.

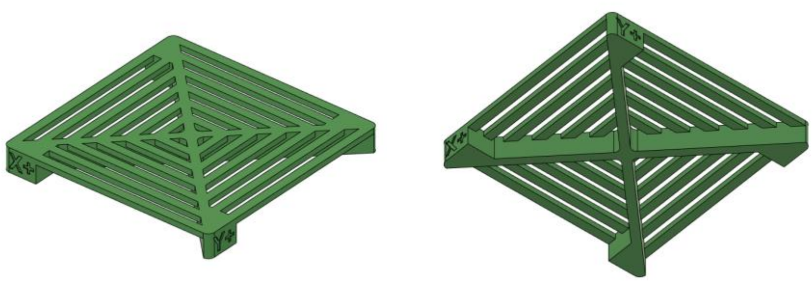

Figure 2: Vector 3D Directional bridging Test; "Bridging Directional Test".

The bridge distance varies from 6mm to 40mm and each bridge is 2mm wide with a gap in between and are setup to be perpendicular to each of the principal axis. They are supported along the diagonals where the directions are also indicated on the end faces.

Settings

As with the overhang test, its best to go with your normal slicer settings as that will allow you to understand what to expect from a typical print. Again, PLA typically performs best in this test as its normal to print with high amounts of cooling as it doesn't warp very much.

If you want to find the maximum, printing at higher speeds and slightly lower temperatures can also help with reducing the drooping. This reduces the time spent on the bridge and the viscosity of the extruded filament.

Reading Results

Use a set of callipers, digital or vernier, to measure the thickness of the bridges all the way from 6-40mm, be careful when measuring not to compress the bridge as this will invalidate the result. If there is a strand or filament touching the build plate at any point then it's a fail for that direction.

For the best understanding of your machines record the bridge for all for directions for a selection of filaments that you use. As with overhangs, you'll typically see that some directions repeatedly perform better than other due to the design of the part cooling system installed. You can use this to your advantage when arranging prints on a bed angling the overhangs into the direction which performs the best.

Making Adjustments

Cooling capacity is most important here so increasing that will provide the best difference but as always be aware that some filaments such as ASA and ABS are significantly affected by this as it reduces the layer strength.

Purchasing a higher power fan, more fans, increasing RPM, or optimising flow characteristics of the fan ducts are all other ways to improve overhang capability.

Adjusting flow rate down for bridging if the option is available in your slicer. For example, a reduction to 95% from 100% will improve bridge performance.

Reducing printing temperatures can be done to increase viscosity of the extruded filament but this can cause adverse effects on the rest of the print.

Increasing printing speed for the bridge will reduce the time that the filament will be drooping. Lots of cooling is still needed to do this to ensure the bridge does not collapse after being printed.

Dimensional Accuracy and XY Skew

Test File

This one is affectionately known as the calibration flower. A dimensional test that determines the ability of your printer, settings and filament to print geometrically accurately and can also be used to calibrate your printer to improve results for future prints.

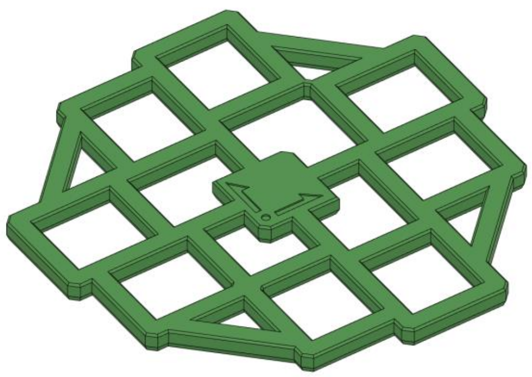

Figure 3: Vector 3D Dimensional and Skew Test: "Calibration Flower"

Settings

As with the other tests, its best to go with your normal slicer settings as that will allow you to understand what to expect from a typical print. The filament choice is important because of different expansion and contraction rates. Repeating the test with different filaments will help you understand how this affects size and what you need to do to get the best results.

Reading Results

There are ten separate dimensions that need to be collected from this print. Every dimension should be approximately 50mm or 100mm. If you get measurements that are more than 5mm from this, and the printed parts looks correct, you may be measuring in the wrong place.

To take the measurements, use a set of callipers, either vernier or digital. You'll need to take both 'inner' and 'outer' dimensions so make sure you are familiar with how to use the callipers to do this. The arrows on the print point towards X+ and Y+ so make sure you identify this before noting down the dimensions.

The measurement locations are shown in Figure 4. Locations 1, 2, 5, 6, 9, 10 should be measured as an 'outer' dimension while 3, 4, 7, and 8, should be measured as internal dimensions. The numbers are positioned on the side where the callipers should measure to help you get this right.

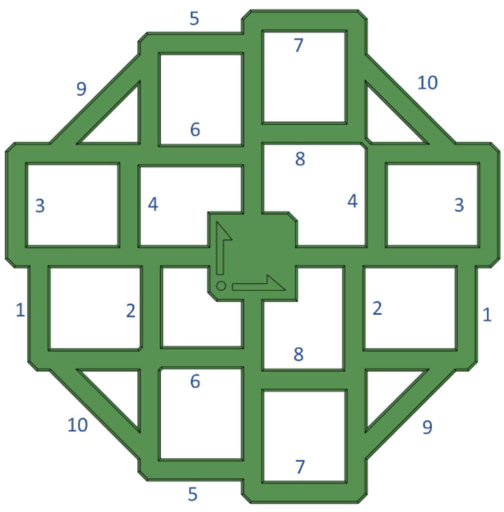

Figure 4: Calibration flow measurement locations.

Log the measurements in the Calibration Calculator provided with this document in the area marked ‘measurements’ in the numbered boxes. The target dimensions are shown by default.

	A	B
1	Measurements	
2	**1**	100
3	**2**	50
4	**3**	100
5	**4**	50
6	**5**	100
7	**6**	50
8	**7**	100
9	**8**	50
10	**9**	100
11	**10**	100

Figure 5: Measurements

Making Adjustments

The errors found in this test come from belt tension, incorrect Steps/mm as well as filament contraction. If your results are more than 1mm away for their intended dimensions, check that your steps/mm are set correctly and that your belt tension is good.

The results for size are provided via the spreadsheet where the measurements are logged. This is provided as a percentage. There are two main methods for correction, changing the firmware steps per mm, or using your slicer to incorporate XY shrinkage compensation for each filament as the shrinkage will change for each type. Testing suggests around 0.45% for ABS and around 0.1% for PLA but your results may vary.

Skew results are shown as an angle, and the methods for correction for Klipper and Marlin are shown to make the process easier, these can be copied from the spreadsheet. Check the skew correction pages for both Marlin Firmware and Klipper Firmware to identify how to implement these changes.

Always reprint this calibration part after making changes to validate.