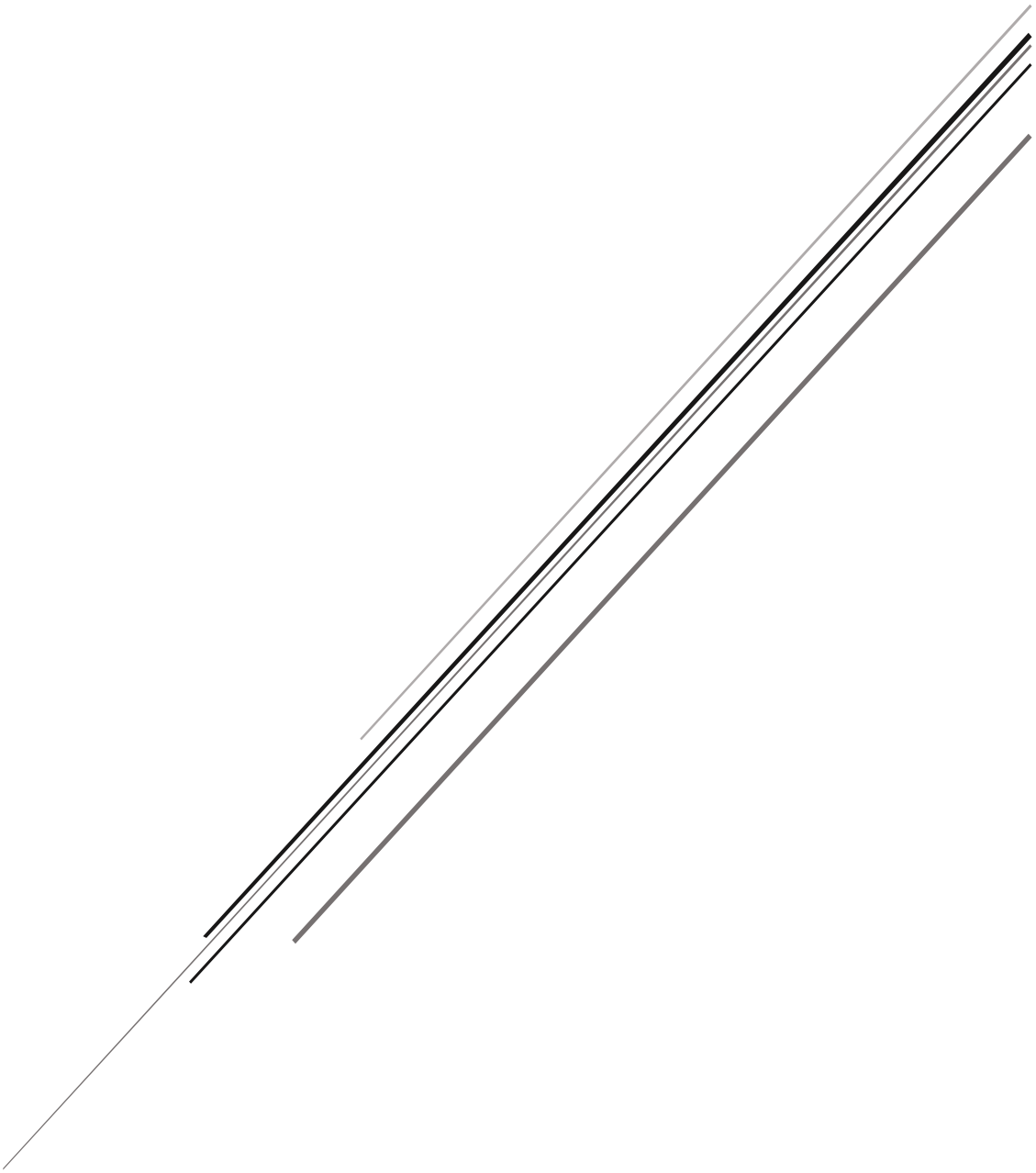
INSTRUCTION GUIDE

Vector 3D Califlower



Version 2.0

Contents

Introduction

Congratulations on getting started with the Vector 3D Califlower. 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 the Califlower has very little functional use (but can be a tea coaster) 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. Please do your part by printing this file as few times as possible to achieve the desired results. Also, avoid trying to print multiple tests on the same print bed as this can affect the print results.

Orientation

All Vector 3D calibration 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

Dimensional Accuracy and XY Skew

Test File

This test is affectionately known as the Califlower. A dimensional test that determines the ability of your printer, settings, and filament to print geometrically accurately and then calibrate your printer to improve results for future prints.

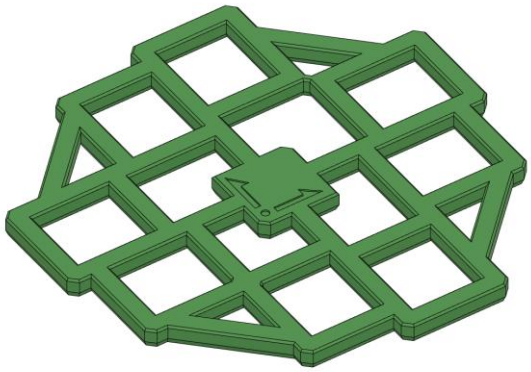


Figure 1: Vector 3D Dimensional and Skew Test: "Califlower"

Settings

As with the other calibration 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

the different expansion and contraction rates. Repeating the test with different filaments will help you understand how this affects the size and what you need to do to get the best results.

The calculator has the option of printing at a larger scale, if you have large vernier/digital callipers and wish to print at a larger size, you can scale the print in your slicer and then just remember to adjust the scale in the calculator by the same amount.

Scale	100%
(if you have a larger printer, and larger tools, you can print at larger scale)	

Figure 2: Adjustment for scale in the cauliflower calculator.

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 3, also repeated in the calculator. 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.

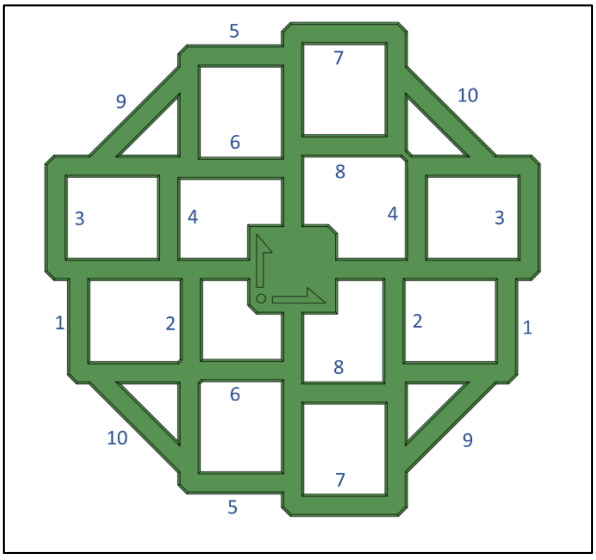


Figure 3: Calibration flow measurement locations.

Log the measurements in the Calibration Calculator provided with this document in the area marked 'measurements' in the numbered boxes. Remember to take each reading three times, the calculator will then determine the average of these readings. The target dimensions are shown by default.

	Reading 1	Reading 2	Reading 3	Average
1	100.00	100.00	100.00	100.00
2	50.00	50.00	50.00	50.00
3	100.00	100.00	100.00	100.00
4	50.00	50.00	50.00	50.00
5	100.00	100.00	100.00	100.00
6	50.00	50.00	50.00	50.00
7	100.00	100.00	100.00	100.00
8	50.00	50.00	50.00	50.00
9	100.00	100.00	100.00	100.00
10	100.00	100.00	100.00	100.00

Figure 4: Measurements

The readings have conditional formatting which means they can change colour based on the value. If the calculator thinks you may have measured the wrong size, the box will read either orange or red depending on the deviation. These highlights do not represent the quality of your print, only an indication that you measured in the right place. The values that determine the colours are shown on the calculator below the measurements table.

Understanding Results

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 of these test come in three forms, XY size, skew, and inner/outer. These are shown in a table under the heading Results.

Results

	Error	Correction
X	0.00%	0.00%
Y	0.00%	0.00%
Skew	0.00°	0.00°
Inner	0.00%	0.00%
Outer	0.00%	0.00%

Figure 5: Results table

For XY size the results are provided as a percentage. At the default size of 100mm, 1%=1mm, so that should help you understand how far off your prints are. Positional error can scale with the travel distance so it's important to get this dialled in accurately.

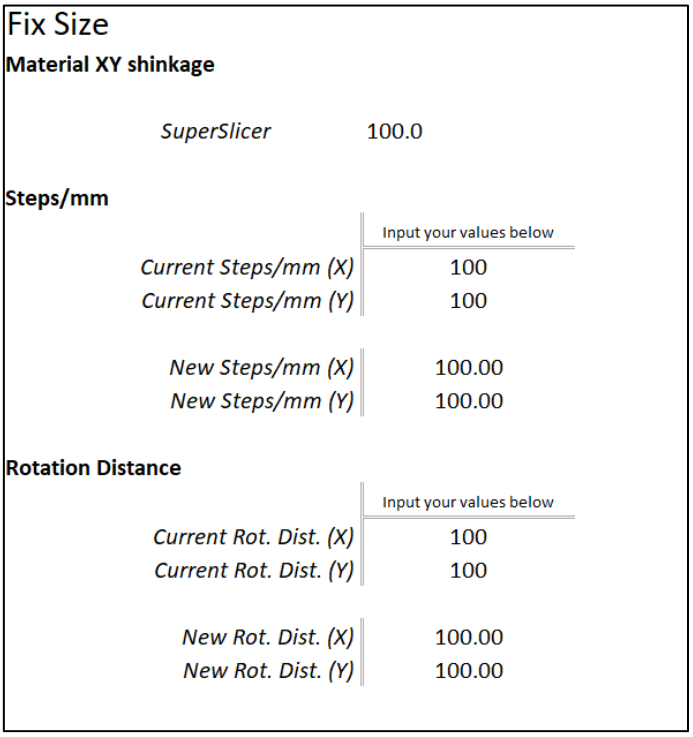
Skew results are shown as an angle. Angles do not change with scale, but the further you travel at an incorrect angle, the larger the error is so. Therefore, the larger your printed parts, the more important this value becomes.

Making Corrections

X Y Adjustments

For adjusting size errors 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 and there is an integrated tool to help you with this.

If the results shown for X and Y are within 0.5% then this is best adjusted via shrinking compensation as the error is likely down to the filament shrinking when it cools. Unfortunately, this feature only seems to be available in SuperSlicer currently. You can copy the value below into the filament shrinking compensation value to implement this correction.



Fix Size

Material XY shinkage

SuperSlicer	100.0

Steps/mm

	Input your values below
Current Steps/mm (X)	100
Current Steps/mm (Y)	100
New Steps/mm (X)	100.00
New Steps/mm (Y)	100.00

Rotation Distance

	Input your values below
Current Rot. Dist. (X)	100
Current Rot. Dist. (Y)	100
New Rot. Dist. (X)	100.00
New Rot. Dist. (Y)	100.00

Figure 6: Integrated shrinkage, steps and rotation calculator

If you don't use SuperSlicer, or have error more than around 0.5%, then its recommended to adjust via changing the steps/mm or rotation distance. Firstly, get the current values from your printer. For Klipper and RRF just check your configuration file, for Marlin use M503 to show current steps/mm. Place these values into the appropriate boxes for X/Y, and then copy the new values into your firmware. For Marlin use M92 (example: M92 X80.67 Y80.32) to send these values to the printer and M500 to save them.

Skew Adjustments

Despite being a more complex calculation, skew is quite simple to implement once it has been evaluated as the calculator does all the hard work for you. Marlin is slightly more difficult as firmware adjustment and compiling may be needed.

Skew adjustment calculations are based on the fact that for a given perfect square with side length x, the diagonal y, across that perfect square must be: $y = x \times 2\sqrt{2}$. This is why you see 70.71 occur quite often. Don't worry about that though, the calculator deals with all this for you.

To implement the fix you can simply follow the instructions for your firmware.

For Klipper, you'll enable skew correction and restart, then send the calibrated values via console, save the profile, and then add commands to load and clear the profile when printing.

For Marlin there are three options as they have two methods for firmware and one for G-code. Both firmware versions SHOULD produce the same results, but some rounding errors can make them different. My current recommendation is to use the first method of adding four new lines to your firmware. If you have Skew_correction_gcode enabled in firmware already you can use the simpler G-code correction method but if you need to change firmware anyway, one of the firmware versions may be just as easy.

Fix Skew

Note: Because of the way excel works, you may find "," and "." get switched for your regional language settings.

Klipper G Code Fix

Add	[skew_correction]	*to printer.cfg*
Send Command	SET_SKEW XY=100,100,70.71	*via console*
Send Command	SKEW_PROFILE SAVE=my_skew_profile	*via console*
Add	SKEW_PROFILE LOAD=my_skew_profile	*to end of start G code*
Add	SET_SKEW CLEAR=1	*to start of end G code*

Marlin Firmware Adjustment

Add	if ENABLED(SKEW_CORRECTION)	*to firmware*
Add	define XY_DIAG_AC 100	*to firmware*
Add	define XY_DIAG_BD 100	*to firmware*
Add	define XY_SIDE_AD 70.7107	*to firmware*

Marlin Firmware Adjustment (alternative)

Add	if ENABLED(SKEW_CORRECTION)	*to firmware*
Add	define XY_SKEW_FACTOR 0	*to firmware*

Marlin G-Code Adjustment

Ensure	#define SKEW_CORRECTION_GCODE	*in firmware*
Send Command	M852 I0	*via console*
Send Command	M500	*via console*

RepRap Firmware Adjustment

Add	M556 S100 X0	*to config.g*

Figure 7: Skew adjustment tools.

RepRap firmware is the easiest to adjust, simply add this M556 command to your configuration file.

There may be circumstances that you get very close to zero error, but your results still show large deviation from the target value. This means that there are other problems that should be addressed. Either your printer is not able to accurately reproduce the same dimension resulting in uneven walls, your slicer has a setting which is expanding or contacting walls, or the amount of filament you're extruding is incorrect resulting in extra thick, or extra thin walls.

This calculator works to ensure that the centre of the line drawn by the nozzle is in the correct position, making it somewhat independent of the amount of filament extruded. If you are over extruding very thick walls, the nozzle may still be in the right place, but your inner and outer dimensions will still be incorrect.

For more detailed information on correction, check the skew correction pages for RepRap Firmware, Marlin Firmware and Klipper Firmware to identify how to implement these changes.

Always reprint this calibration part after making changes to validate they have taken effect.

Happy printing!