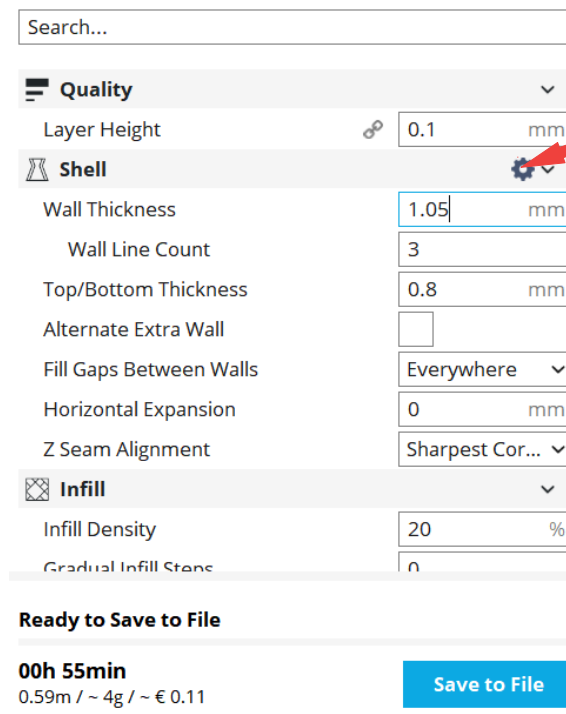


Cura Preferences

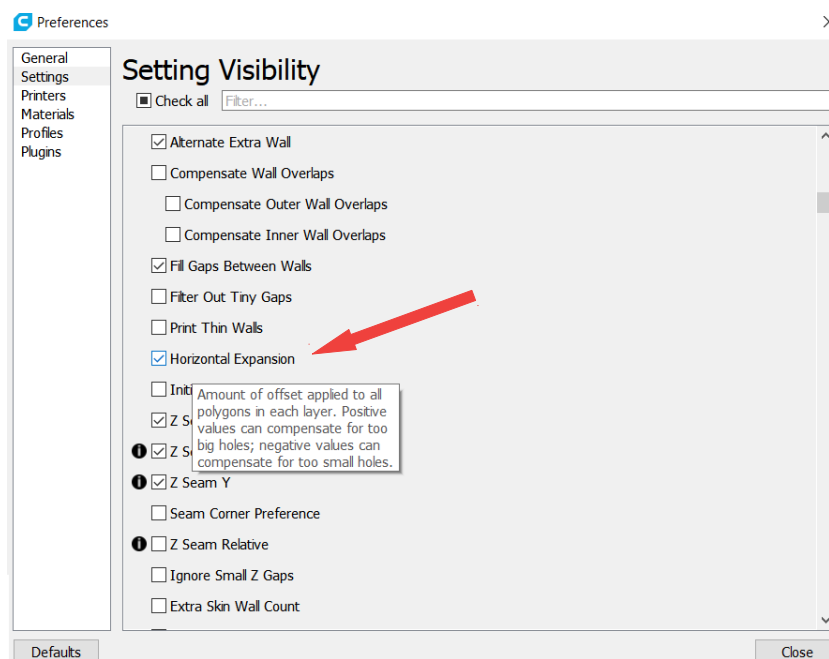
Shell

The shell setting adjusts the thickness of outside walls (on the X/Y axis) of the model. You might want to modify the shell setting based on the material you are using. All plastics shrink as they cool.

Horizontal Expansion



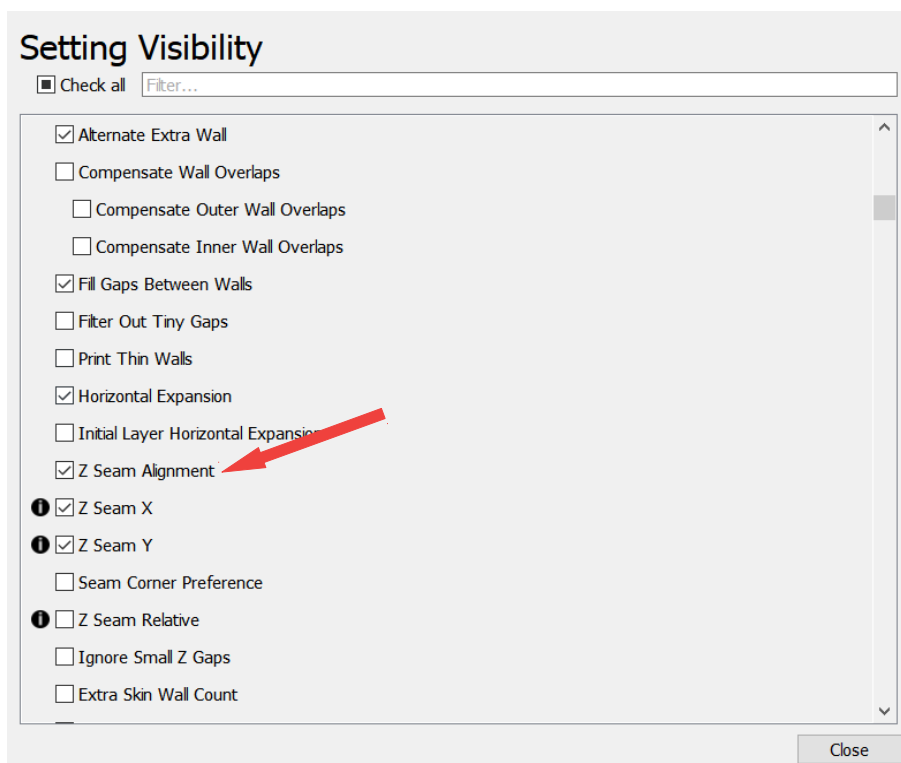
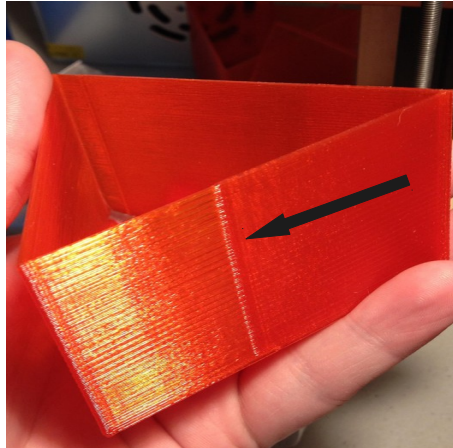
A screenshot of the Cura software's settings panel. At the top is a search bar. Below it is a 'Quality' section with a dropdown arrow. Under 'Quality', 'Layer Height' is set to 0.1 mm. The 'Shell' section is highlighted with a red arrow pointing to its gear icon. Under 'Shell', 'Wall Thickness' is set to 1.05 mm, 'Wall Line Count' is 3, 'Top/Bottom Thickness' is 0.8 mm, 'Alternate Extra Wall' is unchecked, 'Fill Gaps Between Walls' is set to 'Everywhere', 'Horizontal Expansion' is set to 0 mm, and 'Z Seam Alignment' is set to 'Sharpest Cor...'. Below the 'Shell' section is an 'Infill' section with 'Infill Density' set to 20% and 'Gradual Infill Steps' set to 0. At the bottom, it says 'Ready to Save to File' with a timer of '00h 55min' and a 'Save to File' button.



A screenshot of the 'Setting Visibility' dialog box in Cura. The title bar says 'Preferences' and 'Setting Visibility'. On the left is a sidebar with 'General Settings', 'Printers', 'Materials', 'Profiles', and 'Plugins'. The main area has a 'Check all' checkbox and a 'Filter...' input. A list of settings is shown with checkboxes: 'Alternate Extra Wall' (checked), 'Compensate Wall Overlaps' (unchecked), 'Compensate Outer Wall Overlaps' (unchecked), 'Compensate Inner Wall Overlaps' (unchecked), 'Fill Gaps Between Walls' (checked), 'Filter Out Tiny Gaps' (unchecked), 'Print Thin Walls' (unchecked), 'Horizontal Expansion' (checked, with a red arrow pointing to it), 'Infill' (unchecked, with a tooltip: 'Amount of offset applied to all polygons in each layer. Positive values can compensate for too big holes; negative values can compensate for too small holes.'), 'Z Seam' (checked), 'Z Seam Y' (checked), 'Seam Corner Preference' (unchecked), 'Z Seam Relative' (unchecked), 'Ignore Small Z Gaps' (unchecked), and 'Extra Skin Wall Count' (unchecked). At the bottom are 'Defaults' and 'Close' buttons.

Z Seam Alignment

Sometimes there will be a visible mark on the surface of your print at the start of a layer. When these marks are all aligned, they form a visible line called the “Z Seam”. Z Seam Alignment allows you to choose where each new layer in the Z direction starts in order to eliminate the appearance of a Z Seam on the exterior of your model.



Fill Gaps Between Walls

Because your 3D printer includes a fixed size nozzle, you may encounter issues when printing very thin walls that are only a few times larger than the nozzle diameter. For example, if you were trying to print a 1.0mm thick wall with a 0.4mm extrusion width, you may need to make some adjustments to ensure your printer creates a completely solid wall and does not leave a gap in the middle.



Alternate Extra Wall

The Alternate Extra Wall setting adds strength to the outer walls of the print without taking the time required to add additional shells on every layer of the print. Alternate Extra Wall will add one extra interior shell to every other layer. For example, if your print is set for two shell widths Alternate Extra Wall will add an extra one on every odd numbered layer.

Speed



Number of slow layers

Number of Slow Layers will slow down the printing of the bottom layers and defines the number of layers it will take the printer to reach the set print speed. The speed, based on the initial layer speed and print speed, will linearly incline over the number of layers specified in the Number of Slow Layers setting. A higher value will decrease the chance of your print warping, but this setting can also increase your print time significantly.

You can find more here

<https://ultimaker.com/en/resources/21932-mastering-cura>

<https://ultimaker.com/en/resources/20422-cura-support-settings>