MeaDe

FDM 3D Printing & Ulitimaker Cura

GLOSSARY

Welcome to the MeaD MaDe FDM 3D Printing & Ultimaker Cura Glossary! This is an excellent resource for anyone looking to improve their knowledge of 3D printing, no matter their experience level.

3D printing can be an incredibly rewarding hobby or profession, but it can also be complex and challenging. With so many different settings, options, and techniques to consider, it's easy to get overwhelmed or confused. This is where your glossary comes in. By having a detailed list of terms and settings at your fingertips, you can quickly and easily find the information you need to overcome any obstacle.

One of the great things about 3D printing is that there's always something new to learn. Whether you're just starting out or have been printing for years, there's always a new setting to tweak, a new technique to try, or a new material to experiment with. With this glossary, you'll be better equipped to take your printing to the next level.

So whether you're printing functional parts, artistic creations, or anything in between, we hope that this glossary proves to be an indispensable tool for you.

Happy printing!

Chris MeaD

@itsMeaDMaDe

Adhesion:
The ability of the first layer of a print to stick to the printing surface. Good adhesion is important because it ensures that the print remains firmly in place throughout the printing process, resulting in a more accurate and high-quality print.

Brim:
A flat extension added to the base of a 3D print to increase the surface area of the first layer, helping to improve bed adhesion and prevent warping.

Build Plate Adhesion Extruder:
The extruder used to apply the material for build plate adhesion (e.g. a brim, raft or skirt).

Build Plate Adhesion Type:
The method used to improve bed adhesion of the print, such as brim, raft, skirt, or none.

Build Plate Temperature:
The temperature set for the print bed, which affects the adhesion and quality of the first layer.

Concentric Infill:
A type of infill pattern where the print head moves in a circular motion, creating a spiral-like pattern that can provide better strength and stability to the printed object.

Cooling:
The process of cooling the printed material as it is being extruded, which helps to improve print quality by reducing warping, stringing, and other printing defects.

Cross Infill:
A type of infill pattern where the print head moves in a crosshatch pattern, creating a grid-like structure that can provide good strength and stability to the printed object.

Cubic Infill:
A type of infill pattern where the print head moves in a cubic pattern, creating a solid-looking structure that can provide good strength and stability to the printed object.

Enable Print Cooling:
A setting that allows for the use of a fan to cool the printed material as it is being extruded.

Enable Support Interface Cooling:
A setting that allows for the use of a fan to cool the interface layer between the support structure and the printed object, making it easier to remove the support structure after printing.

Enable Ironing:
A setting that allows for the smoothing of the top surface of the print by applying additional heat and pressure to it.

Tree Support:
A type of support structure that is designed to mimic the shape of a tree, providing more support only where it is needed and reducing the amount of material used.

Flow Rate:
The amount of material that is extruded through the nozzle per unit time, which affects the quality and strength of the printed object.

Flow Sensor:
A sensor that detects the flow of material through the nozzle, helping to ensure consistent and accurate printing.

Gradual Infill Step Height:
The height at which the infill pattern transitions from one density to another.

Gradual Infill Step Size:
The distance between the infill pattern transitions from one density to another.

Gradual Infill Steps:
The number of infill pattern transitions between different densities in a gradual infill pattern.

Grid Infill:
A type of infill pattern where the print head moves in a grid pattern, creating a series of squares that can provide good strength and stability to the printed object.

Gyroid Infill:
A type of infill pattern that resembles a 3D labyrinth, which can provide good strength and stability to the printed object while using less material than solid infill patterns.

Infill:
Infill refers to the interior structure of a 3D printed object, which is used to provide strength and stability while minimizing material usage. This is achieved by filling the empty space within the object with a pattern of repeating lines, shapes, or solid layers.

Infill Before Walls:
This setting in 3D printing software instructs the printer to print the infill before printing the outer walls of the object. This can result in a smoother finish on the top surface of the object.

Infill Density:
This setting determines how much of the interior space of the 3D printed object will be filled with infill. A higher density will result in a stronger and more solid object, but will also use more material and take longer to print.

Infill Line Directions:
This setting determines the orientation of the lines that make up the infill pattern. Changing the direction of the infill lines can affect the strength and stability of the printed object.

Infill Line Multiplier:
This setting controls the thickness of the lines that make up the infill pattern, which can be adjusted to achieve a desired level of strength and material usage.

Infill Line Width:
This setting determines the width of the lines that make up the infill pattern. A narrower line width can result in more precise and detailed prints, while a wider line width can make the object stronger.

Infill Only Where Needed:
This setting instructs the printer to only add infill to areas of the object where it is necessary for structural support. This can help reduce material usage and printing time.

Infill Orientation:
This setting determines the orientation of the infill pattern within the 3D printed object. Changing the orientation can affect the strength and stability of the printed object.

Infill Overlap:
This setting controls the amount of overlap between adjacent infill lines, which can affect the strength and durability of the printed object.

Infill Pattern:
This setting determines the specific pattern used for the infill within the 3D printed object. Different patterns can affect the strength, flexibility, and material usage of the printed object.

Infill Wipe Distance:
This setting determines how far the printer will travel to wipe the nozzle after completing an infill pass. This can help prevent excess material from building up on the nozzle and affecting print quality.

Infill Wipe Nozzle Count:
This setting determines how many times the printer will wipe the nozzle after completing an infill pass. This can help ensure that excess material is removed from the nozzle and prevent print quality issues.

Infill Wipe Speed:
This setting determines the speed at which the printer will travel to wipe the nozzle after completing an infill pass. A faster wipe speed can save time, but may not be as effective at removing excess material.

Infill Layer Thickness:
This setting determines the thickness of each layer of infill material. A thicker layer can result in a stronger object, but may also require more material and take longer to print.

Inner Wall Line Width:
This setting determines the width of the lines used to print the inner walls of the object. Adjusting this setting can affect the strength, precision, and finish of the printed object.

Initial Layer Height:
This setting determines the height of the first layer of the object. A thicker initial layer can improve bed adhesion and help prevent warping, but may also affect the quality of the print.

Initial Layer Line Width:
This setting determines the width of the lines used to print the first layer of the object. A wider line width can help ensure that the first layer adheres properly to the print bed.

Initial Layer Print Speed:
This setting determines the speed at which the printer will print the first layer of the object. A slower speed can help ensure that the first layer adheres properly to the print bed and improve print quality.

Infill Print Speed:
The speed at which the printer's nozzle moves when printing infill, which is the internal structure of a printed object.

Infill Travel Speed:
The speed at which the printer's nozzle moves between infill lines.

Ironing Flow:
The flow rate of filament used for ironing, a technique where the printer's nozzle moves over the top layer of a print to smooth out any imperfections.

Ironing Inset:
The distance between the nozzle and the top surface of a print during ironing.

Ironing Line Spacing:
The distance between ironing lines during the ironing process.

Ironing Monotonic:
A setting that ensures that the printer's nozzle only moves in one direction during ironing to prevent any artifacts.

Ironing Only Highest Layer:
A setting that enables ironing only on the top layer of a print.

Ironing Speed:
The speed at which the printer's nozzle moves during ironing.

Jerk:
The sudden change in velocity of the printer's nozzle, which can cause vibrations and ringing in a print.

Layer Height:
The thickness of each layer in a 3D print, which affects the print's resolution and strength.

Line Width:
The width of the lines that the printer's nozzle extrudes, which affects the print's strength and surface finish.

Maximum Resolution:
The maximum level of detail that a printer can achieve.

Maximum Travel Resolution:
The maximum distance that the printer can move accurately, which affects the print's accuracy and quality.

Minimal Layer Time:
The minimum amount of time that each layer of a print should take to ensure that the previous layer has cooled sufficiently and prevent warping.

Minimum Skin Width:
The minimum width of a skin, which is a continuous outer layer of a print that covers the infill.

Nozzle Size:
The nozzle size setting in Cura determines the diameter of the extruder nozzle used to deposit the filament. It is important to choose the correct nozzle size to achieve the desired level of detail and speed for your print.

Octet Infill:
A type of infill that uses a pattern of octagons to fill the internal structure of a print.

Outer Wall Line Width:
The width of the outermost perimeter of a print, which affects the print's surface finish and strength.

Pattern Spacing:
The distance between each repetition of an infill or pattern.

Preview:
The preview mode in Cura allows you to see a 3D representation of your print before you start printing. It can help you to identify any potential issues with the print and make adjustments as necessary.

Print Bed Adhesion:
The method used to ensure that a print adheres to the printer's bed during printing, which affects the print's stability and accuracy.

Print Sequence:
The order in which a printer prints different parts of a print, which affects the print's stability and accuracy.

Print Speed:
The speed at which the printer's nozzle moves when printing, which affects the print's speed and quality.

Print Temperature:
The temperature at which the printer's filament is melted and extruded, which affects the print's strength and surface finish.

Raft:
A raft is a base layer of material that is printed underneath the first layer of the model. It is designed to improve bed adhesion and prevent warping by creating a larger surface area for the print to stick to.

Retraction:
Retraction is the process of pulling the filament back into the extruder to prevent oozing and stringing during printing. It is typically used when the printer moves from one area of the print to another to prevent excess material from being deposited.

Scrolling Mode:
Scrolling mode is a setting that determines how the printer moves between layers. When scrolling mode is enabled, the printer moves diagonally between layers, which can improve print quality by reducing the visibility of layer lines.

Seam Corner Preference:
Seam corner preference determines where the printer begins and ends each layer of the print. Depending on the setting, the printer may start and end layers in a different location, which can affect the appearance of the print.

Shell:
The shell is the outermost surface of a 3D printed object, also known as the perimeters. It is made up of a series of lines printed around the outer edge of the object to create a solid, smooth surface.

Skin Removal Width:
Skin removal width determines the distance between the edge of the model and the outer wall of the print. This setting can be adjusted to remove any thin lines of material that may be visible on the outer surface of the print.

Slicing:
Slicing is the process of converting a 3D model into layers that can be printed. Cura uses a slicing algorithm to create a G-code file that contains the instructions for the printer to follow when printing the object.

Spiralize Outer Contour:
Spiralize outer contour is a setting that causes the printer to print the outermost layer of the model in a continuous spiral, rather than printing each layer separately. This can improve the appearance of the print by reducing the visibility of layer lines.

Speed:
Speed refers to the speed at which the printer moves during printing. This setting can be adjusted to increase or decrease the overall printing time and affect the quality of the print.

Support:
Support is a structure that is printed beneath overhanging parts of the model to prevent them from collapsing. Supports can be manually added to the model or generated automatically by the printer software.

Support Density:
Support density determines how much material is used to print the support structure. Higher densities result in stronger supports but can also increase print time and material usage.

Support Distance Priority:
Support distance priority is a setting that determines how the printer generates support structures for the model. When this setting is enabled, the printer prioritizes creating supports that are farther away from the model.

Support Floor Thickness:
Support floor thickness determines the thickness of the first layer of the support structure. This setting can be adjusted to improve bed adhesion and prevent warping.

Support Horizontal Expansion:
Support horizontal expansion determines how much the support structure extends outward from the model. This setting can be adjusted to improve support strength and prevent the model from sagging.

Support Infill Rate:
Support infill rate determines how much material is used to fill in the support structure. Higher rates result in stronger supports but can also increase print time and material usage.

Support Interface Build Plate Distance:
Support interface build plate distance determines the distance between the support structure and the build plate. This setting can be adjusted to improve bed adhesion and prevent warping.

Support Interface Density:
Support interface density determines how much material is used to create the interface layer between the support structure and the model. Higher densities result in stronger interfaces but can also increase print time and material usage.

Support Interface Line Spacing:
Support interface line spacing determines the distance between each line of material in the interface layer. This setting can be adjusted to improve the strength of the interface layer and prevent it from breaking apart.

Support Interface Line Width:
Support interface line width determines the width of each line of material in the interface layer. This setting can be adjusted to improve the strength of the interface layer and prevent it from breaking apart.

Support Pattern:
This refers to the type of pattern used to create the support structure for overhanging or complex parts of a 3D print. Common support patterns include grid, lines, zigzag, and concentric.

Support Placement:
This refers to where the support structures are placed on a 3D print. This can affect the overall quality of the print and how easy it is to remove the support structures after printing.

Support Roof Thickness:
This is the thickness of the top layer of support structures, which help to hold up overhanging parts of a 3D print.

Support Stair Step Height:
This refers to the height of each step in the support structure, which can affect the overall quality of the print and how easy it is to remove the support structures after printing.

Support Stair Step Maximum Height:
This refers to the maximum height of each step in the support structure, which can affect the overall quality of the print and how easy it is to remove the support structures after printing.

Support Stair Step Maximum Width:
This refers to the maximum width of each step in the support structure, which can affect the overall quality of the print and how easy it is to remove the support structures after printing.

Support Travel Speed:
This refers to the speed at which the print head moves when printing support structures. A higher speed can lead to lower quality prints, while a lower speed can increase the overall printing time.

Support X/Y Distance:
This refers to the distance between the support structures and the object being printed. A larger distance can result in less contact between the support and the object, while a smaller distance can make it more difficult to remove the support structures after printing.

Support Zig Zag:
This is a type of support pattern where the support structures are created in a zigzag pattern. This can provide better stability and support for overhanging parts of a 3D print.

Top/Bottom Line Width:
This refers to the width of the top and bottom layers of a 3D print. A wider line width can lead to stronger prints, but may also reduce the overall detail of the print.

Top/Bottom Print Speed:
This refers to the speed at which the print head moves when printing the top and bottom layers of a 3D print. A faster speed can reduce the overall printing time, but may also reduce the quality of the print.

Top/Bottom Thickness:
This refers to the thickness of the top and bottom layers of a 3D print. A thicker layer can provide more stability and strength, but may also reduce the overall detail of the print.

Tower Bottom Thickness:
This is the thickness of the bottom layer of a tower used for printing multiple small objects at once. A thicker bottom layer can provide more stability for the objects being printed.

Tower Diameter:
This refers to the diameter of the tower used for printing multiple small objects at once. A larger diameter can provide more stability for the objects being printed.

Tower Roof Angle:
This refers to the angle of the top layer of a tower used for printing multiple small objects at once. A steeper angle can make it easier to remove the printed objects from the tower.

Travel:
Travel refers to the movements of the printer's extruder between printing locations. It is important to minimize travel time to reduce the risk of stringing or other defects in the print.

Travel Avoid Distance:
This is the distance the print head moves when traveling between parts of the print. A larger distance can reduce the risk of knocking over the printed object, but may also increase the overall printing time.

Travel Speed:
This refers to the speed at which the print head moves when traveling between parts of the print. A faster speed can reduce the overall printing time, but may also reduce the quality of the print.

Triangle Infill:
Triangle infill is a type of infill pattern where the interior of a 3D print is filled with triangular-shaped structures. This infill pattern can create a strong, rigid print with minimal material usage.

Tri-hexagon Infill:
Tri-hexagon infill is another type of infill pattern where the interior of a 3D print is filled with a combination of triangles and hexagons. This infill pattern can create a strong and lightweight print.

Use Towers:
Use towers is a setting that allows you to create additional support structures to support overhangs or other areas of a model that may need extra support.

Vase Mode:
Vase mode, also known as spiralize outer contour, is a printing mode in which the printer continuously prints a single, continuous wall around the object. It is useful for creating objects with a smooth, seamless surface.

Walls:
The walls are the individual layers that make up the shell or perimeters of the 3D printed object. The thickness of the walls is determined by the wall line count and the wall line width settings in Cura.

Wall Line Count:
Wall line count determines the number of perimeter walls on a 3D print. This setting affects the overall strength and durability of the print.

Wall Line Width:
Wall line width determines the width of each perimeter wall on a 3D print. This setting affects the overall strength and durability of the print.

Wall Thickness:
Wall thickness determines the thickness of each perimeter wall on a 3D print. This setting affects the overall strength and durability of the print.

Z-Hop:
Z-Hop is a setting that controls how far the print head moves up when moving between non-printing areas of the print. This setting can help reduce the risk of the print head hitting previously printed areas of the model, resulting in a smoother print.

Don't for get to Subscribe & Follow me!

@itsMeaDMaDe

YouTube TikTok Instagram