ORNL Slicer 2

User's Guide

Version 1.2

10/27/2021


OAK RIDGE
National Laboratory
MANUFACTURING
DEMONSTRATION
FACILITY

Contents

1 Introduction

ORNL Slicer is a slicing engine and graphical user interface (GUI) used to convert 3D models into specific printing instructions. To do this, the software imports .STL files and slices them into layers to generate toolpaths that are exported as G-Code.

The original ORNL Slicer was initially developed for large format polymer systems. Over the years, the software has grown to support more than fifteen unique machine types and syntaxes. Everything from large industrial systems to small homeowner desktop systems. The ORNL Slicer supports machines printing polymer (filament, resin, and pellet), concrete, and metal (laser wire and wire arc).

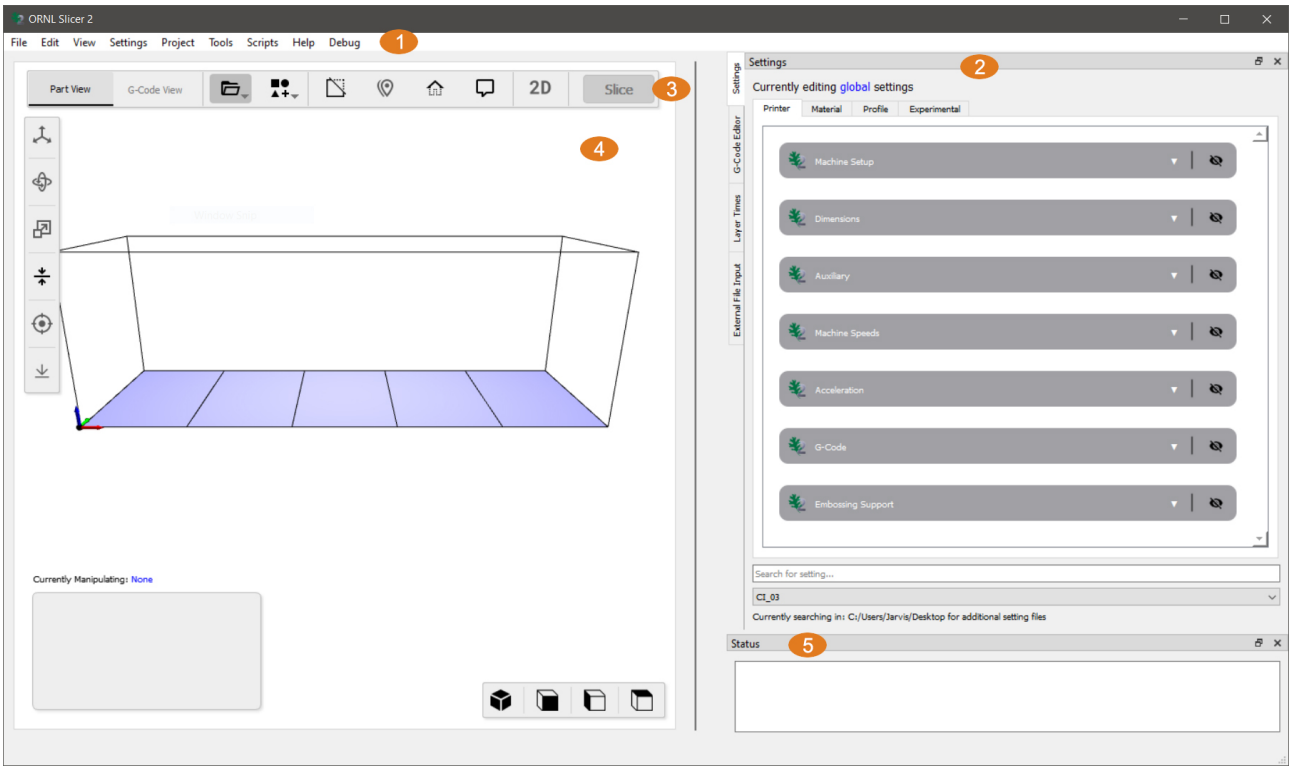
This documentation is for ORNL Slicer 2, and will refer to ORNL Slicer 2 as the Slicer, Slicer 2, ORNL Slicer, and ORNL Slicer 2. These names are synonymous as this is now the only ORNL Slicer. The original ORNL Slicer is no longer under development. All efforts are focused on this package, ORNL Slicer 2.

The ORNL Slicer has plans to be fully open-sourced via Github. This has not yet happened, but we do distribute source code to partners under an NDA. If you have an installable version, we kindly ask that you do not distribute it to anyone.

The ORNL Slicer is in active development. Please report all bugs to roschliac@ornl.gov and/or borishmc@ornl.gov.

2 Navigating the User Interface

2.1 Main Window



1. Menu Bar - See Section 2.1.2 Menu Bar
2. Right Side Panel
    - Settings Toolbar - View and edit Printer, Material, and Profile settings. For more info, see Section 2.1.4 Settings Toolbar.
    - G-Code Editor Toolbar - View and edit the currently loaded G-Code file. For more info, see Section 2.1.5 G-Code Editor Toolbar.
    - Layer Times Toolbar - View the expected layer times for the currently loaded G-Code file. For more info, see Section 2.1.6 Layer Times Toolbar.
    - External File Input Toolbar - Import a .csv file to manipulate the tool-pathing.

3. Visualization Control Bar - Provides functionality to the active visualizer as well as a Slice button and the ability to swap the active visualizer. There are two visualizers:
    - Part View - Align and manipulate STL files within the build volume. For more info, see Section 2.1.2 Part View.
    - G-Code View - Visualize G-Code toolpaths in 3D. For more info, see Section 2.1.3 G-Code View.
4. Visualizer - Used to display solid models or G-Code within the bounds of the defined print volume. There are two visualizers:
    - Part View - Align and manipulate STL files within the build volume. For more info, see Section 2.1.2 Part View.
    - G-Code View - Visualize G-Code toolpaths in 3D. For more info, see Section 2.1.3 G-Code View.
5. Status Toolbar - Outputs information about the current slicing status including estimated print time and weight.

2.1.1 Menu Bar

The menu bar contains 9 different menus: File, Edit, View, Settings, Project, Tools, Scripts, Help, and Debug. Each of these menus and their contents are listed below.

1. File Menu
    - Load Model for Building - Open STL files for building. The hotkey for this action is Ctrl + O.
    - Restore Last Session - Open a project file auto-generated from the previous session with saved STL files, locations, and settings.
    - Slice - Execute the Slicing operation. The hotkey for this action is Ctrl + G.
    - Take Screenshot - Save a screenshot of the active visualizer.
    - Exit - Close the program. The hotkey for this action is Ctrl + Q.
2. Edit Menu
    - Undo - Undo the last operation.

- Redo - Redo the last operation.
- Copy - Copy to the clipboard.
- Paste - Paste from the clipboard.
- Delete - Delete the selected STL file. The hotkey for this action is Ctrl + Del.

3. View Menu

   - Part View - Change the active view to the part view. For more info, see Section 2.1.2 Part View.
   - G-Code View - Change the active view to the G-Code view. For more info, see Section 2.1.3 G-Code View.
   - Reset Camera - Reset camera angle to the default isometric view.
   - Zoom
     - Zoom In - Zoom in on the active viewer tab.
     - Zoom Out - Zoom out on the active viewer tab.
     - Reset - Reset zoom to 100% on the active viewer tab.
   - Hidden Settings
     - Show All Settings - Set all settings categories to visible.
     - Printer Settings - Show a list of all hidden Printer Settings categories. Click on a category in the list to re-add it to the Printer Settings Tab.
     - Material Settings - Show a list of all hidden Material Settings categories. Click on a category in the list to re-add it to the Material Settings Tab.
     - Profile Settings - Show a list of all Profile Settings categories. Click on a category in the list to re-add it to the Profile Settings Tab.
   - Toolbars - This is a list of all the Toolbars within the UI. Toolbars with a check mark next to their name have their view enabled. If no check mark is shown, click the name to enable it.
     - G-Code Editor
     - Layer Time
     - Status
     - Settings
     - Main Control

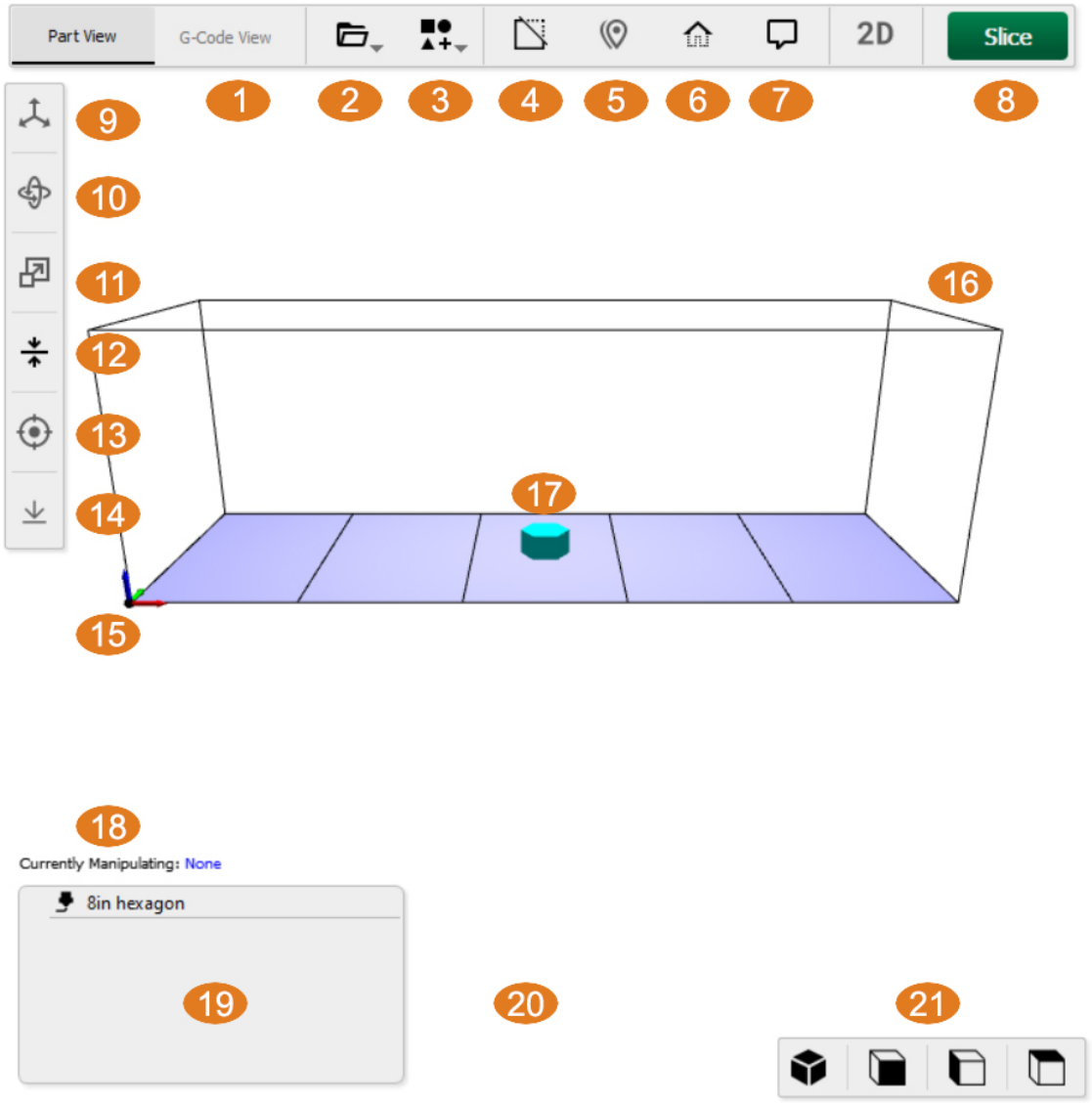
4. Settings Menu
    - Load from Template - Opens a window to select and load an ORNL Slicer 2 settings profile. By default, this window filters to only show files of .s2c, the ORNL Slicer 2 Settings Configuration File Extension.
    - Save as Template - Opens the Save as Template Window. For more info, see Section 2.3.1 Save as Template Window.
    - Additional Setting Location - Opens a window for selecting a folder. All .s2c files within the folder will be loaded in to the active settings list. This folder path will also be displayed at the bottom of the Settings Toolbar. For more info, see Section 2.1.4 Settings Toolbar.
    - Application Preferences - Opens the Preferences Window. For more info, see Section 2.3.2 Preferences Window.
5. Project Menu - For more about working with Projects, see Section 4.4 Saving Projects.
    - New Project - Clears all currently loaded parts to begin a new project file.
    - Load Project - Opens a window select and load an ORNL Slicer 2 project file. By default, this window filters to only show files of .s2p, the ORNL Slicer 2 Project File Extension. The hotkey for this action is Ctrl + Shift + O
    - Save Project - Opens a window to export the current project as a .s2p file. The hotkey for this action is Ctrl + S
    - Import G-Code - Opens a window to select and load a G-Code file to the G-Code Editor and the G-Code Visualizers.
    - Export G-Code - Opens the G-Code/Project Export Window. For more info, see Section 2.3.3 G-Code/Project Export Window.
6. Tools Menu
    - Flowrate Calculator - Opens the Flowrate Calculator Window. For more info, see Section 2.3.4 Flowrate Calculator Window.
    - Xtrude Calculator - Opens the Xtrude Calculator, a flowrate extrusion calculator created by Hybrid Manufacturing Technologies. For more info, see Section 2.3.5 Xtrude Calculator Window.
7. Scripts Menu

- Python Interpreter - A future addition that will allow the user to write Python scripts to interact with the G-Code. Have an interest in using this? Email the developers to prioritize this task.

8. Help Menu
   - User's Manual - Opens this User's Manual as a PDF.
   - Open Website/Repository - Opens the private Github page for ORNL Slicer 2. If you are not logged into Github with an approved account, you will get a Github 404 error.
   - Report Bug - Opens the public Github issue tracker for ORNL Slicer 2 development. For more info, see Section 5.1 Reporting Bugs and Requesting New Features.
   - About ORNL Slicer 2 - Opens an about window that shows the current version of ORNL Slicer 2.
   - About Qt - Opens an about window that shows the version of Qt currently used for development.
9. Debug Menu
   - Run MainWindow::debug() - This feature is currentlty disabled.
   - Cross-section Viewer - Opens the Cross-section Viewer Window. For more info, see Section 2.3.6 Cross-Section Viewer Window.

2.1.2 Part View

Part View is used for loading and orienting STL models within the build volume as building, clipping, and settings models. It is also used to create layer specific settings. For more info about settings models see Section 4.2.4 Adding a Settings Model, and for more info layer specific settings see Section 4.2.2 Layer Specific Settings and Section 4.2.3 Layer Range Settings.



Part View contains the following buttons and items:

1. G-Code View - Switch the viewer to the G-Code View.
2. Load a New Part - Load an STL for building or clipping, or add a settings model.

3. Create Object - Create a box settings region as well as create a rectangular prism or triangular pyramid that can be used as any form of object.

4. Show Slicing Plane - shows a pink plane intersecting the object that represents the plane used for slicing.

5. Show/Hide Optimization Points - Show or Hide markers that display the location of the various seam optimization points. Custom Island Point is in dark blue. Custom Path Point is dark green and the Second Custom Path Point is light green.

6. Show/Hide Support Overhang - When selected, any face of the STL files that exceeds the support overhang threshold will be colored red. This is helpful for quickly looking at an STL to see where supports will be generated during printing.

7. Show/Hide Billboards - When selected, billboards displaying the object name will appear over each object.

8. Slice - Initiate the slicing process. The same as using the hotkey CTRL + G.

9. Translation Controls - Translate the selected STL in X, Y, and Z.

10. Rotation Controls - Rotate the selected STL about Pitch, Yaw, and Roll.

11. Scaling Controls - Scale the selected STL in X, Y, and Z. There is also a drop-down menu to define the units of the STL. STL files don't contain units, so Slicer 2 defaults to millimeters as the assumed unit. Selecting a different unit will scale the STL file accordingly.

12. Alignment Menu - Selecting this button opens a horizontal menu of six alignment buttons. The six buttons are used to align the face of a part in a certain orientation based on the button used: bottom, top, right, left, front, and back. The user selects the alignment button, then selects the STL face to be aligned in that direction. For example, selecting bottom and clicking a face will align the part such that the face is in-plane and sitting atop the build surface.

13. Center Part in Build Volume - Moves the selected STL file to the center of the build volume and places it on the build surface.

14. Drop Part to Floor - Moves the selected STL file to sit directly on the build surface.

15. Coordinate Axes - Displays the X (red), Y (green), and Z (blue) coordinate arrows.
16. Build Volume Wireframe - A wireframe representation of the build volume as defined in the printer settings.
17. STL - A currently loaded STL file within the build volume.
18. Status - Shows what object is currently selected for manipulation.
19. Object List - List of currently loaded STL files with a symbol beside the name to signify if the object is for building or clipping.
20. Warnings - Displays in red text any active warnings such as a part being outside the build volume. If there are no warnings, the area will show no text.
21. View Orientation Buttons - Adjusts the camera angle including isometric, front, side, and top.

Mouse Controls

Selecting STL files and manipulating the camera angle within the Part View is done with the mouse:

- Select Object - STL files can be selected by double left clicking on the object within the viewer or by left clicking on the name in the object list. Object colors change to symbolize what type of object they are. See below for a list of object colors. A selected object can be deleted with the hotkey Ctrl + delete.
- Translate Object - A selected object can be translated by left clicking and dragging the object.
- Rotate Object - A selected object can be translated by right clicking and dragging the object. Note that the rotation auto-snaps to 15 degree intervals.
- Pan View - Hold middle click (clicking the wheel) and move the mouse.
- Rotate View - Hold right click and move the mouse.
- Zoom - Zooming in and out is done by scrolling up and down with the mouse wheel. Zoom can also be adjusted with the hotkeys Ctrl + = and Ctrl + - to zoom in and out respectively. Zoom can also be adjusted from the View Menu on the Menu Bar. For more info, see Section 2.1.1 Menu Bar.

Object Colors

STLs within the part view appear as one of six colors depending upon type of mesh and selected vs unselected.

- Light Blue - STLs that appear blue are objects designated for building that are not selected.
- Green - STLs that appear green are objects designated for building that are selected.
- Red - STLs that appear red are objects designated for clipping that are not selected.
- Orange - STLs that appear orange are objects designated for clipping that are selected.
- Dark Blue - Hollow rectangle objects that appear dark blue are objects designated as settings regions that aren't currently selected.
- Purple - Hollow rectangle objects that appear purple are objects designated as settings regions that are currently selected.

Object List Right Click Menu

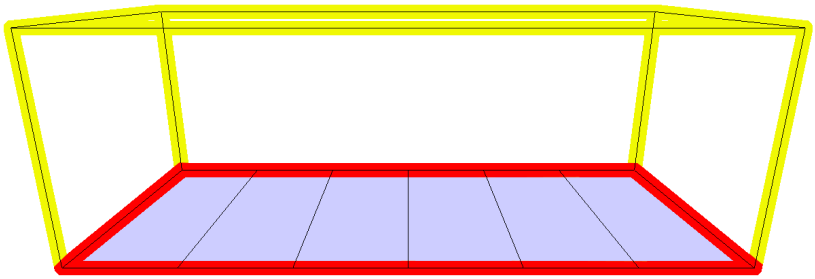
Objects in the object list can be right clicked to bring up a menu. The menu presents the following options:

- Switch to Clipper - Switch the selected object to a clipping model. This option is only available when right-clicking on an object designated for building or setting.
- Switch to Build - Switch the selected object to a building model. This option is only available when right-clicking on an object designated for clipping or setting.
- Switch to Setting - Switch the selected object to a setting model. This option is only available when right-clicking on an object designated for clipping or setting. For more about using settings models see Section 4.2.4 Adding a Settings Model.
- Reset Transformation - Resets all transformations to the object including translation, rotation, and scaling operations.

- Delete Part - Deletes the object from ORNL Slicer 2.
- Transparency - Set the object transparency to make it easier or harder to see.

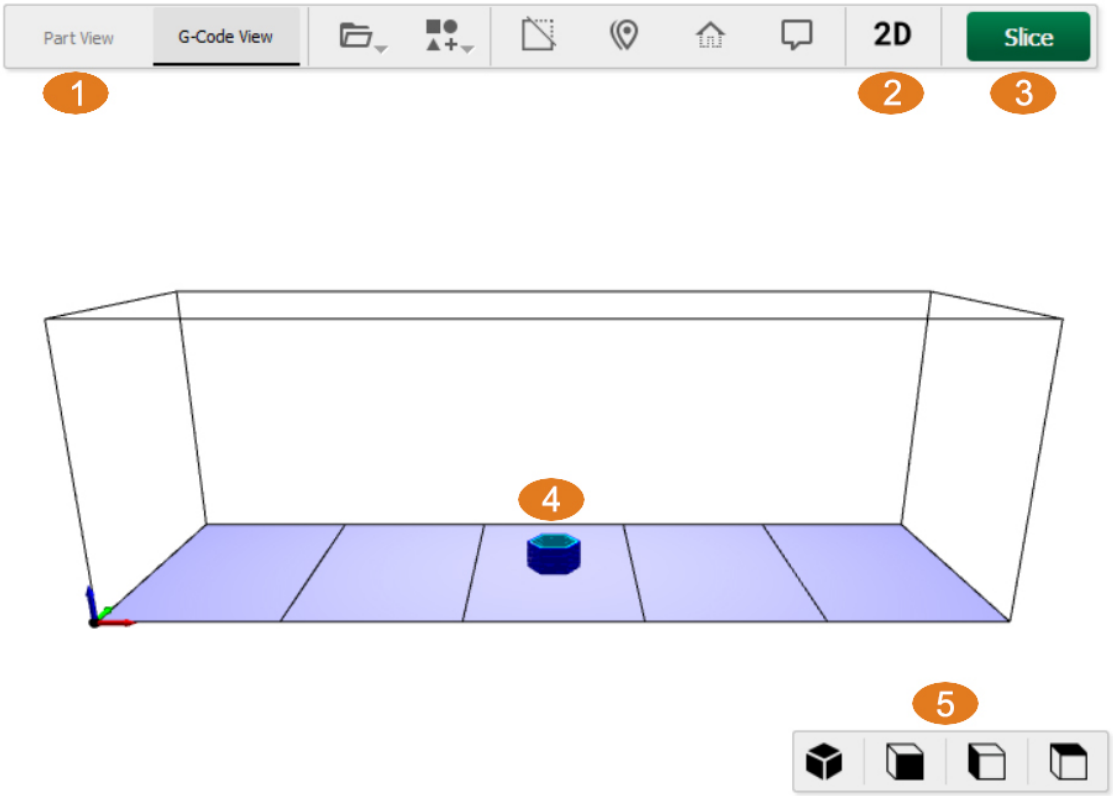
Build Volume vs Build Surface

Build volume and build surface are used repeatedly throughout this guide to explain the visualization of the printer. The build volume (shown outlined with red and yellow lines) encompasses the entire printable volume in X, Y, and Z space. The build volume includes the build surface (shown outlined with just red lines, and highlighted in blue) which is an XY area that is positioned as the bottom surface of the build volume. Both the build volume and build surface are drawn using the dimensions input to the printer settings. The build surface has optional grid lines that can be used to better illustrate the size of the space. For more info, see Section 3.1.2 Dimensions.



2.1.3 G-Code View

The G-Code View is used to visualize G-Code toolpaths in three dimensions within the printer build volume. This can be used to visualize a single layer, or multiple layers by adjusting the minimum and maximum layers at the bottom of the G-Code Editor Toolbar (see Section 2.1.5 G-Code Editor Toolbar). Also at the bottom of the editor toolbar are buttons to hide support and travel which will hide support and travel paths, respectively, from the visualization.



G-Code View contains the following buttons and items:

1. Part View - Switch to the Part View visualizer to manipulate and view models.
2. 2D View - Forces an overhead orthographic 2D view of the toolpaths. Click to enter/exit the 2D view.
3. Slice - Initiate the slicing process. The same as using the hotkey CTRL + G.
4. Toolpaths - Visualized toolpaths appear with the build volume.
5. View Orientation Buttons - Adjusts the camera angle including isometric, front, side, and top.

Mouse Controls

Segments within the visualization can be left clicked to select. This will highlight the segment yellow as well as the corresponding line of G-Code in the editor. Clicking a

line of code in the editor will cause the corresponding segment to be highlighted. Segments in the 2D View cannot be selected from the visualizer, but can be highlighted by selecting the line of code in the editor.

The 3D View can be manipulated in the same ways as the Part View. To pan, hold middle click and move the mouse. To rotate, hold right click and move the mouse. To zoom, scroll up and down with the wheel. Zoom can also be adjusted with the hotkeys Ctrl + = and Ctrl + - to zoom in and out respectively. Zoom can also be adjusted from the View Menu on the Menu Bar. For more info, see Section 2.1.1 Menu Bar.

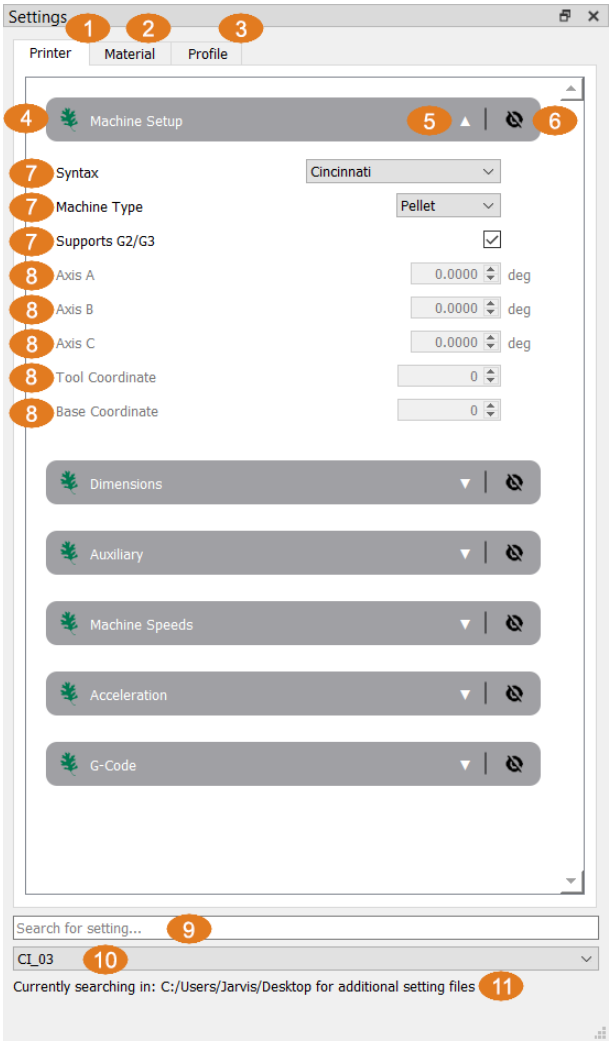
Segment Colors

Every segment in the 2D Viewer and 3D Viewer is color coordinated to the corresponding line of G-Code in the G-Code editor. Colors are set based on path and region type. Here is a full list of all G-Code colors complete with RGB values:

Perimeter (0,0,255)	Prestart (204,0,255)
Infill (0,255,0)	Initial Startup (135,222,205)
Skin (0,128,0)	Slow Down (44,160,137)
Inset (0,204,255)	Tip Wipe (179,128,255)
Travel (233,175,198)	Coasting (211,95,141)
Support (255,102,0)	Spiral Lift (113,55,200)
Support Roof (255,179,128)	Unknown (255,0,0)
Skeleton (160,44,44)	Raft (102,102,102)
Skirt (211,188,95)	Brim (200,113,55)

2.1.4 Settings Toolbar

The settings toolbar is used for viewing and editing settings across three main categories: Printer, Material, and Profile. For detailed descriptions about specific settings and how they are used, see Chapter 3 Settings.



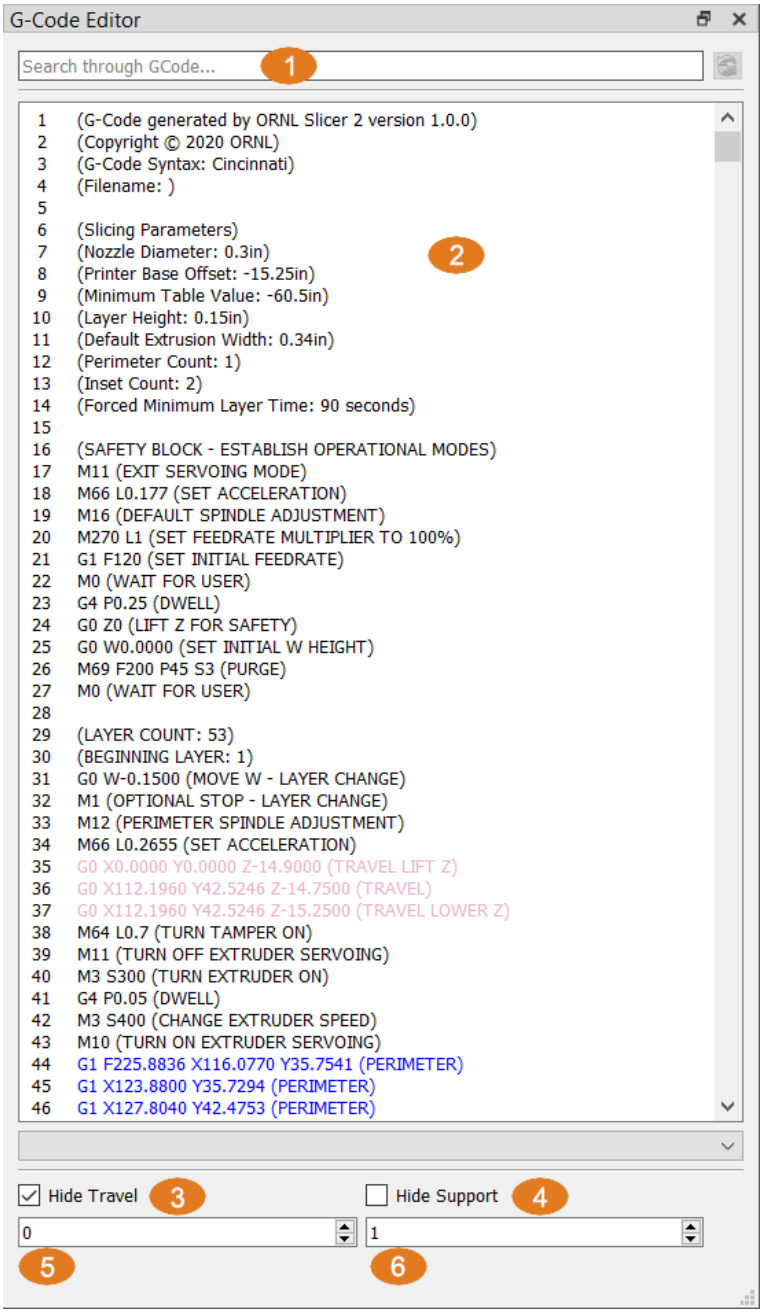
The Settings Toolbar contains the following buttons and items:

1. Printer Settings Category - Contains all the categories and settings for the Printer.

2. Material Settings Category - Contains all the categories and settings for the Material.
3. Profile Settings Category - Contains all the categories and settings for the Profile.
4. Setting Group - A group of similar settings.
5. Show/Collapse Group - Click this arrow to show or collapse the group of settings.
6. Hide Group - Click this eye with a line through it to remove the setting group from the display. The group can be brought back via View Menu -> Hidden Settings. For more info, see Section 2.1.1 Menu Bar.
7. Active Setting - The name of a setting that is active with a value that can be edited. An active setting shows the name in black whereas an inactive setting is gray. Hover the mouse over the setting name to expose the tooltip, a short definition of what the setting does.
8. Inactive Setting - The name of a setting that is inactive with a value that cannot be edited. An active setting shows the name in black whereas an inactive setting is gray. Settings are made inactive by a combination of other settings. To see what settings depend on each other, see Chapter 3 Settings. Hover the mouse over the setting name to expose the tooltip, a short definition of what the setting does.
9. Search for Setting - Type in a string to search for a setting by name.
10. Active Settings File - A dropdown menu showing what settings file is active and what other currently loaded files are available. This menu is unique to each active category such that a different settings file can be set active for each of Printer, Material, and Profile.
11. List of Active Settings Locations - Lists the other location(s) on the computer that settings files are being searched for.

2.1.5 G-Code Editor Toolbar

The G-Code Editor Toolbar is a text editor used to read and edit lines of G-Code. The G-Code Editor is blank until a file is sliced, a G-Code file is imported, or G-Code is manually entered to the editor.



The G-Code Editor Toolbar has the following buttons and items:

1. Search G-Code - Search for specific strings within the G-Code.

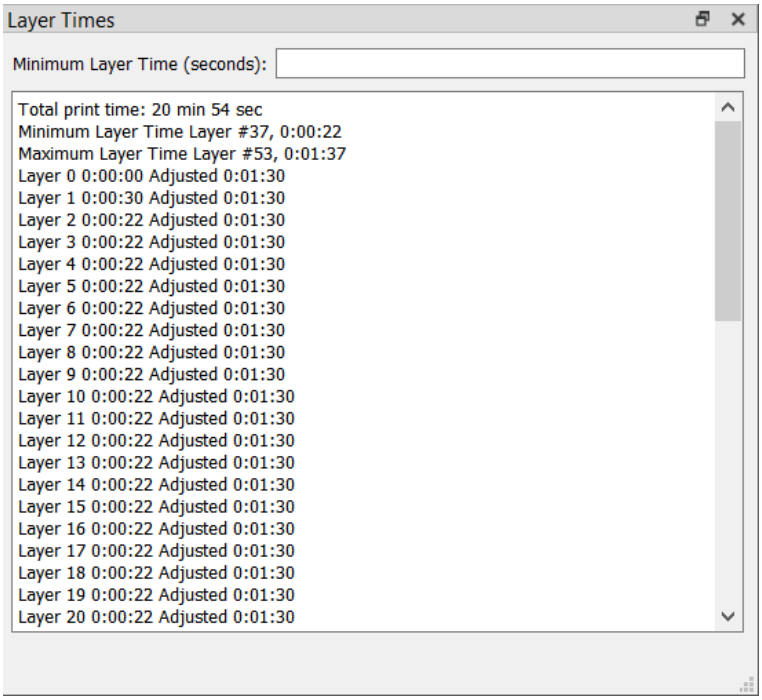
2. G-Code - The currently loaded G-Code file. The code populated here comes from slicing or from importing G-Code. This code can be exported via Project -> Export G-Code.
3. Hide Travel Checkbox - Select this box to hide all travels moves from the visualization. This does not hide them from the G-Code Editor.
4. Hide Support Checkbox - Select this box to hide all support moves from the visualization. This does not hide them from the G-Code Editor.
5. Minimum Layer Number - Set the minimum layer number to appear in the visualization. This doesn't affect the G-Code Editor.
6. Maximum Layer Number - Set the maximum layer number to appear in the visualization. This doesn't affect the G-Code Editor.

Every line of code in the G-Code Editor is color coordinated to the corresponding segment in the 2D Viewer and 3D Viewer. Colors are set based on path and region type. Here is a full list of all G-Code colors complete with RGB values:

Perimeter (0,0,255)	Prestart (204,0,255)
Infill (0,255,0)	Initial Startup (135,222,205)
Skin (0,128,0)	Slow Down (44,160,137)
Inset (0,204,255)	Tip Wipe (179,128,255)
Travel (233,175,198)	Coasting (211,95,141)
Support (255,102,0)	Spiral Lift (113,55,200)
Support Roof (255,179,128)	Unknown (255,0,0)
Skeleton (160,44,44)	Raft (102,102,102)
Skirt (211,188,95)	Brim (200,113,55)

2.1.6 Layer Times Toolbar

The Layer Times Toolbar is used to view the estimated print time for each layer. Data also includes the total print time, minimum layer time, and maximum layer time.
If a minimum layer time is being enforced by the settings, each layer time will also show an adjusted layer time to reflect the actual print time after the adjustment.
A minimum time can be input to the textbox at the top to color layers red that are less than the minimum time. If an adjusted time exists, that value will be used when determining to color red.



2.2 Customizing the Main Window

The Main Window can be customized by reorganizing some of the tabs into different dock locations. The Status Bar, Settings Toolbar, G-Code Editor Toolbar, and Layer Times Toolbar are all capable of being popped out and kept as separate windows or docked in new locations.

A toolbar can be popped out by clicking the overlapping squares in the top right of the toolbar view. This will make the toolbar into a separate window that can be moved to another location on the screen, or another monitor. By dragging this

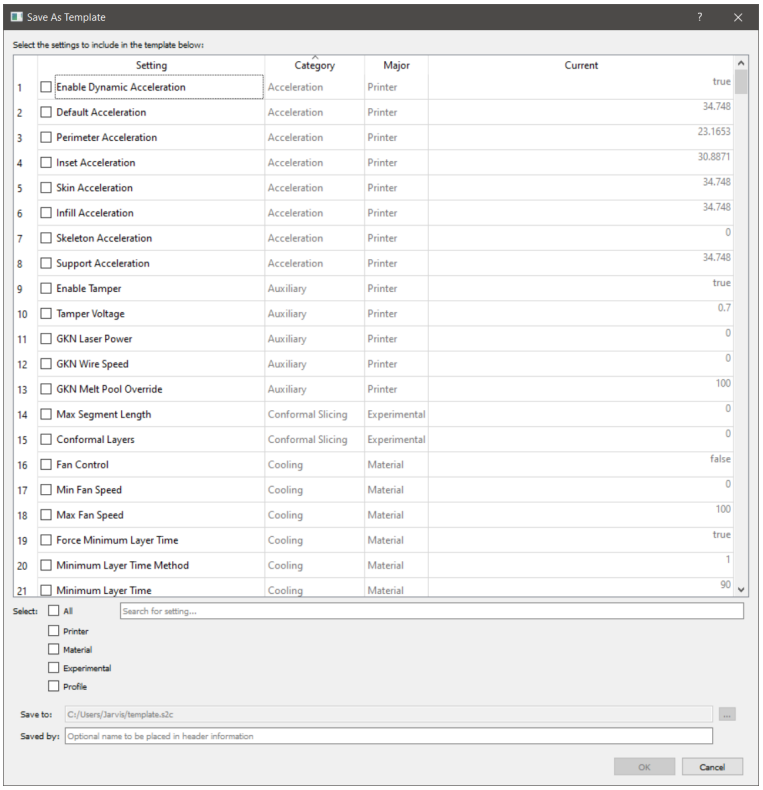
window over the Main Window of the ORNL Slicer 2, areas that are available for docking appear in blue. Available docking locations include the bottom width of the viewer (where the Status Toolbar is located by default), the top width of the viewer, the right side panel (where the Settings, G-Code Editor, and Layer Times toolbars are by default), and above or below the right side panel which is used to split the right side panel into two.

Splitting the right side panel in two can be useful for viewing two toolbars at one time, such as the Settings and G-Code Editor. These can be resized by dragging the divider up and down.

As a reminder, if a window is closed by accident, it's view can be re-enabled from the View Menu along the Menu Bar by going to View -> Toolbars.

2.3 Other Windows

2.3.1 Save as Template Window



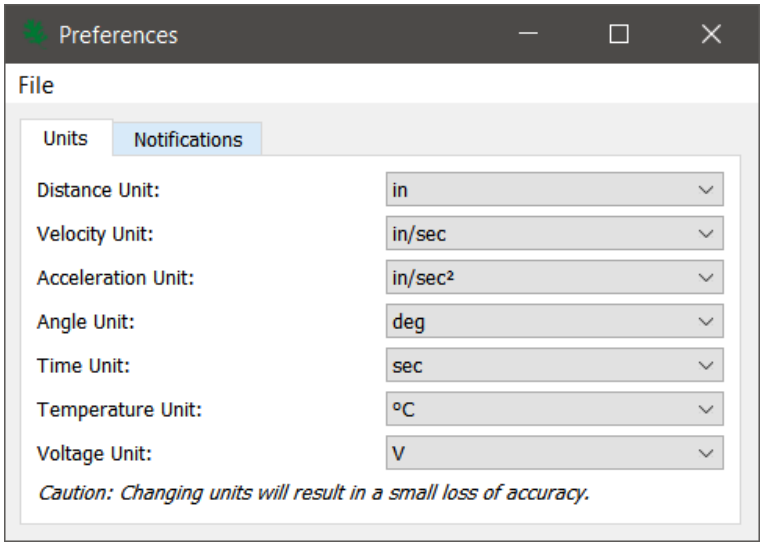
By default, the ORNL Slicer comes preloaded with various settings templates that include settings suggestions that are known to work. A dropdown menu of templates can be seen at the bottom of the settings toolbar. These templates can not be edited and saved. They can be edited during the session, but the changes will be erased when Slicer is restarted. Any edits must be saved as a new template. This is done so that the user will always have settings templates that function properly. To update the default values of an included template, send an email to roschliac@ornl.gov.

The save as template window is used to create new settings templates that are stored as .s2c files on the local machine. It can be opened by going to Settings -> Save as Template. When the window is opened, a list is generated of every setting and the active value. This is populated based on the templates that are currently selected for Printer, Material, and Profile.

To create a new template, you must choose which settings you want to include in the template. Each setting has a checkbox next to it that can be used to include it in the settings to be saved in the new template. There are also radio buttons for each of the four major categories: Printer, Material, Experimental, and Profile as well as a radio button for selecting all settings. There is also a search bar below the list for finding specific settings.

Once all desired settings have been selected for saving as a new template, select the button with the ellipsis (...) to select a file location and input a file name. Finally, pressing the "OK" button will save the new template and load it into the active templates for use.

2.3.2 Preferences Window

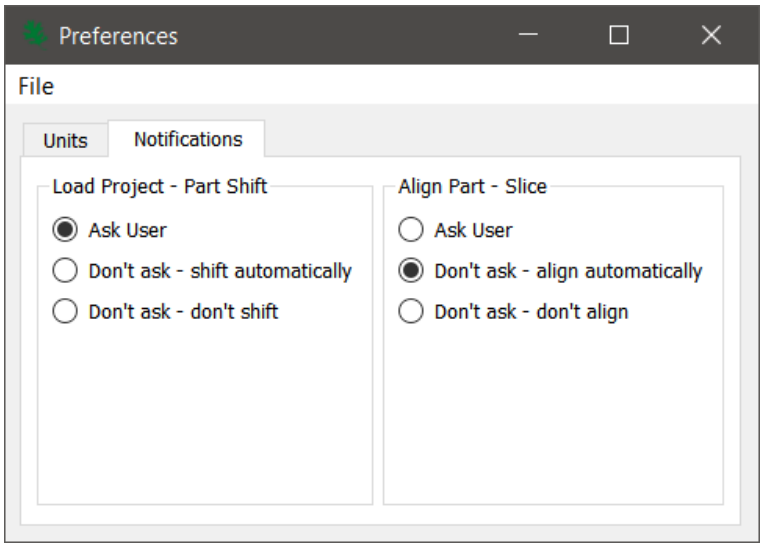


The preferences window is used to define the preferred units and notifications seen throughout the user interface. It can be opened by going to Settings -> Application Preferences. The Units tab, shown above, allows the user to set the desired Distance, Velocity, Acceleration, Angle, Time, Temperature, and Voltage unit. These unit values are used primarily for inputting settings values. The distance and angle values are also used for part positioning within the build volume. Note that these values do not impact the G-Code units. They are just a convenience for the user to more comfortably interact with the values within the interface.

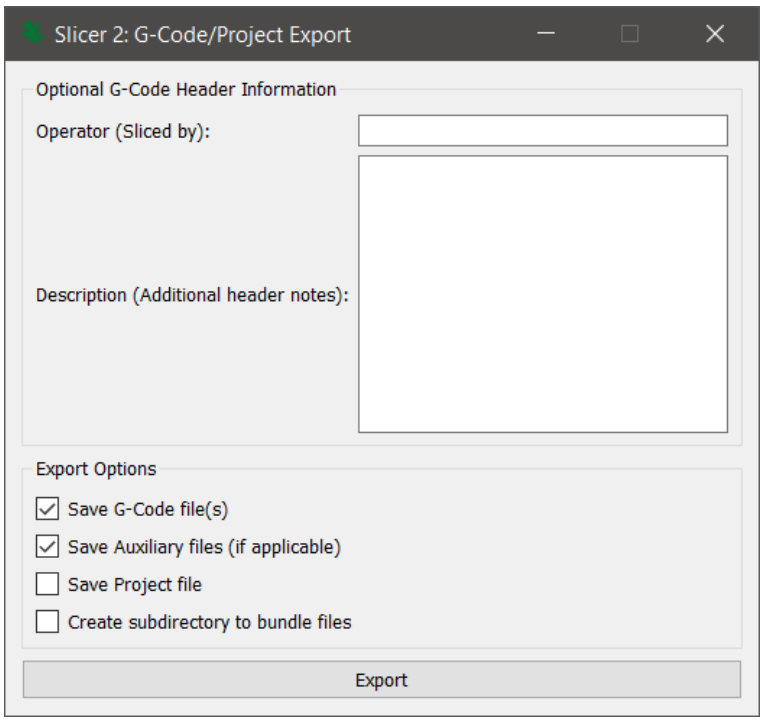
The notifications tab of the preferences, seen below, allows the user to define what popup notifications are seen. The first notification is "Load Project - Part Shift". When a project is saved and exported, the locations of all STL files are saved. Upon reloading the project, the Slicer will place the STL files in their saved locations. If this causes a collision with parts loaded to the Slicer before loading the project, the system can automatically shift the parts being loaded to avoid collisions ("Don't ask - shift automatically"), not shift the parts and maintain all objects in their location with collisions ("Don't ask - don't shift"), or prompt the user with a popup window ("Ask User").

The second notification is "Align Part - Slice". When a part is manipulated within the build volume, it's possible for the part to become unaligned with the build surface, that is to say that the part isn't sitting on the build surface. When this happens and the user presses slice, there are three options. The part can be

automatically aligned with the build surface (”Don’t ask - align automatically”), leave the part as it is in space (”Don’t ask - don’t align”), or prompt the user with a popup window (”Ask User”).



2.3.3 G-Code/Project Export Window



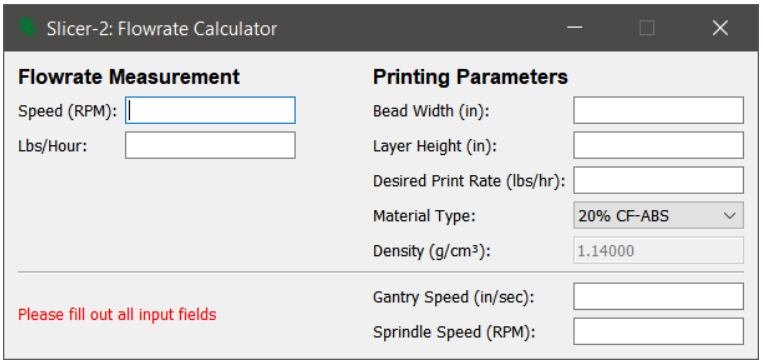
The G-Code/Project Export Window is used for saving the output G-Code and all associated files, as well as the entire project. It can be opened by going to Project -> Export G-Code.

At the top of the window are two input text boxes: operator and description. Both boxes are used to populate fields found in the header of the G-Code file.

At the bottom of the window are four checkboxes and the export button. By default, Save G-Code and Save Auxiliary files are selected. Check the save G-Code file box to export the currently loaded G-Code file. Save Auxiliary files is often not applicable, but it is used for laser scanner ideal files that are needed to operate a laser scanner. Save Project file is used to save the entire project, which is the state of the slicer. The project file includes: all loaded object files and their locations, active settings files, and layer specific settings. The last option, Create subdirectory, is used to create a folder to encapsulate everything that is being exported.

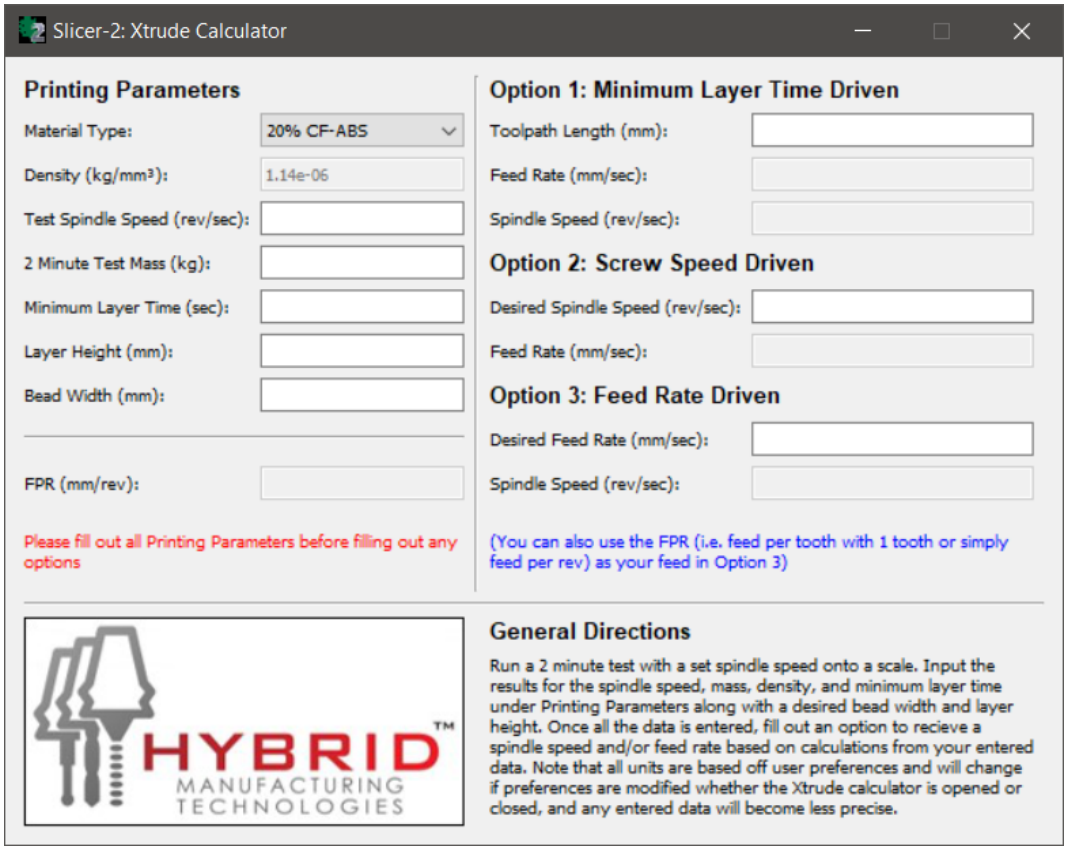
The Export button is used to complete the saving process for all the user desired outputs. Clicking this button will initiate instances of windows save file dialog for each option.

2.3.4 Flowrate Calculator Window



The Flowrate Calculator Window is used to estimate extruder and gantry "feeds and speeds" based on a single datapoint input by the user. It can be opened by going to Tools -> Flowrate Calculator. The left column of the calculator asks for the user to input data from one flowrate measurement: the test extruder speed (in RPM) and the resultant pounds per hour. The right column asks the user to input the desired bead geometry including bead width, layer height, desired print rate, material type OR material density. Once all of the fields have been filled, the suggested gantry speed and extruder spindle speed will be filled in at the bottom two boxes.

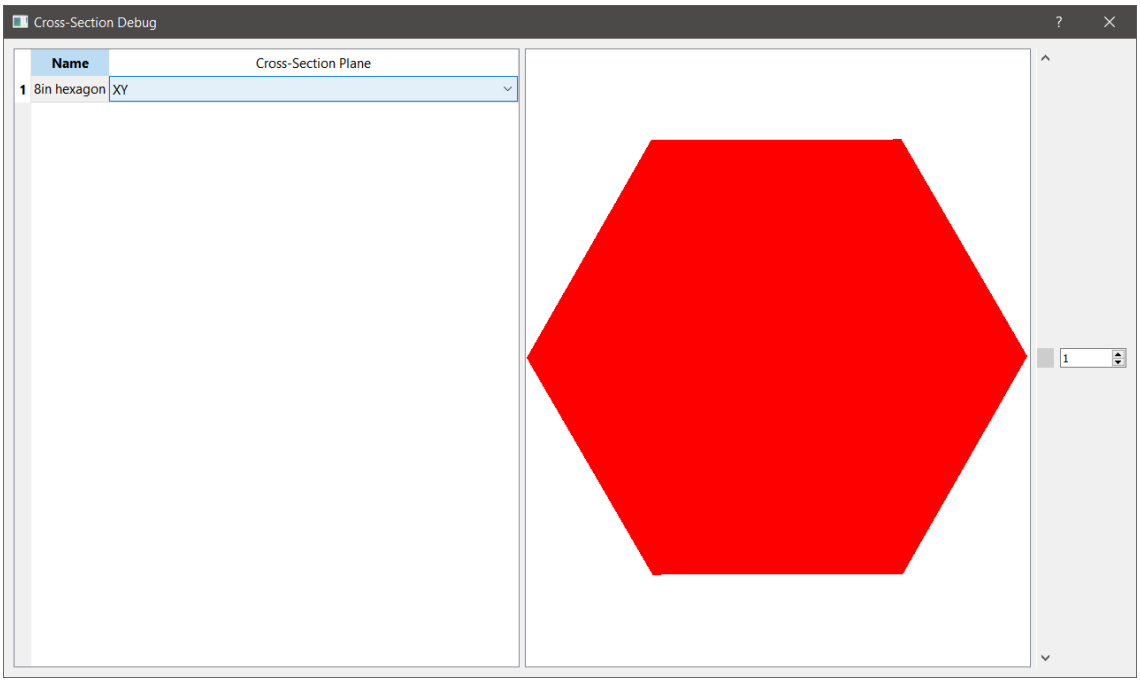
2.3.5 Xtrude Calculator Window



The Xtrude Calculator Window is used to estimate extruder and gantry "feeds and speeds" similar to the Flowrate Calculator. However, the Xtrude calculator requires slightly different input and allows three different options to calculate an output feed rate and spindle speed. The left column of the calculator asks for the user to input data from one flowrate measurement and provide a few desired printing parameters. The right column allows the user to pick three different drivers for calculating the output feedrate and spindle speed: minimum layer time driven, screw speed driven, and feed rate driven. Once all of the fields have been filled, the suggested gantry speed and extruder spindle speed will be filled in at the corresponding boxes.

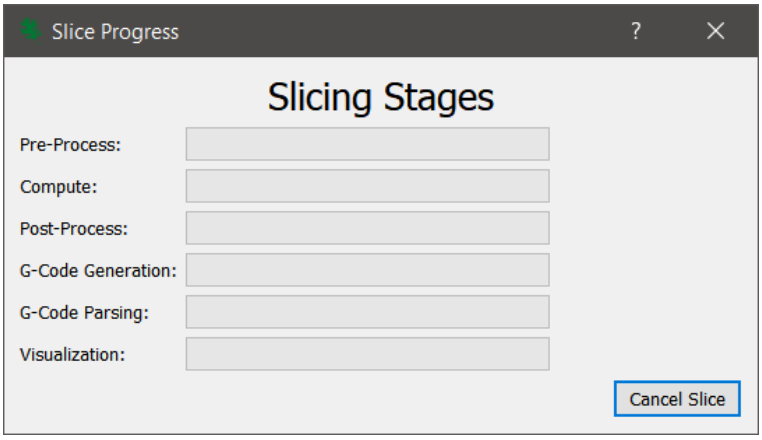
The Xtrude calculator was developed by Hybrid Manufacturing Technologies.

2.3.6 Cross-section Viewer Window



The Cross-section viewer window is meant to be used as a debugging tool. It can be opened by going to Debug -> Cross-section Viewer. When an STL file is loaded to the part view, the Cross-section viewer will show the resultant cross-section of each layer as a polygon. This is useful for helping to "see what the Slicer sees". If a part is not slicing correctly or improper geometry is resulting on a layer or set of layers, the cross section viewer can be used to understand what polygon is created in the slicing step for that layer. Layer number can be adjusted on the right side of the window.

2.3.7 Slice Progress Window



The Slice progress window automatically appears whenever the Slicing operation is initiated. It is meant as a progress window to understand what slicing operation is currently under way and also give the user the ability to cancel slicing.

There are six slicing stages:

1. Pre-Process - this step loads the STL file and location
2. Compute - this step is where slicing into layers occurs. It is also where initial path planning is done.
3. Post-Process - this step is where travels, path modifiers, and path optimizations are applied
4. G-Code Generation - this step creates G-Code from the paths created in the last two steps
5. G-Code Parsing - this step parses all of the resultant G-Code and adds color tags for each type
6. Visualization - this step creates the geometry in the 2D and 3D Viewer from the G-Code that has been parsed

3 Settings

The following subsections will explain every category and setting in the printer, material, profile, and experimental settings. If a setting is only enabled when certain criteria are met, then an "Enabled By" bullet point will be added. If no "Enabled By" bullet point is listed, then the setting should always be enabled. Likewise, there is an "Enables" bullet point to show what settings are enabled the current setting. If no "Enables" bullet point is listed, then the setting does not enable any other settings.

3.1 Printer Settings

3.1.1 Machine Setup

- Syntax
  - **Description:** This is the type or flavor of G-Code that will be output. Syntax is specific to the machine that will be reading the G-Code.
  - **Enables:** Numerous settings in all categories are enabled and disabled based on the active syntax.
- Machine Type
  - **Description:** This defines what type of machine is being used
  - **Options:** Pellet, Filament, Wire Arc, Laser Wire, Concrete, Thermoset
  - **Enables:** Numerous settings are enabled and disabled based on the active machine type. For example, the filament machine type enables the entire filament category.
- Supports G2/G3
  - **Description:** G2 and G3 are arc motion commands. The ORNL Slicer only uses these motions for spiral lifts at this time.
- Axis A
  - **Description:** Used to set the angle for Axis A during print moves on the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.

- Axis B
  - **Description:** Used to set the angle for Axis B during print moves on the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Axis C
  - **Description:** Used to set the angle for Axis C during print moves on the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Tool Coordinate
  - **Description:** Used to set the Tool Coordinate during print moves on the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Base Coordinate
  - **Description:** Used to set the Base Coordinate during print moves on the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Supports E1
  - **Description:** E1 commands are used for table tilting.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Supports E2
  - **Description:** E2 commands are used for table rotation.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.

3.1.2 Dimensions

- Build Volume Type
  - **Description:** Allows the user to determine what shape the printer build volume is. Most printers are rectangle shaped, but other printers (such as Deltas) use a cylindrical build volume.

  - **Options:** Rectangular, Cylindrical, Toroidal
  - **Enables:** The build volume type enables and disables various settings that allow the user to define the dimensions of the machine. For example, Inner and Out Radius are only enabled for toroidal build volumes.

- Minimum X
  - **Description:** Minimum value for the X axis. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for rectangular build volumes.

- Maximum X
  - **Description:** Maximum value for the X axis. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for rectangular build volumes.

- Minimum Y
  - **Description:** Minimum value for the Y axis. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for rectangular build volumes.

- Maximum Y
  - **Description:** Maximum value for the Y axis. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for rectangular build volumes.

- Minimum Z
  - **Description:** Minimum value for the Z axis. This is used to draw the boundary of the printer volume in the visualization.

- Maximum Z
  - **Description:** Maximum value for the Z axis. This is used to draw the boundary of the printer volume in the visualization.

- Inner Radius

  - **Description:** Sets the inner radius of the print volume. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for toroidal build volumes.

- Outer Radius
  - **Description:** Sets the outer radius of the print volume. This is used to draw the boundary of the printer volume in the visualization.
  - **Enabled By:** This setting is only enabled for cylindrical and toroidal build volumes.

- X Offset
  - **Description:** Offsets the X axis position from the minimum X-axis value. The default value of zero places the origin at the center of a rectangular build volume. This setting is often set as the average of the minimum and maximum X values, where the minimum is zero, so that the printer is entirely in quadrant 1.

- Y Offset
  - **Description:** Offsets the Y axis position from the minimum X-axis value. The default value of zero places the origin at the center of a rectangular build volume. This setting is often set as the average of the minimum and maximum Y values, where the minimum is zero, so that the printer is entirely in quadrant 1.

- Z Offset
  - **Description:** Calibrated by measuring the height of the Z axis where the nozzle touches the table. Layer 1 is printed at a Z height of:

$$Zminimum + Zoffset + layerheight$$

- Enable W Axis
  - **Description:** Enables W axis motion which travels in line with the Z-axis.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
  - **Enables:** Enables Minimum and Maximum W settings.

- Minimum W
  - **Description:** Sets the minimum value of the W axis.
  - **Enabled By:** This setting is only enabled when Enable W Axis is selected.
- Maximum W
  - **Description:** Sets the maximum value of the W axis.
  - **Enabled By:** This setting is only enabled when Enable W Axis is selected.
- Layer Change
  - **Description:** Set which axis is moved to transition between layers. Z only will always move just the Z. W only will always move just the W. Z and W will utilize both axes to maximize the amount of travel.
  - **Options:** Z Only, W Only, Z and W
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
- Use Doffing Station
  - **Description:** When selected, the user can input a value to move the W axis to at the end of a print.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
  - **Enables:** This setting enables Doffing Location.
- Doffing Location
  - **Description:** Sets the location to move the W-axis to at the end of a print.
  - **Enabled By:** This setting is only enabled when Use Doffing Station is selected.
- Enable Grid X
  - **Description:** Enables grid lines along the X-axis, traveling in the Y direction.
  - **Enables:** This setting enables the Grid X Distance setting.

- Grid X Distance
  - **Description:** The distance between grid lines along the X-axis.
  - **Enabled By:** This setting is only enabled when Enable Grid X is selected.
- Enable Grid Y
  - **Description:** Enables grid lines along the Y-axis, traveling in the X direction.
  - **Enables:** This setting enables the Grid Y Distance setting.
- Grid Y Distance
  - **Description:** The distance between grid lines along the Y-axis.
  - **Enabled By:** This setting is only enabled when Enable Grid Y is selected.

3.1.3 Auxiliary

- Enable Tamper
  - **Description:** If selected, M-Code commands to turn the tamper on will be inserted whenever the extruder is on.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
  - **Enables:** This setting enables the Tamper Voltage setting.
- Tamper Voltage
  - **Description:** Sets the voltage to be written out in the G-Code that sets the tamper speed.
  - **Enabled By:** This setting is only enabled when Enable Tamper is true.
- GKN Laser Power
  - **Description:** Laser power for GKN printing moves.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- GKN Wire Speed
  - **Description:** Wire speed for GKN printing moves.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.

- GKN Melt Pool Override
  - **Description:** Percentage to change the melt pool size by.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.

3.1.4 Machine Speeds

- Max XY Speed
  - **Description:** User defined maximum XY traverse speed used for speed error checking. The error checking not currently implemented.
- Maximum Extruder Speed
  - **Description:** User defined maximum extruder speed used for speed error checking. The error checking not currently implemented.
- W Table Speed
  - **Description:** Sets the speed for W table motions.
  - **Enabled By:** This setting is only enabled when Enable W Axis is selected.
- Z Speed
  - **Description:** Sets the speed for Z axis motions.
- GKN Print Speed
  - **Description:** Sets the speed for all print moves for the GKN system.
  - **Enabled By:** This setting is only enabled for the GKN Syntax.
- Extruder Gear Ratio
  - **Description:** Sets the gear ratio for the extruder output RPM. This gear ratio is multiplied by any RPM setting when creating the output G-Code.
  - **Enabled By:** This setting is only enabled for the Pellet machine type.

3.1.5 Acceleration

- Enable Dynamic Acceleration
  - **Description:** If selected, custom acceleration rates can be set for each region which will be output as M-Codes.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
- Default Acceleration
  - **Description:** The default acceleration value passed to the G-Code with an M66 command.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.
- Perimeter Acceleration
  - **Description:** The perimeter acceleration value passed to the G-Code with an M66 command at the start of all perimeter paths.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.
- Inset Acceleration
  - **Description:** The inset acceleration value passed to the G-Code with an M66 command at the start of all inset paths.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.
- Skin Acceleration
  - **Description:** The skin acceleration value passed to the G-Code with an M66 command at the start of all skin paths.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.
- Infill Acceleration
  - **Description:** The infill acceleration value passed to the G-Code with an M66 command at the start of all infill paths.

  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.

- Skeleton Acceleration
  - **Description:** The skeleton acceleration value passed to the G-Code with an M66 command at the start of all skeleton paths.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.

- Support Acceleration
  - **Description:** The support acceleration value passed to the G-Code with an M66 command at the start of all support paths.
  - **Enabled By:** This setting is only enabled when Enable Dynamic Acceleration is selected.

3.1.6 G-Code

- Use Default Startup G-Code
  - **Description:** Default startup G-Code is defined specific to each syntax. When this setting is selected, that G-Code will be added to header. The user can also manually add commands via the Start Code setting.
  - The following is a list of all syntaxes and a sample output of the default startup G-Code
    * **Cincinnati** -

```
(SAFETY BLOCK - ESTABLISH OPERATIONAL MODES)
M11 (EXIT SERVOING MODE)
M66 L0.177 (SET ACCELERATION)
M16 (DEFAULT SPINDLE ADJUSTMENT)
M270 L1 (SET FEEDRATE MULTIPLIER TO 100%)
G1 F120 (SET INITIAL FEEDRATE)
M0 (WAIT FOR USER)
G4 P0.2500 (DWELL)
G0 Z0 (LIFT Z FOR SAFETY)
G0 W0.0000 (SET INITIAL W HEIGHT)
M69 F200 P45 S3 (PURGE)
M0 (WAIT FOR USER)
```

* **DMG DMU** -

```
G40 G17 G71 G90
G500
G64
M56
STOPRE
CYCLE800( )
TRAFOOF
L_FREI
L_ZYM91
SIMUL:
L_TIME(0)
M56
T="TPE_2"
M06
D1 ;G43 EQUIVALENT
G90 G54
G0 X0.5 Y0.5
G0 Z100.
G187 P1
```

* **GKN** -

```
&ACCESS RVP
&REL 5
DEF GKN_gcode()
decl real lsr_pwr
decl real wr_speed
decl real print_speed
decl real scan_speed
decl real rapid_speed
;FOLD INI;%[PE]
;FOLD BASISTECH INI
GLOBAL INTERRUPT DECL 3 WHEN $STOPMESS==TRUE DO IR_STOPM
( )
INTERRUPT ON 3
BAS (#INITMOV,0 )
;ENDFOLD (BASISTECH INI)
;FOLD USER INI
```

```
;Make your modifications here
;ENDFOLD (USER INI)
;ENDFOLD (INI)
;Default Values
lsr_pwr = 0.6286
wire_speed = 0.337
print_speed = 0.01
rapid_speed = 0.08
scan_speed = 0.1
$APO.CVEL=100
$ACC.CP=10
;FOLD PTP P1 Vel=10 % PDAT1 Tool[2]:Side Feed Base[0];%[PE]%R
8.3.40,%MKUKATPBASIS,%CMOVE,%VPTP,%P 1:PTP, 2:P1, 3:,
5:10, 7:PDAT1
$BWDSTART=FALSE
PDAT_ACT=PPDAT1
FDAT_ACT=FP1
BAS(#PTP_PARAMS,10)
PTP XP1
;ENDFOLD
;start preparation xxxxxxxxxxxxxxxxxxxxxxxxxxxxxxx
M1() ;initialize lasers
M2() ;laser program select
M3() ;argon gas on
M8(FALSE) ;turn keyence laser scanner shutter off
M0(TRUE) ; wire cutoff
```

* **Gudel** -

```
TRAORI
FFWON
SOFT
G642
G54
G187 P1
```

* **Haas Inch** -

```
M06 T2
G90G00G54X0.Y0.
```

```
G43H2Z100.
G187 P1
```

* **Haas Metric** -

```
M06 T2
G90G00G54X0.Y0.
G43H2Z100.
G187 P1
```

* **Haas Metric No Comments** -

```
M06 T2
G90G00G54X0.Y0.
G43H2Z100.
G187 P1
```

* **Hurco** -

```
T2M6
G00G90G21
G54
(toolbegin)
T2
M06
G90
G43H2Z100.
G05.3P50.
G1 X0 Y0 F4000 (MOVE TO ORIGIN)
G187 P1
```

* **Marlin** -

```
M140 S100 ;SET BED TEMPERATURE
M190 S100 ;SET BED TEMPERATURE AND WAIT
M104 S274.999 T0 ;SET EXTRUDER 0 TEMPERATURE
M109 S274.999 T0 ;SET EXTRUDER 0 TEMPERATURE AND WAIT
G28 ;TRAVEL HOME ALL AXES
G91 ;USE RELATIVE EXTRUSION DISTANCES
G92 E0 ;RESET FILAMENT AXIS TO 0
```

* **Marlin Pellet** -

```
M140 S100 ;SET BED TEMPERATURE
M190 S100 ;SET BED TEMPERATURE AND WAIT
M104 S274.999 T0 ;SET EXTRUDER 0 TEMPERATURE
M104 S274.999 T0 ;SET EXTRUDER 0 TEMPERATURE AND WAIT
T0
G28 ;TRAVEL HOME ALL AXES
M102 S0 ;USE RPM MODE
```

* **Mazak** -

```
#981 = 800.0 (FEEDRATE DURING BURN)
<BASIC_FRAMEWORK>()
(*******)
#901 = 4000.0 (INITIAL LASER POWER)
#902 = 20.0 (SHIELDING LEAVE ALONE)
#903 = 20.0 (SHIELDING LEAVE ALONE)
#904 = 125.0 (F-WIRE FEED SPEED)
#905 = 400.0 (H-HOT WIRE POWER)
(*******)
G57 (4th WORKPIECE)
G461 (WORK SHIFT FOR AM HEAD CANCEL)
(****** SET BS 9 TO RUN WITH LASER OFF *****)
/9(CHECK LASER STATUS BEGIN)
/9IF[#904900EQ0] GOTO110
/9GOTO120
/9N110
/9#3000=72(PRESS_LASER_READY_KEY->RESTART)
/9M30
/9N120
/9(CHECK LASER STATUS END)
(*******)
G90 G80 G69 G21 G17 G49 G94 G97 G40
G91 G28 Z0.
G91 G28 B0.  C0.
G91 G28 X0.  Y0.
(AM CODE BEGIN)
M444 (CLADDING DOOR OPEN)
G0 G90 W-500.
/9 M613 (MIST COLLECTOR ON)
```

```
/9 G910E#901F#904H#905I#902J#903 (SET LASER POWER)
(******CLADDING POWER******)
(E: LASER POWER [W])
(J: SHIELD G [L/min])
(********************)
G460 (WORK SHIFT ON FOR AM HEAD)
G0 G17 G90 G94 G57
G40 G80 G69
G49
(UNCLAMP ROTARIES)
M46
M43
G0 G90 B0.  C0.
(**** COMMENT OUT G61.1 FOR STRAIGHT LINES)
G61.1
(**** USE G68.2 TO ROTATE PATH)
(G68.2 XYZI90.JK)
(G53.1 P1)
(CALL G43.4 TCPC FOR LASER HEAD)
G442 (LASER OFF)
G187 P1
```

* **MVP** -

```
G54 ;ENABLE W01
G1 F30.0000 Z0 ;SET INITIAL TABLE HEIGHT
```

* **RomiFanuc** -

```
T2
M06
G90G00G54X0.Y0.
G43H2Z100.
G187 P1
```

* **Siemens** -

```
;SAFETY BLOCK - ESTABLISH OPERATIONAL MODES
G643 ;
G90 ;USE ABSOLUTE COORDINATES
G54 ;WORK OFFSET
```

    * **SkyBAAM** -
      `There is no default startup G-Code for SkyBAAM`

- Material Load
  - **Description:** Used to add an initial purge to the header of the G-Code used to prime the extruder.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
  - **Enables:** This setting enables the initial purge settings in Materials Settings -> Purge.
- Wait for User
  - **Description:** M0 commands are inserted to the header of the G-Code file so that the operator must manually press the cycle start to advance the print through the initial startup.
  - **Enabled By:** This setting is only enabled for the Cincinnati Syntax.
- Demo Bounding Box
  - **Description:** At the start of the print, travel moves will be added to outline the minimum and maximum x and y values, thus creating a bounding box for where the print will occur.
- Start Code
  - **Description:** Custom code input by the user that will be added to the header of the G-Code file, right before the first travel and print moves.
- Layer Change Code
  - **Description:** Custom code input by the user that will be added to the layer changes of the G-Code file.
- End Code
  - **Description:** Custom code input by the user that will be added to the footer of the G-Code file, before the end of G-Code command.

3.2 Material Settings

3.2.1 Density

- Printing Material
  - **Description:** Select the material being used for the print. The density of this material is used to calculate the expected print weight. The other option should be selected to manually enter a density. To get a material added to the list permanently, or to get a density value updated, email the help team.
  - **Options:** The following are materials available to be selected along with their programmed density value. Numbers denote a percentage of filler material by weight. I.e. ABS20CF is ABS with 20% Carbon Fiber by weight
    - ABS20CF - 1.14 g/cm^3
    - ABS - 1.07 g/cm^3
    - PPS - 1.35 g/cm^3
    - PPS50CF - 1.53 g/cm^3
    - PPSU - 1.29 g/cm^3
    - PPSU25CF - 1.38 g/cm^3
    - PESU - 1.37 g/cm^3
    - PESU25CF - 1.47 g/cm^3
    - PLA - 1.25 g/cm^3
    - Concrete - 2.60 g/cm^3
  - **Enables:** When Other is selected, the Other Density setting becomes available.
- Other Density
  - **Description:** Input the exact density to be used for weight estimation of the toolpaths.
  - **Enabled By:** This setting is enabled when "Other" is selected from the Material Density dropdown list.

3.2.2 Start-Up

Start-up is used for modifying the behavior at the start of an extrusion path, typically to slow the start so the bead properly adheres or so that the extrusion can build pressure. The user defines a length for the start-up, a speed for the extruder, and speed for the motion. For example: if the user enabled perimeter start-up, set a distance of 4 inches, a speed of 5in/s, and an extruder RPM of 100, then the first 4 inches of every perimeter path would be printed at 100 RPM and 5in/s. Following the first 4 inches, the standard perimeter settings would be applied. The transition is seamless, without any pausing or traveling.

Ramp-Up is an option that can be added to the start-up motion for each region. Ramp-Up allows for multiple steps to be applied when transitioning from the start-up speed to the printing speed. The number of steps is defined by the user and is used to break the start-up distance into equal distance steps.

The settings are broken out for each region (Perimeter, Inset, Skin, and Infill) allowing for a custom behavior within each.

- Enable Perimeter Startup
  - **Description:** Enable start-ups for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
  - **Enables:** This setting enables Perimeter Start-Up Distance, Perimeter Start-Up Extruder Speed, and Perimeter Start-Up Speed.
- Perimeter Start-Up Distance
  - **Description:** Set the length of the start-up for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Start-Up is enabled.
- Perimeter Start-Up Speed
  - **Description:** Set the movement speed of the start-up for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Start-Up is enabled.
- Perimeter Start-Up Extruder Speed
  - **Description:** Set the extruder speed of the start-up for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Start-Up is enabled.
- Enable Perimeter Start-up Ramp-Up

  - **Description:** Enables ramp-up capability for perimeter start-ups wherein a series of steps are applied to ramp-up the gantry and extruder speeds.
  - **Enabled By:** This setting is enabled when Perimeter Start-Up is enabled.

- Perimeter Start-up Ramp-Up Steps
  - **Description:** Sets the number of steps taken to ramp-up the extruder and gantry speeds during the perimeter start-up ramp-up motion.
  - **Enabled By:** This setting is enabled when Perimeter Start-Up Ramp-Up is enabled.

- Enable Inset Startup
  - **Description:** Enable start-ups for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
  - **Enables:** This setting enables Inset Start-Up Distance, Inset Start-Up Extruder Speed, and Inset Start-Up Speed.

- Inset Start-Up Distance
  - **Description:** Set the length of the start-up for inset paths.
  - **Enabled By:** This setting is enabled when Inset Start-Up is enabled.

- Inset Start-Up Speed
  - **Description:** Set the movement speed of the start-up for inset paths.
  - **Enabled By:** This setting is enabled when Inset Start-Up is enabled.

- Inset Start-Up Extruder Speed
  - **Description:** Set the extruder speed of the start-up for inset paths.
  - **Enabled By:** This setting is enabled when Inset Start-Up is enabled.

- Enable Inset Start-up Ramp-Up
  - **Description:** Enables ramp-up capability for inset start-ups wherein a series of steps are applied to ramp-up the gantry and extruder speed.
  - **Enabled By:** This setting is enabled when Inset Start-Up is enabled.

- Inset Start-up Ramp-Up Steps

  - **Description:** Sets the number of steps taken to ramp-up the extruder and gantry speeds during the inset start-up ramp-up motion.
  - **Enabled By:** This setting is enabled when Inset Start-Up Ramp-Up is enabled.

- Enable Skin Startup
  - **Description:** Enable start-ups for all skin paths.
  - **Enabled By:** This setting is enabled when skins are enabled.
  - **Enables:** This setting enables Skin Start-Up Distance, Skin Start-Up Extruder Speed, and Skin Start-Up Speed.

- Skin Start-Up Distance
  - **Description:** Set the length of the start-up for skin paths.
  - **Enabled By:** This setting is enabled when Skin Start-Up is enabled.

- Skin Start-Up Speed
  - **Description:** Set the movement speed of the start-up for skin paths.
  - **Enabled By:** This setting is enabled when Skin Start-Up is enabled.

- Skin Start-Up Extruder Speed
  - **Description:** Set the extruder speed of the start-up for skin paths.
  - **Enabled By:** This setting is enabled when Skin Start-Up is enabled.

- Enable Skin Start-up Ramp-Up
  - **Description:** Enables ramp-up capability for skin start-ups wherein a series of steps are applied to ramp-up the gantry and extruder speeds.
  - **Enabled By:** This setting is enabled when Skin Start-Up is enabled.

- Skin Start-up Ramp-Up Steps
  - **Description:** Sets the number of steps taken to ramp-up the extruder and gantry speeds during the skin start-up ramp-up motion.
  - **Enabled By:** This setting is enabled when Skin Start-Up Ramp-Up is enabled.

- Enable Infill Startup
  - **Description:** Enable start-ups for all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.
  - **Enables:** This setting enables Infill Start-Up Distance, Infill Start-Up Extruder Speed, and Infill Start-Up Speed.
- Infill Start-Up Distance
  - **Description:** Set the length of the start-up for infill paths.
  - **Enabled By:** This setting is enabled when Infill Start-Up is enabled.
- Infill Start-Up Speed
  - **Description:** Set the movement speed of the start-up for infill paths.
  - **Enabled By:** This setting is enabled when Infill Start-Up is enabled.
- Infill Start-Up Extruder Speed
  - **Description:** Set the extruder speed of the start-up for infill paths.
  - **Enabled By:** This setting is enabled when Infill Start-Up is enabled.
- Enable Infill Start-up Ramp-Up
  - **Description:** Enables ramp-up capability for infill start-ups wherein a series of steps are applied to ramp-up the gantry and extruder speeds.
  - **Enabled By:** This setting is enabled when Infill Start-Up is enabled.
- Infill Start-up Ramp-Up Steps
  - **Description:** Sets the number of steps taken to ramp-up the extruder and gantry speeds during the infill start-up ramp-up motion.
  - **Enabled By:** This setting is enabled when Infill Start-Up Ramp-Up is enabled.
- Enable Skeleton Startup
  - **Description:** Enable start-ups for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton is enabled.

  - **Enables:** This setting enables Skeleton Start-Up Distance, Skeleton Start-Up Extruder Speed, and Skeleton Start-Up Speed.

- Skeleton Start-Up Distance
  - **Description:** Set the length of the start-up for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Start-Up is enabled.
- Skeleton Start-Up Speed
  - **Description:** Set the movement speed of the start-up for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Start-Up is enabled.
- Skeleton Start-Up Extruder Speed
  - **Description:** Set the extruder speed of the start-up for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Start-Up is enabled.
- Enable Skeleton Start-up Ramp-Up
  - **Description:** Enables ramp-up capability for skeleton start-ups wherein a series of steps are applied to ramp-up the gantry and extruder speeds.
  - **Enabled By:** This setting is enabled when Skeleton Start-Up is enabled.
- Skeleton Start-up Ramp-Up Steps
  - **Description:** Sets the number of steps taken to ramp-up the extruder and gantry speeds during the skeleton start-up ramp-up motion.
  - **Enabled By:** This setting is enabled when Skeleton Start-Up Ramp-Up is enabled.

3.2.3 Slow Down

Slow Down is the companion to Start-Up and is used for modifying the behavior at the end of an extrusion path, typically to slow the ending to prevent over-extrusion. The user defines a length for the slow down, a lift height (optional), a speed for the motion, a speed for the extruder, and a distance from the end to cutoff the extruder (optional).

For example: if the user enabled perimeter slow down, set a distance of 3 inches, lift height of 1 inch, a speed of 2in/s, an extruder RPM of 50, and a cutoff distance

of 1 inch, then 3 inches before the end of every perimeter path the extruder would be adjusted 50 RPM and gantry to 2in/s. There would be a lift of 1 inch during this three inches of travel and 1 inch from the end of the three inches the extruder would turn off. Prior to the first 3 inches, the standard perimeter settings would be applied. The transition is seamless, without any pausing or traveling. Coasting is the term applied to a slow down path where the extruder is disabled, or set to 0 RPM, such as the cutoff distance section of the slow down (or the entire slow down if the slow down extruder speed is set to 0 RPM).

The settings are broken out for each region (Perimeter, Inset, Skin, and Infill) allowing for a custom behavior within each.

- Enable Perimeter Slow Down
  - **Description:** Enable slow downs for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
  - **Enables:** This setting enables Perimeter Slow Down Distance, Perimeter Slow Down Extruder Speed, and Perimeter Slow Down Speed.
- Perimeter Slow Down Distance
  - **Description:** Set the length of the slow down for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Slow Down is enabled.
- Perimeter Slow Down Lift Distance
  - **Description:** Set the distance to lift during the slow down for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Slow Down is enabled.
- Perimeter Slow Down Speed
  - **Description:** Set the movement speed of the slow down for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Slow Down is enabled.
- Perimeter Slow Down Extruder Speed

  - **Description:** Set the extruder speed of the slow down for perimeter paths. If this value is set to 0, the path is named a Coast.
  - **Enabled By:** This setting is enabled when Perimeter Slow Down is enabled.

- Perimeter Slow Down Extruder Cutoff Distance
  - **Description:** Set the length before the end of the perimeter path to turn the extruder off.
  - **Enabled By:** This setting is enabled when Perimeter Slow Down is enabled.

- Enable Inset Slow Down
  - **Description:** Enable slow downs for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
  - **Enables:** This setting enables Inset Slow Down Distance, Inset Slow Down Extruder Speed, and Inset Slow Down Speed.

- Inset Slow Down Distance
  - **Description:** Set the length of the slow down for inset paths.
  - **Enabled By:** This setting is enabled when Inset Slow Down is enabled.

- Inset Slow Down Lift Distance
  - **Description:** Set the distance to lift during the slow down for inset paths.
  - **Enabled By:** This setting is enabled when Inset Slow Down is enabled.

- Inset Slow Down Speed
  - **Description:** Set the movement speed of the slow down for inset paths.
  - **Enabled By:** This setting is enabled when Inset Slow Down is enabled.

- Inset Slow Down Extruder Speed
  - **Description:** Set the extruder speed of the slow down for inset paths. If this value is set to 0, the path is named a Coast.
  - **Enabled By:** This setting is enabled when Inset Slow Down is enabled.

- Inset Slow Down Extruder Cutoff Distance
  - **Description:** Set the length before the end of the inset path to turn the extruder off.
  - **Enabled By:** This setting is enabled when Inset Slow Down is enabled.
- Enable Skin Slow Down
  - **Description:** Enable slow downs for all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.
  - **Enables:** This setting enables Skin Slow Down Distance, Skin Slow Down Extruder Speed, and Skin Slow Down Speed.
- Skin Slow Down Distance
  - **Description:** Set the length of the slow down for skin paths.
  - **Enabled By:** This setting is enabled when Skin Slow Down is enabled.
- Skin Slow Down Lift Distance
  - **Description:** Set the distance to lift during the slow down for skin paths.
  - **Enabled By:** This setting is enabled when Skin Slow Down is enabled.
- Skin Slow Down Speed
  - **Description:** Set the movement speed of the slow down for skin paths.
  - **Enabled By:** This setting is enabled when Skin Slow Down is enabled.
- Skin Slow Down Extruder Speed
  - **Description:** Set the extruder speed of the slow down for skin paths. If this value is set to 0, the path is named a Coast.
  - **Enabled By:** This setting is enabled when Skin Slow Down is enabled.
- Skin Slow Down Extruder Cutoff Distance
  - **Description:** Set the length before the end of the skin path to turn the extruder off.
  - **Enabled By:** This setting is enabled when Skin Slow Down is enabled.

- Enable Infill Slow Down
  - **Description:** Enable slow downs for all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.
  - **Enables:** This setting enables Infill Slow Down Distance, Infill Slow Down Extruder Speed, and Infill Slow Down Speed.
- Infill Slow Down Distance
  - **Description:** Set the length of the slow down for infill paths.
  - **Enabled By:** This setting is enabled when Infill Slow Down is enabled.
- Infill Slow Down Lift Distance
  - **Description:** Set the distance to lift during the slow down for infill paths.
  - **Enabled By:** This setting is enabled when Infill Slow Down is enabled.
- Infill Slow Down Speed
  - **Description:** Set the movement speed of the slow down for infill paths.
  - **Enabled By:** This setting is enabled when Infill Slow Down is enabled.
- Infill Slow Down Extruder Speed
  - **Description:** Set the extruder speed of the slow down for infill paths. If this value is set to 0, the path is named a Coast.
  - **Enabled By:** This setting is enabled when Infill Slow Down is enabled.
- Infill Slow Down Extruder Cutoff Distance
  - **Description:** Set the length before the end of the infill path to turn the extruder off.
  - **Enabled By:** This setting is enabled when Infill Slow Down is enabled.
- Enable Skeleton Slow Down
  - **Description:** Enable slow downs for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton is enabled.
  - **Enables:** This setting enables Skeleton Slow Down Distance, Skeleton Slow Down Extruder Speed, and Skeleton Slow Down Speed.

- Skeleton Slow Down Distance
  - **Description:** Set the length of the slow down for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Slow Down is enabled.
- Skeleton Slow Down Lift Distance
  - **Description:** Set the distance to lift during the slow down for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Slow Down is enabled.
- Skeleton Slow Down Speed
  - **Description:** Set the movement speed of the slow down for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Slow Down is enabled.
- Skeleton Slow Down Extruder Speed
  - **Description:** Set the extruder speed of the slow down for skeleton paths. If this value is set to 0, the path is named a Coast.
  - **Enabled By:** This setting is enabled when Skeleton Slow Down is enabled.
- Skeleton Slow Down Extruder Cutoff Distance
  - **Description:** Set the length before the end of the skeleton path to turn the extruder off.
  - **Enabled By:** This setting is enabled when Skeleton Slow Down is enabled.

3.2.4 Tip Wipe

Tip wipe is a motion added to the end of a path, used to wipe off the tip or break the end of the bead away from the nozzle so that material doesn't stick up when executing a travel. The user defines a length for the tip wipe, a direction, and speed for the motion. For example: if the user enabled perimeter tip wipe, set a distance

of 2 inches, a direction of reverse, and a speed of 20in/s, then immediately after completing the perimeter path, the machine will reverse direction and retrace the last 2 inches of travel with the extruder off.

Tip wipes have four direction options: optimal, forward, reverse, and angled.

- Forward
  - Closed Loop - The forward wipe will continue forward retracing the beginning of the path.
  - Infill/Skin - For infill patterns that aren't closed loop, such as grid or lines, a forward wipe will wipe towards the nearest bounding contour (perimeter or inset) and attempt to follow that path. If inset and perimeter are disabled, the standard open loop wipe is used.
  - Open Loop - The forward wipe will continue along the same vector of travel that the path ended.
- Reverse - Reverse wipes will retrace the current path backwards by the given distance. This is the same for all path types.
- Angled - Angled tip wipes continue forward at a user defined angle. This is the same for all path types.
- Optimal - Optimal will select between forward and reverse based on the path type and nearby regions. For closed loop paths, optimal wipes will always be forward. Open loop paths will be reverse except with the exception of skin/infill where insets or perimeters are enabled.

The settings are broken out for each region (Perimeter, Inset, Skeleton, Skin, and Infill) allowing for a custom behavior within each.

- Enable Perimeter Tip Wipe
  - **Description:** Enable tip wipes for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
  - **Enables:** This setting enables Perimeter Wipe Direction, Perimeter Wipe Distance, and Perimeter Wipe Speed.
- Perimeter Wipe Direction
  - **Description:** Set the direction of tip wipes for perimeter paths.

  - **Options:**
    - Optimal
    - Forward
    - Reverse
    - Angled
  - **Enabled By:** This setting is enabled when Perimeter Tip Wipe is enabled.

- Perimeter Wipe Distance
  - **Description:** Set the length of the tip wipe for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Tip Wipe is enabled.

- Perimeter Wipe Speed
  - **Description:** Set the movement speed of the tip wipe for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Tip Wipe is enabled.

- Perimeter Wipe Angle
  - **Description:** Set the angle of the tip wipe for perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeter Tip Wipe is enabled and Perimeter Wipe Direction is set to Angled.

- Enable Inset Tip Wipe
  - **Description:** Enable tip wipes for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
  - **Enables:** This setting enables Inset Wipe Direction, Inset Wipe Distance, and Inset Wipe Speed.

- Inset Wipe Direction
  - **Description:** Set the direction of tip wipes for inset paths.
  - **Options:**
    - Optimal

    - Forward
    - Reverse
    - Angled
  - **Enabled By:** This setting is enabled when Inset Tip Wipe is enabled.

- Inset Wipe Distance
  - **Description:** Set the length of the tip wipe for inset paths.
  - **Enabled By:** This setting is enabled when Inset Tip Wipe is enabled.

- Inset Wipe Speed
  - **Description:** Set the movement speed of the tip wipe for inset paths.
  - **Enabled By:** This setting is enabled when Inset Tip Wipe is enabled.

- Inset Wipe Angle
  - **Description:** Set the angle of the tip wipe for inset paths.
  - **Enabled By:** This setting is enabled when Inset Tip Wipe is enabled and Inset Wipe Direction is set to Angled.

- Enable Skeleton Tip Wipe
  - **Description:** Enable tip wipes for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
  - **Enables:** This setting enables Skeleton Wipe Direction, Skeleton Wipe Distance, and Skeleton Wipe Speed.

- Skeleton Wipe Direction
  - **Description:** Set the direction of tip wipes for skeleton paths.
  - **Options:**
    - Optimal
    - Forward
    - Reverse
    - Angled
  - **Enabled By:** This setting is enabled when Skeleton Tip Wipe is enabled.

- Skeleton Wipe Distance
  - **Description:** Set the length of the tip wipe for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Tip Wipe is enabled.
- Skeleton Wipe Speed
  - **Description:** Set the movement speed of the tip wipe for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Tip Wipe is enabled.
- Skeleton Wipe Angle
  - **Description:** Set the angle of the tip wipe for skeleton paths.
  - **Enabled By:** This setting is enabled when Skeleton Tip Wipe is enabled. and Skeleton Wipe Direction is set to Angled.
- Enable Skin Tip Wipe
  - **Description:** Enable tip wipes for all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.
  - **Enables:** This setting enables Skin Wipe Direction, Skin Wipe Distance, and Skin Wipe Speed.
- Skin Wipe Direction
  - **Description:** Set the direction of tip wipes for skin paths.
  - **Options:**
    - Optimal
    - Forward
    - Reverse
    - Angled
  - **Enabled By:** This setting is enabled when Skin Tip Wipe is enabled.
- Skin Wipe Distance
  - **Description:** Set the length of the tip wipe for skin paths.
  - **Enabled By:** This setting is enabled when Skin Tip Wipe is enabled.
- Skin Wipe Speed

  - **Description:** Set the movement speed of the tip wipe for skin paths.
  - **Enabled By:** This setting is enabled when Skin Tip Wipe is enabled.

- Skin Wipe Angle
  - **Description:** Set the angle of the tip wipe for skin paths.
  - **Enabled By:** This setting is enabled when Skin Tip Wipe is enabled and Skin Wipe Direction is set to Angled.

- Enable Infill Tip Wipe
  - **Description:** Enable tip wipes for all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.
  - **Enables:** This setting enables Infill Wipe Direction, Infill Wipe Distance, and Infill Wipe Speed.

- Infill Wipe Direction
  - **Description:** Set the direction of tip wipes for infill paths.
  - **Options:**
    * Optimal
    * Forward
    * Reverse
    * Angled
  - **Enabled By:** This setting is enabled when Infill Tip Wipe is enabled.

- Infill Wipe Distance
  - **Description:** Set the length of the tip wipe for infill paths.
  - **Enabled By:** This setting is enabled when Infill Tip Wipe is enabled.

- Infill Wipe Speed
  - **Description:** Set the movement speed of the tip wipe for infill paths.
  - **Enabled By:** This setting is enabled when Infill Tip Wipe is enabled.

- Infill Wipe Angle
  - **Description:** Set the angle of the tip wipe for infill paths.

  - **Enabled By:** This setting is enabled when Infill Tip Wipe is enabled and Infill Wipe Direction is set to Angled.

- Laser Power Multiplier
  - **Description:** Sets a multiplier value applied to the laser power during tip wipe moves for the GKN syntax.
  - **Enabled By:** This setting is enabled for the GKN syntax.

- Wire Feed Multiplier
  - **Description:** Sets a multiplier value applied to the wire feed during tip wipe moves for the GKN syntax.
  - **Enabled By:** This setting is enabled for the GKN syntax.

3.2.5 Spiral Lift

Spiral lift is a motion at the end of the toolpath where the extruder moves in a circular motion while lifting up. This is beneficial for devices with a tamper because it allows the tamper to be moved across the surrounding area making sure the surface is flat. The default is to create this pathing via short G1 commands, but machines that can accept circular arc commands (G2 and G3) can simplify the motion. To use arc commands, Supports G2/G3 must be enabled in Printer Settings -> Machine Setup. It's important to note that spiral lifts often add a few seconds for every instance, which can really affect print time on longer builds.

- Spiral Perimeters
  - **Description:** Enables spiral lifts for the end of all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
  - **Enables:** This setting enables Spiral Lift Height, Number of Points, Radius, and Spiral Lift Speed.

- Spiral Insets
  - **Description:** Enables spiral lifts for the end of all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
  - **Enables:** This setting enables Spiral Lift Height, Number of Points, Radius, and Spiral Lift Speed.

- Spiral Skins
  - **Description:** Enables spiral lifts for the end of all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.
  - **Enables:** This setting enables Spiral Lift Height, Number of Points, Radius, and Spiral Lift Speed.
- Spiral Infill
  - **Description:** Enables spiral lifts for the end of all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.
  - **Enables:** This setting enables Spiral Lift Height, Number of Points, Radius, and Spiral Lift Speed.
- Spiral End of Layer
  - **Description:** Forces a spiral lift at end of the last path of the layer.
  - **Enabled By:** This setting is enabled when Infill is enabled.
  - **Enables:** This setting enables Spiral Lift Height, Number of Points, Radius, and Spiral Lift Speed.
- Spiral Lift Height
  - **Description:** Set the lift height during the spiral lift motion.
  - **Enabled By:** This setting is enabled when Spiral Perimeters OR Spiral Insets OR Spiral Skins or Spiral Infill is enabled.
- Number of Points
  - **Description:** For spiral lifts that don't use arc motions, this sets the number of segments generated to create the circular motion. A larger number of points means more individual segments and a longer duration to execute the path.
  - **Enabled By:** This setting is enabled when Spiral Perimeters OR Spiral Insets OR Spiral Skins or Spiral Infill is enabled.
- Spiral Lift Radius
  - **Description:** Set the radius for calculating the spiral lift motion. A larger radius means a larger area covered by the spiral lift.

  - **Enabled By:** This setting is enabled when Spiral Perimeters OR Spiral Insets OR Spiral Skins or Spiral Infill is enabled.

- Spiral Lift Speed
  - **Description:** Sets the movement speed during the spiral lift path.
  - **Enabled By:** This setting is enabled when Spiral Perimeters OR Spiral Insets OR Spiral Skins or Spiral Infill is enabled.

3.2.6 Purge

Purge is used to prime the extruder before starting the print. The Cincinnati BAAM systems have a defined M-Code to travel to a set location and purge the extruder. The following settings are used to modify the purge M-Code.

- Initial Purge Duration
  - **Description:** Sets the duration for running the extruder during the purge command.
  - **Enabled By:** This setting is enabled when the Syntax is Cincinnati and Material Load is enabled.
- Initial Purge Screw RPM
  - **Description:** Sets the extruder speed (RPM) for the purge command.
  - **Enabled By:** This setting is enabled when the Syntax is Cincinnati and Material Load is enabled.
- Initial Purge Tip Wipe Delay
  - **Description:** Sets the delay after purging, before moving the extruder across the wiper blade.
  - **Enabled By:** This setting is enabled when the Syntax is Cincinnati and Material Load is enabled.

3.2.7 Extruder

- Initial Extruder Speed
  - **Description:** Sets the initial extruder speed, used when turning the extruder on before a path.

  - **Enabled By:** This setting is enabled when the Machine Type is Pellet OR Concrete OR Thermoset.

- Perimeter Extruder On Delay
  - **Description:** Sets the delay duration during which the initial extruder speed is used, while motion is paused, at the start of all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled AND the Machine Type is Pellet OR Concrete OR Thermoset.

- Inset Extruder On Delay
  - **Description:** Sets the delay duration during which the initial extruder speed is used, while motion is paused, at the start of all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled AND the Machine Type is Pellet OR Concrete OR Thermoset.

- Skin Extruder On Delay
  - **Description:** Sets the delay duration during which the initial extruder speed is used, while motion is paused, at the start of all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled AND the Machine Type is Pellet OR Concrete OR Thermoset.

- Infill Extruder On Delay
  - **Description:** Sets the delay duration during which the initial extruder speed is used, while motion is paused, at the start of all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled AND the Machine Type is Pellet OR Concrete OR Thermoset.

- Extruder Off Delay
  - **Description:** Sets the delay duration during which the extruder is kept on, while motion is paused, at the end of the path.
  - **Enabled By:** This setting is enabled when the Machine Type is Pellet OR Concrete OR Thermoset.

- Servo Extruder to Travel Speed

- **Description:** Issues M10 and M11 codes to enable and disable (respectively) the extruder servoing mode on BAAM systems.
- **Enabled By:** This setting is enabled when the Syntax is Cincinnati.

3.2.8 Filament

For filament based systems, an extrusion amount, E###, is added to every printing segment. This extrusion amount (EA) is calculated using the variables filament diameter (FD), segment length (SL), bead width (BW), and layer height (LH). The following formula, where the volume of the bead is set equal to the volume of the input filament, is used to calculate the extrusion amount:

$$SL \times BW \times LH = \pi \times (\frac{FD}{2})^2 \times EA$$

This extrusion amount (EA) is then multiplied by the extrusion multiplier (default value of 1) to calculate the E value output in the G-Code.

- Filament Diameter
  - **Description:** Sets the filament diameter used for calculating the extrusion amount.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament.
- Perimeter Filament Extrusion Multiplier
  - **Description:** Sets the extrusion multiplier used for modifying the extrusion amount during all perimeter paths.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament AND Perimeters are enabled.
- Inset Filament Extrusion Multiplier
  - **Description:** Sets the extrusion multiplier used for modifying the extrusion amount during all inset paths.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament AND Insets are enabled.
- Skin Filament Extrusion Multiplier

  - **Description:** Sets the extrusion multiplier used for modifying the extrusion amount during all skin paths.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament AND Skins are enabled.

- Infill Filament Extrusion Multiplier
  - **Description:** Sets the extrusion multiplier used for modifying the extrusion amount during all infill paths.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament AND Infill is enabled.

3.2.9 Retraction

- Enable Retraction
  - **Description:** Retraction is used to pull back on the filament at the end of an extrusion path to reduce oozing and stringing.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament.
  - **Enables:** This setting enables Retraction Length, Retraction Speed, Retract on Layer Change, Wipe While Retracting, Enable Extra Extrusion on Restart, Prime Length, and Prime Speed.

- Minimum Travel for Retraction
  - **Description:** Travel moves longer than this distance will generate a retraction.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.

- Retraction Length
  - **Description:** Sets the length of filament to be retracted each time. This amount is re-added at the start of the next path to prime the extruder.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.

- Retraction Speed
  - **Description:** Sets the speed of the filament retraction.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.

- Retract on Layer Change
  - **Description:** If enabled, a retraction is forced at every layer change.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.
- Prime Speed
  - **Description:** Sets the speed for the filament priming.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.
- Additional Prime Length
  - **Description:** Sets the length of extra filament to be extruded at the start of each printing path after retracting.
  - **Enabled By:** This setting is enabled when Retraction is Enabled.

3.2.10 Temperatures

- Bed Temperature
  - **Description:** Sets the temperature of the bed to be issued in the header of the G-Code file.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament or the Syntax is Marlin Pellet.
- Extruder 0 Temperature
  - **Description:** Sets the temperature of Extruder 0 to be issued in the header of the G-Code file.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament or the Syntax is Marlin Pellet.
- Extruder 1 Temperature
  - **Description:** Sets the temperature of Extruder 1 to be issued in the header of the G-Code file.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament or the Syntax is Marlin Pellet.
- Extruder 0 Standby Temperature

  - **Description:** Sets the standby temperature of Extruder 0 to be issued in the header of the G-Code file.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament or the Syntax is Marlin Pellet.

- Extruder 1 Standby Temperature
  - **Description:** Sets the standby temperature of Extruder 1 to be issued in the header of the G-Code file.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament or the Syntax is Marlin Pellet.

3.2.11 Cooling

- Fan Control
  - **Description:** Enables dynamic fan speed control.
  - **Enabled By:** This setting is enabled when the Machine Type is Filament.
  - **Enables:** This setting enables Min Fan Speed and Max Fan Speed.
- Min Fan Speed
  - **Description:** Sets the minimum bound of fan speed used for dynamic fan control.
  - **Enabled By:** This setting is enabled when Fan Control is enabled.
- Max Fan Speed
  - **Description:** Sets the maximum bound of fan speed used for dynamic fan control.
  - **Enabled By:** This setting is enabled when Fan Control is enabled.
- Force Minimum Layer Time
  - **Description:** If enabled, settings are available to force a minimum layer time.
  - **Enables:** This setting enables Minimum Layer Time Method and Minimum Layer Time.
- Minimum Layer Time Method

  - **Description:** Sets the method of enforcing a minimum layer time for every layer. For both methods, if the default layer time is greater than the minimum layer time, no action is taken.
  - **Options:**
    * Add Dwell Time - this option adds a pause at the end of every layer to ensure the total layer time meets or exceeds the minimum layer time
    * Modify Feedrate - this option adjusts the feedrate value (F###) for each path segment within the layer to ensure the total layer time meets or exceeds the minimum layer time
  - **Enabled By:** This setting is enabled when Force Minimum Layer Time is enabled.

- Minimum Layer Time
  - **Description:** Sets the minimum layer time to be enforced on every layer.
  - **Enabled By:** This setting is enabled when Force Minimum Layer Time is enabled.

- Pre-Pause G-Code
  - **Description:** Custom G-Code to be executed before the pause command when Minimum Layer Time Method is Add Dwell Time.
  - **Enabled By:** This setting is enabled when Force Minimum Layer Time is enabled AND Minimum Layer Time Method is set to Add Dwell Time.

- Post-Pause G-Code
  - **Description:** Custom G-Code to be executed after the pause command when Minimum Layer Time Method is Add Dwell Time.
  - **Enabled By:** This setting is enabled when Force Minimum Layer Time is enabled AND Minimum Layer Time Method is set to Add Dwell Time.

3.2.12 Platform Adhesion

- Add Raft

  - **Description:** A raft is a solid layer, or layers, of infill printed at the start of the build. The part, or parts, to be printed is printed after the raft completes. The raft is most often used to compensate for a build surface that isn't perfectly flat. Raft uses the default speed and extruder speed.

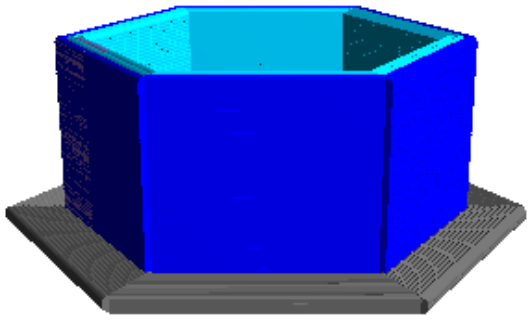


Figure 1: Raft toolpaths, shown in gray, beneath a hexagon.

  - **Enables:** This setting enables Raft XY Offset, Raft Layers, and Raft Bead Width.

- Raft XY Offset
  - **Description:** The XY offset distance is how far the raft extends outside the bounding box of the part.
  - **Enabled By:** This setting is enabled when Add Raft is enabled.
- Raft Layer Count
  - **Description:** The number of raft layers tall to print. The part sits atop all the raft layers.
  - **Enabled By:** This setting is enabled when Add Raft is enabled.
- Raft Bead Width
  - **Description:** The bead width used for raft toolpaths. This value is used as the line spacing because the paths are printed like solid infill.
  - **Enabled By:** This setting is enabled when Add Raft is enabled.
- Add Brim

  - **Description:** A brim is a solid area surrounding the part on the first layer, or layers, that helps adhere the part to the build surface by increasing the surface area. This is especially helpful for parts prone to warping and curling. The brim is created by offsetting the layer boundary of the object outwards to create closed loop toolpath. Brim uses the default speed and extruder speed.

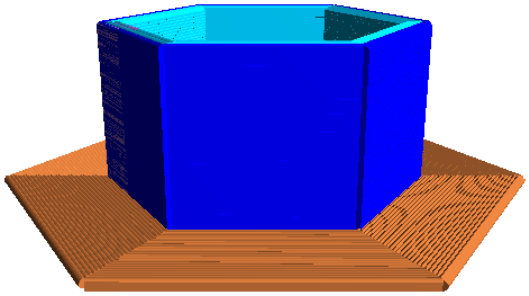


Figure 2: Brim toolpaths, shown in orange, surrounding the base a hexagon.

  - **Enables:** This setting enables Brim Width, Brim Layer Count, and Brim Bead Width.

- Brim Width
  - **Description:** The width of the brim, or how far from the part to start the brim.
  - **Enabled By:** This setting is enabled when Add Brim is enabled.
- Brim Layer Count
  - **Description:** The number of layers to print the brim, starting from layer 1.
  - **Enabled By:** This setting is enabled when Add Brim is enabled.
- Brim Bead Width
  - **Description:** The bead width used for calculating brim toolpaths.
  - **Enabled By:** This setting is enabled when Add Brim is enabled.
- Add Skirt

  - **Description:** Skirt is used to ensure the extruder is properly extruding and also check the build height. Skirt is a closed loop path, or paths, offset from the bounding box of a part at the beginning of the print. The paths do not touch the object and are meant to be purely sacrificial. Skirt uses the default speed and extruder speed values.

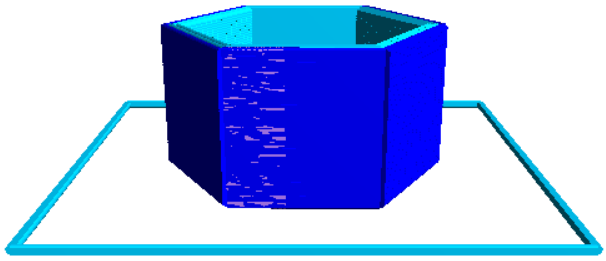


Figure 3: Skirt toolpaths, shown in light blue, offset from the bounding box of a hexagon.

  - **Enables:** This setting enables Skirt Loops, Skirt Distance from Object, Skirt Layer Count, Skirt Minimum Length, and Skirt Bead Width.

- Skirt Loops
  - **Description:** The number of closed loop paths in the skirt layer.
  - **Enabled By:** This setting is enabled when Add Skirt is enabled.
- Skirt Distance from Object
  - **Description:** The distance from the object to create the skirt toolpaths.
  - **Enabled By:** This setting is enabled when Add Skirt is enabled.
- Skirt Layer Count
  - **Description:** The number of layers to print the skirt.
  - **Enabled By:** This setting is enabled when Add Skirt is enabled.
- Skirt Minimum Length
  - **Description:** The minimum print distance for the skirt to ensure the extruder is properly primed. If the minimum distance is not achieved, additional skirt loops are added.

  - **Enabled By:** This setting is enabled when Add Skirt is enabled.

- Skirt Bead Width
  - **Description:** The bead width used for creating the skirt toolpaths.
  - **Enabled By:** This setting is enabled when Add Skirt is enabled.

3.2.13 Multi-Material

- Enable Multi-Material
  - **Description:** Multi-Material is used to assign a specific material to each region. This is currently used for BAAM systems that have material switching capabilities. A material number is assigned to each region and a transition distance, or distances, are assigned to signify how much printing distance is needed to swap materials. The Slicer will automatically calculate where to make the transitions based on these distance inputs and insert the appropriate M-Code to transition materials.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Perimeter is enabled.
  - **Enables:** This setting enables Perimeter Material Number, Inset Material Number, Skin Material Number, Infill Material Number, and Enable Multiple Nozzles with Multi-Material.

- Perimeter Material Number
  - **Description:** Sets the material number for perimeter paths.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Perimeters are enabled.

- Inset Material Number
  - **Description:** Sets the material number for inset paths.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Insets are enabled.

- Skin Material Number
  - **Description:** Sets the material number for skin paths.

  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Skins are enabled.

- Infill Material Number
  - **Description:** Sets the material number for infill paths.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Infill is enabled.

- Support Material Number
  - **Description:** Sets the material number for support paths.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled AND Support is enabled.

- Transition Distance
  - **Description:** Sets the distance used for transitioning between materials. The transition M-Code is inserted during printing at this distance before the next material is needed.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled.

- Enable Second Transition Distance
  - **Description:** Allows for a second transition to be used when transitioning from material 2 to 1.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled.
  - **Enables:** This setting enables Second Transition Distance.

- Second Transition Distance
  - **Description:** Sets the distance used for transitioning between materials 2 and 1. The transition M-Code is inserted during printing at this distance before the next instance of material 1 is needed.
  - **Enabled By:** This setting is enabled by Enable Second Transition Distance.

- Use M222 Code
  - **Description:** Uses the M222 code to swap materials rather than the default M237.
  - **Enabled By:** This setting is enabled when Multi-Material is enabled.

3.3 Profile Settings

3.3.1 Layer

- Layer Height
  - **Description:** Thickness of each layer. This value is used to create the scale seen in the Part View when a building mesh is selected.
- Nozzle Diameter
  - **Description:** Diameter of the main extruder nozzle. This value is currently used only for writing the header.
  - **Enabled By:** This setting is enabled when the Machine Type is Pellet OR Concrete OR Thermoset.
- Default Bead Width
  - **Description:** Default bead width for extrusion paths that don't have a defined width. Also used for the visualization.
- Default Print Speed
  - **Description:** Default printing speed for extrusion paths that don't have a defined speed.
- Default Extruder Speed
  - **Description:** Default extruder speed for extrusion paths that don't have a defined extruder speed.
  - **Enabled By:** This setting is enabled when the Machine Type is Pellet OR Concrete OR Thermoset.
- Minimum Extrude Length
  - **Description:** Defaulted minimum extrude length for paths that don't have a defined length. Currently used for brim, raft, and support paths.

3.3.2 Perimeter

Perimeters are closed loop paths (they start and stop at the same point) printed around the boundary of the object. The pathing is found by offsetting the bounding polygon inwards by a bead width, then offsetting outwards by a half bead width to find the center point of the path. The offset inwards is done by a full bead width, rather than a half bead width thus skipping the step offsetting back outwards by a half width, to ensure the resultant path has sufficient space for a bead. Any area not large enough for a full bead width is eliminated during this offsetting process. Perimeters are typically the first region printed on a layer.

- Enable Perimeter
  - **Description:** Perimeters are the outermost contours of a print. These are closed loop paths. Sometimes they are referred to as horizontal shells.
  - **Enables:** This setting enables the following settings: Number of Perimeters, Perimeter Bead Width, Perimeter Speed, Perimeter Extruder Speed, Minimum Perimeter Path Length, Perimeter Power, Perimeter Focus, Perimeter Spot Size, Perimeter Start G-Code, Perimeter End G-Code, Reverse Perimeter Direction, Reverse Perimeter Order, Perimeter Material Number, Perimeter Extruder On Delay, Spiral Perimeters, Enable Perimeter Tip Wipe, Enable Perimeter Start-up, and Enable Perimeter Slow Down.
- Number of Perimeters
  - **Description:** Sets the maximum number of closed loop perimeter paths to be printed for each island on each layer.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
- Perimeter Bead Width
  - **Description:** Sets the bead width used for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
- Perimeter Speed
  - **Description:** Sets the print speed used for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.
- Perimeter Extruder Speed

  - **Description:** Sets the extruder speed for all perimeter paths.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.

- Minimum Perimeter Path Length
  - **Description:** Sets the minimum path length for all perimeter paths. Paths less than this value are eliminated.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.

- Perimeter Power
  - **Description:** Sets the laser power for perimeter paths with the GKN syntax.
  - **Enabled By:** This setting is enabled when Perimeters are enabled AND the syntax is GKN.

- Perimeter Focus
  - **Description:** Sets the focus size for perimeter paths with the GKN syntax.
  - **Enabled By:** This setting is enabled when Perimeters are enabled AND the syntax is GKN.

- Perimeter Spot Size
  - **Description:** Sets the spot size for perimeter paths with the GKN syntax.
  - **Enabled By:** This setting is enabled when Perimeters are enabled AND the syntax is GKN.

3.3.3 Inset

Insets are very similar to perimeters. They are closed loop paths found by offsetting the boundary geometry inwards. Insets are printed inside of perimeters and allow for additional control of speed and width such that insets, which aren't typically exposed on the outside of a part, can be printed faster and thicker to save print time.

- Enable Inset
  - **Description:** Insets are closed loop contours printed inside of the perimeters. If perimeters are disabled, insets are the outermost contours of a print.

  - **Enables:** This setting enables the following settings: Number of Insets, Inset Bead Width, Inset Speed, Inset Extruder Speed, Minimum Inset Path Length, Inset Start G-Code, Inset End G-Code, Reverse Inset Direction, Reverse Inset Order, Inset Material Number, Inset Extruder On Delay, Spiral Insets, Enable Inset Tip Wipe, Enable Inset Start-up, and Enable Inset Slow Down.

- Number of Insets
  - **Description:** Sets the maximum number of closed loop insets paths to be printed for each island on each layer.
  - **Enabled By:** This setting is enabled when Insets are enabled.
- Inset Bead Width
  - **Description:** Sets the bead width used for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
- Inset Speed
  - **Description:** Sets the print speed used for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
- Inset Extruder Speed
  - **Description:** Sets the extruder speed for all inset paths.
  - **Enabled By:** This setting is enabled when Insets are enabled.
- Minimum Inset Path Length
  - **Description:** Sets the minimum path length for all inset paths. Paths less than this value are eliminated.
  - **Enabled By:** This setting is enabled when Insets are enabled.

3.3.4 Skeleton

Skeletons are open-loop paths meant to fill voids and compensate for parts not designed for the proper bead width. Perimeters and Insets are strictly closed-loop paths, that is to say they start and stop at the same point. Skeletons are open-loop meaning they start and stop at different points. The math for finding skeletons is

very complex, and at this point, not incredibly robust. Skeletons are still under development.

- Enable Skeletons
  - **Description:** Enables the generation of open-loop skeleton paths.
- Skeleton Input
  - **Description:** Select between sending points or segments of border geometry to the Voronoi Diagram generator.
  - **Options:**
    - Segments
    - Points
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
- Skeleton Input Cleaning Distance
  - **Description:** Removes vertices: that join co-linear edges, or join edges that are almost co-linear (such that if the vertex was moved no more than the specified distance the edges would be co-linear), that are within the specified distance of an adjacent vertex, that are within the specified distance of an adjacent vertex, that are within the specified distance of a semi-adjacent vertex together with their out-lying vertices.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
- Skeleton Output Cleaning Distance
  - **Description:** Removes vertices: that join co-linear edges, or join edges that are almost co-linear (such that if the vertex was moved no more than the specified distance the edges would be co-linear), that are within the specified distance of an adjacent vertex, that are within the specified distance of an adjacent vertex, that are within the specified distance of a semi-adjacent vertex together with their out-lying vertices.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
- Skeleton Bead Width
  - **Description:** Sets the bead width used for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.

- Skeleton Speed
  - **Description:** Sets the speed used for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
- Skeleton Extruder Speed
  - **Description:** Sets the extruder speed used for all skeleton paths.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.
- Minimum Skeleton Path Length
  - **Description:** Sets the minimum path length used for all skeleton paths. Paths less than this value are eliminated.
  - **Enabled By:** This setting is enabled when Skeletons are enabled.

3.3.5 Skin

The skin region is used to create solid layers on vertical surfaces. Skins are printed using space filling approaches, similar to those in infill, but with 100% density (pattern images can be seen in the infill section, 3.3.6). The user sets the number of top and bottom skins, which correlates to the number of solid layers at each top and bottom surface. A top skin count of three would mean three solid layers of fill leading up to a top surface. This top surface doesn't have to be the final layer of the part. Top surfaces can occur at any layer of the part and are found by checking the polygon area to see what polygons appear or disappear layer to layer during slicing. Any polygon that disappears, is known to be a top of the last layer that it appears. Likewise, the same approach is used for finding bottom skins/surfaces that can appear on any layer.

- Enable Skin
  - **Description:** Skins are solid layers of infill on top and bottom surfaces. Sometimes referred to as vertical shells.
  - **Enables:** This setting enables the following settings: Top Skin Count, Bottom Skin Count, Skin Pattern, Skin Fill Angle, Skin Fill Angle Rotation, Skin Bead Width, Skin Speed, Skin Extruder Speed, Minimum Skin Path Length, Skin Exterior Overlap, Enable Skin Prestart, Enable Gradual Infill Steps, Skin Start G-Code, Skin End G-Code, Skin Material Number, Skin Extruder On Delay, Spiral Skins, Enable Skin Tip Wipe, Enable Skin Start-up, and Enable Skin Slow Down.

- Top Skin Count
  - **Description:** The number of solid top skin layers generated for each top surface.
  - **Enabled By:** This setting is enabled when Skins are enabled.
- Bottom Skin Count
  - **Description:** The number of solid bottom skin layers generated for each bottom surface.
  - **Enabled By:** This setting is enabled when Skins are enabled.
- Skin Pattern
  - **Description:** Sets the pattern for the skin fill on both top and bottom skins.
  - **Options:**
    - Lines
    - Concentric
    - Inside Out Concentric
  - **Enabled By:** This setting is enabled when Skins are enabled.
- Skin Fill Angle
  - **Description:** The angle of the lines for the line skin pattern. 0 degrees is in line with the Y axis.
  - **Enabled By:** This setting is enabled when Skins are enabled and the pattern is set to Lines.
- Skin Fill Angle Rotation
  - **Description:** The angle rotation for the line skin fill pattern. The pattern is rotated by this amount each layer with respect to the previous layer.
  - **Enabled By:** This setting is enabled when Skins are enabled and the pattern is set to Lines.
- Skin Exterior Overlap

  - **Description:** Sets the amount of overlap for the skin region with the bounding contour. To do this, the polygon for the skin is offset by this value.
  - **Enabled By:** This setting is enabled when Skins are enabled.

- Skin Bead Width
  - **Description:** Sets the bead width used for all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.

- Skin Speed
  - **Description:** Sets the print speed used for all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.

- Skin Extruder Speed
  - **Description:** Sets the extruder speed for all skin paths.
  - **Enabled By:** This setting is enabled when Skins are enabled.

- Minimum Skin Path Length
  - **Description:** Sets the minimum path length for all skin paths. Paths less than this value are eliminated.
  - **Enabled By:** This setting is enabled when Skins are enabled.

- Enable Skin Prestart
  - **Description:** Prestart adds extra segments to the start of the skin path. These extra segments are printed on top of the bounding contour. The idea is to help anchor the fill and the contour.
  - **Enabled By:** This setting is enabled when Skins are enabled AND Perimeters OR Insets are enabled.

- Skin Prestart Distance
  - **Description:** Sets the total length of the skin prestart path. This distance is back tracked away from the start point of the skin path.
  - **Enabled By:** This setting is enabled when Skin Prestart is enabled.

- Skin Prestart Speed
  - **Description:** Sets the speed for the skin prestart path.
  - **Enabled By:** This setting is enabled when Skin Prestart is enabled.
- Skin Prestart Extruder Speed
  - **Description:** Sets the extruder speed for the skin prestart path.
  - **Enabled By:** This setting is enabled when Skin Prestart is enabled.
- Enable Gradual Infill Steps
  - **Description:** Gradual infill steps are used to help print solid skins over sparse infill. Solid infill tends to droop down in the gaps of sparse infill and can take many layers to print successfully. Gradual infill steps are used to densify the infill pattern as the infill approaches a top surface. The area for apply gradual infill is found using the same algorithm as skins.
  - **Enabled By:** This setting is enabled when Skins are enabled.
- Number of Gradual Infill Steps
  - **Description:** The number of steps is the number of layers of gradual infill. With each step the infill gets more dense. The density is calculated based on the infill density making steps towards skin density (which is 100%). For example: if infill density is 50% and gradual infill is step to 1 step, the 1 layer of gradual infill will be 75% dense. If there are two steps, the first layer will be 66.66% dense and the second layer will be 83.33% dense.
  - **Enabled By:** This setting is enabled when Gradual Infill is enabled.
- Gradual Infill Pattern
  - **Description:** Sets the pattern for the gradual infill steps.
  - **Options:**
    - Lines
    - Grid
    - Honeycomb
    - Hexagons and Triangles
  - **Enabled By:** This setting is enabled when Gradual Infill is enabled.

- Gradual Infill Angle
  - **Description:** The angle of the lines for the gradual infill pattern. 0 degrees is in line with the Y axis.
  - **Enabled By:** This setting is enabled when Gradual Infill is enabled and the pattern is set to Lines or Grid.
- Gradual Infill Angle Rotation
  - **Description:** The angle rotation for the gradual infill pattern. The pattern is rotated by this amount each layer with respect to the previous layer.
  - **Enabled By:** This setting is enabled when Gradual Infill is enabled and the pattern is set to Lines or Grid.

3.3.6 Infill

Infill is the region used for filling the center of a part. The infill is often fully enclosed, and thus not seen after printing is completed. Infill comes in a variety of patterns, shown in the following image. Unlike skins, infill density can be varied based on the needs of the object being printed.

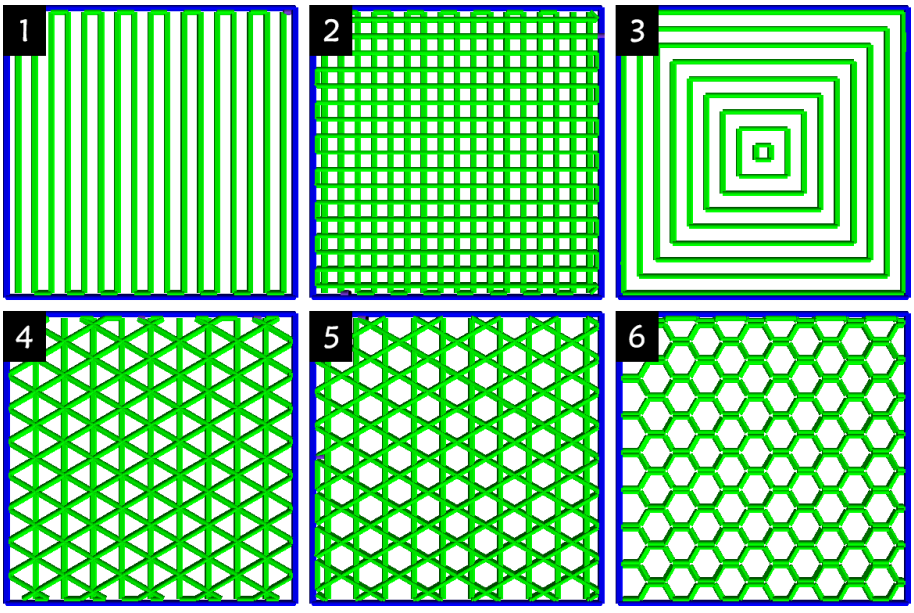


Figure 4: Various infill patterns

1. Lines - a fill pattern characterized by parallel lines. This pattern can be printed at a defined angle and can be set to rotate by a set amount layer to layer. Line spacing sets the distance between parallel lines of infill. Minimum Travel Length (Profile Settings -> Travel -> Minimum Travel Length) determines if the lines are connected at the end, or if a travel move is inserted to connect the lines.

2. Grid - a fill pattern using the same approach as the lines pattern, but creating two iterations of the lines within each layer, rotated by 90 degrees, so that a grid is formed on every layer. Half of the line spacing is used as the distance between parallel lines.

3. Concentric - creates closed loop paths that are inwards offsets of the layer boundary, like with perimeters and insets. Line spacing determines how far to offset.

4. Triangles - a fill pattern using the same approach as the lines pattern, but creating three iterations of the lines within each layer, rotated by 60 degrees and shifted, so that a grid of triangles is formed on every layer. One third of the line spacing is used as the distance between parallel lines.

5. Hexagons and Triangles - a fill pattern using the same approach as the lines pattern, but creating three iterations of the lines within each layer, rotated by 60 degrees, so that a grid of hexagons and triangles is formed on every layer. One third of the line spacing is used as the distance between parallel lines.

6. Honeycomb - a fill pattern that creates hexagons in a honeycomb shape.

- Enable Infill
  - **Description:** Infill is used to fill the center, or void, of the object that remains after printing all perimeter, inset, and skin paths. It comes in a variety of patterns, representing space filling approaches, and can have a specific density to make the infill sparse or solid as needed by the object.
  - **Enables:** This setting enables the following settings: Infill Line Spacing, Infill Density, Infill Pattern, Infill Based on Printer Area, Infill Angle, Infill Angle Rotation, Infill Overlap Distance, Infill Bead Width, Infill Speed, Infill Extruder Speed, Minimum Skin Path Length, Skin Exterior Overlap, Enable Skin Prestart, Enable Gradual Infill Steps, Skin Start G-Code, Skin End G-Code, Skin Material Number, Skin Extruder On Delay,

Spiral Skins, Enable Skin Tip Wipe, Enable Skin Start-up, and Enable Skin Slow Down.

- Infill Line Spacing
  - **Description:** The distance between centers of parallel infill beads. Infill spacing equal to double the bead width would result in a 50% dense infill whereas spacing equal to the bead width would be 100% dense.
  - **Enables:** This setting is enabled when Infill is enabled.
- Infill Density
  - **Description:** THIS SETTING IS CURRENTLY NOT IMPLEMENTED AND HAS NO EFFECT.
  - **Enables:** This setting is enabled when Infill is enabled.
- Infill Pattern
  - **Description:** Sets the pattern for the infill.
  - **Options:**
    - Lines
    - Grid
    - Concentric
    - Inside out Concentric
    - Triangles
    - Hexagons and Triangles
    - Honeycomb
    - Radial Hatch - only available for the Saturn Slicer
  - **Enabled By:** This setting is enabled when Gradual Infill is enabled.
- Infill Based on Printer Area
  - **Description:** By default, infill is generated for the bounding box of the part and clipped to the region that needs infill. This means that multiple parts can be printed with the exact same infill path design. This setting allows infill to be generated across the bounding box of the printer and then clipped to the infill region. This means that the infill pathing for a part is specific to the location of the part within the printer volume. It

also means that multiple parts printed at the same time will have slightly different infill path design. The benefit is that objects can be moved within the workspace to generate slightly different fill, which is helpful for thin sections.

  - **Enabled By:** This setting is enabled when Infill is enabled.

- Infill Angle
  - **Description:** The angle of the fill for all infill patterns, except Concentric and Inside Out Concentric.
  - **Enabled By:** This setting is enabled when Infill is enabled and the pattern is not Concentric or Inside Out Concentric.

- Infill Angle Rotation
  - **Description:** The angle rotation for the infill pattern. The pattern is rotated by this amount each layer with respect to the previous layer.
  - **Enabled By:** This setting is enabled when Infill is enabled and the pattern is not Concentric or Inside Out Concentric.

- Infill Overlap Distance
  - **Description:** Sets the amount of overlap for the infill region with the bounding contour. To do this, the polygon for the infill is offset by this value.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Infill Bead Width
  - **Description:** Sets the bead width used for all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Infill Speed
  - **Description:** Sets the print speed used for all infill paths.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Infill Extruder Speed
  - **Description:** Sets the extruder speed for all infill paths.

  - **Enabled By:** This setting is enabled when Infill is enabled.

- Minimum Infill Path Length
  - **Description:** Sets the minimum path length for all infill paths. Paths less than this value are eliminated.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Combine Infill Every X Layers
  - **Description:** Used to speed up printing by combining of layers together to print at once. For example, combine infill every 2 layers would print infill every other layer. The idea is that the infill is extruded at a higher rate and/or printed at a slower speed so that more material fills the multi-layer void.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Enable Infill Prestart
  - **Description:** Prestart adds extra segments to the start of the infill path. These extra segments are printed on top of the bounding contour. The idea is to help anchor the fill and the contour.
  - **Enabled By:** This setting is enabled when Infill is enabled AND Perimeters OR Insets are enabled.

- Infill Prestart Distance
  - **Description:** Sets the total length of the infill prestart path. This distance is back tracked away from the start point of the infill path.
  - **Enabled By:** This setting is enabled when Infill Prestart is enabled.

- Infill Prestart Speed
  - **Description:** Sets the speed for the infill prestart path.
  - **Enabled By:** This setting is enabled when Infill Prestart is enabled.

- Infill Prestart Extruder Speed
  - **Description:** Sets the extruder speed for the infill prestart path.
  - **Enabled By:** This setting is enabled when Infill Prestart is enabled.

- Infill Sector Count
  - **Description:** The number of sectors used to generate the radial hatch fill pattern.
  - **Enabled By:** This setting is enabled when Infill is enabled and the pattern is set to Radial Hatch.
- Infill Power
  - **Description:** Sets the laser power for infill paths with the GKN Syntax.
  - **Enabled By:** This setting is enabled when Infill is enabled and the syntax is GKN.
- Infill Focus
  - **Description:** Sets the focus size for infill paths with the GKN Syntax.
  - **Enabled By:** This setting is enabled when Infill is enabled and the syntax is GKN.
- Infill Spot Size
  - **Description:** Sets the spot size for infill paths with the GKN Syntax.
  - **Enabled By:** This setting is enabled when Infill is enabled and the syntax is GKN.

3.3.7 Support

- Enable Support
  - **Description:** Support structures are used to support bridges and overhangs that exceed the bounds of what is printable.
- Print Support First
  - **Description:** Print support structures before printing the other regions on each layer.
  - **Enabled By:** This setting is enabled when Support is enabled.
- Taper Support

  - **Description:** Taper supports inward to save material by making the supports less dense.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Support Threshold Angle
  - **Description:** Minimum angle at which supports are generated. Measured from the vertical axis.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Support XY Distance
  - **Description:** XY Offset from object to generate support.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Support Layer Offset
  - **Description:** Z offset, measured in integer layer numbers, to offset support from object.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Minimum Support Infill Area
  - **Description:** Minimum area of a support structure. Supports smaller than this are often unstable and not likely to print successfully.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Minimum Support Area
  - **Description:** Minimum overhanging area on object to generate support. Areas smaller than this aren't worth generating supports for.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Support Pattern
  - **Description:** Sparse infill pattern used for support structures
  - **Options:**
    * Lines
    * Grid

  - **Enabled By:** This setting is enabled when Support is enabled.

- Support Line Spacing
  - **Description:** Distance between centers of parallel support infill beads.
  - **Enabled By:** This setting is enabled when Support is enabled.

3.3.8 Travel

- Travel Speed
  - **Description:** Sets the movement speed used for all travel segments. This speed is used to calculate time estimates for syntaxes that travel with G0.

- Minimum Travel Length
  - **Description:** The minimum distance for which a travel segment is generated rather than an infill or skin linking print segment.
  - **Enabled By:** This setting is enabled when Infill OR Skin is enabled.

- Minimum Travel Length for Lift
  - **Description:** Minimum travel segment length required to generate a lift. Travel segments less than this length are executed without lifting.

- Travel Lift Height
  - **Description:** Height to lift extruder at the start of travel moves.

3.3.9 G-Code

- Perimeter Start G-Code
  - **Description:** User input G-Code that is executed at the start of every perimeter path.
  - **Enabled By:** This setting is enabled when Perimeter is enabled.

- Perimeter End G-Code
  - **Description:** User input G-Code that is executed at the end of every perimeter path.

  - **Enabled By:** This setting is enabled when Perimeter is enabled.

- Inset Start G-Code
  - **Description:** User input G-Code that is executed at the start of every inset path.
  - **Enabled By:** This setting is enabled when Inset is enabled.
- Inset End G-Code
  - **Description:** User input G-Code that is executed at the end of every inset path.
  - **Enabled By:** This setting is enabled when Inset is enabled.
- Skeleton Start G-Code
  - **Description:** User input G-Code that is executed at the start of every skeleton path.
  - **Enabled By:** This setting is enabled when Skeleton is enabled.
- Skeleton End G-Code
  - **Description:** User input G-Code that is executed at the end of every skeleton path.
  - **Enabled By:** This setting is enabled when Skeleton is enabled.
- Skin Start G-Code
  - **Description:** User input G-Code that is executed at the start of every skin path.
  - **Enabled By:** This setting is enabled when Skin is enabled.
- Skin End G-Code
  - **Description:** User input G-Code that is executed at the end of every skin path.
  - **Enabled By:** This setting is enabled when Skin is enabled.
- Infill Start G-Code

  - **Description:** User input G-Code that is executed at the start of every infill path.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Infill End G-Code
  - **Description:** User input G-Code that is executed at the end of every infill path.
  - **Enabled By:** This setting is enabled when Infill is enabled.

- Support Start G-Code
  - **Description:** User input G-Code that is executed at the start of every support path.
  - **Enabled By:** This setting is enabled when Support is enabled.

- Support End G-Code
  - **Description:** User input G-Code that is executed at the end of every support path.
  - **Enabled By:** This setting is enabled when Support is enabled.

3.3.10 Special Modes

- Enable Smoothing
  - **Description:** Smoothing is used to simplify the polygons used for slicing. This works by eliminating points that are too close together and therefore redundant. Having extra points makes for larger G-Code files and more print segments that the printer must interpret. A smoothed file is faster and smaller, at the cost of being slightly less accurate. Loss of accuracy is dependent upon the tolerance.

- Smoothing Tolerance
  - **Description:** Smoothing tolerance determines how many points are removed during the smoothing process. A large tolerance will create more smoothing and remove a larger quantity of points in the polygon. A suggested starting point for tolerance is 1/10th the bead width.
  - **Enabled By:** This setting is enabled when Smoothing is enabled.

- Enable Spiralize Mode
  - **Description:** Spiralize mode prints without starts and stops by eliminating all insets, skins, and infill. It prints one single perimeter and slowly increments the Z-axis on each move so that there is not a defined lift at the layer change. The very first segment will be printed with the extruder nearly touching the build surface. Perimeter settings, such as extruder speed and gantry speed, are applied to spiralize toolpaths.
- Enable Fix Model
  - **Description:** Fix model attempts to repair STL issues such as inverted normals and lack of water tightness. This feature is still under development.
- Oversize Part
  - **Description:** Oversizing is used to grow a part in the XY axis. It is primarily used to add a machining offset. Oversizing is done at the time of slicing a layer into polygons. It only applies to the XY directions, not Z.
- Oversize Distance
  - **Description:** Distance to oversize the part by. A positive value offsets the part outward, grows the part. A negative value offsets the part inward, shrinks the part.
  - **Enabled By:** This setting is enabled when Support is enabled.
- Use Width and Height
  - **Description:** THIS SETTING IS NOT YET IMPLEMENTED.
  - **Enabled By:** This setting is enabled when the syntax is Cincinnati.

3.3.11 Optimizations

- Enable GPU Acceleration
  - **Description:** GPU acceleration is used to expedite the process of the optimization strategies by offloading the computation to the GPU. If a GPU is not present, or the setting is left disabled, the CPU will handle all calculations.

- Island Order Optimization
  - **Description:** Determine the ordering strategy for island printing within the layer.
  - **Options:**
    * Next Closest - finds the island with closest start point to the current island.
    * Next Farthest - the island at the farthest distance from the end point of the current island.
    * Shortest Distance (approximate) - island geometry is simplified and the center of mass is compared to the current island to find the shortest distance.
    * Shortest Distance (brute force) - islands are compared using their center of mass. This is more exact than the approximate method, but takes more time to compute for slices with numerous islands.
    * Least Recently Visited - the island that least recently had a layer printed.
    * Random - a randomly selected island will be printed next.
    * Custom Location - selects the island closest to a location defined by the user with additional settings.
  - **Enables:** The Custom Location option enables the settings Custom Island Point X Location and Custom Island Point Y Location.
- Custom Island Point X Location
  - **Description:** The X position for island order optimization.
  - **Enabled By:** This setting is enabled when Island Order Optimization is set to Custom Location.
- Custom Island Point Y Location
  - **Description:** The Y position for island order optimization.
  - **Enabled By:** This setting is enabled when Island Order Optimization is set to Custom Location.
- Path Order Optimization
  - **Description:** Determine the ordering strategy for the points within a path.

  - **Options:**
    * Next Closest - the point along the next path that is closest to the current point.
    * Next Farthest - the point along the next path that is the farthest from the current point.
    * Least Recently Visited - the path that has least recently been printed.
    * Random - a randomly selected point within the next path.
    * Consecutive -
    * Custom Location - selects the point along the next path closest to a location defined by the user with additional settings.
  - **Enables:** The Custom Location option enables the settings Custom Path Point X Location, Custom Path Point Y Location, Add Second Custom Location, Second Custom Point X Location, and Second Custom Point Y Location. The Consecutive option enables Consecutive Distance Threshold.

- Custom Path Point X Location
  - **Description:** The X position for path order optimization.
  - **Enabled By:** This setting is enabled when Path Order Optimization is set to Custom Location.

- Custom Path Point Y Location
  - **Description:** The Y position for path order optimization.
  - **Enabled By:** This setting is enabled when Path Order Optimization is set to Custom Location.

- Add Second Custom Location
  - **Description:** Allows for a second path order optimization location to be added. This will cause the optimization locations to alternate between layers with the second point being used on even numbered layers.
  - **Enabled By:** This setting is enabled when Add Second Custom Location is enabled and Path Order Optimization is set to Custom Location.

- Second Custom Point X Location

  - **Description:** The X position for path order optimization on even numbered layers.
  - **Enabled By:** This setting is enabled when Add Second Custom Location is enabled and Path Order Optimization is set to Custom Location.

- Second Custom Point Y Location
  - **Description:** The Y position for path order optimization on even numbered layers.
  - **Enabled By:** This setting is enabled when Add Second Custom Location is enabled and Path Order Optimization is set to Custom Location.

- Consecutive Distance Threshold
  - **Description:** The minimum distance to rotate the seam along the outer contour with the consecutive pathing strategy.
  - **Enabled By:** This setting Path Order Optimization is set to Consecutive.

3.3.12 Ordering

- Region Order
  - **Description:** Set the printing order for each region within the layer. This is a list box where each item can be click and dragged to rearrange the order. Item #1 prints first and item #5 prints last. The default order is Perimeter, Inset, Skeleton, Skin, and Infill.

- Reverse Perimeter Direction
  - **Description:** Reverse printing direction for perimeters from clockwise to counterclockwise.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.

- Reverse Perimeter Order
  - **Description:** Print perimeters from the inside out. By default, perimeters are printed from the outside in but this setting reverses the order.
  - **Enabled By:** This setting is enabled when Perimeters are enabled.

- Reverse Inset Direction

  - **Description:** Reverse printing direction for insets from clockwise to counterclockwise.
  - **Enabled By:** This setting is enabled when Insets are enabled.

- Reverse Inset Order
  - **Description:** Print insets from the inside out. By default, insets are printed from the outside in but this setting reverses the order.
  - **Enabled By:** This setting is enabled when Insets are enabled.

3.3.13 Laser Scanner

- Enable Laser Scanner
  - **Description:** Use a laser scanner to scan the printed paths and compare the printed geometry to the planned geometry. The Slicer will create ideal files to be exported and used for the comparison.
  - **Enabled By:** This setting is enabled when the syntax is set to Cincinnati or GKN.
  - **Enables:** This setting enables the following settings: Laser Scan Speed, Laser Scanner Height Offset, Laser Scanner X Offset, Laser Scanner Y Offset, Laser Scanner Height, Laser Scanner Width, Laser Step Distance, Laser Scan Line Resolution, Scanner Axis, Invert Laser Head, Enable Bed Scan, Scan Layer Skip, Enable Scanner Buffer, Buffer Distance, Transmit Height Map, Global Scan, Orientation Axis, Orientation Angle, Custom Orientation Definition, Angle A, Angle B, and Angle C.

- Laser Scan Speed
  - **Description:** Set the scan speed for laser scan motions.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scanner Height Offset
  - **Description:** Set the measured height offset between the extruder tip and the laser scanner.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scanner X Offset

  - **Description:** Set the X offset used when creating scanning toolpaths.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scanner Y Offset
  - **Description:** Set the Y offset used when creating scanning toolpaths.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scanner Height
  - **Description:** Set the ideal height for laser scanning motions.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scanner Width
  - **Description:** Set the laser scanner width at the Laser Scanner Height.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Step Distance
  - **Description:** Step distance of scanner along layer.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Laser Scan Line Resolution
  - **Description:** Resolution of a single scan line.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Scanner Axis
  - **Description:** Axis the scan line will travel.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Invert Laser Head
  - **Description:** Direction of mounted laser head with respect to machine axis.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Enable Bed Scan

  - **Description:** Scan the bed before printing the first layer.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Scan Layer Skip
  - **Description:** Multiple of layer to scan (1 = every layer, 2 = every other layer, etc.)
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Enable Scanner Buffer
  - **Description:** Allow scan lines to be offset by buffer distance to allow acceleration.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Buffer Distance
  - **Description:** Distance to offset scan lines to allow acceleration.
  - **Enabled By:** This setting is enabled when Laser Scanner AND Scanner Buffer are enabled.

- Transmit Height Map
  - **Description:** Transmit height map via socket to controlling software.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Global Scan
  - **Description:** Scan all objects as part of a global bounding box.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Orientation Axis
  - **Description:** Orientation axis of scan (use Z for vertical).
  - **Options:**
    * X
    * Y
    * Z
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.

- Orientation Angle
  - **Description:** Orientation angle of scan (use 0 for vertical).
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.
- Custom Orientation Definition
  - **Description:** Define angles that represent scanner orientation.
  - **Enabled By:** This setting is enabled when Laser Scanner is enabled.
- Angle A
  - **Description:** X axis rotation (pitch)
  - **Enabled By:** This setting is enabled when Laser Scanner AND Custom Orientation Definition are enabled.
- Angle B
  - **Description:** Y axis rotation (yaw)
  - **Enabled By:** This setting is enabled when Laser Scanner AND Custom Orientation Definition are enabled.
- Angle C
  - **Description:** Z axis rotation (roll)
  - **Enabled By:** This setting is enabled when Laser Scanner AND Custom Orientation Definition are enabled.

3.3.14 Thermal Scanner

- Enable Thermal Scanner
  - **Description:** Generates G and M-Codes for IR camera measurements between layers.
  - **Enabled By:** This setting is enabled when the syntax is Cincinnati.
  - **Enables:** This setting enables Thermal Scanner X Offset, Thermal Scanner Y Offset, Thermal Scanner Temperature Cutoff, and Move to Pyrometer Location.
- Thermal Scanner X Offset

  - **Description:** X offset from extruder to thermal scanner.
  - **Enabled By:** This setting is enabled when Thermal Scanner is enabled.

- Thermal Scanner Y Offset
  - **Description:** Y offset from extruder to thermal scanner.
  - **Enabled By:** This setting is enabled when Thermal Scanner is enabled.

- Thermal Scanner Temperature Cutoff
  - **Description:** Desired temperature to wait for.
  - **Enabled By:** This setting is enabled when Thermal Scanner is enabled.

- Move to Pyrometer Location
  - **Description:** Move to the pyrometer location after lifting during travel.
  - **Enabled By:** This setting is enabled when the syntax is GKN.

3.4 Experimental Settings

3.4.1 Printer Configuration

- Slicer Type
  - **Description:** The number of conformally mapped layers to generate for the conformal slicer.
  - **Options:**
    * Polymer
    * Conformal
    * Metal Embossing
    * Metal
    * Saturn
  - **Enables:** This setting, when set to Conformal, enables Max Segment Length and Conformal Layers. When set to Saturn, this enables Sector Overlap, Sector Stagger Enable, and Sector Stagger Angle.

- Layer Ordering
  - **Description:** How to order layers across multiple part.

  - **Options:**
    * By Height
    * By Layer Number
    * By Part (Sequential)

3.4.2 Curve Fitting

- Enable Arc Fitting
  - **Description:** Circular arc motions (G2/G3) will be fit along curves to reduce the amount of G1 motions.
  - **Enabled By:** This setting is enabled when Supports G2/G3 is enabled.
  - **Enables:** This setting enables Minimum Fitting Angle, Supporting Segment Distance, Arc Error Threshold, Arc Differentiation Tolerance, and Spline Differentiation Tolerance.
- Enable Spline Fitting
  - **Description:** Spline motions (G5) will be fit along curves to reduce the amount of G1 motions.
  - **Enabled By:** This setting is enabled when the syntax is not Marlin.
  - **Enables:** This setting enables Minimum Fitting Angle, Max Supporting Segment Distance, Arc Differentiation Tolerance, and Spline Differentiation Tolerance.
- Minimum Fitting Angle
  - **Description:** Minimum angle arcs and splines can be fit to.
  - **Enabled By:** This setting is enabled when Enable Arc Fitting OR Enable Spline Fitting is enabled.
- Max Supporting Segment Distance
  - **Description:** Maximum distance any segment can be to support an arc or spline.
  - **Enabled By:** This setting is enabled when Enable Arc Fitting OR Enable Spline Fitting is enabled.
- Arc Error Threshold

  - **Description:** Maximum allowed error when fitting arcs.
  - **Enabled By:** This setting is enabled when Enable Arc Fitting is enabled.

- Arc Differentiation Tolerance
  - **Description:** The percentage change in curvature that will result in an arc fit.
  - **Enabled By:** This setting is enabled when Enable Arc Fitting OR Enable Spline Fitting is enabled.

- Spline Differentiation Tolerance
  - **Description:** The percentage change in curvature that will result in a spline fit.
  - **Enabled By:** This setting is enabled when Enable Arc Fitting AND Enable Spline Fitting is enabled.

3.4.3 Single Path

- Enable Single Path Generation
  - **Description:** Single path generation employees an ORNL Patented algorithm to create a single extrusion path through the entire layer. This eliminates starts and stops and greatly speeds up printing time. When paired with spiralize, a single path can be generated across the entire print.
  - **Enabled By:** This setting is enabled when the Single Path Library DLL files are saved on the local computer.
  - **Enables:** This setting enables Disable Bridge Exclusion, Enable Single Path Zippering, Previous Layer Exclusion Distance, Corner Exclusion Distance, Max Bridge Beads, and Min Bridge Separation.

- Enable Bridge Exclusion
  - **Description:** Enforces the requirement that bridges for each layer not lie on top of previous layer (will allow you to not get the same location for bridges each layer).
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.

- Enable Single Path Zippering
  - **Description:** Attempts to alternate single path bridges from one layer to the next creating a zippered look on the part.
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.
- Previous Layer Exclusion Distance
  - **Description:** The minimum distance a bridge can be away from one on the previous layer
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.
- Corner Exclusion Distance
  - **Description:** The minimum distance from a corner where a bridge can occur.
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.
- Max Bridge Beads
  - **Description:** The maximum number of beads that a bridge can cross.
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.
- Min Bridge Separation
  - **Description:** The minimum distance between bridges.
  - **Enabled By:** This setting is enabled when Single Path Generation is enabled.

3.4.4 Slicing Angle

- Auto Slicing Angle
  - **Description:** Automatically calculates the slicing angle for each layer based on the geometry. Meant to be used for 5-axis printing.
- Slicing Axis

  - **Description:** The axis that the slicing plane will traverse for angled slicing.
  - **Options:**
    * X
    * Y
    * Z

- Stacking Direction Pitch
  - **Description:** The angle the slicing plane will be rotated about the x-axis. Note that if this value is invalid based on the other rotations and the slicing axis, the entire box will be colored red to warn the user that the operation is not possible.

- Stacking Direction Yaw
  - **Description:** The angle the slicing plane will be rotated about the y-axis. Note that if this value is invalid based on the other rotations and the slicing axis, the entire box will be colored red to warn the user that the operation is not possible.

- Stacking Direction Roll
  - **Description:** The angle the slicing plane will be rotated about the z-axis. Note that if this value is invalid based on the other rotations and the slicing axis, the entire box will be colored red to warn the user that the operation is not possible.

3.4.5 Conformal Slicing

- Max Segment Length
  - **Description:** The maximum length for any segment in the conformal slicer.
  - **Enabled By:** This setting is enabled when Slicer Type is Conformal.

- Conformal Layers
  - **Description:** The number of conformally mapped layers to generate for the conformal slicer.
  - **Enabled By:** This setting is enabled when Slicer Type is Conformal.

3.4.6 Saturn Slicing

- Sector Overlap
  - **Description:** Distance to extend each sector by.
  - **Enabled By:** This setting is enabled when Slicer Type is Saturn.
- Enable Sector Stagger
  - **Description:** Enables sector alignment alterations.
  - **Enabled By:** This setting is enabled when Slicer Type is Saturn.
- Sector Overlap
  - **Description:** Amount to alternate sector stagger distance.
  - **Enabled By:** This setting is enabled when Slicer Type is Saturn and Sector Stagger is Enabled.

3.4.7 Multi-Nozzle

- Enable Multiple Nozzles
  - **Description:** Enables options for experimental multi-nozzle features.
  - **Enables:** This setting enables Number of Nozzles, Nozzle Offset X, Nozzle Offset Y, Nozzle Offset Z, Nozzle Material, Enable Multiple Nozzles with Multi-Material, Remove Duplicate Paths, Duplicate Path Similarity, Enable Independent Nozzles, and Independent Nozzle Assignment Method.
- Number of Nozzles
  - **Description:** The number of nozzles available.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.
  - **Enables:** This setting enables Enable Multiple Nozzles with Multi-Material and Remove Duplicate Paths.
- Nozzle Offset X
  - **Description:** X-Distance of nozzle offset.

  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.

- Nozzle Offset Y
  - **Description:** Y-Distance of nozzle offset.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.

- Nozzle Offset Z
  - **Description:** Z-Distance of nozzle offset.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.

- Nozzle Material
  - **Description:** Material type used in nozzle.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.

- Enable Multiple Nozzles with Multi-Material
  - **Description:** If selected, segments of different materials will be printed with different nozzles.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled AND Nozzle Count is larger than 1 AND Enable Multi-Material is enabled.
  - **Enables:**

- Remove Duplicate Paths
  - **Description:** Removes duplicate paths if the paths can be printed simultaneously by multiple nozzles.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled AND Nozzle Count is larger than 1.
  - **Enables:**

- Duplicate Path Similarity

  - **Description:** Percent similarity required for paths to be considered duplicates.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled AND Remove Duplicate Paths is enabled.

- Enable Independent Nozzles
  - **Description:** Enables optimizations for multiple independent extruders.
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled.
  - **Enables:** This setting enables Independent Nozzle Assignment Method.

- Independent Nozzle Assignment Method
  - **Description:** Method to use for assigning islands to nozzles.
  - **Options:**
    * X Location
    * Y Location
    * Area
  - **Enabled By:** This setting is enabled when Enable Multiple Nozzles is enabled AND Enable Independent Nozzles is enabled.

3.4.8 Visualization

- Turn Off Visualization
  - **Description:** Enables 3D visualization of g-code.
  - **Enables:**

- Visualization Layer Skip
  - **Description:** Specifies a layer multiple to skip (ex: 1 = visualize every layer, 2 = visualize every 2nd layer, etc.)
  - **Enabled By:** This setting is enabled when Turn Off Visualization is NOT enabled.

4 Slicing Workflow

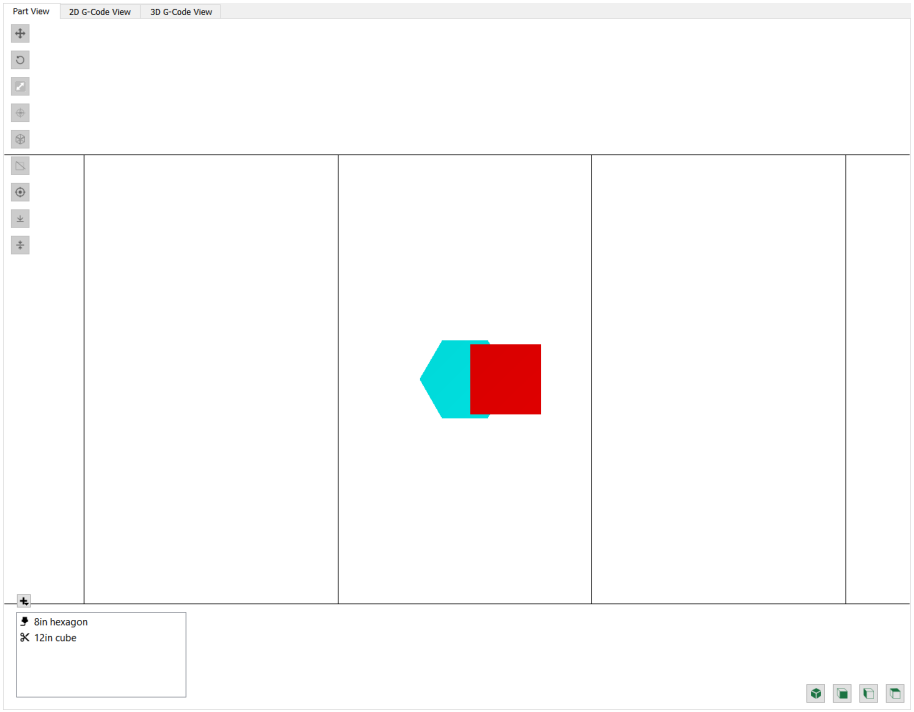
4.1 Loading and Orienting a Part

The first step of setting up a slice is to load and orient a model. In this context, model, part, STL, mesh, and object are all synonymous. A model can be loaded in three different ways: File -> Load Model for Building, Using the Add New Part Menu at the bottom left of the Part View, or by using the hotkey CTRL + O. When an object is loaded, by default it will be placed in the center of the build volume. The object can be translated, rotated, and scaled to meet the needs of the object and the print process. By default, the object is loaded with mm units but some CAD packages use inch as their default. This can be updated in the Scale Menu of the Part View. The object can be scaled about each axis, by a percentage. It can also be scaled in the X and Y by a distance using the Oversize Part settings in Profile Settings -> Special Modes. Additional parts can be added in the same way that the first part is loaded. The additional parts when loaded will default to the center, and then shift along the X-axis until there is no collision with previously loaded parts.

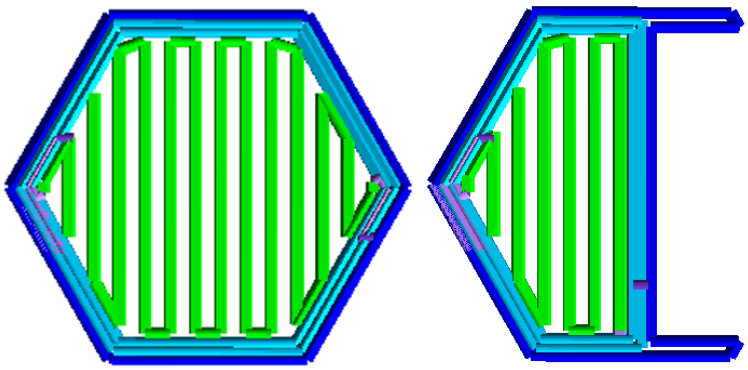
4.1.1 Building vs Clipping Meshes

A loaded mesh, or STL, can be set as a building or a clipping mesh. Once a mesh has been loaded, it can be interchanged between the two by using the right click menu within the viewer or within the object list.

- Building Mesh - an object that is meant to be built. When a building mesh is loaded, the slicing function will create G-Code toolpaths for the entirety of the building object. Slice settings can be applied to building meshes.
- Clipping Mesh - an object that is meant to be used to clip, or cut, from a building mesh. Clipping meshes do not create toolpaths when sliced; they alter the geometry of all building meshes that they intersect. Slice settings cannot be applied to clipping meshes.



In the above photo, a hexagon (blue) is loaded as a building mesh and a cube (red) is loaded as a clipping mesh. The objects are intersecting. The result is that the volume of the hexagon (the building mesh) that is intersecting with the cube (the clipping mesh) will be cut away from the output. The volume of the cube that is not intersecting with the hexagon does nothing, and the volume of the hexagon that does not intersect the cube will print as normal. The result can be seen below on the right. On the left is the output of just the hexagon without the clipping mesh.



Clipping meshes were implemented with the idea of simulating a non-flat build surface. A 3D scan of the build surface could be imported to the ORNL Slicer 2 as an STL and used to clip the bottom the object to be printed so that it appropriately prints on the new surface.

4.2 Adjusting Settings

Settings for an object can be manipulated in four different ways: Global Settings, Layer Specific Settings, Layer Range Settings, and Settings Region.

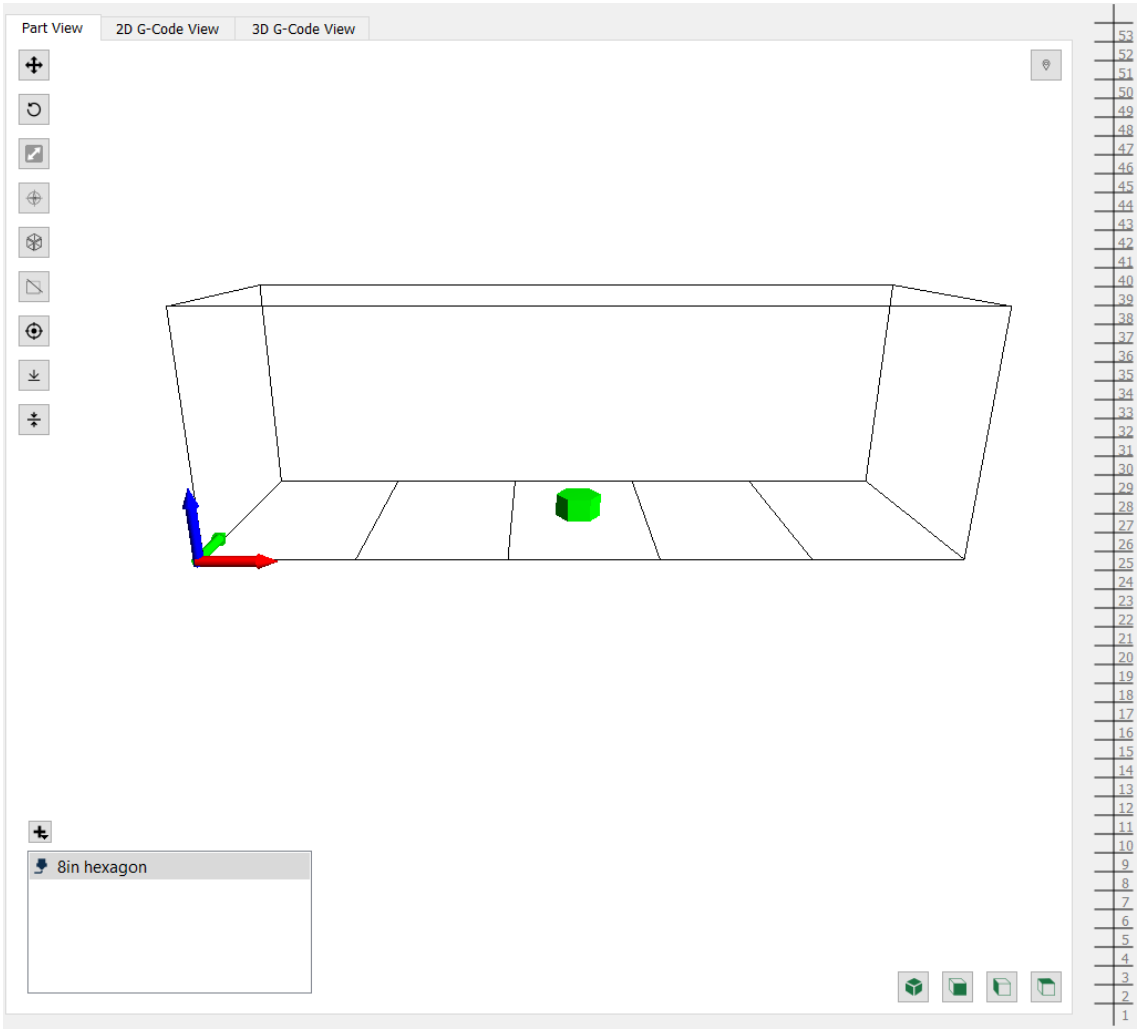
4.2.1 Global Settings

Global Settings are settings that are applied globally, which is to say to all parts and all layers. By default, settings that are being edited are global settings. Layer Specific, Layer Range, and Settings Regions are used to override the global settings for specific layers or regions of an object. How those are edited will be outlined in the following sections.

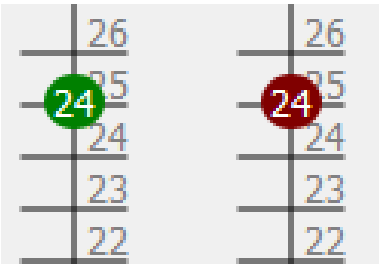
4.2.2 Layer Specific Settings

Layer Specific Settings are used to override the global settings on a specific layer. Layer Specific Settings can be as simple as changing the infill density on a single layer or as complex as changing the entire settings profile on a specific layer. Different layer specific settings can be applied to various layers throughout the part for as many layers as the part has.

To create specific settings for a layer, select the object in the Part View. This will generate a scale on the right side of the window that represents the number of layers based on the Layer Height set in the Profile Settings -> Layer -> Layer Height. This 8in Hexagon generates 53 layers, as seen in the scale.



Clicking on a number, will create a marker at that layer. The marker is initially green, to show that it is not selected and thus that the Global Settings are being viewed/edited. Clicking on the marker colors it red to show that it is now actively being edited. Any changes made to the settings are applied just to that specific layer. Clicking on it again will return it to green so that the Global Settings can be viewed/edited.

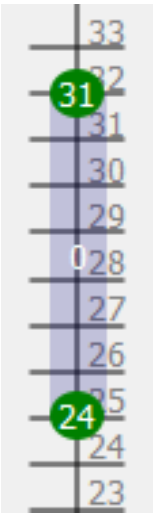


When a marker is selected (red), right clicking brings up a menu with three options:

- Set Layer - brings up a window to manually enter a layer number to move the marker to a different position
- Clear Selection - clear any layer specific changes that have been applied to that layer
- Delete Selection - remove the marker

4.2.3 Layer Range Settings

Layer Range settings work the same way as Layer Specific Settings, but allow the settings modifications to be applied to a consecutive range of layers. This is done by adding two markers, selecting both markers so that they are red, and then right clicking on one of the markers and selecting Maker Pair. This will create a blue rectangle between the two markers to show that they are paired.



4.2.4 Adding a Settings Model or Settings Region

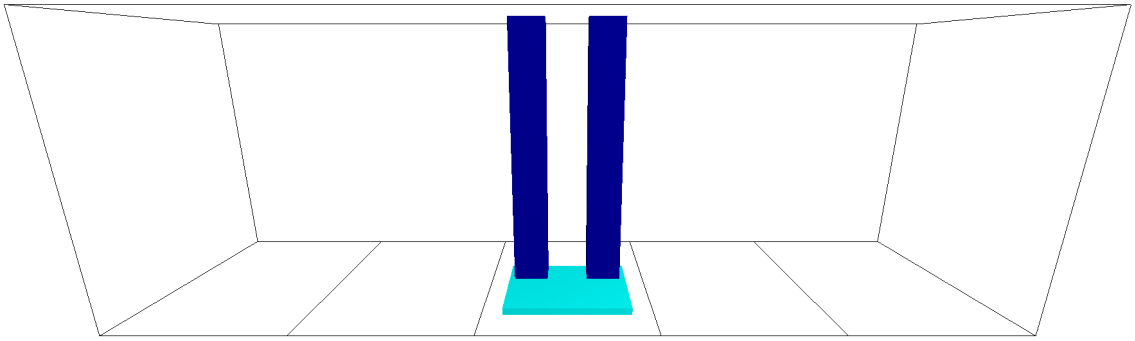
Settings models and settings regions are volumes where settings can be changed uniquely within the volume. The volume is meant to overlap with a building mesh so that within the settings model/region the speeds and/or infill density are different. Currently, perimeter speed, inset speed, and infill density are the only settings that can be changed within the settings model/region, but future work will allow the user to adjust more settings.

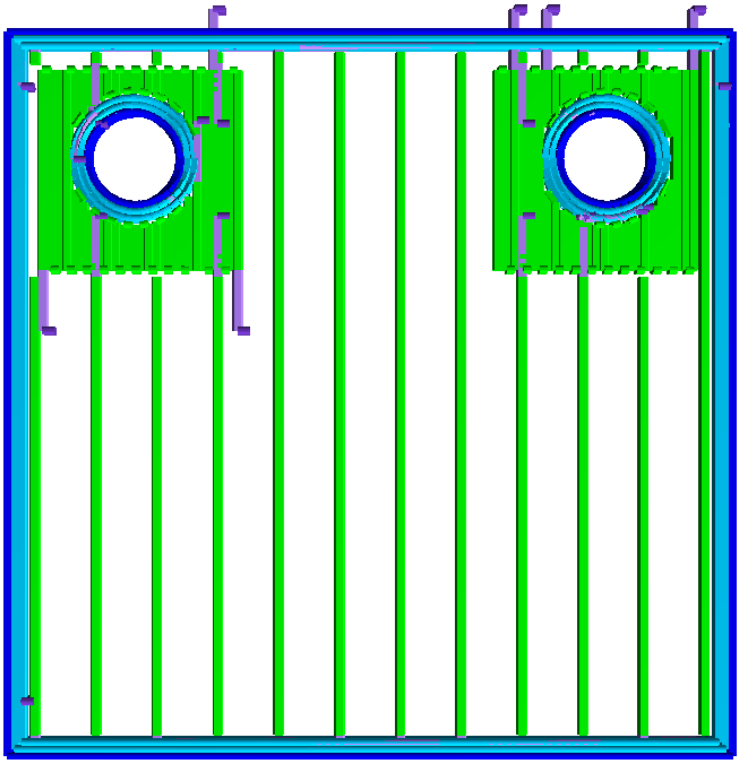
A settings region can be added as a box settings region in the create object menu of the Part View. This will prompt the user to enter a name for the settings region, then a hollow column shaped object will appear in the visualizer.

A settings model can be added in the load object menu of the Part View.

Settings regions and models are blue when deselected and purple when selected. They can be manipulated about the workspace in the same way as any other mesh (including rotation, translation, and scaling). When the settings region is selected, the slice settings for a settings region can be edited. When deselected, the global settings for the build meshes can be edited.

The typical use cases would be to create a settings region/model and move it over a section of a part that needs to be printed at a different speed (like printing slow on a critical face) or placing the region over an area that needs a different infill density (such as increasing the density around a hole that will be tapped/threaded). In the case of adjusting the speed, the paths entirely within the region are maintained with an updated speed. The paths that intersect the region are broken at the intersection so that the portion within the region is updated and the portion outside the region is maintained as is. In the case of adjusting the infill density, the volume within the region will have infill toolpaths planned at the new density. These paths will go right up to the edge to the region. The volume outside the region will keep the infill density of the building mesh and continue right up to the edge of the region. Below is an example of an object with settings regions used to create more dense infill immediately around holes.





4.3 Slicing a Part

Once a part has been loaded, oriented, and settings have been adjusted, it's time to slice the part. This can be done with the Slice Button, File -> Slice or by using the hotkey CTRL + G.

After Slicing, the 2D view is automatically pulled up along with the the G-Code editor. The G-Code can now be viewed, edited, and manipulated as needed. G-Code can be refreshed to show the changes or a reslice can be initiated if the object needs moved or settings need changed. Reslicing will only reslice what has changed from the previous slice in order to save time. This is done by tracking what changes have been made and what aspects of the slice are affected by these changes.

4.4 Saving Projects

The project system within ORNL Slicer 2 is helpful for saving your work to pickup at a later time or for referencing on a future build. A project is automatically created whenever Slicer 2 is running and the data is stored in the background. When the Slicer is closed, the current session and its data is autosaved and can be restored on the next startup by going to File -> Restore Last Session. Only one autosave session is stored at a time and it is therefore best to Save/Export any project that is of value. A project can be saved and exported as a single file by going to Project -> Save Project. This will open windows explorer to pick a location and name to save the project. The project is saved as a single file with the extension .s2p.

A project stores the following:

- All Loaded Meshes - this includes building meshes, clipping meshes, and settings regions. It also includes the location information of the meshes such as translation, rotation, and scaling data.
- Settings - this includes what profiles are loaded, what layer specific settings are in place, and what layer region settings are in place

4.5 Exporting G-Code

The last part of the ORNL Slicer workflow is to export the resultant G-Code so that it can be printed. The G-Code can be exported by itself, with the project, or with other associated files. All of this is available in the export window found at Project -> Export G-Code. A more detailed writeup of how to use the Export Window can be found in Section 2.3.3 G-Code Project Export Window.

5 Conclusion

The ORNL Slicer 2, and this documentation, are under active development at the ORNL MDF with new changes being made frequently. Development is spearheaded by Michael Borish (borishmc@ornl.gov) and Alex Roschli (roschliac@ornl.gov). The Slicer, it's code, and this documentation are all considered to be business sensitive and should not be distributed at this time.

To follow along with development, a public update log is visible at https://github.com/mdfbaam/ORNL-Slicer-2-Issue-Tracker/wiki/Update-Log This includes all updates as well as information about releases. The current development schedule plans for one major release each month. The expected date for the next upcoming release can be seen in the update log as well as the release dates for all previous versions.

5.1 Reporting Bugs and Requesting New Features

ORNL Slicer 2 is still in its infancy and is therefore prone to experiencing bugs. To see a list of all known bugs, as well as report any new bugs, please visit the public issue tracker at https://github.com/mdfbaam/ORNL-Slicer-2-Issue-Tracker/issues

Every issue in the list is assigned a priority and includes a detailed description of what is going wrong. The issue also includes a section for comments. We encourage all our users to follow along with the issue tracker to avoid any issues that may arise from known bugs. Users are also invited to create a free Github account and submit any bugs that are encountered, or comment on existing bugs.

The issue tracker is also used for requesting new features. Here one can create a new issue (only when logged into a Github account, which is free) and document the desired new functionality.