3D Printing: Design to Manufacturing

Fundamental Knowledge for a Successful Print

Corey Murphy
corey_murphy8@uri.edu

Overview

- CAD
  - Software options
  - What designs can and can’t be printed
  - Tips to ensure design is “printable"
- Slicing and 3D Printing
  - 3D Printers available to MCE Capstone students
  - Slicing software
  - Materials
  - How to choose the ideal slice settings for your print

3D Printing Design to Manufacturing Process

1. Design prototype/model via preferred CAD software
2. Evaluate and confirm “printability” of design
3. Export CAD design as STL file
4. Import STL file to slicing software
5. Optimize slice settings
6. Slice model, note print time/amount of material required for print
7. Save .gcode/.ufp file to flash drive
8. Load required filament to printer
9. Insert flash drive to printer and begin print
10. Monitor print to ensure successful completion

CAD Software

- Solidworks
  - Available through URI ECC computers
    - Remote access available
  - Student Editions available for purchase
    - 3DEXPERIENCE cloud-based package ($48 /year)
    - Desktop package ($99 /year)

CAD Software

- Autodesk Products
  - Inventor
    - Same capabilities as Solidworks, different UI
  - Fusion 360
    - Cloud-based integrated CAD/CAM/CAE and PCB software
      - Teams can all work together on CAD files
    - Excellent features geared toward 3D printing
  - All Autodesk products available for free to students
    - 1 year license, renewable as long as you remain eligible
    - Full professional editions, no “student” editions

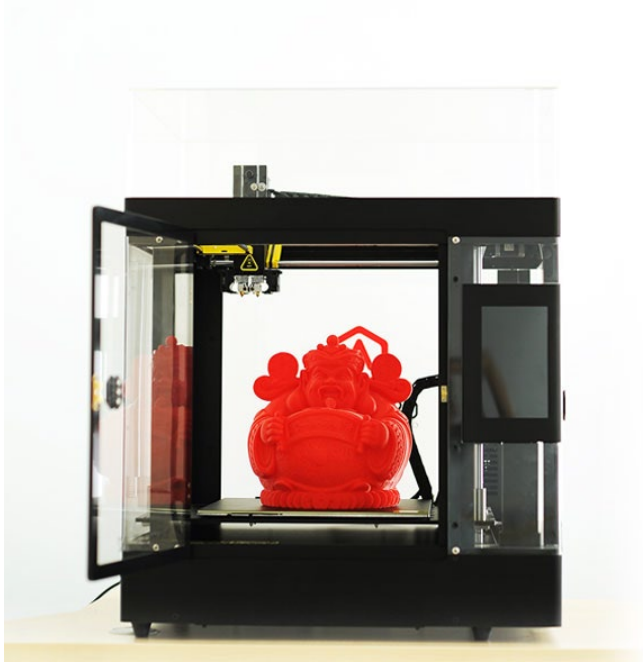
MCE Capstone 3D Printers

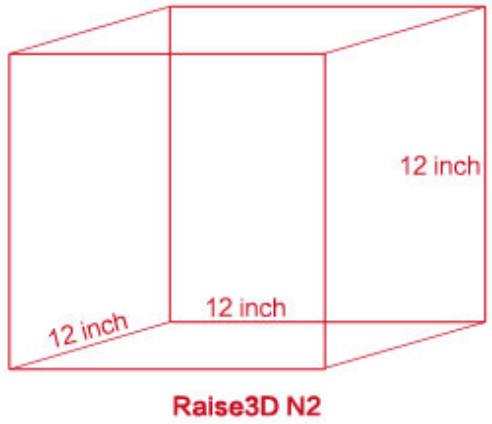
- Printers located in Makerspace of MCE Capstone room
- (2) Raise 3D N2
  - Ideal for initial prototypes and basic geometries
- (5) Ultimaker S5
  - Ideal for final prototypes and complex geometries
- Fused Deposition Modeling (FDM) 3D printers

Raise 3D N2

- Initial prototypes and basic geometries
- 2 printers operational in Makerspace
  - Single extruder w/ 80mm nozzle
  - Dual extruder w/ 80mm/60mm nozzles
- Larger nozzles = faster print times, decreased resolution
- 1.75mm filament
- Ideamaker slicing software

Raise 3D N2 Build Volume





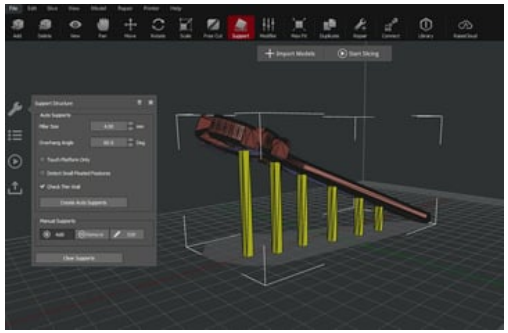
Raise3D N2

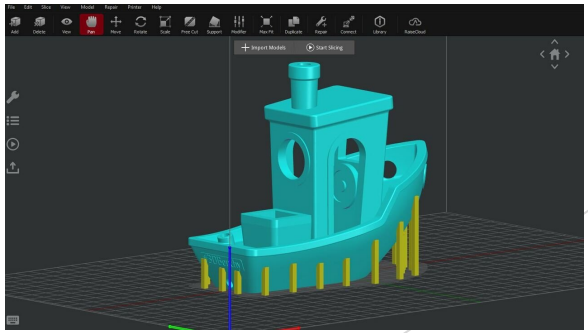
Build Volume
(W×D×H)

12×12×12 in (305×305×305 mm)

Ideamaker

- Default profiles for Raise 3D N2
- Configurable automatic presets
- Ideamaker Library/Raise3D Academy
  - User submitted profiles
  - Tutorials
- Slice converted to .gcode file
- Can be difficult to use
  - Must thoroughly check over settings prior to slicing



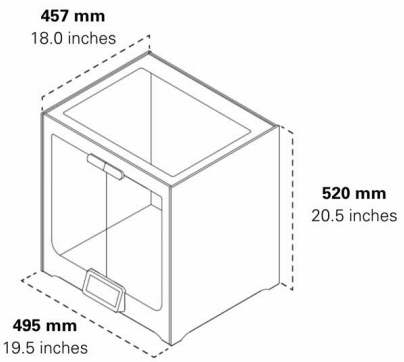


Ultimaker S5

- Final prototypes and complex geometries
- 5 printers operational in Makerspace
  - All dual extruder, 40mm nozzles
- Best choice for models that require support
  - Ultimaker specific support materials
- 2.85mm filament
- Cura slicing software

Ultimaker S5 Build Volume

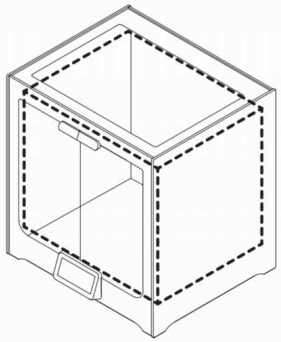
Outer Dimensions



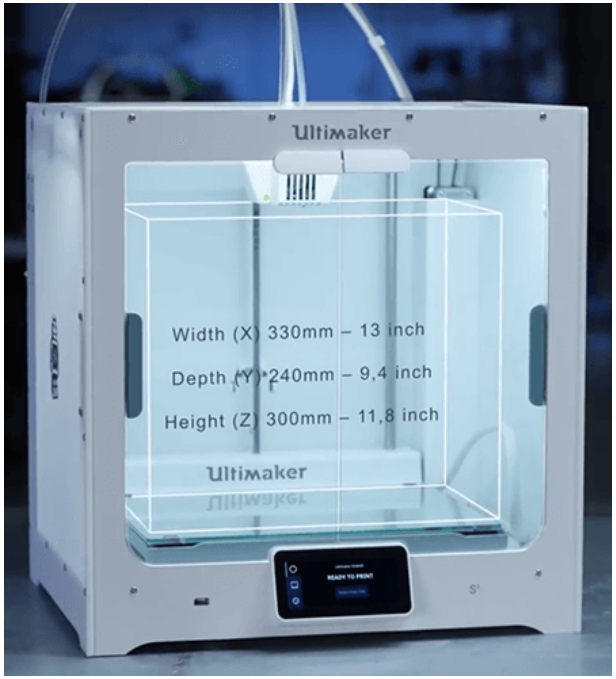
Dimensions with bowden tube and spool holder:
495 x 585 x 780 mm
(19.5 x 23 x 30.7 inches)

Weight:
20.6 kg (727 ounces)

Build Area

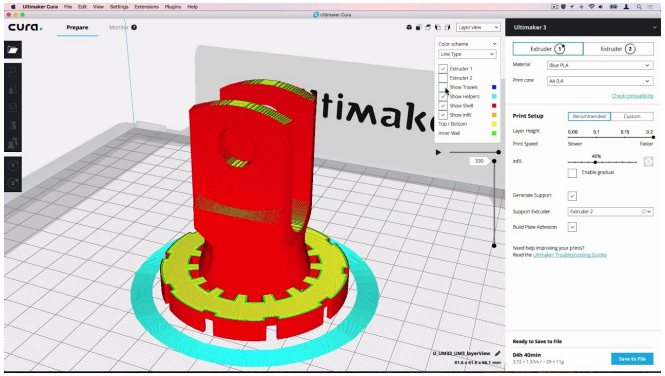


Left or right nozzle, or dual extrusion:
330 x 240 x 300 mm
(13 x 9.4 x 11.8 inches)



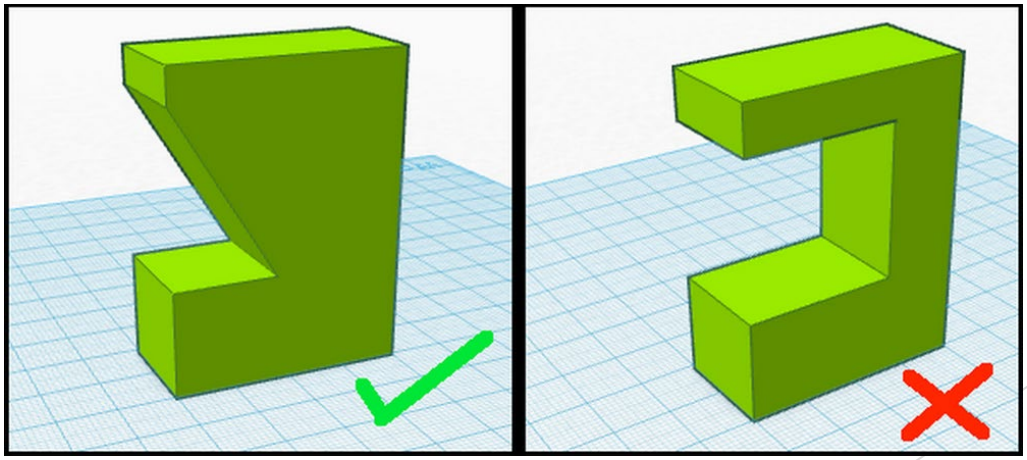
Cura

- Many default profiles for Ultimaker S5
  - Different materials
  - Default/Engineering/Visual/Draft
- Most popular slicing software
  - Plenty of helpful resources/forums/tutorials
  - Regular updates
- Customize scripts with plug-ins
- Slice converted to .ufp file



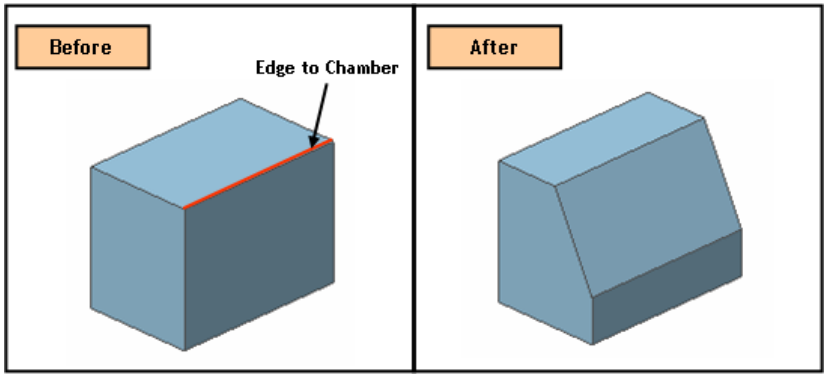
CAD Design Tips: Do I Need Supports?

- Consider the overhang angles on your model
  - Raise 3D N2: Overhangs ≥ 45 deg → Need supports
  - Ultimaker S5: Overhangs ≥ 60 deg → Need supports
- Supports are reliable if necessary, however:
  - Supports require more filament, longer print times
  - If using build material for support (Raise3D N2), surface finish can be compromised



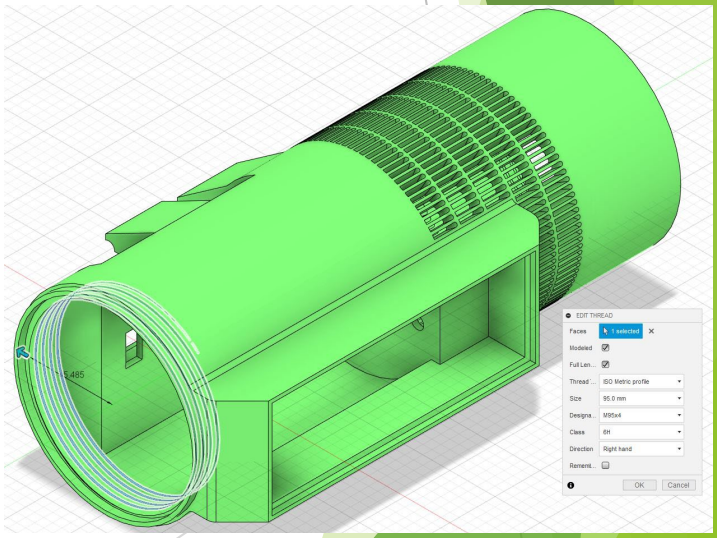
CAD Design Tips: Do I Need Supports?

- Avoid unnecessary overhangs ≥ max suggested angle
- Add chamfer to steep overhangs
  - Creates wider incline split into 45 deg segments
- Add anchors to design (create your own supports)
- Consider the base of the model for print
  - Orient model to appropriately to reduce need for supports
- Split model into multiple parts



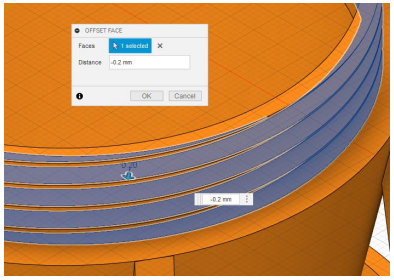
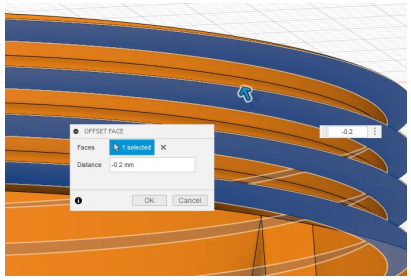
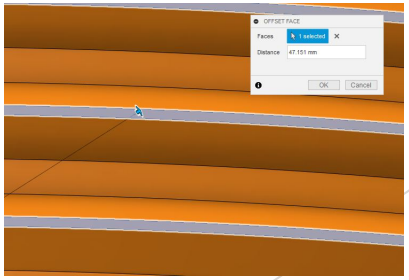
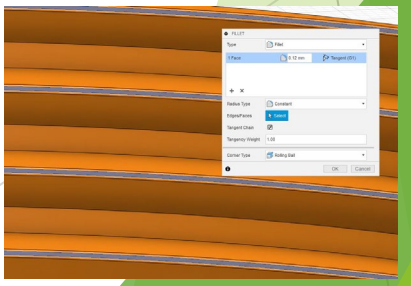
CAD Design Tips: 3D Printed Threads

- Solidworks/Fusion 360 support automated modeled thread creation
  - Inventor does not (can open .ipt files in Fusion 360)
  - Can create custom threads if necessary
- 3D Printed threads require additional tolerance
  - Tolerance changes based on:
    - CAD software (pre-programmed tolerances)
    - Printer
    - Material
    - Print settings (layer height, speed, etc.)



CAD Design Tips: 3D Printed Threads

- Ultimaker S5/Fusion 360 thread tolerance/settings
  - Tolerance applied only to outer thread:
    - 0.2mm offset applied to thread flanks/top of thread
    - 0.12mm fillet on top of thread
  - 0.15mm layer height
  - PLA/PLA+
- Threads DO NOT require supports

Available Filament

- Raise 3D N2 (1.75mm)
  - Build Material
    - PLA
    - PLA+
    - ABS
    - TPU
- Ultimaker S5 (2.85mm)
  - Build Material
    - PLA
    - PLA+
    - Tough PLA
  - Support Material
    - Breakaway White
    - PVA

PLA

- Pros
  - Low cost
  - Stiff, good strength
  - Easy to print
  - Versatile
  - Many color options
- Cons
  - Low heat resistance
  - Oozing
  - Filament can get brittle

PLA Printing Tips

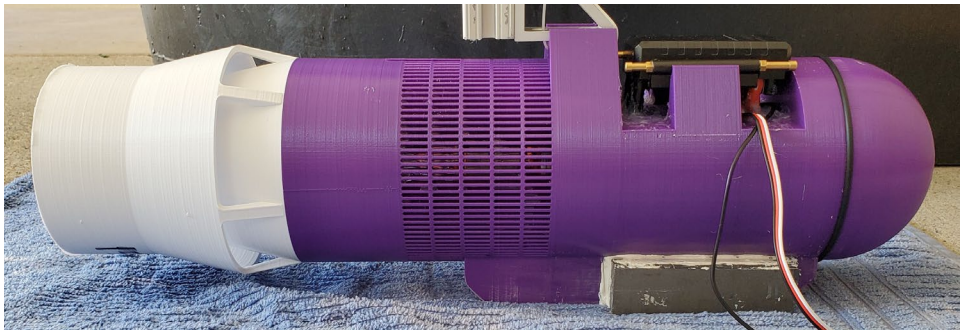
- Ultimaker S5
  - Load to nozzle 1
  - Requires AA 0.4mm nozzle
  - 200-210C nozzle temp
  - 60C bed temp
  - 100% cooling fans
  - No hair spray/glue required
  - Default/Engineering Cura profiles work well
- Raise 3D N2
  - 205-210C nozzle temp
  - 60C bed temp
  - 100% cooling fans
  - Apply hair spray to bed
  - Default print speeds
  - Always use brim

PLA+

- Pros
  - Similar benefits of PLA
  - Improved strength/thermal characteristics
  - Smoother finish
  - Water resistant (less prone to micro cracks)
- Cons
  - Relatively low heat resistance
  - Oozing
  - Slightly more expensive than standard PLA

PLA+ Printing Tips

- Ultimaker S5
  - Load to nozzle 1
  - Requires AA 0.4mm nozzle
  - 212C nozzle temp
  - 60C bed temp
  - 100% cooling fans
  - Set material to “Generic Tough PLA” in Cura
  - 9.5mm retraction distance
  - 45 mm/s retraction speed
  - No hair spray required
- Raise 3D N2
  - 210-215C nozzle temp
  - 60C bed temp
  - 100% cooling fans
  - Apply hair spray to bed
  - Default print speeds
  - Always use brim



ABS

- Pros
  - Low cost
  - Good impact resistance
  - Durable
  - Less oozing/stringing (smoother finish)
  - Good heat resistance
- Cons
  - Heavy warping
  - Dimensional inaccuracies due to parts shrinking
  - Requires enclosure
  - High print/bed temps
  - Can be difficult to print

ABS Printing Tips

- Raise 3D N2
  - Make sure to use enclosure
  - 230-250C nozzle temp
    - First few layers 10-20C higher
  - Avoid using cooling fans
  - 100-110C heated bed
  - Apply hair spray to heated bed
  - See Ideamaker site for specialized profiles

TPU

- Pros
  - Flexible, elastic characteristics
  - Excellent vibration dampening
  - Good impact resistance
  - Durable
- Cons
  - Difficult to print
  - Poor bridging characteristics
  - Blobs and stringing
  - Slow print times

TPU Printing Tips

- Raise 3D N2
  - Slow, consistent feed rate/print speeds
  - Print at lower layer heights (0.1mm recommended)
  - Reduce resistance from filament spool
  - Optimize retraction speeds to reduce sensitivity to quick movements, improve oozing/stringing
  - Optimize travel movements to reduce quick movements

Breakaway White

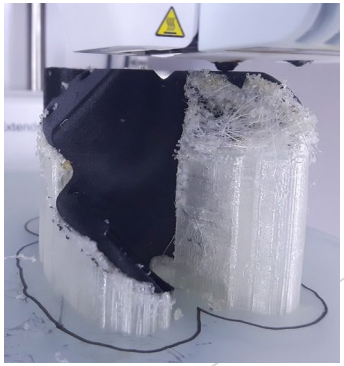
- Pros
  - Easy to remove
  - Easy to print
  - Leaves smooth surface finish (no sanding)
  - High dimensional accuracy
  - Adheres to many filaments
- Cons
  - Expensive
  - Oozing
  - Exclusive to Ultimaker printers

Breakaway White Printing Tips

- Ultimaker S5
  - Load to Nozzle 2
  - Requires AA 0.4mm nozzle
  - 227C nozzle temp
  - 7.5mm retraction distance, 45mm/s retraction speed
  - Concentric infill for circular models, triangular infill for most other models
  - 10-15% infill
  - Always use brim
  - Apply hairspray to bed

PVA

- Pros
  - Water soluble
  - Great for support required in tough to reach places
  - Adheres to PLA/CPE/Nylon/ PETG
- Cons
  - Expensive
  - Requires proper storage
  - Difficult to print
  - Removal can be time consuming
  - Stringing/oozing



PVA Printing Tips

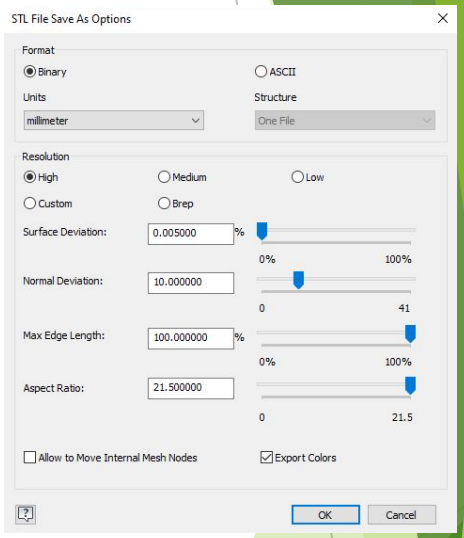
- Ultimaker S5
  - Load to nozzle 2
  - Requires BB 0.4mm nozzle
  - Apply hairspray to bed
  - Triangular infill for most models
  - 20% infill
  - 215C nozzle temp
  - Always use brim
  - Default/Engineering Cura profiles work well
  - ALWAYS store PVA in resealable bag with desiccant after use to prevent moisture absorption

Slicing Basics

1. Exporting/Importing STL File
2. Print Temperatures
3. Layer Height
4. Infill Patterns/Percentages
5. Print Speed
6. Wall/Shell Quality
7. Retraction
8. Supports/Adhesion

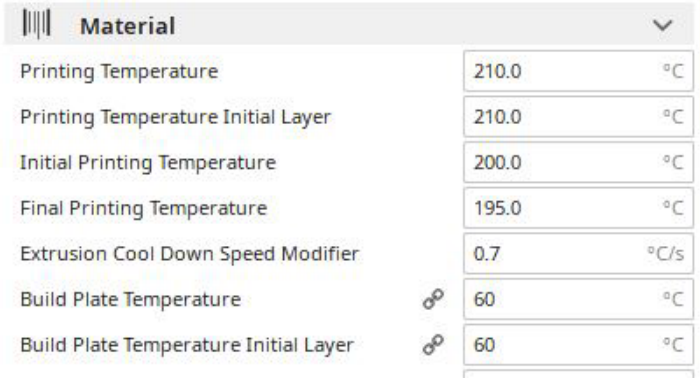
Exporting/Importing STL File

- Must export CAD model as STL file
  - Inventor: File > Export > CAD Format > Save as type: STL Files (from dropdown)
  - Prior to saving, select Options
    - Format = Binary, Units = millimeter, Resolution = High
  - Hit OK and save
  - Similar process for Solidworks/Fusion360
- Import STL file to slicing software
  - Cura: File Icon > Select STL File > Open
  - Ideamaker: File > Import Models > Select STL File > Open



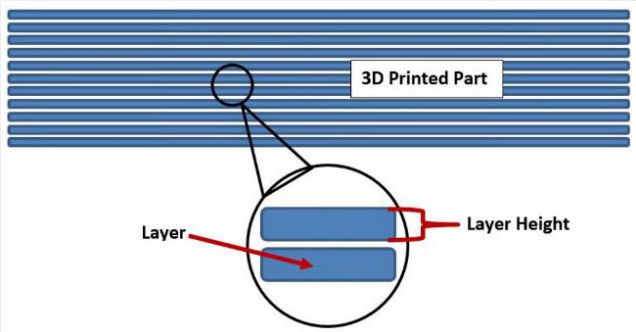
Print Temperature

- Nozzle/bed temp dependent on specific filament
  - See filament spool/material data sheet for info
- Optimizing temps ensures material extrudes and adheres properly
  - Some materials are more temp resilient than others
  - Higher temps can cause oozing/stringing
  - Lower temps can cause under extrusion
- Check to see if material requires cooling
  - Always use cooling fans with PLA
  - Do not use cooling fans with ABS/Nylon

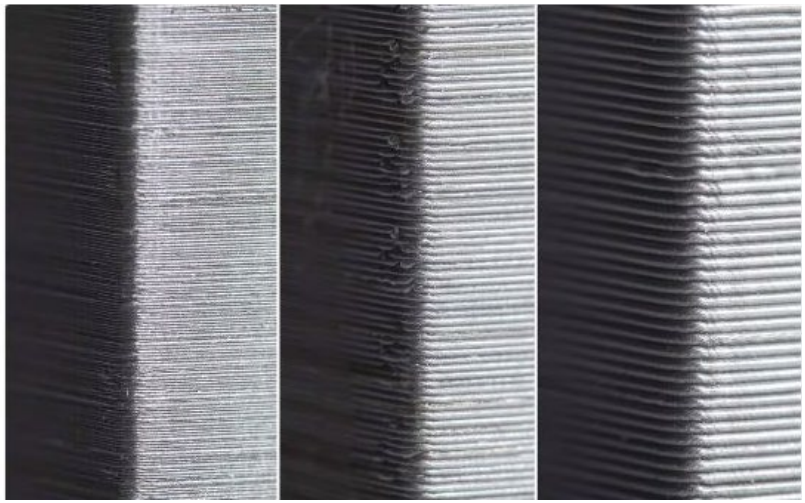


Layer Height

- Controls the height of each individual printed layer (mm)
- Smaller layer height:
  - Better quality print
  - Uses less material
  - Print will take proportionally longer to complete
    - 0.1mm layer height print takes ~2x longer than 0.2mm layer height print
- Consider the requirements for your design
  - Initial prototype → larger layer height (0.2mm)
  - Time constraints → larger layer height
  - Final prototype → smaller layer height (0.15mm/0.10mm)
  - Complex geometry → smaller layer height

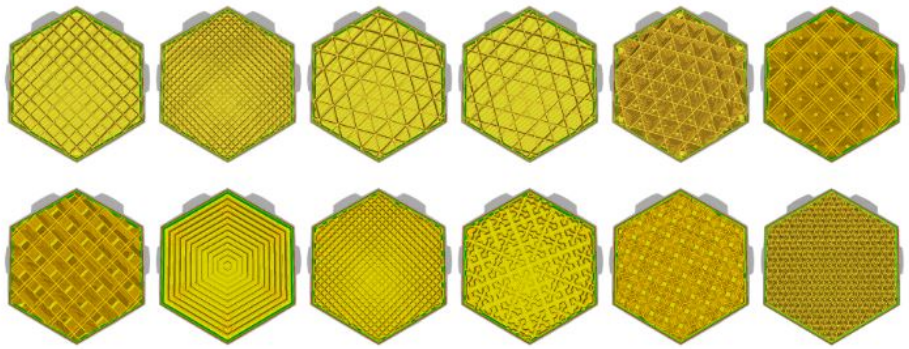


Layer Height



Infill Patterns

- **Grid:** Strong 2D infill
- **Lines:** Quick 2D infill
- **Triangles:** Strong 2D infill
- **Tri-hexagon:** Strong 2D infill
- **Cubic:** Strong 3D infill
- **Cubic (subdivision):** Strong 3D infill (this saves material compared to Cubic)
- **Octet:** Strong 3D infill
- **Quarter cubic:** Strong 3D infill
- **Concentric:** Flexible 3D infill
- **Zig-zag:** A grid shaped infill, printing continuously in one diagonal direction
- **Cross:** Flexible 3D infill
- **Cross 3D**: Flexible 3D infill
- **Gyroid infill**: Infill with increased strength for the lowest weight.



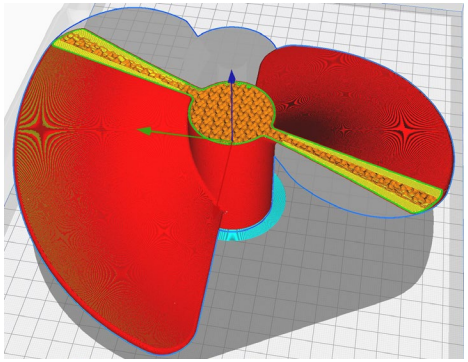
The infill patterns are displayed in the order of the list above, from left to right.

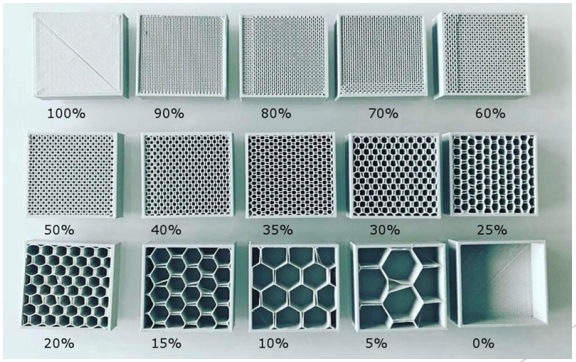
Choosing Infill Pattern/Percentage

- Consider the requirements for your design
  - Non-functional models/figurines (low strength)
    - Pattern: Lines, Zig-zag
    - Percentage: 0-15%
  - Functional models/prototypes (medium strength)
    - Pattern: Grid, Triangles, Tri-hexagon
    - Percentage: 15-30%
  - Functional, load bearing models/prototypes (high strength)
    - Pattern: Cubic, Cubic Subdivision, Octet, Quarter Cubic, Gyroid
    - Percentage: >30%
  - Flexible models
    - Pattern: Concentric, Cross, Cross 3D
    - Percentage: Variable, depends on flexibility required

Additional Infill Tips

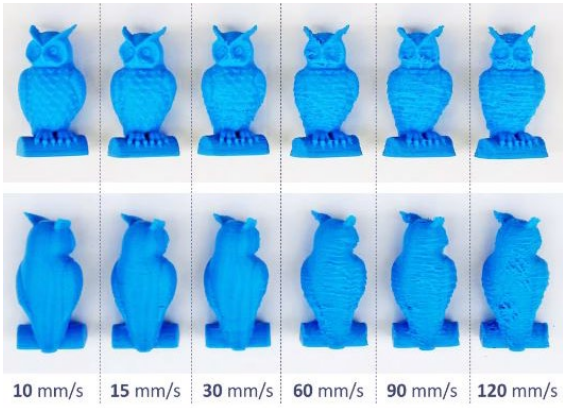
- Some patterns increase print time and material usage more than others
- Consider direction of load applied to your design
  - Triangles strong when load applied perpendicular to model's face
  - Gyroid strong for model stressed in multiple directions
- Infill percentage should be increased to enhance strength
  - Parts requiring additional mass should use ballast (lead)





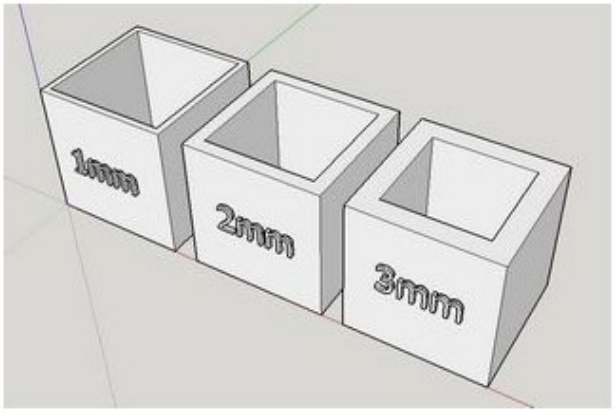
Print Speed

- Print Speed controls print time and affects print quality
  - Increase speed → reduce print time, reduce quality
    - Filament might not have sufficient time to melt, results in brittle model
    - Increased vibrations could lead to ripples
  - Print speeds vary based on material
    - PLA ~ 60 mm/s, TPU ~ 30 mm/s
  - Optimize speeds with 5 mm/s increments or Cura plugin



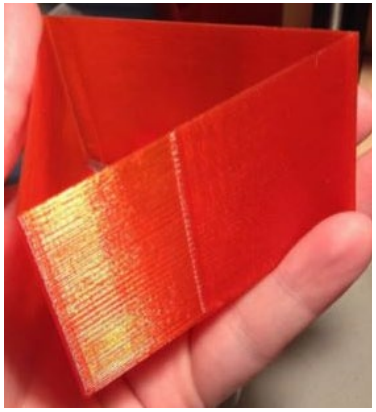
Wall/Shell Quality

- Wall Thickness adjusts total thickness of outer walls
  - Value should be integer multiple of nozzle size
    - ie: 0.4mm nozzle, 3 walls required → Wall Thickness = 1.2mm
    - Higher value = sturdier model, increased print time/material usage
  - If width of your wall is between multiple of nozzle size
    - Use Fill Gaps Between Walls setting to prevent gaps
  - Different wall thickness setting for top/bottom walls



Z-Seam Alignment

- Z-Seam Alignment adjusts where the printer starts each layer
  - Model will have prominent line along this location
- 4 different Z-Seam Alignment settings
  - Shortest: Instructs printer to start new layer from endpoint of previous layer
  - User Specified: Start each layer at user specified area on model
  - Sharpest Corner: Start each layer from sharpest corner on model
  - Random: Start each layer at random position



Supports

- Model will require supports for overhangs
  - ≥ 45 degrees (Raise 3D N2)
  - ≥ 60 degrees (Ultimaker S5)
  - Bridges > 5mm
- Raise 3D N2 limited to build material for support (PLA)
  - Fine for simple supports/initial prototypes
- 2 support specific filaments available for Ultimaker S5
  - Best for complex supports/final prototypes

Important Support Settings

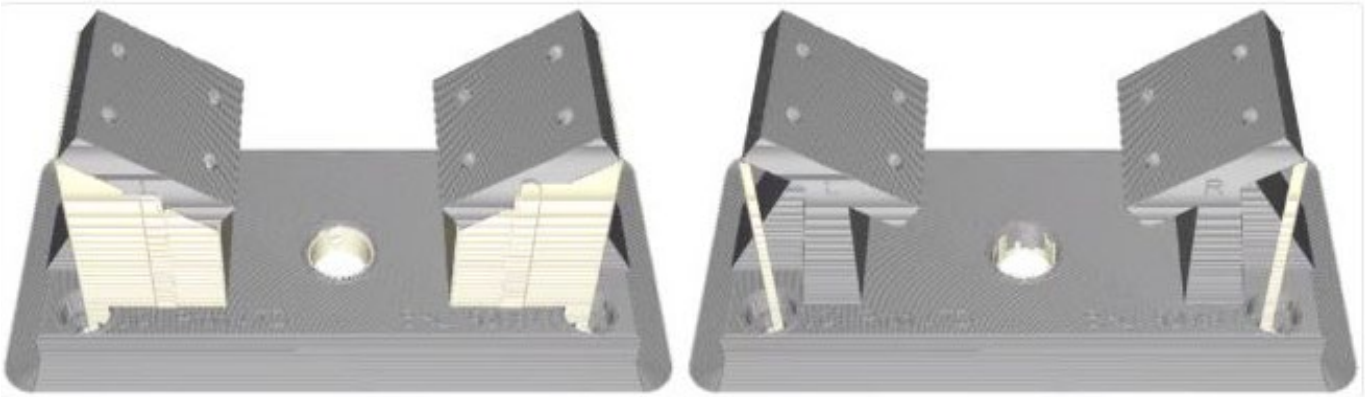
- Overhang Angle
  - Specifies minimum overhang angle for supports to be printed
- Support Infill Pattern/Percentage
  - Different from build material infill pattern/percentage
  - 5 support infill patterns offered
    - Zig-zag/Lines: Fastest to print, easiest to remove, weakest
    - Grid/Triangles: Sturdiest, vibration resilient, increase print time/material usage
    - Concentric: Best for cylindrical/spherical models
  - Infill percentage between 10-20% sufficient for most cases

Important Support Settings

- Support Z and X/Y Distance
  - Z distance defines distance from top and bottom of support to the model
  - X/Y distance adjusts defines horizonal distance b/w model and support
  - High value = larger gap
    - Easier removal, smoother surface
  - Low value = smaller gap
    - Complex overhangs, stronger supports

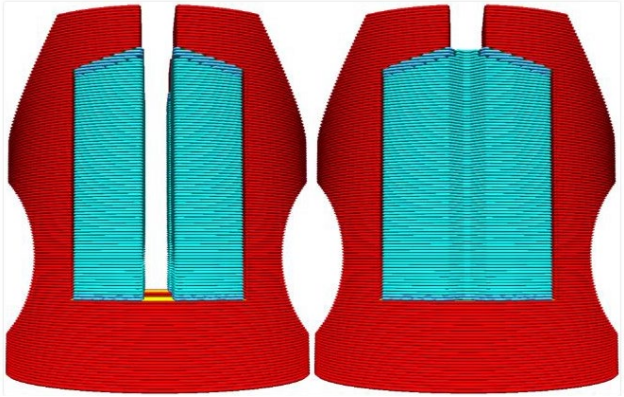
Important Support Settings

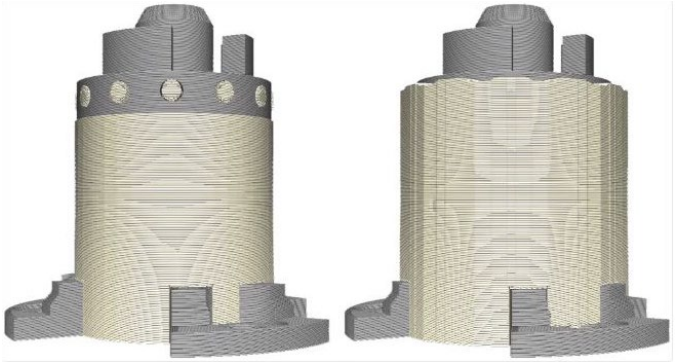
- Support Placement
  - Touching Buildplate
    - Supports built only originating from buildplate
  - Everywhere
    - Supports built everywhere Cura deems necessary



Important Support Settings

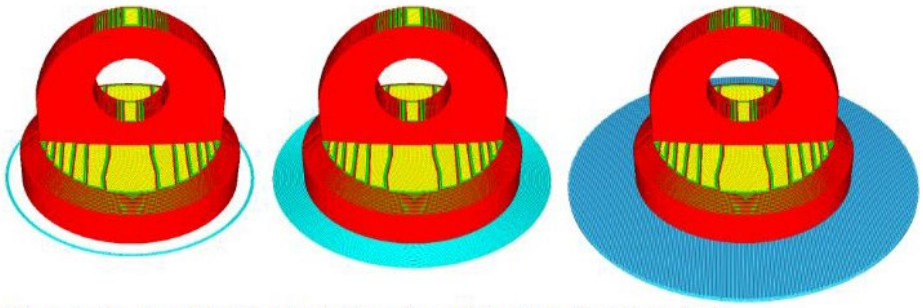
- Join Distance
  - Adjusts the max horizontal distance between support structures
  - Support structures closer than this value will be printed as one structure
- Horizontal Expansion
  - Set value for horizontal expansion to ensure small support areas are adequately supported





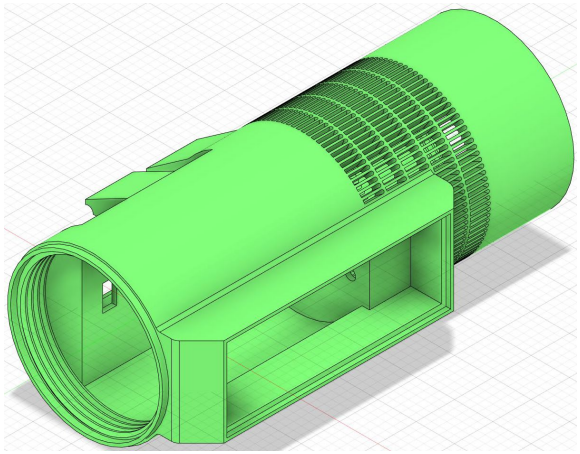
Adhesion

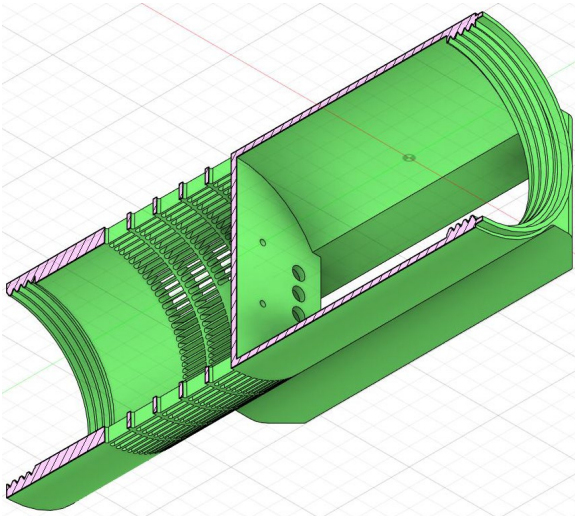
- Skirt
  - Line printed around model on first layer
  - Helpful for priming extrusion nozzle
- Brim
  - Single layer flat area around the base of model
  - Helps prevent warping and improve adhesion
- Raft
  - Thick grid with a roof between model and build plate
  - Useful when bottom surface of model is not completely flat
- Consider geometry of model base/material used
  - Large contact area will only require skirt

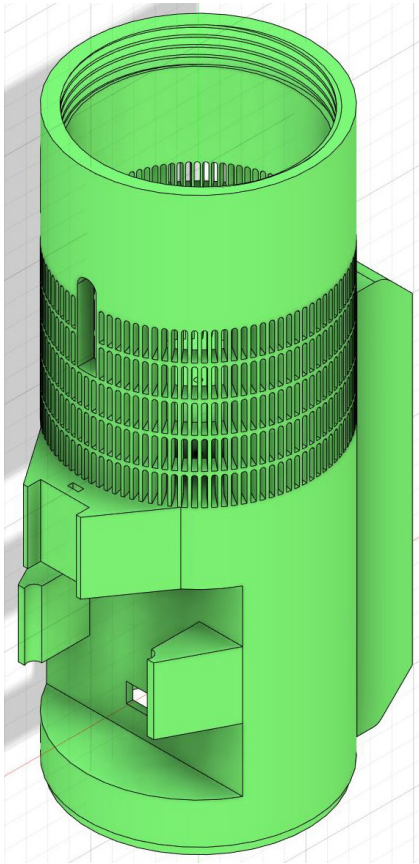


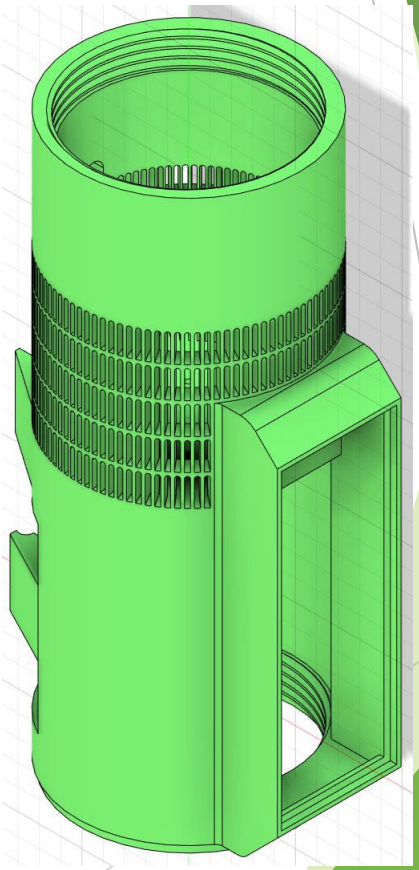
This model has the build plate adhesion types from left to right: skirt, brim and raft

Complex Support Example



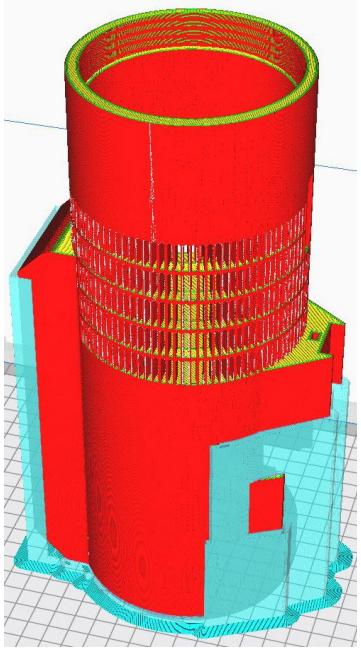


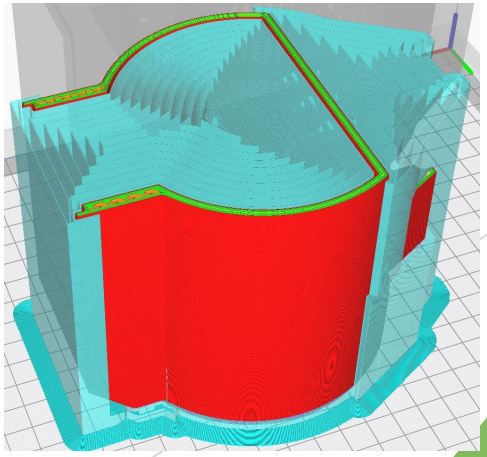




Complex Support Example

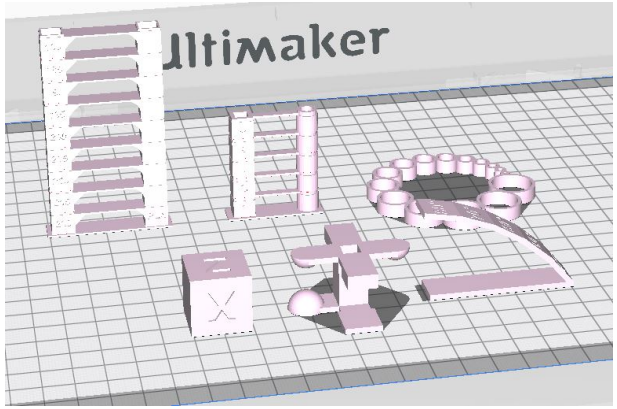
- Overhang Angle = 86 deg
- Infill Pattern = Concentric
- Infill Percentage = 10%
- Z Distance = 0.3mm
- X/Y Distance = 0.875mm
- Support Placement = Everywhere
- Join Distance = 5.0mm
- Horizontal Expansion = 3.6mm





Additional Cura Tips

- Some settings are hidden by default
  - Print settings > Custom > ≡ > Select desired amount of visible settings
- Cura supports plugins to modify print code
  - Cura > Marketplace > Plugins
  - Calibration Shapes Plugin great for settings optimization
    - TempTower, Retract Tower, Tolerance Test, etc.



Once Slice Settings are Optimzied

- Cura
  1. Select ‘Slice’
  2. Note Time Estimation
  3. Note Material Estimation
  4. Select ‘Save to Removable Drive’
  5. Save .ufp file to USB
- Ideamaker
  1. Slice > Start Slice
  2. Select Template
  3. Select ‘Slice’
  4. Note Estimated Print Time
  5. Note Estimated Amount
  6. Select ‘Export’
  7. Save .gcode file to USB

Weigh Required Material

1. Take empty spool of filament you will use for print
   - Empty spools of various filaments located in corner Raise 3D N2
2. Place empty spool on scale next to printer
3. Tare scale and remove empty spool
4. Place full/semi-full spool of material you will use on scale
5. Note the displayed mass
   - This is the amount of material left on spool
6. Ensure this amount is sufficient for your print
   - Typically want +20g more material on spool than material estimation from slicing software

Load Material Raise 3D N2

1. Navigate to ‘Utilities’ menu
2. Select ‘Load’ on appropriate extruder
3. Wait for nozzle to heat up
4. Open side door, load material onto spool
5. Feed filament through the guide tube
6. Push filament down to the feed gear until gear grips filament
7. Allow new filament to extrude for a few seconds
8. Select ‘OK’ when complete

Start Print Raise 3D N2

1. Plug in USB Drive to printer
2. Navigate to ‘Print’ Menu
3. Choose ‘USB Storage’
4. Navigate to the .gcode file you wish to print and select
5. Select ‘Print’ to start
6. Apply hairspray if necessary
7. Printer will extrude some material right before starting print, make sure to remove

Load Material Ultimaker S5

Load material 2

Material 2 will be loaded first because this is the material that must be placed closest to the back of the printer. Select Material 2 from the list shown on the touchscreen, select *Start*, and then take the following steps to load the material.

1. Place the spool with material 2 (PVA) onto the spool holder and select *Confirm*. Make sure the end of the material points in a clockwise direction, so that the material can enter feeder 2 from the bottom
2. Wait until the Ultimaker S5 detects the material and *Confirm*
3. Insert the end of the material into feeder 2 and gently push it until the feeder grips it and the material is visible in the Bowden tube. Select Confirm to continue
4. Wait for the Ultimaker S5 to heat up print core 2 and load the material into the print head
5. Confirm when the new material extrudes consistently from print core 2
6. Wait a moment for print core 2 to cool down

Tip: When using a third-party material you can select the material type manually.

Note: You can straighten the end of the material a bit so it can enter the feeder easier.

Load material 1

Material 1 will be put on the material guide first before placing it on the spool holder in order to avoid any tangling of the 2 materials during printing. Select material 1 from the list on the touchscreen, select *Start*, and follow the steps below.

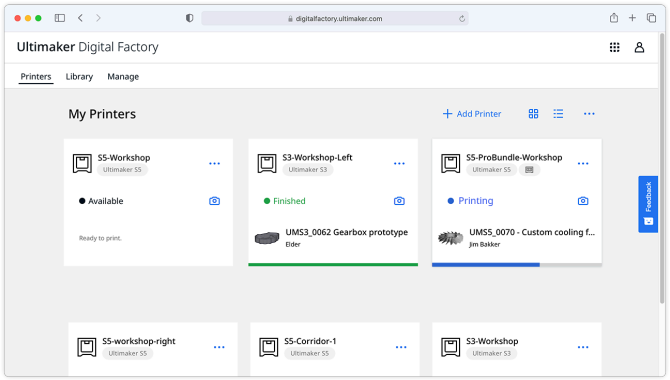
1. Take the material guide and hold it with the outer part towards you
2. Place the material spool with material 1 (Tough PLA) on the material guide with the material in a counter-clockwise direction, and guide the end of the material through the hole in the material guide
3. Place the material guide with material 1 on it onto the spool holder behind material 2, and select *Confirm*
4. Wait until the Ultimaker S5 detects the material and select *Confirm*
5. Insert the end of the material into feeder 1 and gently push it until the feeder grips it and the material is visible in the Bowden tube. Select *Confirm* to continue
6. Wait for the Ultimaker S5 to heat up print core 1 and to load the material into the print head
7. Confirm when the new material extrudes consistently from print core 1
8. Wait a moment for print core 1 to cool down

Tip: When using a third-party material, you can select the material type manually.

Note: You can straighten the end of the material so it can enter the feeder easier.

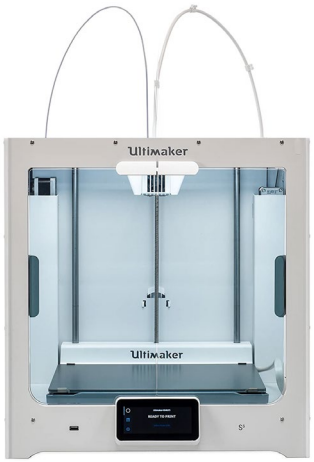
Connect to Ultimaker Digital Factory

1. Create Ultimaker Digital Factory Account
2. Navigate to printer ‘Settings’ menu
3. Select ‘Network’ settings
4. Scroll down and select ‘Ultimaker Digital Factory’
5. Select ‘Connect’
6. The printer will display a code, copy this down
7. Go to the Digital Factory site and sign in
8. On the ‘Printers’ page, click ‘Add printer’
9. Type in code to connect to printer



Start Print Ultimaker S5

1. Plug in USB Drive to printer
2. Select ‘Print’
3. Navigate to the .ufp file you wish to print and select
4. Start print
5. Apply hairspray if necessary, close print doors



Monitoring Print

1. Always stay in Makerspace for first layer
   - Most print failures occur during first layer
2. Routinely check on print
   - Use Digital Factory for Ultimaker S5 prints
   - Make sure to terminate print job ASAP if print fails
     - Printers don’t automatically detect all print failures

Post-Processing

1. Remove print, confirm removal on printer
2. Unload material and return material to filament workbench
3. Remove heated bed and clean with soap and water
4. Use shop vac to remove extra extruded material in printer
5. Reinstall heated bed

Software Links

- Autodesk Student Products: https://www.autodesk.com/education/edu-software/overview?sorting=featured&filters=individual
- Solidworks Student Products: https://www.solidworks.com/product/students
- Raise3D Ideamaker: https://www.raise3d.com/ideamaker/
- Ultimaker Cura: https://ultimaker.com/software/ultimaker-cura

Useful Resources

- Cura Slicing Tutorial: https://all3dp.com/1/cura-tutorial-software-slicer-cura-3d/
- Optimize Cura Support Settings: https://all3dp.com/2/cura-support-settings-optimize-your-supports/
- Ideamaker Library: https://www.ideamaker.io/index.html
- Raise3D Academy: https://www.raise3d.com/academy/
- Fusion360 Tutorials: https://www.youtube.com/c/ProductDesignOnline/videos
- 3D Printing Troubleshooting Guide: https://all3dp.com/1/common-3d-printing-problems-troubleshooting-3d-printer-issues/#print-looks-stringy-and-droopy-over-extrusion
- Infill Patterns: https://all3dp.com/2/cura-infill-patterns-all-you-need-to-know/
- Cura Calibration Shapes Plugin: https://marketplace.ultimaker.com/app/cura/plugins/5axes/CalibrationShapes
- Cura Calibration Shapes Plugin Guide: https://github-wiki-see.page/m/dotdash32/Calibration-Shapes/wiki
- Ultimaker S5 User Guide: https://support.ultimaker.com/hc/en-us/sections/360003504180-Ultimaker-S5
- Raise 3D N2 User Guide: https://support.raise3d.com/list.html?cid=3&pid=-1

Questions?