3D Printing for Model Railroading

- Also called “Additive Manufacturing”
- Two Popular Consumer Technologies:
  - Fused Deposition Modeling (FDM)
  - Resin

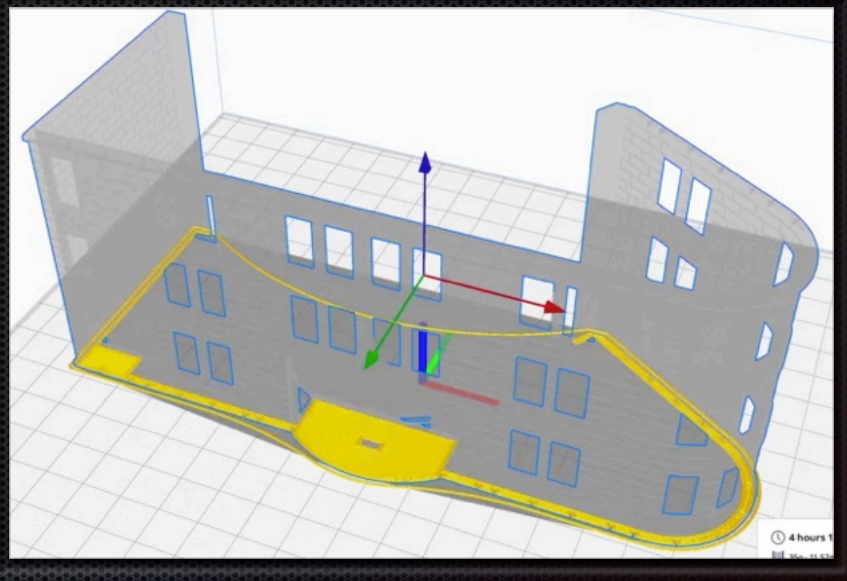
Pros and Cons of FDM vs. Resin

Excellent comparisons at https://3dsolved.com/fdm-filament-vs-resin-printers

- FDM
  - Inexpensive printers and filaments are available
  - Non-toxic/no fumes options are common
  - Large build volumes are available
  - FDM printers are easier/cheaper for beginners
- Resin
  - Resin printers usually have thinner layers and can print smaller more detailed parts
  - Resin printers typically have smaller build volume
  - You must have protective gear
  - Materials and equipment are generally more expensive

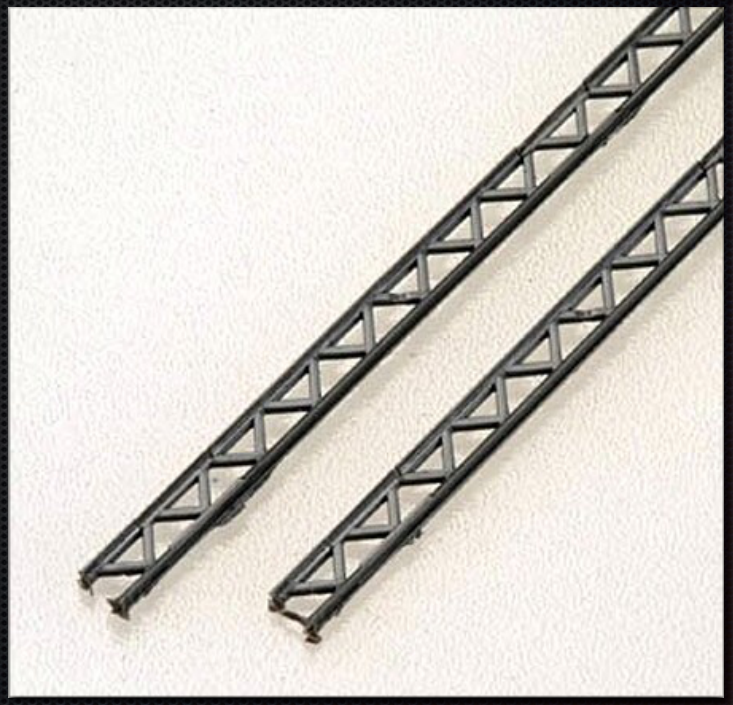
Fused Deposition Modeling (FDM)

- Thermoplastic filament is squeezed through a nozzle to build layer after layer of a model.



Many Materials Are Used With FDM

- Polylactide (PLA)
  - PLA mixed with wood pulp
- Acrylonitrile butadiene styrene (ABS)
  - ABS mixed with brass or other metals
- Polyethylene terephthalate (PTE)
- Polyvinyl chloride (PVC)



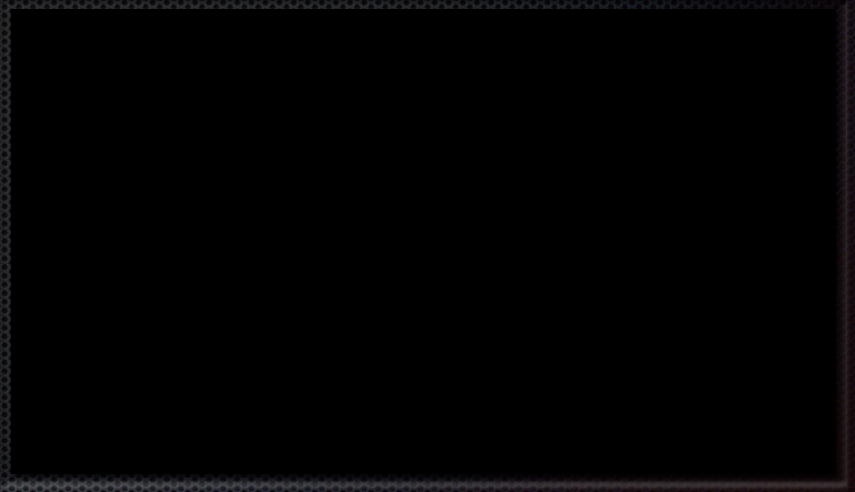
Some Plastruct products and commercial kits are ABS plastic

Choice of Thermoplastic

- Considerations
  - Fumes
  - Strength
  - Ease/Difficulty gluing
  - Ease/Difficulty printing (temperature, environment, specialized equipment)

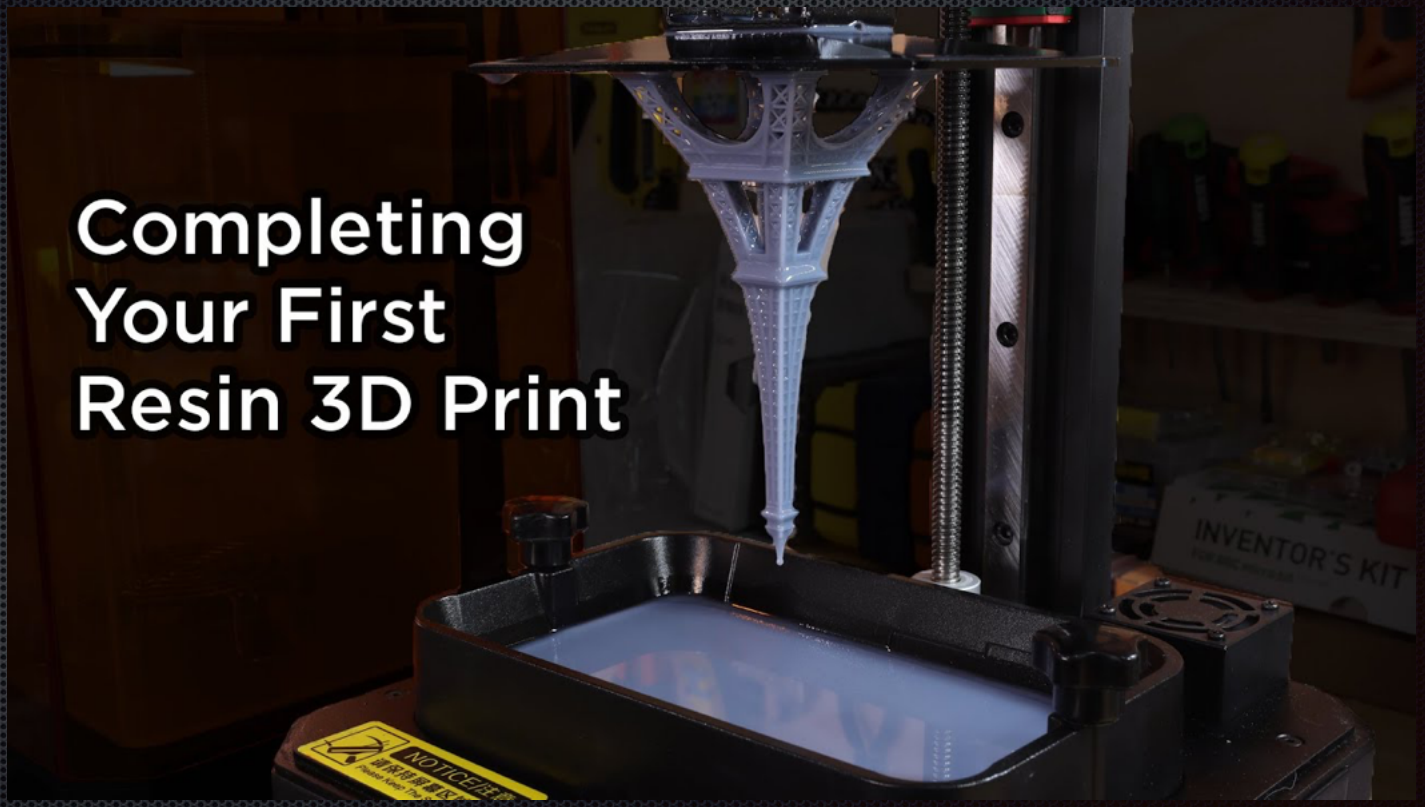
Fused Deposition Modeling (FDM)

- Polylactide (PLA) plastic
  - Economically produced from renewable resources (Sugar Cane Husks)
  - Biodegradable
  - No fumes



Resin Printing

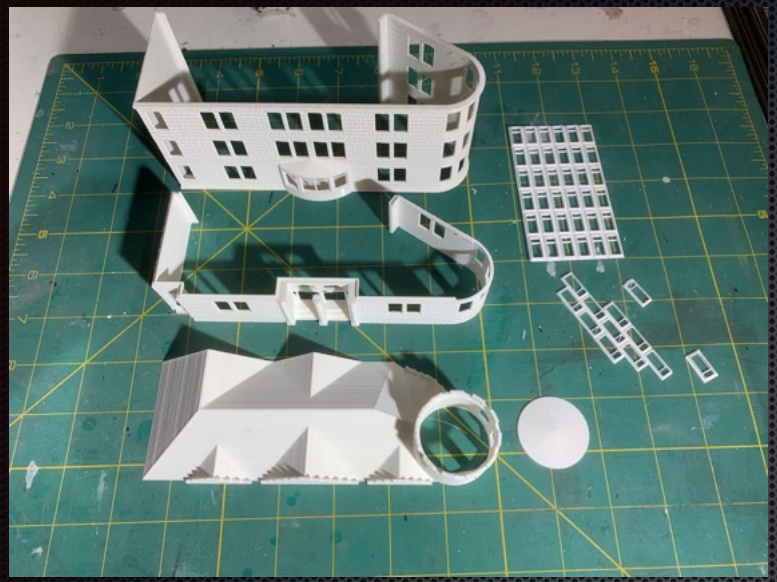
- Layers of ultraviolet light sensitive resin are exposed by light passing through a Liquid Crystal Display (LCD)
- Similar to light coming through a laptop display
- Can be high resolution

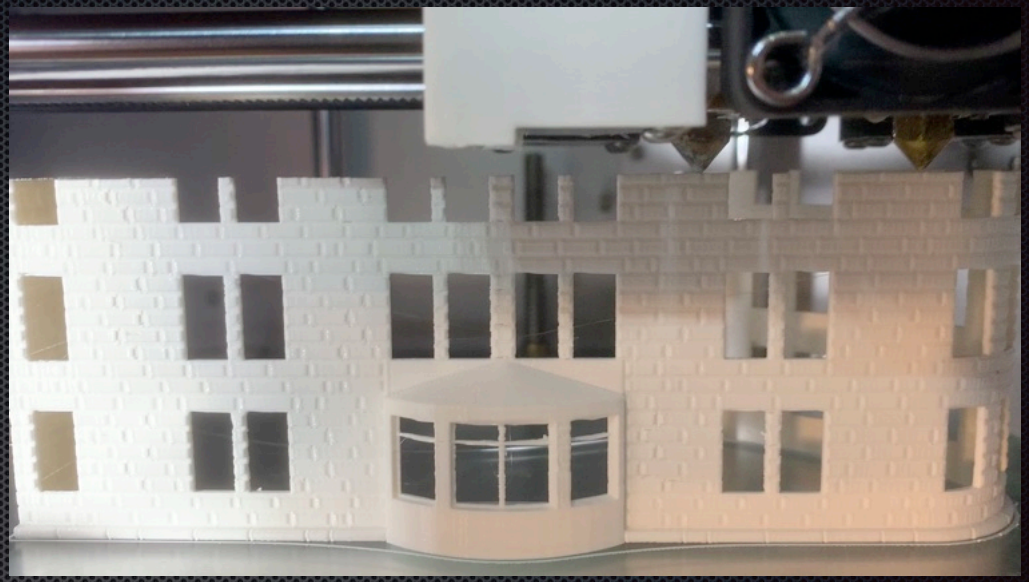


Resin Drawbacks

- Liquid causes skin irritation
- Liquid causes serious eye irritation
- Liquid is toxic to aquatic life with long-lasting effects.
- Fumes irritate eye, nose, throat
- Fumes cause headaches, dizziness etc.
- Long term exposure to fumes damages internal organs - cancer

UV Curing Chamber





All of My Experience Is With Inexpensive FDM

First “BIBO” and now Bambu A1

What Can I Print?

- Layer heights 0.05mm = 0.3 N Scale inches
- Width 0.4mm with current nozzle = 2.52 N Scale inches (height of a brick)

N Scale Brick Magnified 20x

N Scale Brick Magnified 2x

What Can I Print?

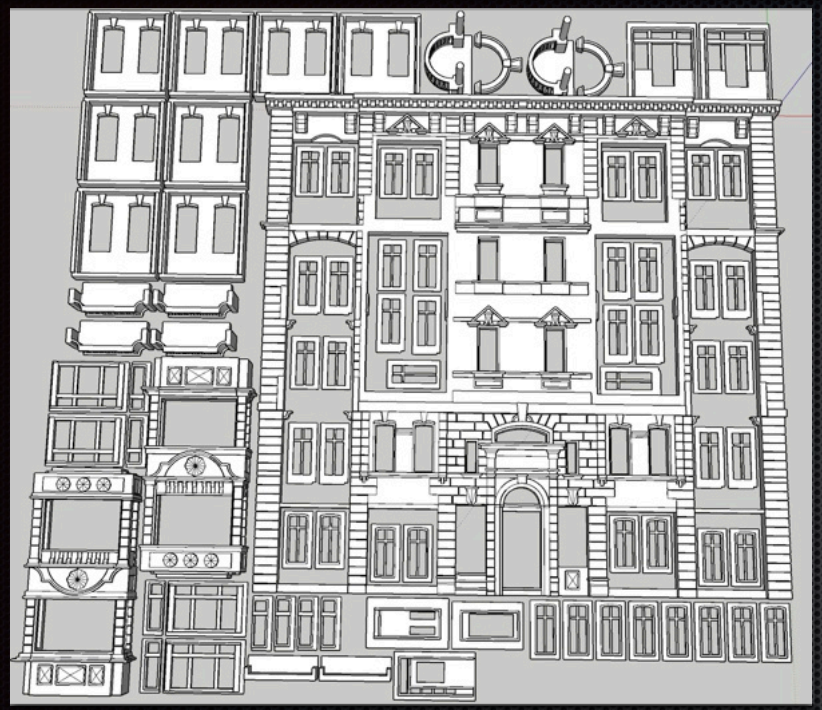
- Layer heights 0.05mm = 0.17 HO inches (1 HO inch is 0.29mm)
- Width 0.4mm with current nozzle = 1.37 HO inches

In Practice, I use 0.2mm Layer Height

From Idea to Finished Product Steps

- Design
- Export as STL format
- “Slice” into layers using Cura, PrusaSlicer, or similar
- Transfer GCode to 3D printer
- Print (Keep an eye on it for failures)
- Post-process (Remove supports, tags, webbing, stringing, possibly sand it)
- Paint and assemble or visa versa

These steps are the same for FDM and Resin



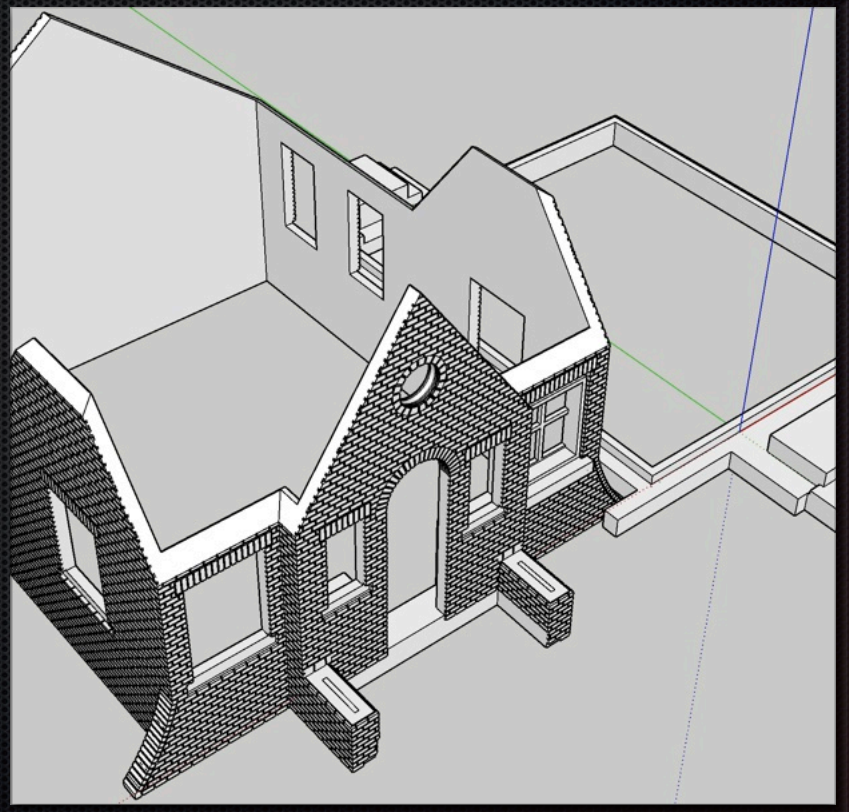
Designing 3D Models

Hard Learned Lessons: plan for mishaps and ease of assembly

- All of my structures were designed using the Free SketchUp Make 2017
  - Easy for beginners but best used for architectural models (not human figures, trees, …)
  - Free SketchUp Make may no longer be available
  - I don’t recommend current SketchUp Free (Runs in browser, limited features, poor export for printing)
  - Gifted SketchUp Version 24.0.598 for Christmas 2024
- Other popular modeling programs:
  - Blender (Free - World’s most difficult software to learn but powerful)
  - SolidWorks (Popular with Engineers)
  - Autodesk Fusion 360 (Free 1 year subscription for hobbyist)
  - FreeCAD (Free - I have never used it)
  - TinkerCAD (Free - Claims to be easy to learn - I have never used it)

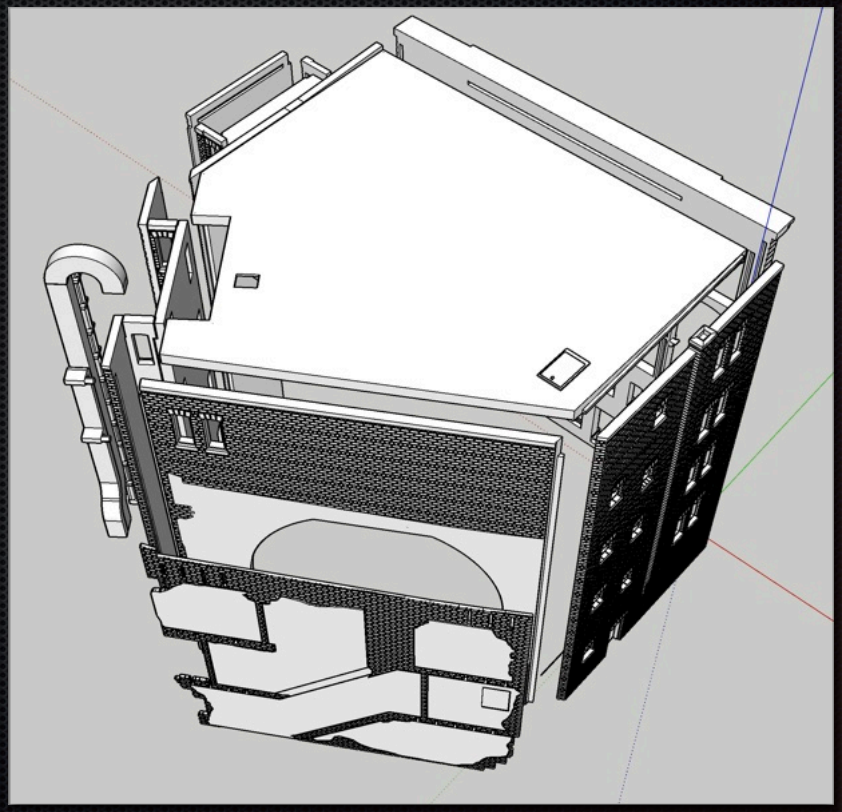
May Print as One or a Few Pieces

- Pro: Simple to assemble
- Con: If anything goes wrong during print, the whole thing may be scrapped



Usually Better to Print Separate Parts

- Pro: One failed part does not impact others
- Pro: Design next parts while other parts print
- Con: Must plan for assembly
  - Slots and tabs for alignment and surface area for glue
  - Tolerances for joining parts
  - Can be difficult to assemble



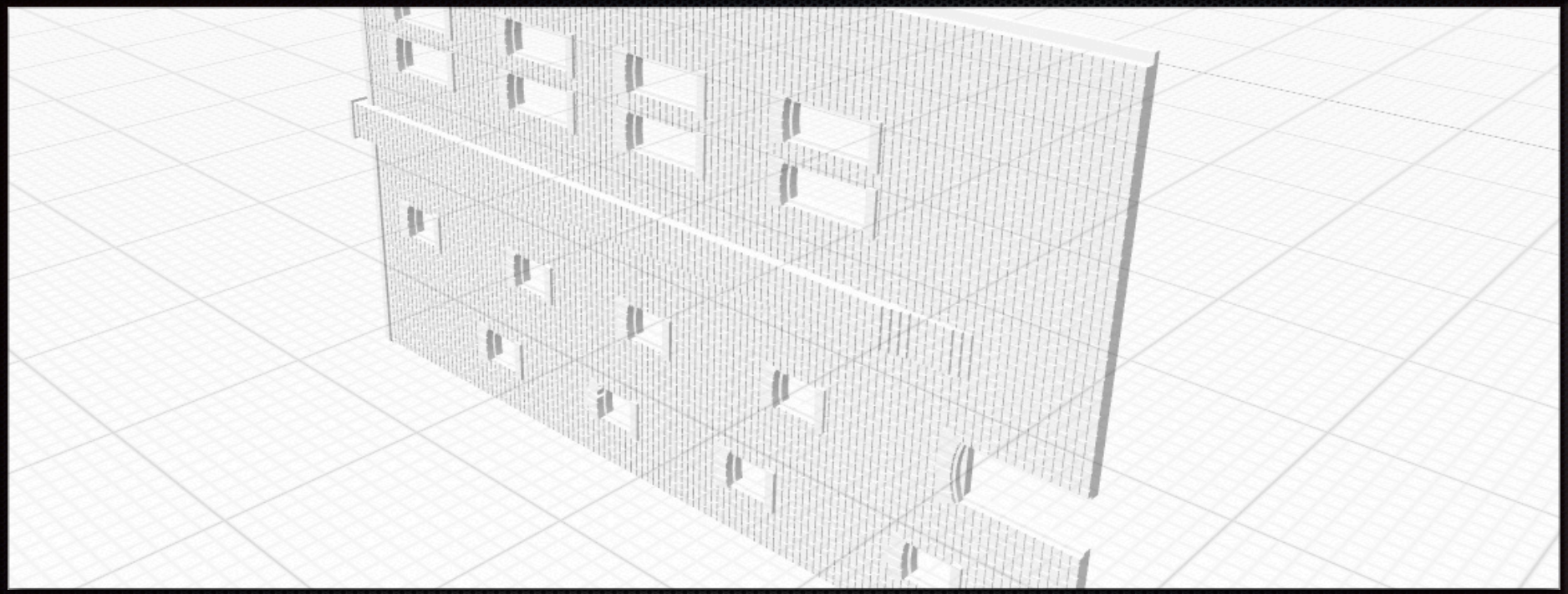
Critically Important: Design for Printing

- No features smaller than printer can print
- Avoid “lofting” (where parts are printed above air)
- Leave space to account for printing tolerances (or parts won’t fit together)
- Dimensions should usually be integer multiples of print resolution
- Resizing models almost always ruins printability by violating above guidance
  - Shrinking or enlarging doesn’t preserve space for printing tolerances, distorts integer multiples, may make printable lofts unprintable, may make features too small to print

Critically Important: Design for Printing

- Parts must almost always be “manifold”
  - Mainfold: Imagine filling the model with water: could the water get everywhere inside the model with no leaks?

In my experience, SketchUp Make usually produces non-manifold models. This means revising/cleanup necessary prior to printing often takes longer than original modeling.

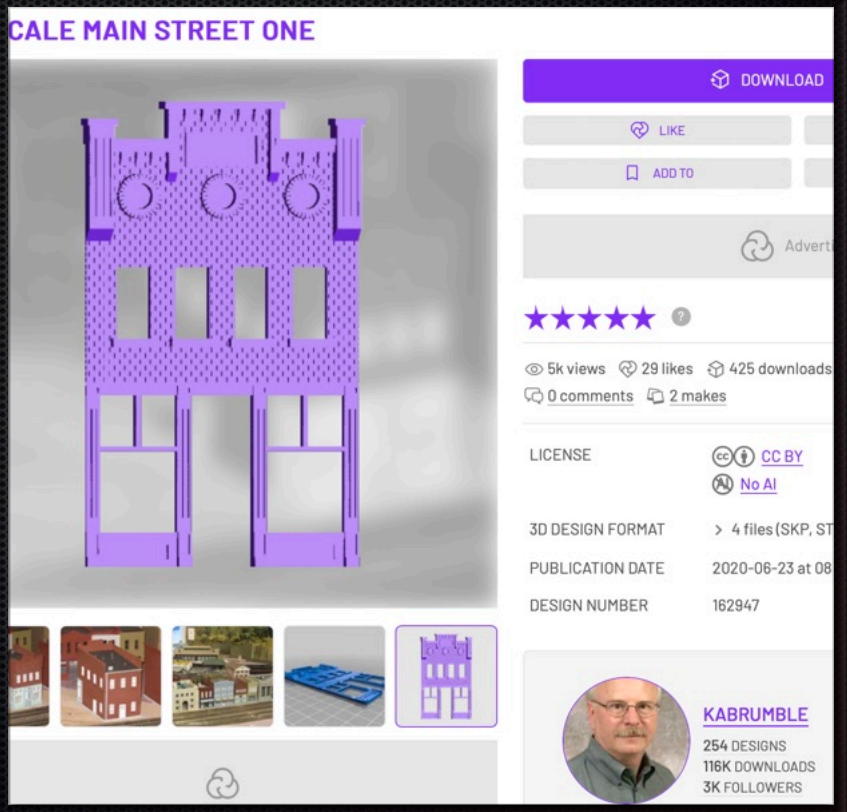


Export For Printing

The standard format is STL (standard tessellated)

Be Wary of STLs Available Online

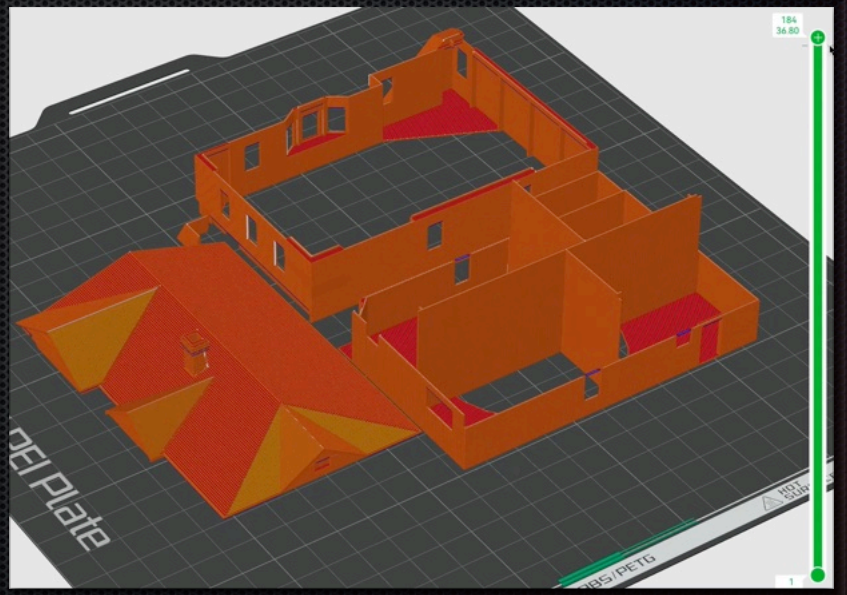
- They are unlikely to match your printer's capabilities
- They may have never been printed before on anyone's printer
- They are likely not to scale, and resizing is problematic
- STL format models are difficult to edit/customize for your use case



https://cults3d.com/en/3d-model/architecture/ho-scale-main-street-one

Slice STL Into Layer By Layer Instructions

- Both FDM and Resin printers build the model layer by layer
- The 3D Model must be “sliced” into individual layers for printing.
- FDM printers use a simple programming language called GCode that describes the path the print head must follow on each layer.



Slicer Settings

- When I started with original Bibo printer, I needed to fuss with slicer settings for hours or days to get a good print
  - Change filament brand or color and new settings were needed
  - Change room temperature, humidity and new settings were needed
  - Age of filament affected settings
  - Speed of print, layer height, amount and placement of supports, “skirts”, etc. all needed to change frequently

Slicer Settings

- Things are much better now.
  - So far, the Bambu Labs A1 I am using has not required any slicer tinkering beyond changing nozzle width from 0.45mm to 0.39mm and sometimes changing layer height to improve print quality.

Transfer GCode To Printer

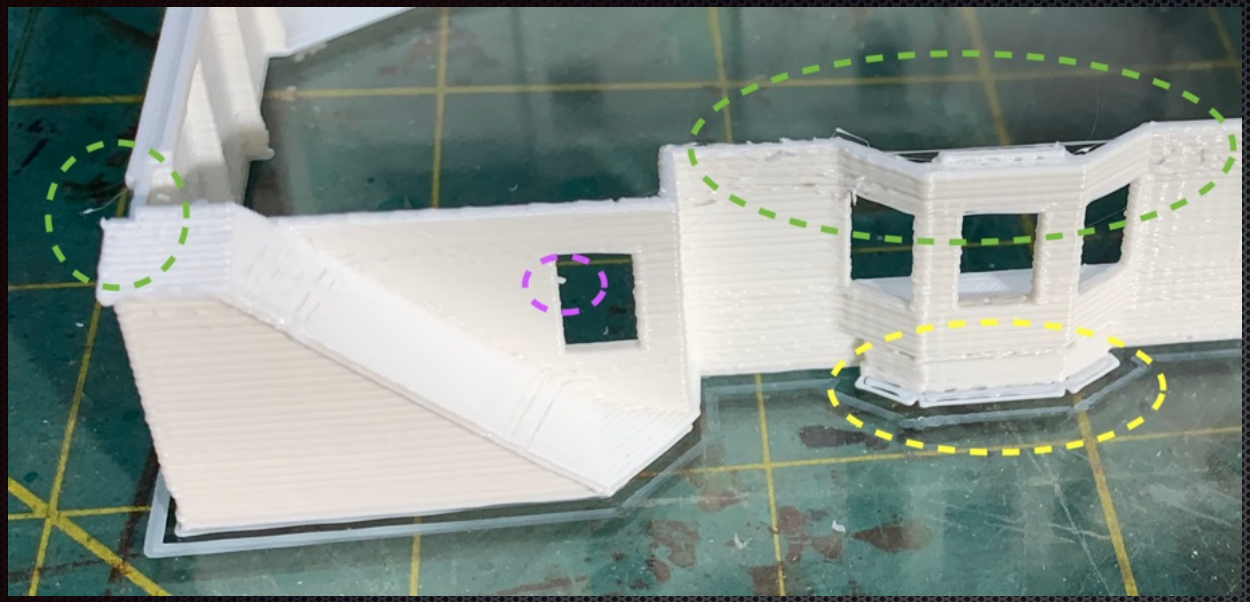
- USB Thumb Drive, SD-Card, Home WiFi, and Over Internet have all worked for me
- Be wary of printers the REQUIRE an Internet connection. It should not be needed but may be handy.

Actually Printing

- Some prints take many hours to print. My longest has been 12+ hours
- You need to monitor long prints to stop things as soon as a problem arises
  - Avoid wasting further filament
  - Avoid clogs and messes that may damage printer
- I recommend using a web cam. Some printers come with one. You can monitor and cancel the print via your phone from a restaurant …

Post-processing

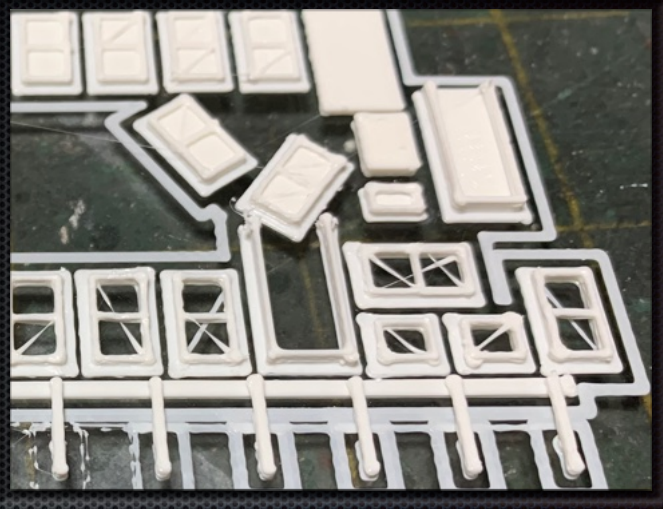
Remove supports - Clean any stringing - Sand the model - Glue & assemble if needed - Priming & painting



Examples of supports, stringing, and tags

A sharp hobby knife takes care of most defects.

Trick: Nail polish fills grooves, hides imperfections, and acts as weak solvent for PLA plastic.



Examples webbing

Painting

- I prefer to paint before assembly
- Rust-oleum products have worked well fro me with PLA. Krylon paints tend to flake off PLA.
- I prefer to print parts that will be painted different colors separately. Window casing separate from brick walls, etc.
- Gluing can be a problem:
  - Plastruct Plastic Weld works OK for PLA. It has several solvents in it, and only some work for PLA
  - Superglue (Cyanoacrylate) always works but is difficult to apply, difficult to clean excess, and unforgiving