COLGATE UNIVERSITY

Introduction To

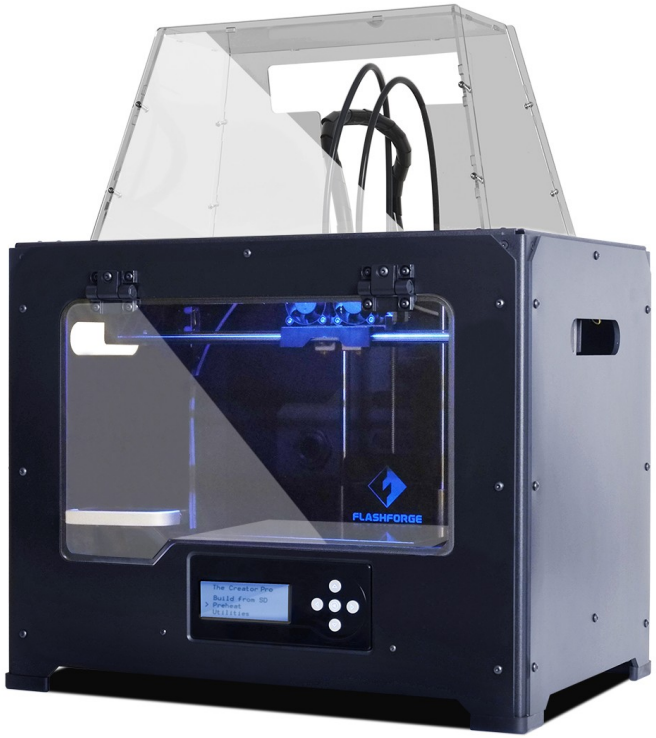
3D Printing

What is 3D Printing?

- An additive process where materials are joined, layered, or solidified with the assistance of computer hardware and software to create a three-dimensional object
- The first additive manufacturing equipment and materials were developed in the early 1980s
- Today there are a number of processes and materials to create 3D prints including:
  - Powder
  - Plastics
  - Resins
  - Metals

Types of Printers at Colgate

- Fused Filament Fabrication
  - Plastic filament is melted and built up layer by layer to create the object
- Stereolithography
  - Light (lasers) causes chemical monomers (resin) to link together to form polymers (plastic)



Flashforge Creator Pro

Formlabs Form 2

Wanhao Duplicator 5S

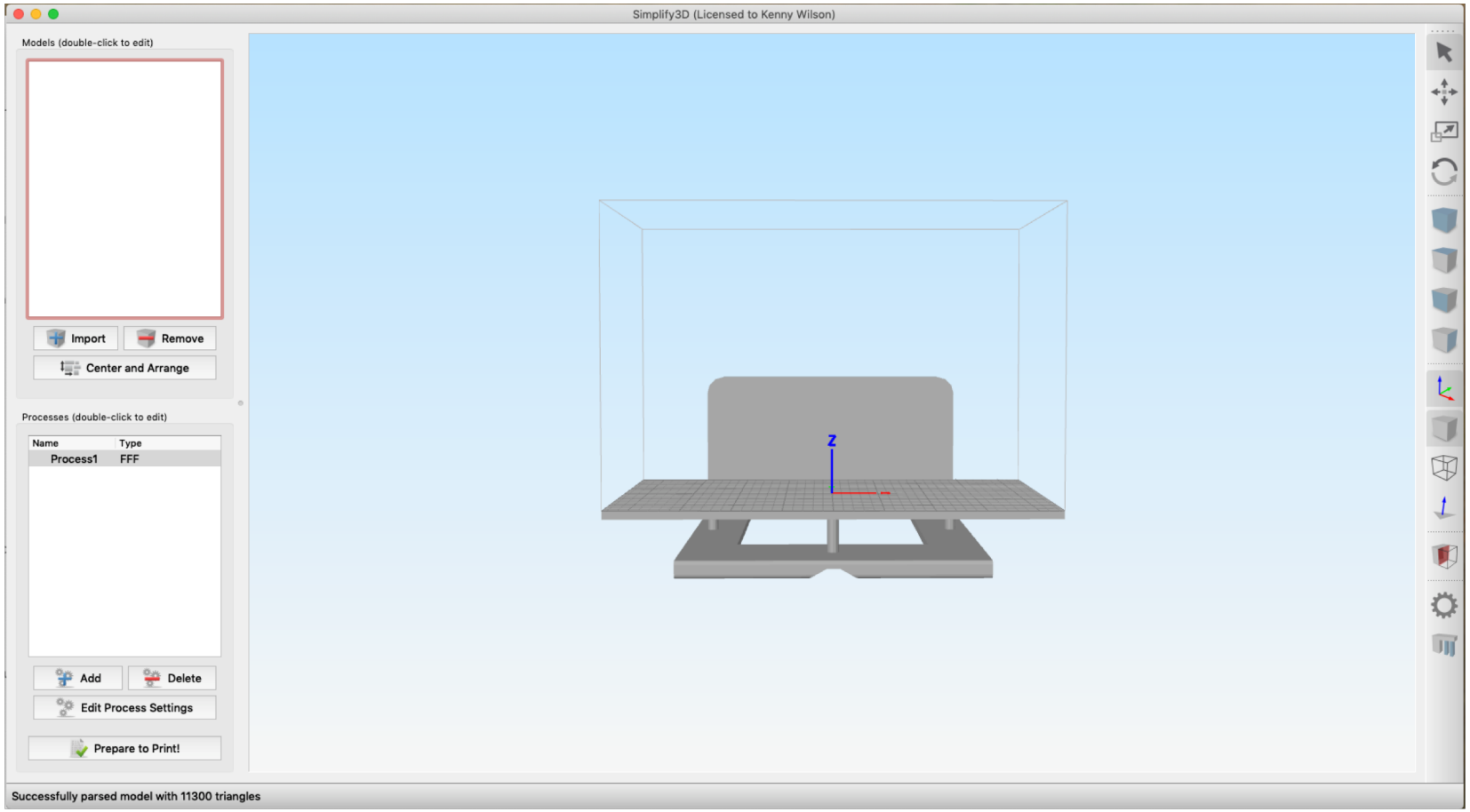
How Do You Create A 3D Print?

- Design A File
  - Tinkercad
  - Maya
  - AutoCAD
- Find A File
  - Thingiverse
    - www.thingiverse.com
  - Cults
    - https://cults3d.com/en
- Slice the model
  - Simplify3D
- Print

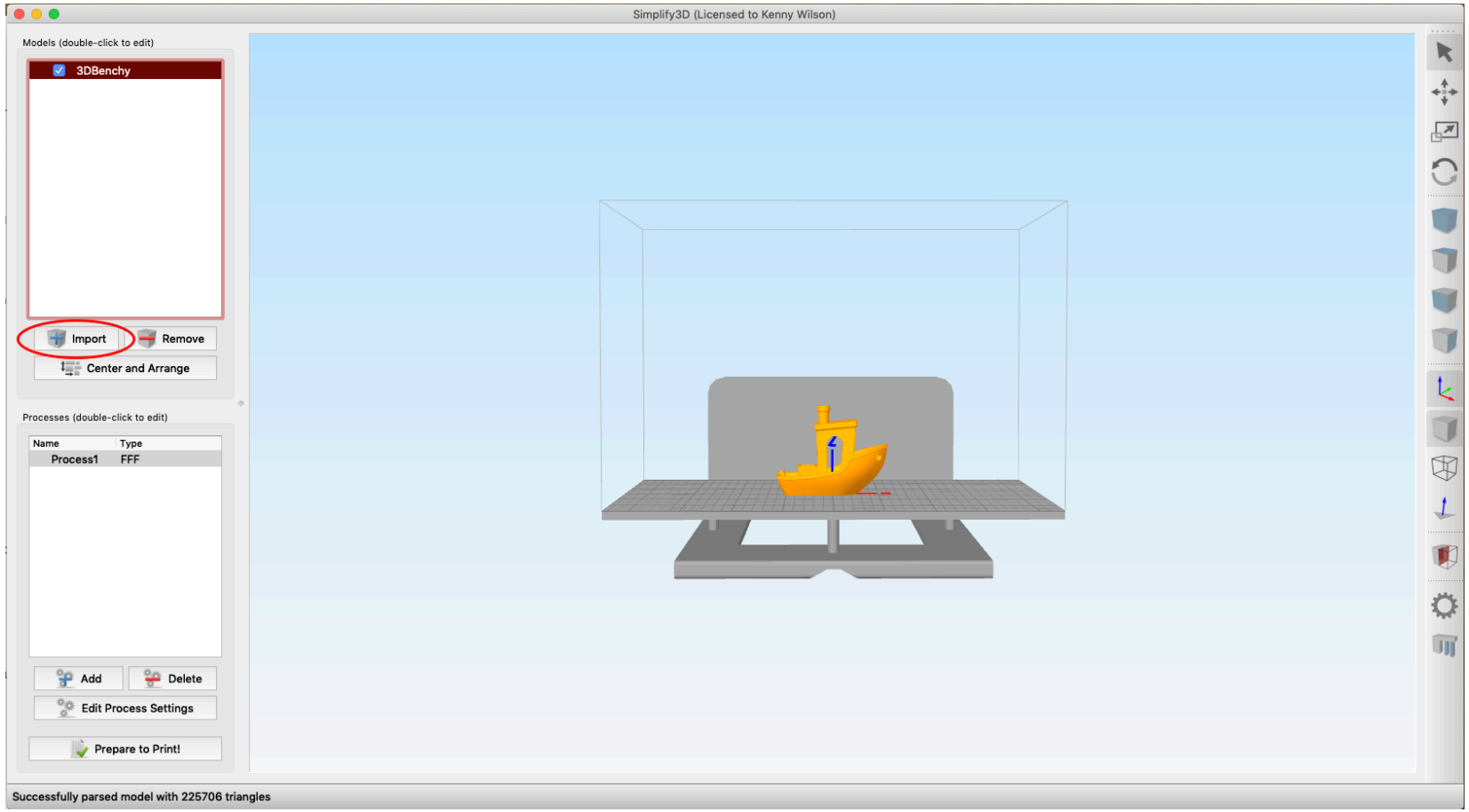
- CAD = Computer Aided Design
- CAD software takes your design and turns it into a compatible file, most commonly a STL file

- Slicing software takes your CAD file and turns it into a set of instructions that the printer uses to construct your object
- gcode

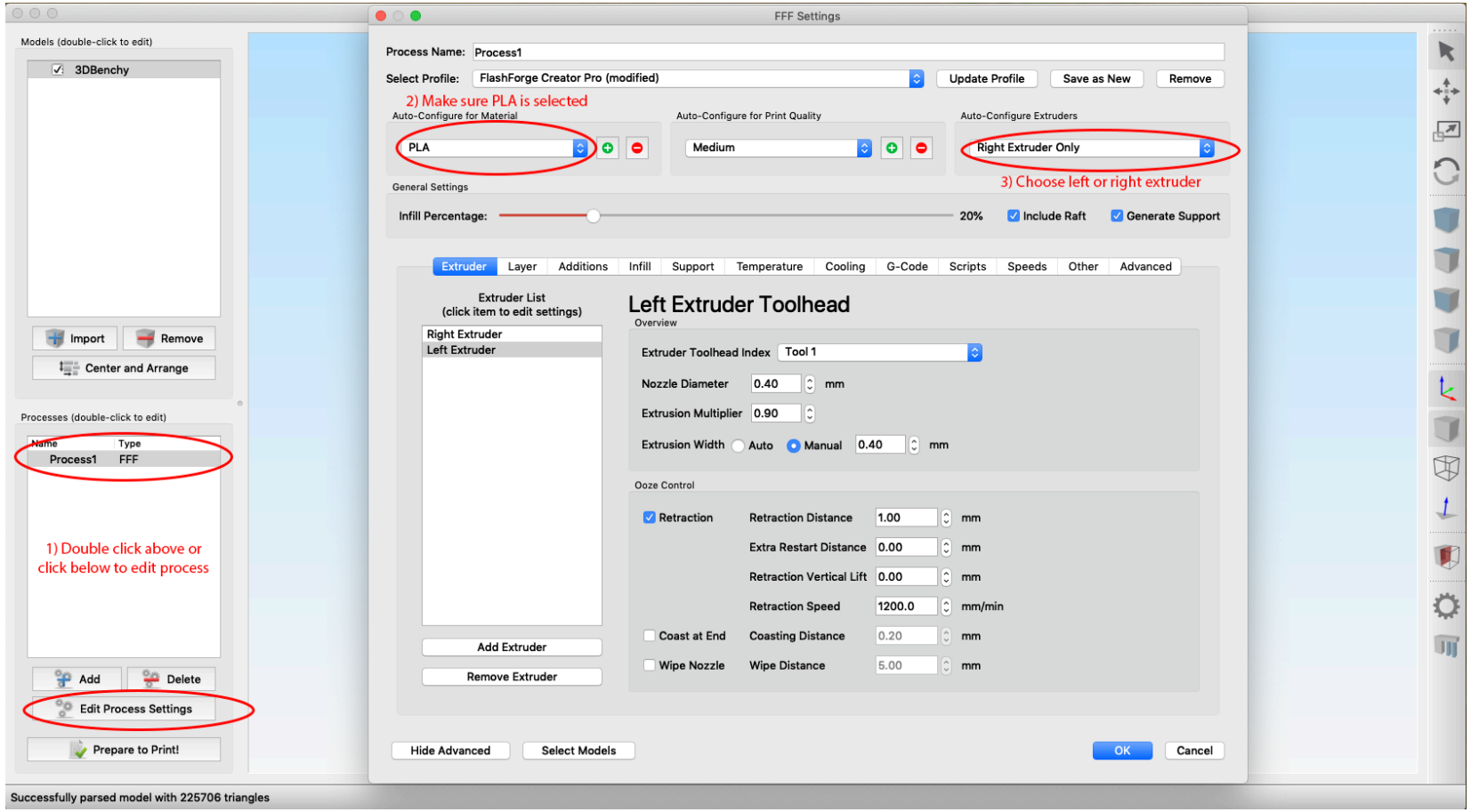
Opening Simplify3D



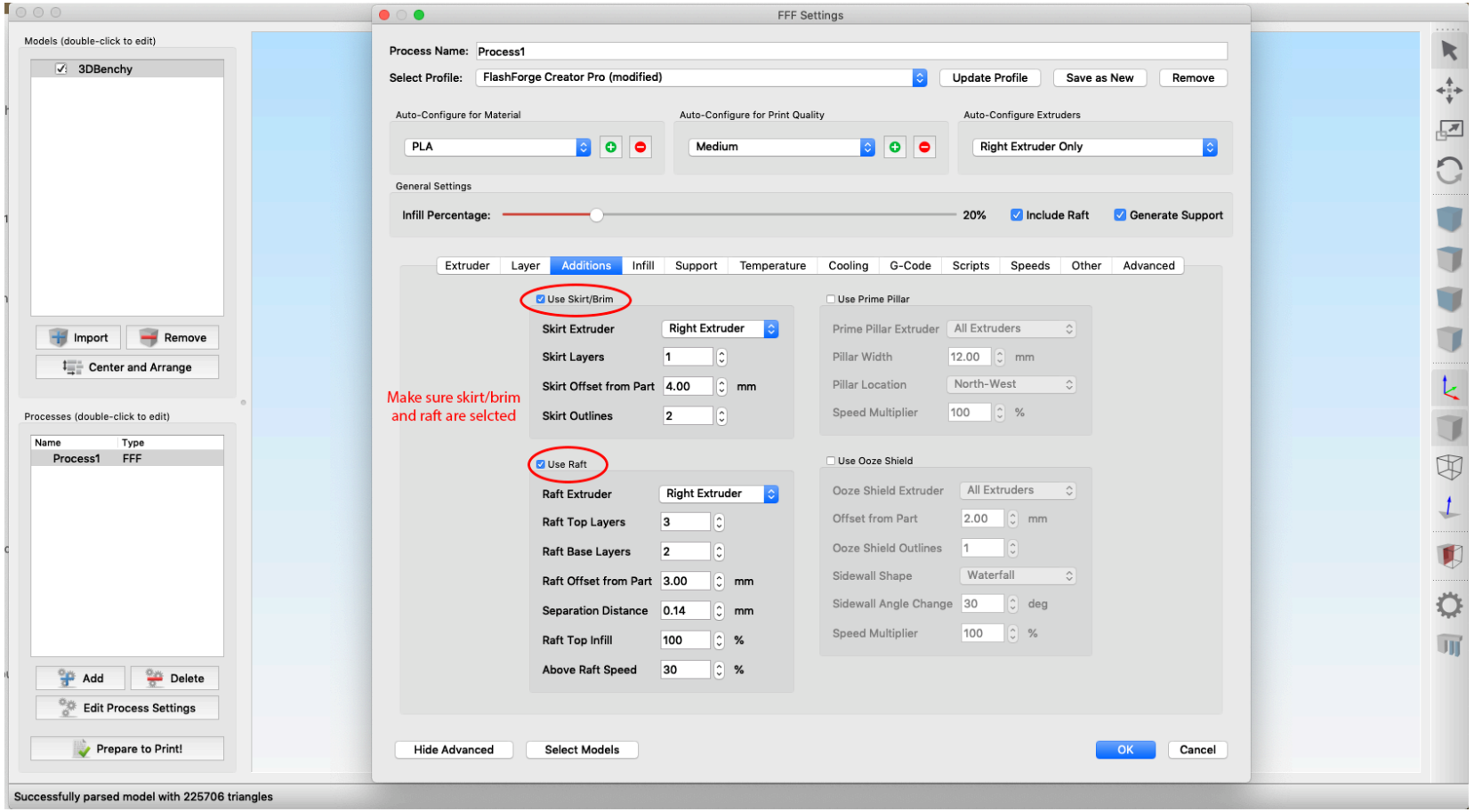
Import A STL File



Edit Process

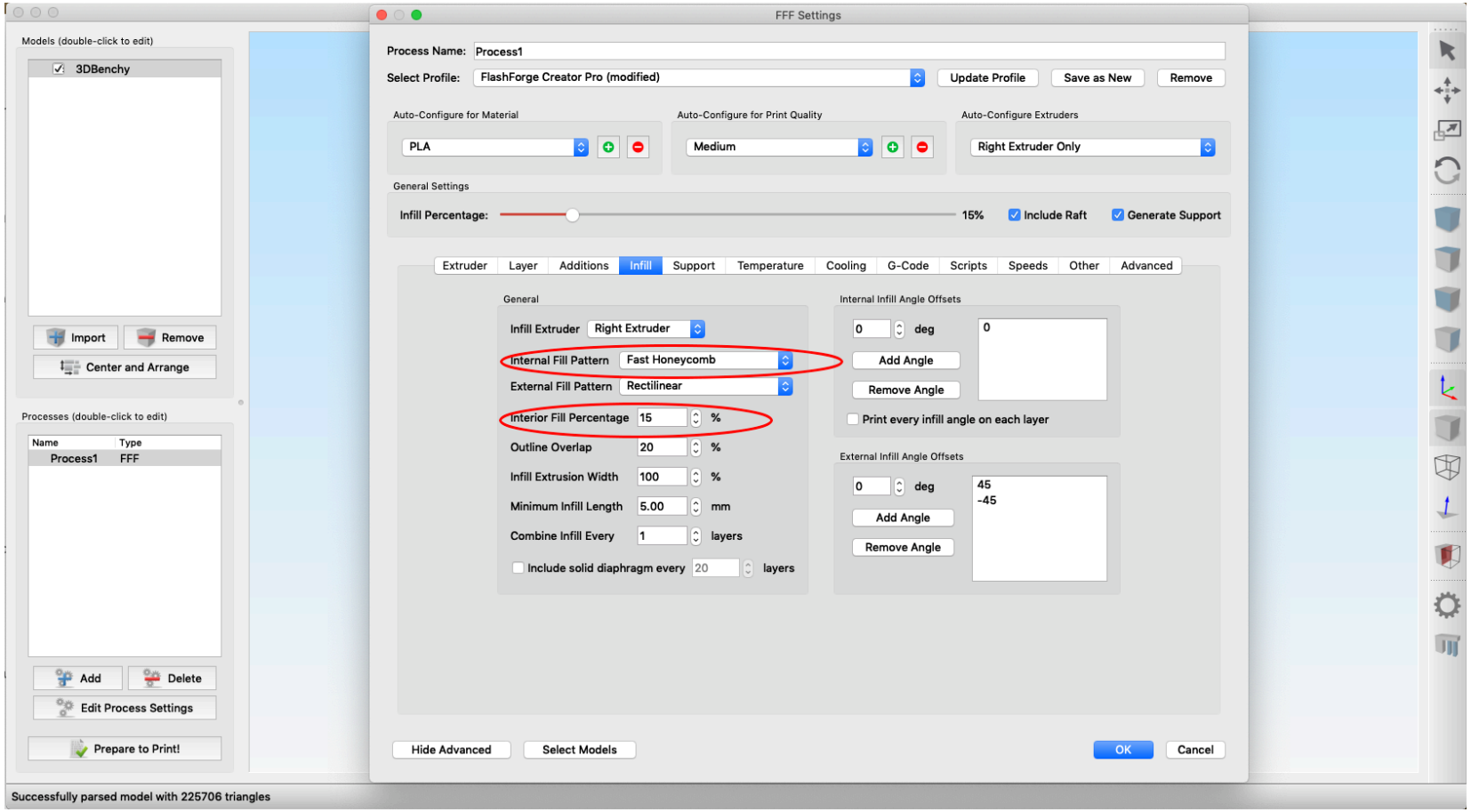


Select Additions

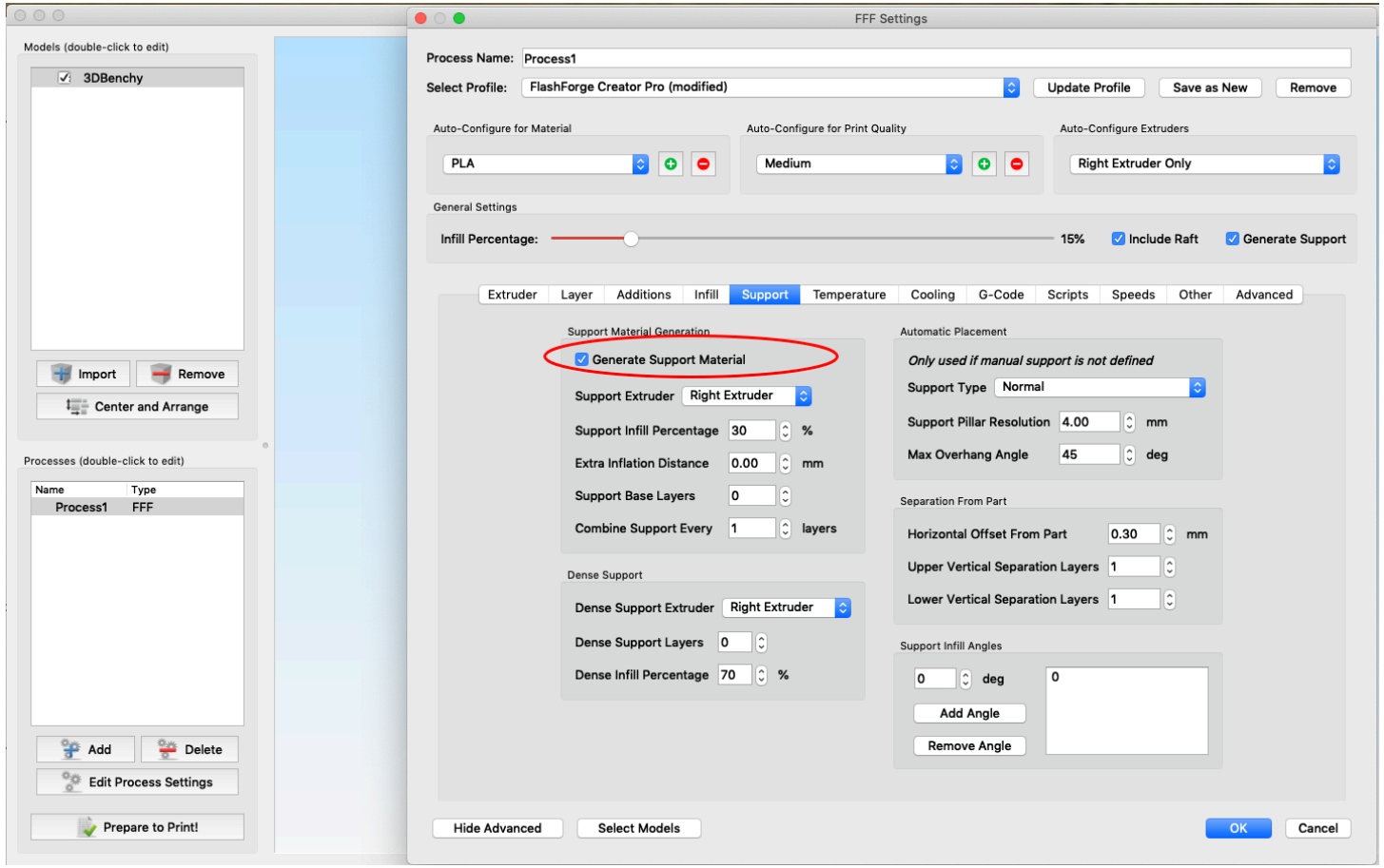


Infill

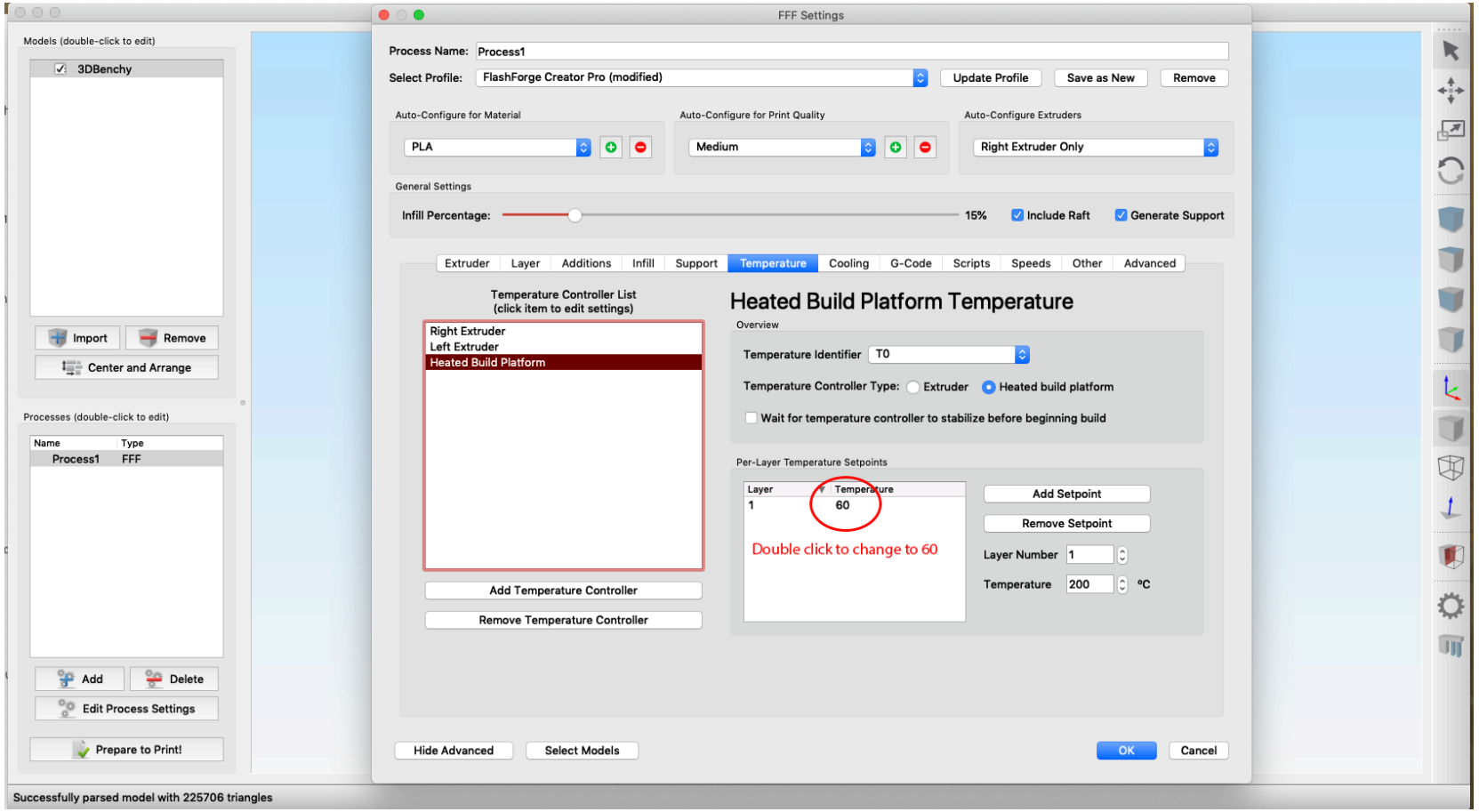
- 15 to 20 percent infill is usually adequate
- Fast Honeycomb improves print time



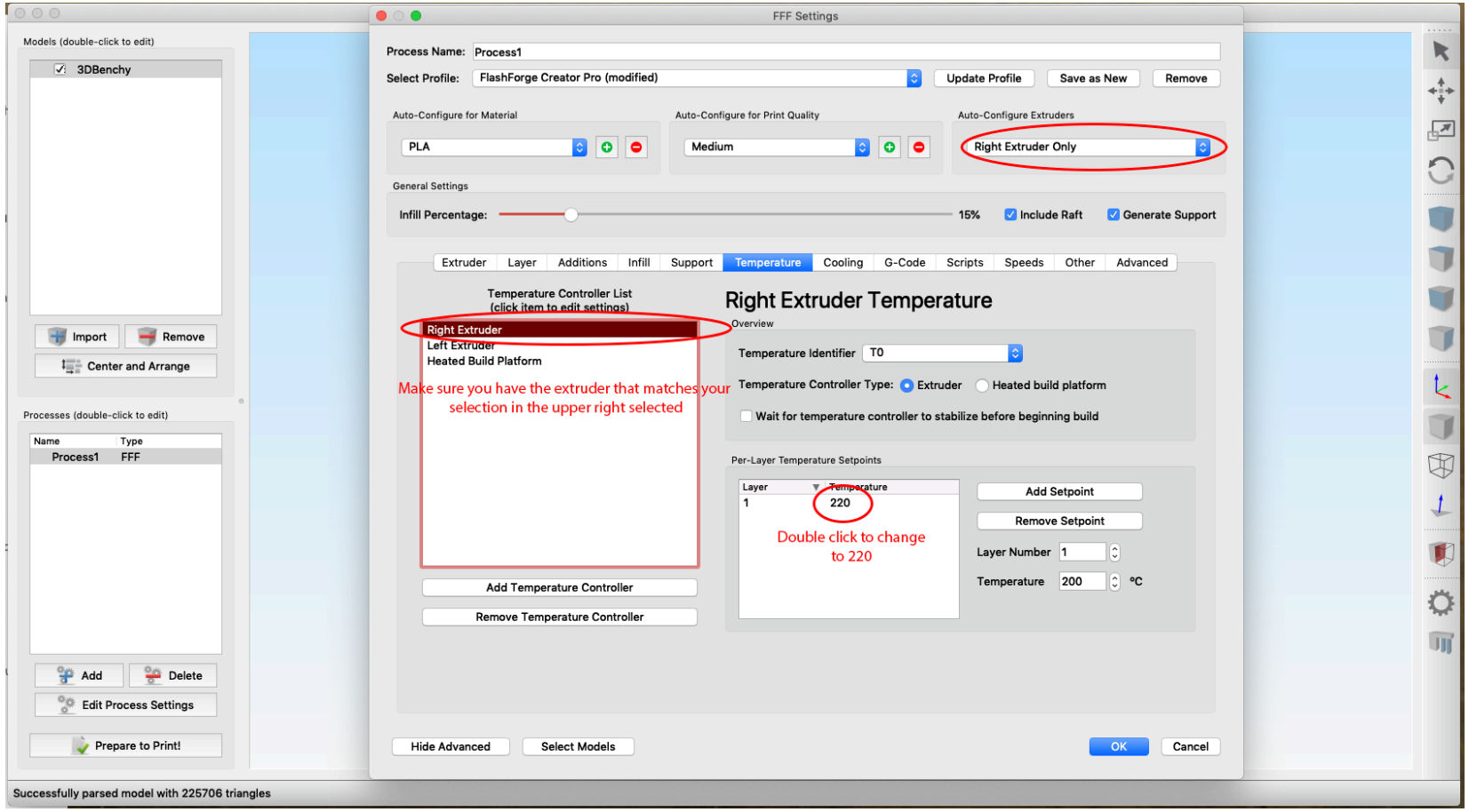
Support


Models (double-click to edit)
3DBenchy
Import
Remove
Center and Arrange
Processes (double-click to edit)
Name
Type
Process1
FFF
Add
Delete
Edit Process Settings
Prepare to Print!
FFF Settings
Process Name:
Process1
Select Profile:
FlashForge Creator Pro (modified)
Update Profile
Save as New
Remove
Auto-Configure for Material
PLA
Auto-Configure for Print Quality
Medium
Auto-Configure Extruders
Right Extruder Only
General Settings
Infill Percentage:
15%
Include Raft
Generate Support
Extruder
Layer
Additions
Infill
Support
Temperature
Cooling
G-Code
Scripts
Speeds
Other
Advanced
Support Material Generation
Generate Support Material
Support Extruder
Right Extruder
Support Infill Percentage
30
%
Extra Inflation Distance
0.00
mm
Support Base Layers
0
Combine Support Every
1
layers
Dense Support
Dense Support Extruder
Right Extruder
Dense Support Layers
0
Dense Infill Percentage
70
%
Automatic Placement
Only used if manual support is not defined
Support Type
Normal
Support Pillar Resolution
4.00
mm
Max Overhang Angle
45
deg
Separation From Part
Horizontal Offset From Part
0.30
mm
Upper Vertical Separation Layers
1
Lower Vertical Separation Layers
1
Support Infill Angles
0
deg
Add Angle
Remove Angle
0
Hide Advanced
Select Models
OK
Cancel

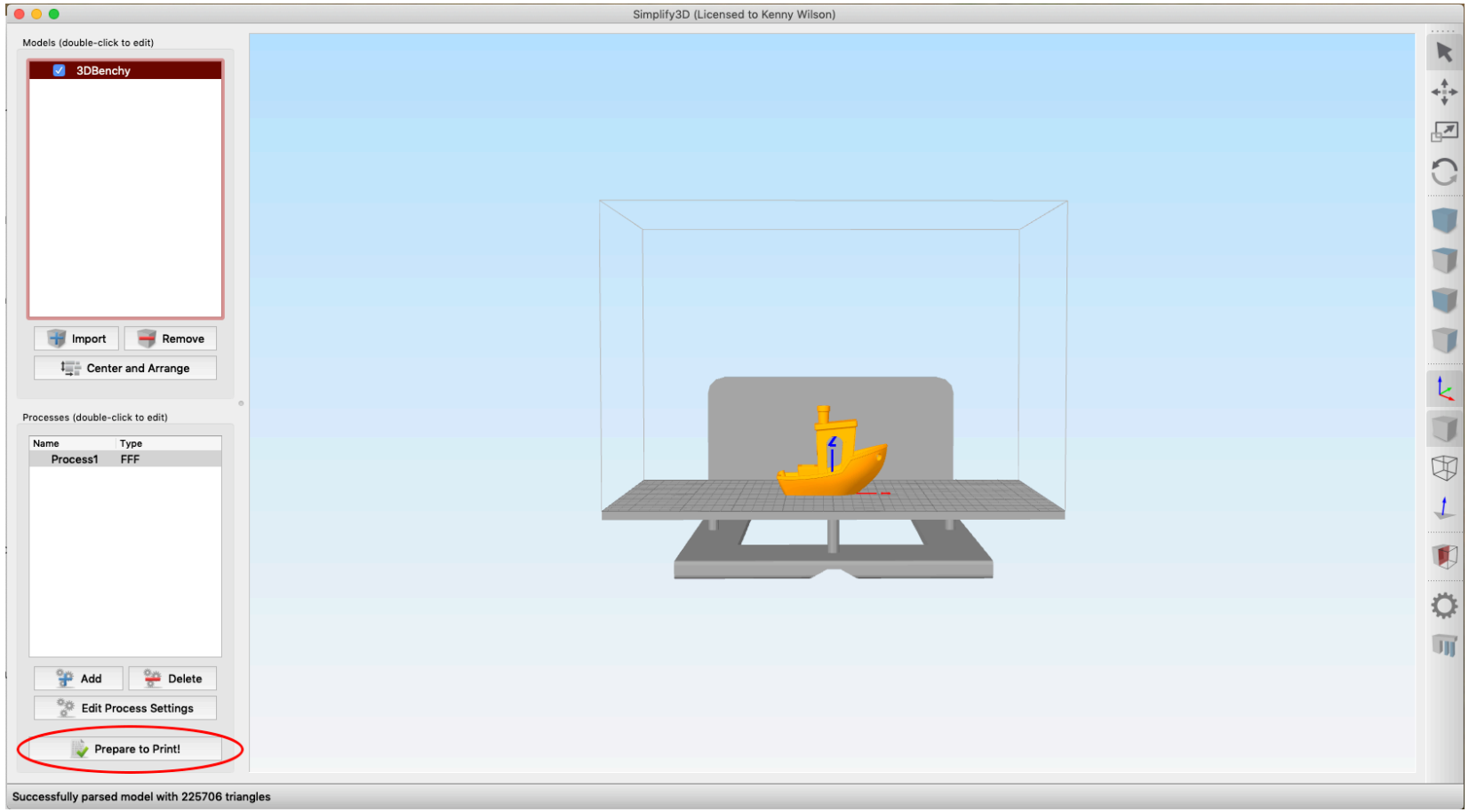
Bed Temperature



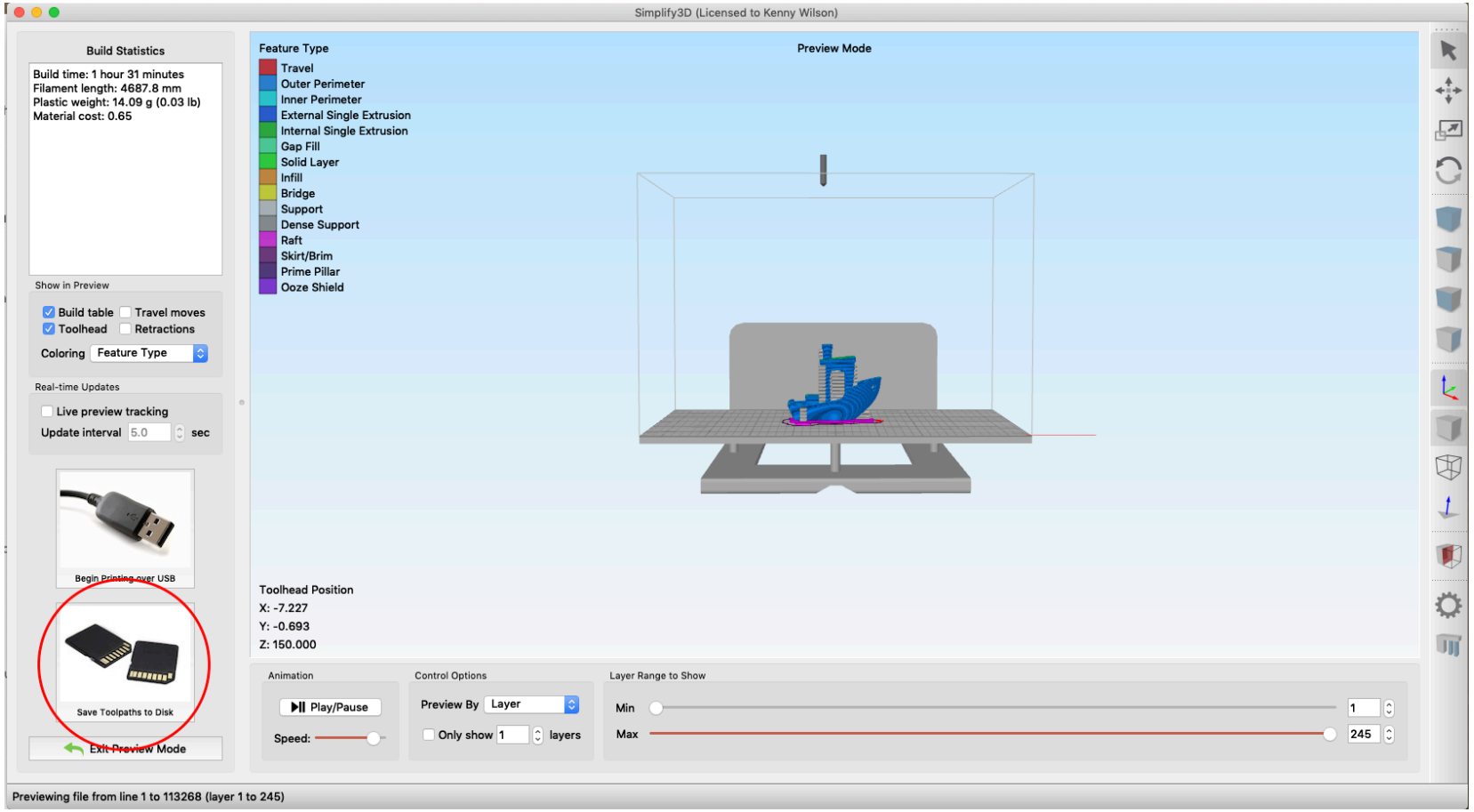
Extruder Temperature



Prepare To Print



Save To SD Card



Let’s Make A Name Tag

- On your computer, open the MediaSAN folder on the desktop
  - Locate the student work folder
  - Locate ‘Name Plate’ folder
  - Locate ‘Media Mentor Name Plate.stl’
  - Drag it to your desktop
- Create a Tinkercad account
  - Import the name plate file
  - Add a text object and edit it
  - Export a STL file
- Save your file to the desktop
  - Move it to the MediaSAN server
  - YourName.stl
  - MediaSAN -> Student Work -> Name Plate

Thank You

- Feel free to contact me at kjwilson@colgate.edu