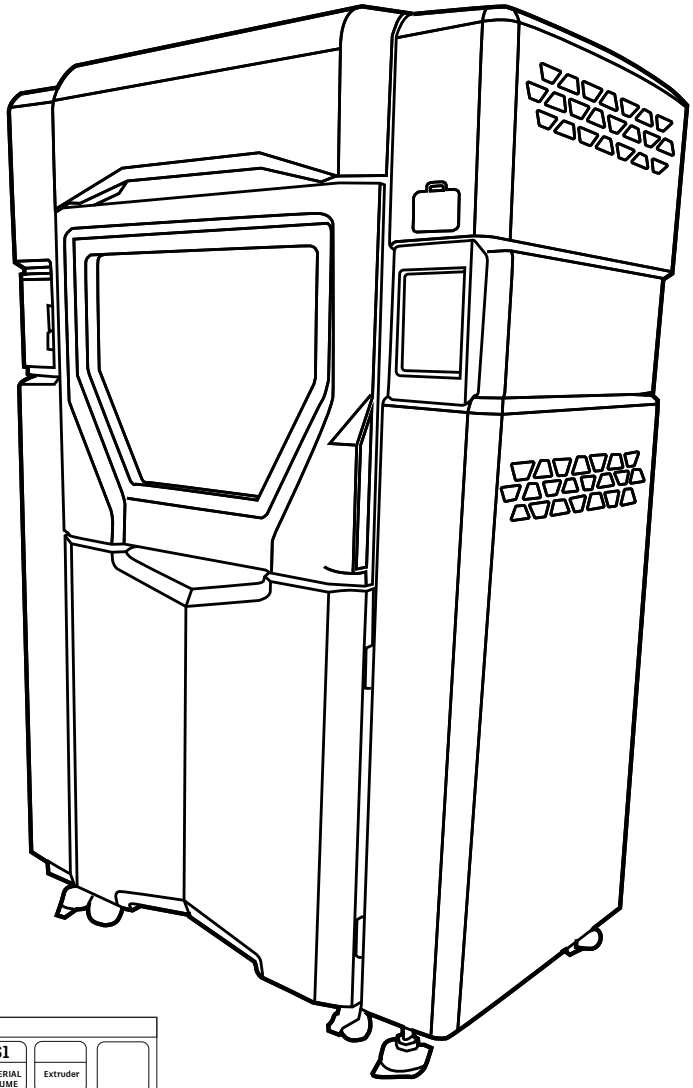
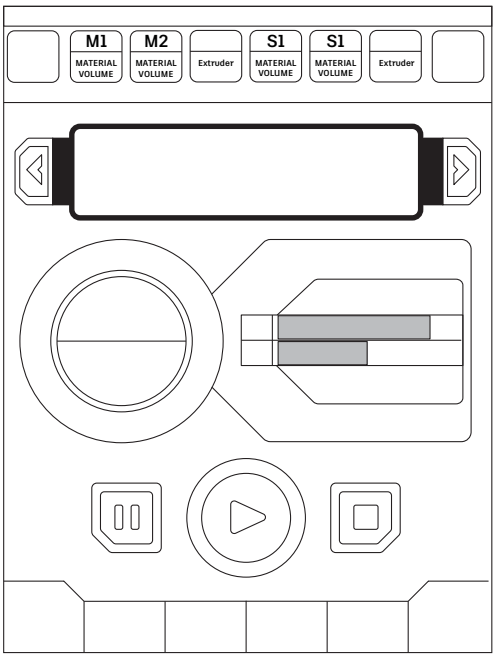
PRINTSHOP FORTUS 450

VERSION 3.1

AUTODESK
PIER 9



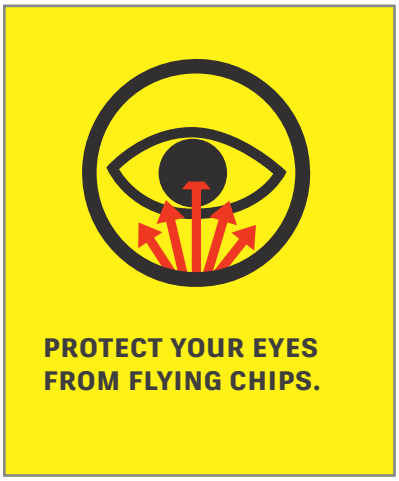


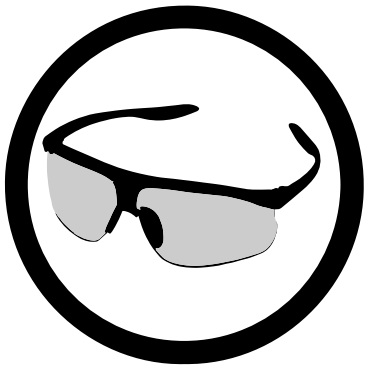
CONTROL PANEL

MATERIALS

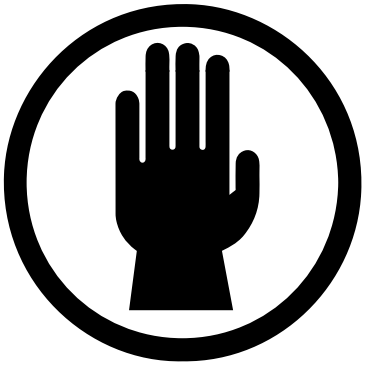
ALLOWED MATERIALS

- Only materials approved and loaded by shop staff.
- Available materials include:
  - ABS
  - ASA
  - PLA
  - Nylon
  - ULTEM
  - Polycarbonate

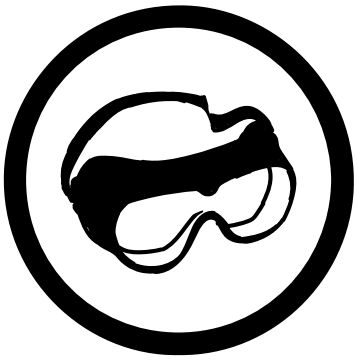




Always wear safety glasses when cleaning parts.



Wear heat-resistant gloves and non-synthetic clothes when working with the oven or hot parts.



Wear nitrile gloves, chemical goggles, face shield, long pants, a liquid-resistant lab coat and closed toe shoes when working in the bath and rinsing parts.

The Fortus 450mc uses an additive manufacturing technology called *Fused Deposition Modeling* (FDM). FDM Printers work by heating and extruding a filament of plastic along a path to form a single layer. This process is repeated layer upon layer until the final model is created.

+ Parts with an overhang will require *support material.*
  - ► It will need to either be removed manually or dissolved in a chemical bath.
+ The printer uses a 3D mesh file type called an .STL.
  - ► This can be created in programs like Fusion 360.
+ To print on the Fortus, you must use *Insight* and *Control Center*.
  - ► These software packages are installed on all of the computers in the 3D Printshop.
+ Each printer is loaded with a single Model Material and a single Support Material
  - ► If your desired material is not loaded in any of the Fortus machines you can request a material change from Shop Staff.

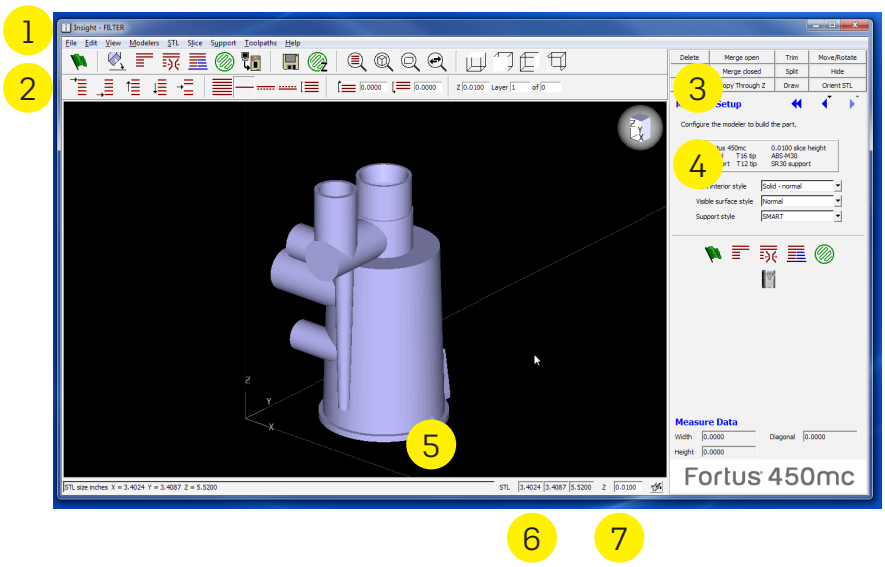
+ Build Envelope : 14" X 16" X 14" (35.5 x 40.6 x 35.5 cm)
+ Layer Heights: 0.005" to 0.020" (0.125 to 0.5 mm)

STEPS FOR SUCCESSFUL USE

To make a part, you need to complete each step in order.

1. Open the STL in Insight
2. Configure the modeler (printer)
3. Select infill and support styles
4. Orient the model
5. Slice the model
6. Create supports
7. Create toolpaths
8. Identify problems
9. Send to Control Center
10. Send to the modeler
11. Post process (clean) the model

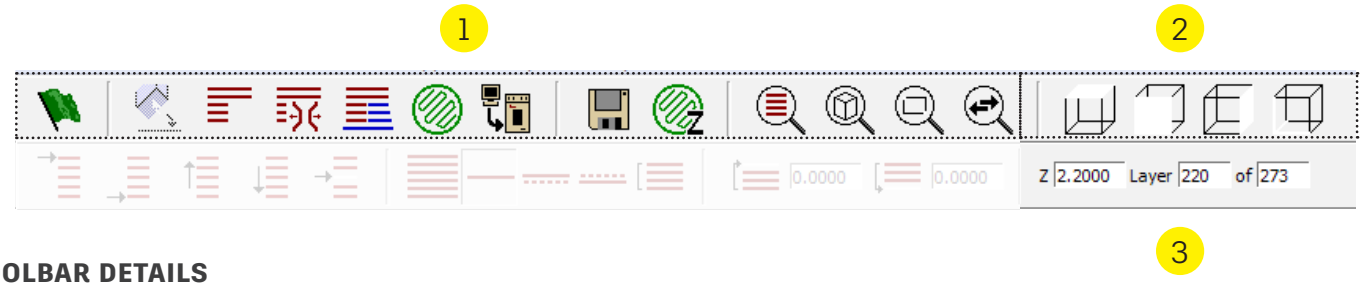
INSIGHT MAIN SCREEN



1. Menus
2. Toolbar
   - Many commands are available in menus and the toolbar.
   - They are arranged in the order of use.
3. Toolbox
4. Current Operation Window
   - Has options related to the current operation.
5. Geometry Window
   - Shows the model.
6. Model Size
7. Layer Height

Navigation tips for the Geometry Window:

+ Rotate
  - Hold the mouse wheel and move the mouse.
+ Pan
  - Hold Ctrl + the mouse wheel and move the mouse.
+ Zoom
  - Rotate the mouse wheel.



TOOLBAR DETAILS

1. Main Toolbar - for general workflow
2. View Toolbar - for changing your viewing angle
3. Z Toolbar - for layer inspection

OPEN THE STL IN INSIGHT

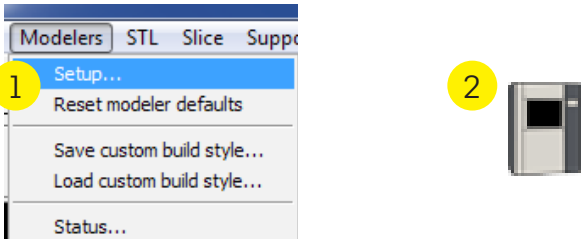
+ Use the file menu.

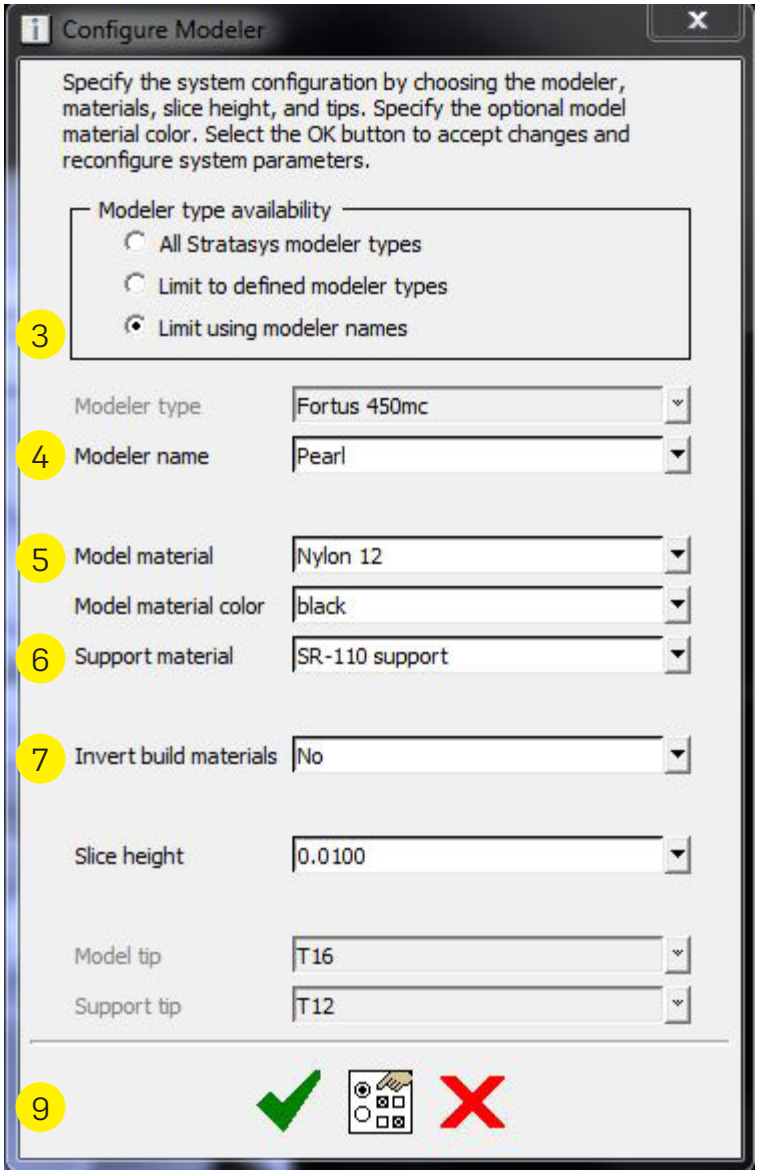
CONFIGURE THE MODELER

1. In the **Modelers** menu, select **Setup...**
2. Click the **Modeler Setup** (printer) button.

Select the modeler

3. Select **Limit using modeler names**.
4. Choose the Modeler name from the menu.
   - ▸ The name of each modeler is on the front of the machine.
5. Choose the Model material that is physically installed in the modeler.
   - ▸ The material is listed on the front of the machine.
6. Choose the Support Material.
7. Ensure that Invert build materials is **No**.
8. Do not change the other options.
9. Click the green check mark.

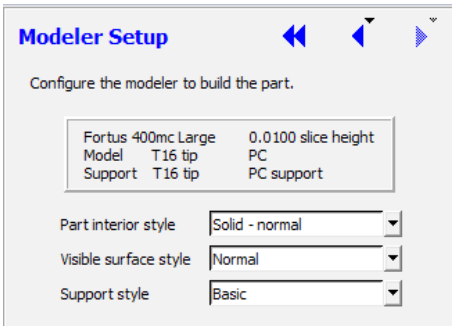




SELECT THE INFILL AND SUPPORT STYLE

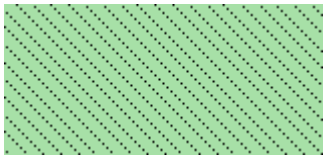
The model will be printed with a solid exterior, but there are options for how the rest of the model is created.

+ In the Modeler Setup dialog, select options for Interior (infill), Surface & Support.



Part interior options

+ **Solid - normal**
  - ▸ This will create a completely solid model.
  - ▸ It is the strongest option, but will be heavier, more expensive and take longer to print.
+ **Sparse**
  - ▸ Alternating layers of diagonal lines will be printed.
  - ▸ This option creates a mesh interior that is lighter, less expensive and faster to print.
+ **Sparse - double dense**
  - ▸ Every layer gets diagonal line in both directions.
  - ▸ This option creates a mesh interior that is stronger than sparse, but still light, inexpensive and fast to print.



Solid - normal

Sparse

Sparse - double dense

Visible surface style

+ In most cases, you should select Enhanced. It creates a slightly nicer print and will take slightly longer to print.

Support style options

+ **SMART**
  - ▸ SMART uses the least amount of support material and prints faster than the other options.
+ **Sparse**
  - ▸ This option is stronger than SMART, but uses more material.
+ **Basic**
  - ▸ Basic uses even more material.
+ **Surround**
  - ▸ Designed for tall thin parts, it completely surrounds the model.
+ **Break-away**
  - ▸ Designed to be easy to remove, but is not as supportive as the other options.

CHOOSE AN ORIENTATION

Changing the orientation of the part (which face sits on the build platform) is influenced by five factors.

+ **Height and size**
  - Z is the slowest axis to print.
  - A tall print will take longer to print than a short print of the same volume.
+ **Support material**
  - Switching build material and support material takes at least 30 seconds per layer. A layer without support will print faster than the same size layer with support.
  - Some support material may be hard to remove, such as inside of pockets.
+ **Part strength**
  - Since the model is printed layer by layer, the joint between layers is the weakest spot.
+ **Surface quality**
  - Parts with a gentle curve on the top surface will have a stair-step effect.
+ **Airflow inside the build chamber**
  - Consistent airflow and temperature helps model quality. Air flows in from the sides of the oven.
  - Placing tall models in the center, rather than the edges, helps to keep airflow regular.
  - Placement in the build chamber is handled at a later step.

ORIENT THE PART

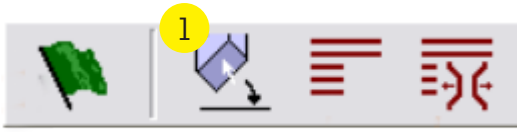
There are two fast options for part orientation.

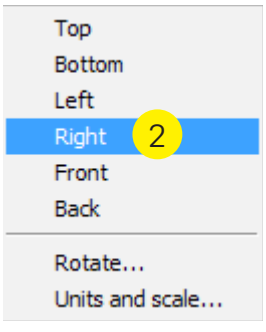
Automatic Orientation

+ Select this option from the STL menu.

Manual Orientation

1. Click the **Orient Part** button in the toolbar.
2. Click the desired orientation from the menu.
3. Click that surface of the model.
   - For example, clicking **Right**, then selecting a surface of the model will orient the model so that face is to the right.





More options

+ **Rotate...** will allow you to rotate the model around the X, Y or Z axis.
+ **Units and scale...** will allow you to change the units (inches or mm) or to scale the model.

SLICE THE MODEL

1. Click the **Slice** icon.

Models generally have hundreds of slices.

The STL file will be hidden. The red lines represent the slices.

+ **Once the model is sliced, it cannot be modified without deleting the slices**.
  - ▸ Select **Slice** > **Setup**.
  - ▸ Click the **Delete Slices** icon.

CREATE SUPPORTS

Models with overhanging or cantilevered elements will distort if they are not supported. The software creates supports for any surface at less than a 45 degree angle, but this can be changed in the software. See Shop Staff for assistance.

+ See page 8 for a review of support styles.

1. Click the **Create Supports** icon.

CREATE TOOLPATHS

1. Click the **Create Toolpaths** icon.

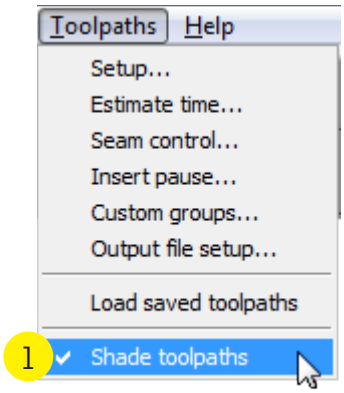
  - ▸ A toolpath is the path that the extruder head will follow to lay down the filament.

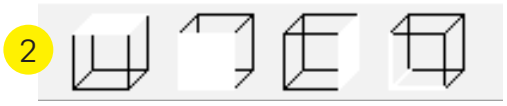
INSPECT THE LAYERS

Inspecting individual layers can help identify problems with the model.

Show the toolpaths

1. Choose **Shade toolpaths** from the Toolpaths menu.
2. Click the **Display Top** icon.





INSPECT TOOLPATHS & SUPPORTS

You may want to review the toolpaths and supports. Deleting unneeded support can speed your print times.

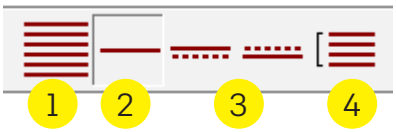
+ Slice lines will be red.
+ Model material will be green.
+ Support material will be blue.

To estimate the print time, select **Toolpaths** > **Estimate Time**.

This will allow to see the time savings when editing the model settings.

Using the Layers toolbar

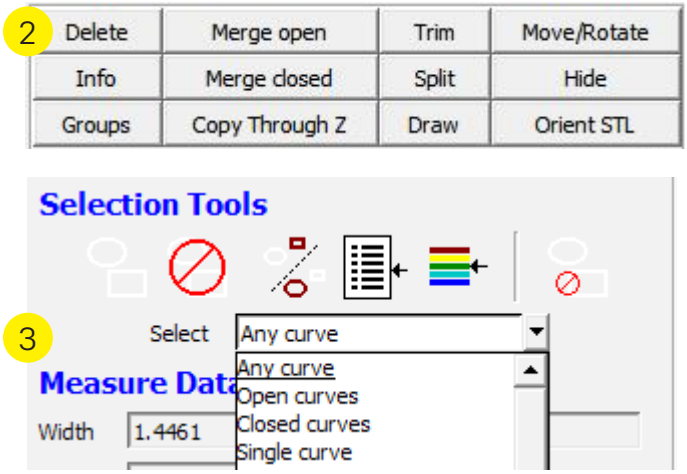
1. View all layers
2. View the current layer only
3. View the current layer, plus the layer above or below
4. View a range of layers
   - ▸ Select the layers in the panel to the right



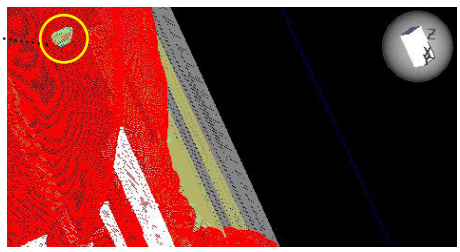
Inspect individual layers

+ Pressing Page Up or Page Down on the keyboard will move up or down through the model one layer at a time.

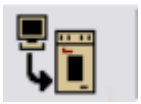
1. To delete unnecessary support material, select **Toolbox** from the View menu.
2. Click **Delete** in the Toolbox.
3. Select **Support Material Curves** from the Selection Tools menu.
   - ▸ Curves are what the software calls the lines.
   - ▸ This option will only select support material, so you don't accidentally delete model material.
4. Click a line (or drag for multiple).
5. Click OK to delete the support path.



The blob is an example of support material that can be safely deleted with the **Support Material Curves** command.



SEND THE JOB TO CONTROL CENTER



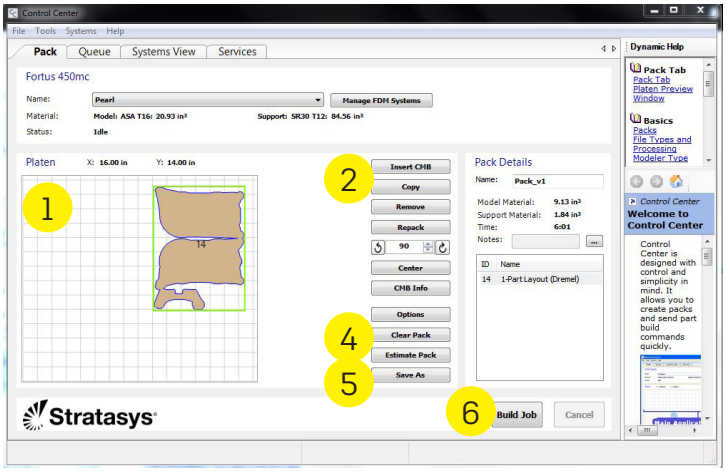
Click the printer icon to send the job to Control Center.

+ This will save your file as a .CBM in the same directory as your .STL file.
+ To print the file again, simply open the .CBM,

USING CONTROL CENTER

Control Center is the software used to pack a tray with parts and send to the printer.

1. If there is an existing model on the grid, select it and click **Remove**.
2. To add additional models, click **Insert CMB**.
   - ▸ To add an additional copy of your model, select it and click **copy**.
3. Click and drag your model to change their position if desired.
4. Click **Estimate Pack** to estimate print time and material use.
   - ▸ Jobs that use 100 cubic inches (or more) of support and model material require Shop Staff approval.
5. Click **Save As** to save the job.
   - ▸ Use this naming convention: *team_firstname_lastname_sheetname*
6. Click **Build Job** to send the job to the printer.



THE OVEN IS HEATED WHILE PRINTING, AND MAY BE AS HOT AS 500° F (240° C).

+ Wear heat resistant gloves before opening the door or handling hot parts.
+ When removing the build sheet it will be hot.
   - ▸ Clear a space for it before opening the door.
   - ▸ Open the door all the way.

PREPARING TO PRINT

Inside the oven is an aluminum plate called the **build platform**. A plastic **build sheet** needs to be placed on the build platform, and the model is built on the sheet.

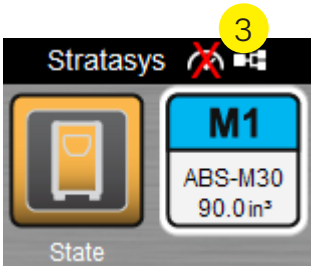
A sheet can be reused, but the print must be made on an unused section.

Note: Different materials require unique build sheets. For example, printing with nylon requires the use of a nylon compatible build sheet.

Using the wrong build sheet may cause it to melt inside the printer.

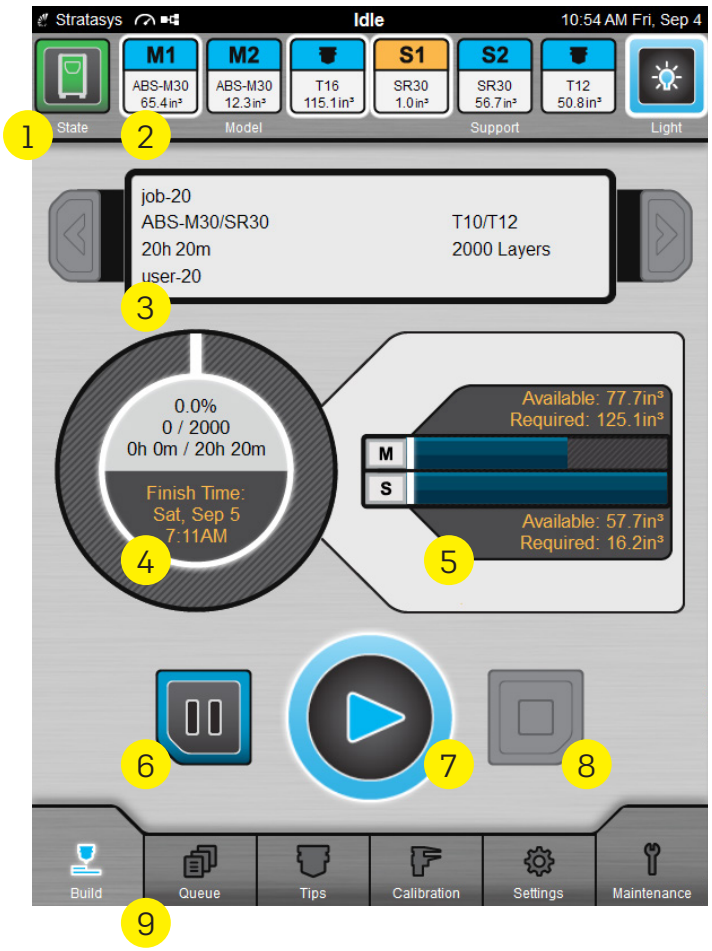
+ There are 3 types of build sheets
  - ABS, ASA and Polycarbonate
  - Nylon
  - Ultem

1. Open the lower printer door and empty the trash can.
2. Insert the correct build sheet for your selected material.
   - Ensure the build platform is clean.
   - Any filament adhered to the sheet must be on the top side.
   - Place the sheet on the platform and slide it to the back.
3. Close the door and check the Vacuum Seal Indicator on the screen.
   - If the sheet doesn't seal after a minute, reposition the seal.



Understanding the Control Panel

1. Machine State (touch for more info)
2. Material and Support status
   - S1 (support spool 1) is yellow, meaning that it is almost empty.
3. Current job stats
   - Use the arrows to scroll through current jobs.
4. Completion data
   - Percent completed
   - Layers completed
   - Elapsed time
   - Expected completion time
5. Material and support, required and available
   - Note that 125 cubic inches are required for this job. Remember that Shop Manager approval is required for all jobs of 100 ci or larger.
6. Pause job
7. Start job
8. Cancel job
9. Queue



Check the printer status

1. Activate the machine by touching the control panel.
2. Check the material available.
   - If the material quantity is lower than the required material, see Shop Staff.
3. Press Start.

Locate parts on the build sheet

Using the touchscreen, you'll tell the machine where to place the parts on the build platform & sheet.

4. There may be "shadows" from the last job that printed.
   - Delete them with the trash can icon.
5. Your print job is displayed as a blue box.
   - Navigate your part to a free space on your build sheet.
   - Grid lines on the software correspond to the grid lines on the print bed.
6. Press OK.
7. The oven will begin to heat, and eventually start printing.
   - ULTEM jobs can take up to 3 hours to preheat the oven.
   - Other materials heat much faster.
8. The display will show the estimated completion time.
   - Daytime jobs must be collected at completion.
   - Overnight jobs must be picked up first thing the next morning.
   - ULTEM and Polycarbonate will have an additional cool-down period at the end. See Shop Staff for details.

Start the print job

1. Press the start button on the control panel.
2. Stay by the printer for a few minutes to ensure that it starts printing normally.

CLEANING THE PRINT

After printing, the part will need to be cleaned by removing the support material. Large pieces of support can be broken off, but soaking the part in a chemical bath will remove virtually all traces of the support material.

Manual removal of support material

+ When removing support material, safety glasses must be worn.
+ Removing support material must happen at the print cleaning station.

Dissolving support material in the bath

+ The following PPE must be worn:
  - Long pants and closed toe shoes
  - Chemical goggles and face shield
  - Nitrile gloves
  - Lab coat
+ PPE must be worn when working with the chemical bath, including:
  - Placing parts in the bath
  - Removing parts from the bath
  - Rinsing parts in the sink

THE CHEMICAL BATH IS AN ALKALI SOLUTION THAT WILL BURN SKIN OR CLOTHING; FOLLOW ALL PPE REQUIREMENTS.

ABOUT THE BATH

The bath has two functions to help remove support material: heat and ultrasonic vibrations. The two have separate controls, and are not related.

Use the wall chart to determine the settings.

USING THE BATH

1. Put on PPE.
2. Turn the Sound off to stop the vibrations.
3. If the power is off, press the green button.

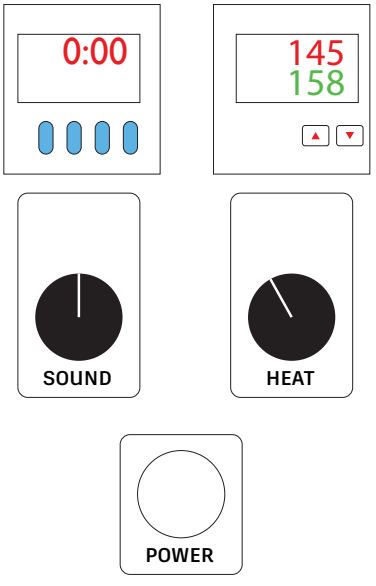
Placing Parts in the Bath

1. Place the parts in the basket.
2. Gently lower the basket into the bath.
3. Set the timer and temperature.
   - Use the blue keys to set the timer.
   - Turn the SOUND switch to ON.
   - Use the red arrows to set the target temperature.
   - Turn the HEAT switch to ON.

Removing Parts from the Bath

1. Put on PPE
2. Remove the basket and place parts into the sink.
3. Put the basket back in the bath.
   - If there are still parts in the basket, turn the bath back on.
4. Take the container to the sink in the mold room.
5. Rinse the parts
   - Use running water for 1 minute or still water for 15.
6. Leave the parts next to the sink to dry.

BEWARE OF SPLASHING WHEN USING THE BATH.



AREA AND MACHINE PREPARATION

1. Open the STL in Insight.
2. Configure the Modeler.
3. Select the infill and support style.
4. Orient the model.
5. Slice the model.
6. Create supports.
7. Create toolpaths.
8. Inspect the layers.

SELECT THE SAME MATERIAL IN YOUR MODEL THAT IS INSTALLED IN THE PRINTER.

PRINTING THE MODEL

1. Send the model to Control Center.
2. Send the model to the printer.
3. Return when the part is complete.
4. Use the chemical bath and mechanical tools to remove support material.
5. Rinse and dry the part.

WEAR SPECIFIED PPE WHEN USING THE CHEMICAL BATH AND REMOVING SUPPORT.

CLEANUP

1. Empty the trash before and after using the printer.
2. Place a build sheet into the oven.
   - This extends the life of the vacuum pump components.
3. Wipe down the counter at the part cleaning station.

LEAVE THE PRINTER CLEAN AND READY FOR THE NEXT PERSON BEFORE YOU WALK AWAY.