MSMHA 3D Printing Guide

An instructional booklet for scanning and printing models within the Modern Human Anatomy Suite

A booklet created by Zachary Stetter

Contents

Artec Scanner Overview

Computer Bag

Contains the computer alongside the charger, power strip, and rotating turntable to place samples on.

Scanner Carrying Case

Heavy duty carrying case for the Artec scanner and its power cord.

Object Turntable

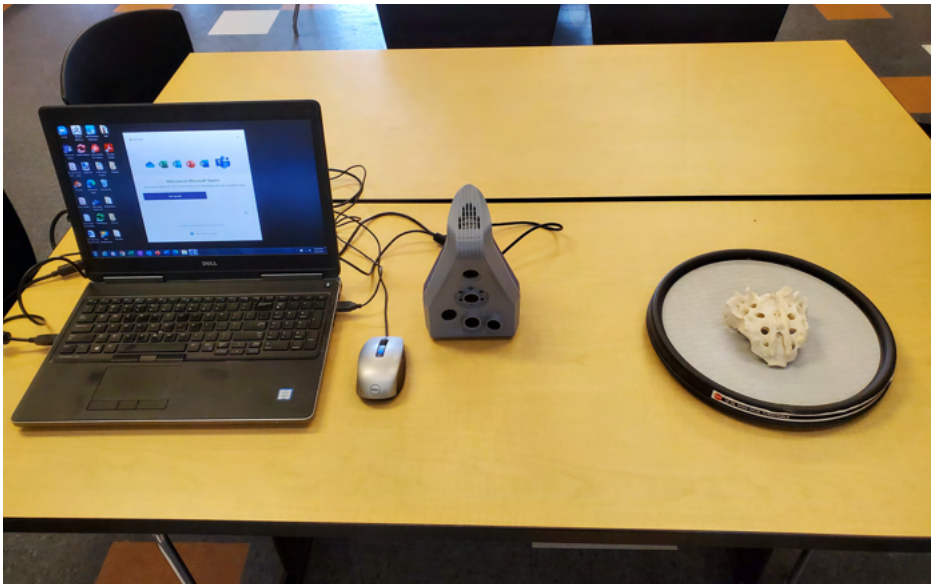
Laptop

Artec Power Cord

Artec Scanner

Setting up your Workspace

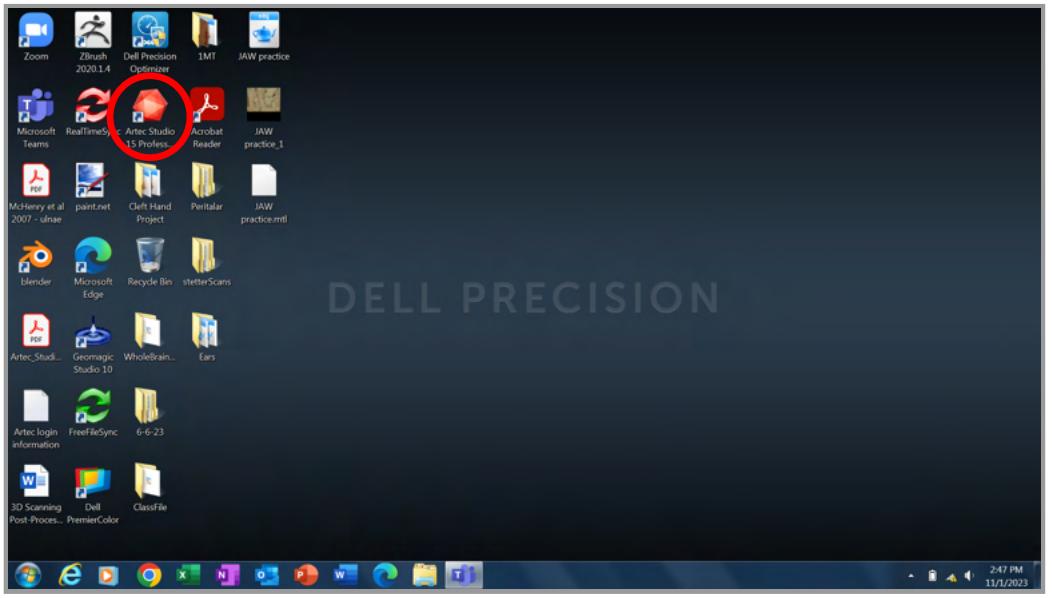
It is always a good idea to set up the area you're going to be working at before you start scanning. This include getting your model properly placed, pulling up the Artec program, making sure that everything is plugged in and your wires are out of the way, and you have plenty of room to maneuver around if you need to.

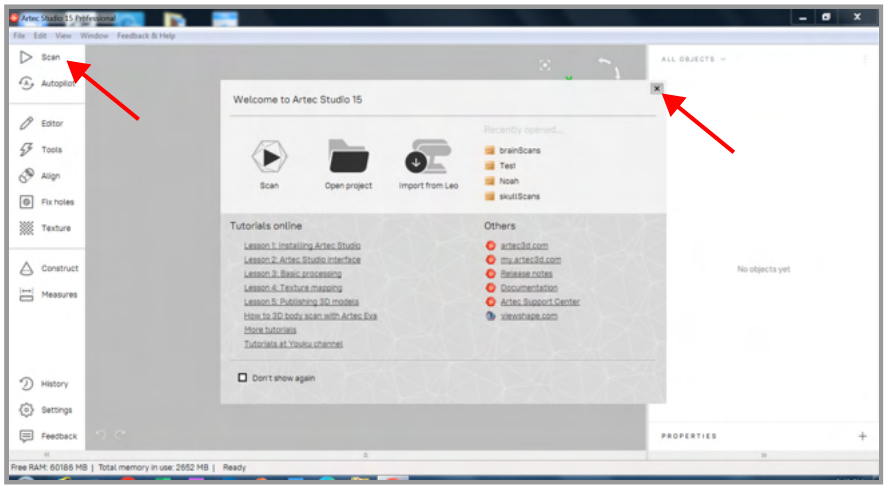


Logging In

The password to the computer is MHA. An internet connection should not be required for login.

Double click to open the Artec scanning software



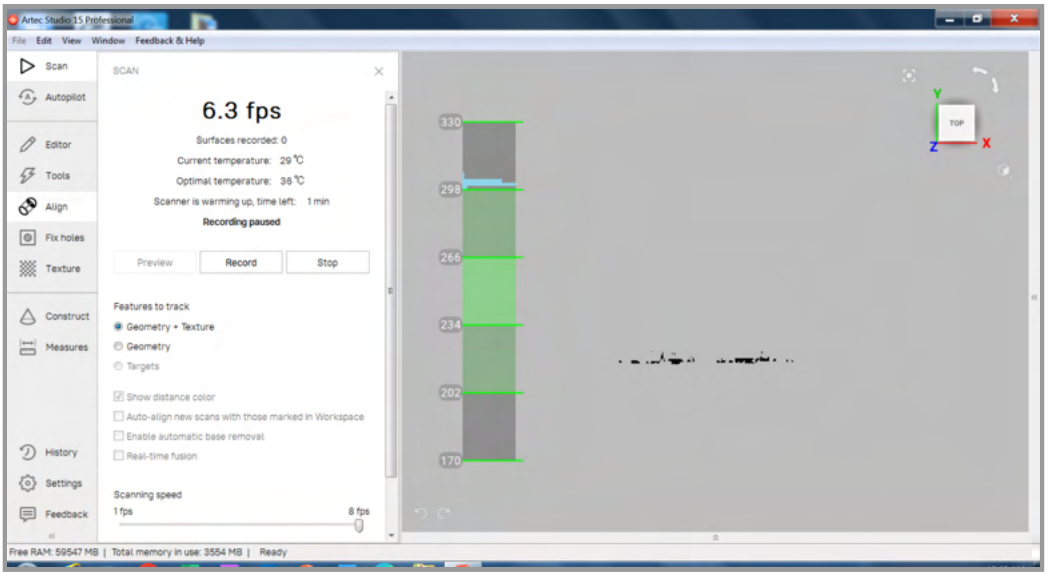


Get the software running

Click the 'x' on the welcome tab, then click the play button next to 'Scan' to start setting up the scanner. Make sure that the scanner is plugged into the USB port of the computer.

Warming Up

The scanner will take some time to warm up. While you do not necessarily have to be at the optimal temperature, it is better to allow the scanner to be around 34-35 C before beginning.

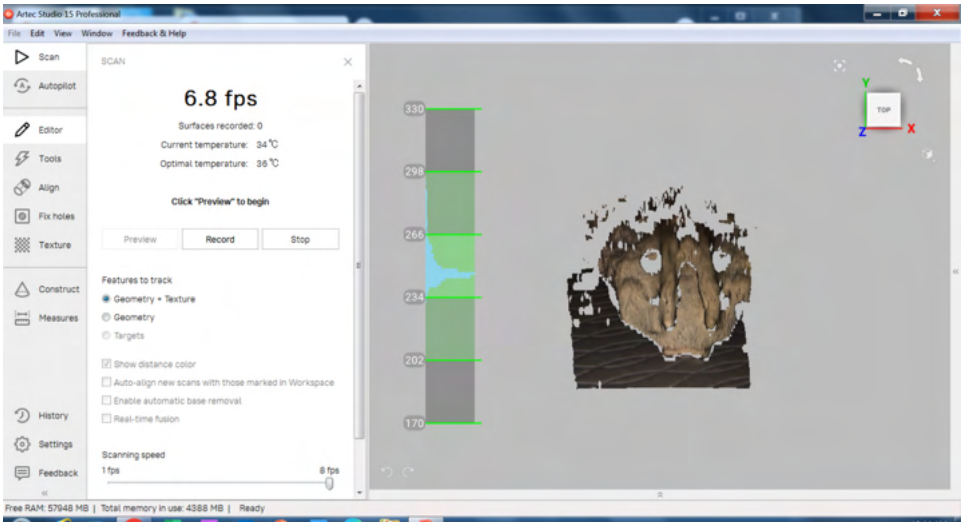


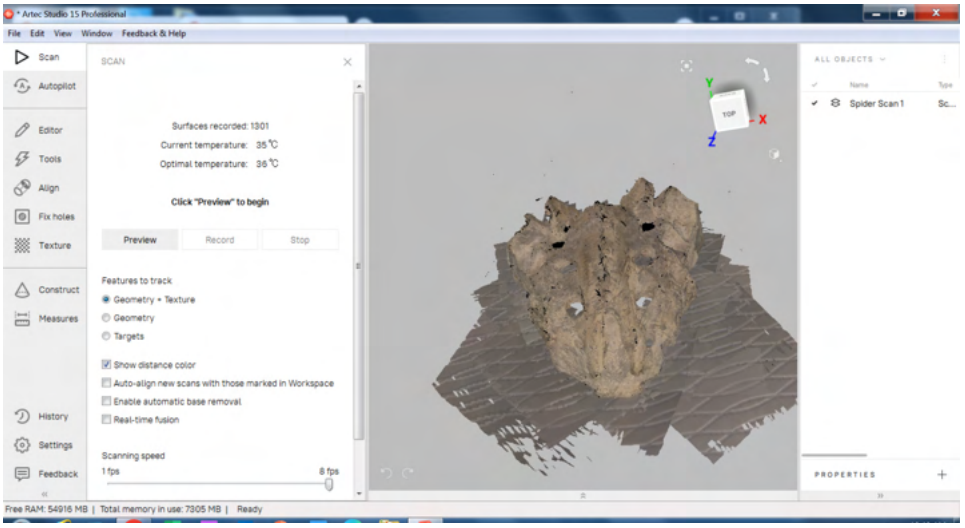
Scanner Controls

On the back of the scanner there is a switch near the top of the handle. Pressing the switch up once will begin the calibration of your scanner with the object that you are trying to scan. Pressing the switch up the second time will begin the process of scanning the object.
Once you have finished your current scan, pressing down on the switch will stop scanning and save the collected data.

Calibration

Press up once on the scanner switch to start calibration. When you're calibrating, keep your eyes on the screen to make sure that the object is being properly captured. Watching the histogram on the side, move back and forth while keeping the visual peaks within the middle three bars.



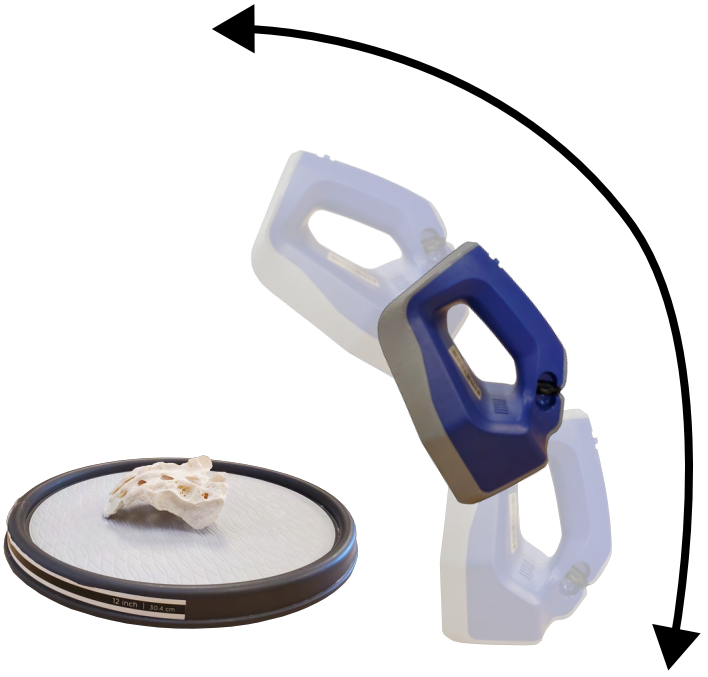


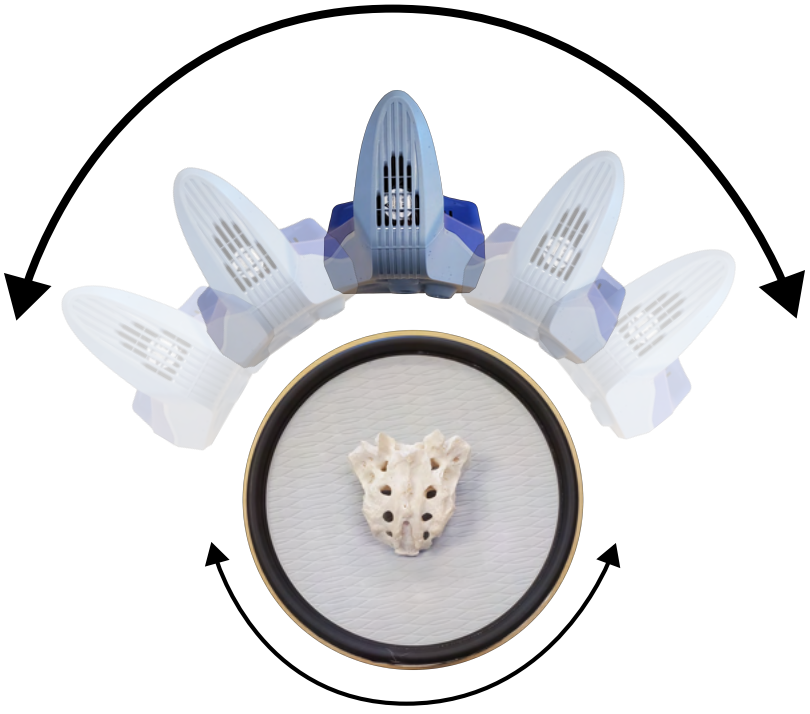
Scanning

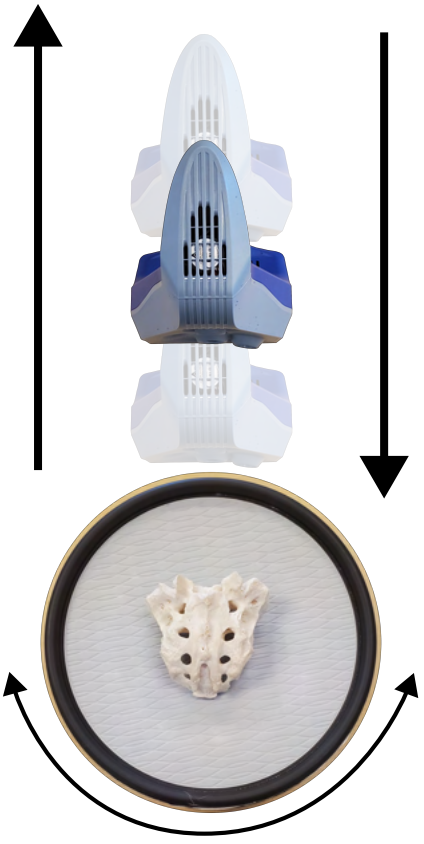
Press up again on the switch to start scanning your object. As you scan, slowly move up, down, around, back, and forth with the scanner around your object. Once again, keep the histogram within the middle three bars on the side of your screen for the best results. Press down on the switch to stop scanning and save your data.

Scanner Movement

When using the scanner, try to use slow, steady, sweeping movements. You never want to twist your wrist or turn the scanner upside down when scanning your object. Use the turntable the object is resting on to your advantage to capture as many angles as you can in a single scan.

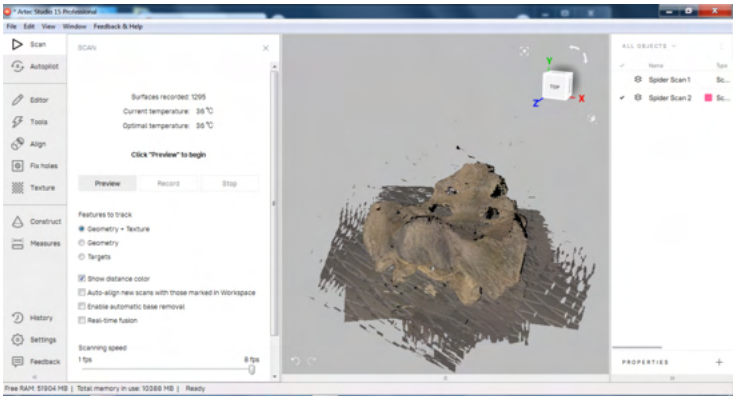


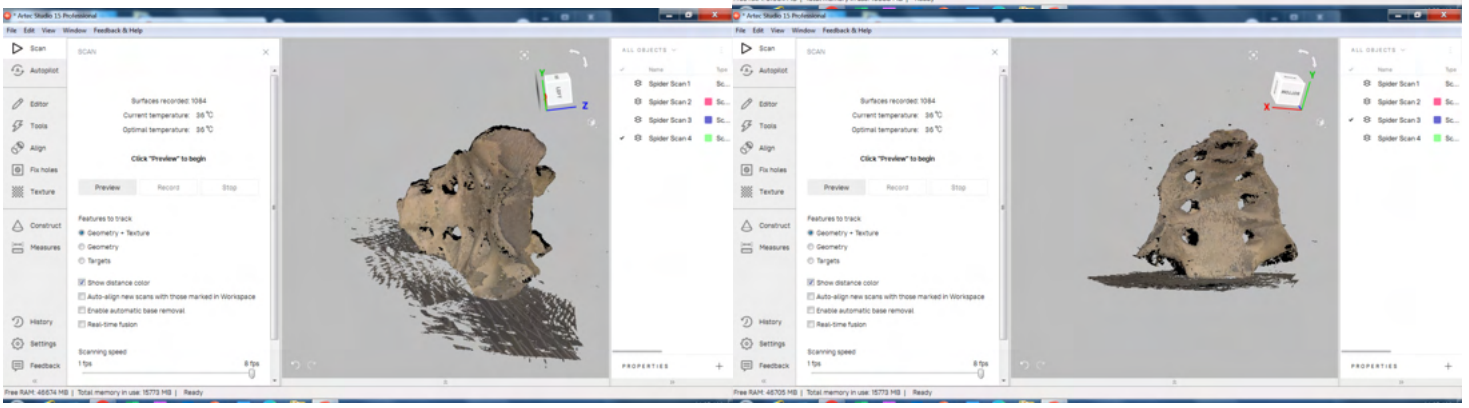


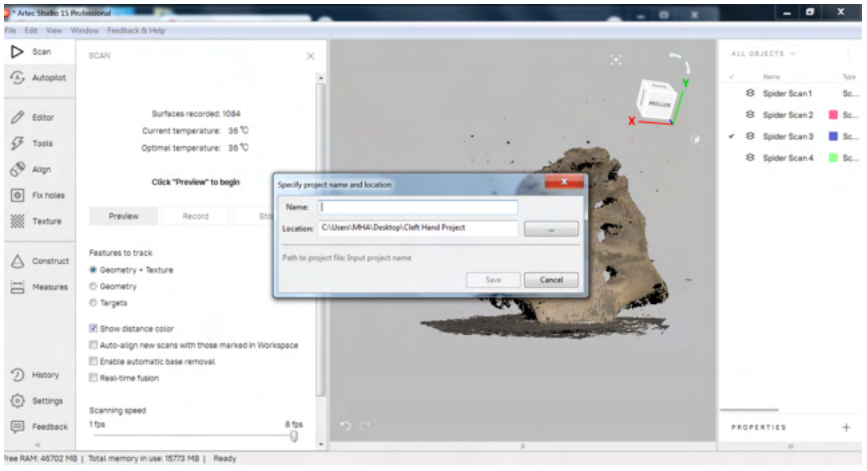


Capturing Multiple Angles

Once you've pressed down on the scanner switch to save a scan, readjust the object you are scanning. You want to capture as many surfaces as possible so that when you compile them later the program will have a lot of data to pull from. Turn your scanned object on its side/top/etc. on the turntable, then press up on the scanner switch to start another calibration. Every time you press down on the switch it will save a separate scan within the same project. Each scan you do in a single project will be saved in the tab on the right of the program, and their visibility with the checkmark next to them.



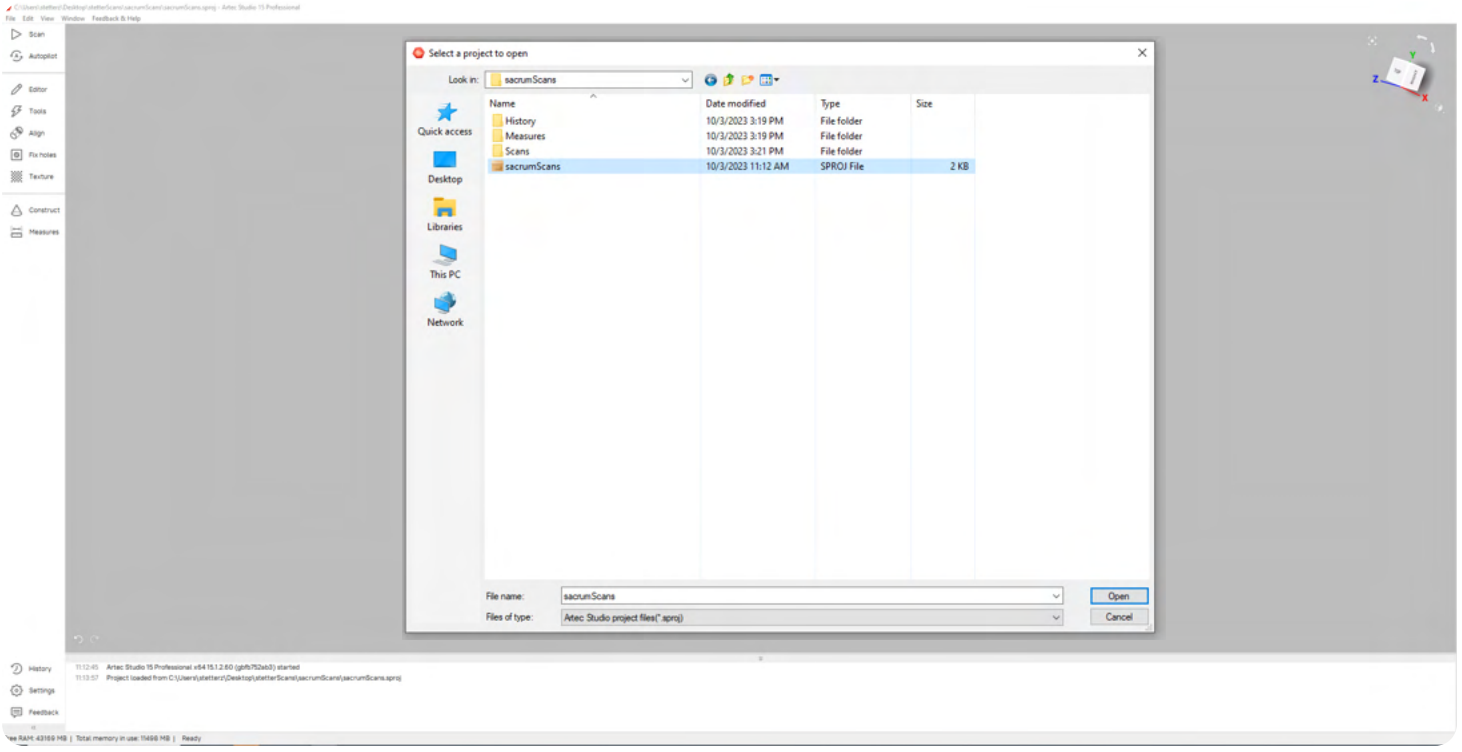




Saving Your Project

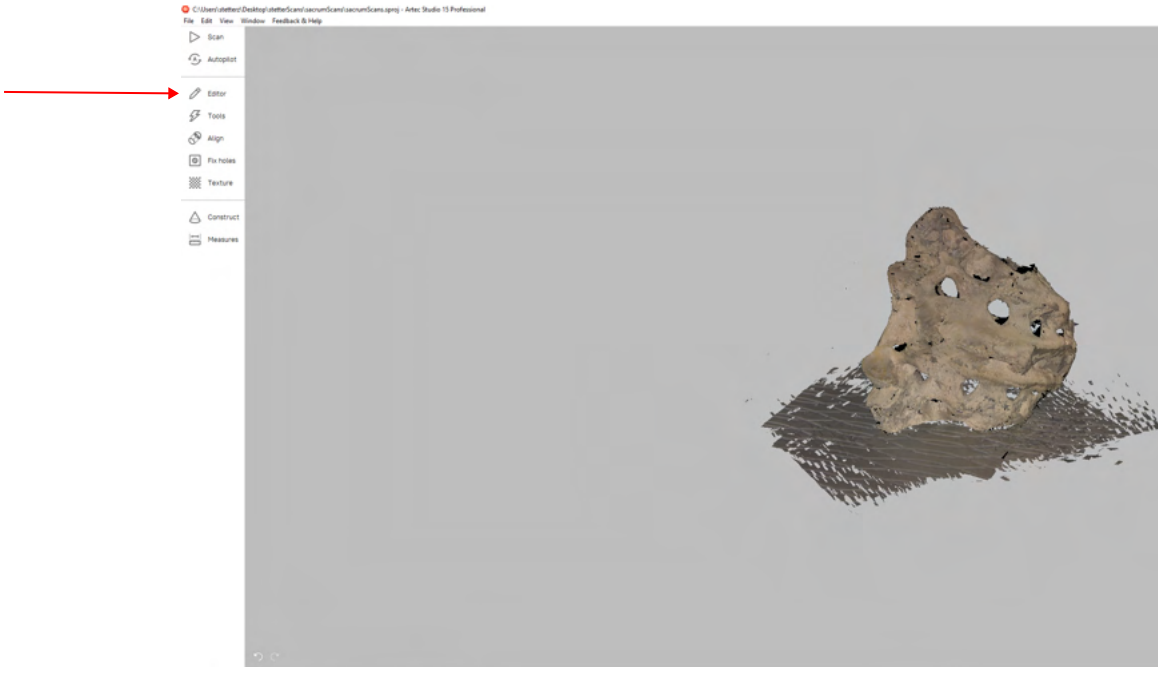
After you have all of the scans that you need, save the project as a whole. You can chose a specific location in the file explorer by clicking the 3 dots next to the location tab.

When compositing your scans into a single model, I recommend using one of the lab computers as they tend to be more powerful than the laptop. You can create a model on the laptop with the same steps, but the program may take longer or crash in the process.



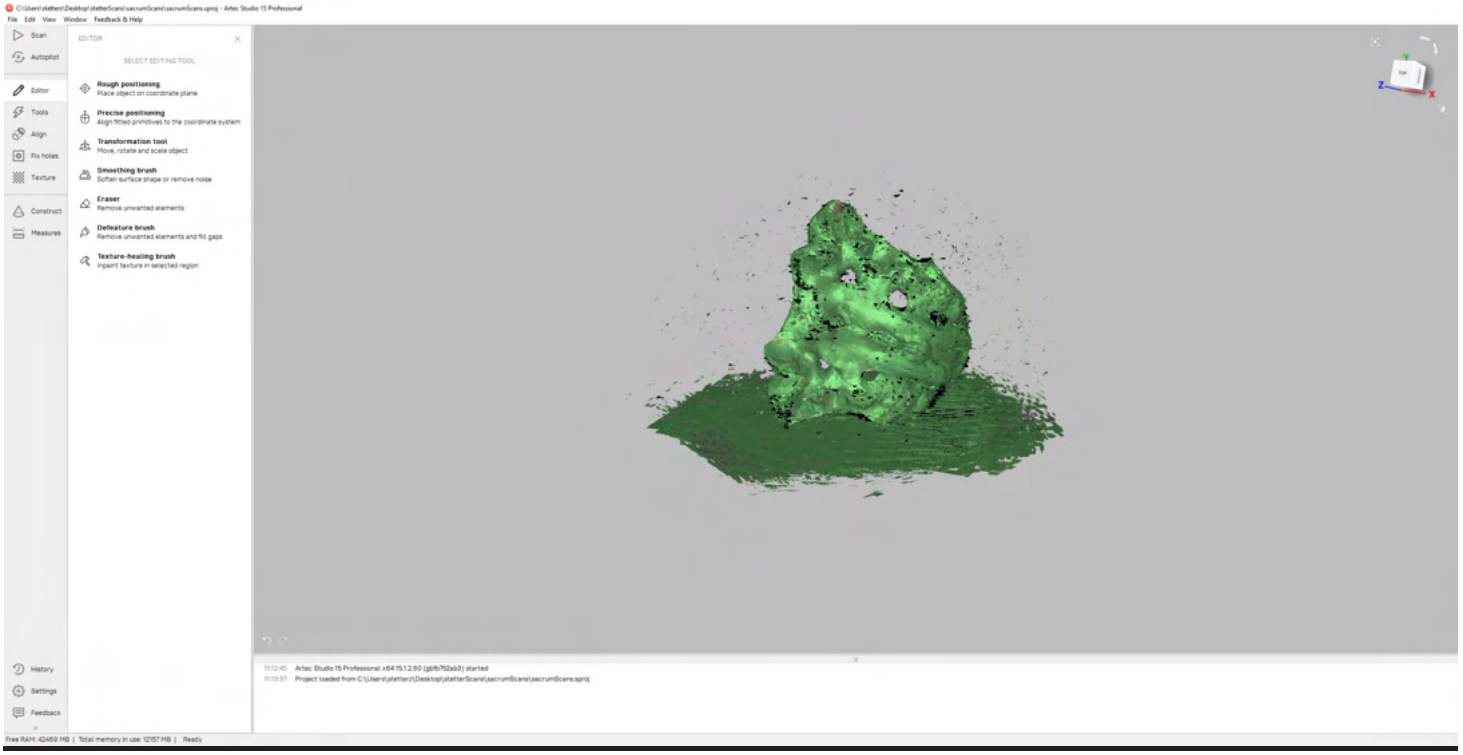
Opening Your Project

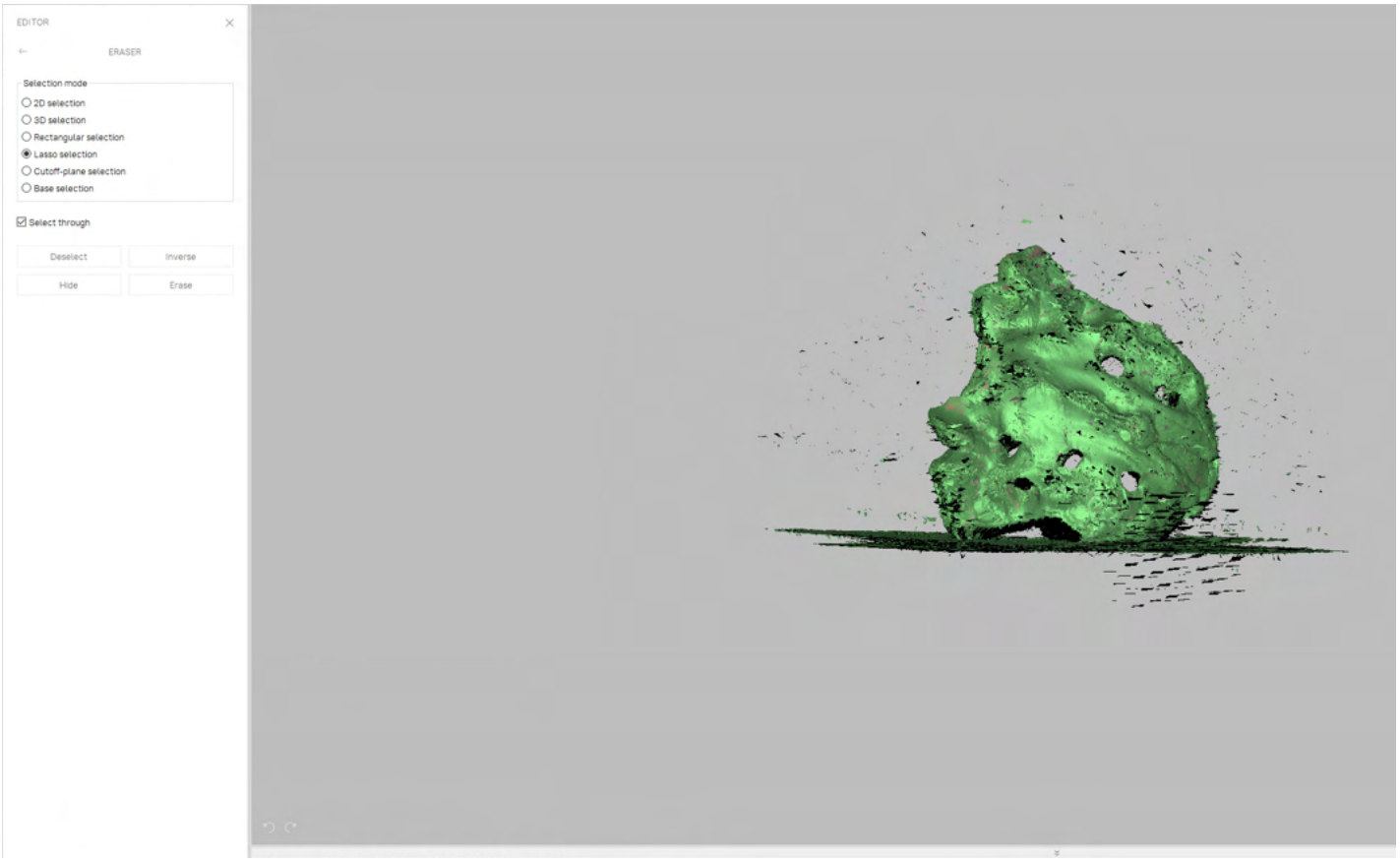
Once your project is opened, all of your scans should appear layered on top of each other. To start we will clean up the scans to only have the data that we want to preserve. We'll work through the scans one by one, so use the check marks on the right hand bar to make only one scan visible at a time.

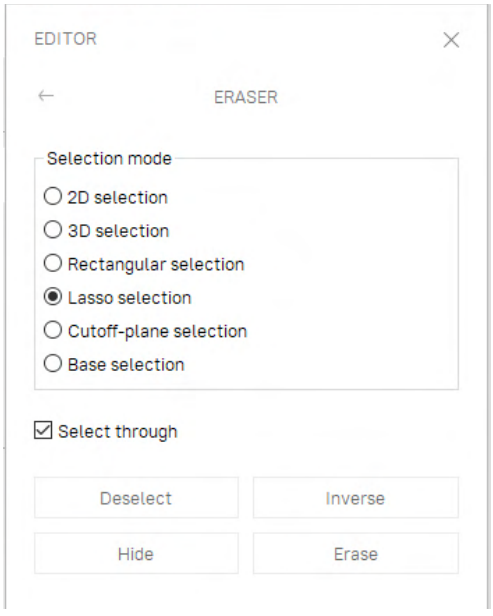


Select The 'Editor' Tab

On your left hand bar you'll see a variety of tools that can be used to clean up your scans into proper models. Luckily, this program is smart enough that the 'Autopilot' tool that we use later will get you 99% of the way there. However, using the eraser tool within the editor tab makes the cleanup process a lot quicker and easier. Make sure that you have only a single scan visible before opening the eraser tool.







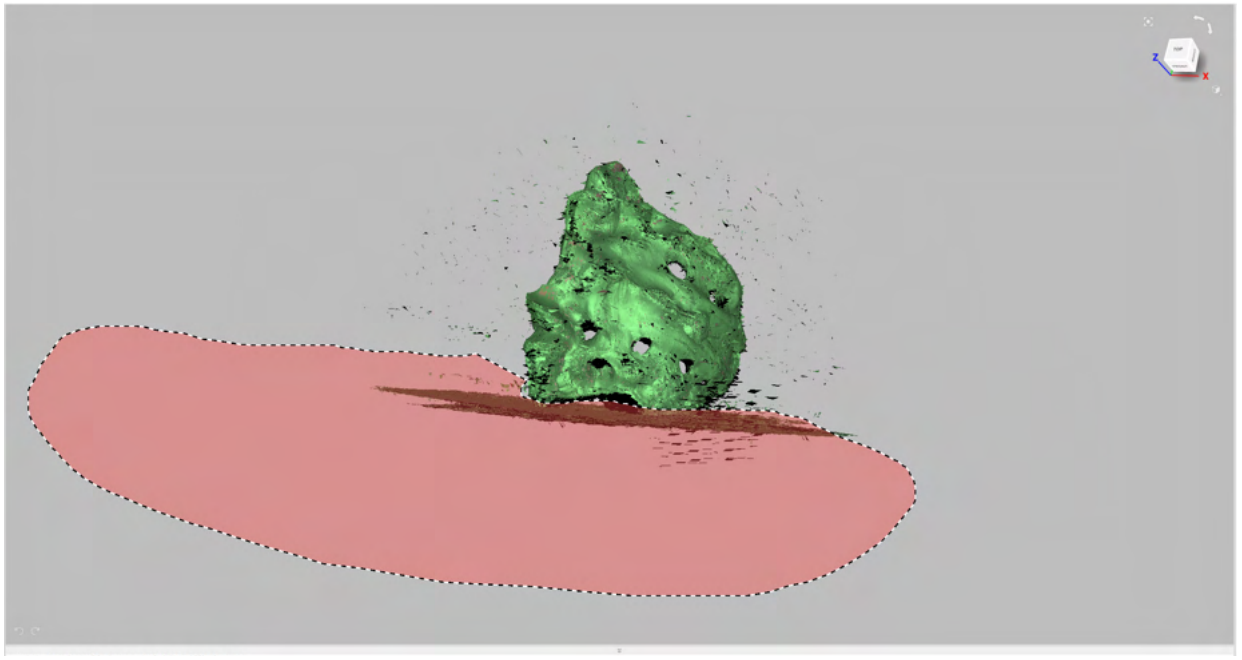
The Eraser Tool

There are 3 main tools that are used for efficient cleanup of models.

-Base Selection: This is the fastest tool in removing the unwanted base of your scan when it works. Simply click on the plane that the model is sitting on and it should select the surface to be erased. However, it can be inconsistent.

-Rectangular Selection: Use this tool to click and drag a rectangular shape that either selects or deselects parts of the mesh.

-Lasso Selection: This is the most flexible and consistent tool. Use the mouse to draw the outline of a shape, and everything within that shape will either be selected or deselected.

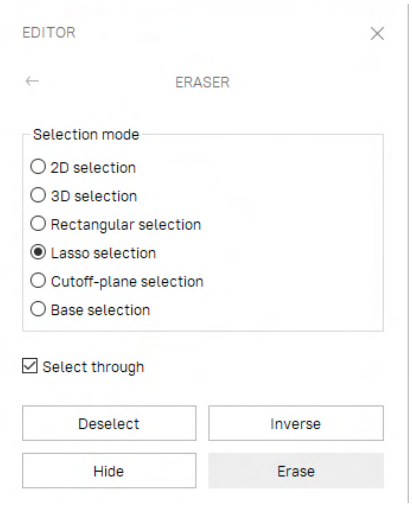


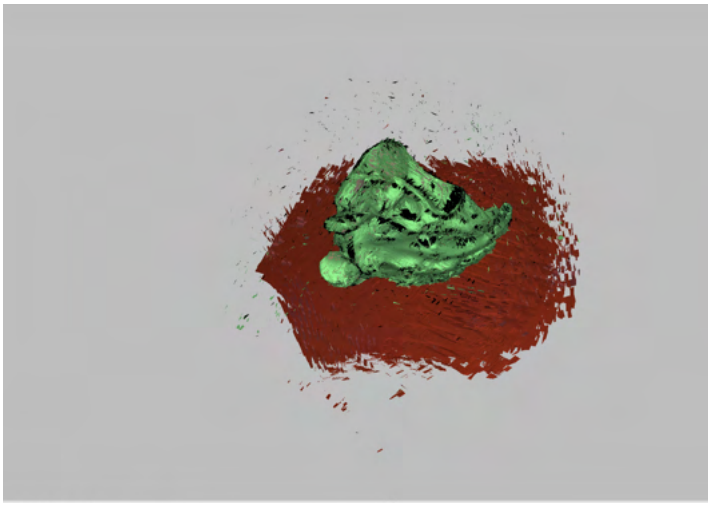
Cleaning Excess Data

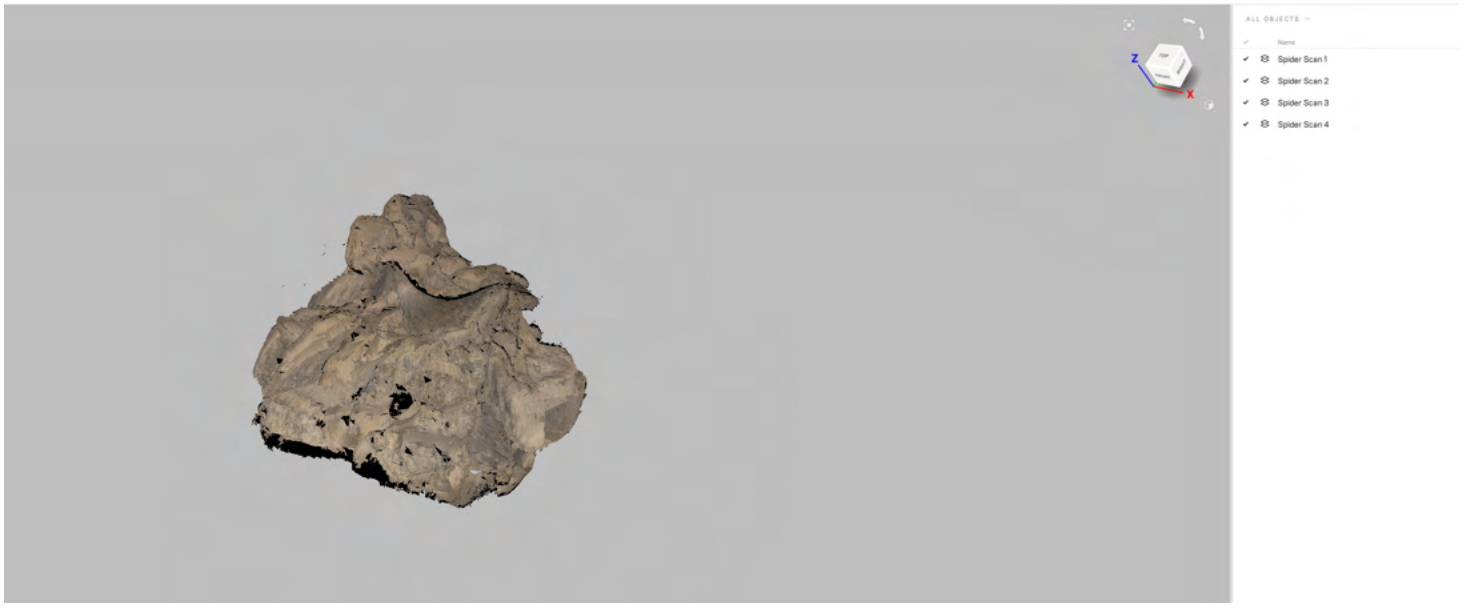
When selecting unwanted meshes with the eraser tool, hold down control and click your left mouse button to highlight the data you want erased. The lasso selection tool allows for great flexibility in tracing around the curves of organic objects. By turning the scanned object so the ground of your scanned object rests on the 'floor' of the scene, the lasso tool makes quick work of highlighting unwanted data. While you can also use the same tool to select the many floating bits of data surrounding the main model, don't feel the need to get every last one as the autopilot tool will help to clean up the floating bits outside of the main model.

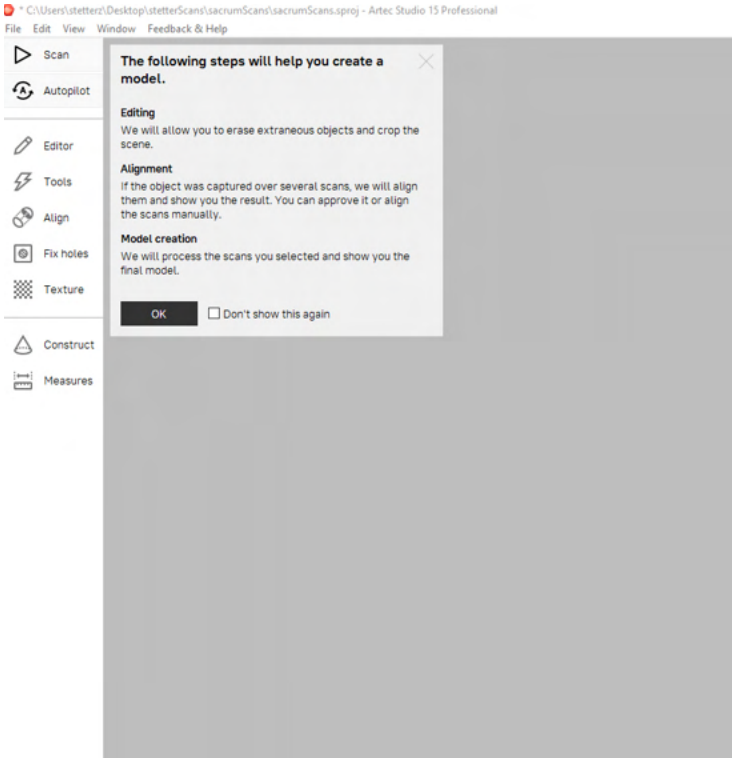
If an area of the model is highlighted that you need to keep, you can deselect it by holding down control+Shift+left mouse button. The selection area will highlight green instead of red.

Once you've selected your unwanted data, click the 'Erase' button to clear it from your dataset.





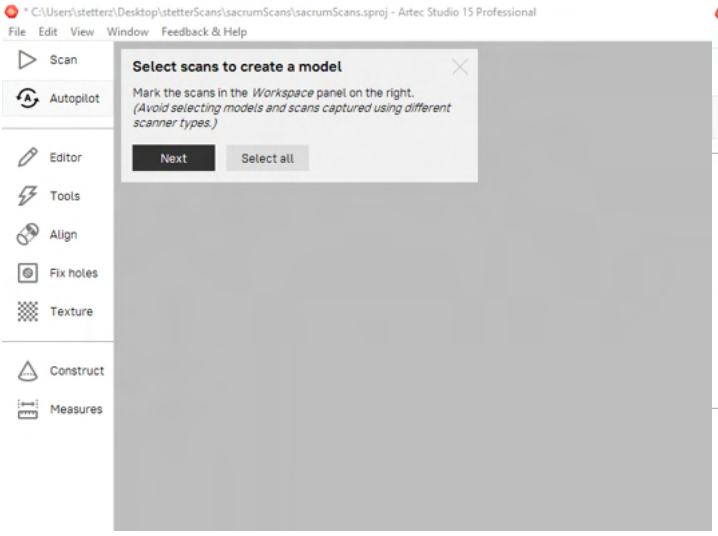




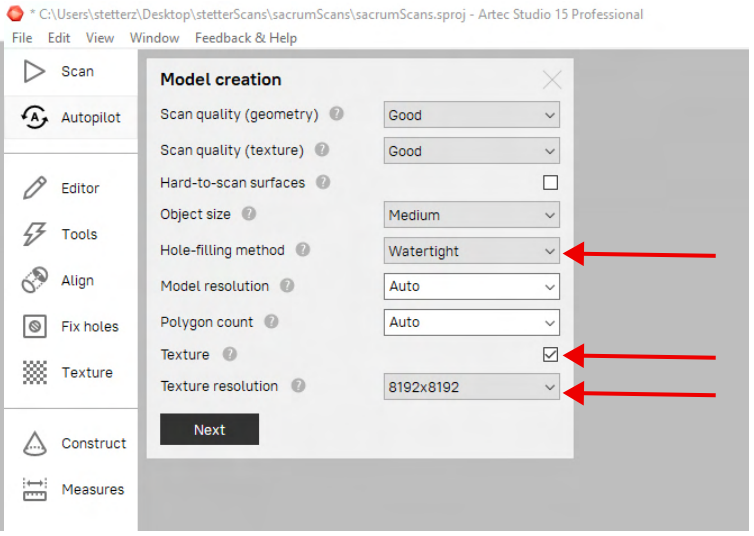
Autopilot Tool

Go through your scans one by one to erase the unneeded data. Once each scan is cleaned, make them all visible by turning on all of their check marks. Don't worry if there are some holes in the meshes or if their surfaces look rough, it is expected at this stage.

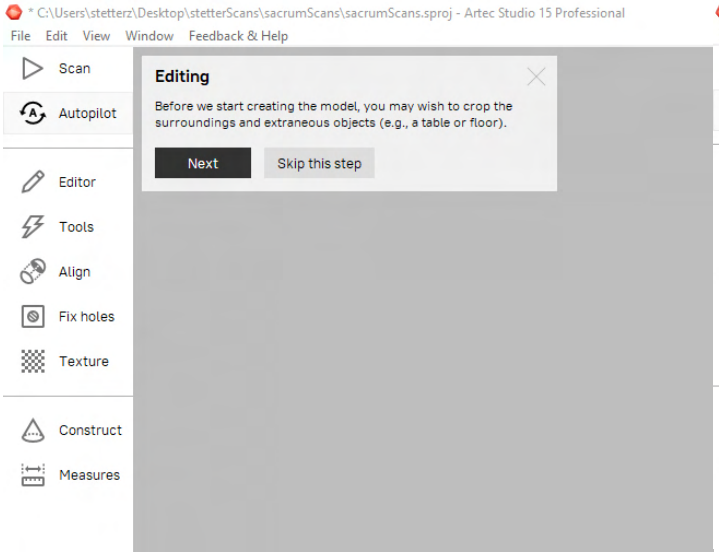
Select the 'Autopilot' tab on the left side of your screen and then click 'OK'.



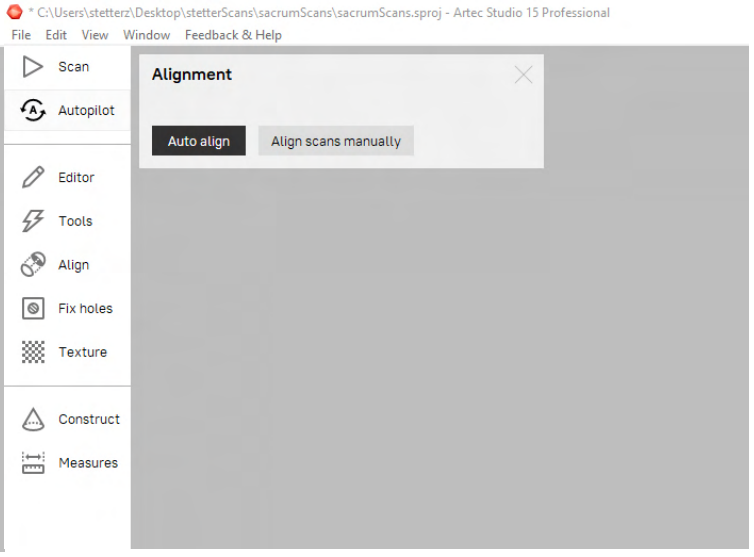
Double check that you've made all of the scans that you want to use visible, then select 'Next'.



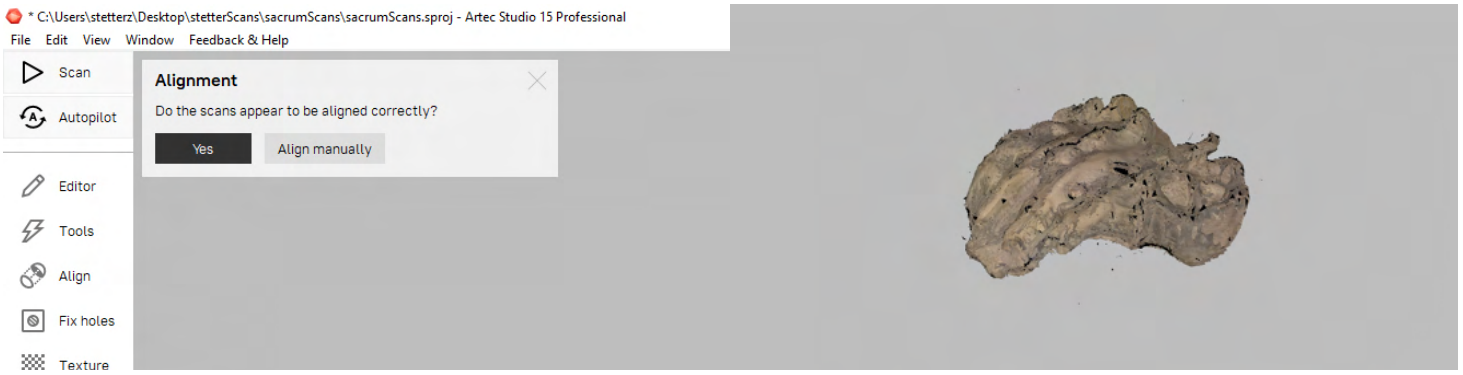
Within the 'Model Creation' tab, most settings will be optimal by default. Double check to make sure that 'Hole-filling method' is 'Watertight', which is best for 3D printing. Alongside that, make sure that 'Texture' is checked and select a high resolution if you want to create any renders of your scanned object. Texture files are rarely large in size, so it is often better to have it if you need it at any point.



Because we took care of editing out the floor and surroundings in a previous step, you can select 'Skip this Step'. If there is still a scan that needs cleaning, press 'Next' to open up the editor tools to finish cleaning the scan of excess data.

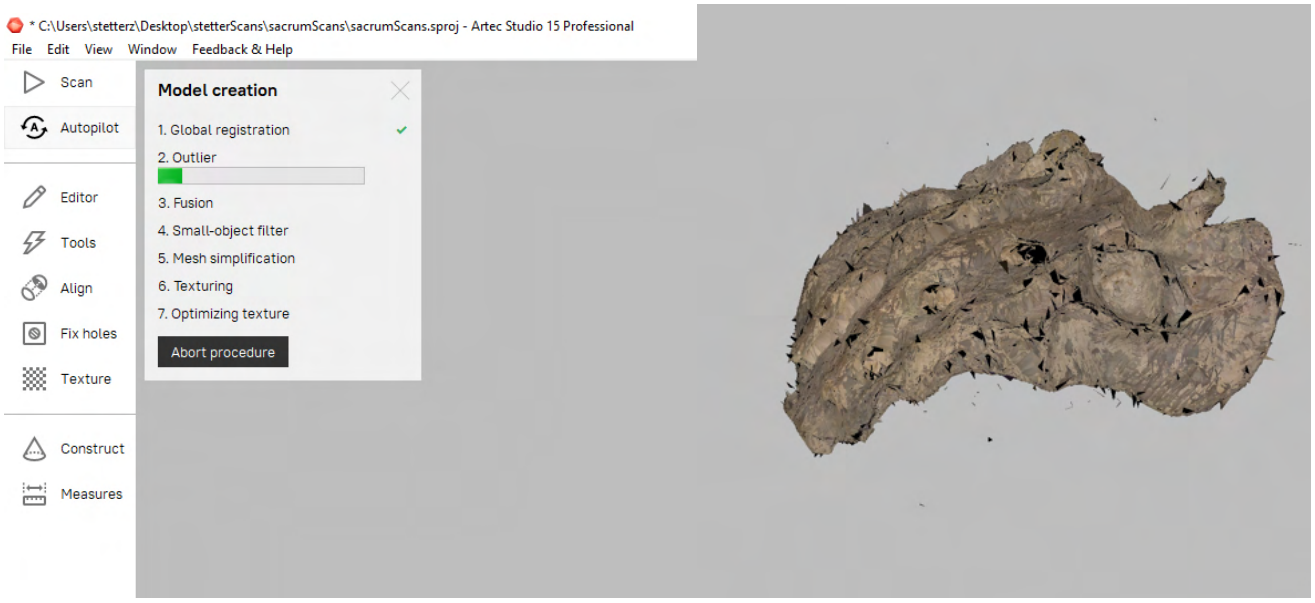


When aligning the scans, the more there are and the cleaner they are the more accurate the auto align function tends to be. Even with 4 scans the tool is very good at layering the scans atop each other. It is a big time save to click 'Auto Align' to test how accurately the program aligns the scans.

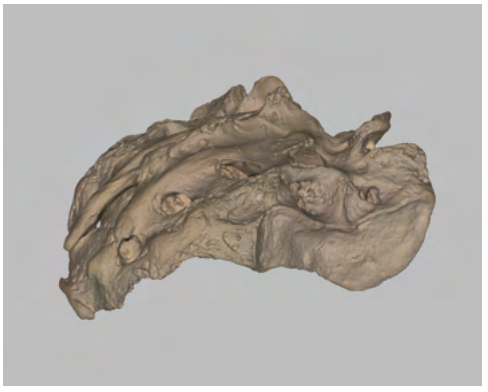


Once auto align has done its job, rotate around the model a few times to make sure that they are sitting atop each other properly. If they are not, click the 'Align manually' tool to give the program a helping hand in making sure the scans are properly aligned.

At this point there will still probably be some unexpected holes and 'debris' floating around the scans. Don't worry! These will be taken care of in the next step. Select 'Yes' to proceed.



Next, the program will automatically go through the process of Model creation. The more scans you provide the longer it will take, but the more accurate the final model will be. Be patient as the program works through all of the steps.



Congratulations!

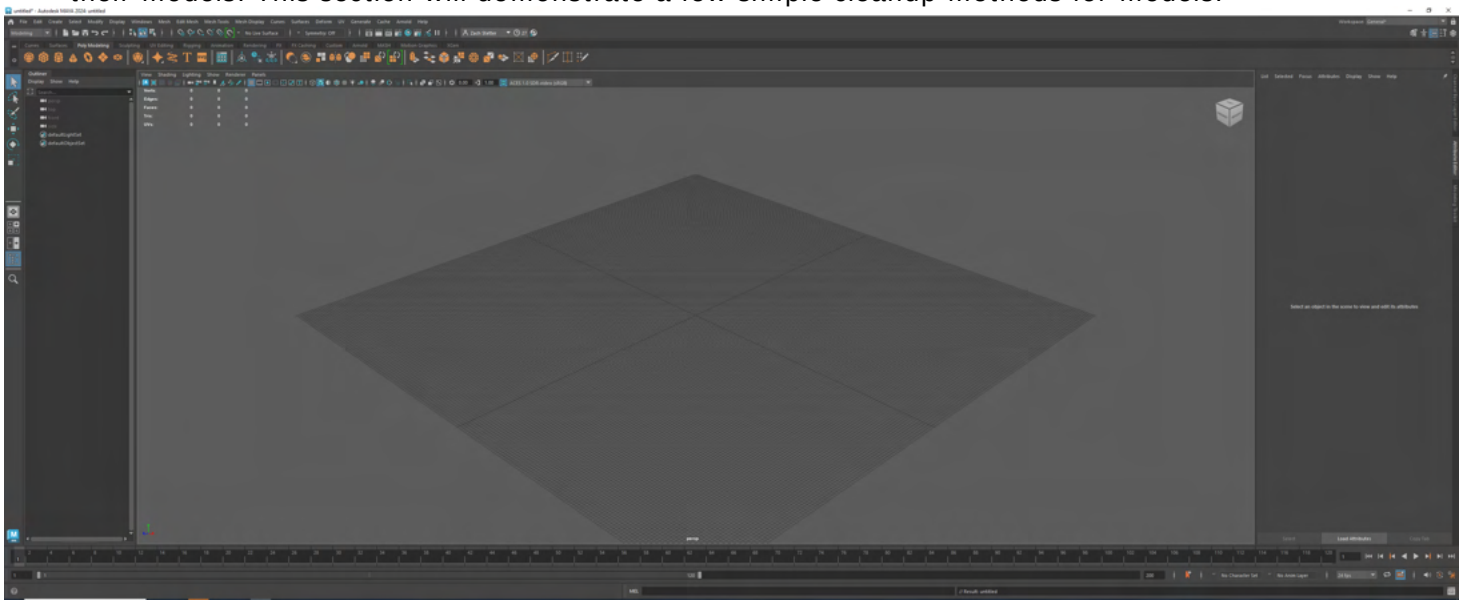
You've successfully cleaned and created your model.

Exporting Your Mesh

Now you're ready to export your mesh to use elsewhere. Go to File -> Export -> Meshes to export, then select the location you want your file saved. An easy file type to work with is an .obj. Your textures will export to the same location.

Cleanup in Maya

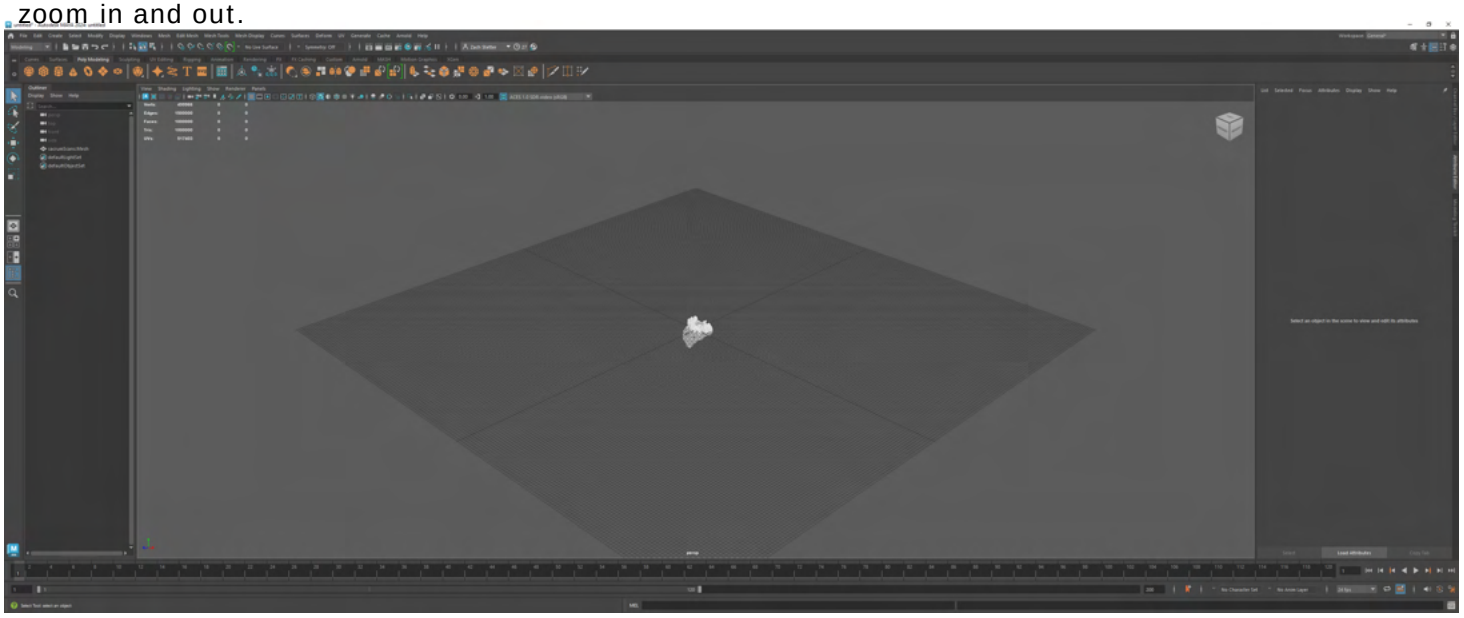
Sometimes a mesh needs additional cleanup before printing, especially if it has too many polygons or there are errors within the mesh. Maya can help quickly resolve these problems, and more advanced users can use the tools within the program to add extra features like bases or stands for their models. This section will demonstrate a few simple cleanup methods for models.

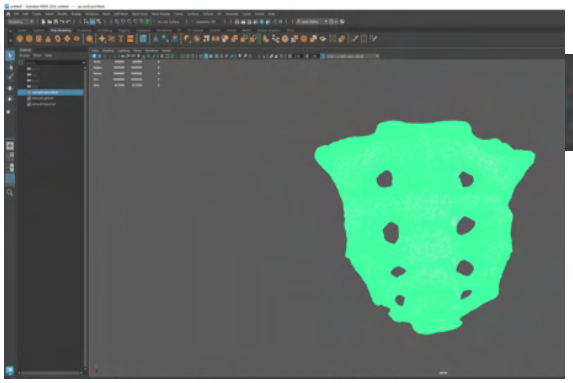


Import Model

To import an .obj, you can either drag and drop the file into the workspace of Maya or go to File -> Import and select the file that you wish to work on.

Once the model is within the scene, it may be either too small or too large to see depending on the scale you are working at. Select the model and press 'F' on your keyboard to focus it within your viewport. To move the camera within your scene, hold down 'Alt' on your keyboard. From there, click and hold your left mouse button to rotate around, use your middle mouse button to pan, or use your right mouse button to zoom in and out.

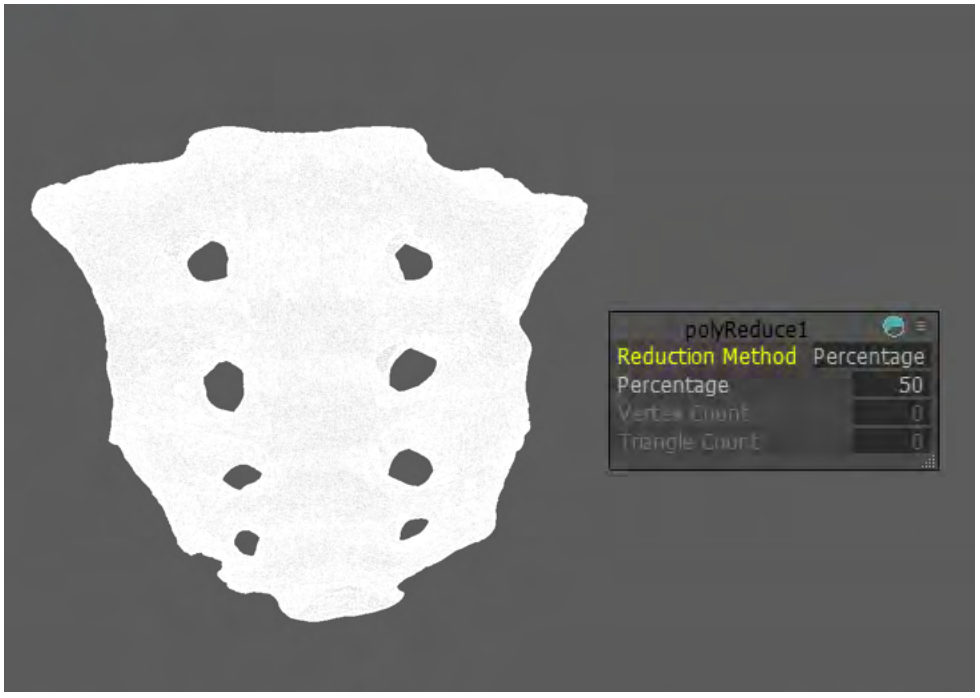




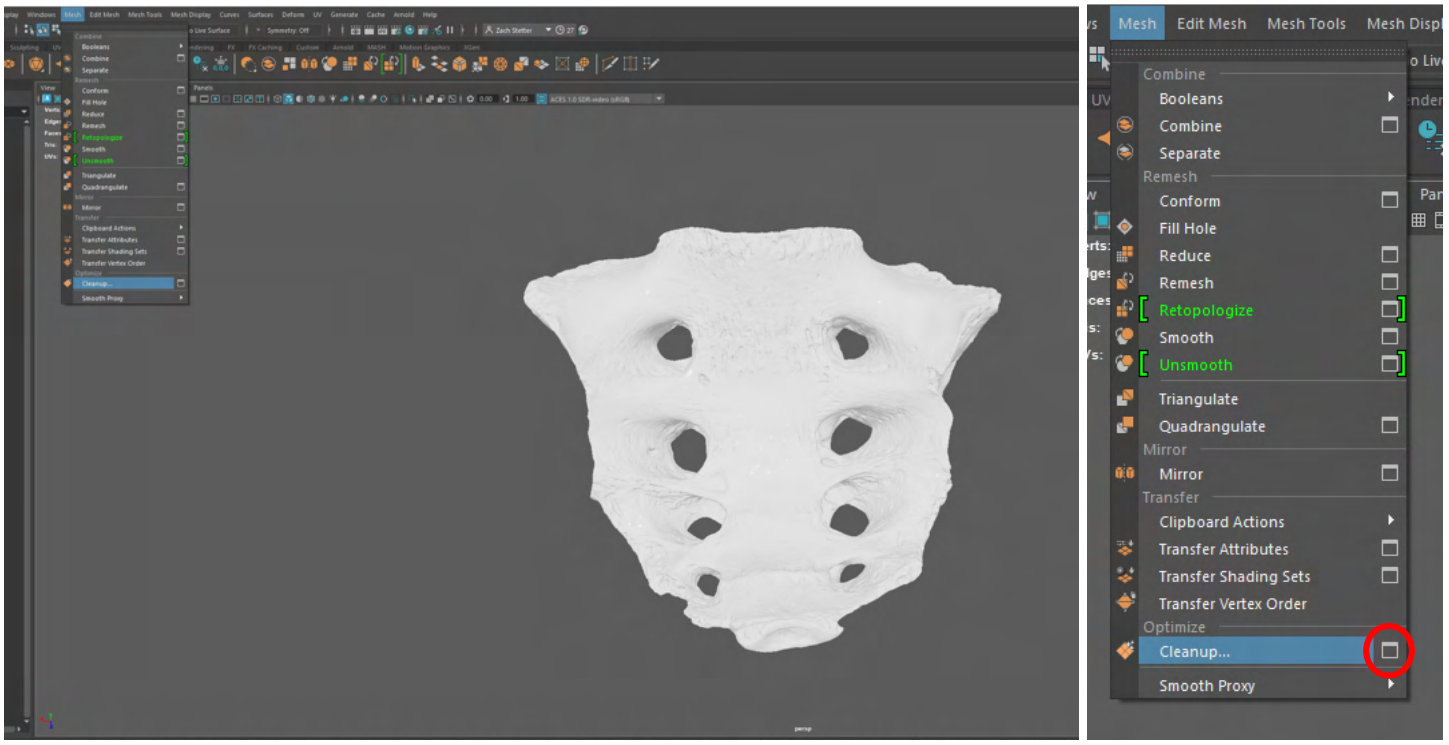
Reducing Polycount

Sometimes 3D printing programs have a hard time importing models to print when there are too many polygons (the little shapes that make up the model in 3D space). Maya can help by reducing the polygon count of your model while retaining its shape.

In the 'Poly Modeling' tab in the top toolbar, there is a tool called 'Reduce'. With your model selected, click the 'Reduce' tool and allow it some time to process.



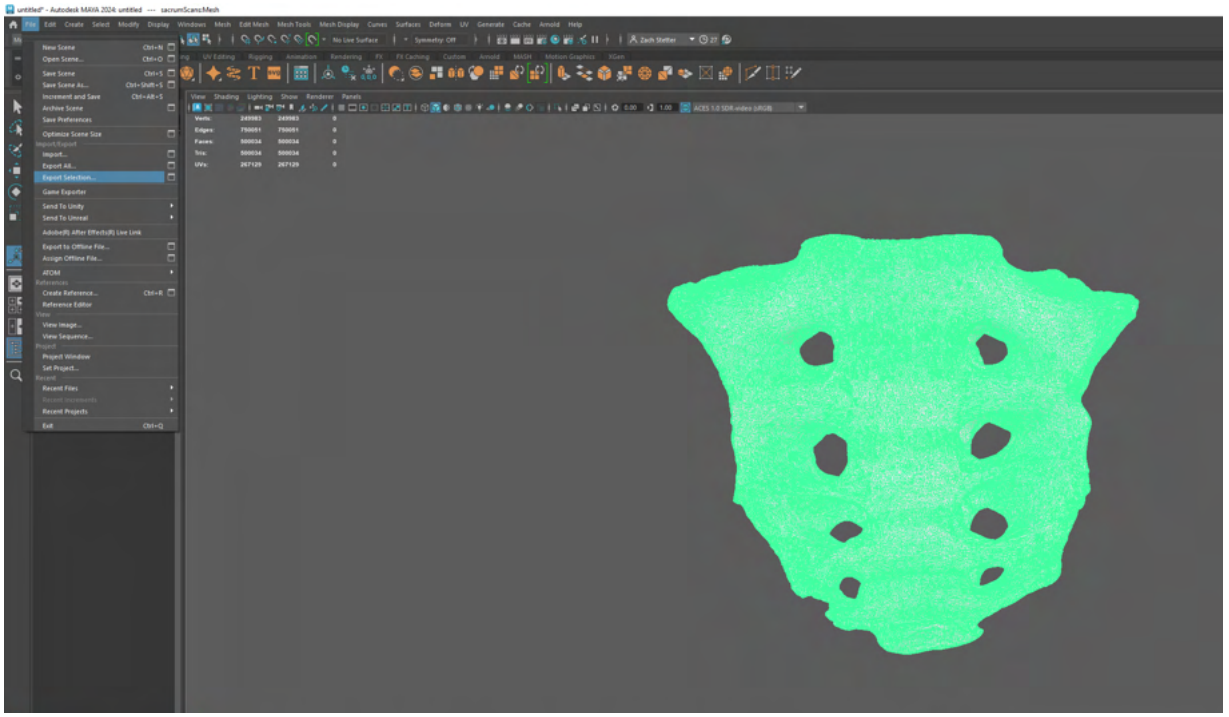
Once the computer has finished calculating the lines of the model will turn white instead of green. Alongside that, an information box will pop up informing you of how the model was reduced and by what percentage. Unless you're aiming for a specific polycount, reducing by percentage is the most reliable method. If you need to reduce the polycount further you can either increase the percentage that the polycount is reduced, or put the model through another Reduce cycle.



Mesh Cleanup

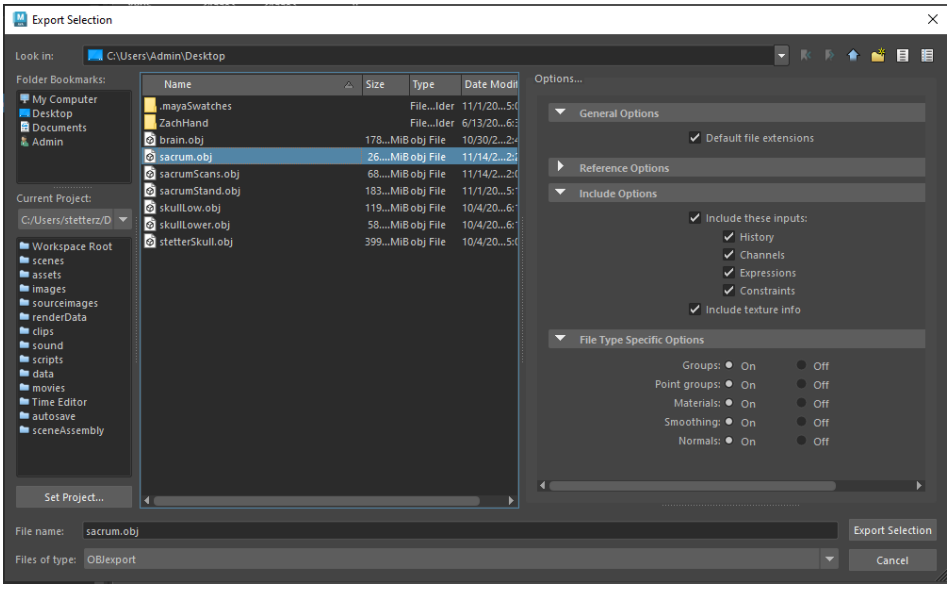
Alongside reducing your polycount, it is also a good idea to cleanup your mesh if a program is having a hard time importing an .obj file. Luckily, Maya also has an easy tool for this. It is also a good idea to clean up a mesh after reducing its polycount.

Underneath your 'Mesh' tab along the top of the program, move down to 'Cleanup' and click on the small box on the right side of the button. This will open up another window with a variety of options. Make sure that 'Faces with more than 4 sides', 'Faces with holes', 'Non-planar faces', and 'Nonmanifold geometry' are selected, then hit either cleanup or apply (both buttons perform the same action, one simply closes the popup window). Once again, make sure your mesh is selected before using the tool!



Export Your Mesh

Congratulations! You've cleaned up your mesh in Maya. Now all you need to do is export it for printing. First, select your mesh within the viewport. This will insure that you don't also export any cameras or other objects that may be within the Maya scene. Next, go to File -> Export Selection.

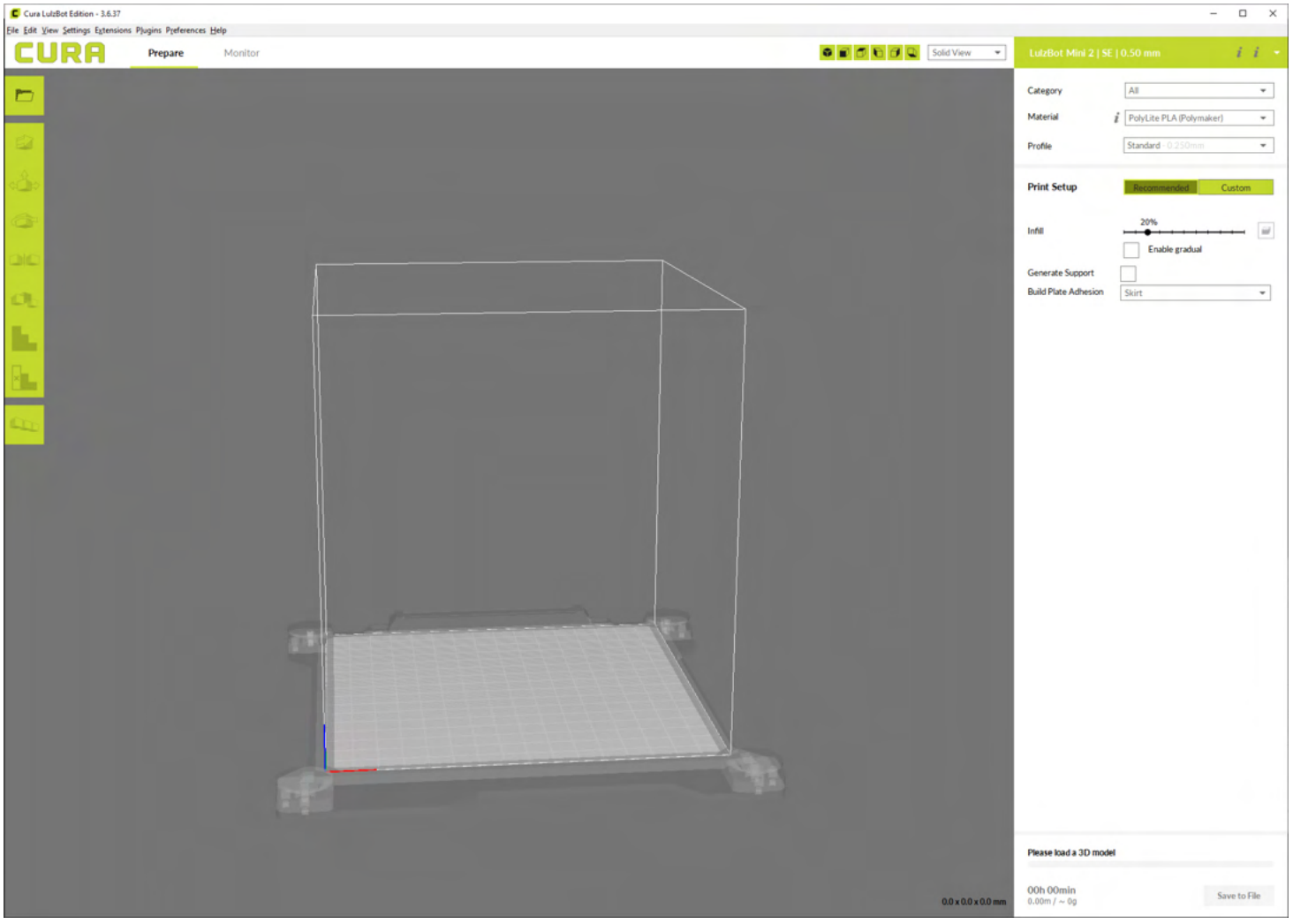


Make sure that the file type being selected is OBJexport, then navigate to where you want the file saved and click 'Export Selection'.

Printing with the Lulzbot

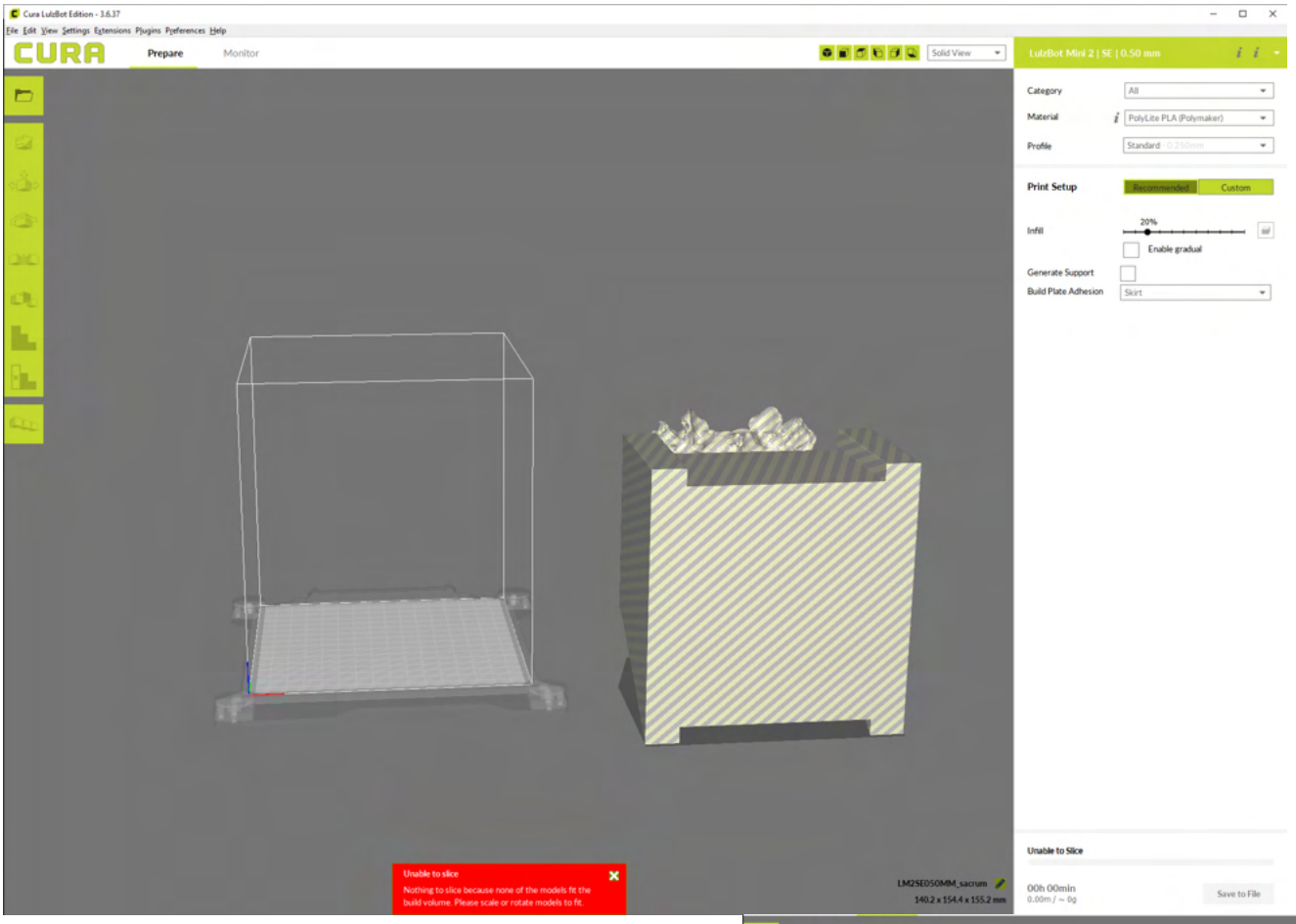
The Lulzbot printer is the biggest of the MHA printers with a large printer bed. However, it also provides the least amount of detail of the printers and is slower than the Prusa printer. While you can still achieve a high level of detail with this printer, it is best used for prototyping or roughly printing models to get a sense of scale, or testing how pieces fit together.

The program used to set up prints is called Cura, and prints can be sent directly to the printer from the program.



Importing your Model

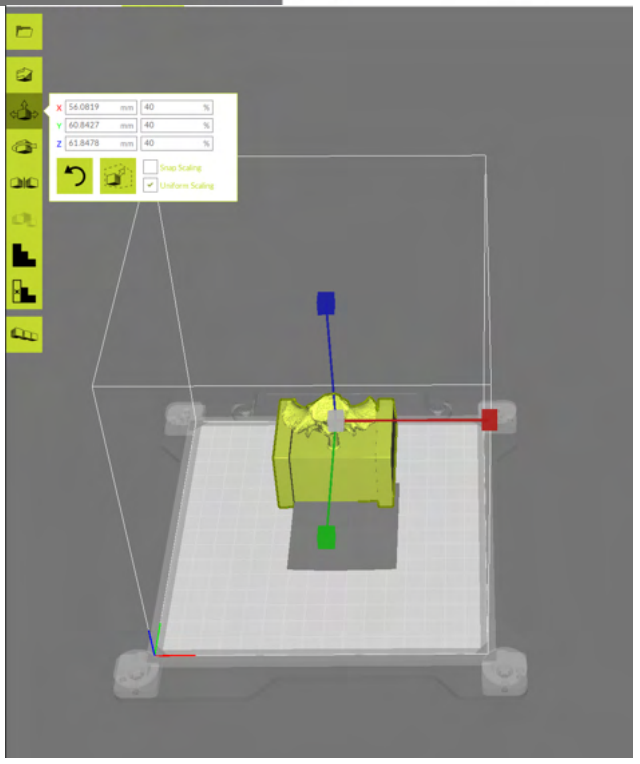
You can bring your .obj file into the program either through File -> Import, or you can simply drag and drop the model's file into the program's workspace.



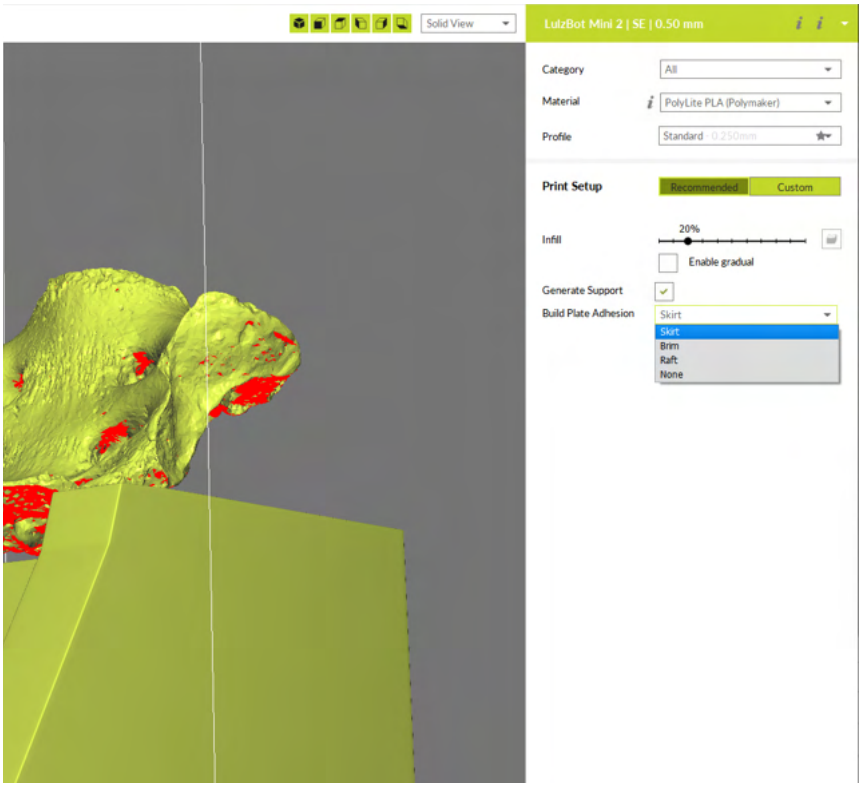
Prepping your Model

Upon importing your model it will probably not be aligned the exact way you want it on the build plate. On the left side of your workspace there is a green bar that has a Move, Scale, and Rotate tool to help you position your model to where it needs to be. After being rotated so that the base would fit upon the build plate, this model was also scaled down to 40% of its original size.

Move
Scale
Rotate
Mirror



Prepping your Print

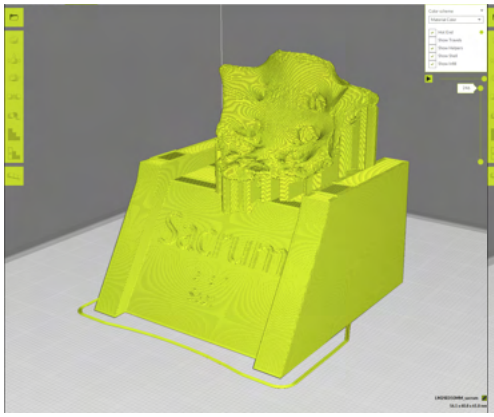


When preparing your print, there are a few things to consider on the right hand toolbar. First, make sure that you select the correct material you are using from the list, as this will affect the temperatures that the material is heated to. While you can adjust details of your print setup in the 'Custom' tab, the 'Recommended' tab will help ensure consistent quality prints.

Infill describes the printed structures within your print that keep it strong and supported. 0% infill will create a hollow print, and 100% will create a completely solid one at the cost of more filament being used. With most prints it is recommended to go between 15% and 25% infill, working in 5% increments.

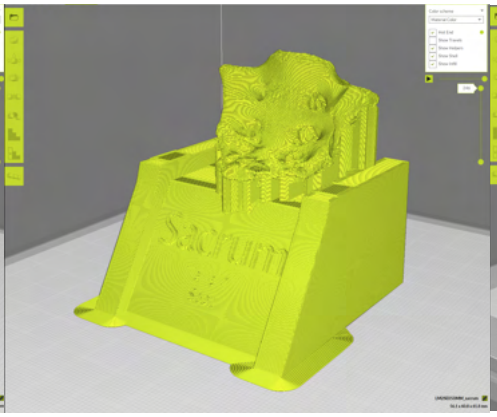
Make sure you have 'Generate Support' checked on, as it will automatically create support structures for your print.

With your Build Plate Adhesion, click one of the options from the list depending on how secured you want it to the plate, as described below.



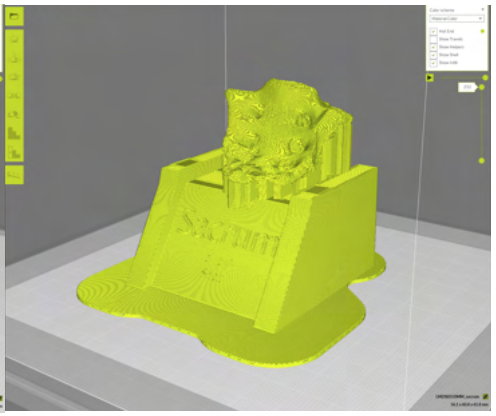
Skirt

The skirt is the thinnest form of plate adhesion, drawing a line or two around your printed model.



Brim

The brim more firmly attaches your model to the buildplate with a thicker line of printed filament.

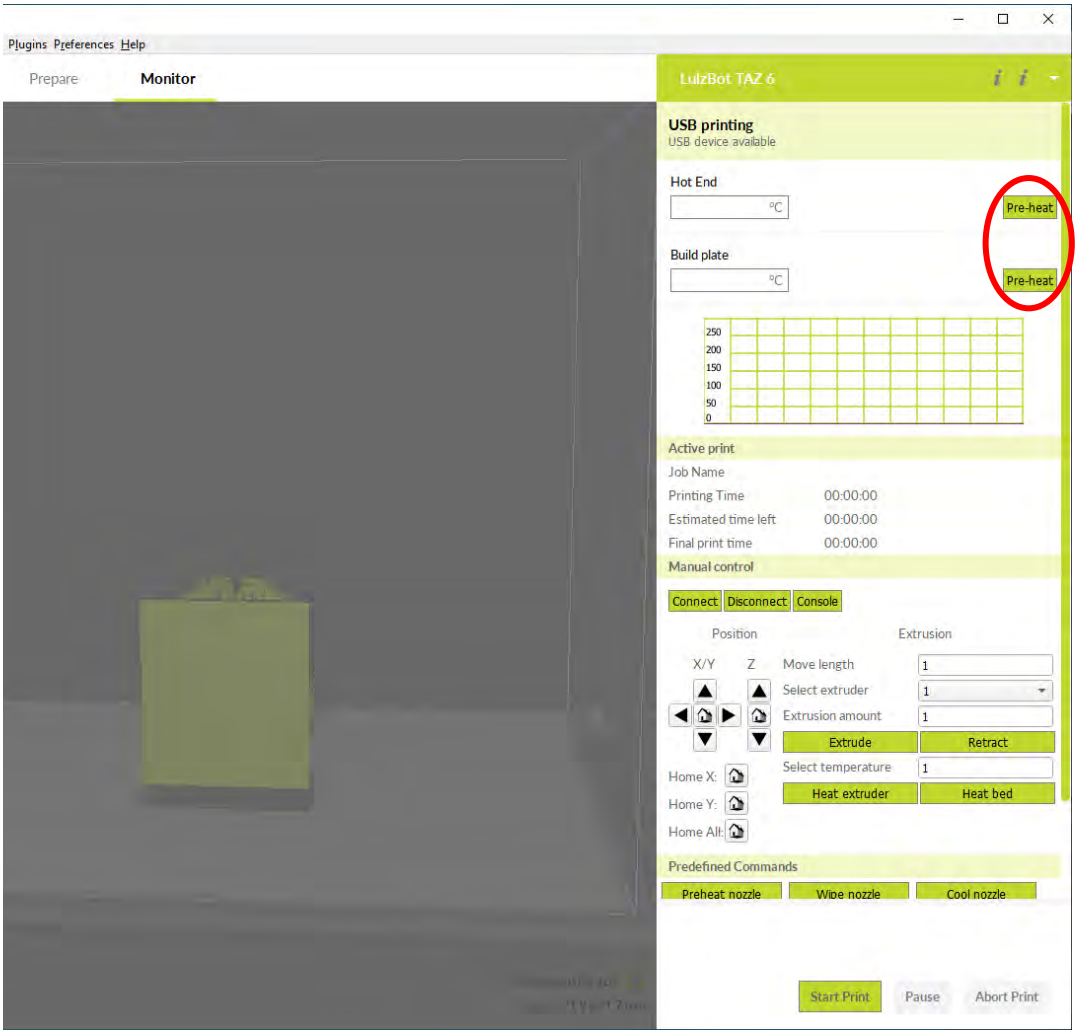


Raft

The raft creates a thin platform of printed filament for your print to sit on as it is being printed.

Printing your Model

Once you are ready to print, navigate to the top of your workspace and switch from the 'Prepare' tab to the 'Monitor' tab.



While not needed, pre-heating your build plate and hot end will decrease the time it takes to start your print. Alongside that, if your final print is having a hard time sticking to the build plate it is a good idea to pre-heat it in order to remove it more easily without any damage to either the build plate or the model.

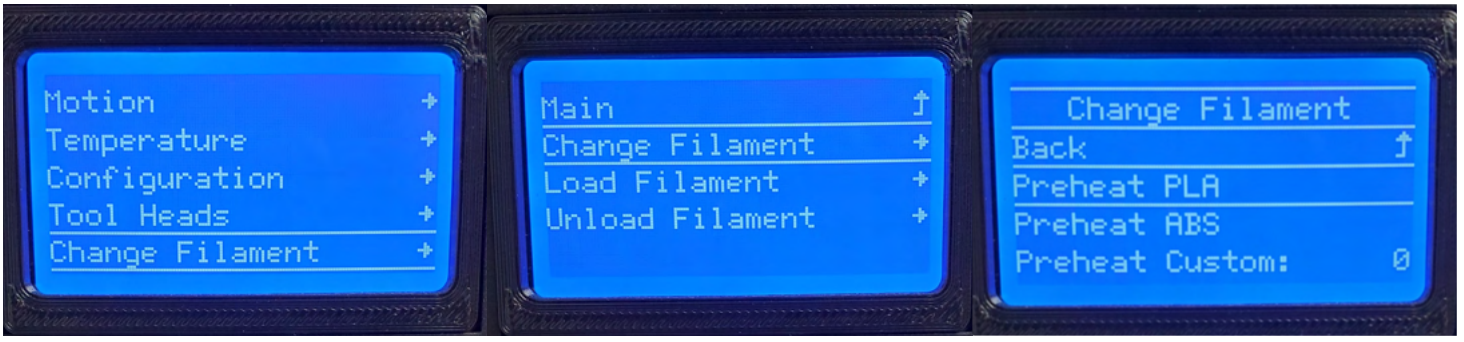
Once you've double checked all of your settings, click on the 'Start Print' button at the bottom of the tab. As long as the printer is plugged into the computer, all of the data should be sent over. It is generally a good idea to check back in 15 to 20 minutes to make sure that everything started and is on track.

Changing Lulzbot Filament

Either when you've run out of filament or you wish to use a different kind, you'll need to change the filament going into the printer. Luckily, much of this process is automated for you.

Make sure that when you insert filament into the Lulzbot printer you are using the correct size. Lulzbot uses 2.85mm filament.

Use this dial to navigate through your menus. Twist the dial to scroll through options, and then press it in to 'click' on them. Press it in once to bring up your menu options.



Scroll down to 'Change Filament'. Press in to select.

'Change Filament' will let you swap out one filament for another if one is already inserted.

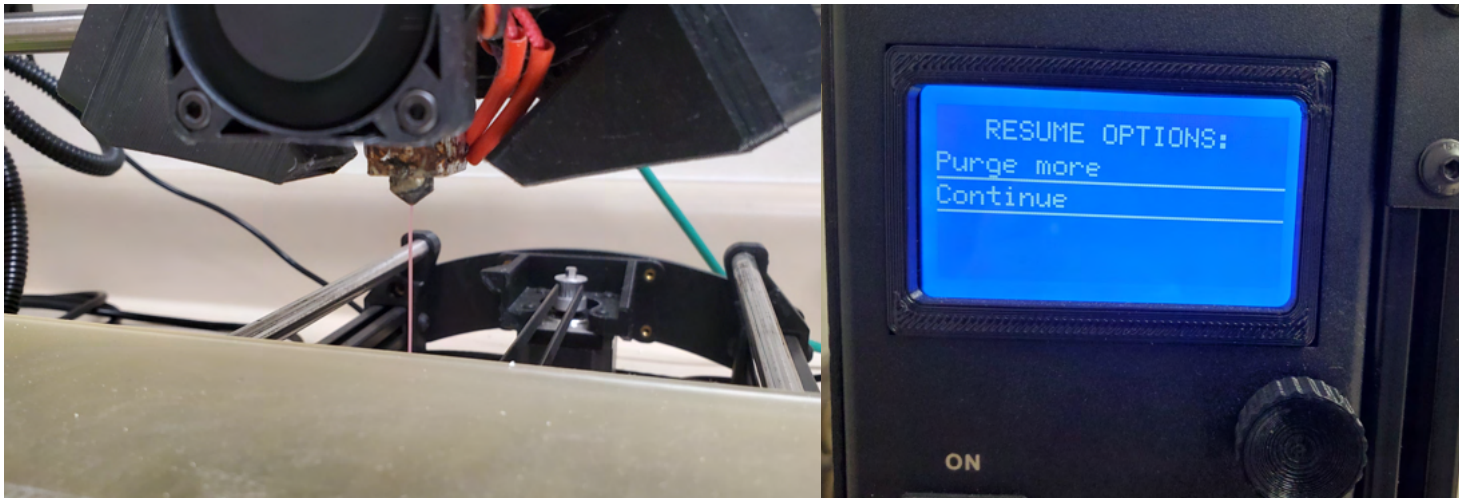
'Load Filament' is for loading when there is no current filament.

'Unload Filament' unloads whatever filament is loaded.

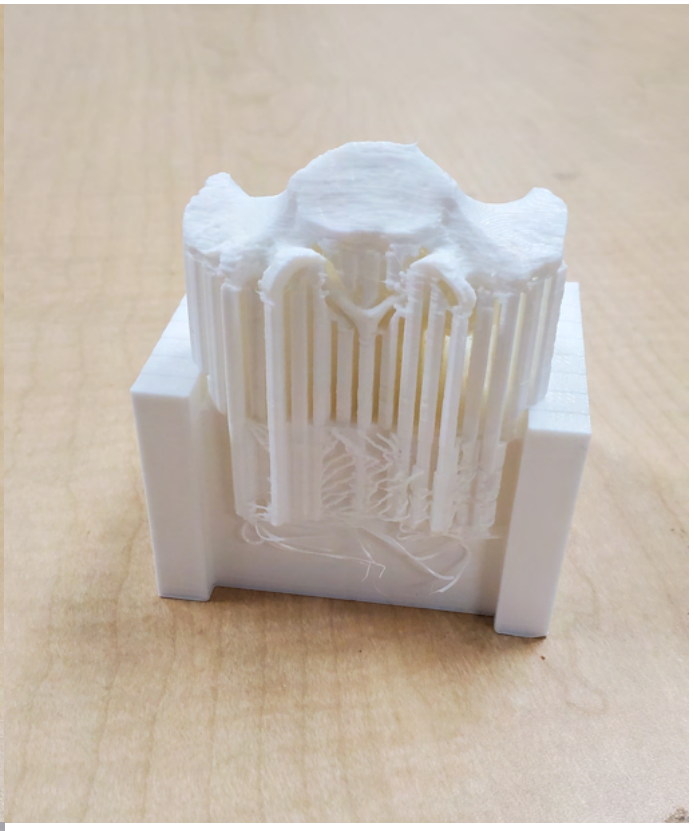
Use the wheel to select what kind of filament will be used so it will heat to the correct temperature.

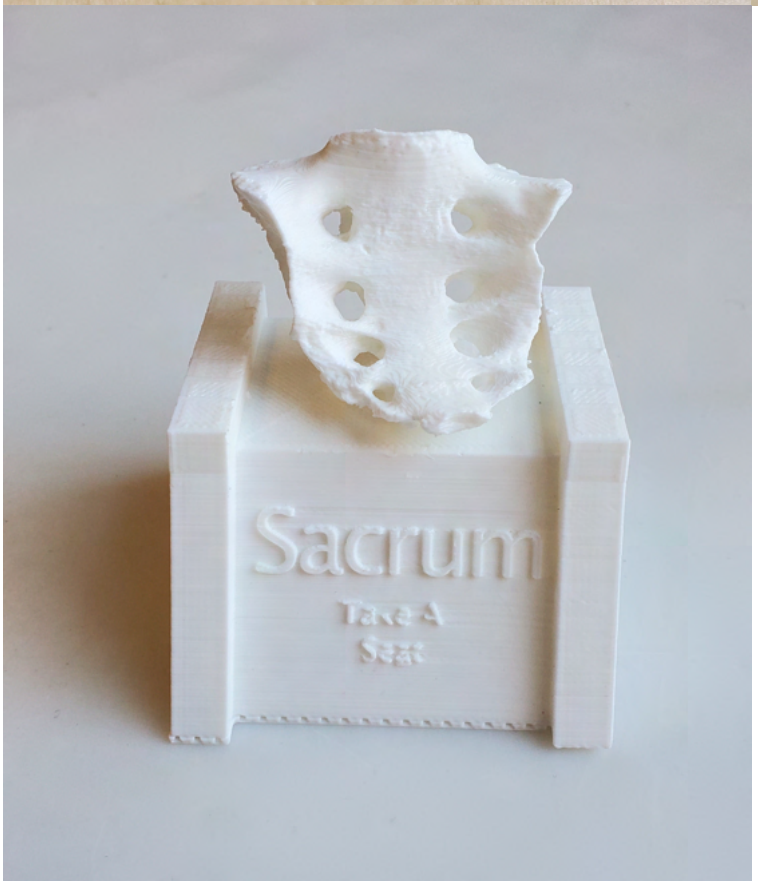
Once the nozzle is preheated, the printer will automatically begin to release the filament currently inserted within the nozzle. When it is ready for you to pull the heated filament out it will beep, and the screen will indicate that it is ready for a filament change. Remove the unwanted filament and replace it with what you are planning on using instead.

Feed your new filament into the top of the nozzle, making sure that it is pressed all the way down to the bottom of the feeder. You should feel a slight resistance as you push on the filament. Click the button on the dial and the printer should finish feeding the filament the rest of the way into the nozzle.



If you inserted the filament correctly, you should see a steady stream of filament coming from the nozzle as the printer purges the old filament to be replaced by the new. If the filament is coming out the correct color, you can click continue on the screen. If the filament isn't coming out of the nozzle or if it is coming out the wrong color, click on 'Purge More' until the correct color is flowing out of the nozzle below.

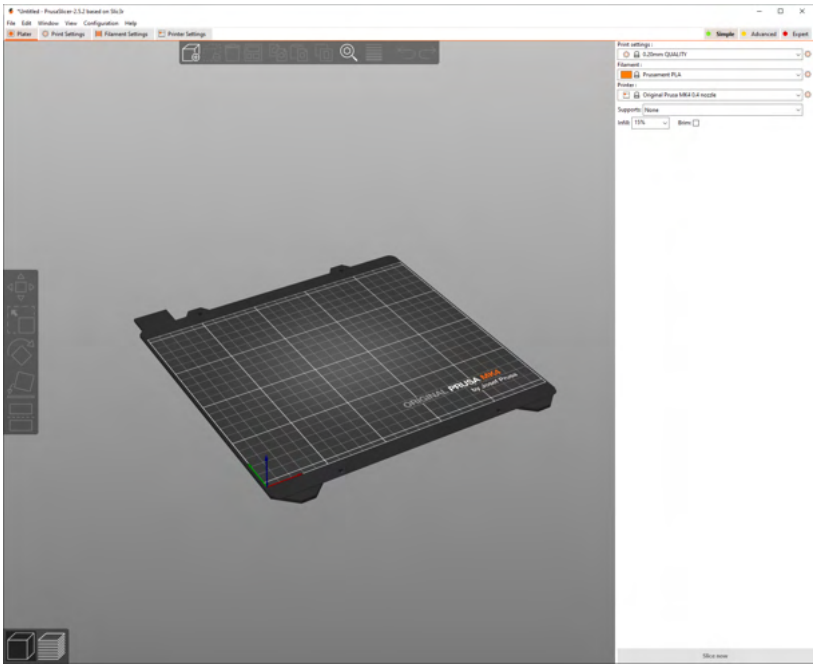




Congratulations!

You've completed a print with the Lulzbot! Once you have it separated from the printing bed, you'll need to pull off the supports that were created during printing. This is best done with a set of pliers and/or a small flush cutter. Make sure that you are wearing eye protection while doing this, as sometimes supports can fly off when under pressure.

The Lulzbot is a powerful printer, but its strengths lie in its large printing area. It is very good for creating rough prototypes of 3D prints as it uses fairly cheap filament to print but doesn't achieve the same level of detail that other printers can. This can be seen in some of the lettering on the front of the created stand.

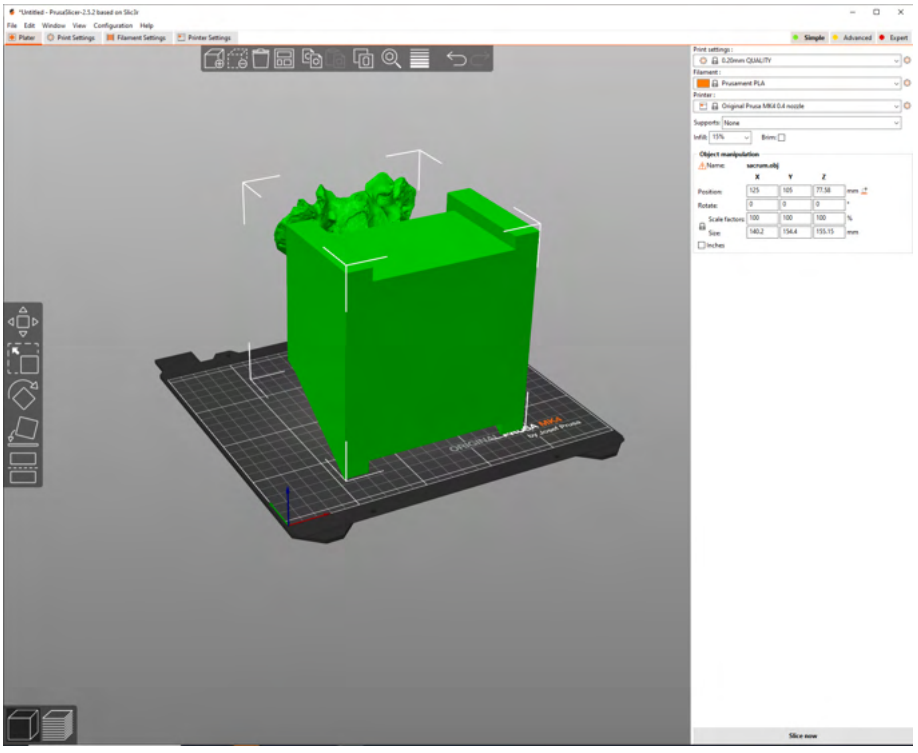


Printing with the Prusa

While it uses filament like the Lulzbot printer, the Prusa is a relatively faster printer with higher attention to detail. It uses a thinner filament (1.75mm) and has a removable metal printing bed that will flex to allow for the easy removal of prints. However, it does have a smaller printing bed than the Lulzbot and any larger prints may need to be printed in pieces before being fit together.

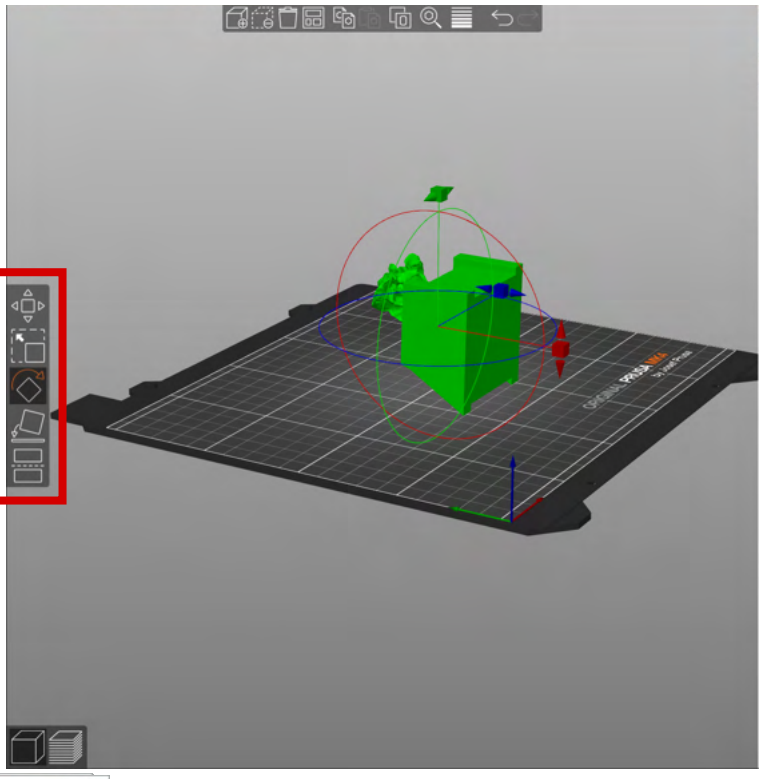
Importing your Model

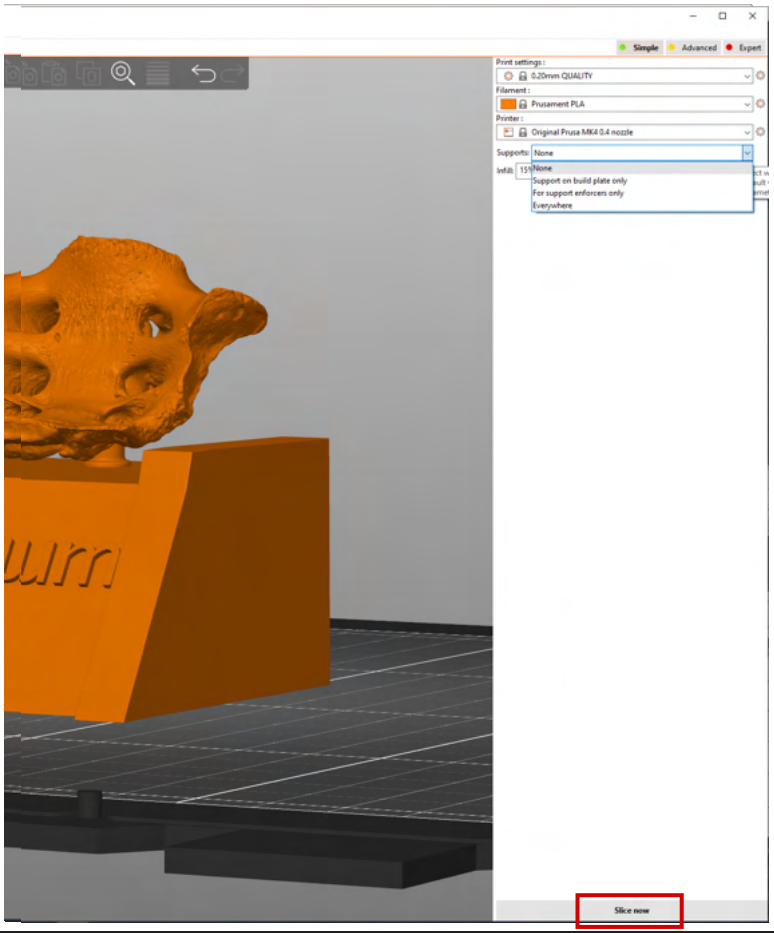
You can bring your .obj file into the program either through File -> Import, or you can simply drag and drop the model's file into the program's workspace.



Prepping your Model

Upon importing your model it will probably not be aligned the exact way you want it on the build plate. On the left side of your workspace there is a gray bar that has a Move, Scale, and Rotate tool to help you position your model to where it needs to be. After being rotated so that the base would fit upon the build plate, this model was also scaled down to 40% of its original size.





In the right hand tab, there are a few different print settings that can be edited to change how your object will print. Underneath the print settings tab, you can choose to either have your object print faster (Draft settings) or with finer detail (Quality Settings). Considering the speed of the printer, quality settings are often the best choice overall.

Under the filament tab, make sure that you select the correct type of filament that you are using so that the printer heats itself to the correct temperatures.

The default settings of the printer and nozzle should be correct when printing from the MHA printer.

Under the Supports drop down, you can chose where supports are printed on your object. Everywhere is the recommended option as it will automatically generate supports wherever is needed on your model. If you select 'Support on build plate only', no supports will be printed on your model which may cause issues if there are overhangs that don't directly overlap with the build plate.

Infill describes the printed structures within your print that keep it strong and supported. 0% infill will create a hollow print, and 100% will create a completely solid one at the cost of more filament being used. With most prints it is recommended to go between 15% and 25% infill, working in 5% increments.

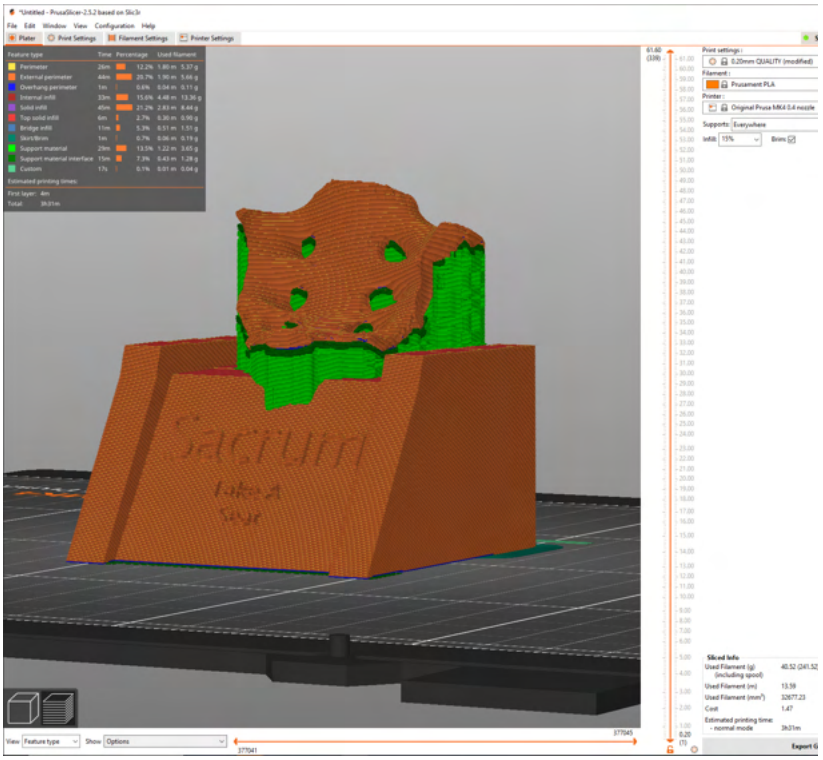
Once you are satisfied with your settings, click "Slice Now"

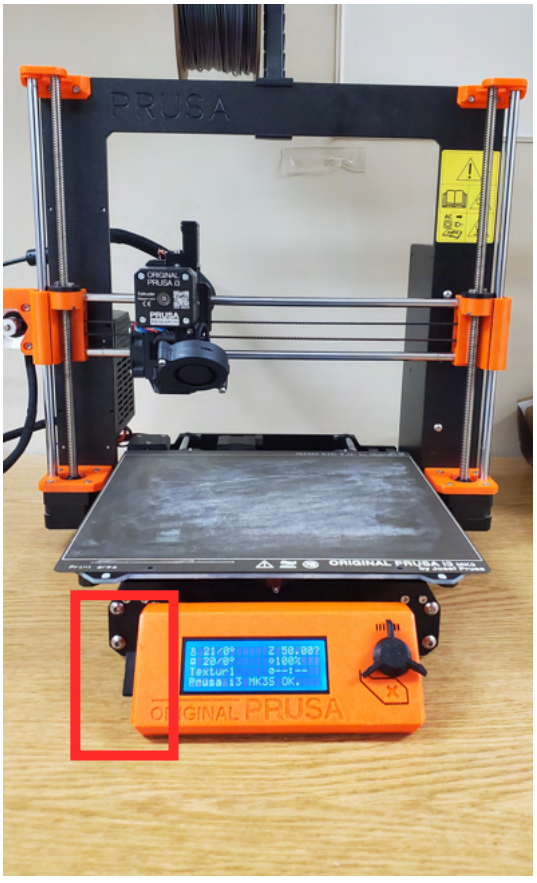
Prepping your Print

After you click 'Slice Now', your workspace will change to show you the layers of your print alongside the supports that will be added during the print. A gray box in the upper left corner will detail the amount of filament that will be used for the print, including where it will be used and how long each section will take. An estimate for total print time will be at the bottom of the square.

If you need to edit the position of your print or change any other settings, click on the cube icon in the bottom left corner of the workspace. Once you've made the edits you need, re-slice before saving out the G-Code.

If you're satisfied with how everything looks, export the G-Code and get it on the printer!

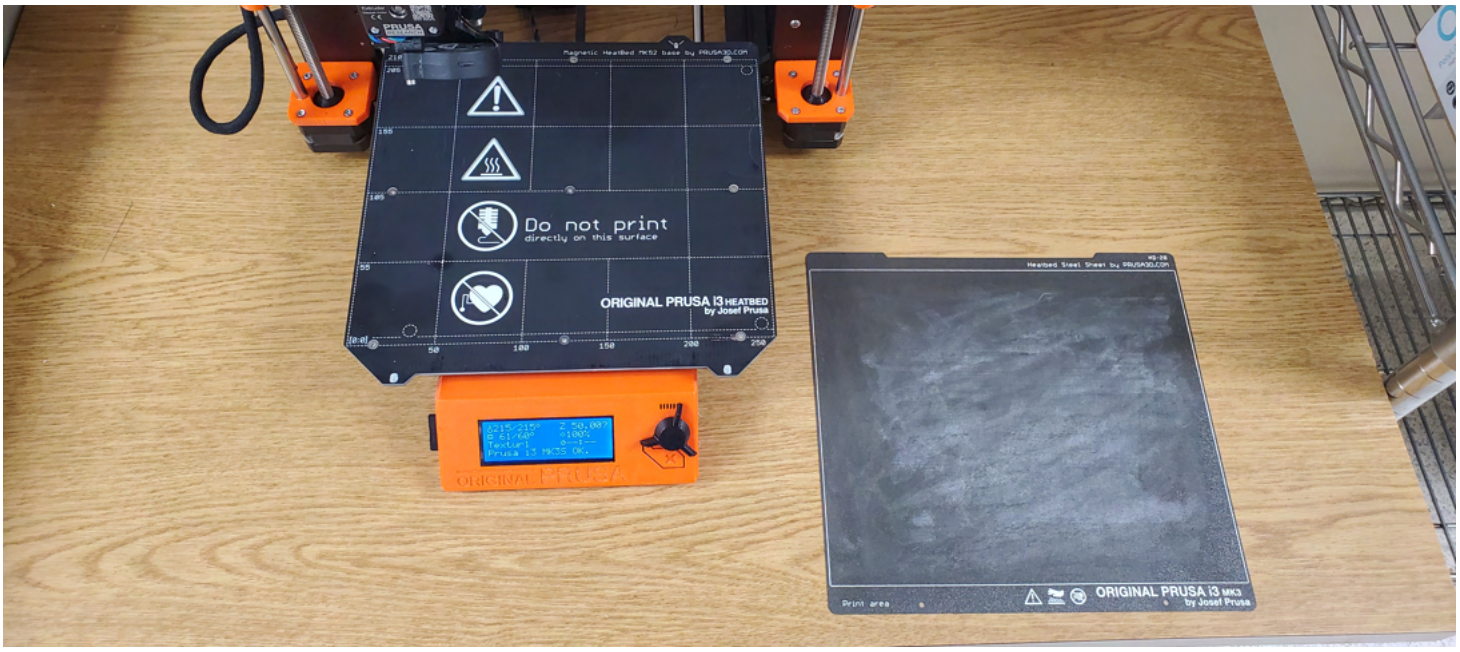




Transfer your Print to the Prusa

On the left hand of the Prusa monitor (shown in the red box), there is a slot for a SD card. In order to give the printer the data for printing, you'll need to save your G-Code to this SD card.

Remove the SD card from the Prusa printer and insert into the SD slot on the front of the computer tower near the power button. From here, you can directly save the G-Code that you export from PrusaSlicer onto the SD card. Eject the SD card from the computer and replace it in the slot of the Prusa printer. From here, you can select the print you saved out by clicking down on the selection wheel and the printer will take it from there.



Changing Filament in the Prusa

Sometimes you will need to either change the filament within the Prusa or add new filament if the roll currently being used has run out.
Make sure that whenever you're printing or changing filament you have the printing sheet on top of the heat bed. It is magnetic and removable, which allows for prints to be peeled off much easier, but you never want to get filament stuck to the heatbed itself.

Navigating the Menus

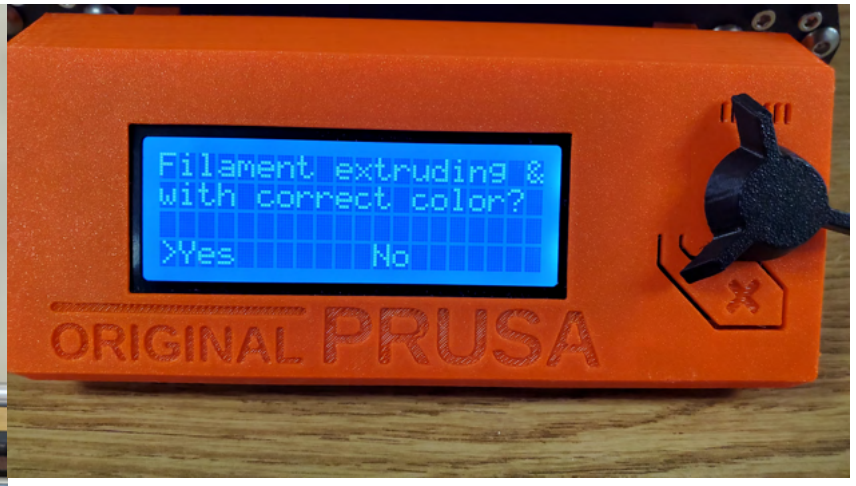
To bring up the menu options for the Prusa, click once on the rotating wheel. Then you can rotate the wheel itself to highlight what you want done with the printer before clicking again to select.

Scroll down to 'Unload Filament' and select it, then select what kind of filament is being unloaded/loaded.

Changing the Filament

Once you've made your selection, the printer will heat the nozzle to allow for the removal of filament. The machine will beep when the correct temperature has been reached, and prompt you to unload the filament from the nozzle.

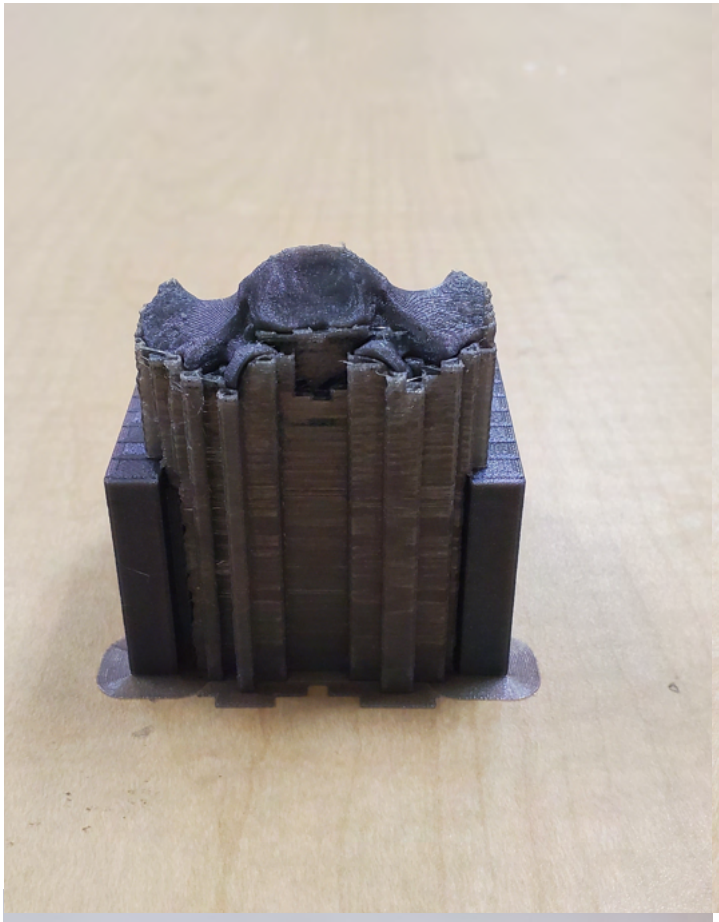
From here, you can load the new filament into the preheated nozzle, sliding the new filament into the space at the top. Press down firmly on the filament to make sure it is as deep as it needs to be, then click the button again to tell the printer to start to load the new filament.



Purge old Filament

To make sure that your filament is properly loaded, look at the color of the filament being pushed out of the nozzle. If it is either not the right color or not extruding filament, scroll the selection to 'No' and continue to purge until the correct color is being pushed from the nozzle.

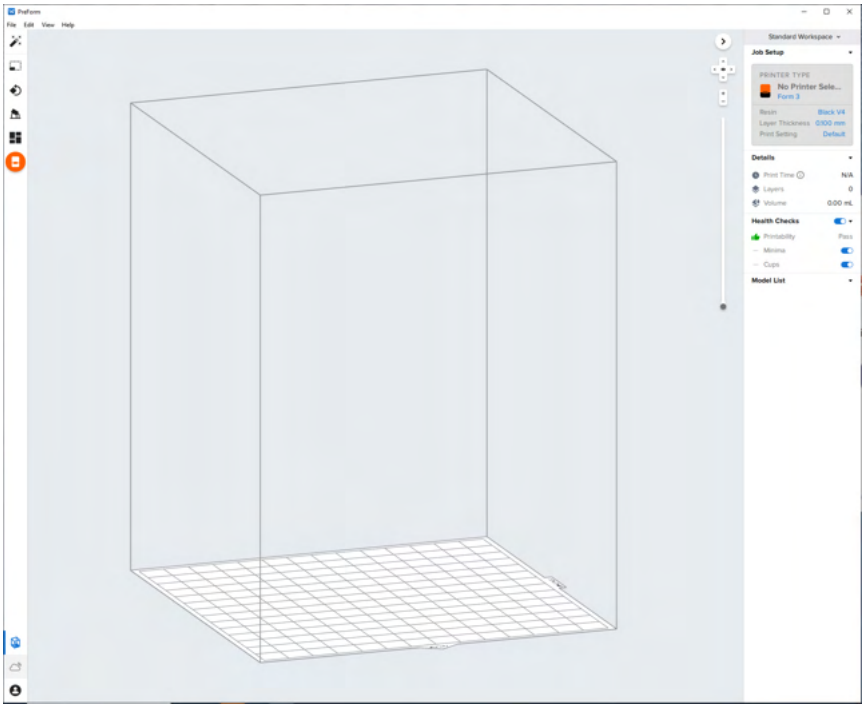
Once you have the correct color coming from the nozzle, select 'Yes' and the printer is ready to print.



Congratulations!

You've completed a print with the Prusa! Once you have it separated from the printing bed, you'll need to pull off the supports that were created during printing. This is best done with a set of pliers and/or a small flush cutter. Make sure that you are wearing eye protection while doing this, as sometimes supports can fly off when under pressure.

The Prusa's strength lies in its ability to print quickly and with a high level of detail. At this size, the printer still struggled with creating text outside of the title of 'Sacrum', but you can still see more legibility than the Lulzbot. When printing with the Prusa, it is a good idea to keep the size limitations of the print bed in mind and be prepared to glue together multiple prints if need be.



Printing with the Formlabs

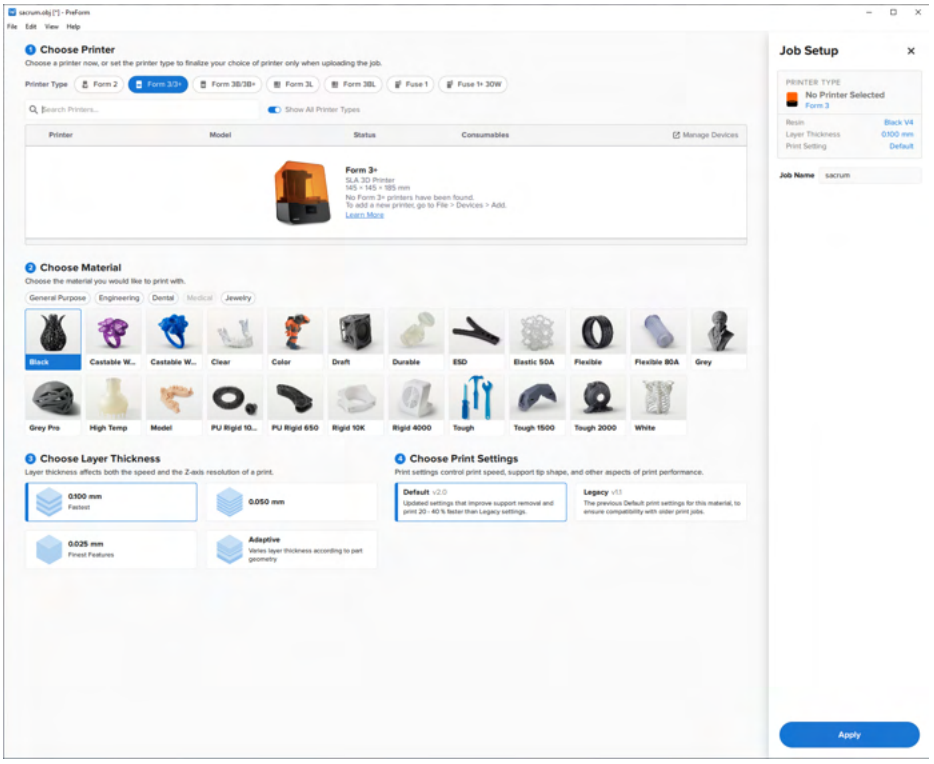
The Formlabs printer is unique in how it uses resin to print from a pool rather than extruding filament onto a printing bed. Because of this, the cleanup process is much different to that of the Prusa or Lulzbot. However, the resin surface usually requires much less smoothing and can print in higher detail than the previous printers. The major drawback of this printer is its price, as resin is a much more expensive printing material than filament.

When working with the Formlabs printer it is a good practice to always wear gloves and a mask before the final print is cured. Resin is toxic, and you don't want any of it to get onto your hands.

Job Setup

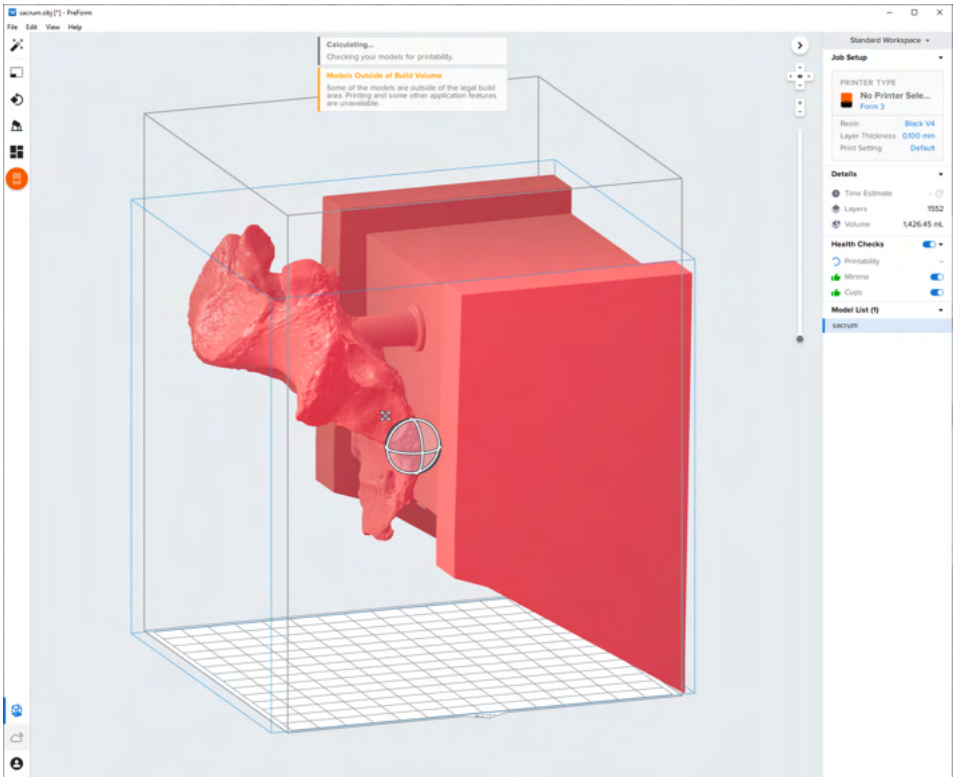
The program used to setup prints and send them to the Formlabs printer is called Preform. Before importing objects and adjusting them to the printing bed, you must make sure that the proper printer is selected and confirm what resin is being used to print the piece. Different types of resins allow for different support structures to be made.

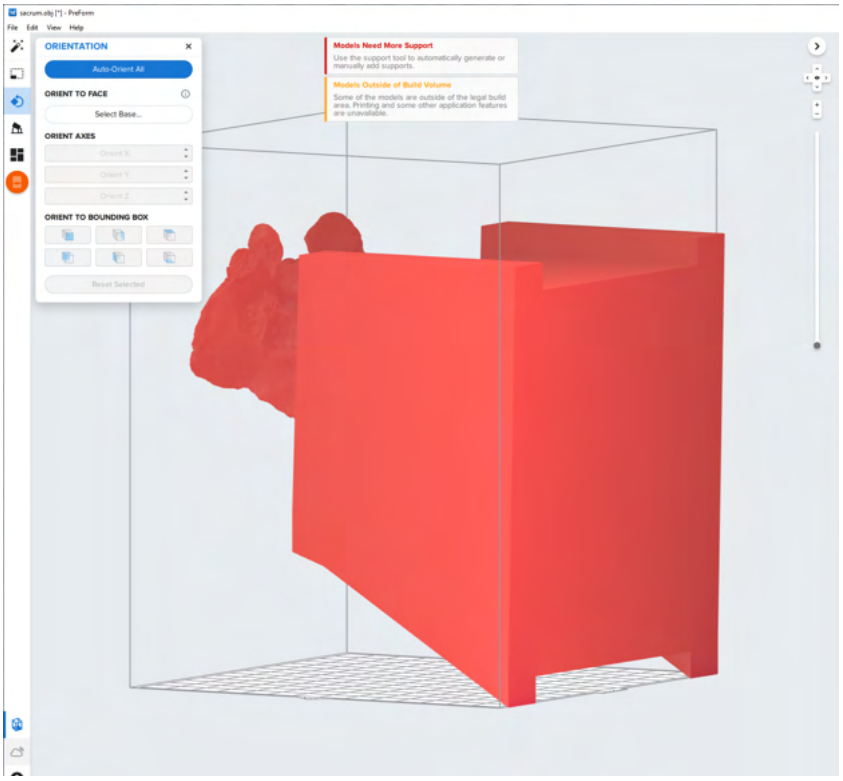
Navigate to the right tab and click on the 'Printer Type' bar. This should open up the menu to allow you to select the resin being used and confirm that you are on the right printer.



Importing your Model

You can bring your .obj file into the program either through File -> Import, or you can simply drag and drop the model's file into the program's workspace.

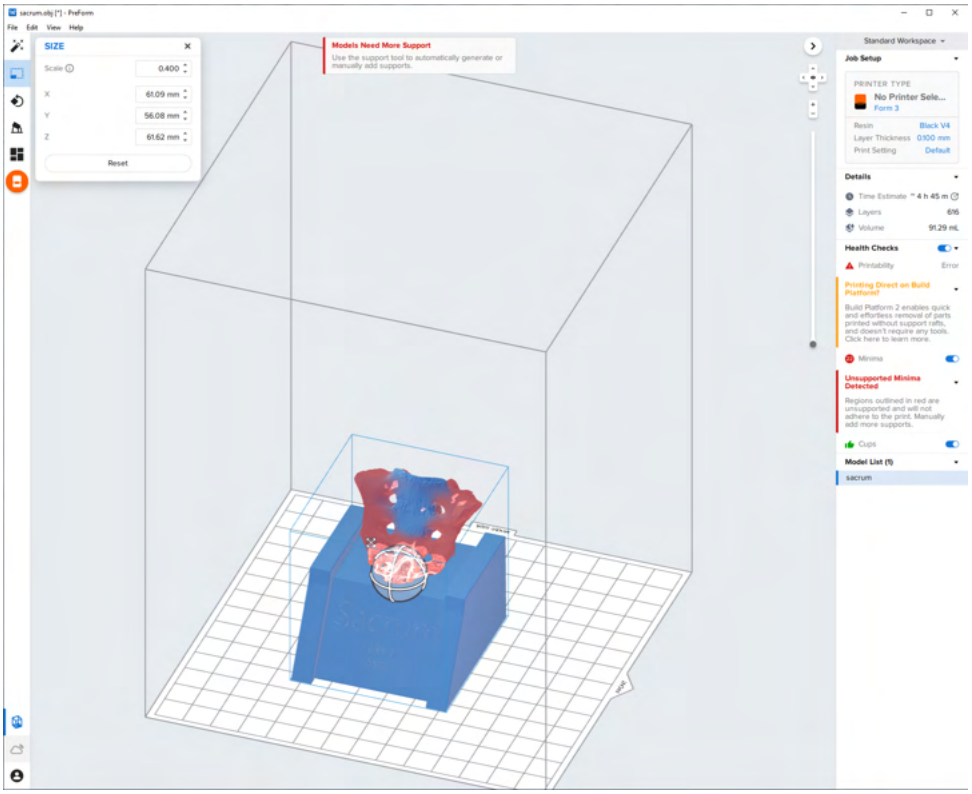




Prepping your Model

Upon importing your model it will probably not be aligned the exact way you want it on the build plate. On the left side of your workspace there is a gray bar that has a Move, Scale, and Rotate tool to help you position your model to where it needs to be.

In the rotate tool of Preform, you have the option to 'Auto-Orient All' or 'Orient to Face'. If you select the first, it will automatically rotate objects on the build plate for you. However, if you choose 'Select Base' under 'Orient to Face', you can click on one of the faces of the model to lay flat against your printing bed. This is a quick option to help you orient your model without messing with rotate tools.



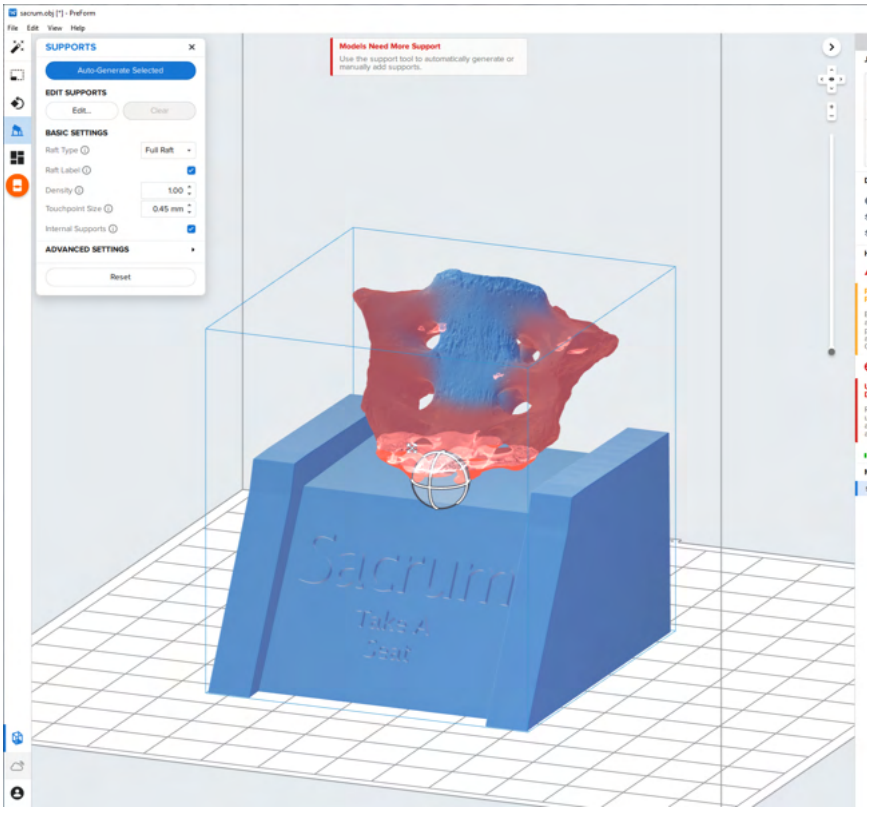
Scale your Model

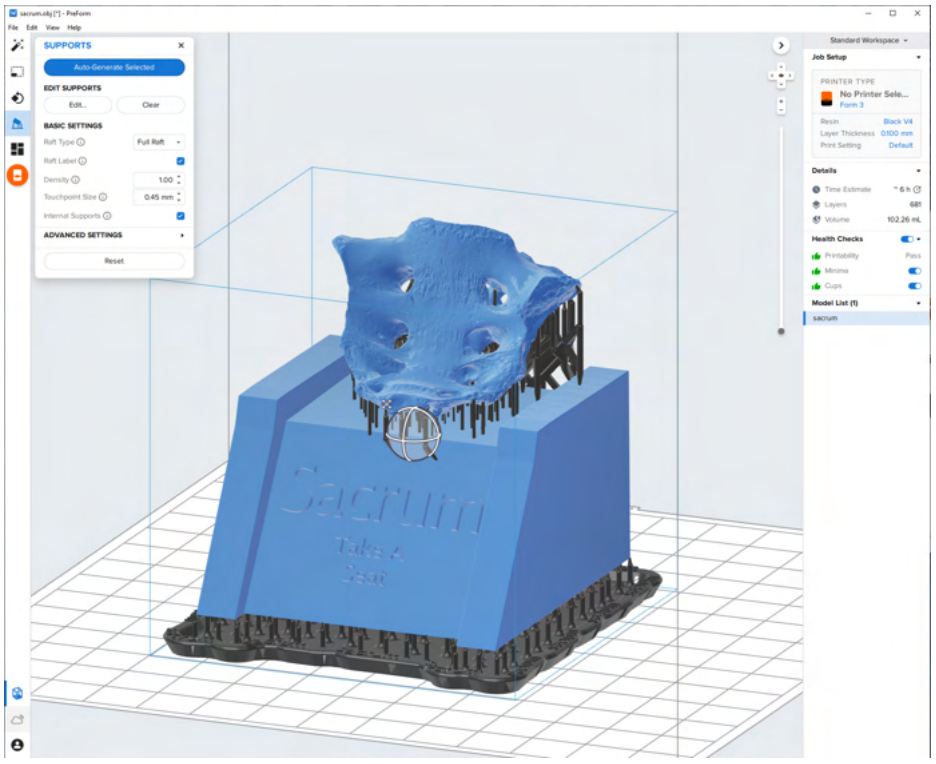
After being rotated so that the base would fit upon the build plate, this model was also scaled down to 40% of its original size. However, as can be seen with the warnings in the sidebar on the right hand side, the print is not yet ready and still needs supports.

Generating Supports

To create supports for the print, first navigate to the left hand tab and select the 'Supports' button. This will open a tab with a series of support settings.

By selecting the type of resin being used before importing our object, many of these support options are already optimized. Click on the 'Auto-Generate Selected' to have the program create a series of supports for the print.



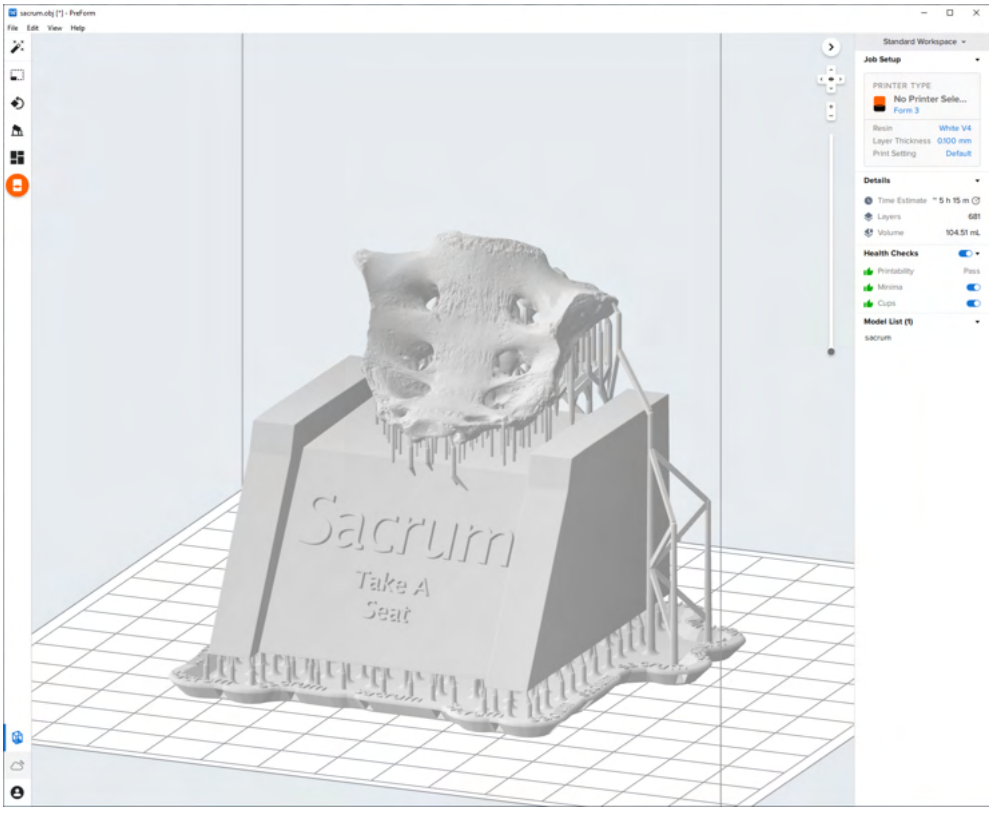


Double Checking Supports

Once the supports are generated for your print, you can rotate around to see where they've all been placed. If there are any in spots where you think they will not be needed, you can click 'Edit' within the supports tab to manually go in to change or delete them.

Sending to the Printer

Once you've double checked that all of your settings are correct and you're happy with how the print preview looks, it is ready to be sent to the printer. Make sure that the 'Health Checks' on the right hand tab are green, and once they are click on the orange circle icon on the left hand side of your workplace. A confirmation tab will pop up asking if you're ready to print, and as long as you're connected to the network that the printer is connected to the print will be sent on its way.



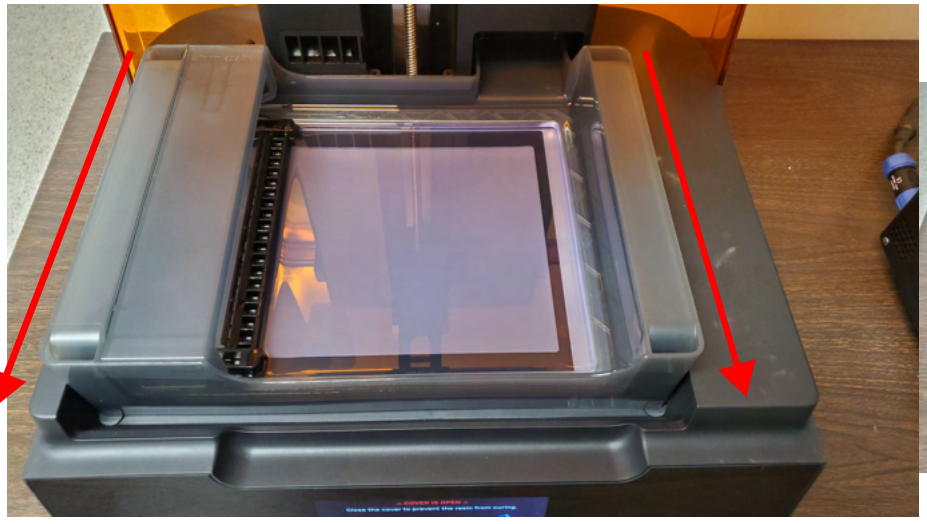
Changing Formlabs Resin

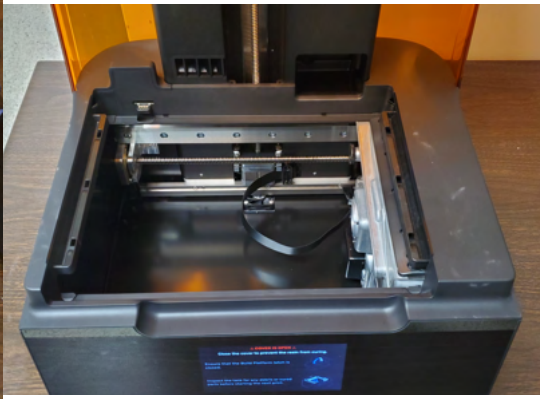
WARNING: When working with resin, ALWAYS wear gloves and a mask until the resin is properly cured.

While changing out the resin being printed is slightly easier than changing out filament, it can also be far more harmful to the machine when done improperly. When in doubt, ask a faculty member familiar with the machine for assistance.

Changing the resin for the Formlabs comes in 2 parts. First, the tank that sits on the base of the printer must be changed for each type of resin. The storage case where the tank is held should be labeled with the type of resin that is used with it.

To remove the tank currently on the printer, hold it by the handles and pull towards yourself (not up). This should release the tank. Put the current tank into its case, and replace it with the tank you want to use for your resin. It should click into place.





The second part of changing the resin has to do with the tank located on the back of the printer. Each tank can be cleanly and safely lifted from the back of the machine and placed away in storage. When not printing and when storing the tank, make sure that the pour nozzle located on the top of the tank is closed. When printing an object, press down on the tab to open the pour nozzle to allow the resin to flow into the printer properly.

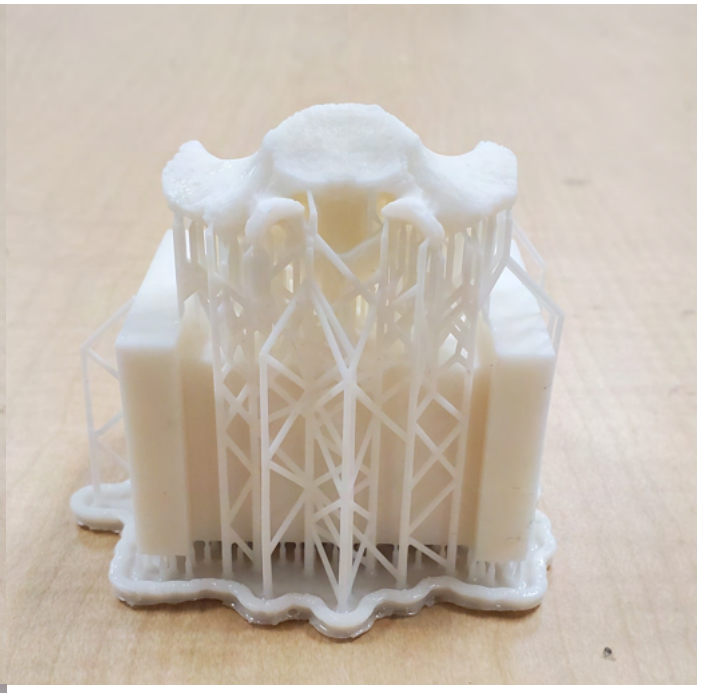
Curing and Cleaning your Print

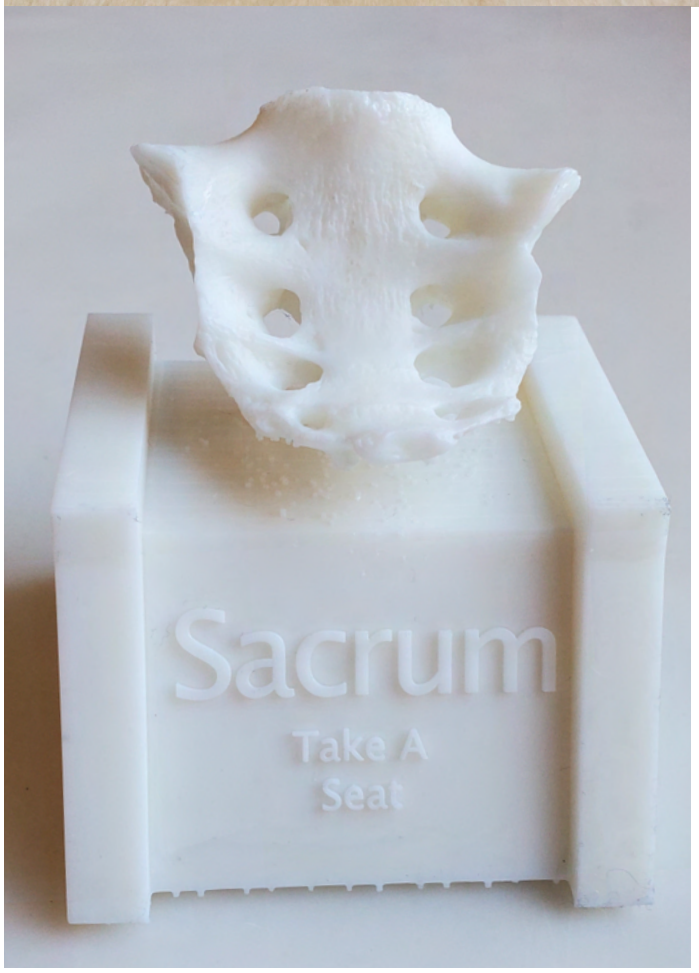
WARNING: When working with resin, ALWAYS wear gloves and a mask until the resin is properly cured.

Once your print is finished in the Formlabs, it still needs to be cured. Unlike filament in the Lulzbot and Prusa, resin prints require extra steps to make sure that they are safe to handle.

To begin, remove the print bed from the Formlabs by lifting the handle in the middle that holds it in place and sliding it towards you. Turn on the alcohol wash station by pressing its knob, and place the build plate in the slot that rises from the alcohol bath. Select a 30 minute wash minimum and let the object descend into the machine to be washed. It will automatically rise back out once it is done.

Once the alcohol bath is finished, remove the build plate and detach the print by flexing the metal ends. Make sure to do this over a soft surface like a box of towels so that the print doesn't fall onto the floor and break. With gloves on, take the print and place it into the UV curing box. Use the knob once more to select the type of cure needed for the print and the machine will set a timer. Sometimes the curing times vary per print, so check the surface of the print with a gloved finger before grabbing it. If the surface is still sticky or tacky place it back into the UV curing chamber.





Congratulations!

You've completed a print with the Formlabs! Once you have finished curing the model, you'll need to pull off the supports that were created during printing. This is best done with a set of pliers and/or a small flush cutter. Make sure that you are wearing eye protection while doing this, as sometimes supports can fly off when under pressure.

The Formlabs' strength lies in its ability to print in high level of detail with a very smooth finish and easy to remove supports. As can be seen, the printer was able to print even the small text legibly. When printing with the Formlabs, it is a good idea to keep the cost of the resin materials in mind, as well as the length post-print process. This printer is best used when you are sure you're ready for a final product.