

Quick Start Guide

for Pegasus Touch 3D Printers

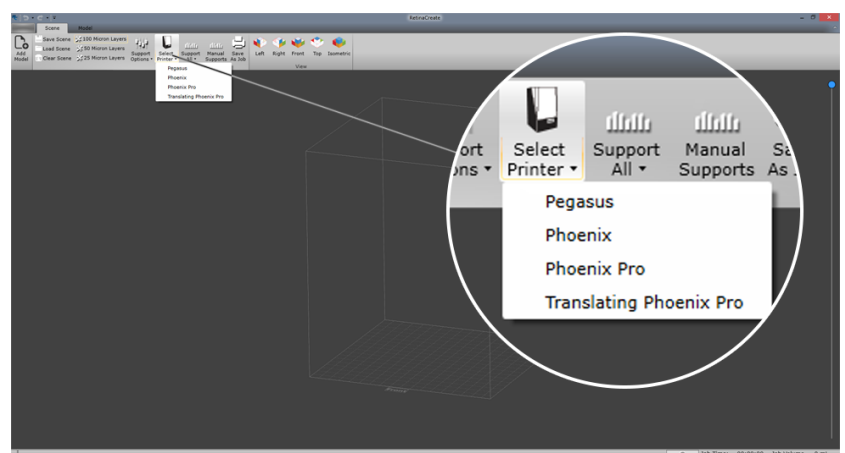
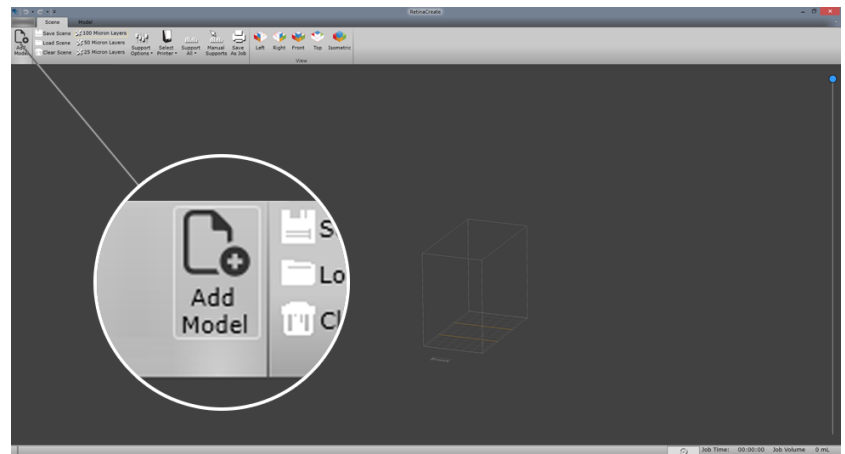
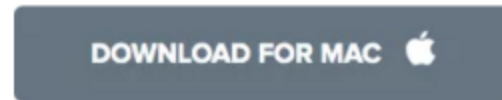
Carefully read and understand all of the contents before operating your printer. For troubleshooting and in depth instruction, please refer to the complete Pegasus Touch manual.

Take the Pegasus 3D printer out of the box and remove wrapping. Plug the printer's power supply cord into an available outlet and then into the power supply input on the printer. The input will be located on the left side of the printer.

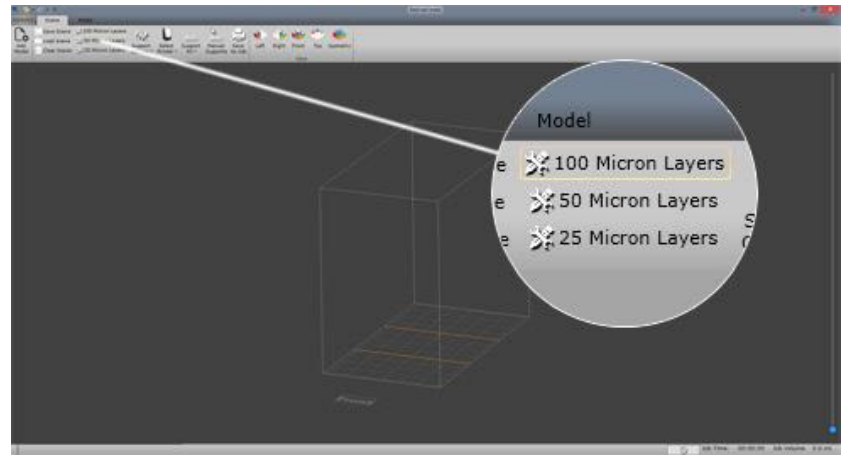
To set up a print job, download the *RetinaCreate* software onto a computer. It can be found here: <http://fslaser.com/Software/RetinaCreate>

The first step to prepare a 3D model for printing is to ensure it is in a ".stl" or ".obj" file format. It can then be dragged into the program or opened using the "Add Model" option in the top left corner of the screen.

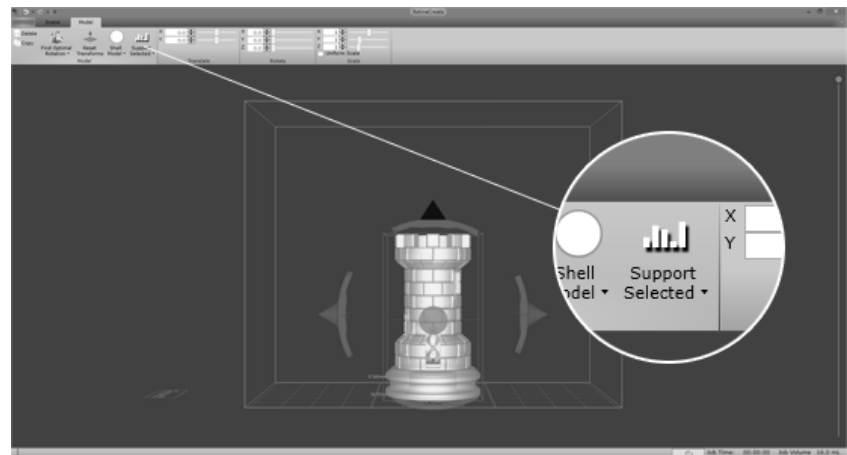
From here, a printer-type can be chosen by using the "Select Printer" button along the top toolbar. A dropbox will be displayed and the printer being used can be selected. In this case we are using the "Pegasus".



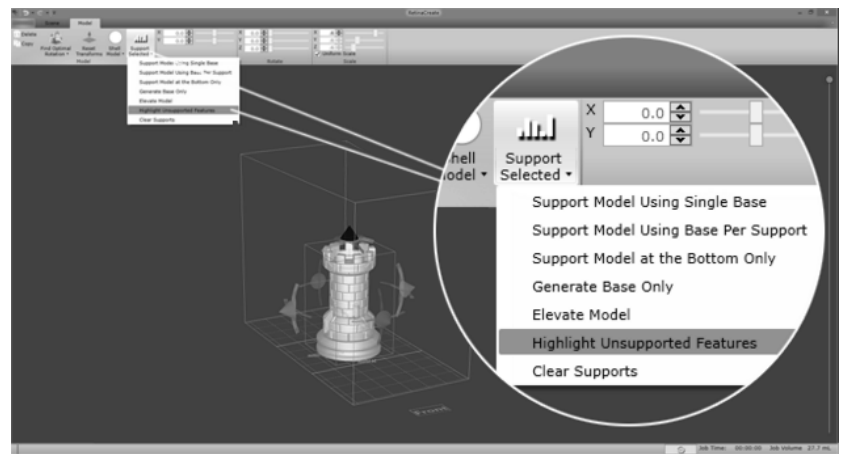
Once the correct printer has been selected, layer thickness can be chosen using the “100 Micron”, “50 Micron”, and “25 Micron” buttons along the top toolbar.



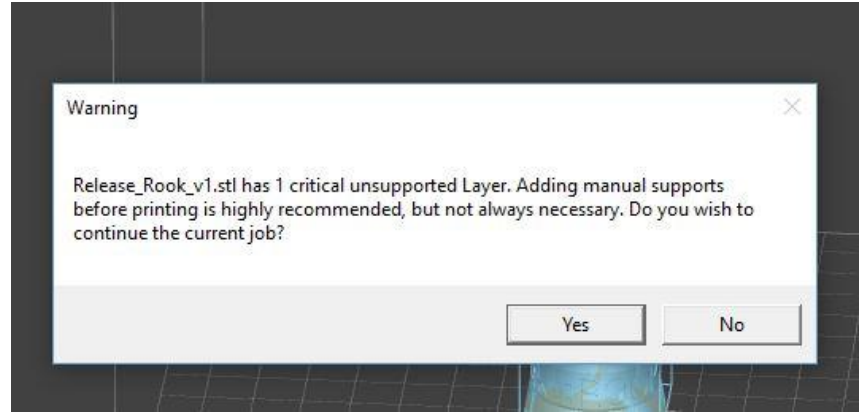
Next, the model(s) will need to be supported properly. In many cases the automatic supports generated by *RetinaCreate* will work in creating a successful print. To use automatic supports select the “Support Selected” button along the top toolbar and choose a support option from the drop box.



RetinaCreate offers a “Highlight Unsupported Features” tool under the “Support Selected” drop box on the toolbar that can be used as a guide for the manual support placement. To add manual supports select the “Manual Support” option and then place the support by moving it to the appropriate area and clicking.



RetinaCreate performs a final print analysis to check for any unsupported features. When this happens a dialog box will appear. If automatic supports are being used on the print it is safe to proceed with current print job. If manual supports have been used, it can be helpful to check that all overhanging areas are properly supported before continuing.



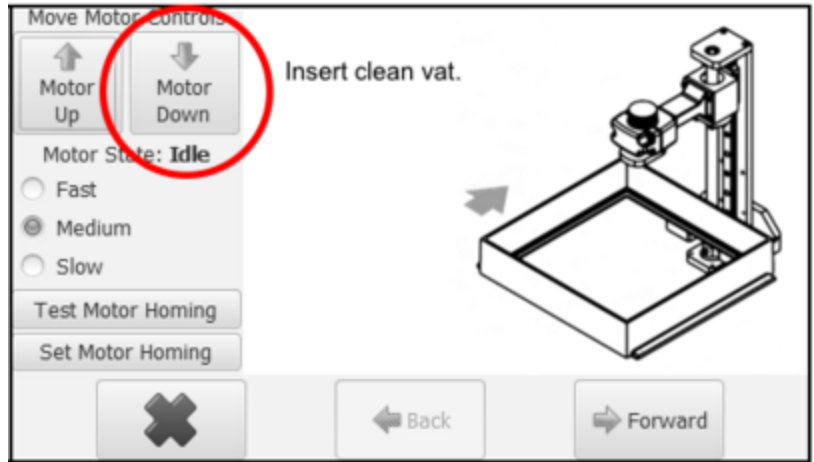
Before starting a print job the Pegasus must be leveled. It is best to level the printer before resin is placed in the vat. First, remove the shipping screws located at the front of the vat frame.

Then slide the vat into the vat frame and lightly tighten the four ball nose screws located at each corner of the vat using a flat head screwdriver. Make sure that the steps on the left and right side of the vat are facing outward.

Next loosen the thumbscrew on the build head and completely slide it onto the holder and hand tighten the thumb screw.

Add equally sized spacers to the four corners of the vat. Batteries, LEGOS, and building blocks all work well. The blocks will be used to measure the space between the vat and the build head to make sure they are parallel.

On the printer touch screen select the gear symbol in the top right corner and select the “Calibrate Motor” screen. Use the down arrow located in the top left corner to move the build head closer to the leveling blocks. Once near the blocks, manually lower the spine until the build head is touching the top of the spacers.

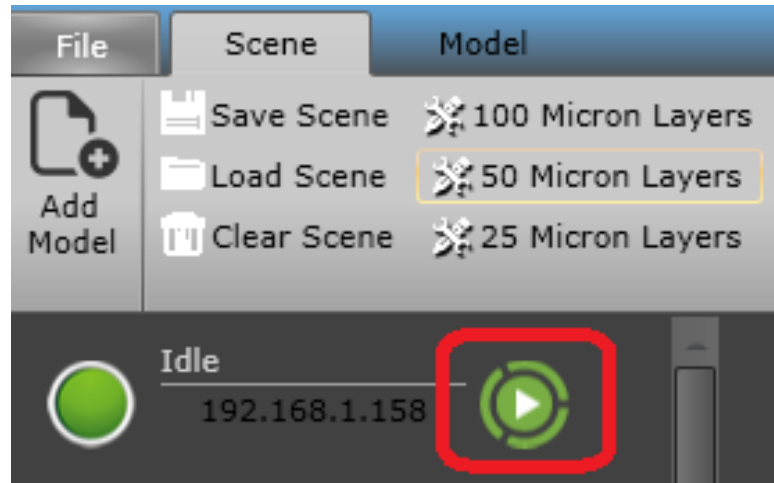


Tighten or loosen the hex screws around the four corners until the build head and spacers are flush, but do not tighten the hex screws all the way down. It is very important that all four corners of the vat are as level as possible. **Improper leveling will result in print failures.**

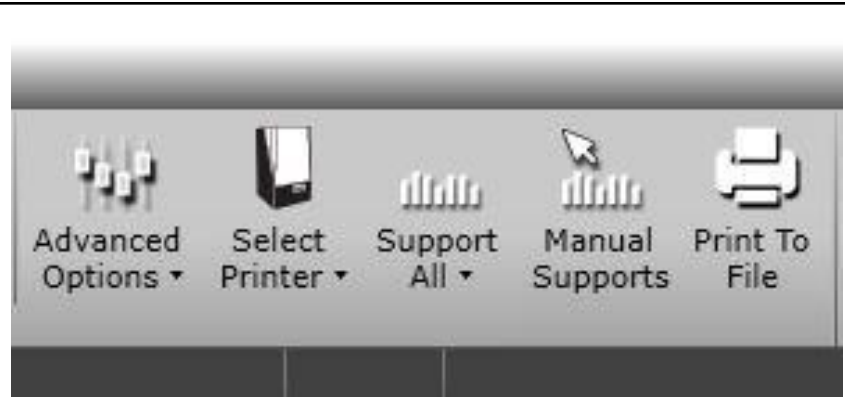
Once the vat is leveled, remove the spacers and lower the build head into the vat. Use the threads on the back of the machine to gently touch the build head to the vat. In the “Calibrate Motor” screen select “Set Motor Homing.” Once this is complete select the “Test Motor Homing” option to make sure the vat and build head are flush to each other.

Once this is completed, raise the build head out of the vat. Shake the resin well and pour the necessary amount into the vat. It is important not to fill the resin over the top of the steps located at the corners of the vat. Over filling the vat can cause resin to overflow and damage the machine.

There are two options for printing. If the printer is connected to the internet via ethernet, an icon with the printer IP address can be seen on the left side of the screen in *RetinaCreate*. By clicking the “play” icon to the right of the IP address, the job is sent directly to the printer.



In other cases, a USB flash drive can be used. Once the model is ready to be printed it must be saved as a “.fsljob2” file on the flash drive. To save the print as a “.fsljob2” file click on the “Print to File” Icon along the top toolbar.



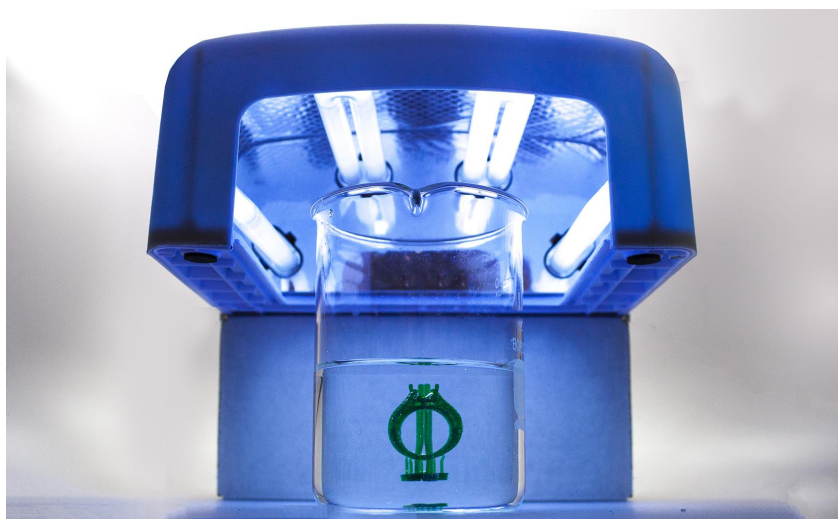
The USB flash drive can then be plugged directly into the side of the printer and a USB icon will show up on the home screen of the printer. Click this icon to browse through the flash drive files and select the desired file by clicking the printer icon next to it. If the file is not saved as a “.fsljob2” file it will not show up on the USB flash drive selection.

Once the print job is selected a screen will appear with options of “default settings” and “custom settings.” We recommend using the default settings options. Once the settings are selected a resin selection screen will be displayed. Use the arrows to scrolls to the appropriate resin being used for your print. One final screen will appear confirming your print and then the job can then be started.

After printing the model and removing it from the build head, it should be swirled vigorously in an IPA (isopropyl alcohol) bath for 2 minutes. Then it should be soaked in fresh IPA for 10 minutes followed by another vigorous swirl in another fresh batch of IPA for 2 minutes. Air must be blown on the model to fully dry the alcohol. (This can be done with a blow dryer). It's normal that the surface of models feels slightly tacky at this point.



If desired the model can be post-cured in Glycerin for 30 minutes in a curing station with a rotating platform. If the curing station doesn't rotate, the parts need to be cured for at least 60 minutes and the parts need to be flipped every 15 minutes. The more models there are to be cured, the longer the time necessary to fully cure the models.



After post-curing is done, swirl vigorously in an IPA (isopropyl alcohol) bath in a beaker or cup for 2 minutes, soak in fresh IPA for 10 minutes, swirl vigorously again in a fresh IPA bath for 2 minutes and finally blow dry the model. At this point, all surfaces of models should be cured. If any surface of the model still feels tacky or can be scratched with a fingernail, repeating the post-curing process is recommended.

