

Support and Print bigger 3D models



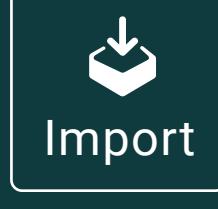
Congrats on the purchase of your new 3D model! Now it's time to make it come to life by printing it. And, of course, we're here to help you every step of the way!

Follow our guide to print easily with Lychee Slicer and get the best result!

1

IMPORT

Import your 3D model in Lychee Slicer (you can **drag and drop** your model).



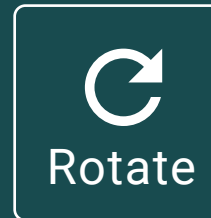
Move

Click on **"Move"** to adapt the position of your 3D model. Place the 3D model in the middle of your printer plate and move it vertically to lift it from the plate. Lift it enough so it doesn't touch the plate and leave room for the supports underneath.



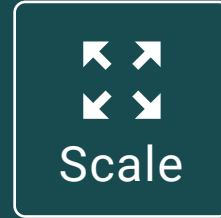
MOVE + ROTATE

2



Click on **"Rotate"** to incline your 3D model at a **45° angle**, with its **back facing the plate**. This way, the supports won't be on the face side of the 3D model. It will be easier to clean without leaving any support marks on the tiny details of your 3D model.

Click on **"Scale"** if you want to change the size of your model. You can increase or decrease the percentage to make it smaller or bigger.



Scale



BONUS STEP

This step is optional. You can save some resin by **hollowing** your 3D model and adding holes for drainage. By doing this, you may increase the risk of a **suction cup effect** and fail the whole print.



Hollow



Holes



This step can be long and complicated if you're a beginner. But thanks to our awesome feature: **the Suction Cup Detector**, you can avoid the suction cup effect and place holes easily in just a few clicks.

We strongly suggest you to watch our [YouTube tutorial](#) to learn more about this process and to be able to reproduce it.



SUPPORT

3



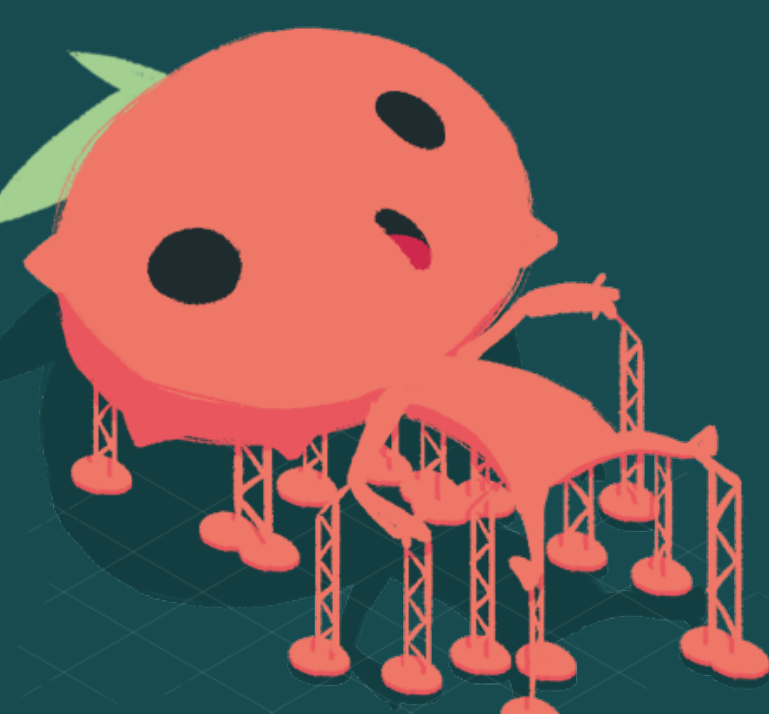
Support



Auto

Click on the **"Support" menu**, choose your support size between light, medium and heavy. Choose the support thickness depending on the size of your 3D model. The bigger the model, the thicker your supports should be. Click on the **"Auto" menu**, then **"Generate Automatic Supports"**.

Note: Automated supports can be modify at any time in the **"Manual" menu**.



4

RAFT: ADD A BASE TO YOUR SUPPORTS

Click on the **"Raft" menu**. Choose the **raft** that you want: it needs to be adapted to your model and to the size of your printer. We suggest you to add a raft so the 3D model is easier to remove from the printer plate once it's printed.



Raft



EXPORT

5



Export

Go to **"Export"**: click on **"Export slices to file"**. Once it's done, your file is ready to go on a USB key and uploaded on your 3D printer.

Still having failed prints? Don't worry, it takes time for everyone. But we're lucky to have an awesome community here to answer questions and help! You can also take a look at our tutorials on our social media.



Try Lychee Slicer here!

Want some friends?

YES, DO IT!

Join our [Discord Community](#)



FOLLOW US ON SOCIAL MEDIA



Discover more Creations & 3D Models

