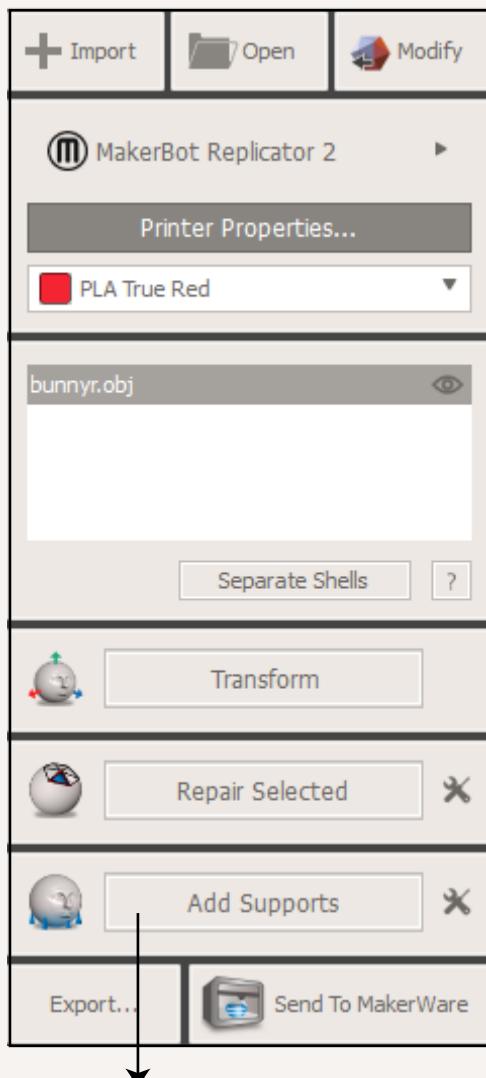


Printing models using your 3D printer



Use this 3D print preparation workspace to prepare your models for 3D printing on your 3D printer.

1. CHOOSE PRINTER

Start here by choosing your 3D printer

2. CHOOSE MATERIAL

Assign materials to your selected 3D printer

3. USE TRANSFORM TOOL

Use the Transform tool to move, rotate and scale your model within the print volume.

Use Move to Platform to place the model(s) on the platform.

Fit To Build Volume will ensure your selected model(s) fits within the printer's print volume.

4. USE REPAIR TOOL

Automatically heal your model for 3D printing with Repair Selected.

Click Advanced button  to view more features

Analyse to check your prints' Strength, Stability, and Slicing resolution.

Hollow your 3D model with a wall thickness by checking the box: Hollow with wall thickness.

Thicken parts of your model that are too small to print by checking the box: Thicken thin parts by, then entering a minimum feature size.

5. ADD SUPPORT STRUCTURE

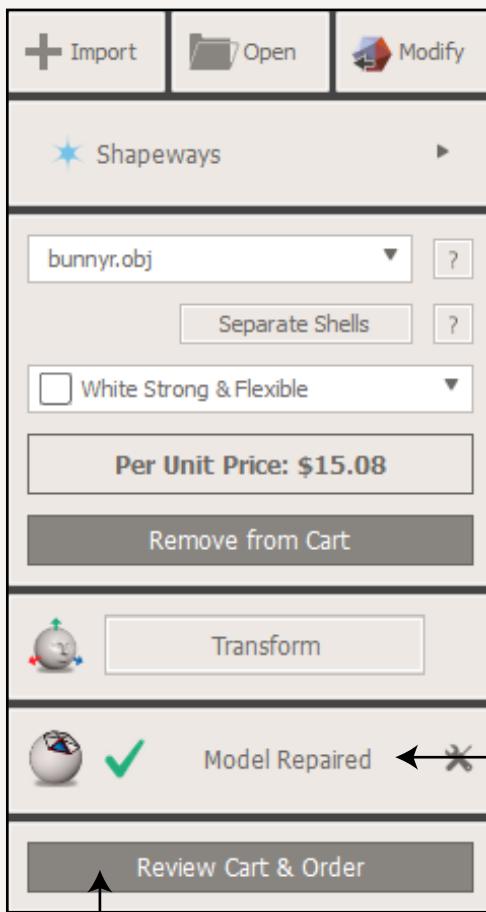
Automatically add Meshmixer's unique support structures, saving your printing time and materials

Click Advanced button  to view advanced properties

Manually add Meshmixer's unique support structures or finetune the properties for support structures for more precision

Finally, send your project directly to MakerWare or Makerbot 3D printer (Windows 8.1). Alternatively, export an STL, THING, or OBJZF for you to 3Dprint.

Printing 3D using online print services



Use this 3D print preparation workspace to prepare your models for 3D printing using online 3D Print Services

1. CHOOSE PRINTING SERVICES

- Start here by choosing your Printing Services.
(3D Hubs, Shapeways, imaterialise, Sculpteo)

2. CHOOSE MODEL FROM THE DROPODOWN

- All the models in scene automatically are sent to cart.
You need to choose each model to apply material and add further settings to make it printable.

3. CHOOSE MATERIAL

- Assign materials to the model and check the price per unit at the Printing Services immediately.

4. USE TRANSFORM TOOL

- Use the Transform tool to move, rotate and scale your model within the print volume.
Use Move to Platform to place the model(s) on the platform.
Fit To Build Volume will ensure your selected model(s) fits within the printer's print volume.

5. USE REPAIR TOOL

- Automatically heal your model for 3D printing with Repair Selected.

Click Advanced button  to view more features

Analyse to check your prints' Strength, Stability, and Slicing resolution.

Hollow your 3D model with a wall thickness by checking the box: Hollow with wall thickness.

Thicken parts of your model that are too small to print by checking the box: Thicken thin parts by, then entering a minimum feature size.

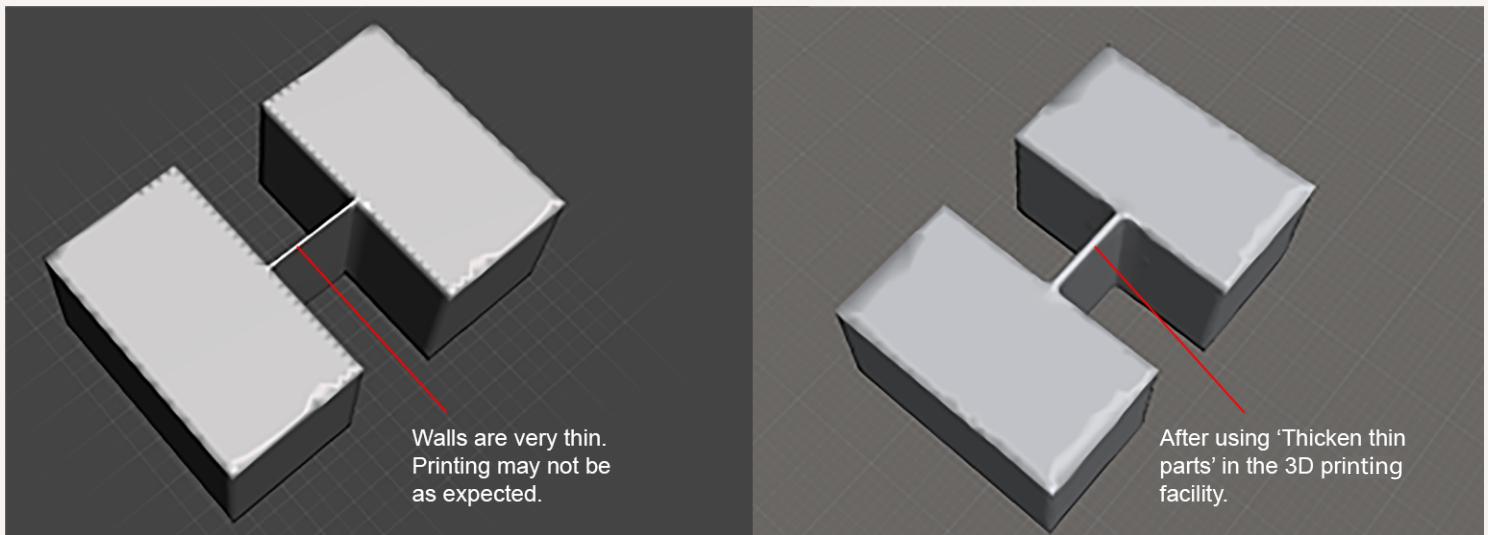
6. REVIEW CART AND ORDER

Click here to see all the models in your cart.
Models with errors are marked, so that you can either remove them or rework on them to make them printable.

Finally, you will see all the models from your Cart on the Printer Services website, from where it is very easy to order and checkout.

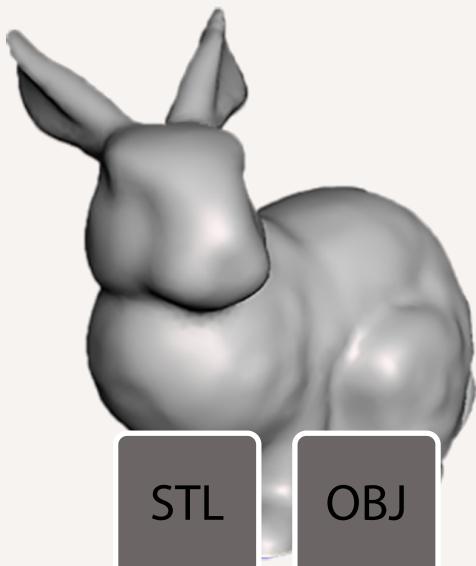
What you need to know before printing your first 3D Model.

Recommended wall thickness



DO'S

1. Each Printer/Print Service Provider has their own specifications for the minimum wall thickness that is achievable through the printing process. Please make sure to check it.
2. It is preferable to create the models in single piece of mesh, for the ease of printing. If you plan to glue the pieces together you can print them separately. If you wish to print moving models, make sure there is enough space between moving parts such as gears, chains, arms ...etc so as to facilitate the relative motion.
3. You may thicken the thin parts of your model in the 3D printing utility. Enter a value, such as 2mm, then check the Thicken thin parts by checkbox.



DON'TS

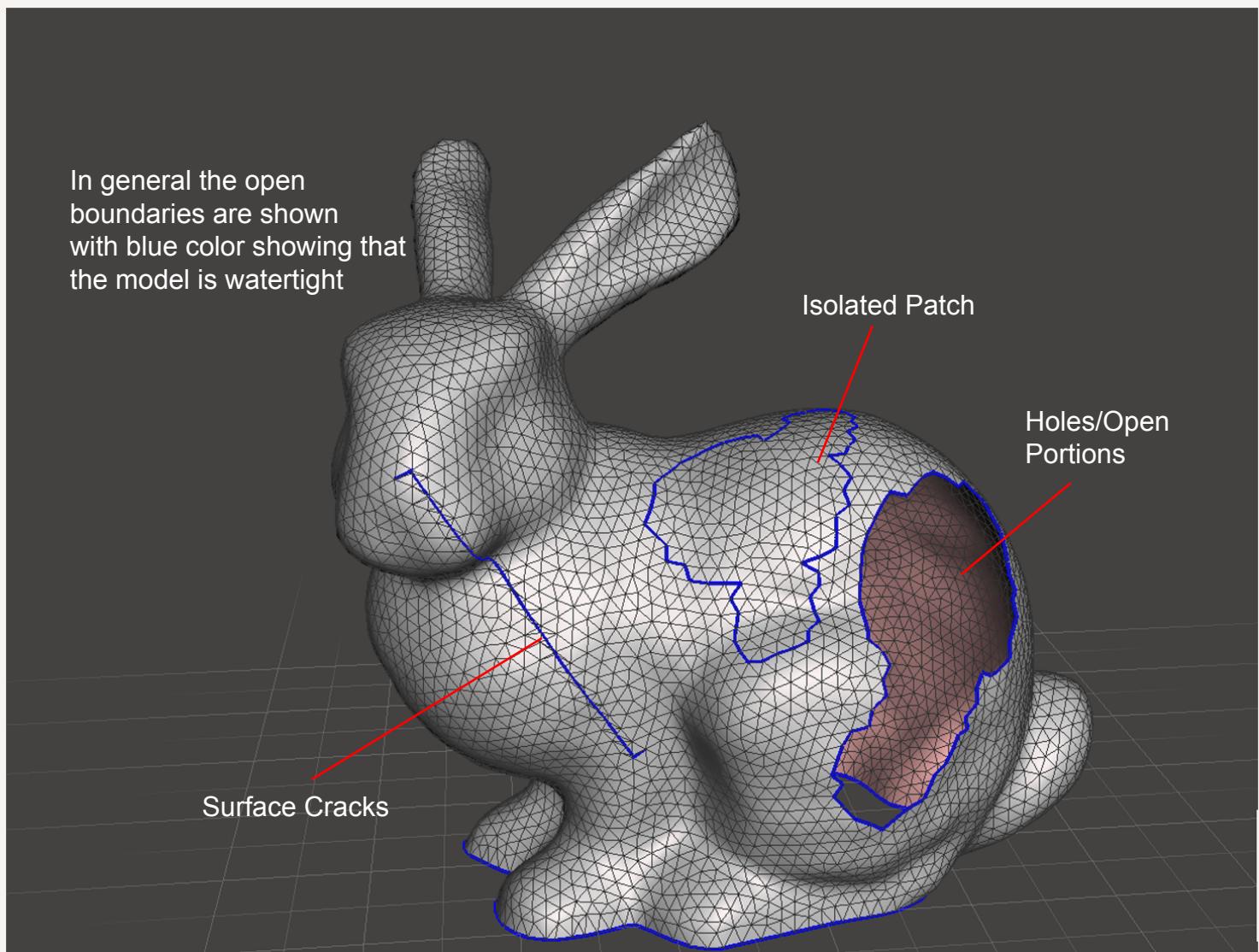
1. Do not make very thin or very small models as they may break due to various reasons. such as self weight, while removing supports etc
2. To avoid making the model delicate, never make large portions of your model very thin.
3. Taking out support material from the model becomes difficult if it is very thin. Excessive thinness could also lead to errors in the final build. Also, if the supports are very thin then, they may not be stable during the printing process. So carefully choose the model thicknesses.

Most common file formats

If you are using Online Print Services, most common file formats for exporting are stl and obj. It is safe to find out which formats do your Service Provider accept. With Meshmixer as 3D Print utility, additional formats such as .ply, .dae, and .wrl are also supported. 3D printing a model you have painted in full color typically will be exported as a .wrl file format. If the object's color comes from a bitmap, then use .OBJ with the associated texture.

Make the model watertight

It is advisable to make the model watertight prior to 3D print. The concept of model being watertight is explained below:
 Think of it as an object that would not leak if filled with water. It would not have holes, intersecting faces, or cracks to be watertight.
 To ensure your object is watertight in Meshmixer, you can click on the "Repair Selected" button.



Proper filament storage

Perhaps this is not an issue in places where the weather is perfect, but in places where the climate is humid, the filament stops working after sitting out for a few days. It is a good idea to store the filament in plastic bags with desiccant when not in use.
 (This is applicable in case of extrusion printers.)

Adding supports for overhanging parts

For extrusion printers such as a MakerBot, the overhanging portions of the model (with respect to printing direction) need to be supported. Meshmixer offers a unique way to analyze and create supports for overhangs on your objects.

1. You can manually design the supports while creating the model or use the feature, [Generate Support Structure](#) to create the supports automatically.
2. In case of Polyjet printers, if the print volume is filled with solid powder, that itself acts as a support material. However, if the print volume is filled with liquid (SLA printers), support structure is necessary.
3. Make sure you set the scale of the objects you are printing prior to generating support structures.

