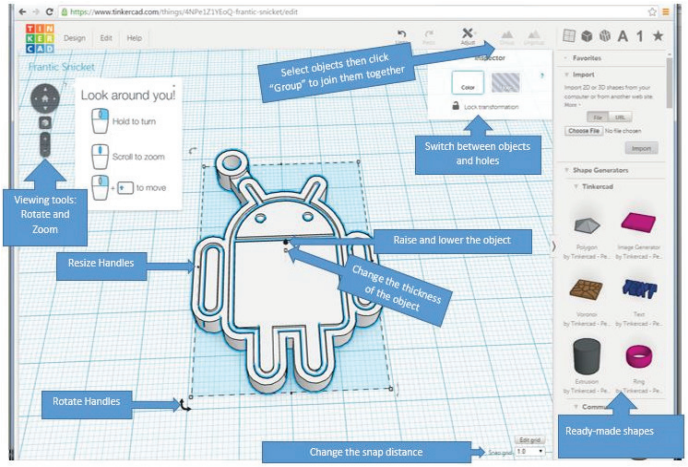
3D CAD MODELLING IN TINKERCAD

Learning TinkerCAD

The easiest way to learn TinkerCAD is to visit the “Learn” section on the TinkerCAD website at https://www.tinkercad.com/learn/.

This area contains lots of of tutorials and step-by-step design projects. You can work your way through these in your own time to learn the software basics and develop your CAD skills. The tutorials are visible on-screen alongside the TinkerCAD design interface making it very easy to follow them. At the end of each tutorial you will have mastered new skills as well as having a completed 3D model.



TinkerCAD is a straightforward 3D CAD program to learn to use, the software has a simple interface and is based around building up a design by using 3D shapes from the library and combining them (by adding or subtracting shapes) to build your model. TinkerCAD also has “circuits” functionality which allows you to design circuits, program an Arduino and incorporate the electronics directly into your 3D designs.

Designing for 3D Printing

3D printing allows for models with very complex geometries to be printed that often can not be manufactured in other ways. However when designing models for 3D printing you need to be aware of the physical limitations of the 3D printer and design your models accordingly. The two main considerations are the nozzle size of the printer and how the printer deals with overhangs.

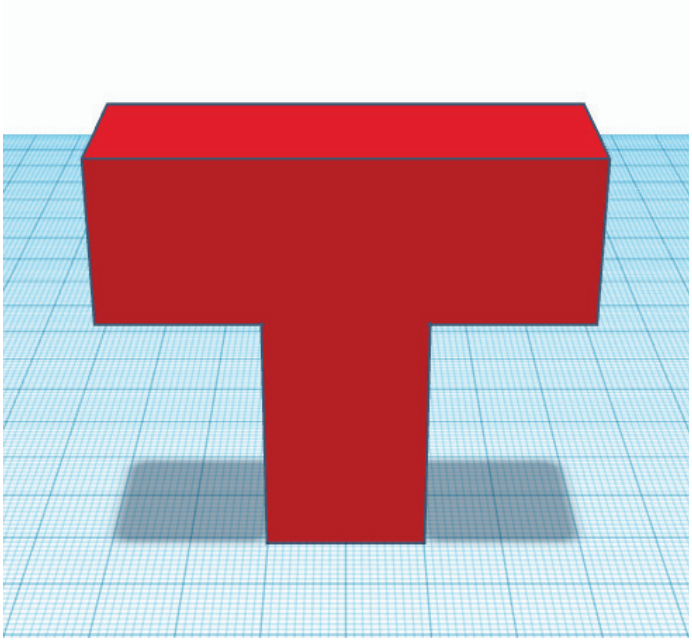
1. Nozzle Size

   The standard nozzle size on an Ultimaker is 0.4mm. This means that any lines or faces smaller that 04.mm will not print. The majority of the time you would never reach dimensions this small, however if you are trying to print small text on the surface of an item, you might encounter this problem.

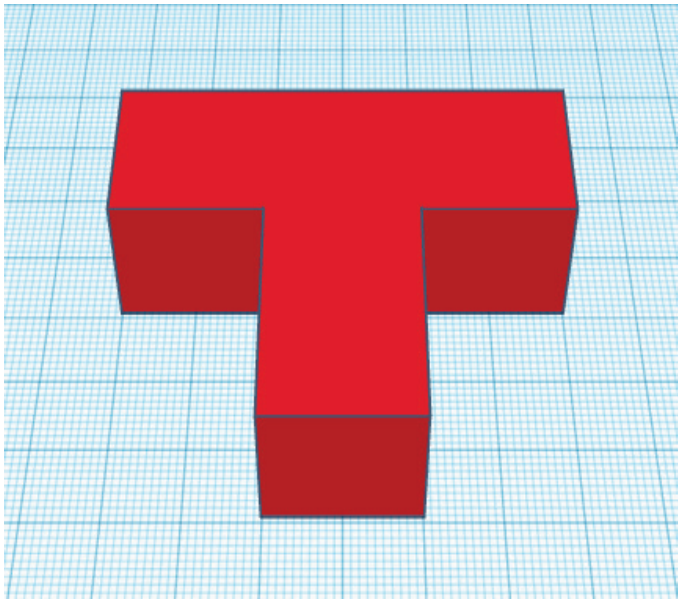
2. Overhangs

   When a 3D printer tries to print parts of the model with no structure underneath (overhangs) the filament has nothing to rest on so this can cause a print to fail. To deal with overhangs the slicing software can generate supports. These are additional support structure that are printed underneath the overhangs to support them. The support structures then need to be removed from the finished print by hand. The problem with this is that this can leave a messy print surface which needs to be filed and/or sanded smooth which can be a time consuming process. With the Ultimaker 3 support structures can be printed in PVA which dissolves in water, but it can take several hours to dissolve and PLA is much more expensive than the standard PVA you will be printing with.

The best way to deal with overhangs is to limit them as much as possible in your design. As a general rule of thumb the 3D printer will print overhangs of up to 45° from vertical, overhangs greater than this will generally require support. The 3D printer will also deal with small bridges where the filament is strung between two supporting areas, some drooping will occur on the bottom edge but the shorter the distance, the less drooping occurs.



Model with overhangs

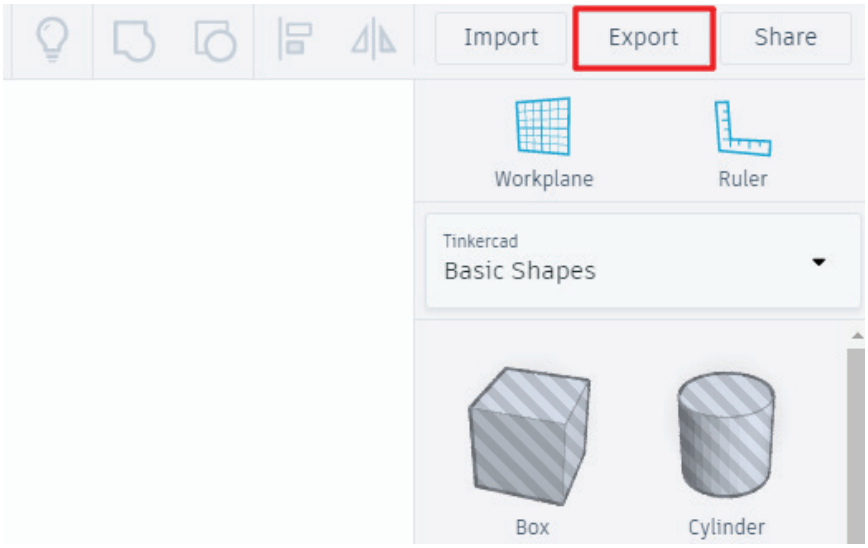


Model lying flat with no overhangs

Saving a Model for 3D Printing

When you have finished your 3D model, before saving, it is a good idea to rotate your view around the model to check that all the individual shapes that make up the model are connected as they should. Sometimes from one viewing angle it looks like a shape is touching another when it isn't and this only becomes apparent when you change the viewing angle.

When you have checked your model and are ready to save, simply click on the Export button in the top right corner of the interface.



Finally, select .STL as the file type, this will download the 3D model file ready for slicing.