3D PRINTING IN AN ARCHITECTURAL PRACTICE
6 STEP GUIDE WITH TIPS AND TRICKS

1. GET TO KNOW THE MACHINE

There are many, many different types of 3D printers around these days, each slightly different so you need to get familiar with the one at hand.

If you are just starting out, what resources do you have access to? Are there any good 'how to' tutorials online?

Don't underestimate the wealth of knowledge available online. There are really good tutorials out there. We use the Ultimaker App on our phones which we refer to constantly. There are also really good community based forums online – typically if you have a question, someone else has had the same question before you and found the answer!

- **What slicer software do you need to run the machine, and does it need to be installed on your computer?**

    What material(s) (and colour(s)) do you want to print with? Does it need to be ordered in?

    There are many, many different materials available to print with on the market. If you are doing a lot of printing, make sure that you order from the same supplier to ensure that colours match!

    Some machines are able to print with more than one filament at a time, so you might be able to choose multiple colours or print with dissolvable supports. There are also printers out there that are able to print in full colour, if that is something that you would like to achieve.

- **How big a model can you print?**

    The build volume of the machine will determine the maximum size of a part, you will need to fit your design within this volume. For larger models you may need to split it up into smaller pieces. This varies from machine to machine.

2. CREATING A 3D PINTABLE MODEL

There are many ways of creating a model that is ready to print, depending on which program you are working in, each much too specific to go into in this guide. So here are some easy to remember rules of thumb:

- **What scale do I want to model at, and how long will it take to print?**

    This is really important to consider early, especially if you intend to do an iterative series of models. Remember that the bigger the model is, the longer it will take to print. A model at 1:100 will take about 4 times as long as the same model at 1:200. Ask yourself, is the time required to print this going to become a barrier to your designing?

    As a rule of thumb, I tend to use a scale 1 step smaller than if I were modelling by hand. So if I would build a 1:500 model by hand, it is probably OK to print it at 1:1000. The printer will generally be able to handle the detail at the smaller scale, and in my opinion a larger 3D printed model can run the risk of looking a bit unresolved.

    If asked to give a quick estimate of time when beginning it will take to print a model I usually estimate about 8-10 hours a piece. This of course varies wildly depending on the machine and what you want to print (I have seen print times ranging from 20 minutes to 100 hours), but at the time of publishing 8-10 hours seems to be a good goal, as it allows you to put on one print in the morning, and one as you leave to go home.

- **Is the model able to be printed?**

    Typically you want to ensure that the model you are creating is watertight (no holes or surfaces with no thickness), with all elements at a suitable thickness for printing (as a rule of thumb I suggest you start with 1-2mm, and then get thinner as you get more experienced).

    As you would expect, the better your drafting is, the better the quality of the resulting print.

- **What file do I need to output?**

    Our printers can take .stl and .obj files. Check what files your slicer software can use, and work out how to export this file type from your modelling software. You may need to investigate plugins, but a quick google search or a question to a colleague will usually give you an idea what you need to do.

3. PREPARE YOUR FILE IN YOUR SLICER SOFTWARE

This process differs between different softwares, but here are some tips:

- **Optimise your model for speed**

    Time is usually the biggest constraint when 3D printing, so we want to make sure we optimise our file set up to speed things up!

    Most slicer software comes with default 'simple' settings which will work perfectly fine for producing a model, however there are usually massive time savings if you delve into the more advanced settings.

    Typically a 'low res' model is perfectly acceptable for an architectural model and will usually go unnoticed, so I generally advise to print at a lower quality setting than is typically standard.

    At a minimum, you will want to understand how nozzle type, layer height, wall thickness and infill density affect the time required to print.

- **Check if you need supports**

    If you are using a FFF (Fused Filament Fabrication) machine (like a Makerbot or Ultimaker) the majority of overhangs will need to be supported by automatically generated supports, to avoid the model slumping. These supports can be torn away after your print has completed.

    If you don't want to use supports, you can also look into printing your model on its side or upside down to avoid overhangs.

4. PRINT YOUR MODEL

This process will also differ machine to machine, but here are a few things to be aware of to ensure your print does fail.

- **Make sure your print will stick to the build plate**

  The majority of prints fail because they don't stick correctly to the build plate. A bit of care is recommended when setting up to ensure this doesn't happen

  Firstly, ensure that the machine, especially the build plate it properly calibrated. There are typically instructions in your printers documentation on how to do this. Take the time to go through these step by step if you are having issues.

  A very thin layer of glue from a glue-stick will also help PLA prints stick. Only a very fine layer is needed, and If there is accumulated glue on the glass plate give it a quick wash in warm water and start fresh.

  If you are printing in ABS, a small amount of filament dissolved in Acetone and brushed onto the plate will help your print stay put.

- **Make sure you have enough material (properly) loaded.**

  Kind of obvious, but this happens surprisingly often. Make sure that there is enough material loaded to complete the print. Also check for tangles, as these will cause your print to fail.

- **Remove your print with care**

  Prints can often be very rigidly stuck to the build plate, so be careful when removing them. If you are having difficulty, you can always soak everything in warm water for 15 minutes or so, this should make it easier to remove.

  Try and minimise the amount of glue you use, as this will make your print easier to remove!

5. SHARE YOUR MODEL!

Every print you do is a little bit of design research, make sure you share what you learn with others so everyone can be involved in the fun. We all come across different problems and opportunities while we work, so it is really important to share what you learn! I find a carefully photographed model to be a very powerful resource to share.

6. TEACH SOMEONE TO 3D PRINT

3D printing is quickly become standard practice in Architecture firms, so do your part for your practice and industry by helping others to use this awesome technology!