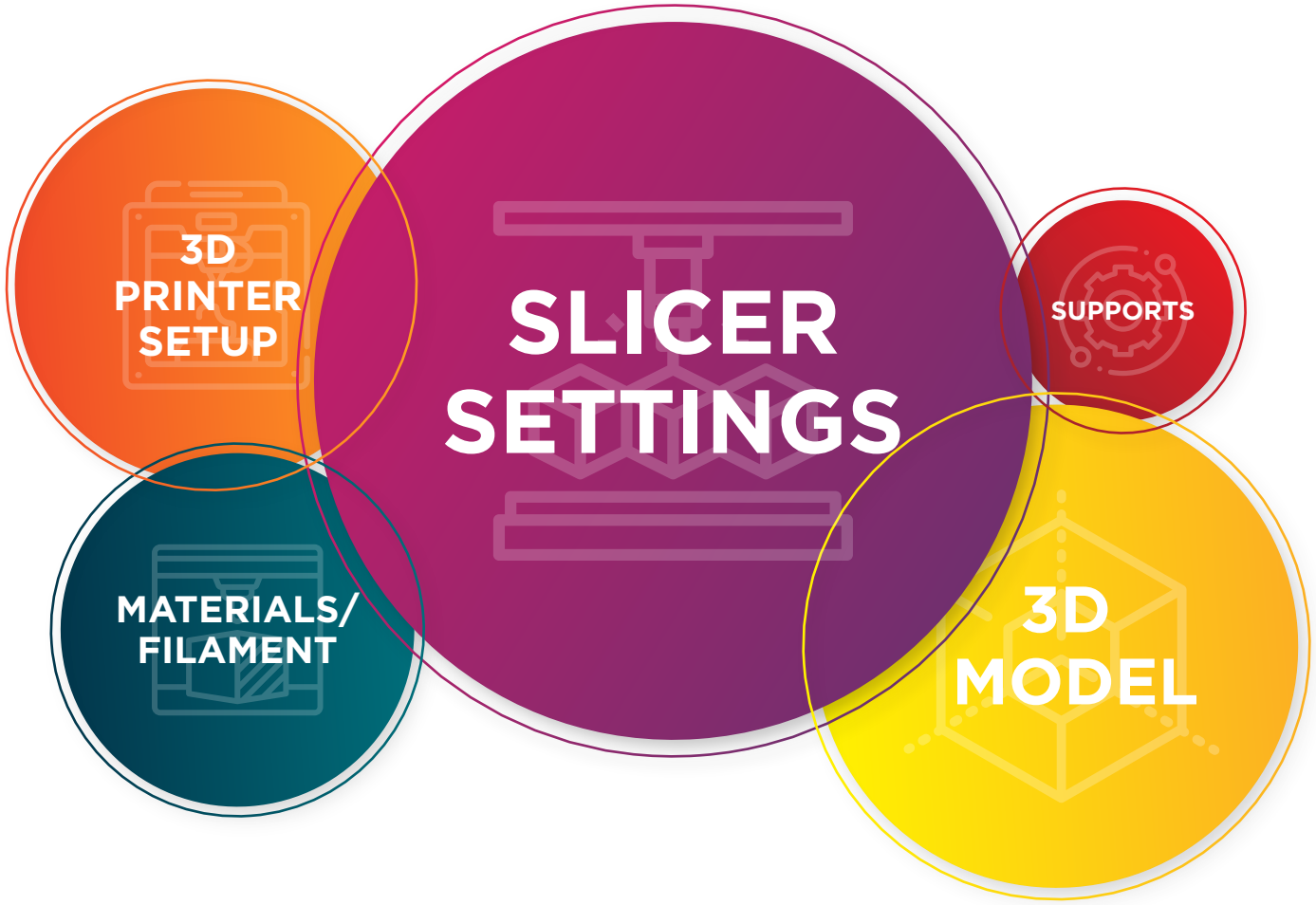
5 PILLARS OF PRINTING

CONFUSED WITH WHAT AFFECTS WHAT?
ALL THE KEY CAUSE AND EFFECTS EXPLAINED.



BEGINNERS 3D PRINTING CHECKLIST

Institute of 3D Printing

Institute of 3D Printing

Before you print, make sure you've checked everything off this list! This checklist serves as a reminder to prevent printing mishaps, but will also give you an idea of what parameters can cause which issues and how they all relate to each other. This is not an exhaustive list, but includes the most common issues and their causes.

3D PRINTER SETUP

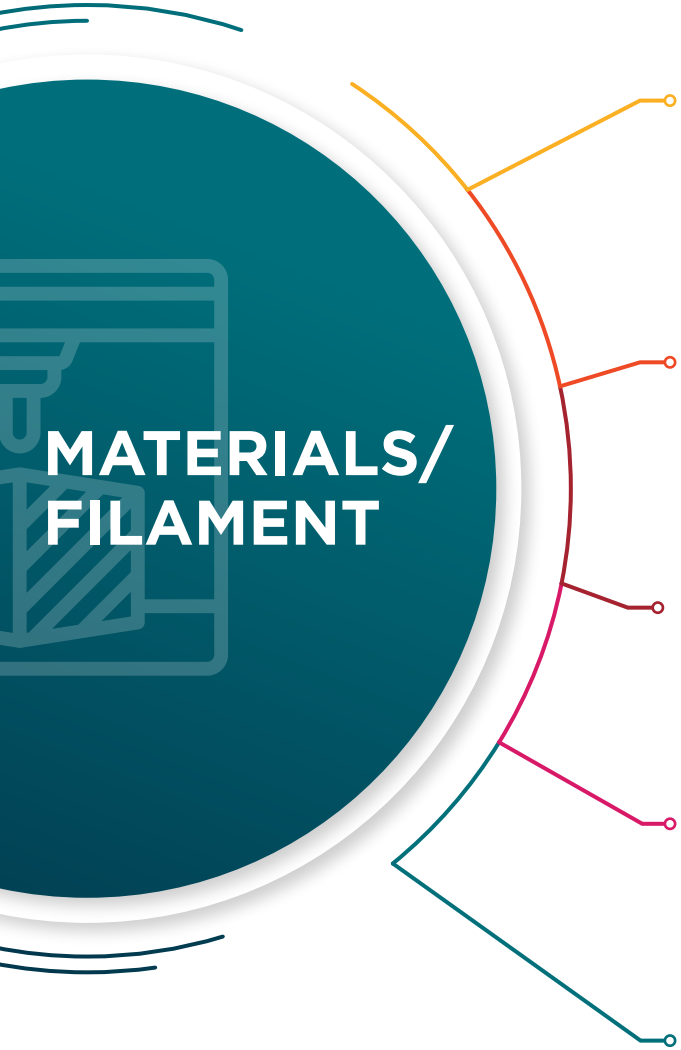
CHECK THIS	AFFECTS & SUGGESTIONS
E-steps & Flow Rate Calibrated and accurate? Filament isn't slipping on feeder gear and no over - or under-extrusion.	• Layer to layer adhesion • Finish quality of print
Check Belts turn freely without excessive slack.	• Ghosting on features
No Wobble on moving components; bed, extruder gantry etc.	• Can lead to layers mis-aligned
Optimal Z Gap (distance between nozzle and bed), nice level of "squish".	• Critical for adhesion of first layer
Bed Perfectly Levelled CRITICAL for reliability of printing in general	• Can cause the first layer to not stick • Prints to warp or break away from bed during printing • Poor overall finish quality
Using The Optimal Adhesion Surface For Your Filament?	• First layer won't adhere • Print will warp away from the bed after a few layers

TOP TIPS:

Always watch the first layer, if this goes right the rest of the print should complete well.

Institute of 3D Printing

MATERIALS/ FILAMENT



CHECK THIS	AFFECTS & SUGGESTIONS
High Quality Filament Around $25-$30 per 1kg spool	• Finish quality • Extrusion (jams) • Layer to layer adhesion • Build plate adhesion • Print strength • Your sanity!
Filament is Dry	• Damp filament can cause poor finish quality • Weak prints • Popping during printing
Filament Delivery No tangles in filament, and spool holder turns freely. Ensure no excess slack, keep it in tension at all times.	• Can cause random printer jams, stopping the print.
Printing Profiles Tune for each filament	• Printing with sub-optimal temperatures and speeds affects: – Layer to layer adhesion (finished part strength) – Finish quality
Profile vs. Material The material you're printing with is the same profile you selected in your slicer.	• Nozzle clogging

Institute of 3D Printing

SLICER SETTINGS

CHECK THIS	AFFECTS & SUGGESTIONS
Use a Powerful Slicer With the features you need, such as Simplify3D, Cura or PrusaSlicer	• Spending too much time learning a slicer only to later find out the feature you need isn't available can be frustrating
Print Speed In a "sensible" range for your setup. For example, 50mm/s is the default in Cura. You can print faster, but slower is always recommended until everything else is optimised.	• Printing too fast can cause many issues. • There are few issues that printing slower does not improve.
Layer Height Correct for the level of detail you want	• Use smaller layer heights when you want higher detail. • Don't print too-thin layers if you don't need the extra detail. • Use adaptive layer height if you want the best of both worlds.
Correct Amount of Shells	• Increase shells for a stronger print (at the cost of more time and material). • A minimum of 2 shells is recommended for most prints.
Infill Percentage	• Weight • Strength • Printing time (10-20% infill is good for most prints)

Institute of 3D Printing

05

SLICER SETTINGS

CHECK THIS	AFFECTS & SUGGESTIONS
Cooling Is enabled & sufficient	• Insufficient cooling causes saggy or molten-looking prints and poor overhangs and bridges. • Too much cooling for some materials (like ABS) can cause layer splitting. • Most materials benefit from reduced cooling on the first layer to promote bed adhesion.
Feature Select Check no special features are accidentally selected, like “Vase Mode”	• “Vase Mode” prints a single-walled print with no infill and no top layers.
Preview Function Use this to check through layers in G-Code.	Look for: • Missing layers • Non-flat bottom of the print • Unsupported support sections

Institute of 3D Printing

Always prepare your print with the aim to minimise supports. Excessive supports are wasteful, take more time and filament and increase chances of failure. They can also increase the amount of post-print finishing work required.

SUPPORTS

CHECK THIS	AFFECTS & SUGGESTIONS
Does your model need supports? Turn off if it doesn't, turn on if it does.	• Ensure all angles over 50* have some support • Look for unsupported parts of the model (like arms or tails when printing figures) • Ensure supports are stable. Tall, narrow supports may topple over mid-print!
Easy Removal Do you have a layer of separation between the top of the supports and the bottom of the print?	• This will make it much easier to remove supports. 1 layer (commonly 0.2mm) is a good starting point.
Orient Your Print To reduce or eliminate the need for supports.	• Sometimes you manage a tradeoff between strength, finish quality, and support volume when orienting the print. • Choose what matters most to you – decorative parts benefit from finish quality but functional parts benefit more from extra strength. And all prints benefit from reduced support volume. • Use Meshmixer's "orientation" function to calculate the best orientation for your part based on these three factors.

Institute of 3D Printing

3D MODEL

CHECK THIS

AFFECTS & SUGGESTIONS

Working File The STL/OBJ/3MF file you use is watertight, manifold (where intentioned) and not corrupt. Corrupt files can be recovered in Meshmixer (a free tool) using **Analysis -> Inspector**, or using Autodesk's free online service, Netfabb.

- File may result in unprintable G-code or strange, unexpected behavior during printing.
- Can also cause missing faces
- Poor resolution
- Parts printed entirely solid instead of with infill, and other geometric inaccuracies.

File Has a Flat Bottom Note: It might look flat, but make sure you check in a tool like meshmixer. Most apparent when the first layer is sliced as only a section of the bottom of the print touches the build plate. Plane-cut the bottom if in doubt.

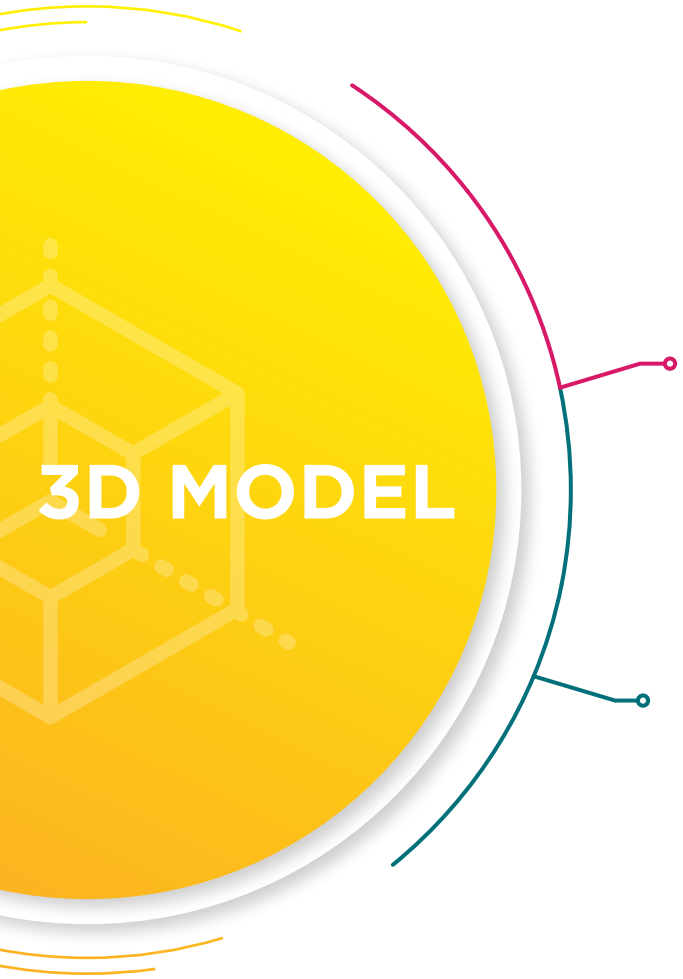
- Can affect the first few layers / general printability if the first layer is uneven. The printer may even attempt to print layers in thin air.

Designed For 3D Printing? Was the file designed with 3D printing in mind? Look for no overhangs over 50%, preferably with chamfers and fillets added, and sufficient wall thickness throughout (minimum recommended thickness = extrusion width. Usually 0.4-0.48mm).

- Many 3D files were not truly designed to be 3D printed, causing multiple issues during printing. Thin walls can disappear, extended features on Z axis can be prone to snapping off.

Institute of 3D Printing

08



CHECK THIS

AFFECTS & SUGGESTIONS

Scale / Resolution Is the file at an appropriate resolution?

- Too high can cause a very resource-intensive file, potentially leading to the slicer crashing or producing a G-Code your printer cannot take.
- Too low and the model may look blocky and/or miss key features or details.

Orientation on the Build Plate
FDM printing is “anisotropic” meaning it’s strongest along the layer lines, like wood grain. Ensure the sections you need to be strongest run along these layer lines, not perpendicular to them

- This affects strength or finish quality. Ensure you know which you desire more.

Optional Upgrades?

Be very careful when upgrading. Generally if you're new to 3D printing, do not upgrade. Get good results on the stock hardware first, and then build from there. Many novices jump straight into upgrades and make things much harder for themselves because it's now 10x harder to troubleshoot!

However, there are some universal upgrades that are well worth having. Here are the ones we suggest are good to have from the beginning. Please note, the first suggestion is extremely important and highly advised:

Enclosure with HEPA / Carbon Filter: Typical advice is to print in a well ventilated area, but many users assume this just means having the window open.

This isn't really sufficient. It's important you do not breathe in the VOCs (volatile organic compounds) and UFPs (Ultra-Fine Particles) your printer creates as they can be bad for your health. Luckily, a simple enclosure with a Carbon or HEPA filter extractor can be built cheaply, or more robust options purchased from companies like Kora 3D.

Typical advice is to print in a well ventilated area, but many users assume this just means having the window open.

Octoprint: This is software you can use to essentially remote monitor and control your printer. Use it to deliver files and start the print, instead of carrying over an SD card and clicking print.

Use it to watch the print with a basic webcam hookup, and a whole host of other useful features and plugins are available - the list is growing all the time. It's the sort of thing that once you start using it, you'll wonder how you ever printed without it!

Removable Flexible Bed Surface or UltraBase Fixed Glass Bed:

Both options let the prints pop right off, so you don't need to scrape the bed and risk damaging it (the printer or your print).