Level the bed

Adjust the height of the bed so that when the nozzle is at height zero \((Z = 0 \text{ mm})\), a thin piece of paper (like receipt paper) will barely pass between the nozzle and the bed. Repeat this process for each corner of the bed and verify in the center of the bed. If you can’t get the corners and the center at the same level, your bed may be warped. In this case, you’ll either have to replace the bed, or enable mesh-bed-levelling (consult with our guides for more instructions).

Clean Nozzle

It goes without saying that printer cleanliness and operational maintenance should be periodically carried out. Remove carbon build up in nozzle with a nozzle cleaner or cleaning filament. Remove built-up plastic from the outside of the nozzle by wiping with a tissue while the nozzle is hot. Be careful not to burn your finger! You can grasp the tissue with tweezers to keep your fingers away from it.

Bearings & Belts

Keep the bearings lubricated and belts tight – you can retro-fit a simple belt tensioner.

First Layer

Don’t print your first layer too close to the bed, you could cause the extruder to skip and the nozzle may block as the filament can’t escape. Likewise, don’t print too far from the bed, or the first layer won’t stick and the print will come off.

Hot Level

Only level your bed when both your extruder and heated bed are hot. When hot, these parts of your printer expand. So a bed levelled when cold is not actually calibrated for hot printing. Watch our complete guide to bed levelling masterclass inside the Io3DP training suite. Find out more at io3dp.com

Bowden Tube

Ensure your Bowden tube is clean as it can quickly harbour dust particles. Sometimes it only takes a 0.2mm particle to block a 0.4mm nozzle.

Bed Surface

Ensure bed surfaces are maintained. Alcohol wipe down after each use and for PEI use fine grit sandpaper to clean surface every few prints. Don’t touch the bed with your fingers (grease). Remove specks of dust using a clean wide paintbrush.
Step 2: Environment

1. No Draughts
   Keep draughts low where required to prevent cooling too fast (e.g. ASA), or use a desk fan to aid faster cooling where necessary (e.g. when printing bridges with PLA).

2. Temperature Swings
   Watch out for wild fluctuations in ambient temperature. Printing in a garage in summer could result in temperature swings of 20°C between day and night, leading to warping and cracking.

   A printer enclosure may help with this – ABS nearly always prints better in an enclosure, or a consistently warmer ambient room temperature.

3. Tidy Up
   Ensure your environment is clean and not overly dusty.

4. Ventilation
   It goes without saying that you should always print in a well-ventilated environment, regardless of the printer or filament being used.
Step 3: File/Slicer Settings

1. Watertight
   Only use watertight 3D printer ready files. Sketchup is not suitable for 3D printing.
   We recommend using approved printer-ready files where possible, from sites like MyMiniFactory.com.

2. Check Files
   If a 3D file is suspected to be unusable, quickly check it over using Meshmixer (a powerful, free tool) to inspect it.
   We’ve recorded a brilliant Meshmixer masterclass for everything 3D printing. Contact us for more details at success@io3dp.com.

3. Fast Layers Need Faster Cooling
   If any layer takes less than 15 seconds to print (nozzle passing over the same point), consider slowing these layers down or increasing cooling (using a desktop fan).
   Insufficient cooling times can lead to deformation, warping and poor print quality on small features.

THE FAIL-FASTER HACK
If you have a print with a particularly difficult feature (for example, a long bridge, steep overhang, narrow gap or very fine feature), and you’re not confident if your printer and settings are up to it - extract that part from the print (using Meshmixer’s “plane cut” feature for example) and print just that.
This way you can test it in isolation without spending the time and material of printing the whole model only to find that that part fails. This is particularly useful for fine-tuning print settings because you can iterate far more quickly with smaller models.
Get all the handy meshmixer tips with our complete Meshmixer. Head to https://www.io3dp.com/ to find out more.

Don’t Knock It
Smaller prints getting knocked off the bed during printing?
Use a skirt or brim to aid bed adhesion on tall & narrow or smaller surface prints.
## Step 4: Filament

### 1. Clean Filament
Ensure filament is clean. A dusty environment can lead to dust on your spool.

If you have a dusty environment, or you’re printing with an especially small nozzle size, use a clip-on filament cleaner to scrape off particles before the filament enters your extruder.

### 2. Nozzle Width
Composite materials print better with wider nozzles. Generally, 0.5mm+ nozzle sizes are optimum for composite filaments.

### 3. Use Uprated Nozzles
Composite materials (other than soft wood/woodfill) will wear down standard nozzles and filament feeder hobbed gears. Use uprated hardened nozzles and hobbed gears when printing with composite filaments, especially glow in the dark (strontium aluminate is extremely abrasive) metal, glass or carbon fibre-filled filaments.

Ensure hobbed feeder gear is kept clean from particle build-up, or it will slip and you’ll under-extrude.

### 4. Raft
If you’re using a filament that’s designed to have lower adhesion (e.g. Break-away), it will be easier to print using a raft.

### 5. Filament Quality
This is more important than most realise.

*Use good quality filament.*

### 6. Snap Test
To check if PLA is still fresh, do the snap test: see if a piece of the filament breaks by bending it back on itself. It should fold without snapping. If it snaps, throw it away – it’s gone stale.

Usually, PLA should be replaced 6 months after opening, but good quality filament stored under the correct conditions (e.g. airtight bag/container, out of direct light) can last much longer.

### 7. If In Doubt, Dry It Out
Dry out older filament, or filament exposed to air for long periods, before using. Most FDM filaments are hygroscopic, meaning they absorb water. Always dry Nylon before use, as this absorbs the most moisture.
8 **Firmer Filaments**
If using a ‘regular’ (non-flexible) extruder to print flexible filament, use the firmest filament you can, around 95A Shore hardness (e.g. TPU), or those with less friction (e.g. Flexible PLA). These will be much easier to print with.

9 **Don’t Leave It Hot**
When using soluble support filaments like PVA, if the filament will be waiting in the nozzle for extended periods of time, bring the extruder temperature down by around 30°C while it’s on standby. Hot molten PVA left in the nozzle at printing temperature for long periods of time can lead to blockages.

10 **Storage**
Filament should be stored in a cool, dark place when not in use, ideally in a resealable bag. UV light, moisture and warm temperatures will degrade your filament over time. Keep a desiccant in the bag to keep moisture at bay.

11 **Low & Slow**
Always print flexible materials at a slower speed. As a rule of thumb, try about half the speed you would print standard materials, such as PLA or ABS.

12 **Keep It Tight**
When unspooling and loading filaments, always keep hold of the end so there’s no slack and it cannot unravel, even just a small amount. Loosened rings of filament can expand around your spool and fall over the more tightly-wound filament, causing it to lock up during printing. Similarly, ensure your spool holder is not too loose or allowing excess slack when printing, as this will cause the same issue.
If in doubt, add a small amount of resistance to your spool holder using some cardboard between the spool and the side of the holder. This will keep the spool in constant tension with the printer and minimise the risk of your prints failing.

**PRO TIP!**
Want to know what will get you perfect prints, from your machine, faster than anything else?

... keep scrolling ↓
How To Get The Best Prints Possible, From Your Machine

Reliably, Quickly & Easily
(without having to spend ages “tinkering” with your printer, or slicer)

We make it all simple, inside The Institute of 3D Printing Training suite. It’s THE LARGEST and MOST COMPREHENSIVE dedicated 3D printing education platform.

Learn how to setup your printer, calibrate it, optimise it for perfect prints, learn all the features in your slicer (and how to use them) and so much more.

No contracts or tie-ins & you can cancel easily whenever you like. TRY IT TODAY.

CLICK HERE TO LEARN MORE
HEAD TO HTTPS://WWW.IO3DP.COM/