



Technical guide

A Technical Guide – Part One

At first glance, 3D printing can seem deceptively simple. You load a file, press print, and a physical object appears. But anyone who has tried to use a printed part in the real world quickly learns that how a part is printed matters just as much as what is printed. Strength, durability, and reliability are all shaped by a small number of key decisions made before the print ever begins.

Among the most important of these decisions are layer height, infill, and print orientation. These settings control how material is laid down, how layers bond together, and how forces travel through the finished part. Understanding how they interact is what separates fragile display pieces from robust, functional components.

This guide is written for readers who want to move beyond beginner assumptions and start thinking like designers and engineers people who want their prints not just to look good, but to work properly.

What Layer Height Really Means

Layer height describes how thick each individual printed layer is. In most desktop FDM printers, this value is measured in millimetres and typically falls somewhere between 0.12 mm and 0.28 mm. While it's often described as a "quality setting", layer height influences far more than surface finish.

Smaller layer heights create smoother surfaces and finer detail because the steps between layers are less visible. This is why detailed models and visual parts often use thinner layers. Larger layer heights, on the other hand, reduce print time and can actually improve layer bonding when temperature and flow are set correctly.

It's important to challenge a common misconception here: thinner layers do not automatically make stronger parts. In fact, thicker layers can sometimes produce stronger results because there is more molten material available to bond each layer together.

With a standard 0.4 mm nozzle, most reliable results are achieved when layer height stays between roughly 25% and 75% of the nozzle diameter. That places the practical range between about 0.16 mm and 0.28 mm. Outside this range, layer bonding and extrusion consistency begin to suffer.

How Layer Height Influences Strength

To understand strength in 3D printing, you need to understand anisotropy. This simply means that printed parts are not equally strong in all directions. The weakest direction is almost always between layers, along the vertical (Z) axis.

Layer height plays a direct role here. Very thin layers increase the number of thermal cycles during a print, which can introduce subtle weaknesses between layers. Thicker layers reduce the total number of layer interfaces and, when printed at the correct temperature, allow stronger fusion between layers.

This does not mean “thicker is always better”, but it does mean that chasing ultra-fine layers for strength is often counterproductive. Strength comes from good bonding, not just visual smoothness.

Understanding Infill from the Inside Out

Infill controls how much material exists inside the part. Rather than printing objects completely solid, most prints use an internal structure to support the outer walls. This saves material, reduces print time, and—when done correctly—maintains strength.

Infill is usually expressed as a percentage, but it’s important not to think of it as a direct measure of strength. A part printed at 30% infill is not “half as strong” as one printed at 60%. In practice, strength increases rapidly at first and then levels off.

For many functional parts, infill values between 20% and 30% are more than sufficient. Higher infill levels add weight and time but often provide diminishing returns unless the part is under extreme load.

Why Infill Pattern Matters

Infill is not just about quantity; it’s also about structure. Different infill patterns distribute forces in different ways.

Patterns such as gyroid and cubic are popular in functional printing because they distribute stress evenly in multiple directions. This makes them well suited for parts that experience complex or unpredictable loads. Simpler patterns like grid print quickly, but they tend to perform poorly under twisting or bending forces.

For most real-world applications, a gyroid or cubic infill at a moderate density offers the best balance between strength, material use, and print time.

Why Wall Thickness Often Matters More Than Infill

One of the most important lessons in functional 3D printing is that outer walls do far more work than internal infill. Walls resist bending, impact, and surface damage, while infill mainly supports the walls from collapsing inward.

A useful rule of thumb is this: adding one extra wall often improves strength more than increasing infill by 20%. This surprises many beginners, but it reflects how forces actually move through printed parts.

For most functional prints, using three to five walls combined with moderate infill produces parts that are both strong and efficient. Increasing the number of top and bottom layers further improves durability, especially for parts under compression or impact.

Print Orientation: Strength Without Changing Settings

Print orientation is one of the most powerful—and most overlooked—tools in strength optimisation. Because layer bonding is weakest between layers, parts are strongest when forces act *along* the layers rather than *across* them.

A bracket printed flat can support far more load than the same bracket printed upright. Holes and threads tend to perform better when printed horizontally. Long beams are far stronger when laid flat rather than standing vertically.

In many cases, simply rotating a part in the slicer improves strength more than changing any numerical setting.

Temperature, Cooling, and Layer Bonding

Layer bonding depends heavily on thermal behaviour. If plastic cools too quickly, layers do not fuse properly. If it is printed too cold, the material becomes brittle and weak.

Functional parts often benefit from slightly higher extrusion temperatures and reduced cooling. This allows layers to remain molten long enough to bond effectively. Materials such as PETG, ABS, and ASA are particularly sensitive to cooling, and excessive fan use can dramatically reduce strength.

Even PLA, often printed with high cooling for visual quality, can produce stronger parts when fan speeds are reduced.

Material Choice and Mechanical Behaviour

No amount of tuning can compensate for the wrong material. PLA is stiff and dimensionally accurate, but it is brittle and prone to creep. PETG is tougher and more forgiving. ABS and ASA tolerate heat and impact far better. Nylon offers exceptional strength and fatigue resistance but demands careful moisture control.

Material choice often has a greater impact on real-world performance than infill percentage or layer height. Choosing correctly at this stage prevents failure later.

Knowing When Professional Printing Makes Sense

Some designs demand consistency, precision, and material control that are difficult to achieve reliably on home machines. Parts that must fit accurately, perform under load, or be produced repeatedly benefit from professional printing workflows.

At BritForge3D, prints are optimised for mechanical performance using calibrated machines, proven material profiles, and controlled environments. The goal is not just to make parts look good, but to make them work as intended—every time.

If your part needs to function reliably, not just print successfully, professional support can make all the difference.

Final Thoughts

Understanding layer height, infill, and print orientation changes the way you approach 3D printing. Instead of guessing, you begin making informed decisions that save time, reduce material waste, and dramatically improve performance.

These principles apply whether you are printing at home or working with a professional service. Once mastered, they form the foundation for every successful functional print.

In the next technical guide, we will explore **tolerances, shrinkage, and dimensional accuracy**—essential knowledge for parts that must fit together precisely.

Guide 1 Summary – The Basics of 3D Printing

This guide introduced the fundamental process behind 3D printing, from digital model to finished physical part. By understanding how slicing, layer formation, and material deposition work together, you can begin to see why certain print decisions matter more than others.

Rather than treating 3D printing as a black box, this guide encourages you to think of it as a controlled manufacturing process. Once you understand how layers are created and bonded, settings such as orientation, material choice, and strength optimisation start to make sense.

In the next guide, we build on this foundation by exploring how layer height, infill, and internal structure influence the strength and reliability of printed parts.

