BRITFORGE3D
3D PRINTING SERVICES

Technical guide

A Technical Guide – Part two

Understanding Tolerances in 3D Printing

One of the most important concepts to understand in 3D printing is that printed parts are never exact copies of their digital designs. This can be surprising at first, especially for people coming from traditional manufacturing or CAD backgrounds. On screen, a model appears perfect and precise. In reality, the printed result is always an approximation — and that's completely normal.

The way we manage this approximation is through tolerances. A tolerance is simply an accepted range of variation between parts that are meant to fit together. Rather than expecting absolute accuracy, designers allow for small differences so components can still assemble and function correctly.

In traditional manufacturing methods such as machining, tolerances can be extremely tight because material is removed in a controlled, predictable way. In 3D printing, parts are built layer by layer from molten plastic. This introduces more variability, which means tolerances must be wider to remain realistic and reliable.

Several factors contribute to this variation. Each layer is placed individually, material flow can vary slightly from one moment to the next, plastic contracts as it cools, and even a well-maintained machine has mechanical limits. Because of this, designing without tolerance allowances is one of the most common causes of failed assemblies in 3D printing.

For most FDM printers, a realistic expectation is accuracy within about two tenths of a millimetre for general-purpose parts. Well-tuned machines can sometimes achieve tighter results, but designing around ideal conditions rather than typical ones often leads to frustration. Good design is about predictability, not perfection.

Why Holes Almost Always Print Smaller

A classic example of tolerance issues appears with holes. If you design a hole at a specific diameter, it will almost always print slightly smaller than expected. This behaviour is not a printer fault — it's a natural outcome of the printing process.

As the nozzle moves in a circular path, molten plastic tends to curve inward rather than forming a perfect circle. Layer stacking adds a stepped edge to what should be a smooth surface, and cooling causes the material to contract slightly. When these effects combine, the final hole ends up undersized.

Because this behaviour is consistent, it can be designed around. A practical approach is to increase hole diameters in the design stage, typically by two to four tenths of a millimetre. For example, a hole intended for a 5 mm bolt may be designed closer to 5.3 or 5.4 mm to ensure reliable fit after printing.

For applications where precision really matters, post-processing is often the best solution. Drilling or reaming printed holes removes variability and produces clean, accurate results that are difficult to achieve through printing alone.

Shrinkage: Why Parts Change Size as They Cool

Shrinkage is another unavoidable aspect of 3D printing. When plastic is heated and then allowed to cool, it contracts. Different materials shrink at different rates, which is why the same design can behave very differently depending on the filament used.

PLA is one of the most dimensionally stable materials and tends to shrink very little, which is why it prints so predictably. PETG shrinks slightly more but compensates by being more flexible, reducing the risk of cracking. ABS, ASA, and nylon shrink significantly more and are far more sensitive to cooling conditions, which is why enclosures and controlled environments are often required.

Shrinkage affects more than just overall size. It influences flatness, causes warping, and distorts holes and mating surfaces. This is why material choice plays such a critical role in dimensional accuracy. Choosing the wrong material can introduce accuracy issues that no amount of slicer tuning will fully correct.

How Print Orientation Affects Accuracy

Accuracy in 3D printing is not uniform in all directions. The printer controls movement in the horizontal directions using motors and belts, which are generally very precise. Vertical movement, however, is governed by layer height, which limits resolution in that direction.

As a result, parts tend to be more accurate in the horizontal plane than in the vertical one. Vertical holes are often less accurate than horizontal holes. Tall, thin parts can drift slightly as layers accumulate, and changes in layer height directly affect vertical resolution.

For parts where accuracy is critical, orienting important surfaces in the horizontal plane often produces better results. This does not eliminate the need for tolerances, but it does reduce variability.

Layer Height and Dimensional Control

Layer height plays an important role in how accurately features are reproduced, particularly in the vertical direction. Smaller layer heights improve resolution and produce smoother mating surfaces, while larger layer heights exaggerate the stepped nature of the print.

For parts where fit matters, consistency is more important than ultra-fine detail. Moderate layer heights tend to produce more predictable results, especially when combined with sufficient top and bottom layers. Adaptive layer height can be useful, but only after careful testing, as it introduces additional variability.

Calibration: The Hidden Foundation of Accuracy

No amount of clever design can compensate for a poorly calibrated printer. Dimensional accuracy begins with consistent extrusion, stable motion, and reliable bed positioning.

Extrusion accuracy ensures that the correct amount of material is deposited. Belt tension affects positional accuracy in the horizontal plane. Bed leveling and Z-offset determine how accurately the first layer sets the foundation for the entire print. Advanced features such as pressure advance help manage material flow during acceleration and deceleration.

At BritForge3D, printers are routinely calibrated and verified to ensure that parts behave predictably across multiple prints. This consistency is essential for professional prototyping and repeat production.

Designing for Fit: Clearance as a Design Tool

Rather than trying to eliminate variation, experienced designers use clearance intentionally. Clearance is the space between mating parts that allows for assembly after printing.

Loose fits allow parts to move freely. Sliding fits enable controlled movement. Snug fits hold parts in place without force. Press fits require testing and careful iteration. These categories are not fixed rules, but starting points that must be validated through prototypes.

Fit-critical parts should always be tested before final production. Small changes in clearance can dramatically change how a part behaves.

Threads, Inserts, and Reliable Fastening

Printed threads can work, but they have limitations. Coarse threads perform better than fine ones, and orientation matters greatly for strength. For assemblies that will be taken apart repeatedly, printed threads are rarely the best solution.

Heat-set inserts provide a far more durable alternative. Installed correctly, they create strong metal threads within printed parts, allowing repeated assembly without wear. Designing for inserts requires careful attention to hole size and installation temperature, but the improvement in reliability is significant.

When Precision Truly Matters

Some projects demand tighter control than most home printers can reliably provide. Mechanical assemblies, snap-fit enclosures, replacement parts, and engineering prototypes often fall into this category.

This is where a professional print service adds real value. At BritForge3D, tolerances are adjusted per material, critical components are test-fit, and print orientation is chosen specifically to maximise accuracy. The result is consistent, dependable production for customers across the UK.

Final Thoughts

Dimensional accuracy in 3D printing is not about achieving perfection. It is about achieving predictability. Once you understand how tolerances, shrinkage, orientation, and calibration interact, you can design parts that fit first time and perform exactly as intended.

In the next technical guide, we will explore **heat resistance, load, and long-term durability** — essential knowledge for parts designed to survive real-world use.

Guide 2 Summary – Layer Height, Infill, and Strength

In this guide, we explored how strength in 3D printing is created internally rather than through surface appearance alone. You learned why thicker walls often matter more than high infill, why thinner layers are not always stronger, and how print orientation can dramatically affect performance.

The key takeaway is that strength comes from balance. Smart combinations of layer height, wall thickness, infill, and orientation produce better results than extreme settings in any single area.

The next guide focuses on one of the most common challenges in functional printing: understanding tolerances, shrinkage, and why parts do not always fit together as expected.