OSR 3D Printing Essentials

A concise guide to the basics of 3D printing for the OSR. It covers filaments, settings, and tips to achieve ideal prints. Perfect for newcomers or anyone seeking OSR-specific printing insights.

- Filaments
- Print Settings

Filaments

Polylactic Acid (PLA) filament is commonly recommended for its ease of use, safety, and versatility. PLA adheres well to a variety of surfaces and does not necessitate an enclosure for effective printing. It offers moderate impact resistance, an important consideration given the potential for mechanical interactions. However, PLA has limitations in heat resistance, softening at approximately 50°C (122°F) and deforming at temperatures around 60°C (140°F). While standard operational temperatures of servomotors generally remain below these thresholds, awareness of these properties is advisable.

Polyethylene Terephthalate Glycol (PETG) may present a more suitable alternative for certain applications. Though its impact resistance is lower than PLA, PETG is less prone to brittle failure and exhibits more elasticity before breaking. Notably, PETG maintains its structural integrity until reaching temperatures of about 80°C (176°F), a threshold unlikely to be surpassed under typical conditions.

Acrylonitrile Butadiene Styrene (ABS) is another filament option, known for its robust mechanical and thermal properties. However, printing with ABS requires specific equipment, such as a 3D printer with an actively heated chamber and minimal air drafts. Additionally, ABS printing can *release harmful gases*, necessitating proper safety precautions. For users seeking ABS-like properties without the associated printing requirements, Acrylonitrile Styrene Acrylate (ASA) may be a viable alternative.

Other filament types, such as PET, Polycarbonate (PC), or Polyethylene Terephthalate Glycol Copolymer (PCTG), are less commonly used. Flexible filaments such as Thermoplastic Elastomers (TPE) and Thermoplastic Polyurethane (TPU) may be used for specific components, such as gaskets, but are not generally required for the majority of parts.

Print Settings

For most components, a layer height of **0.2mm** is sufficient, balancing print resolution and time efficiency. A minimum of two walls and **30+% infill** is recommended, with **three walls** preferable for increased structural integrity. Parts subjected to greater stress, such as mechanical arms, may benefit from a **fourth wall** and an infill percentage of 40% or higher.

The **line width** for printing should correspond to the printer nozzle size, typically around 0.4mm. Adherence to manufacturer recommendations for nozzle and bed temperatures—as well as fan speed—is advised for optimal print quality and material handling.