**Overview**

This 3D printable vase design is available in two different models:

1. **Standard Mode Model** – A robust, fully solid version that ensures durability and watertightness.
2. **Vase Mode Model** – A lighter, single-wall version optimized for quick printing with strong, aesthetically pleasing layer lines.

**How to Print a Watertight Vase**

To ensure that the vase is completely watertight and doesn't leak, follow these essential guidelines:

- **Use at least 3 wall lines** – This provides extra thickness and improves the overall structure.
- **Scarf seams** – Adjust your slicer settings to place seams strategically, reducing weak points.
- **Print with PETG** – PETG is naturally more waterproof than PLA and provides better strength.
- **Increase print temperature** – A higher temperature ensures proper layer bonding, reducing gaps.
- **Overextrude slightly (flow ratio 1.04)** – This compensates for small gaps between extrusion lines.
- **Slow down print speed** – Reduce the flow speed in filament settings to improve adhesion.

**Printing the Standard Mode Model**

If you're printing the **standard mode model**, follow these additional tips:

- **No infill at the bottom** – Ensure that the slicer settings do not add infill to the base. Instead, use a sufficient number of **full top and bottom layers** to guarantee a solid foundation.
- **Set at least 4-5 top and bottom layers** – This helps create a strong, sealed bottom to prevent leaks.
- **Use a slightly higher flow rate** – Helps with sealing gaps between layers.

**Printing the Vase Mode Model**

For those printing in **vase mode (spiralized outer contour)**, use these settings for best results:

- **Adjust line width for strength** – If using a **0.4mm nozzle**, set the **line width to 0.8mm** to reinforce the single-wall structure.
- **Use larger nozzles for more strength** – A **0.6mm or 0.8mm nozzle** provides additional thickness and highlights visible layer lines for a stylish finish.
- **Enable spiralized contour mode (vase mode)** – This setting in your slicer ensures smooth, continuous extrusion without retractions or unnecessary seams.
- **Slow down print speed** – Reduces the chance of gaps between layers and improves bonding.