

3D Model Print Instructions

Grace Sculpture Keychain

This model includes **two file options**:

- **Supported STL** – Pre-supported and ready to print
- **Unsupported STL** – Ideal if you want to scale the model or create your own custom supports

The figure can be printed using either a **resin** or **filament (FDM)** 3D printer.
At 100% scale, the figure prints at approximately **2.75 inches tall**.

Printing Steps

- **Step 1 – Choose Your File Type**
 - **Option A: Supported STL**
 - Supports are already included and optimized for printing
 - No additional edits are required before slicing
 - **Option B: Unsupported STL**
 - Scale the model to any size you prefer
 - You will need to generate supports before printing
 - Most slicers offer both **automatic** and **manual** support options
- **Step 2 – Import File**

Download and import the STL file into your slicer program (such as Chitubox, Bambu Studio, or Lychee).
- **Step 3 – Prepare the Print**

Use your slicer to generate the print layout and settings required for your specific 3D printer.
- **Step 4 – Post-Print**

Once printing is complete:

 - Carefully remove the supports
 - For easier removal, place the model in **warm water** to soften the supports and reduce surface damage
 - Once model is cleaned - use a standard key ring and place in right hand of model to complete your keychain

Pro Tips:

- Resin prints will capture finer detail than filament prints
- For best results, print at a **0.03–0.05 layer height (resin)** or **0.1–0.2 (FDM)**
- Always cure resin prints fully after cleaning