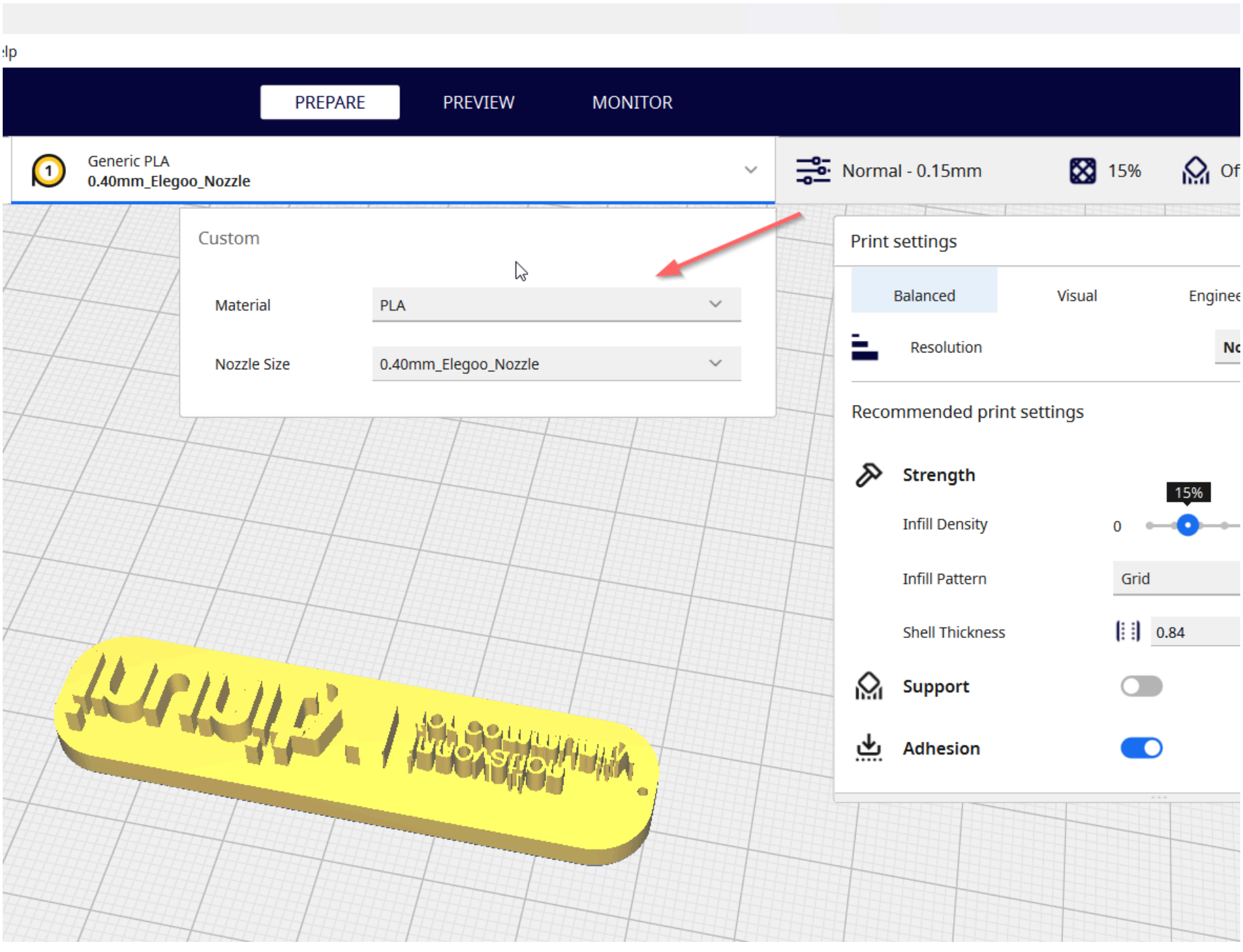
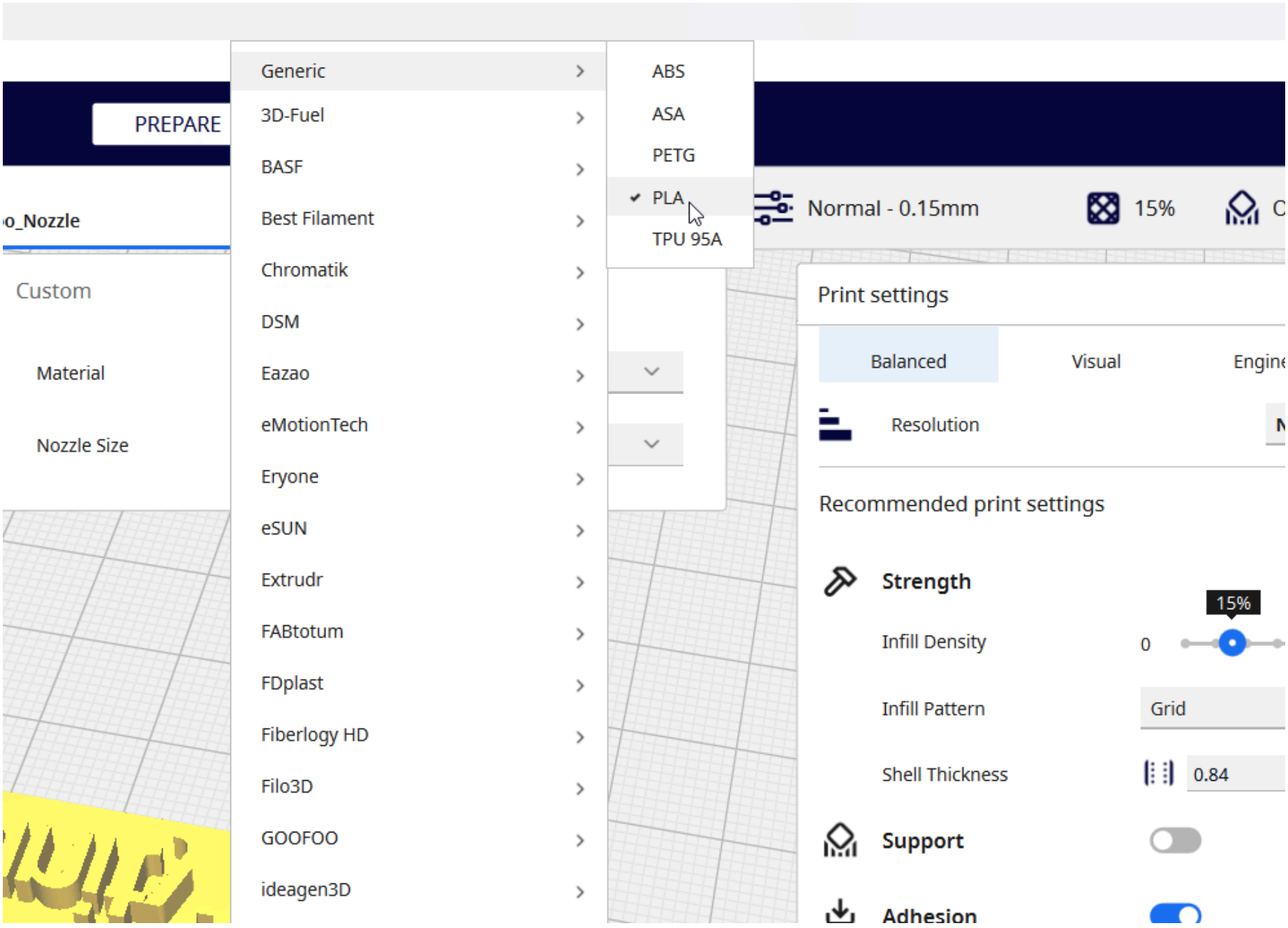
Slicing Settings

Step 1: Select Material

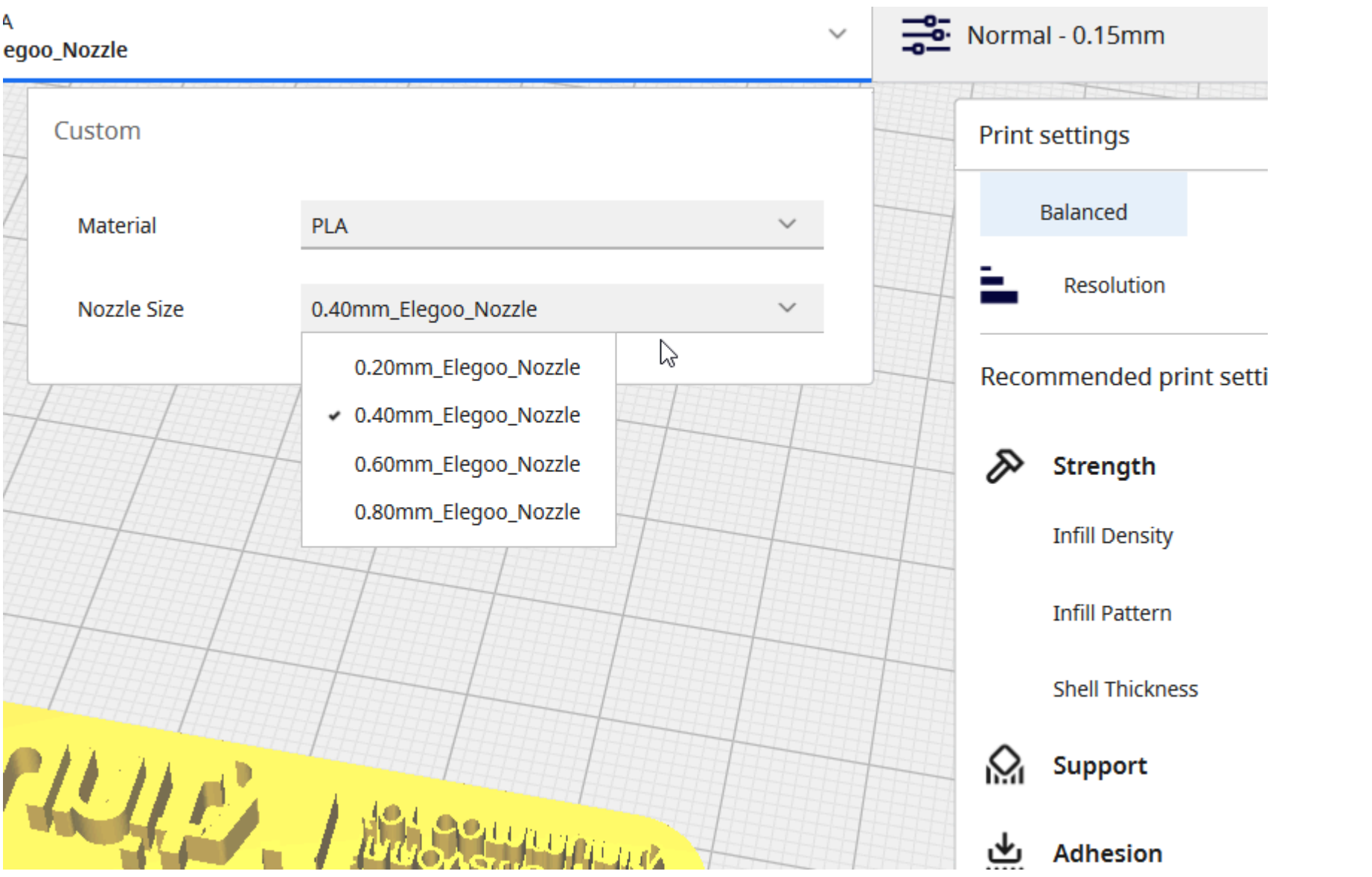


From here, you can choose different types of **printing materials**.



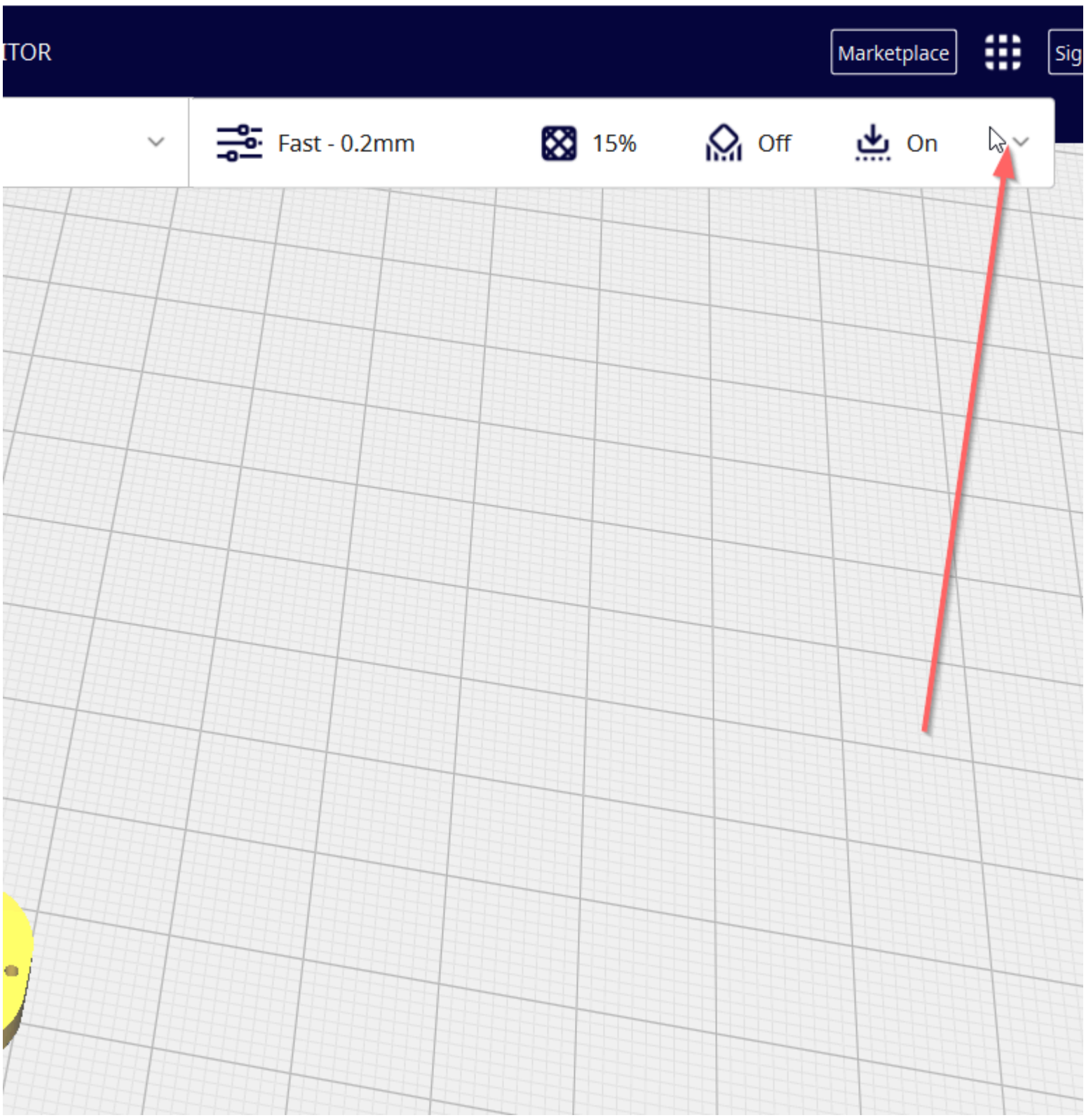
For this print, I selected **Generic → PLA**.

Step 2: Select Nozzle Size



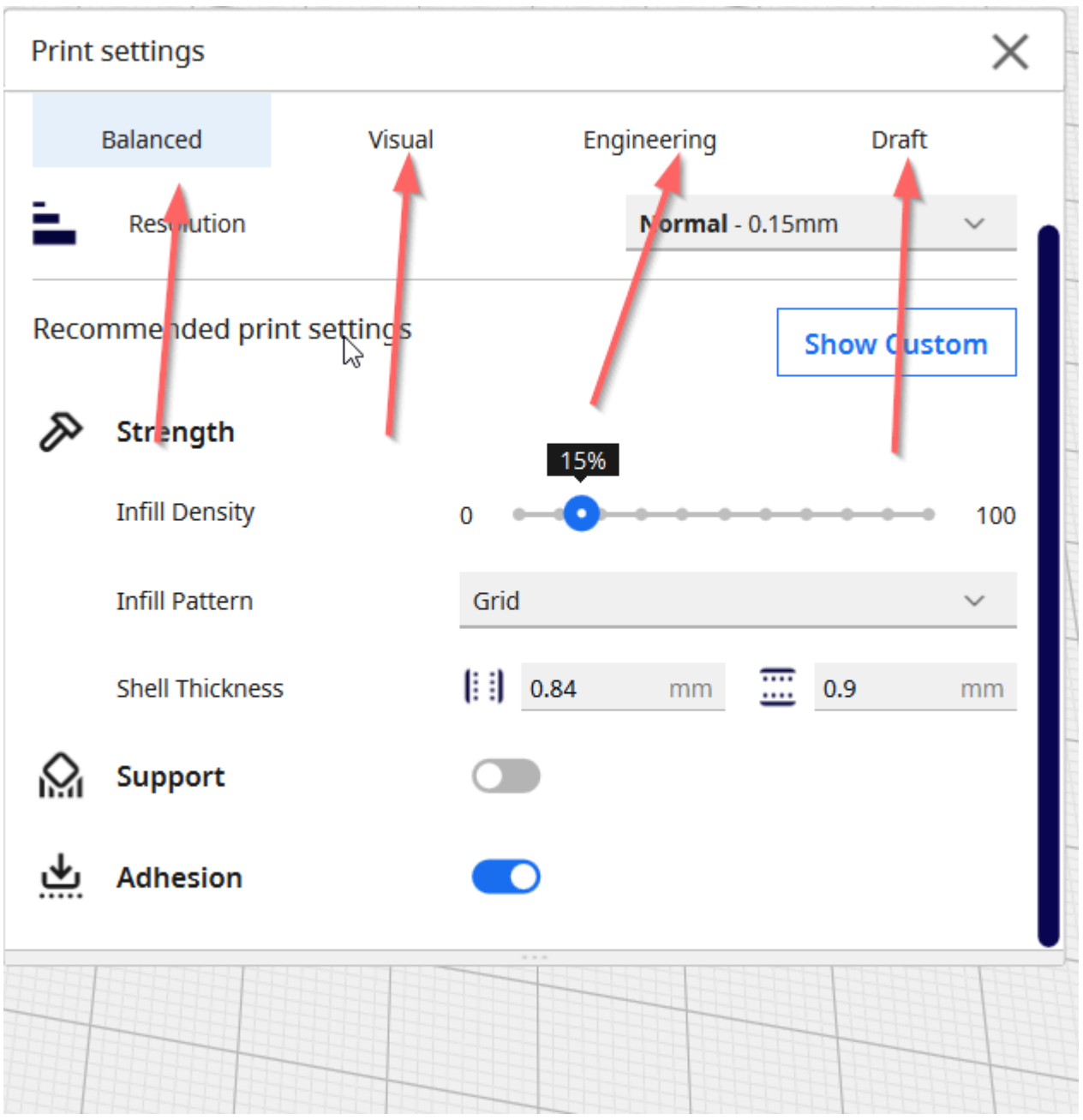
You can change the **nozzle size** from this option.
Always select the same nozzle size that is physically attached to the printer.
I am using a **0.40 mm nozzle**.

Step 3: Open Slicing Settings



Click on this option to view **more slicing settings**.

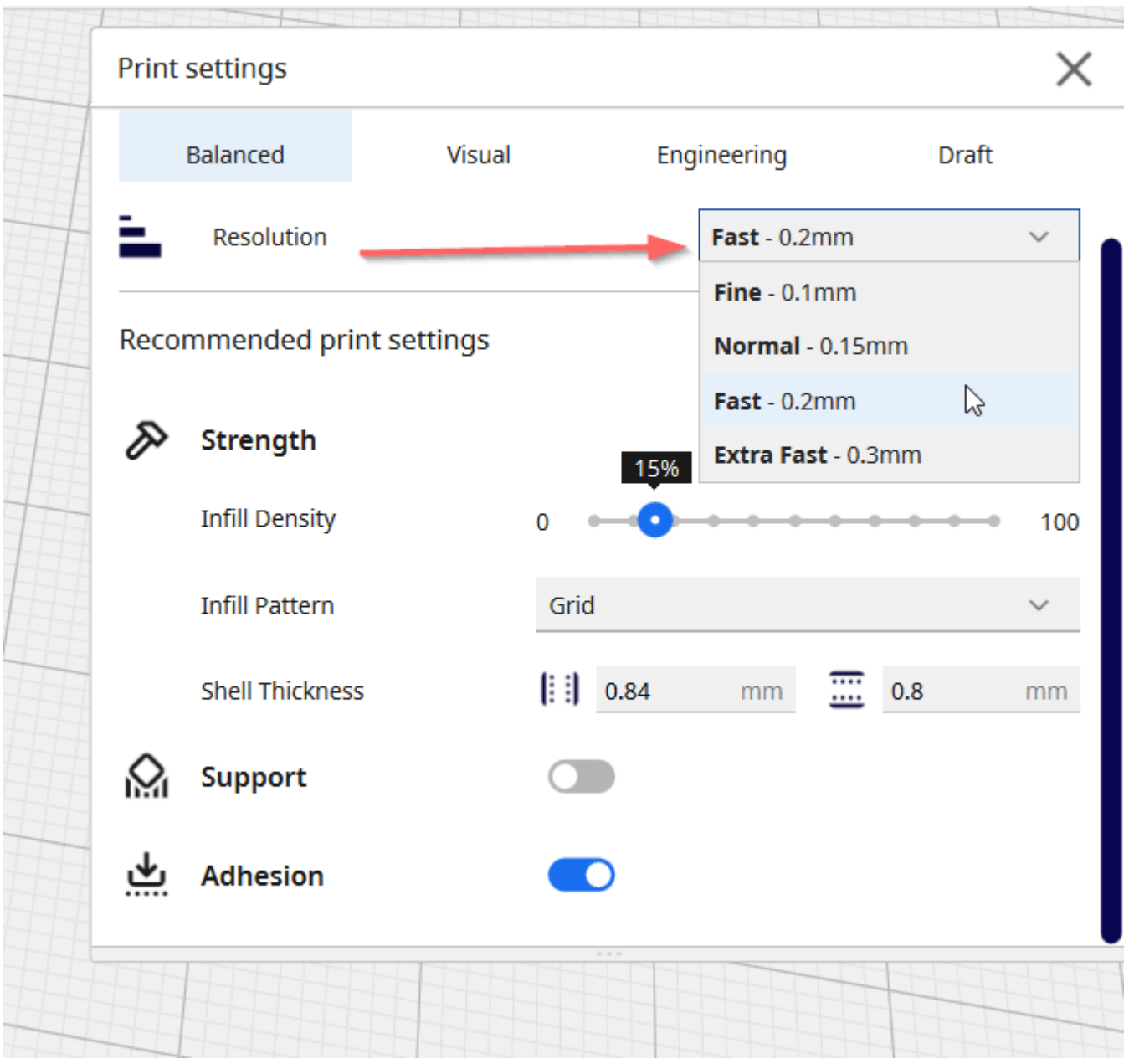
Step 4: Choose Recommended or Custom Settings



You can:

- Use **Recommended** settings
- Or switch to **Custom** and edit the settings manually

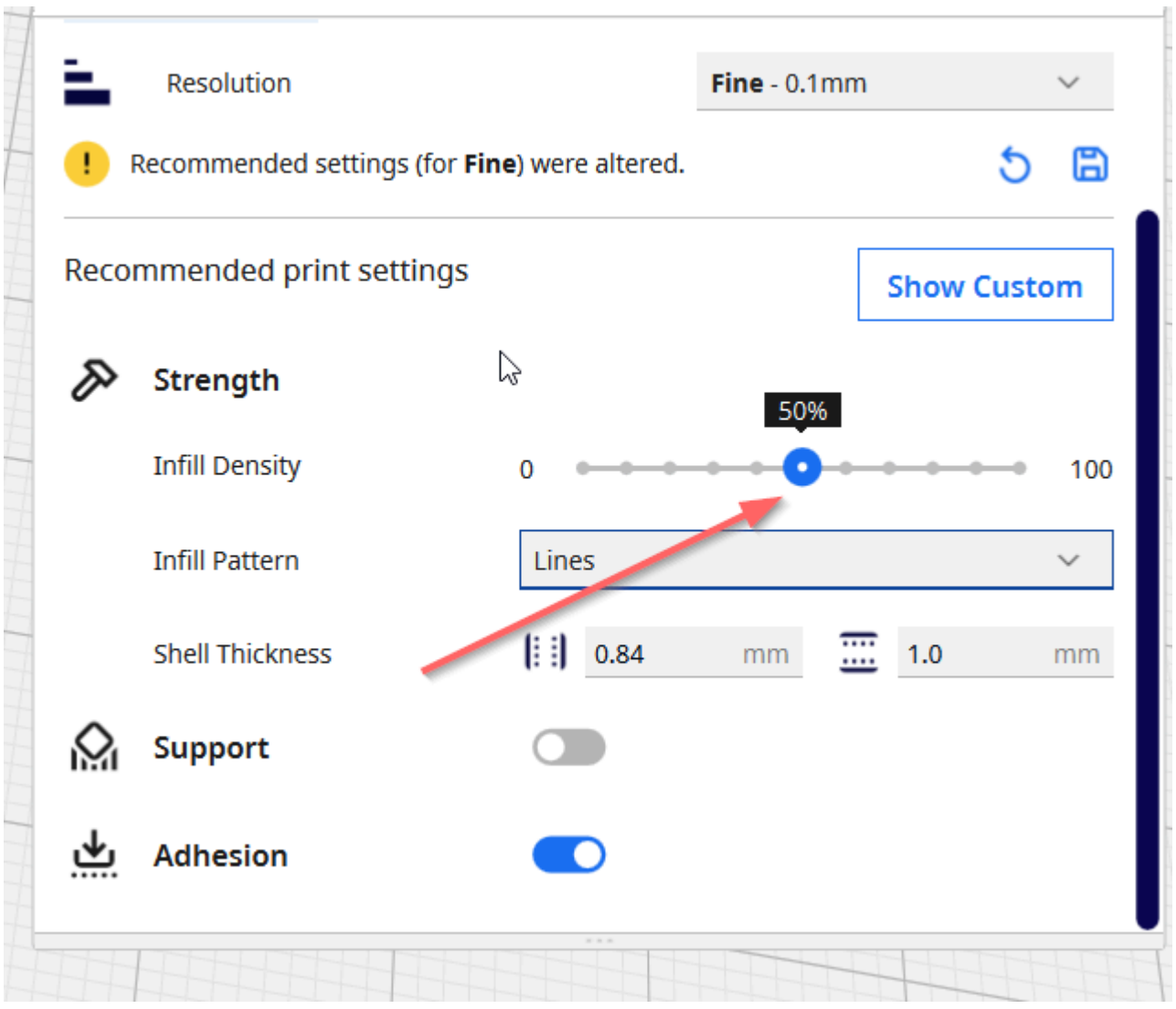
Step 5: Resolution / Quality



Resolution controls the **print quality**:

- Higher quality → better finish → more print time
- **Fine quality** is usually the best balance

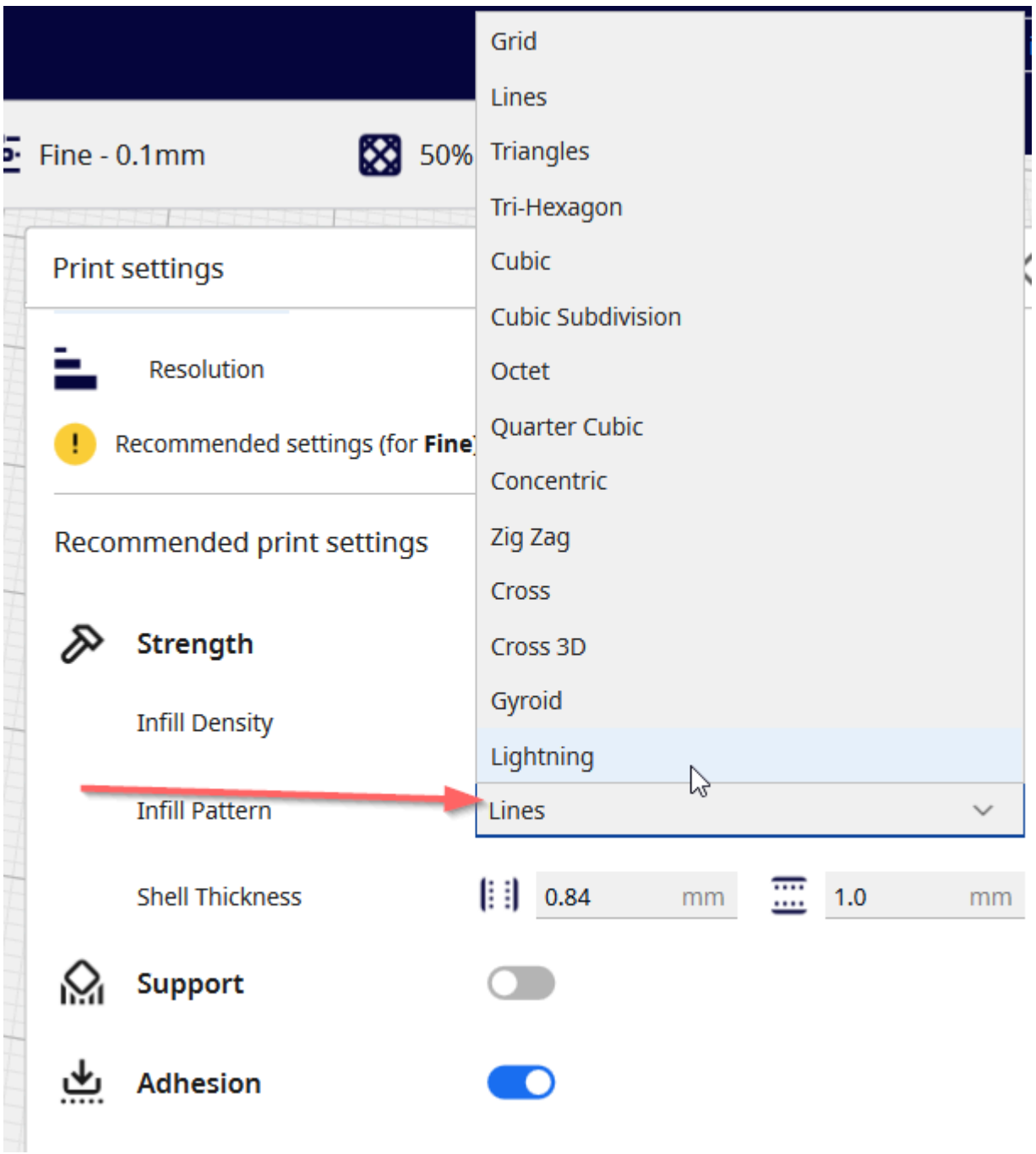
Step 6: Infill Density



Infill density decides how much material is filled **inside the walls**:

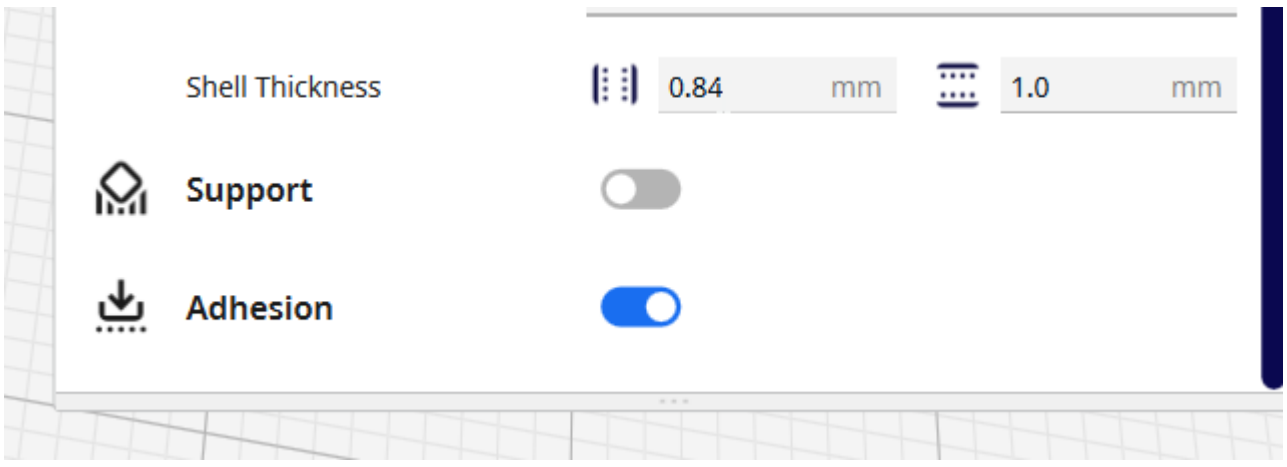
- **100%** → fully solid and strong
- **10–20%** → lighter and faster prints

Step 7: Infill Pattern



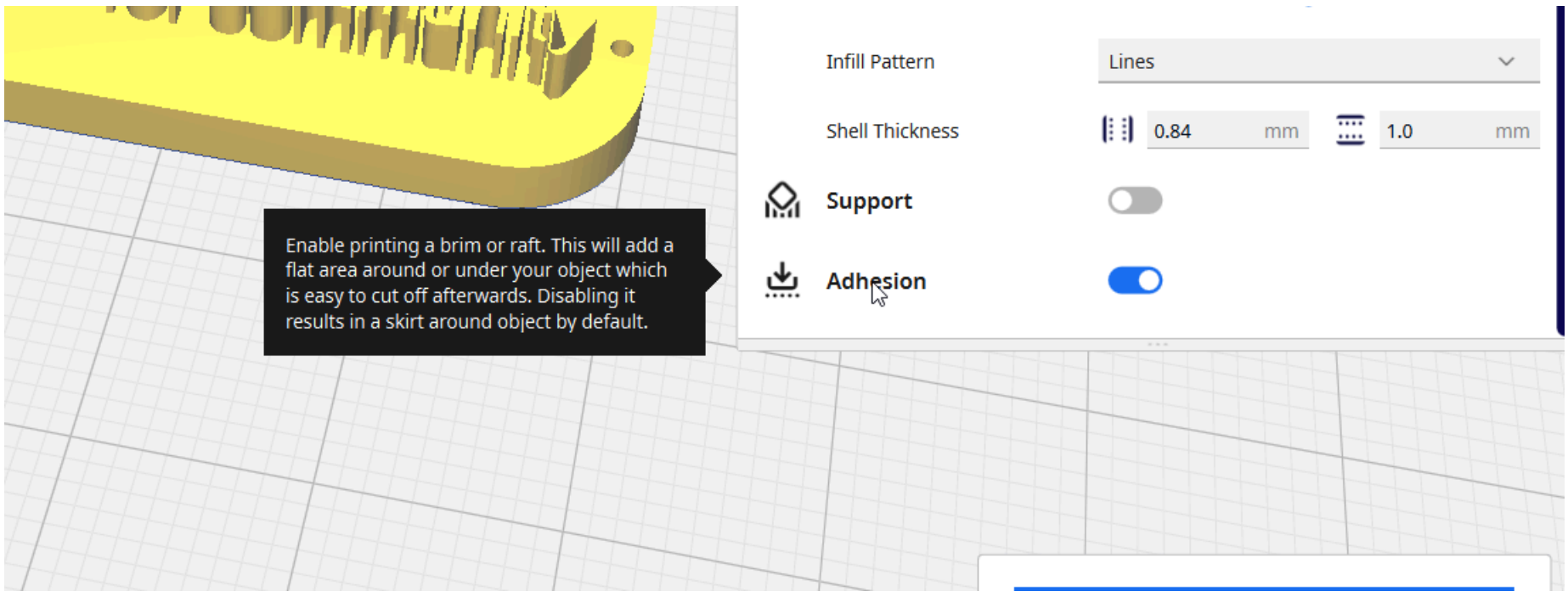
Infill pattern defines the **internal structure** of the print.
Different patterns are used for different purposes like strength or speed.
I selected **Lines**.

Step 8: Shell Thickness (Walls & Top/Bottom)



Shell thickness controls the **wall width and top/bottom layers**.
Walls are printed first, followed by infill.

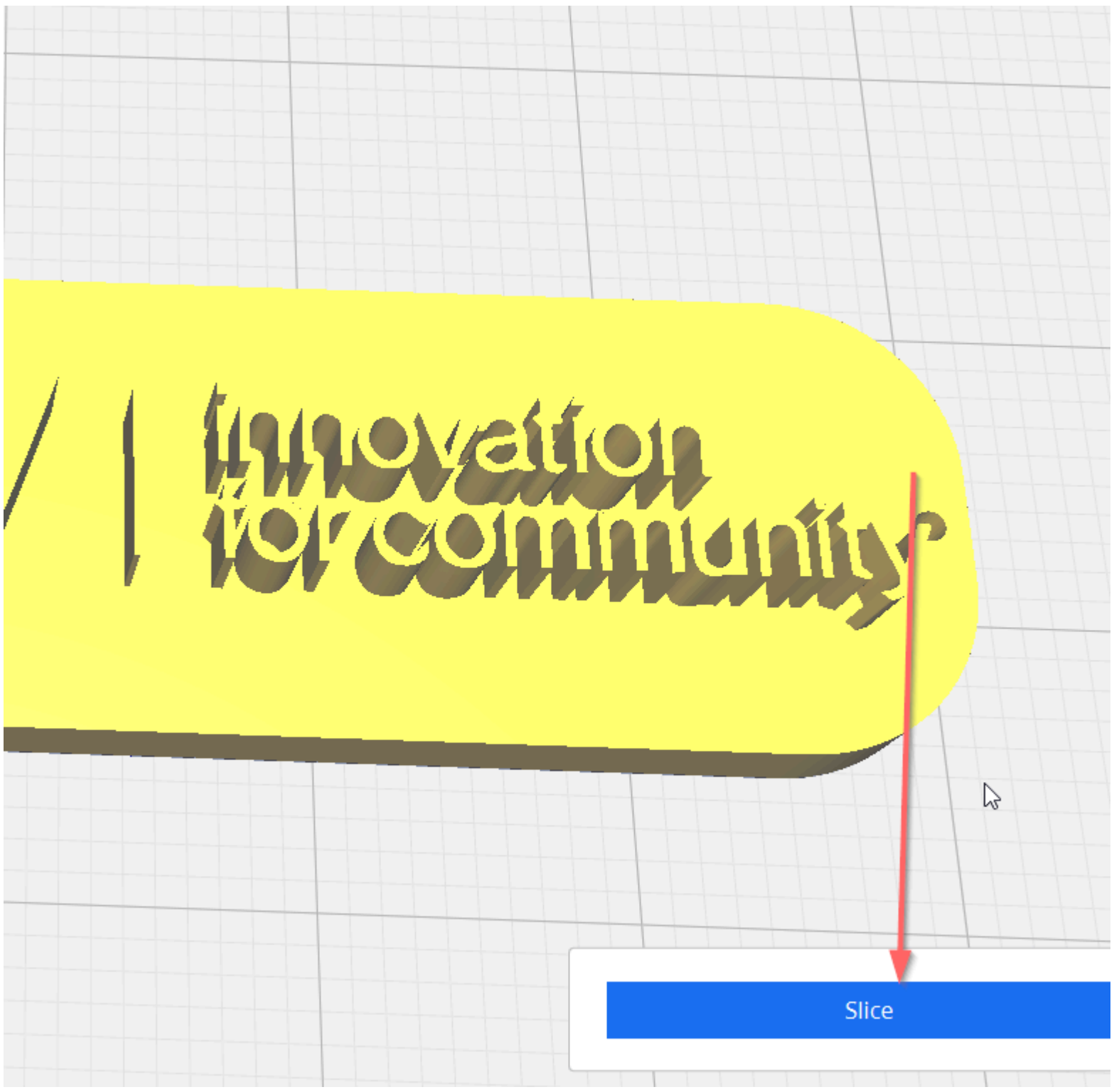
Step 9: Build Plate Adhesion



Adhesion adds a layer at the bottom to ensure:

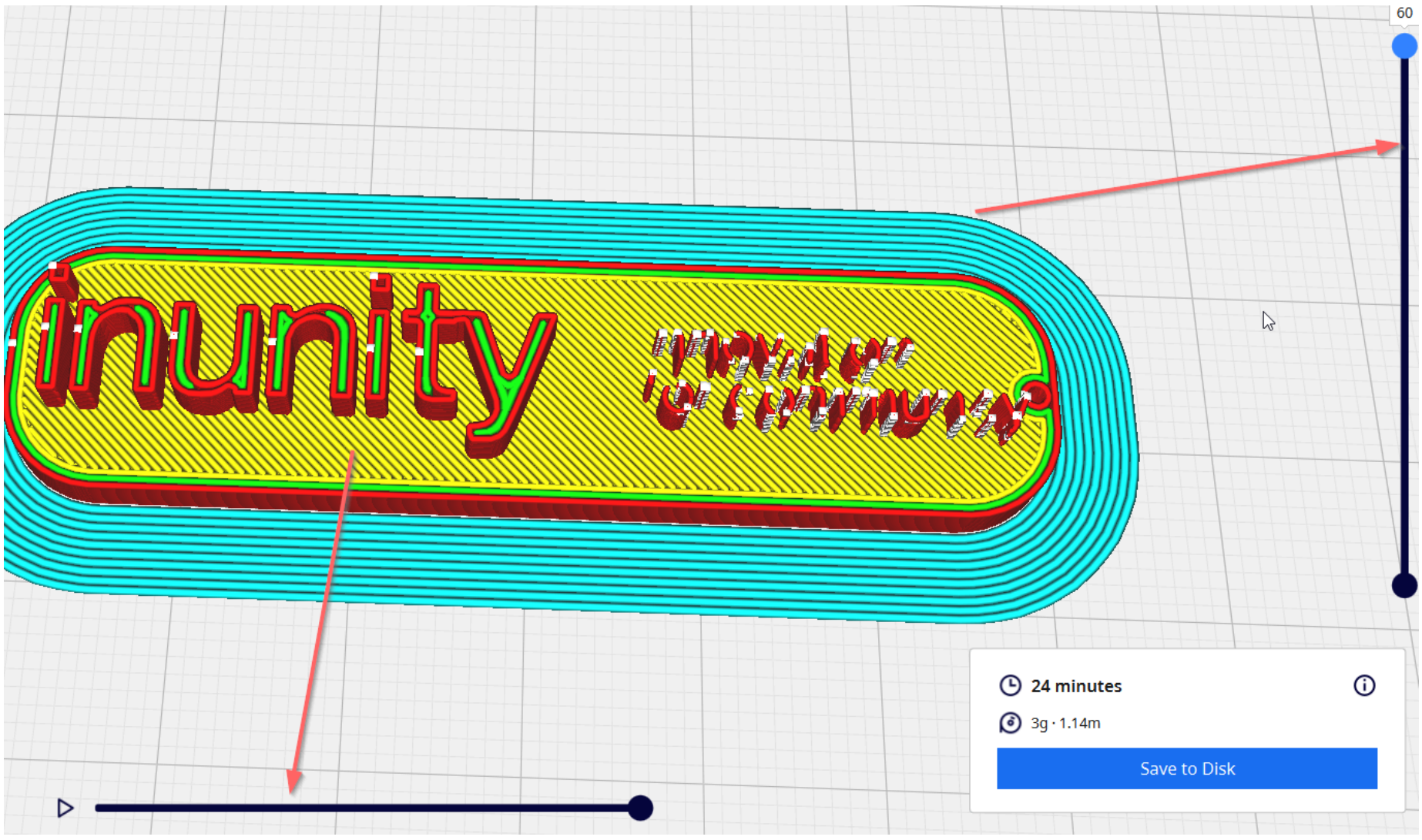
- Proper sticking to the build plate
- Reduced warping and print failure

Step 10: Slice the Model

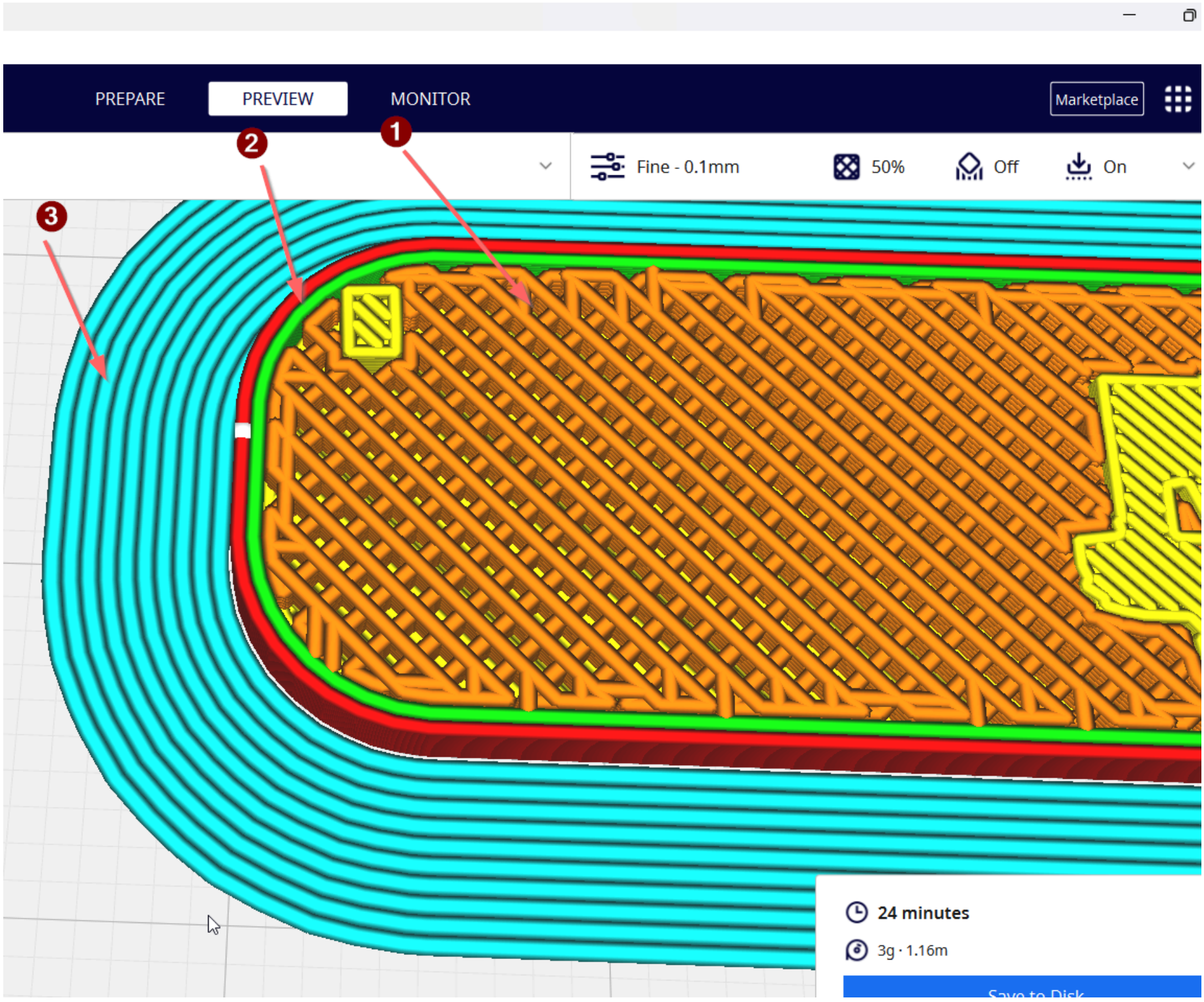


Click on **Slice** to generate the file for 3D printing.

Sliced File Preview



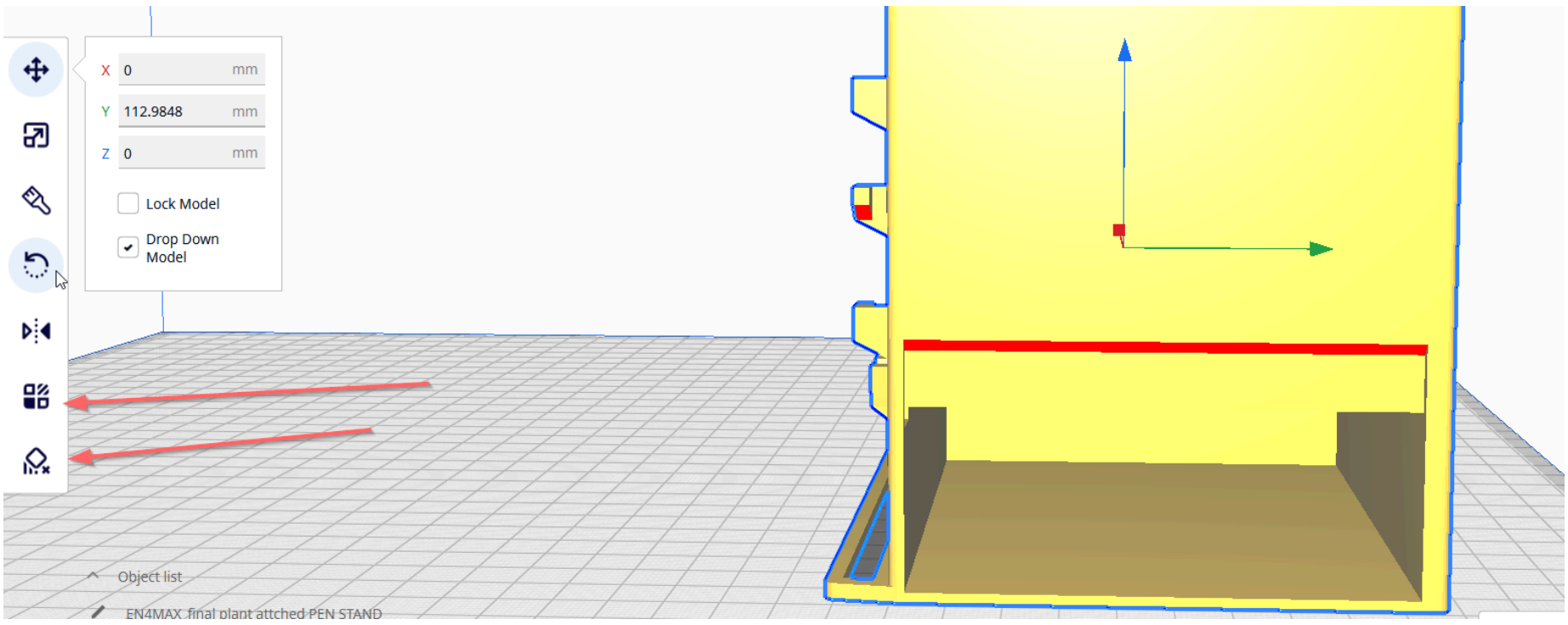
These sliders help you preview **each printing layer**.



Explanation:

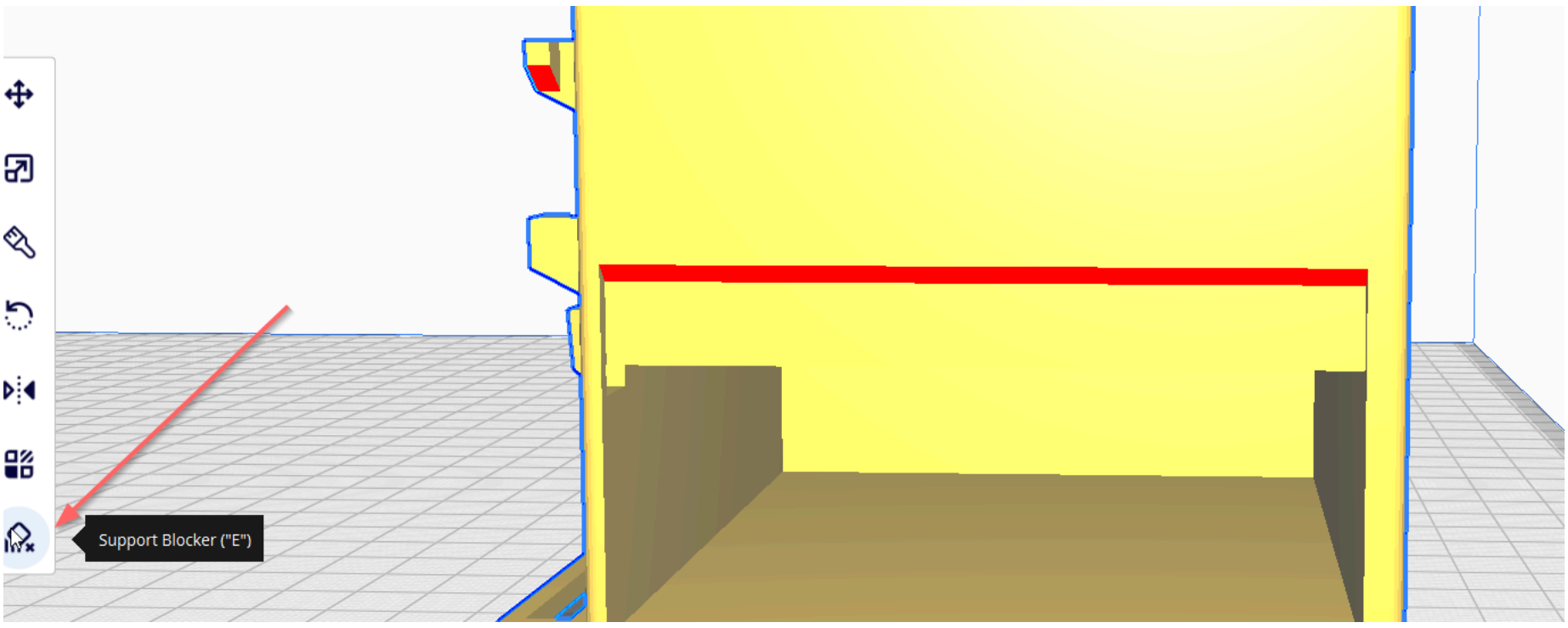
1. Shows the **infill pattern** with **50% infill**
2. Shows the **shell thickness (walls)**
3. Blue area shows **build plate adhesion**

Per Model Settings & Support Blocker



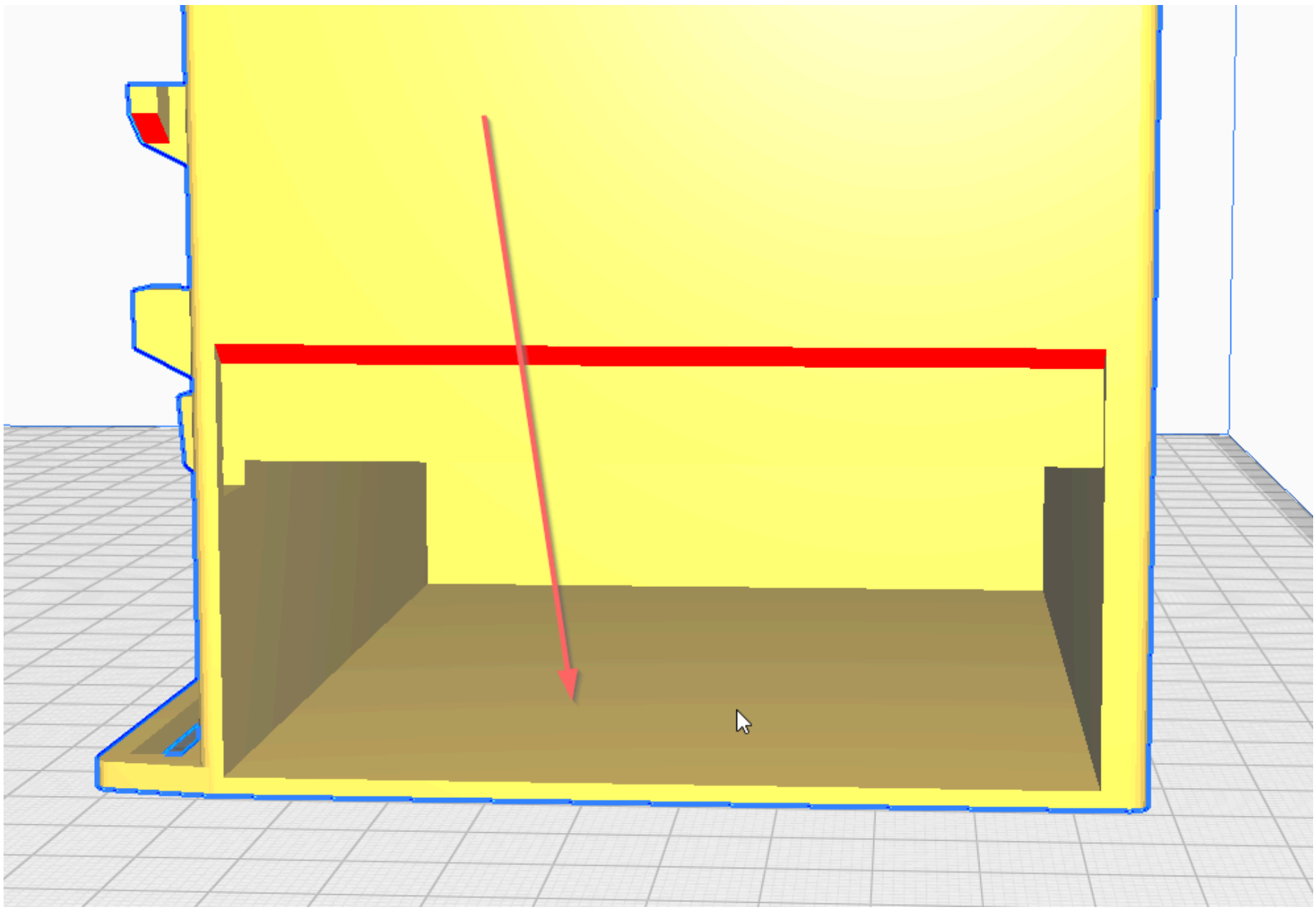
Now we will learn about **Per Model Settings** and **Support Blocker**.

Step 11: Select Support Blocker



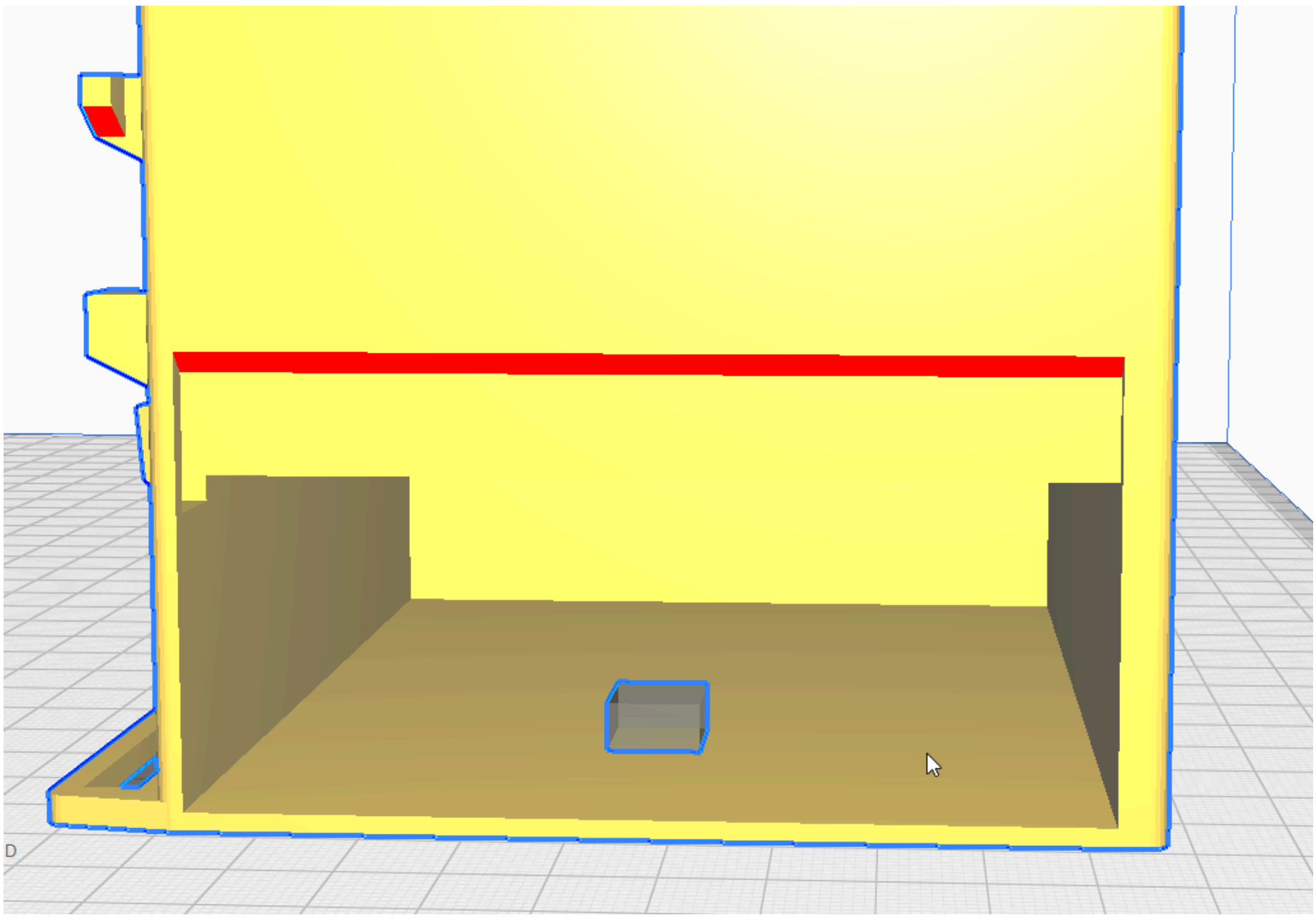
Click on **Support Blocker**.

Step 12: Place the Support Blocker



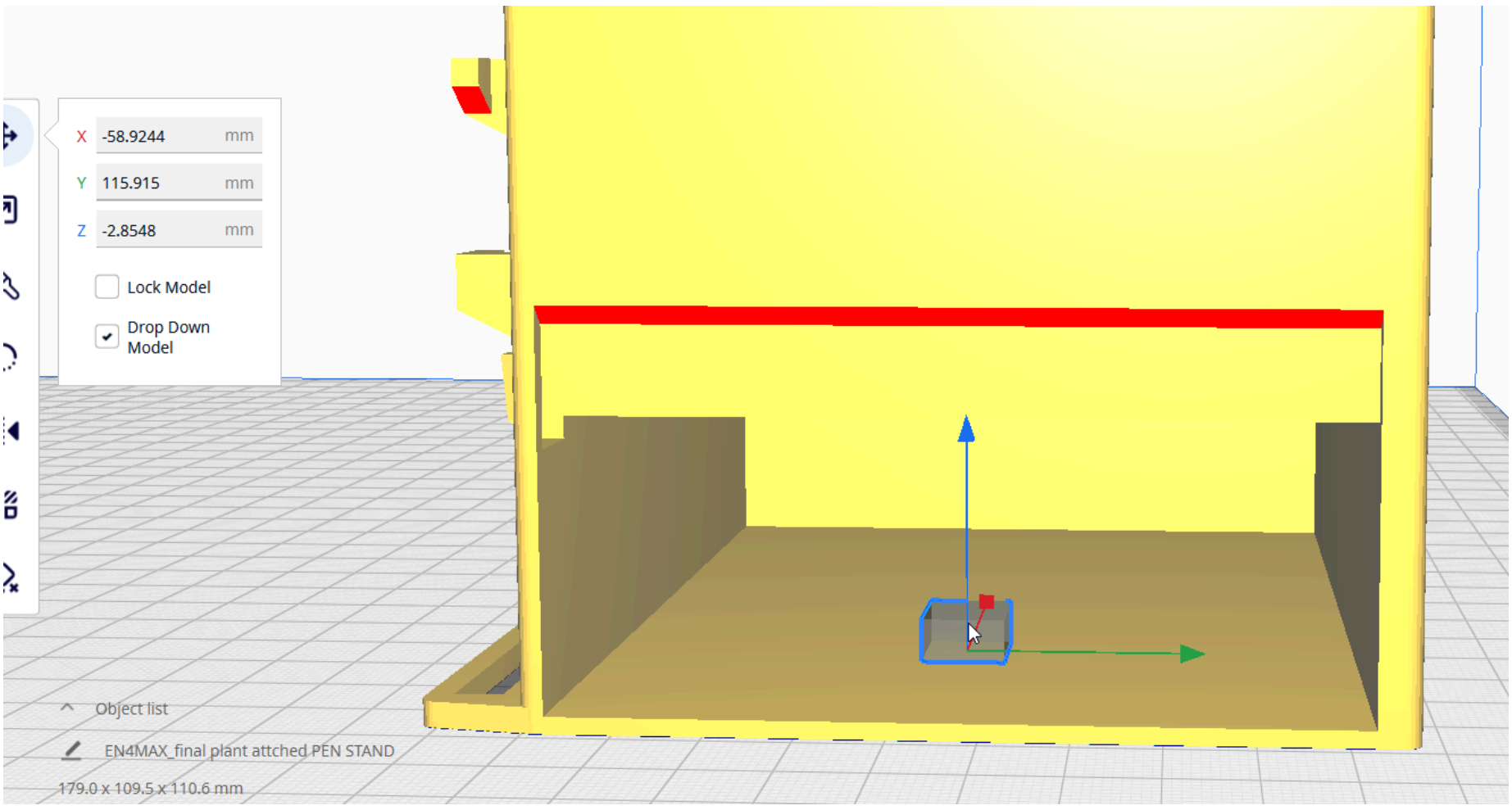
Click on the surface where you want to place the support blocker.

Step 13: Support Blocker Placed

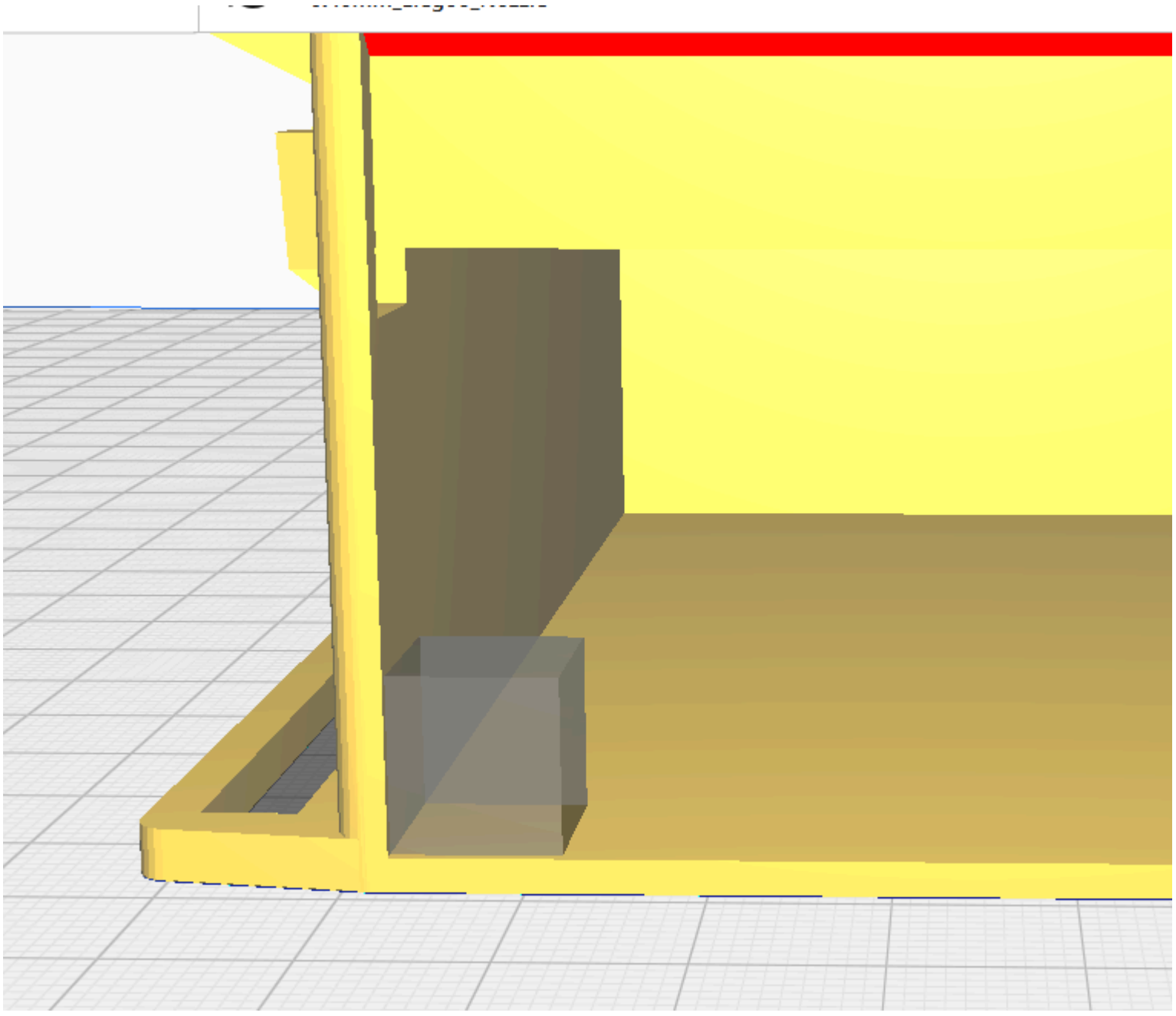


The support blocker is now placed.

Step 14: Move the Support Blocker

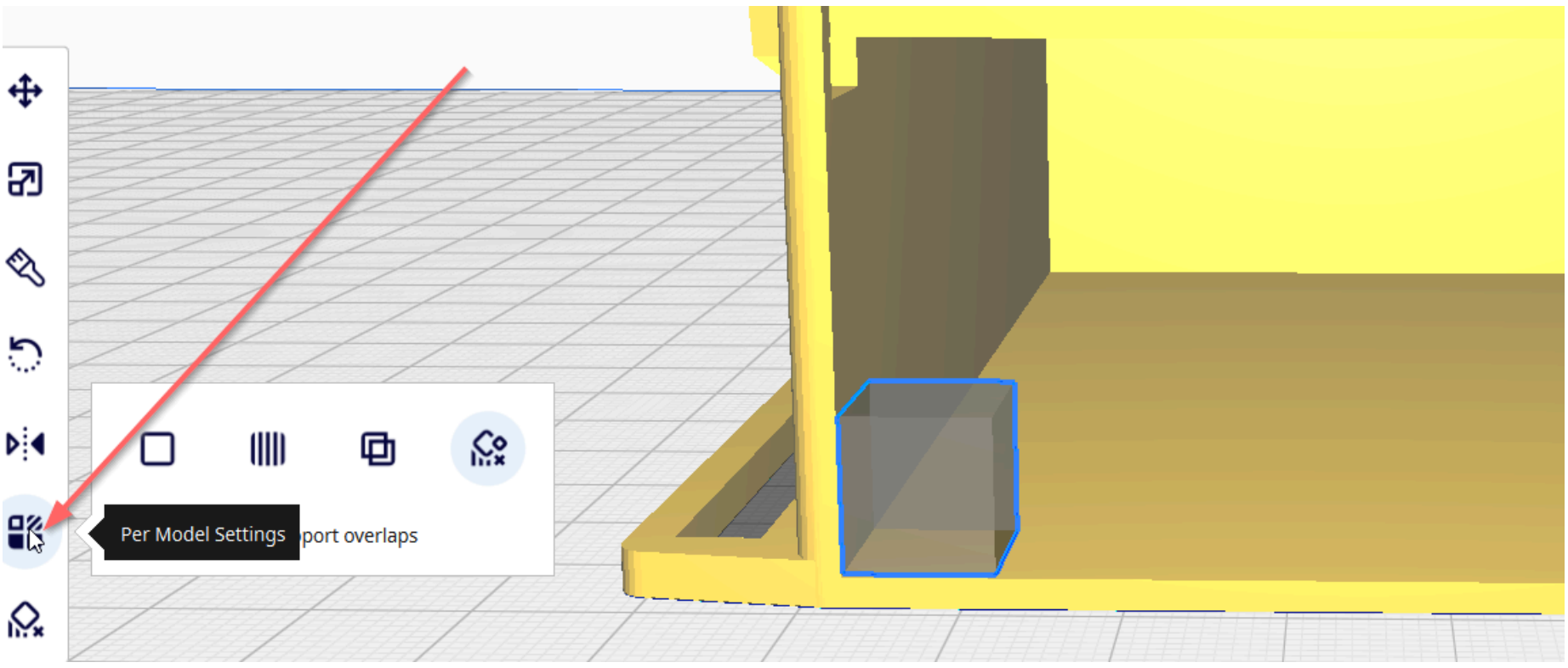


Select the block and move it.



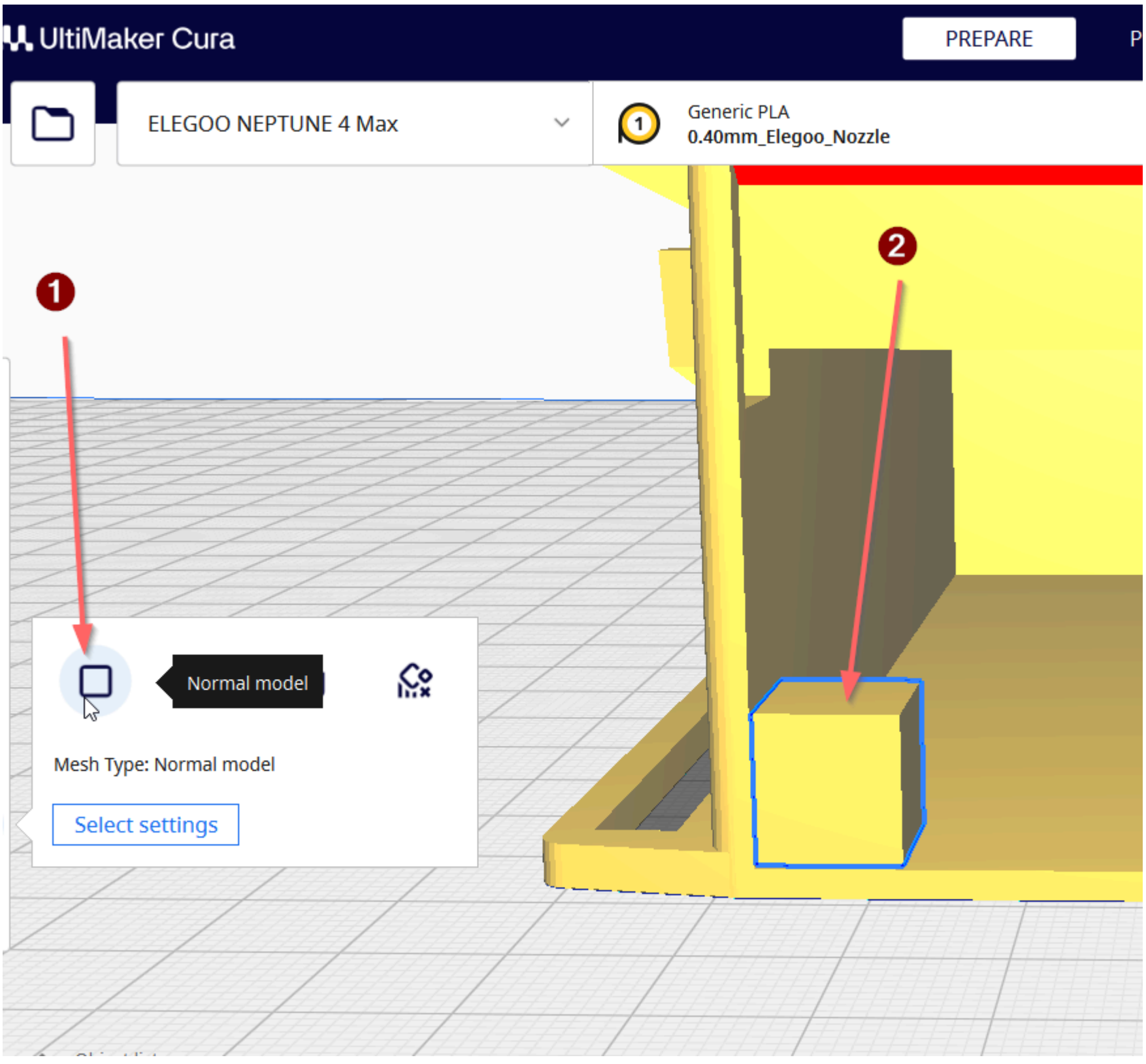
For example, I placed it here.

Step 15: Open Per Model Settings



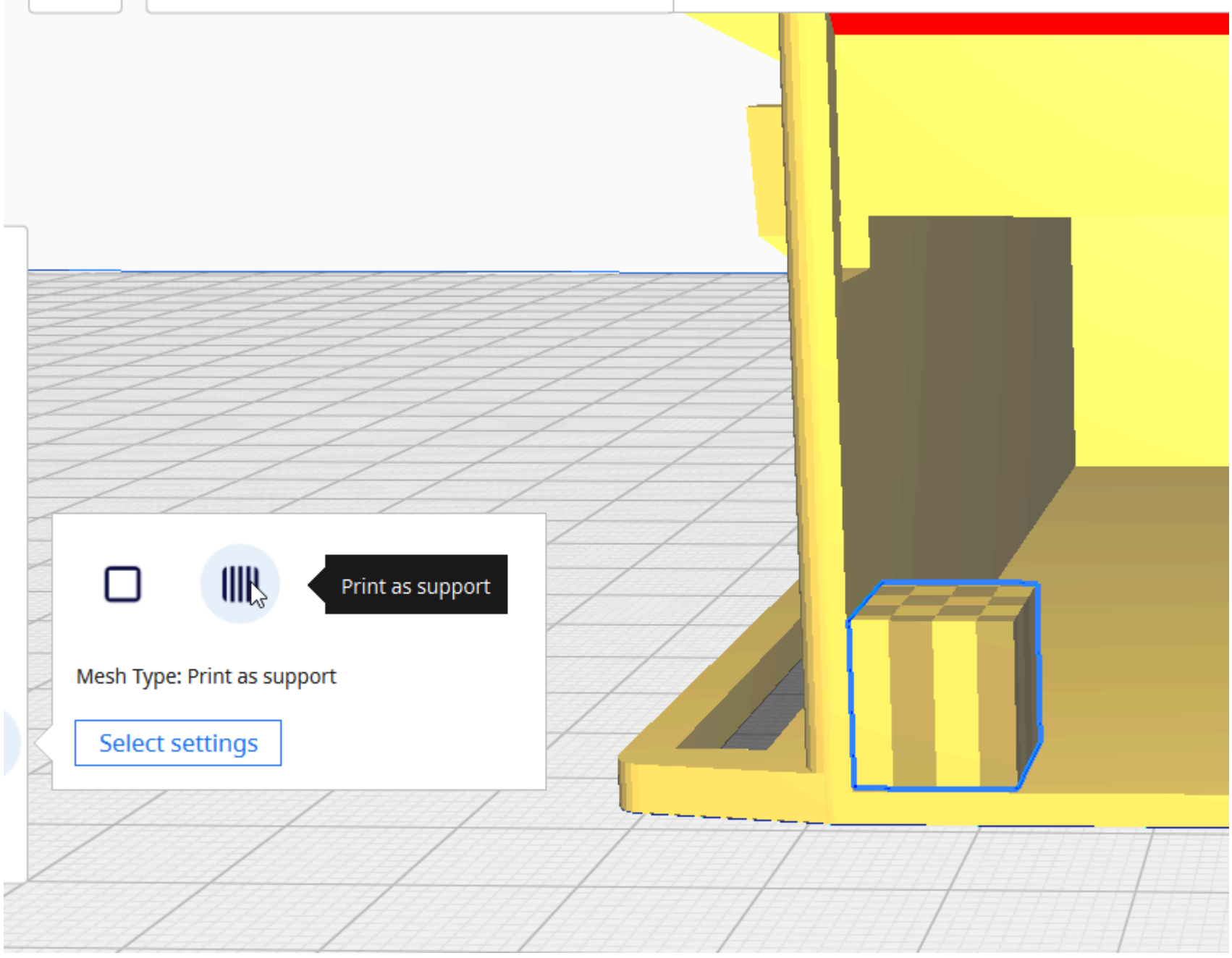
Select the block and go to **Per Model Settings**.

Step 16: Normal Model Option



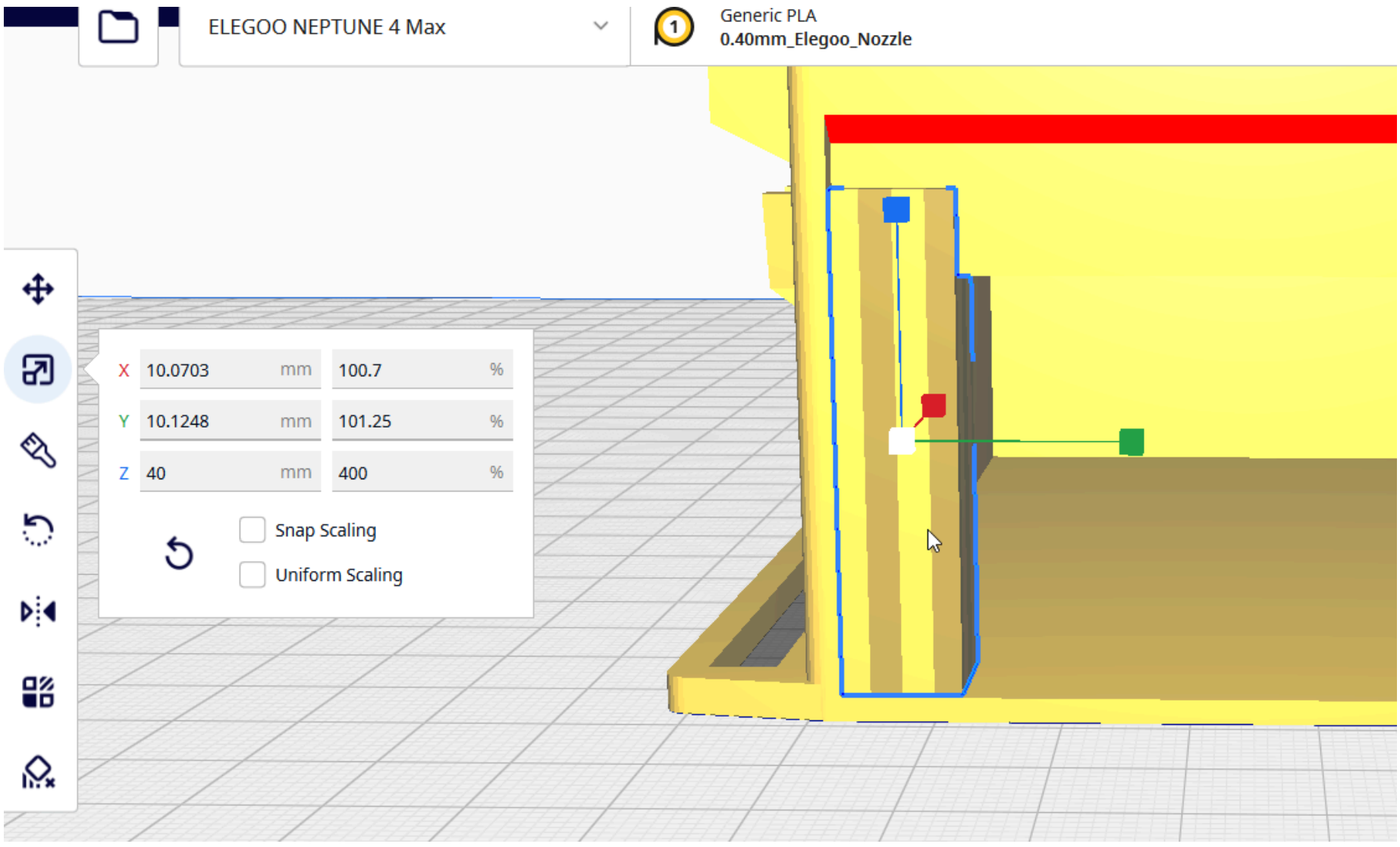
If **Normal Model** is selected, the block becomes a **solid model** and merges with the main model.

Step 17: Convert Block to Support



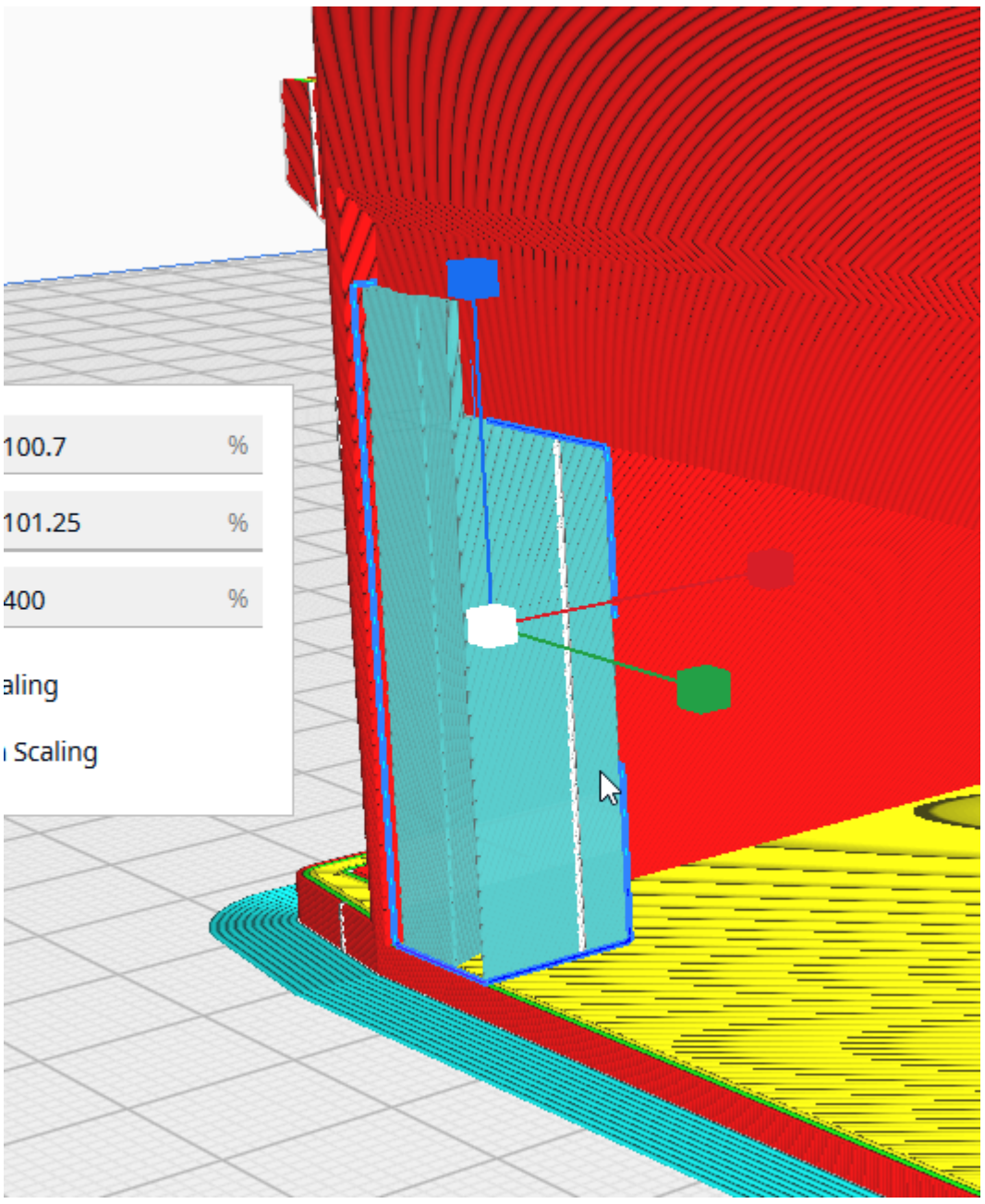
Change the block type to **Support** to add support **only where required**.

Step 18: Resize the Support Block



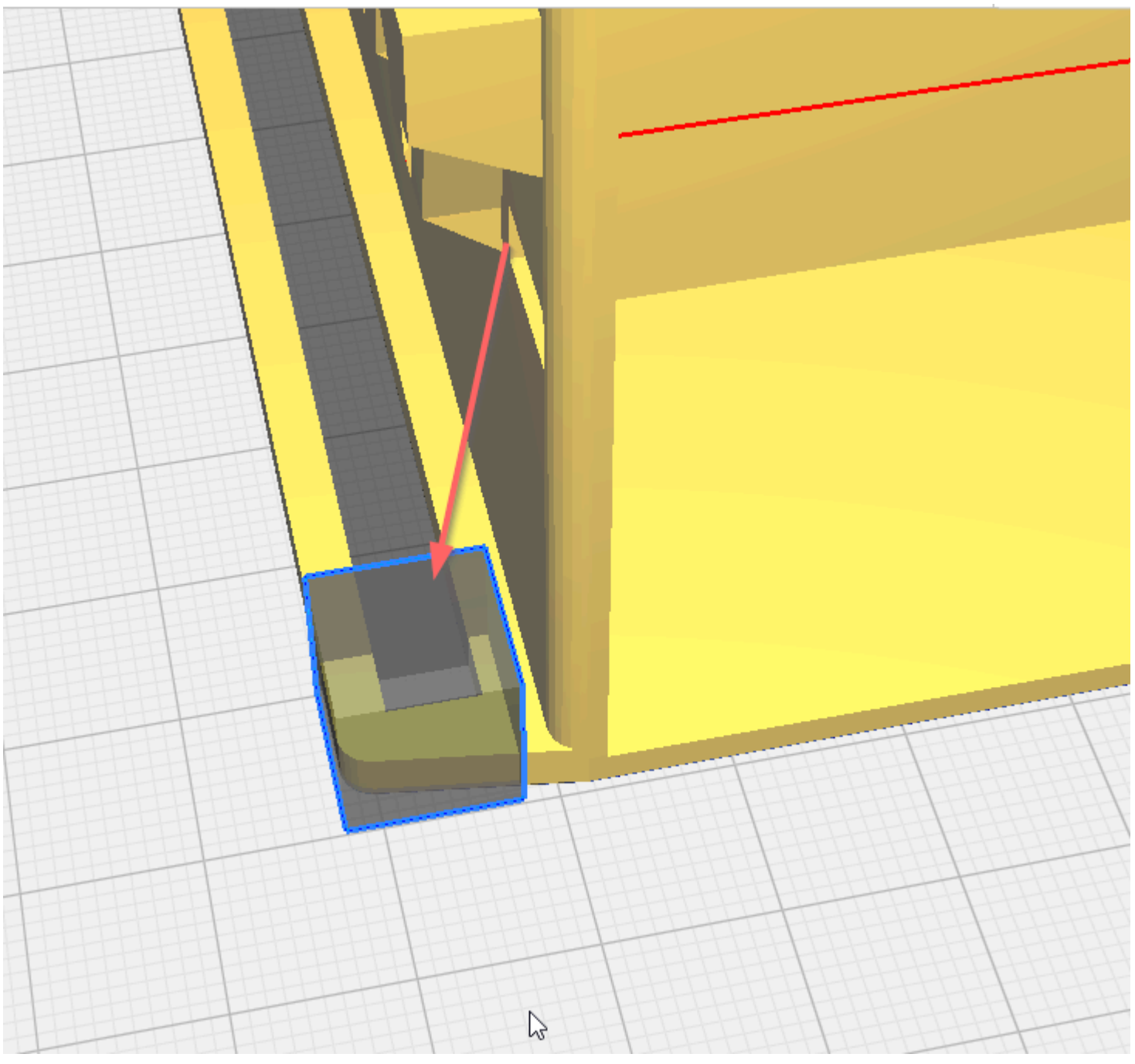
You can increase or decrease the **size of the support block**.

Step 19: Result After Slicing

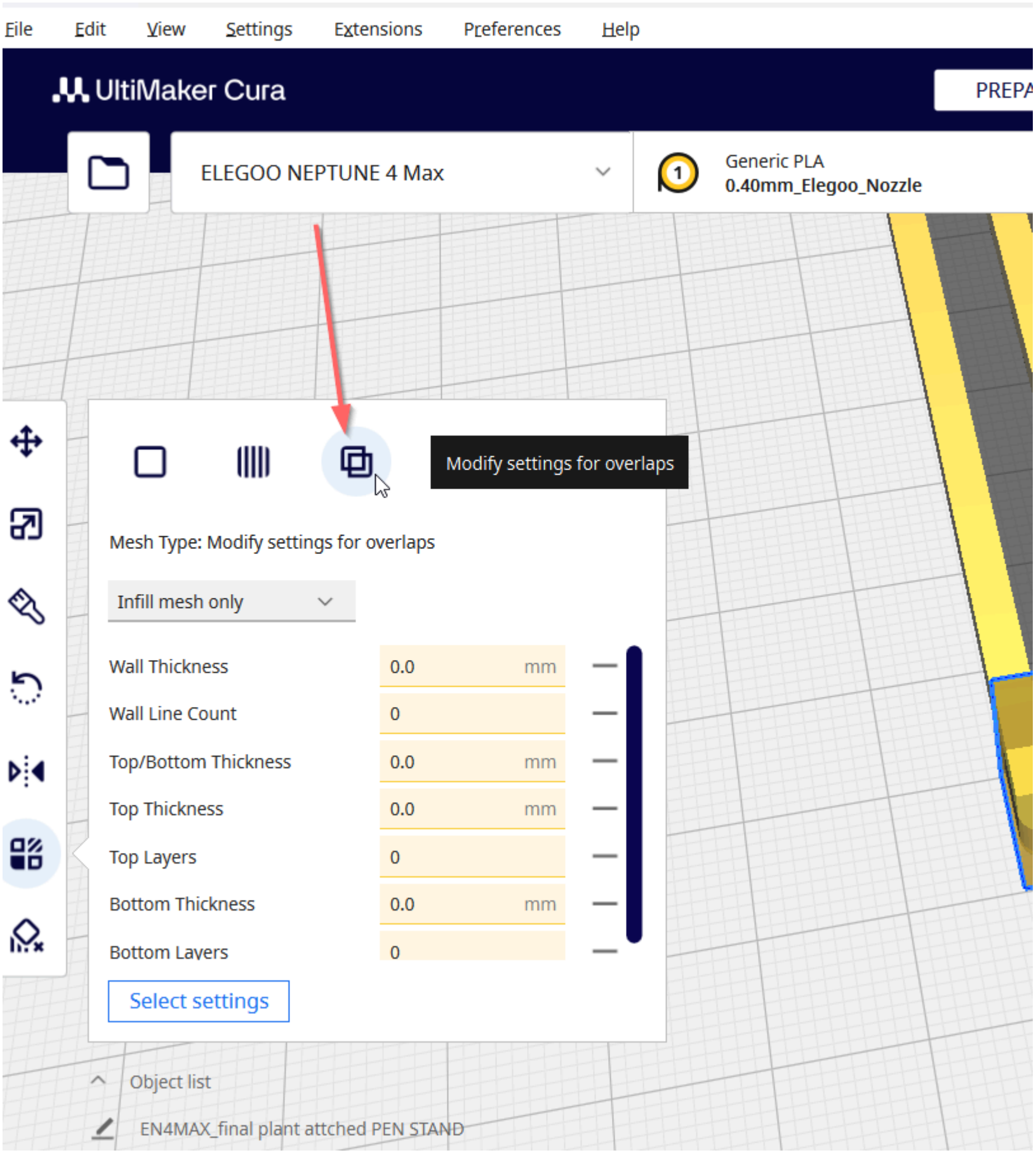


After slicing, the block remains as **support only**, which can be removed later.

Step 20: Advanced Use – Overlap Settings

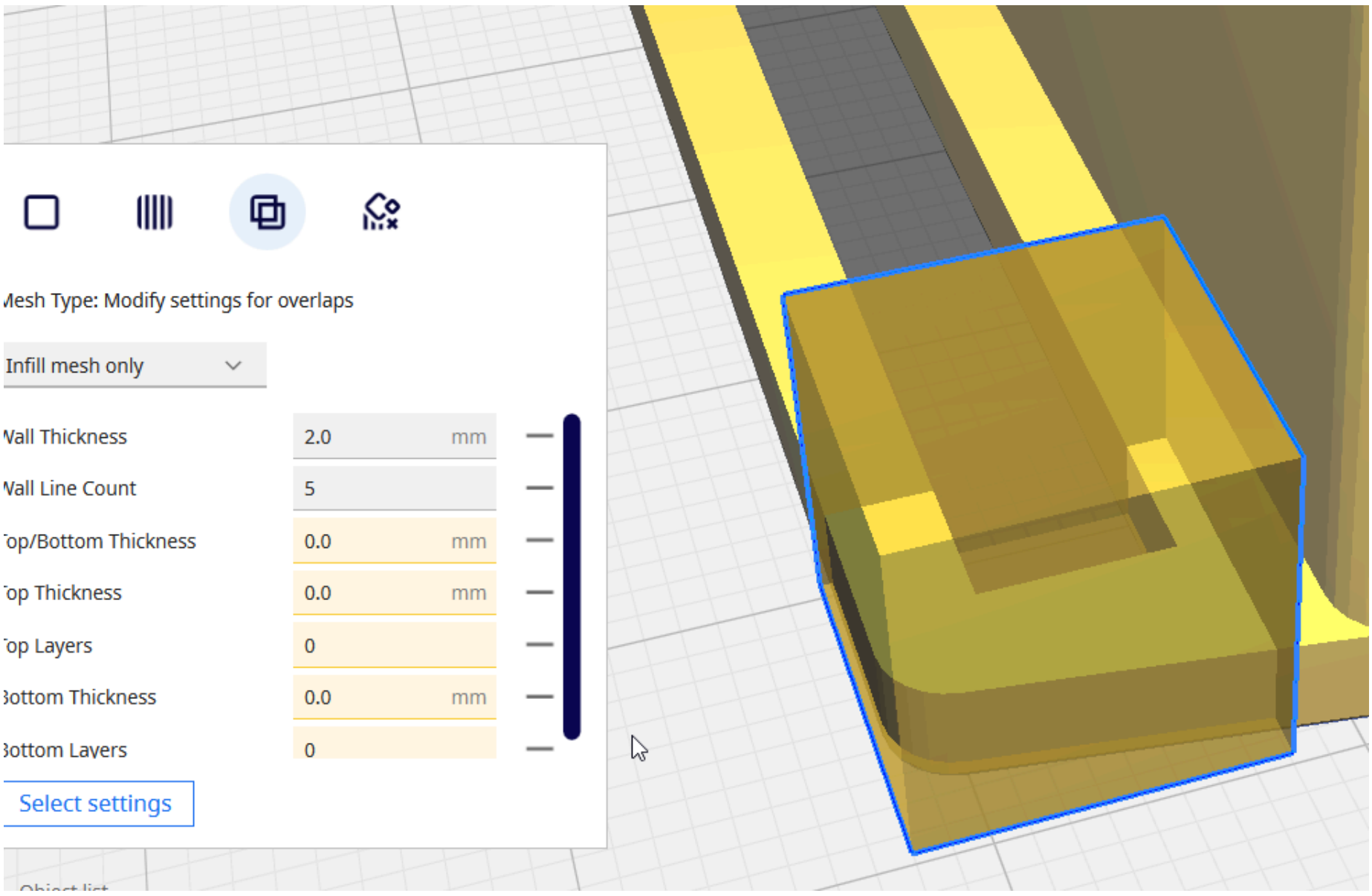


Support blockers can also be used for advanced control.



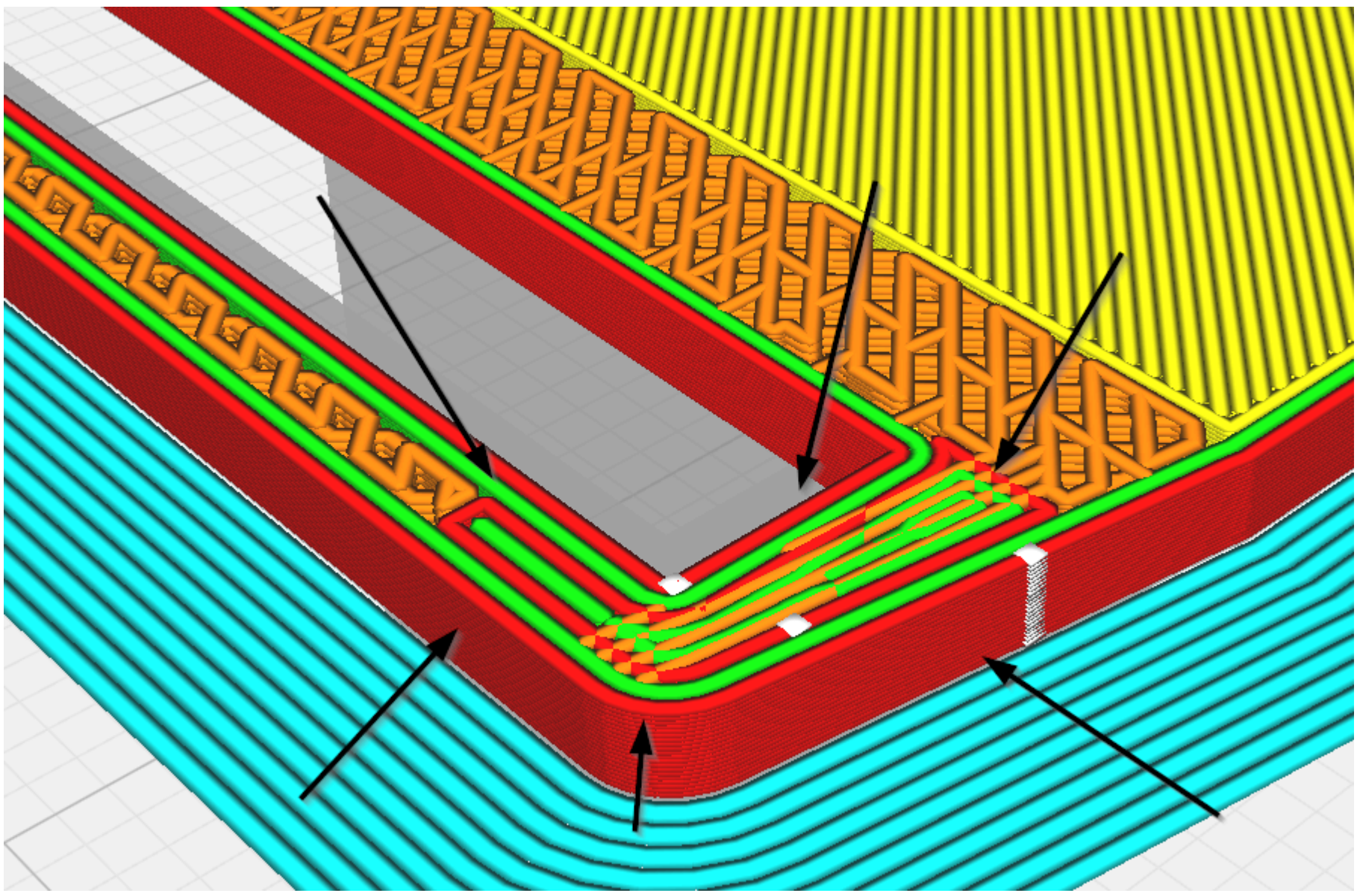
By selecting **Modify settings for overlaps**, you can apply special settings only to this region.

Step 21: Apply Custom Parameters



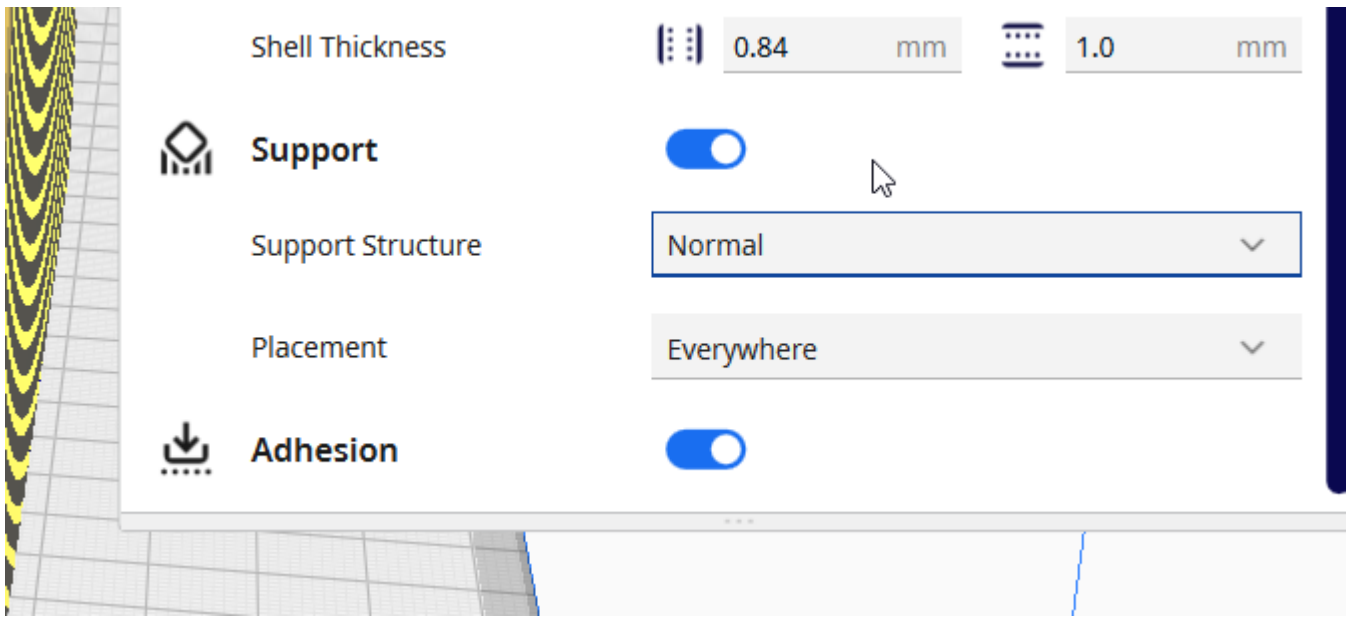
Custom parameters are added for this area.

Step 22: Result After Slicing



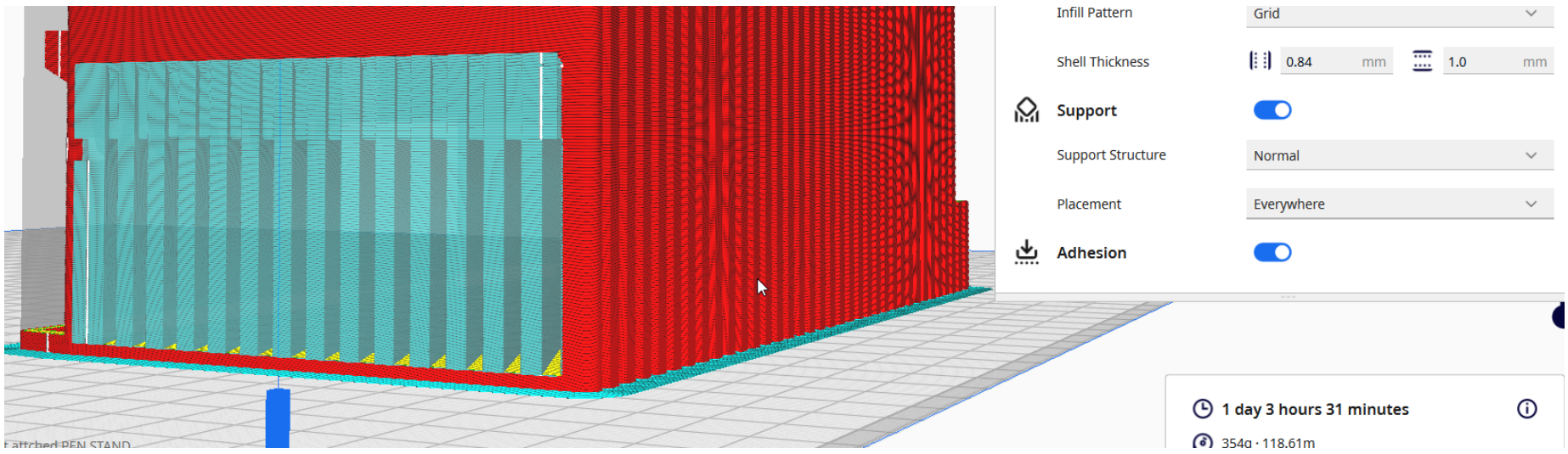
Only this **specific area** changes after slicing.

🧩 Support Options in Ultimaker Cura



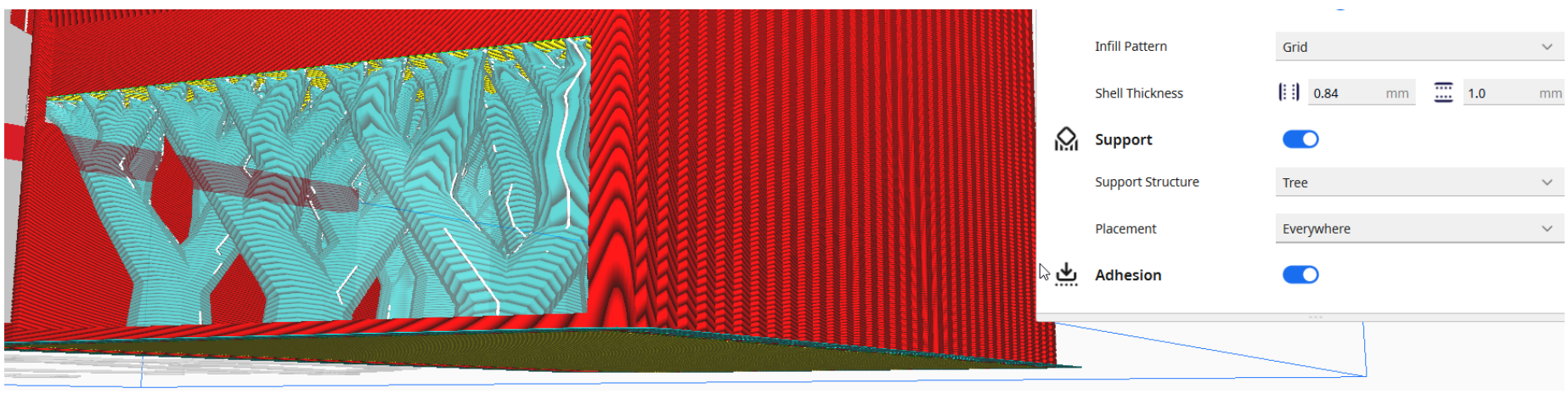
Some parts of a model require **supports** for proper printing.

Normal Support



Normal supports create straight support structures.

Tree Support



Tree supports:

- Use less material
- Are easier to remove
- Work well for complex shapes
