

Altair Inspire™

Tutorial: Optimizing Lattice

Generate a PolyNURBS from an optimized shape, then run and review a lattice optimization.

In this lesson, you will:

- Run an initial topology optimization
- Generate a PolyNURBS from an optimized shape
- Set the element size for the new design space
- Run a lattice optimization
- Review lattice results

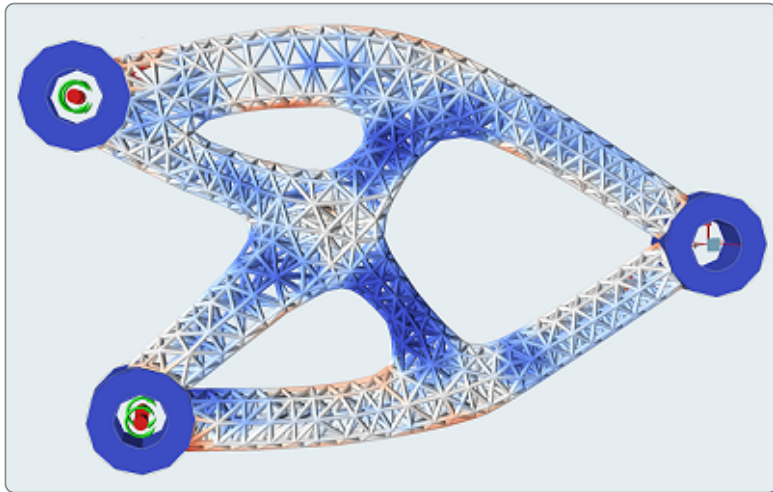


Figure 1. Lattice Optimization Results

Step 1. Overview

Lattice structures typically consist of different types of cells. In Altair Inspire, each beam can be optimized, filling your design space with an optimized lattice structure rather than repetitive patterns. It only works on solids, and design spaces have to be separated with non-design space.

While unique to 3D printing, lattice structures bear several desirable characteristics from a design perspective. Due to the large network of structural members, optimized lattice designs tend to exhibit better stability and more desirable thermal behavior. They also have desirable weight characteristics, and are used as an approach to target weight reduction. Lattice designs are particularly well-suited to biomedical applications such as implants, as the porous nature of the structure facilitates bone and tissue growth.

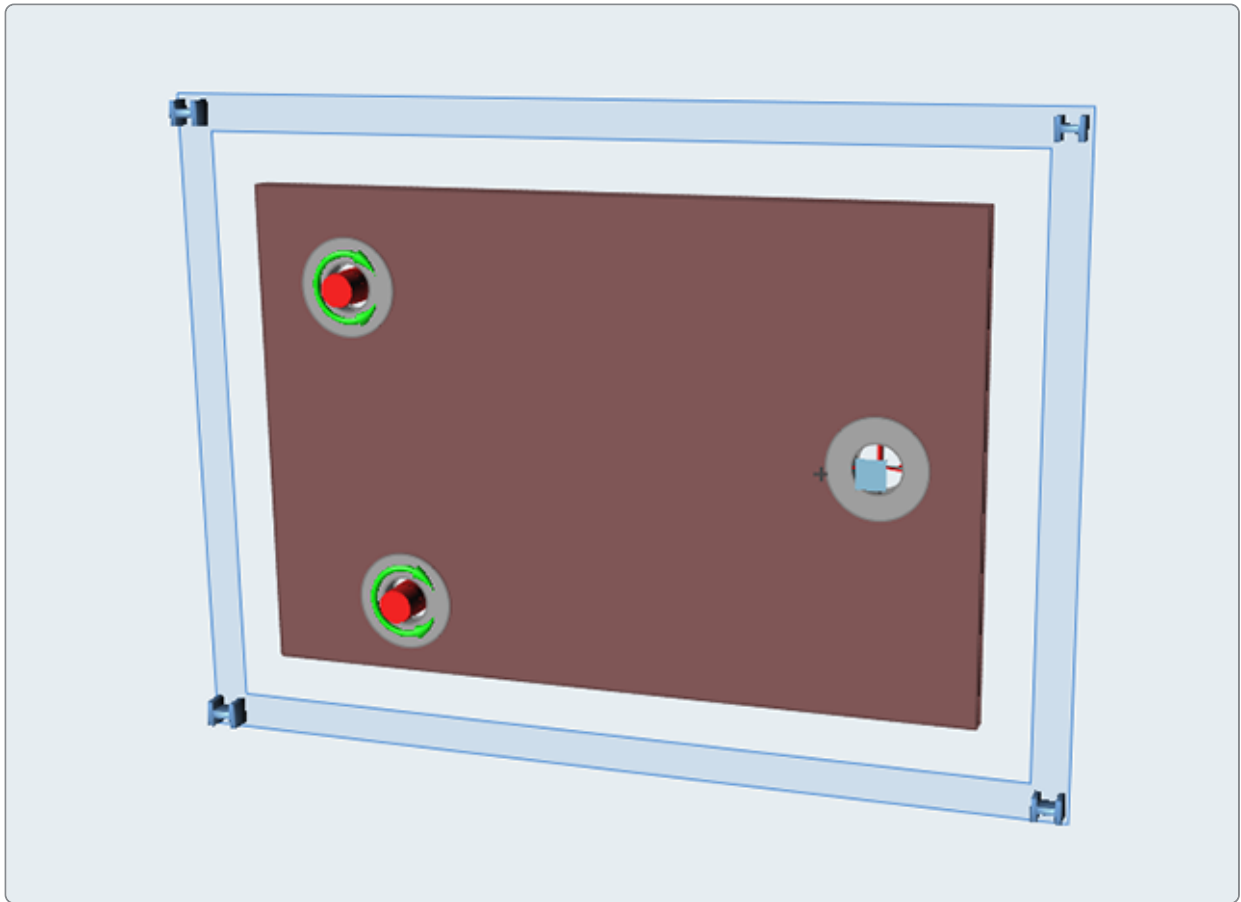
A lattice structure occupying the same design space as a solid structure will be less stiff and have higher stresses. For this reason, it is often necessary to set your design requirements more conservatively than you normally would for a traditional topology optimization. It is not uncommon for displacements and stresses to be five to ten times greater in a lattice structure compared to that of a solid structure occupying the same region. As it is not always possible to have an accurate estimate of the degradation, it may be necessary to start the optimization with increasingly strict constraints before a desired result from the lattice optimization is obtained.


Step 2. Run Topology Optimization

First, let's run an initial topology optimization on the design space of our example model. We'll use a stress constraint five times the desired performance for the final design, as the lattice occupying the optimal topology will have increased stress and displacement.

1. Press F7 to open the Demo Browser.
2. Double-click the `1.0_hanger_topology.stmod` file to load it in the modeling window.

Note that it is set up with a maximum displacement of 0.15 mm at the load application point. This will become important in the next step.



3. Make sure the display units in the **Unit System Selector** are set to **MPA (mm t N s)**.
4. Click **Run Optimization**  on the Optimize icon on the Structure ribbon.



5. Set the options in the **Run Optimization** window as shown below:


Run Optimization [Close]

Name:

Type:


Objective:

Stress Constraints

 ☐ None

☒ Minimum safety factor:

Frequency Constraints


 ☒ None

☐ Maximize frequencies

☐ Minimum:

Use supports from load case:


Thickness Constraints

 Minimum:

☐ Maximum:

Speed/Accuracy [v]

Contacts [v]

 ☒ Sliding only

☐ Sliding with separation

Gravity [v]

Load Cases [v]

6. Select **Topology** for the optimization **Type**.

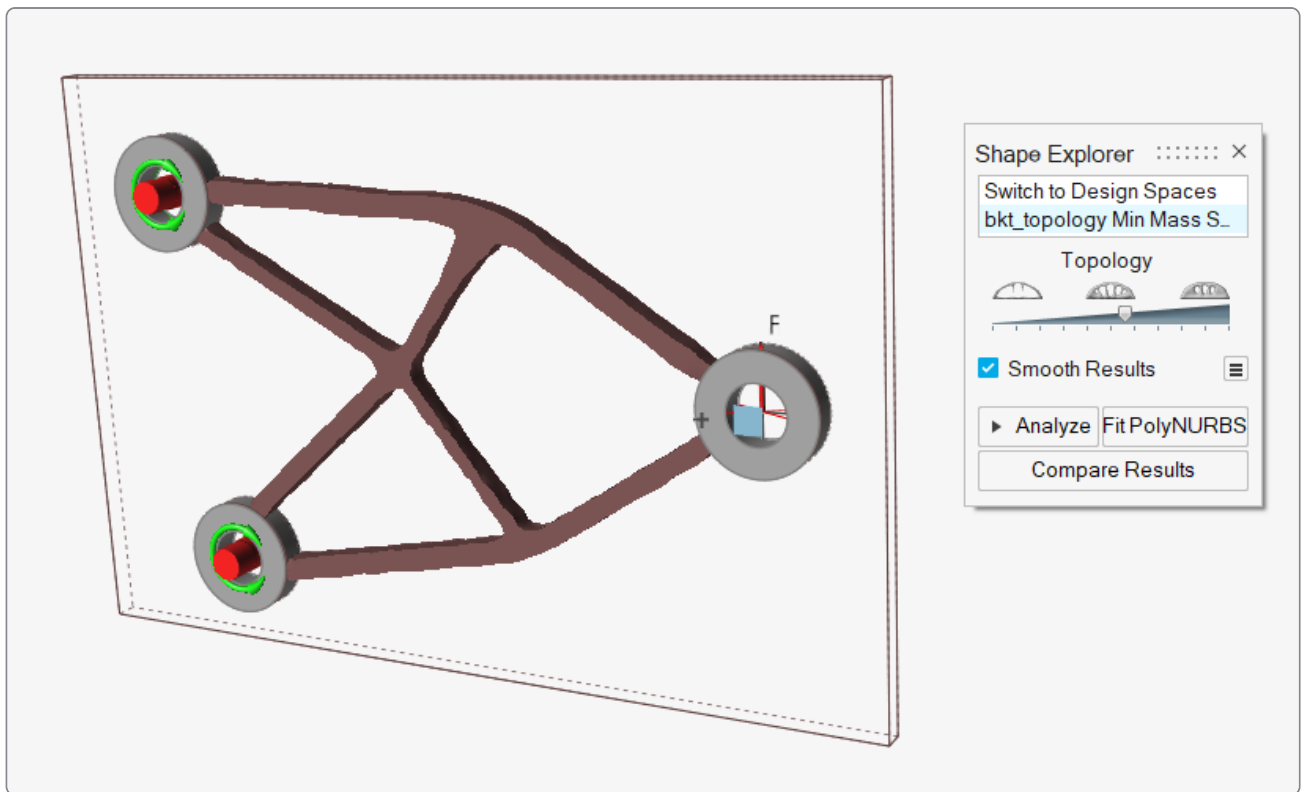
7. Select **Minimize Mass** for the optimization **Objective**.

8. Under **Stress constraints**, set the **Minimum safety factor** to 5.0 .

9. Enter a **Minimum Thickness Constraint** of 15 mm .

10. Click **Run**.


- When the run is complete, double-click the run name to view the results. (Alternatively, you can press F7 and double-click the 1.1_hanger_topology_run.stmod file in the Demo Browser to load a file that includes the results.)

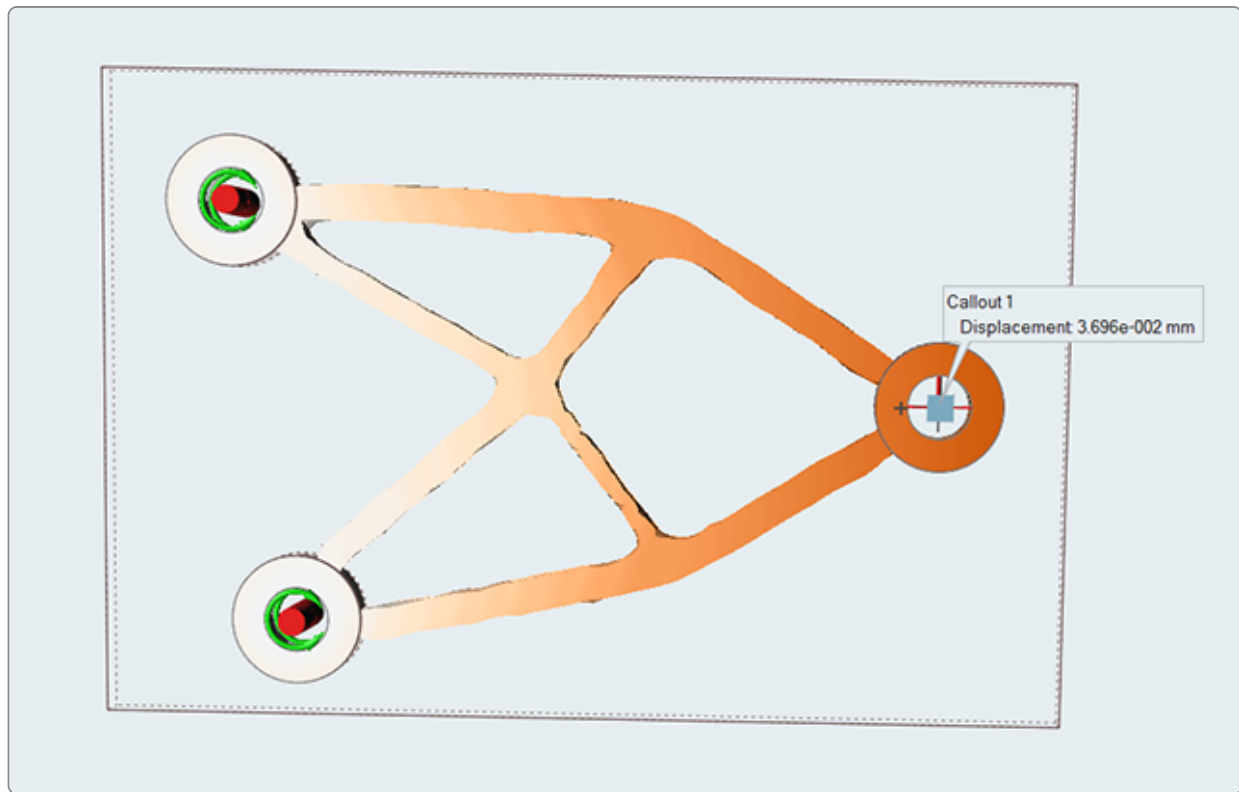


Step 3. Run Analysis on the Optimized Result

Next, reanalyze the optimized result at different thresholds until the displacement at the load point is less than 0.02 mm (a fifth of the ultimate target).

- Click the **Analyze** button on the Shape Explorer.
- When the run is complete, double-click the run name to view the results.
- View the **Displacement** result type.

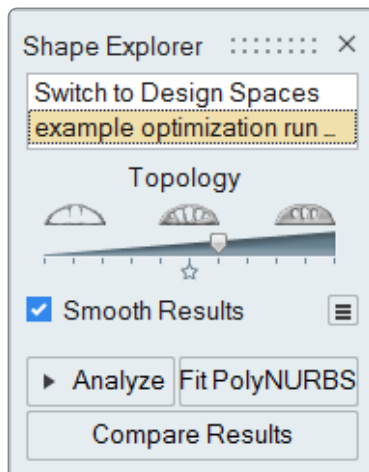
4. Click the  icon under Callouts and place a callout on the load point.



5. The displacement is greater than 0.02 mm, so click the **Show Optimization Results** icon to return to the initial optimized result.

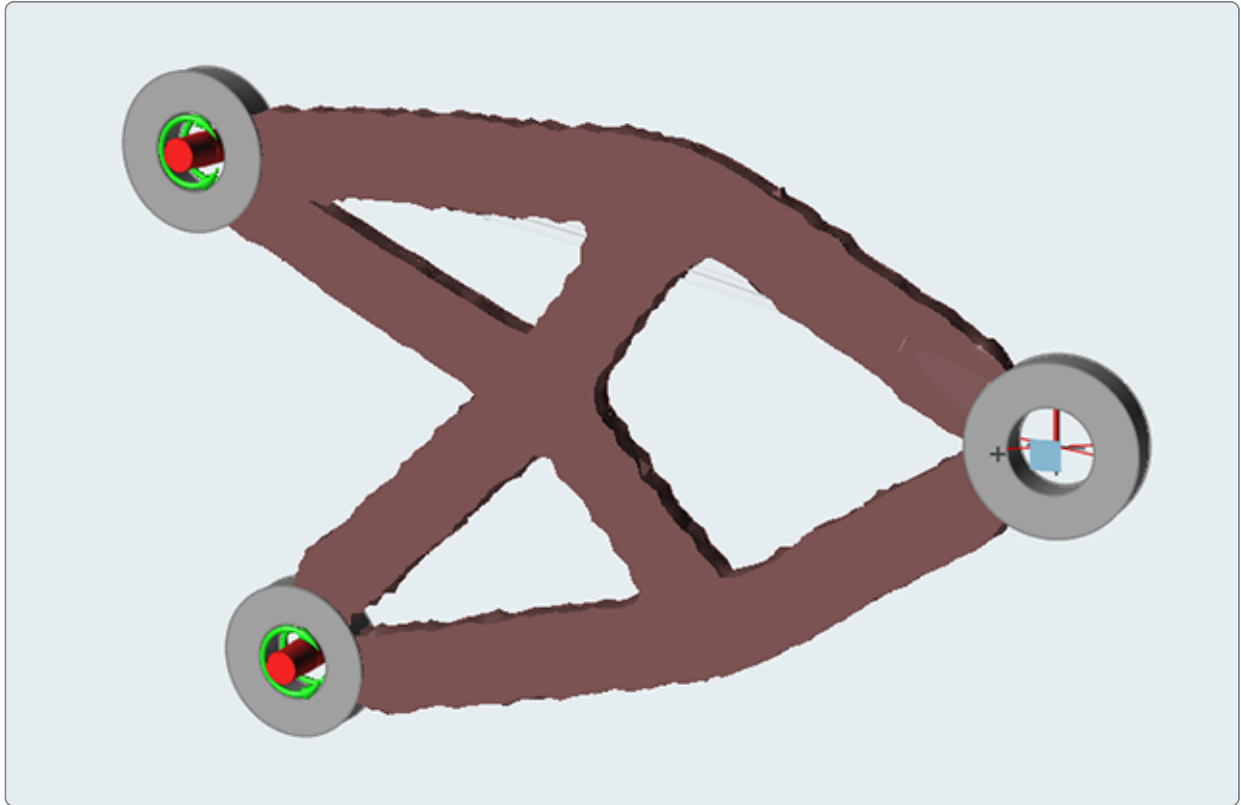


6. On the Shape Explorer, slide the topology slider to the right to increase the threshold.



7. Click the **Analyze** button to run another analysis.
8. Repeat the process until the displacement at the load point is less than 0.02 mm.

9. Double-right-click to exit the Analysis Explorer.



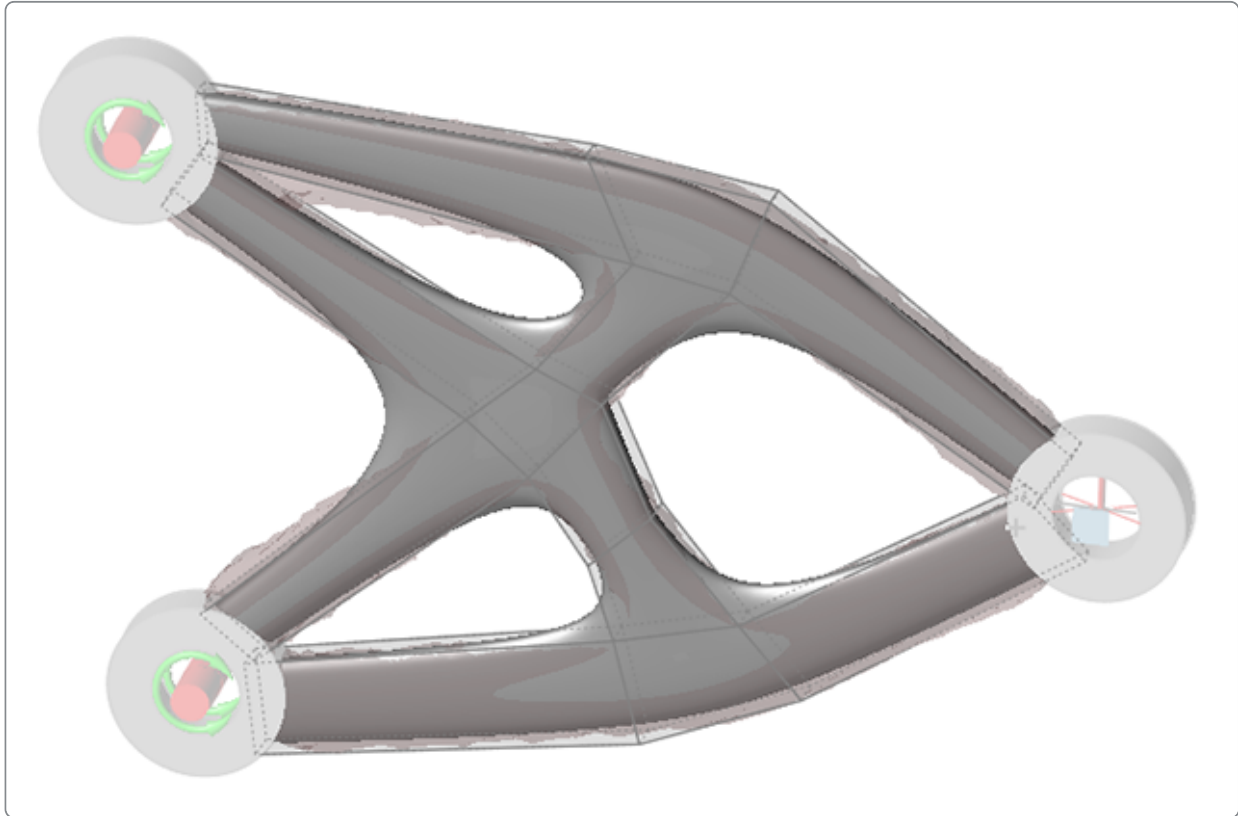
Step 4. Generate a PolyNURBS Part from the Optimized Result

Use the PolyNURBS tools to create a new design space based on the topology results generated. (To skip this step, load the 2.0_hanger_lattice_PN.stmod file from the Demo Browser and proceed to step 5.)

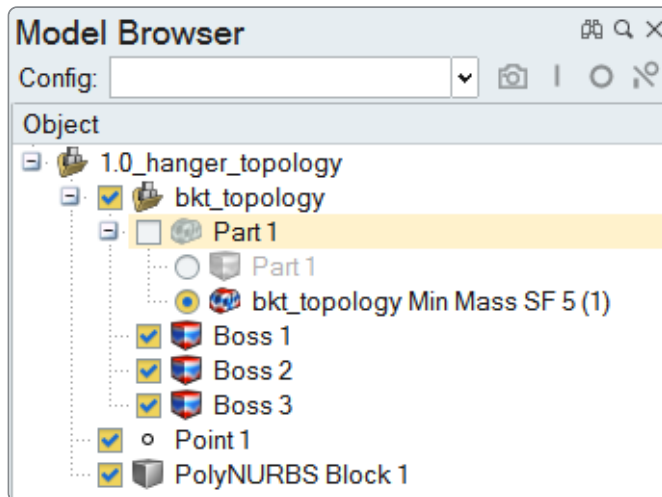
1. Select **PolyNURBS** on the geometry ribbon.



2. Use **Wrap** and the other PolyNURBS tools as needed to create a PolyNURBS part around the optimized shape.

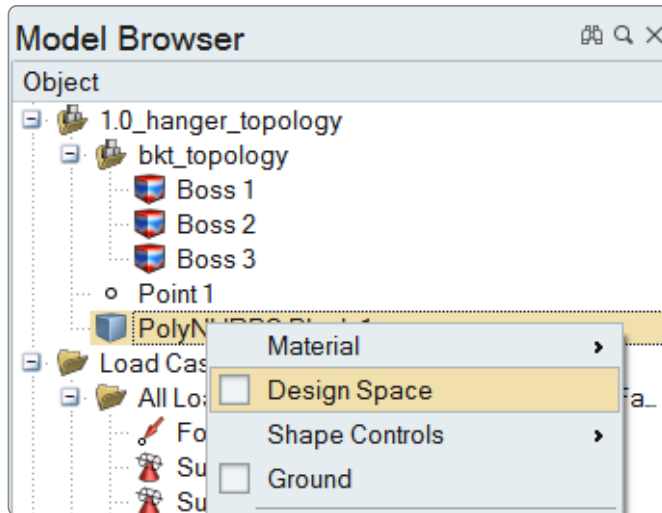


3. Press **F5** to open the Model Configuration toolbar.
4. Deselect the check box next to Part 1 in the Model Browser to configure the original design space off.

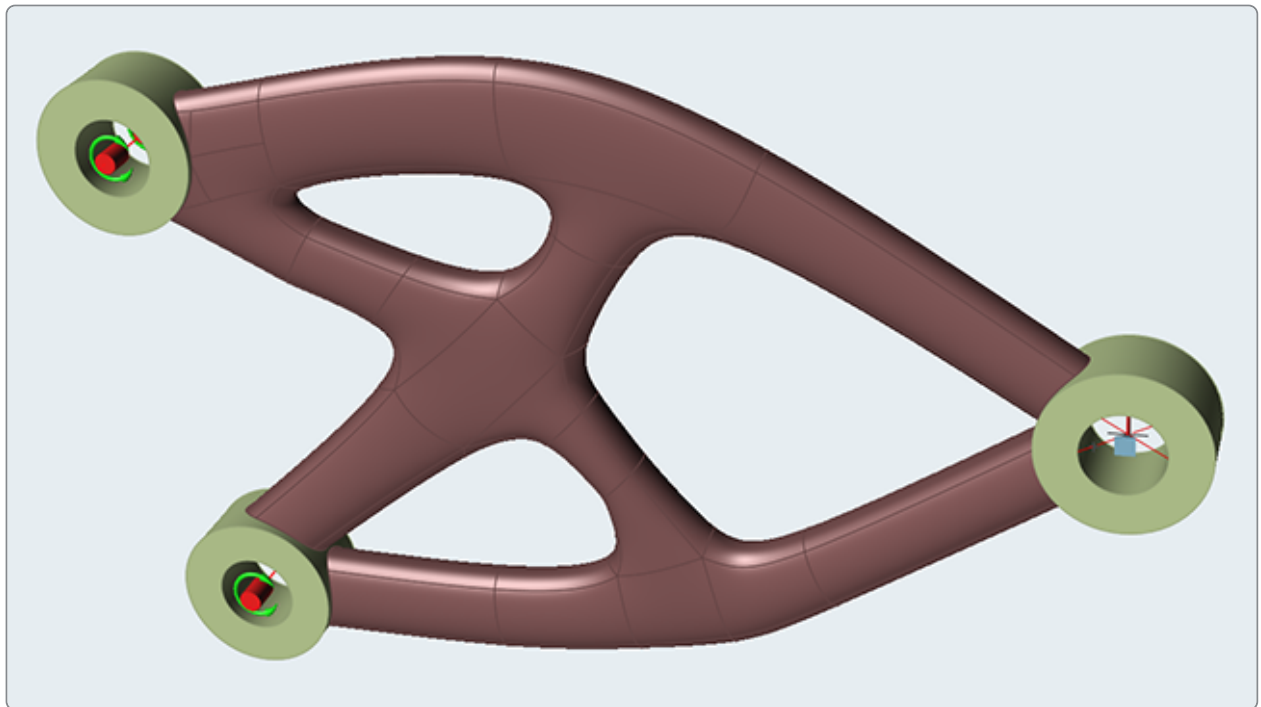


5. Press **F5** to close the Model Configuration toolbar.

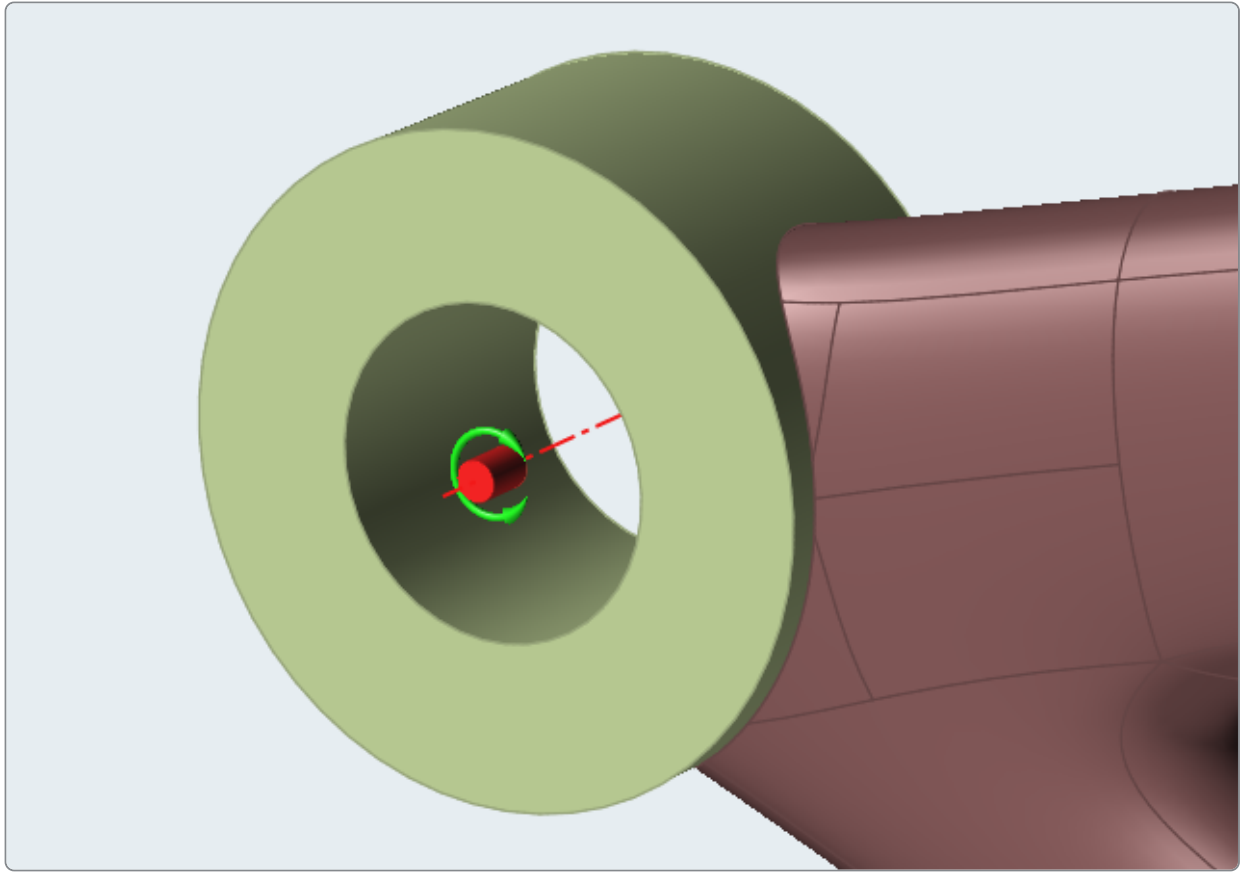
6. Right-click the PolyNURBS part and select Design Space from the context menu.



7. The PolyNURBS part that was fit to the baseline topology optimization results is now the new design space.



8. Note the slight offset of the design space from the non-design boss parts, which prevents the resulting lattice from protruding.

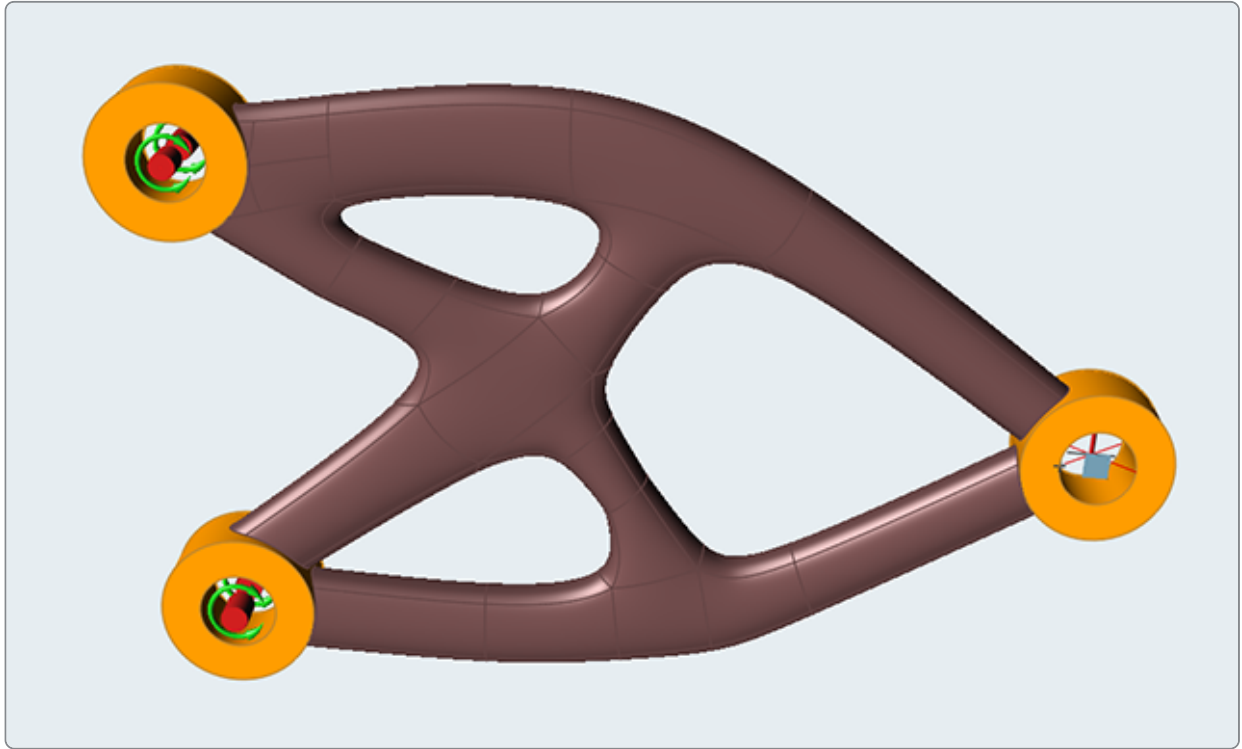


Step 5. Turn Off Autocalculate Element Size

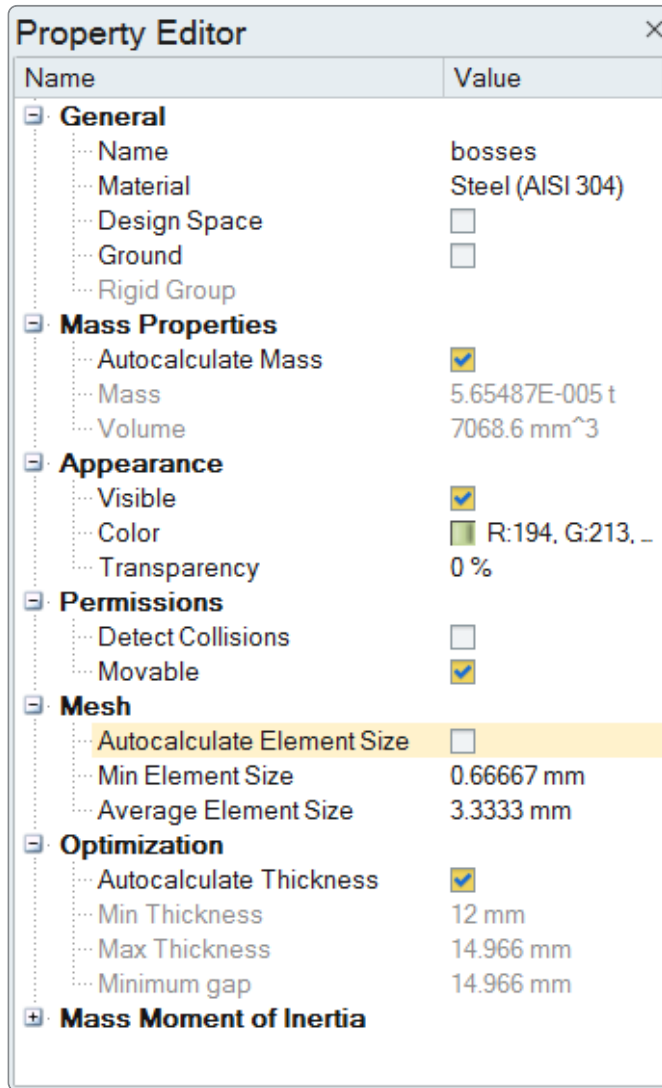
The target lattice length is often larger than the ideal element size for the non-design parts. So before running the lattice optimization, we'll turn off Autocalculate Element Size in the Property Editor.

1. Press **F3** to open the Property Editor.

2. Select the boss parts.



3. Deselect the **Autocalculate Element Size** check box in the Property Editor.



Step 6. Set Up and Run Lattice Optimization

Now we'll run a lattice optimization. (To skip this step, load the 3.0_lattice_hanger_PN_RUN.stmod file from the Demo Browser and proceed to step 7.)

1. Click **Run Optimization** on the Optimize icon on the Structure ribbon.



2. Set the options in the Run Optimization window as shown below:



Run Optimization


Name:


Type:

Objective:

Lattice


 Target length: 

Minimum diameter: 


Maximum diameter: 

Fill with:

Stress constraints

 Minimum safety factor:

Frequency constraints

 ☒ None


☐ Maximize frequencies

☐ Minimum:

Use supports from load case:

Speed/Accuracy ▾

Contacts ▾

 ☒ Sliding only

☐ Sliding with separation

Gravity ▾

Load cases ▾

3. Select **Lattice** for the optimization Type.

4. Select **Minimize Mass** for the optimization Objective.

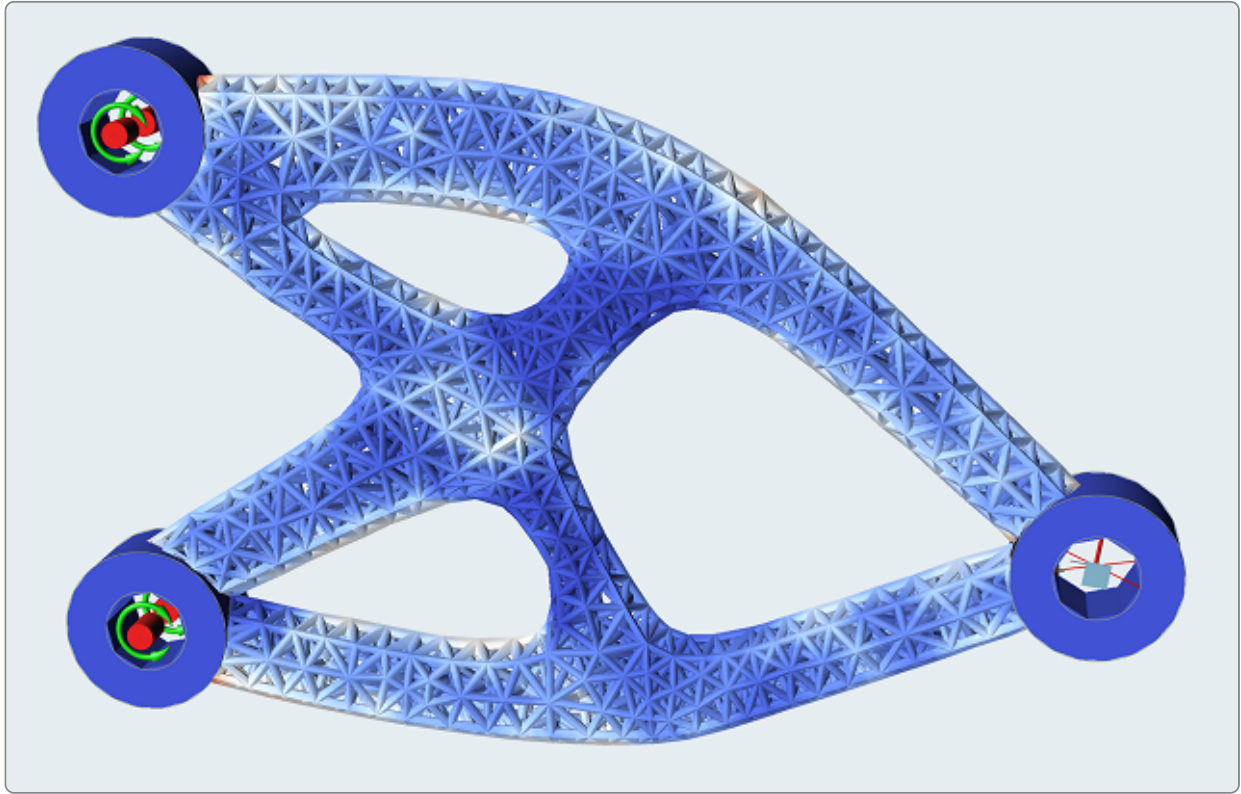
5. Under Lattice, set the **Target length** to 6 mm, the **Minimum diameter** to 1 mm, and the **Maximum diameter** to 3 mm.

6. Set the **Fill with** percentage to 100% Lattice.

7. Under Stress Constraints, set the **Minimum safety factor** to 1.0.

8. Click **Run**.


9. When the run is complete, double-click the run name to view the results.

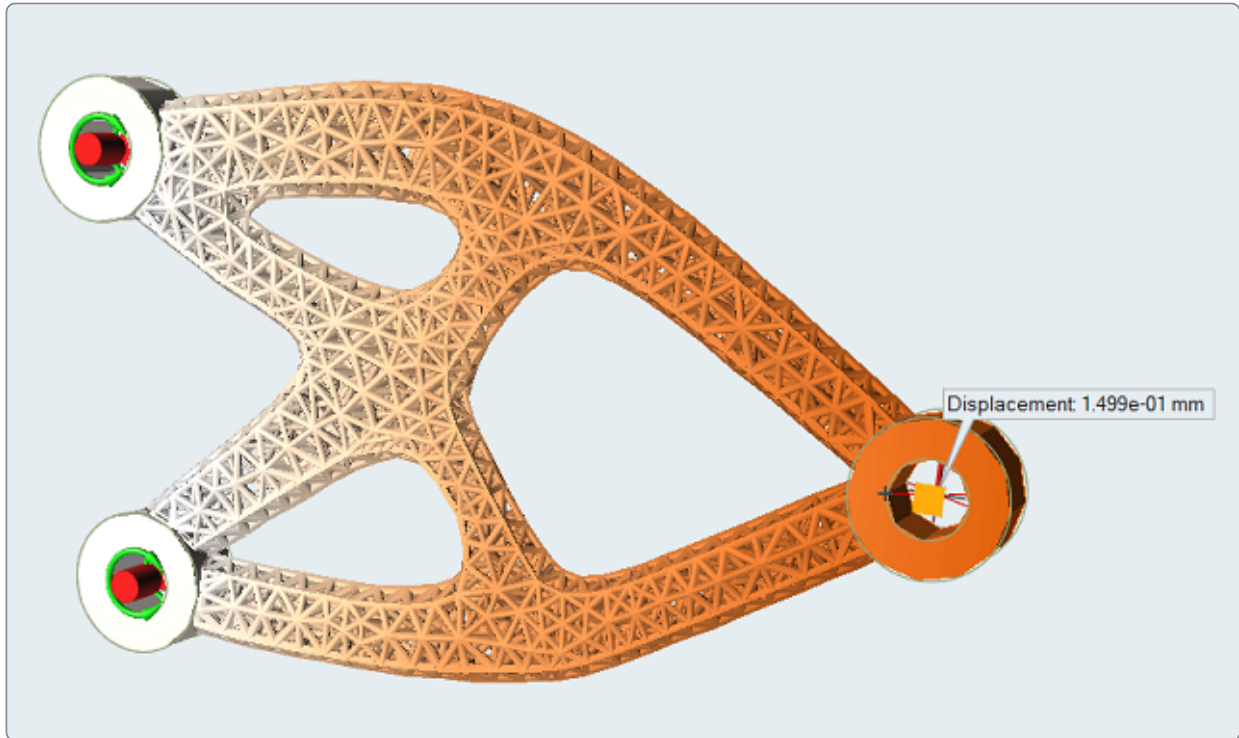



Step 7. Review the Lattice Optimization Results

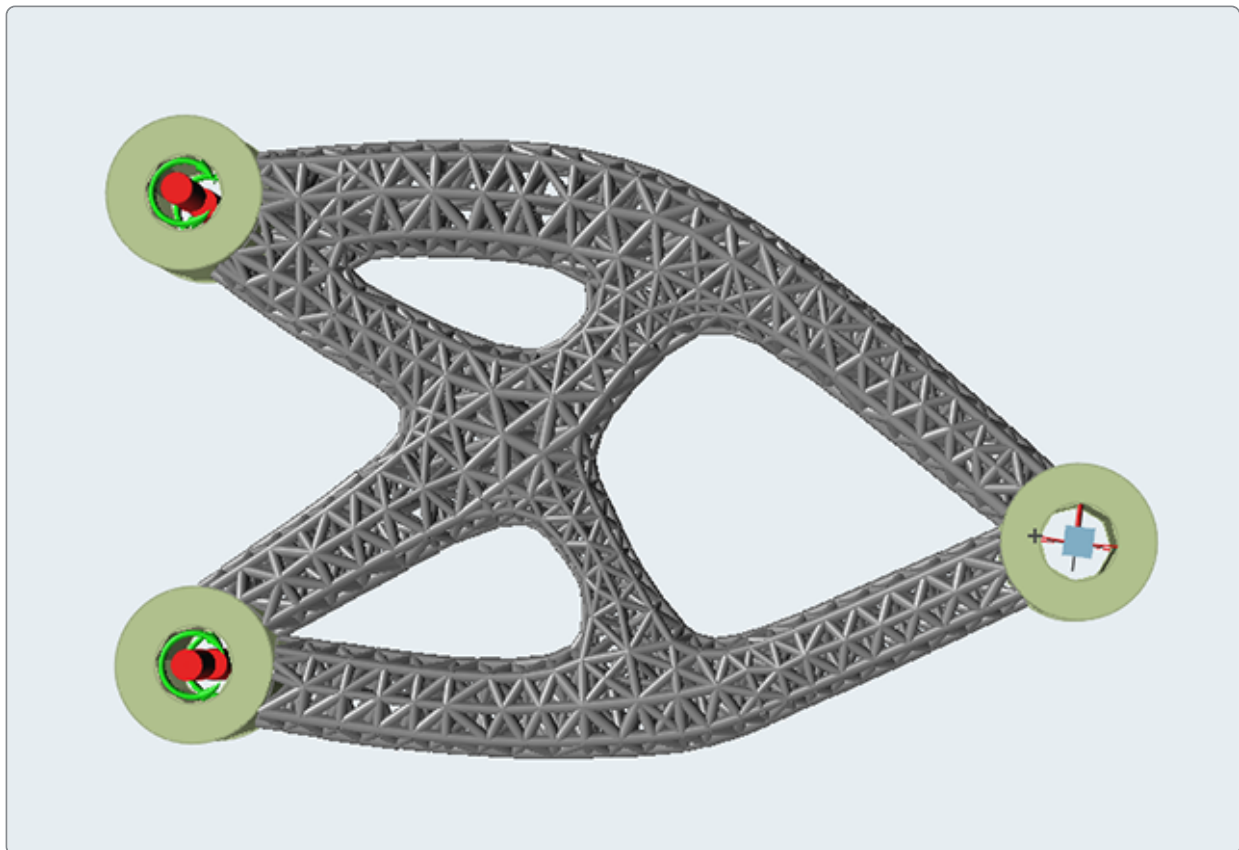
Review the results of the lattice optimization run in the previous step. Lattice optimization results appear in the Analysis Explorer rather than the Shape Explorer.


1. Examine the results for Factor of Safety. Note that the minimum is 2.0 (above our target of 1.0).
2. View the **Displacement** result type.

3. Click the  icon under **Callouts** and place a callout at the load point. Note the value of .1499 mm is within the constraint of 0.15 mm.



4. Click the  icon under Show on the Analysis Explorer to hide the contours.



5. Click the  icon and select **Smooth Lattice**. This displays an alternate visualization of the resulting lattice structure with radii between the lattice beams.

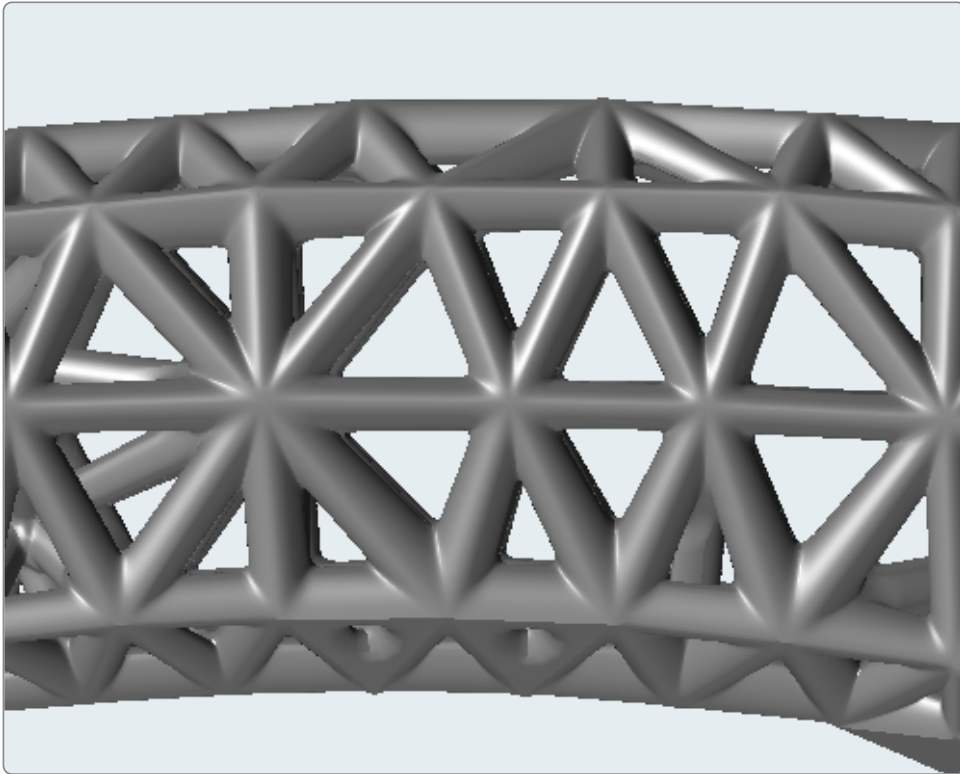


Figure 2. Smooth Lattice On

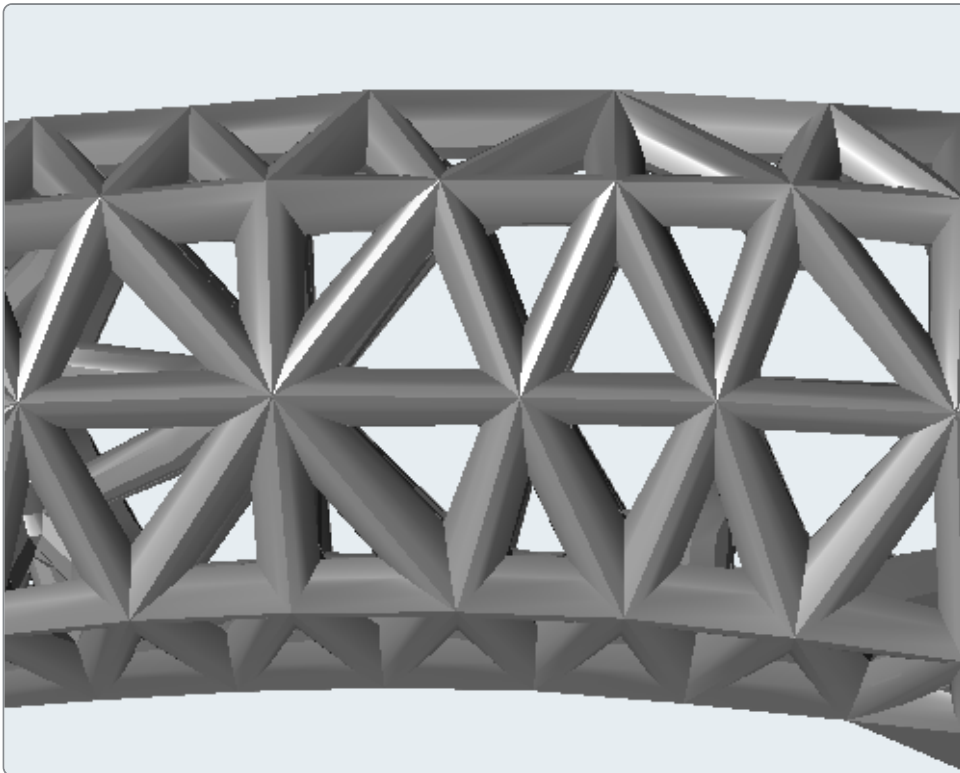


Figure 3. Smooth Lattice Off

6. Experiment with different lattice parameters, then click the **Compare Results** button at the bottom of the Analysis Explorer to compare the different results and total mass of the optimized lattice structures.