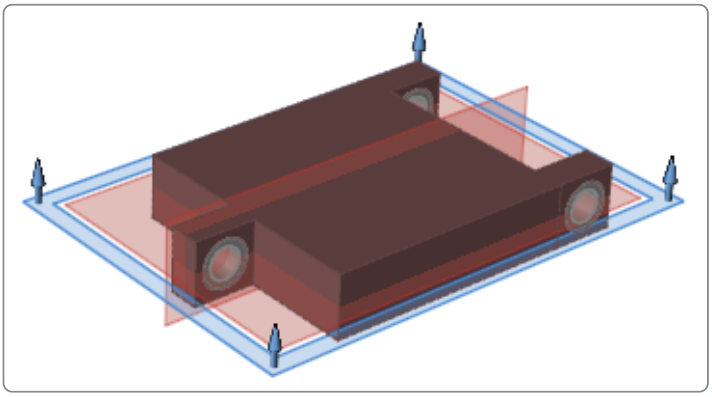
Altair Inspire™

Tutorial: Optimizing Topology - Maximize Stiffness

Run a topology optimization to maximize stiffness and explore the generated shape.

In this lesson you will learn how to:

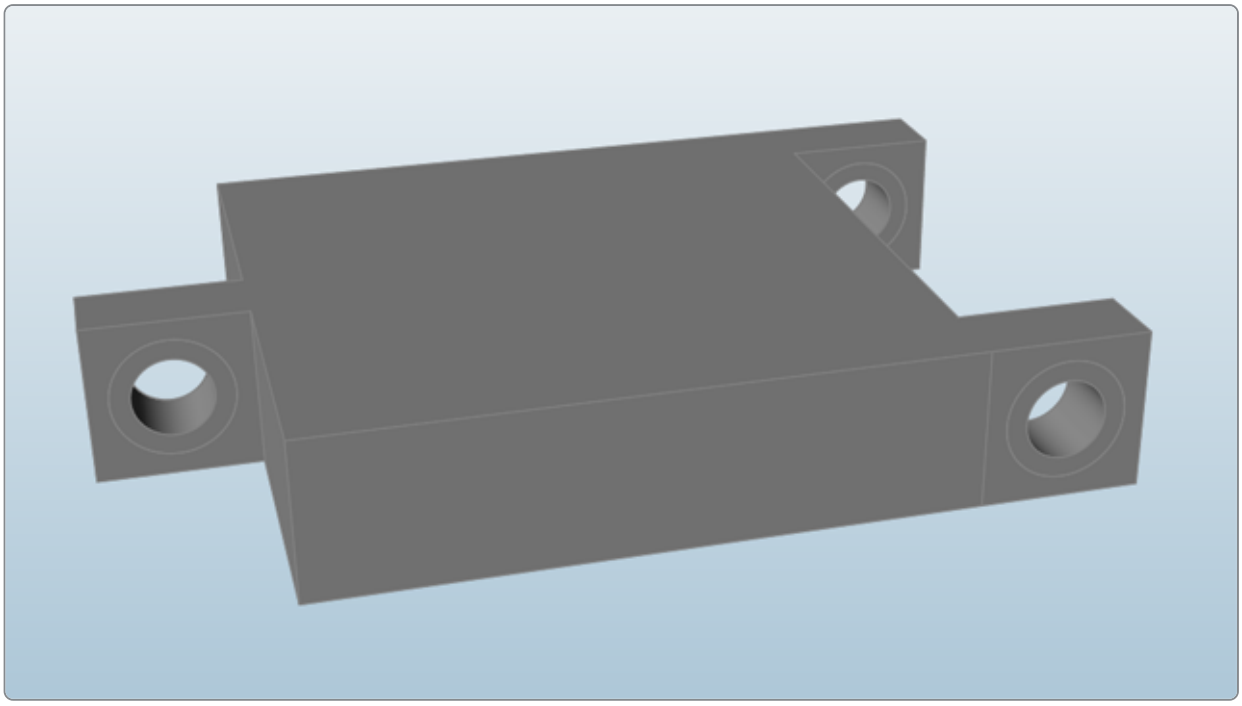
- Create multiple load cases with unit loads in x, y, and z directions
- Create forces and supports
- Apply symmetry planes
- Apply a single draw direction
- Run a topology optimization to maximize stiffness
- Explore generated shapes



Step 1. Open the Y-bracket Model

1. Press F7 to open the Demo Browser.
2. Double-click the `y-bracket.stmod` file to load it in the modeling window.
3. Make sure the display units in the Unit System Selector are set to **MKS (m kg N s)**.

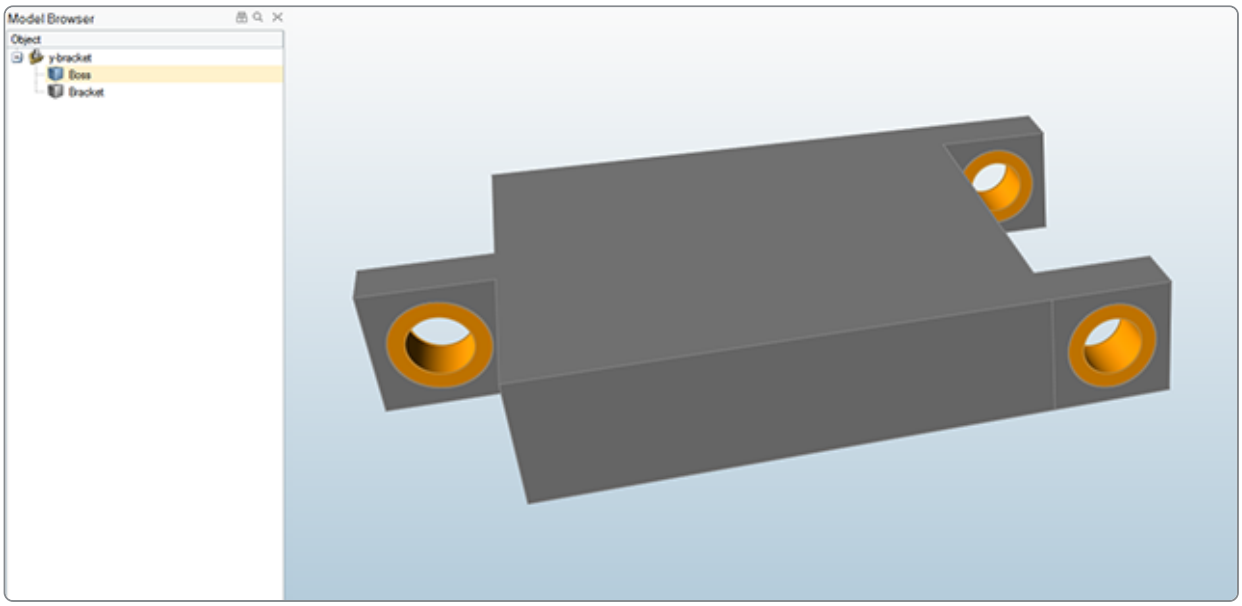
4. Use the **right mouse button** and the **middle mouse button** to pan and rotate the view so the y-bracket is positioned as shown below:



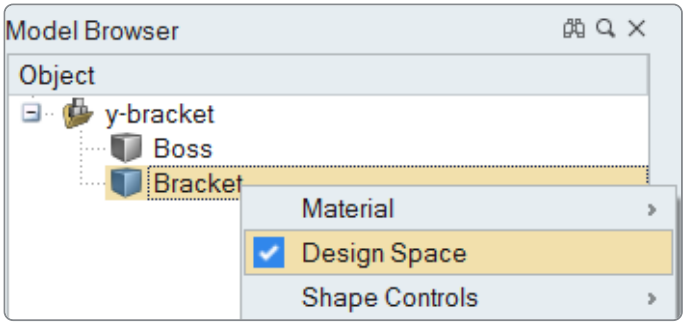
Step 2. Define the Design Space

1. If not already visible, press **F2** to open the Model Browser.
2. In the Model Browser, two parts are listed: Boss and Bracket. Click **Boss** to select it.

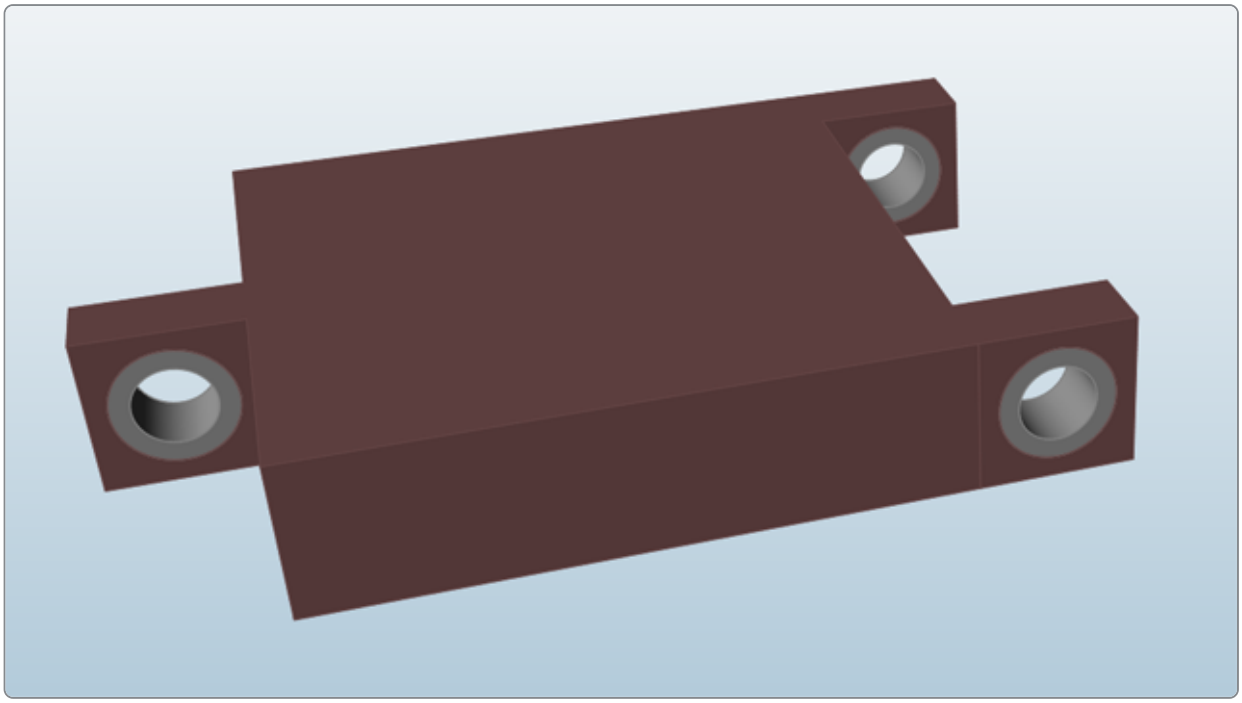
   The three cylindrical holes in the bracket turn yellow. While we will be placing loads and supports on the boss materials, we do not want to subtract any material from this part during optimization, so we do not want to include it in the design space.



3. Right-click **Bracket** in the Model Browser and select **Design Space**.



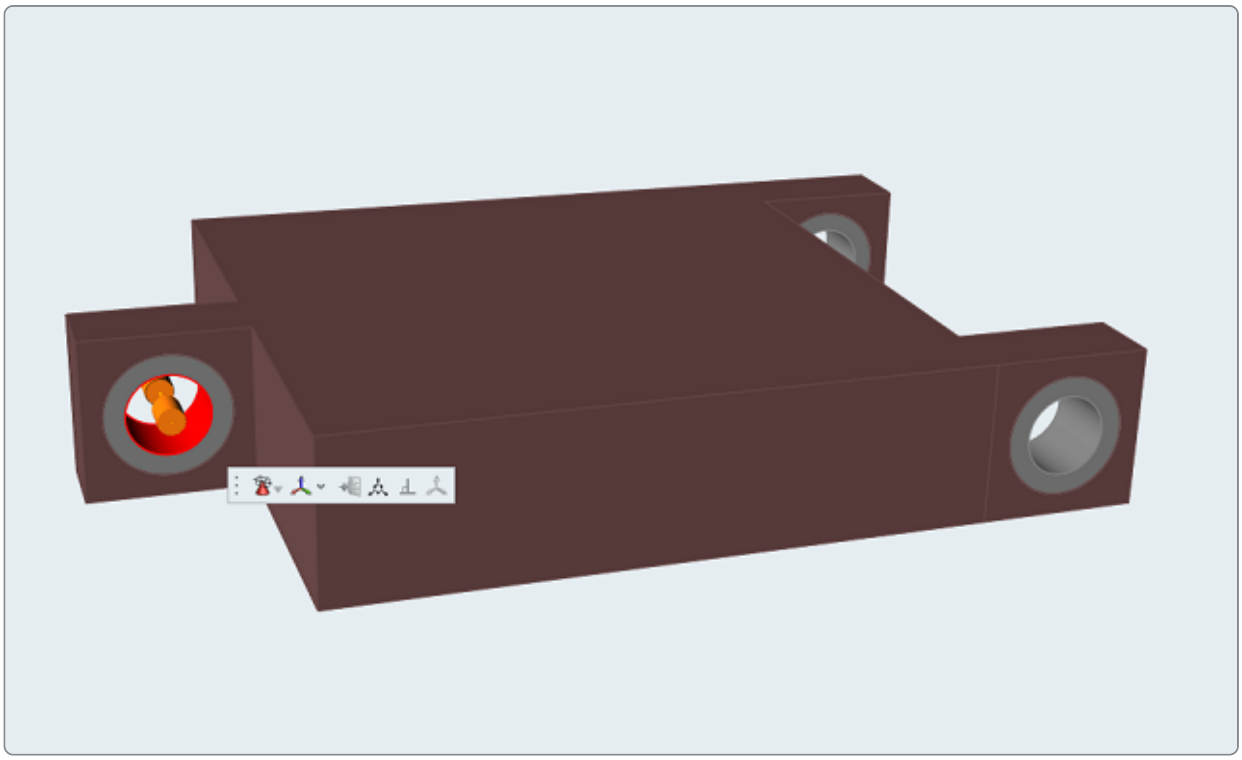
4. Click on an empty space in the modeling window. The red-brown color indicates the area that material will be carved from during optimization.



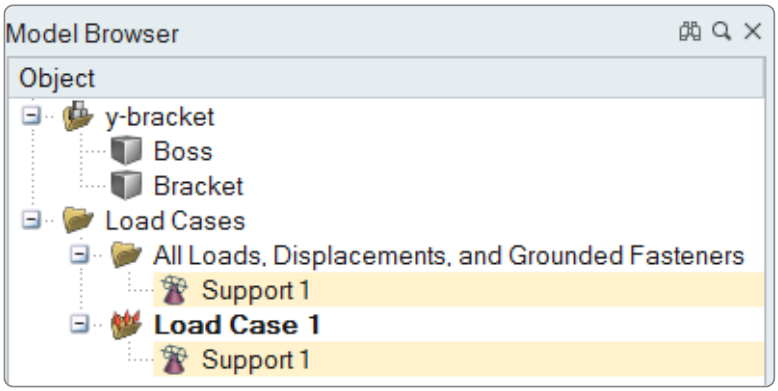
Step 3. Create a Center Hole Support and the First Load Case

1. Click the **Structure** tab on the ribbon.
2. Select the **Apply Supports** tool on the Loads icon.

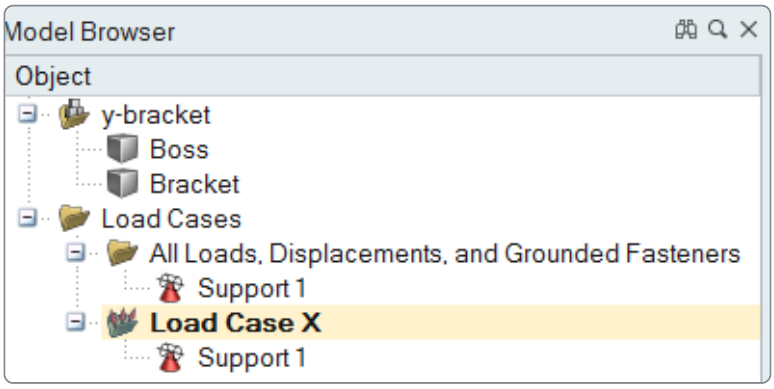
3. Click on the front boss material to apply the support.



In the Model Browser, two new folders are created, one called Load Case 1 and the other called All Loads, Displacements, and Grounded Fasteners. Support 1 is added to both.



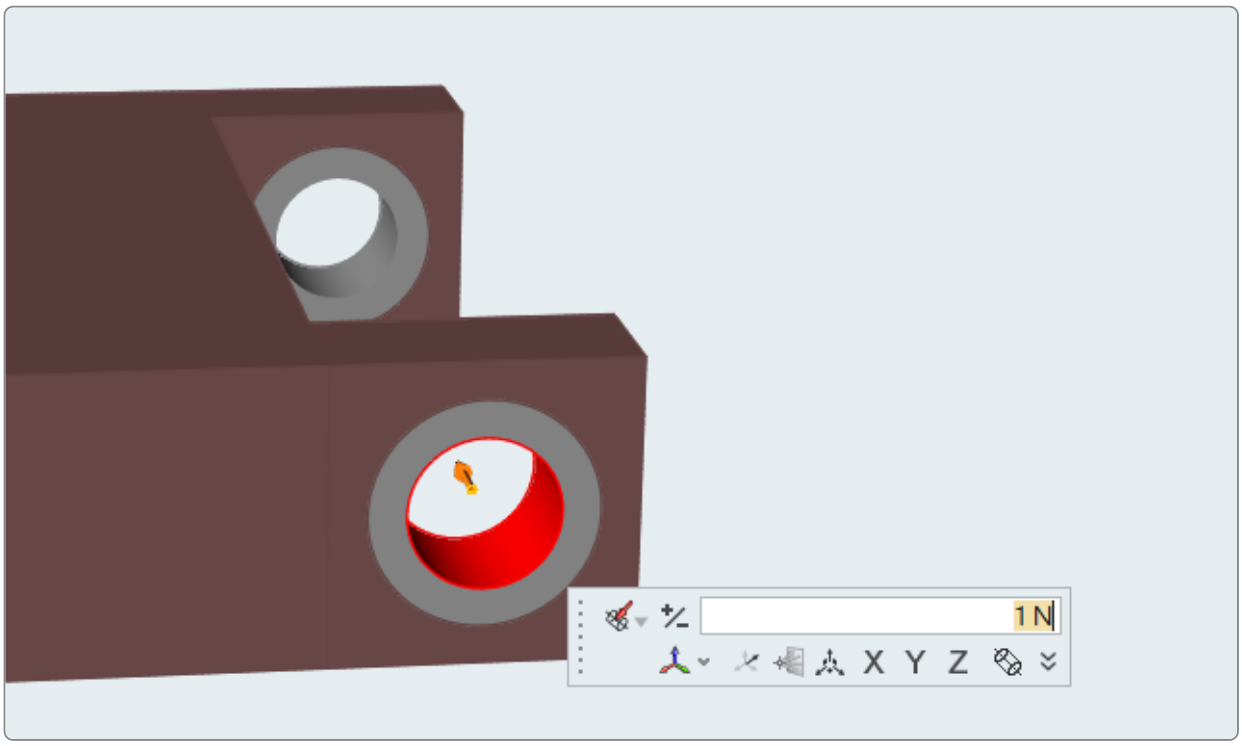
4. Click **Load Case 1** twice to make it editable, rename it `Load Case X`, and press **Enter**.
   The name of the load case is shown in bold, indicating that it is the current load case. Any new loads or supports that you create will be added to it automatically.



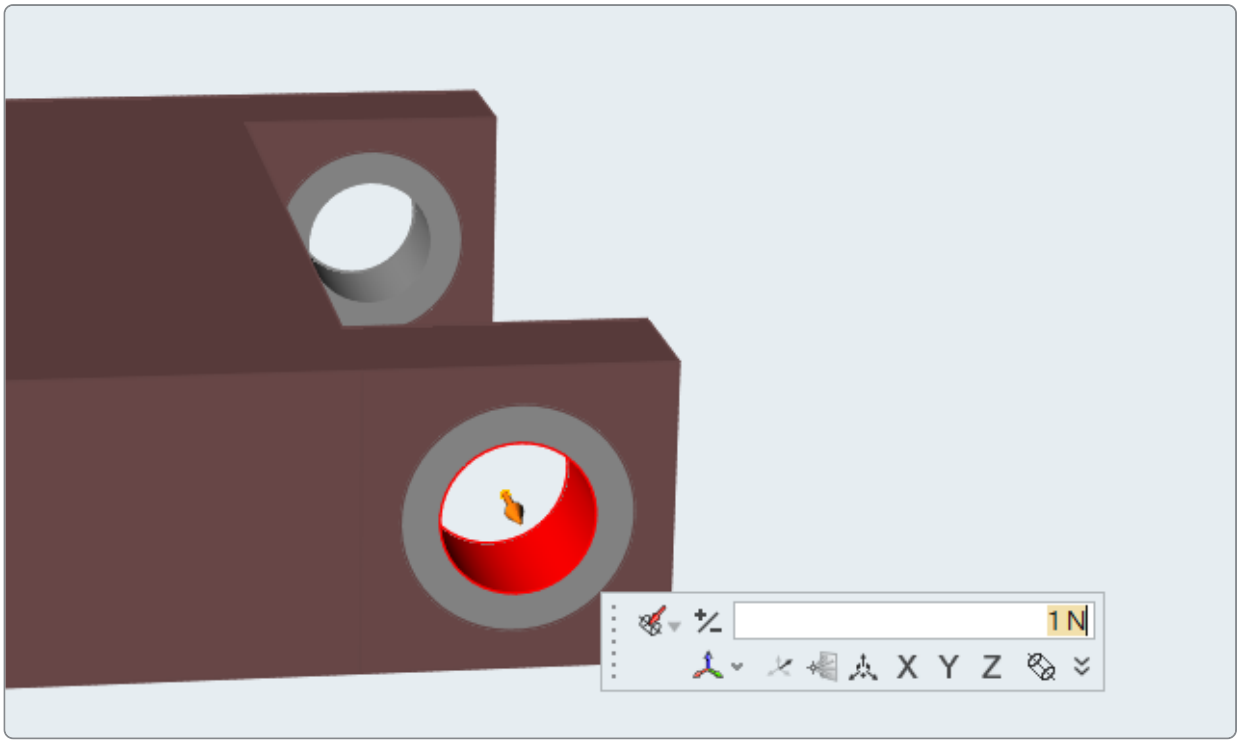
Step 4. Apply Forces to Boss Materials

1. Select the **Apply Force** tool on the Loads icon.

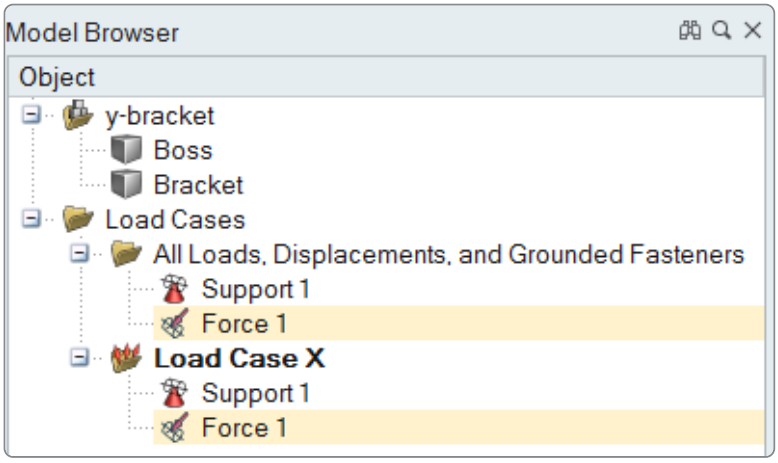
2. Click on one of the rear boss materials to apply the force.



3. The force is initially applied in the negative x direction. Click the **+/-** icon in the microdialog to reverse it to the positive x direction.

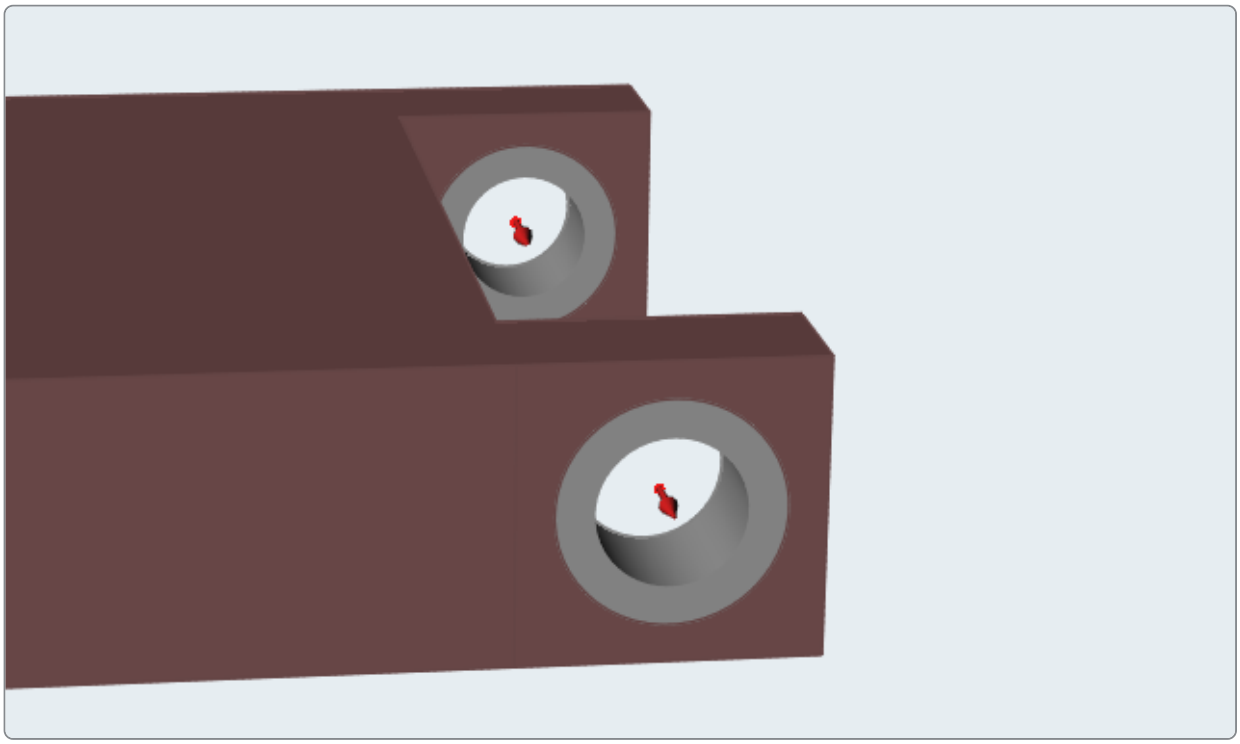


4. Force 1 appears in the Model Browser in both the All Loads and Displacements folder and Load Case X.

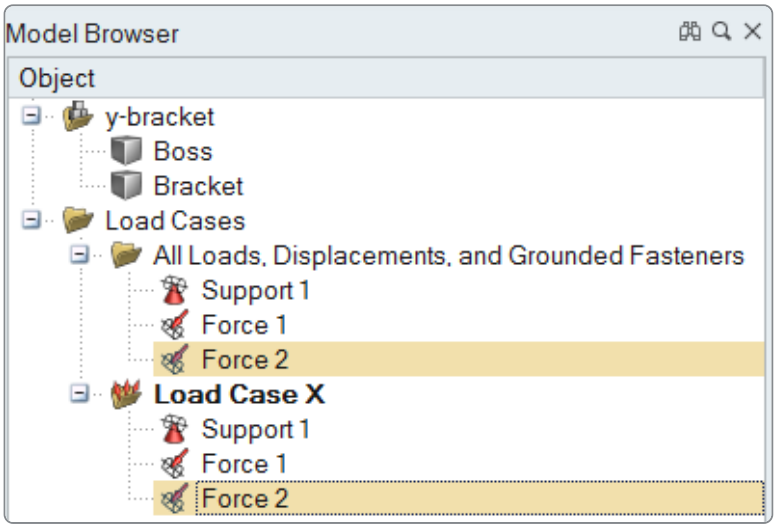


5. While the Apply Force tool is still active, click on the other rear boss material and use the **+/-** icon to reverse the direction.

   Both Force 1 and Force 2 should now be applied in the positive x direction, as shown in the image below:



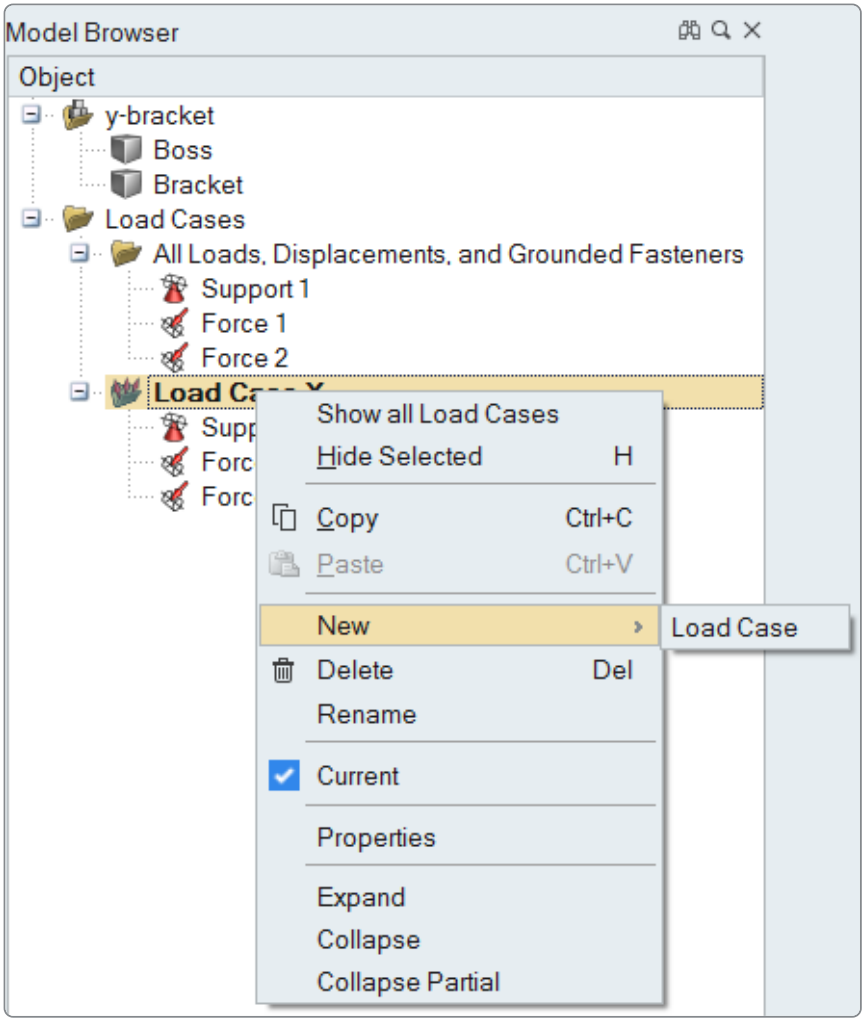
6. Force 2 appears in the Model Browser in both the All Loads and Displacements folder and Load Case X.



7. Double right-click to exit the tool.

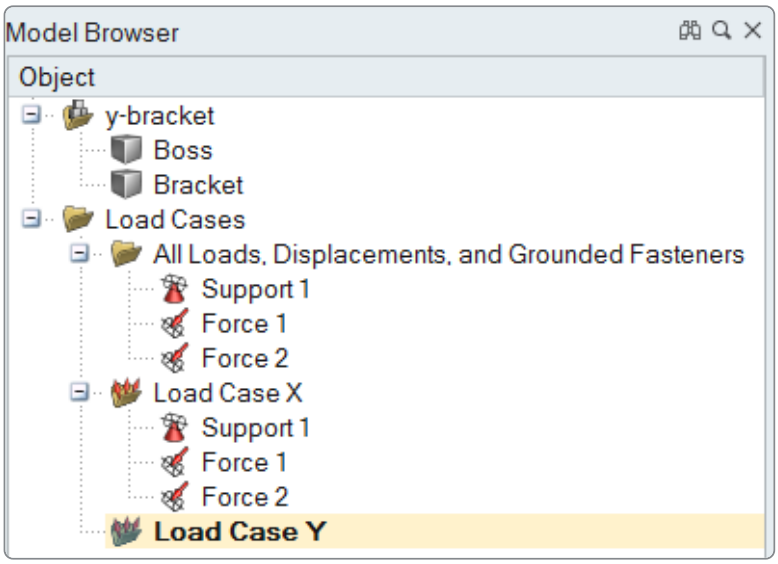
Step 5. Create the Second Load Case

1. In the Model Browser, right-click on **Load Case X** and select **New** > (and then) **Load** Case.

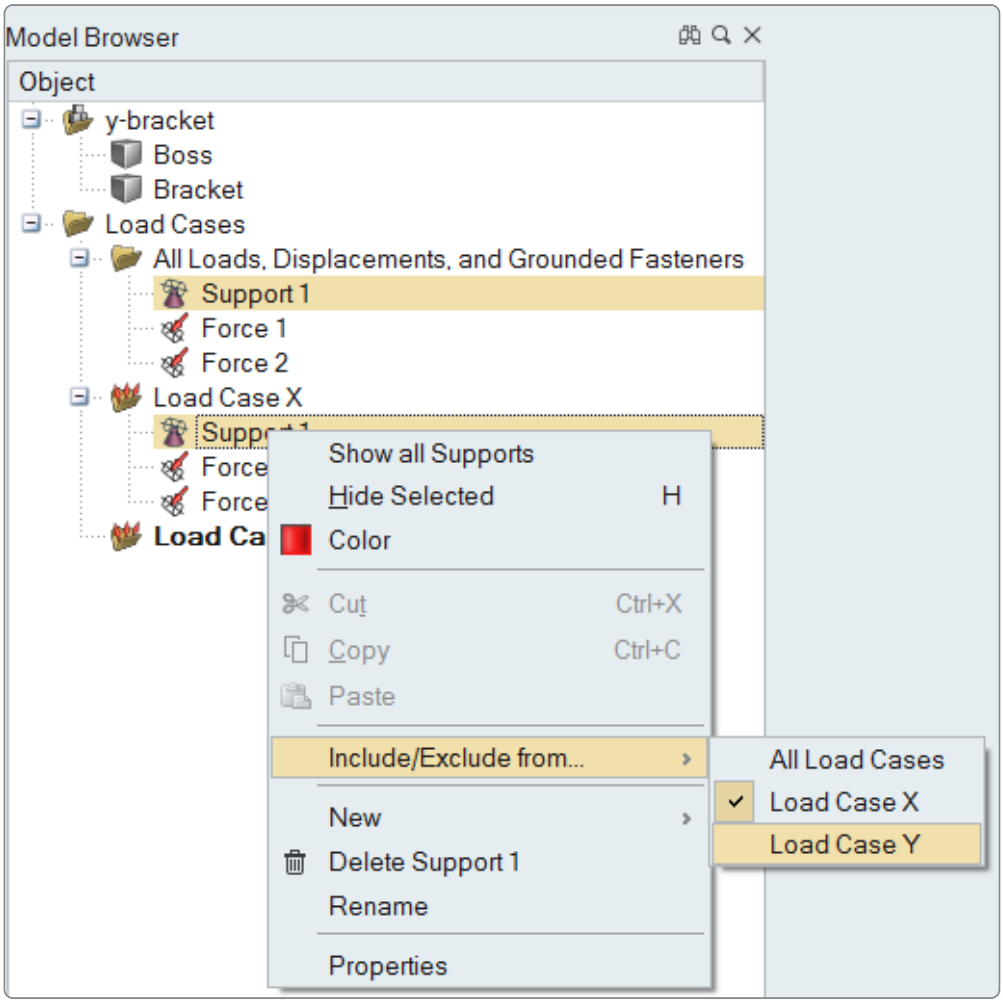


A new load case is added in the Model Browser.

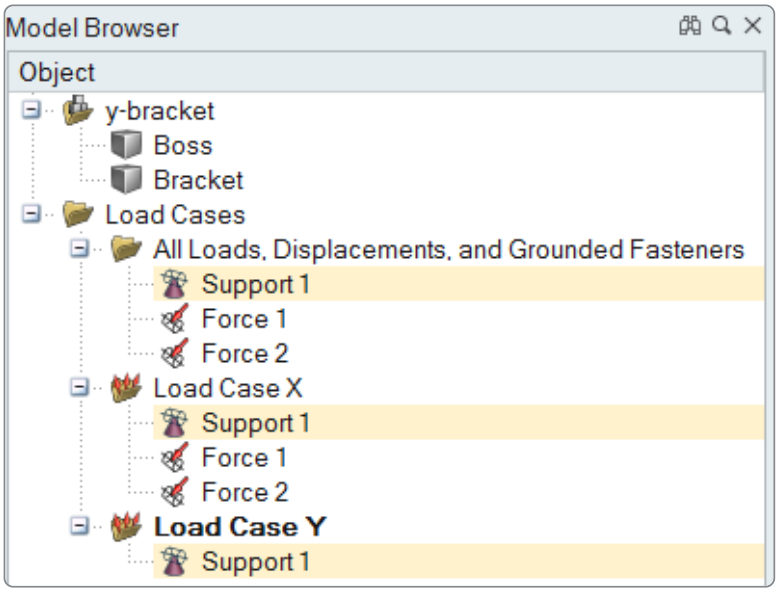
2. Rename the load case `Load Case Y` and press **Enter**.
This is now the current load case.



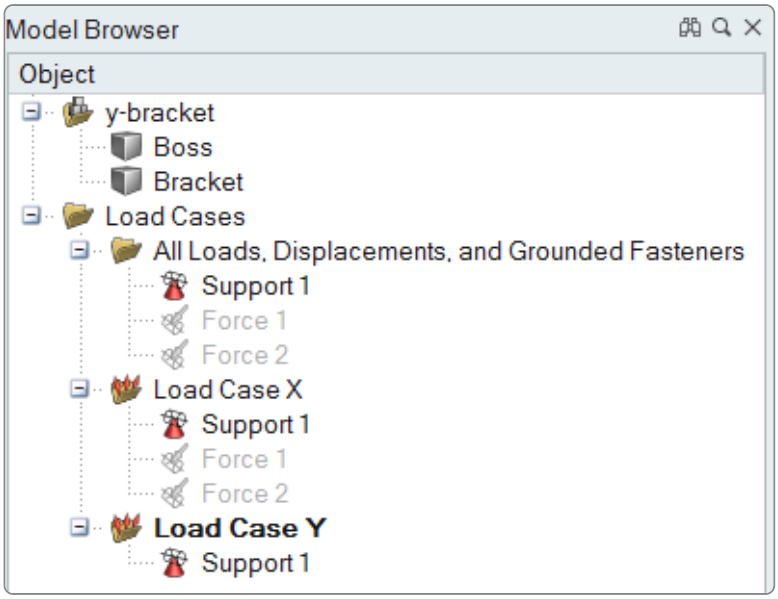
3. We want to use the same support from Load Case X in Load Case Y, so right-click on **Support 1** in the Model Browser and select **Include/Exclude from... > Load Case Y**.



4. Support 1 is added to Load Case Y in the Model Browser.

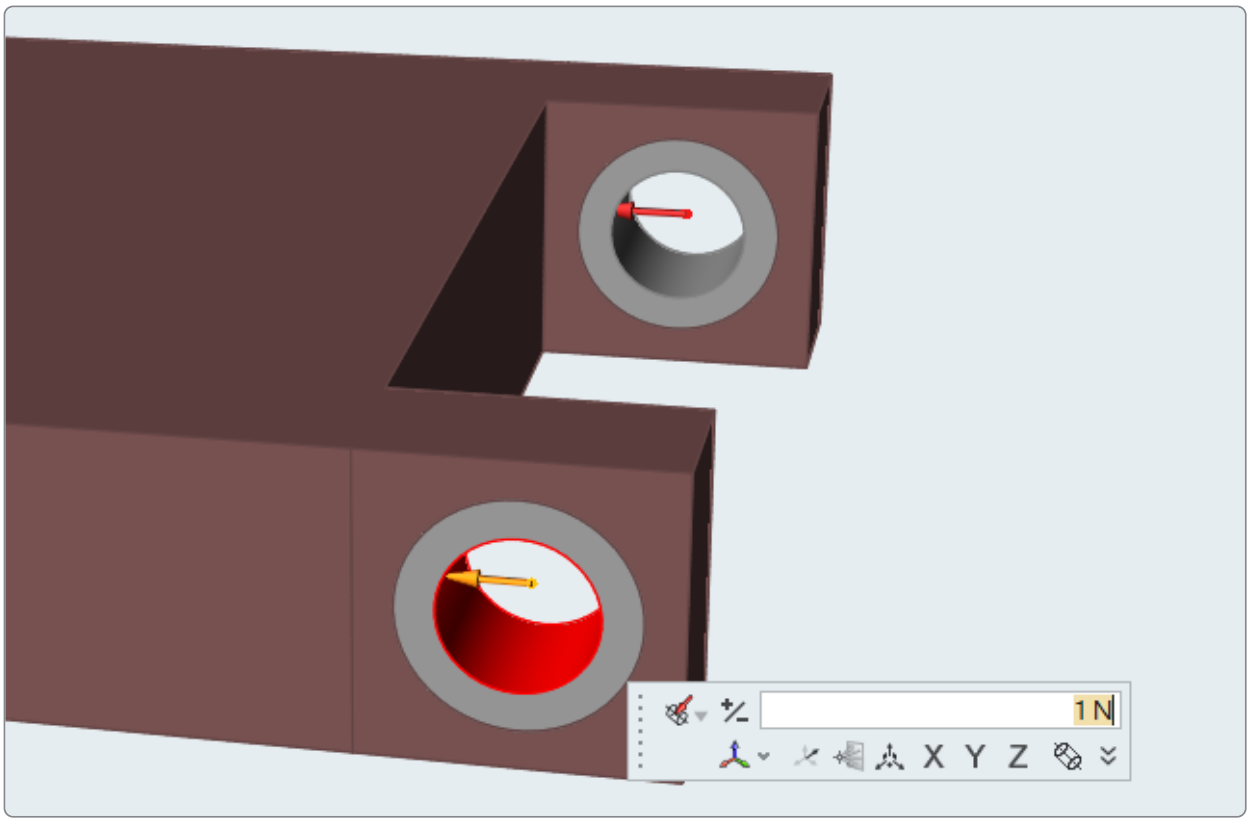


5. Click the icons next to Force 1 and Force 2 in the Model Browser to temporarily hide these forces in the modeling window.

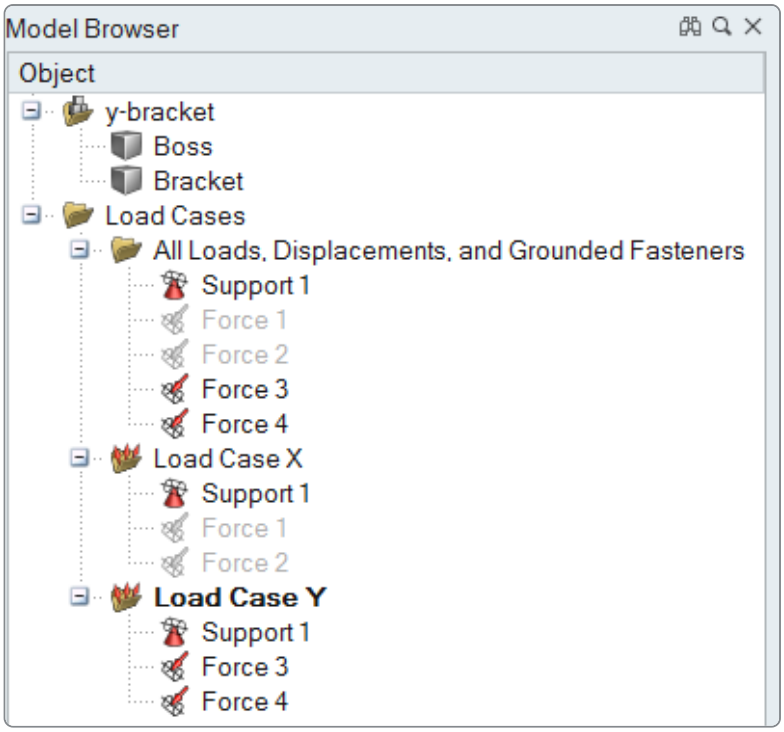


6. Select the **Apply Force** tool on the Loads icon.

7. Add two more forces, one to each of the rear boss materials in the negative y direction.



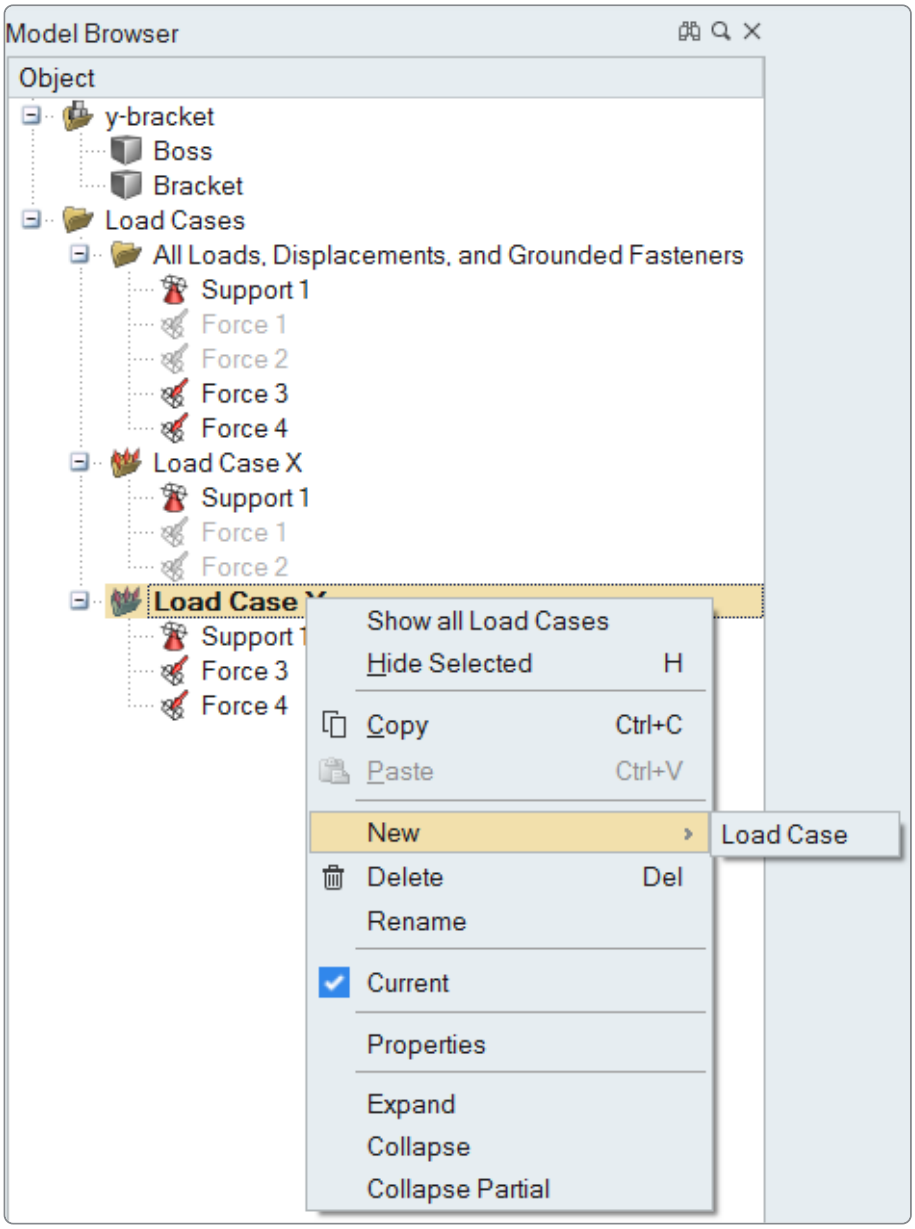
8. Force 3 and Force 4 have been added to Load Case Y in the Model Browser.



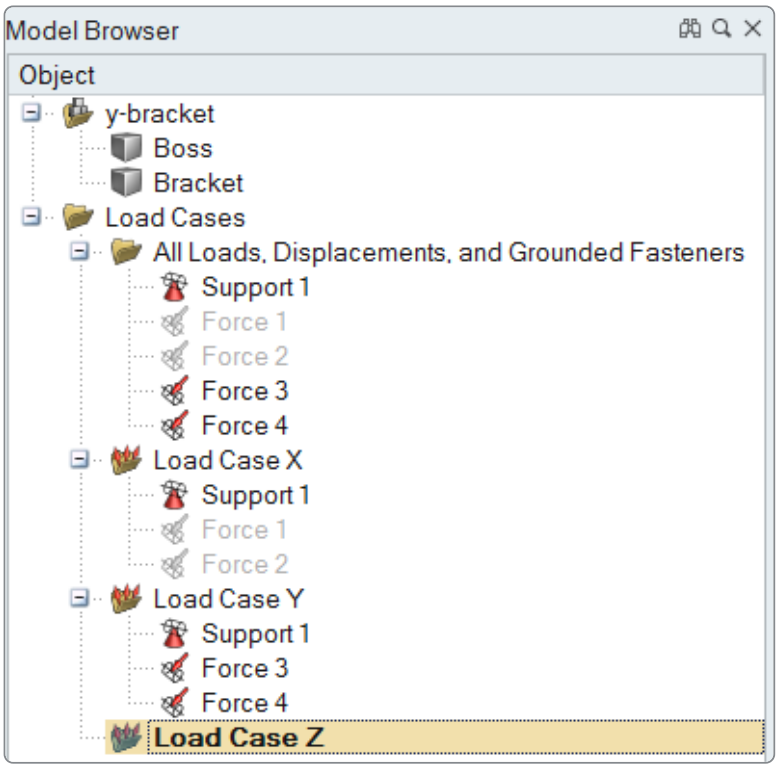
9. Double right-click to exit the tool.

Step 6. Create the Third Load Case

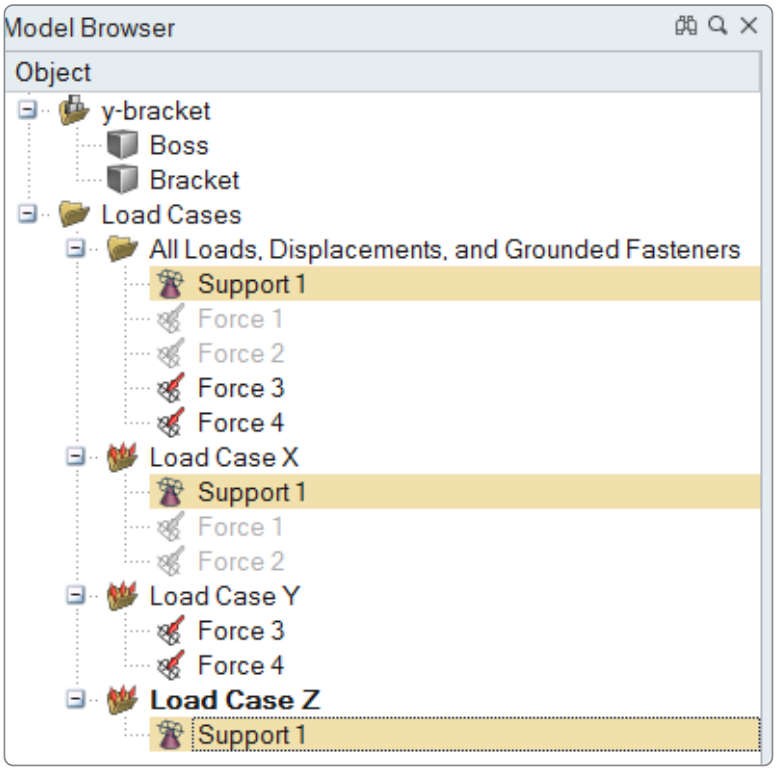
1. In the Model Browser, right-click on **Load Case Y** and select **New** > (and then) **Load Case**.



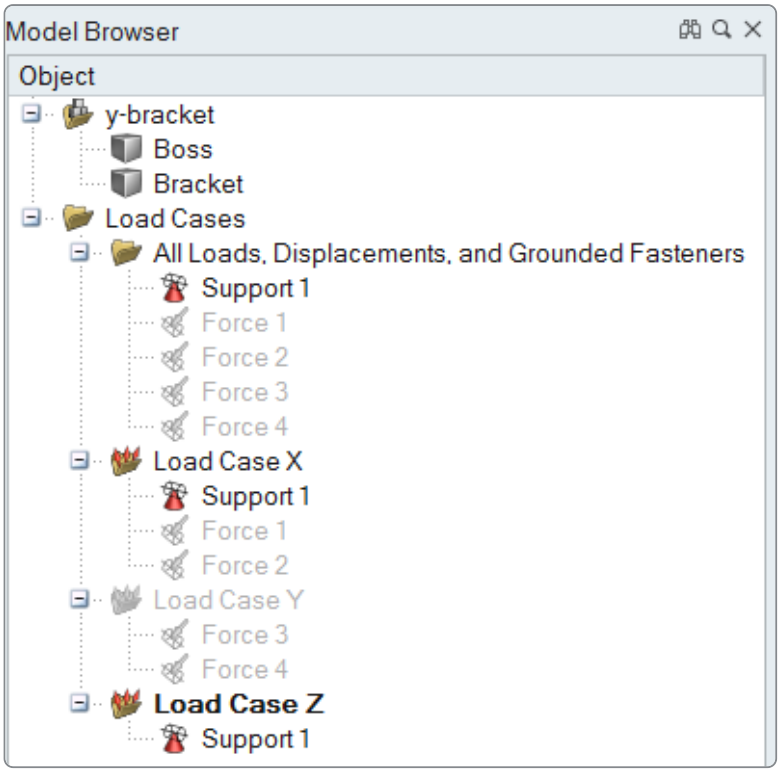
2. Rename the load case `Load Case Z` and press **Enter**. This is now the current load case.



3. Click on **Support 1** in the All Loads, Displacements, and Grounded Fasteners folder in the Model Browser and drag it to Load Case Z.

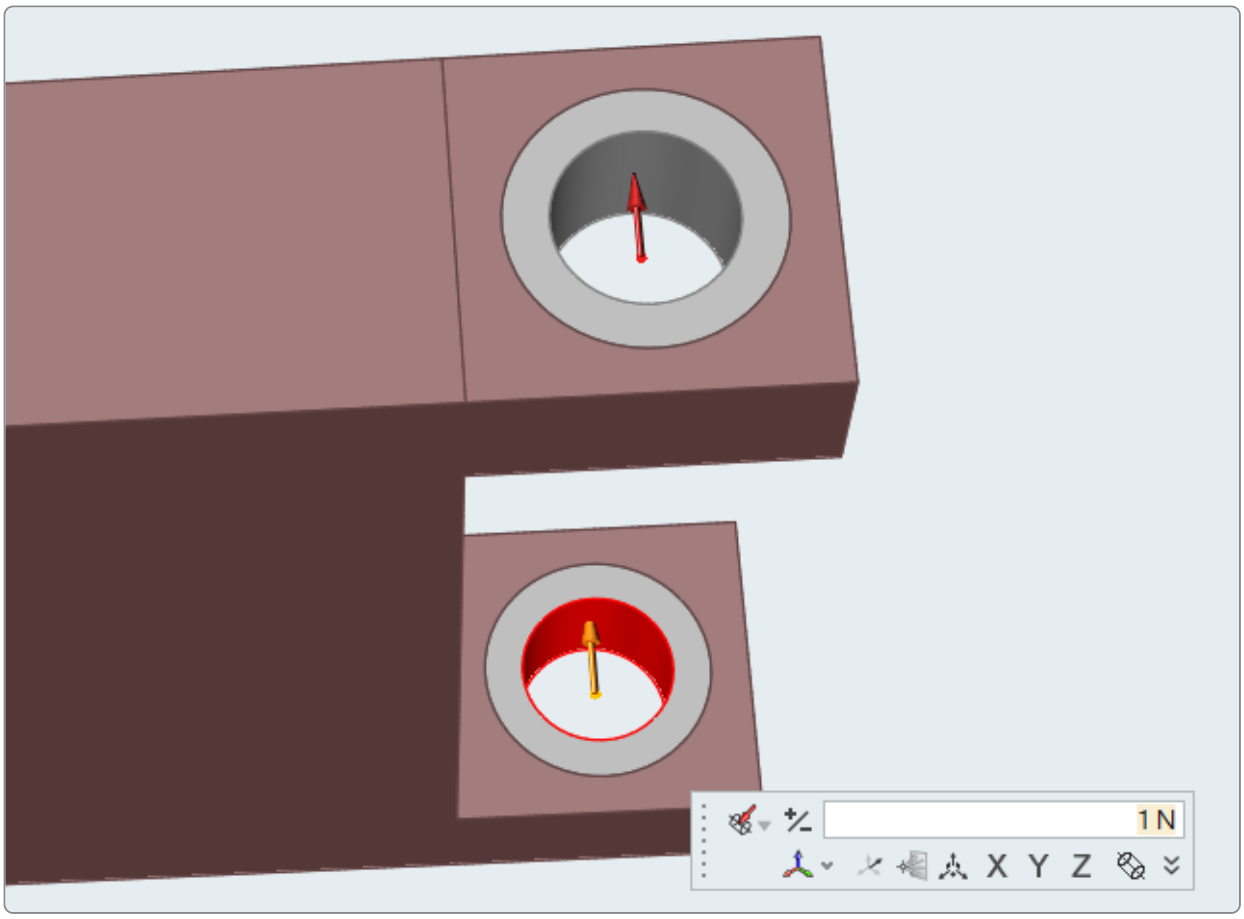


4. Click the icons next to Force 3 and Force 4 in the Model Browser to temporarily hide these forces in the modeling window.

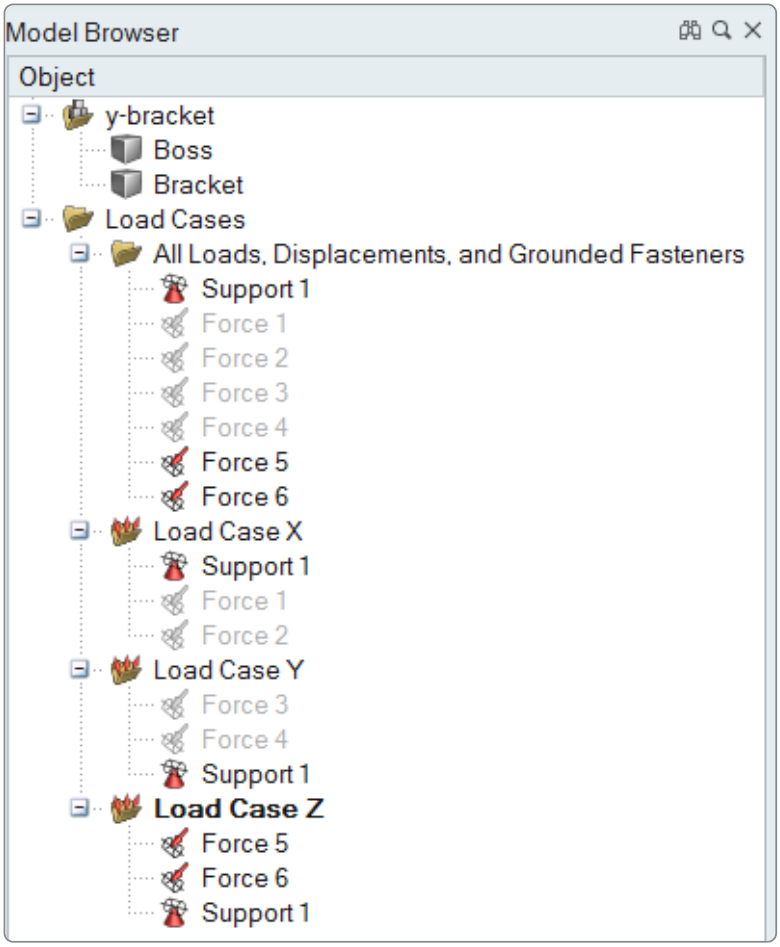


5. Select the **Apply Force** tool on the Loads icon.

6. Add two more forces, one to each of the rear boss materials in the positive z direction.

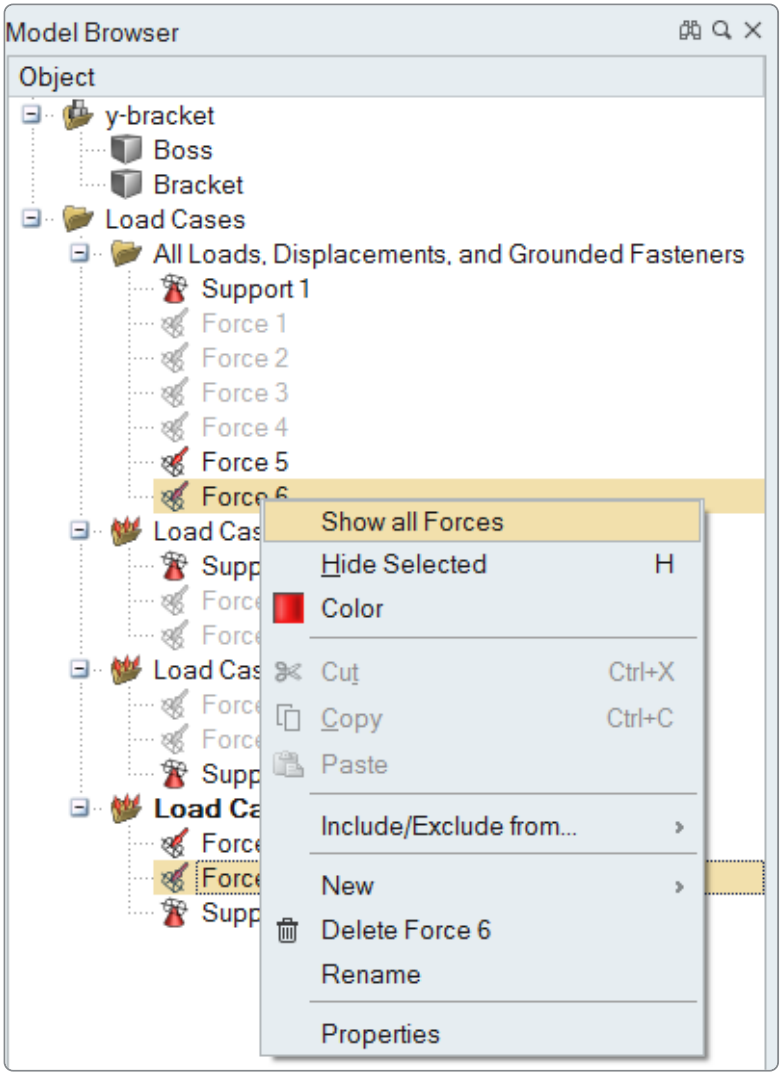


7. Check that Force 5 and Force 6 have been added to Load Case Z in the Model Browser.



8. Double right-click to exit the tool.

9. Right-click one of the forces in the Model Browser and select **Show all Forces** in the context menu.

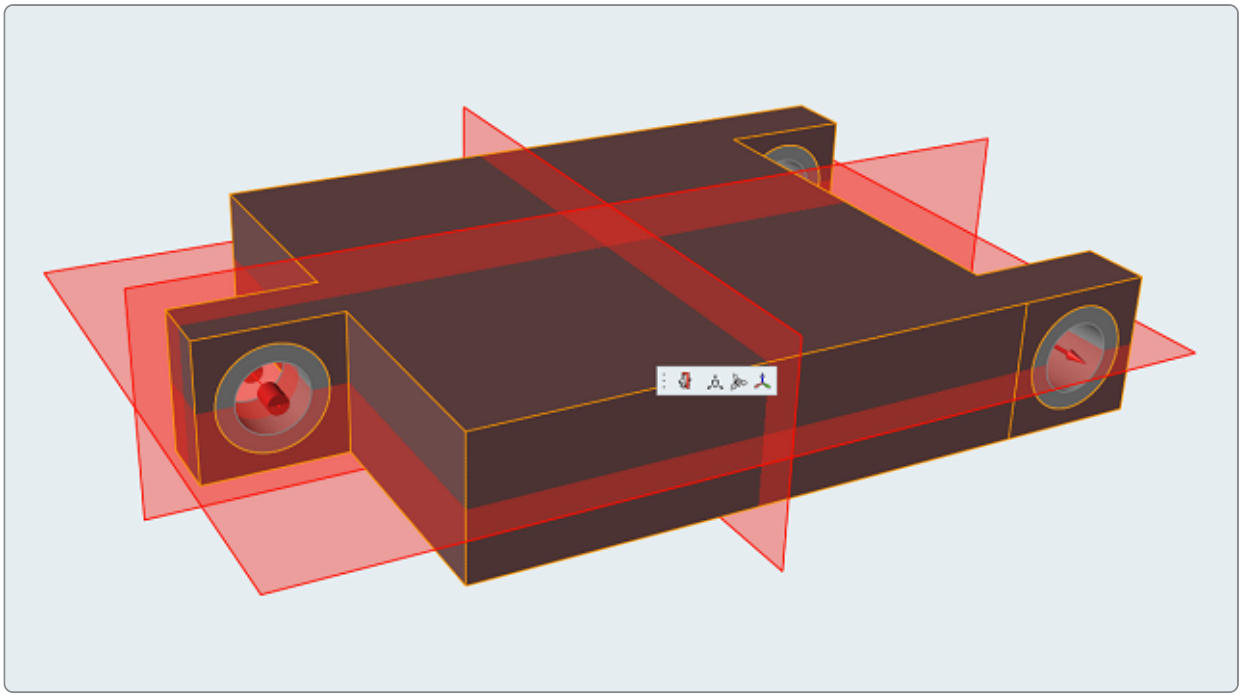


Step 7. Add Symmetry Planes

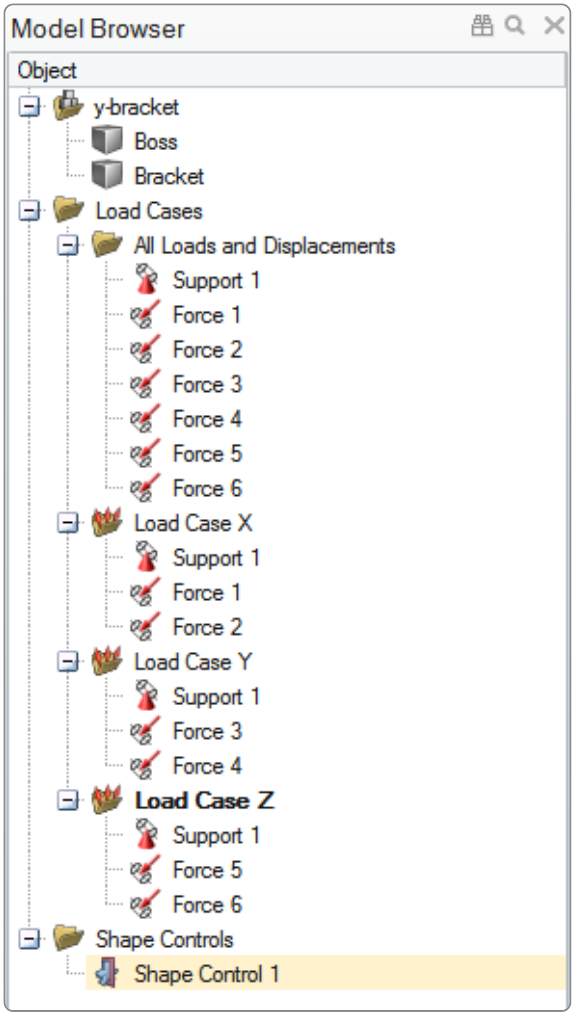
1. Select the **Symmetry** tool on the Shape Controls icon.

2. Select the **Symmetric** tool from the secondary ribbon.

3. Click on the bracket in the modeling window to select it.
   Three red symmetry planes appear.

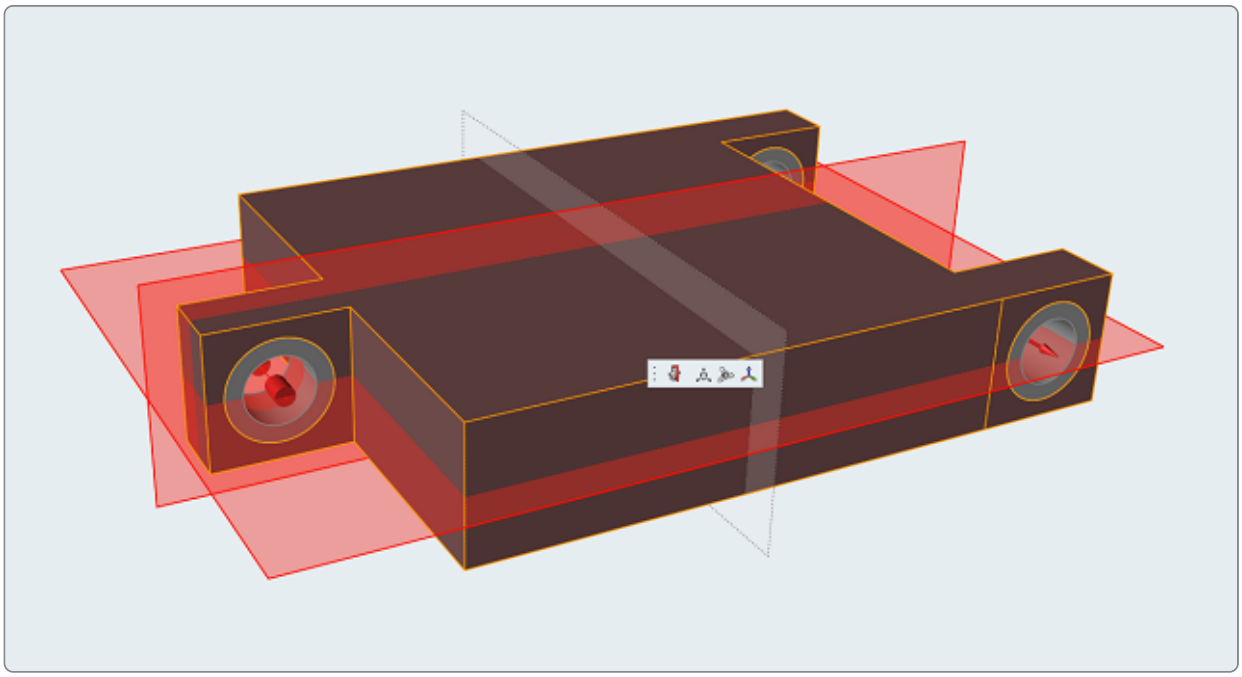


4. Check that Shape Control 1 has been added to the Shape Controls folder in the Model Browser.

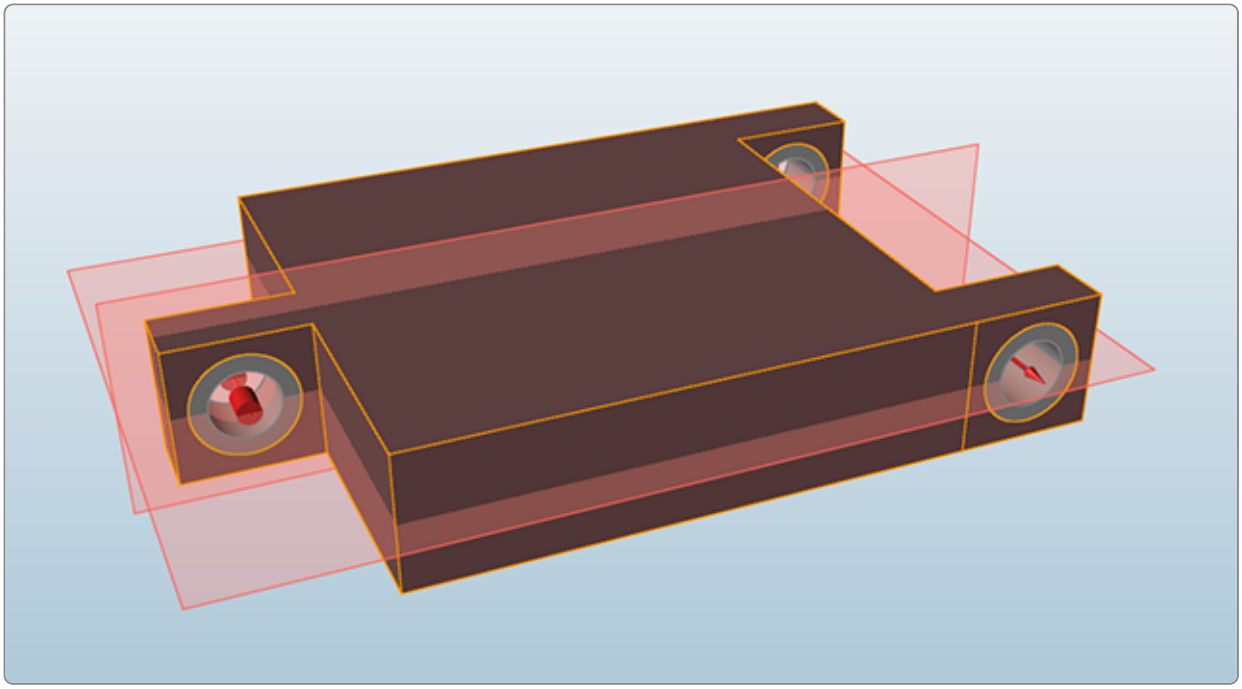


5. Click the plane shown below to deselect it.

   The plane turns gray.



6. Double right-click to exit the tool.

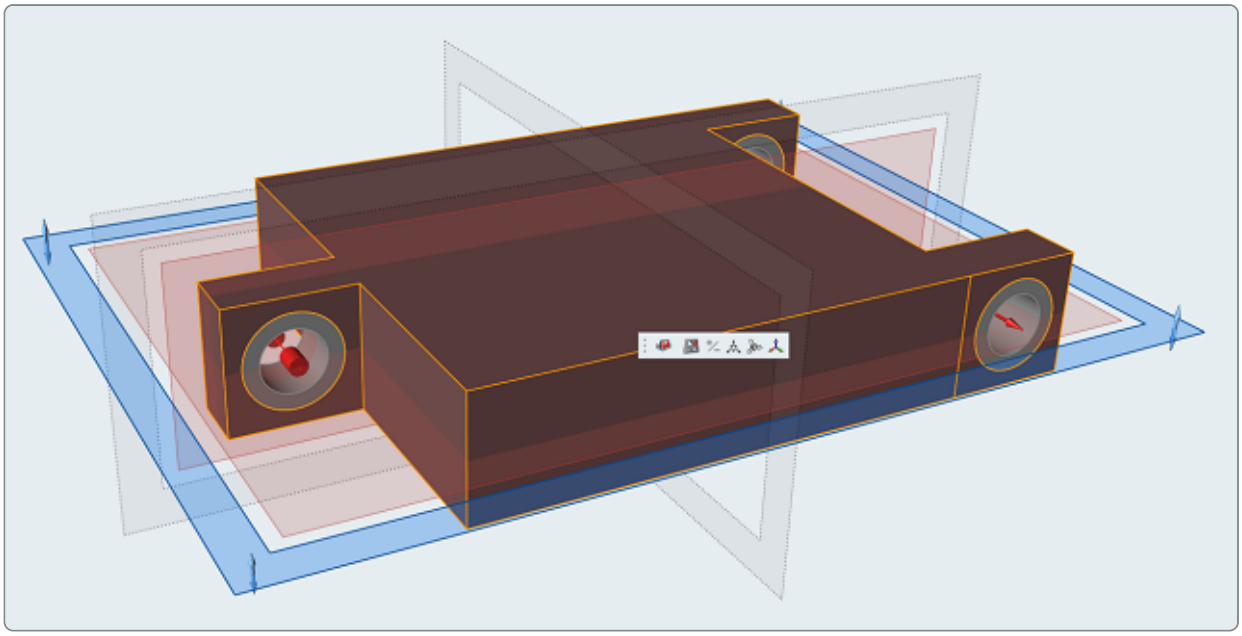


Step 8. Add a Draw Direction

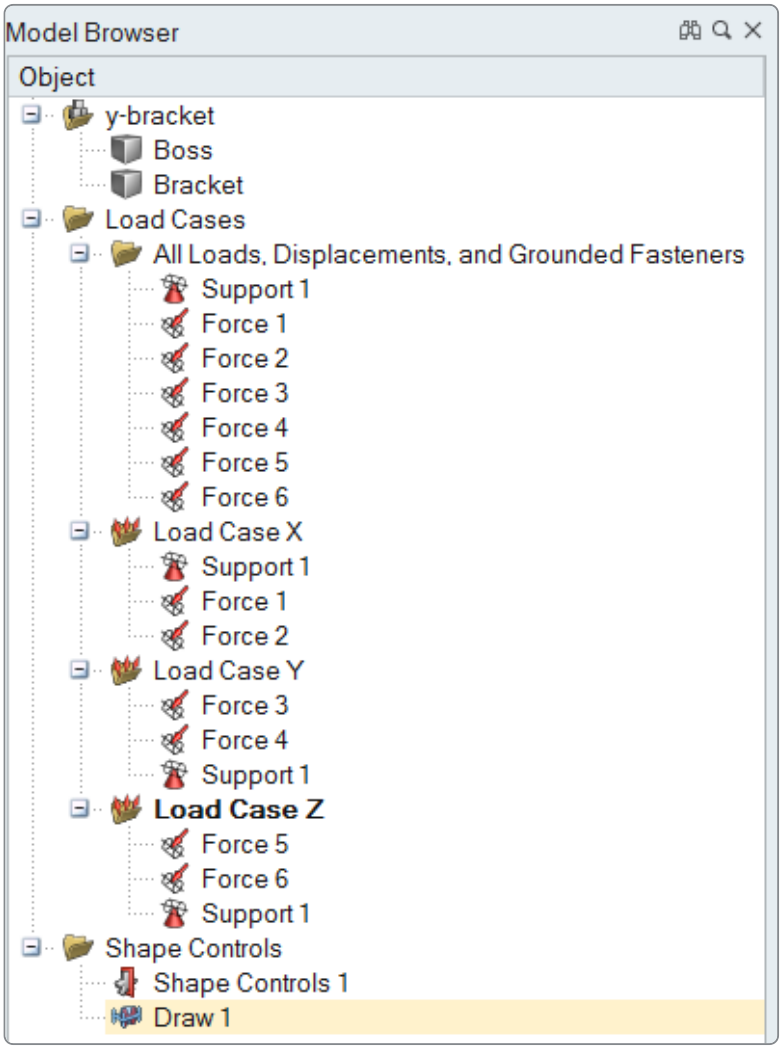
1. Select the **Draw Direction** tool on the Shape Controls icon.

2. Select the **Split Draw** tool on the secondary ribbon.

3. Click on the bracket in the modeling window to select it.
   Three planes appear; the blue plane indicates the currently selected parting plane.



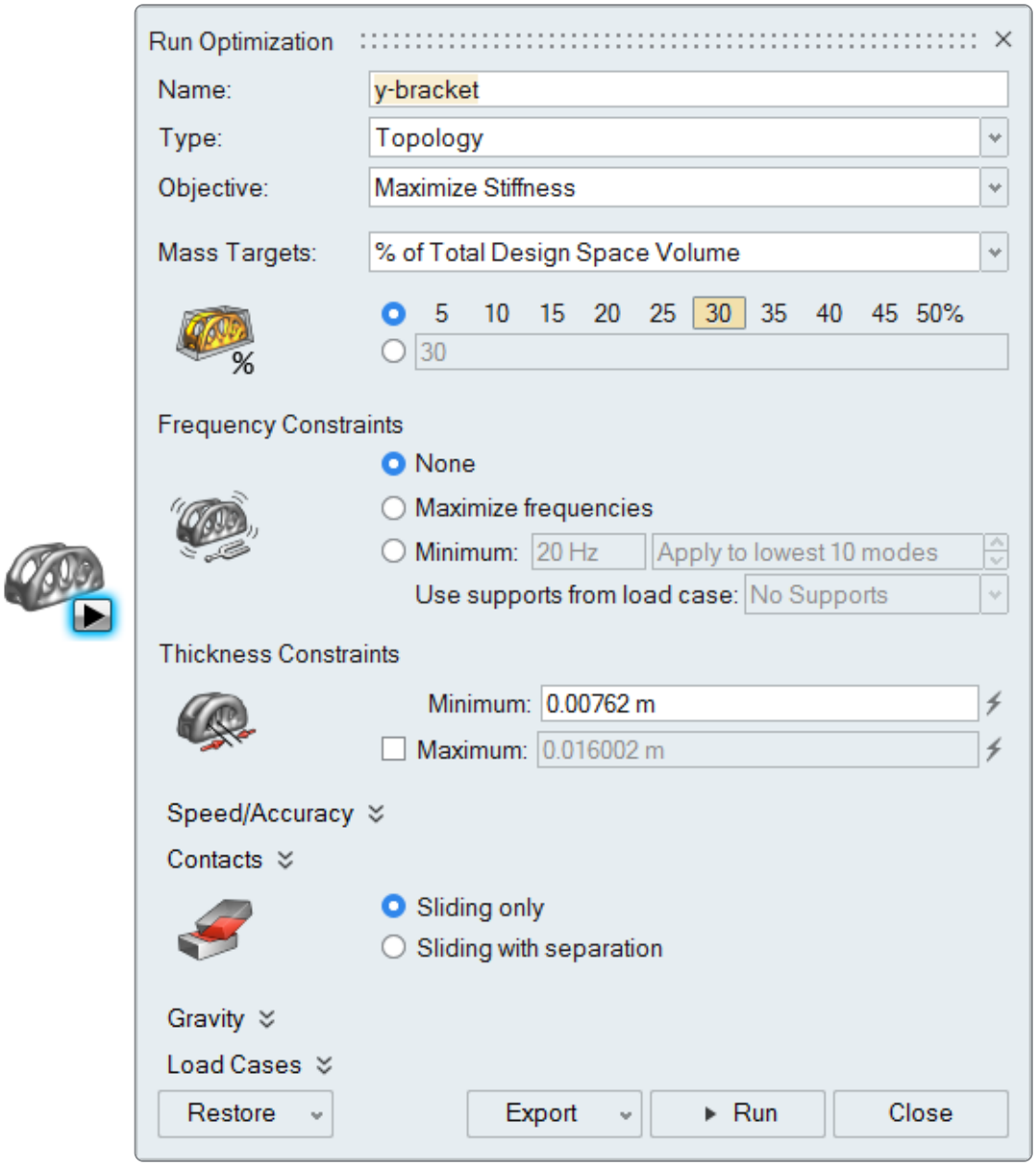
4. Check that Draw 1 has been added to the Shape Controls folder in the Model Browser.



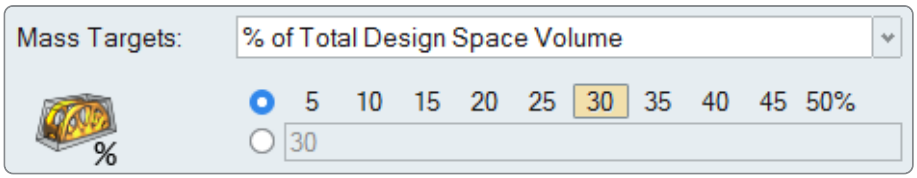
5. Double right-click to exit the tool.

Step 9. Run a Topology Optimization

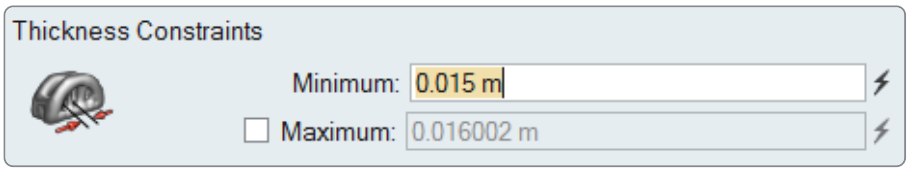
1. Click on the **Optimize** icon to open the Run Optimization window.



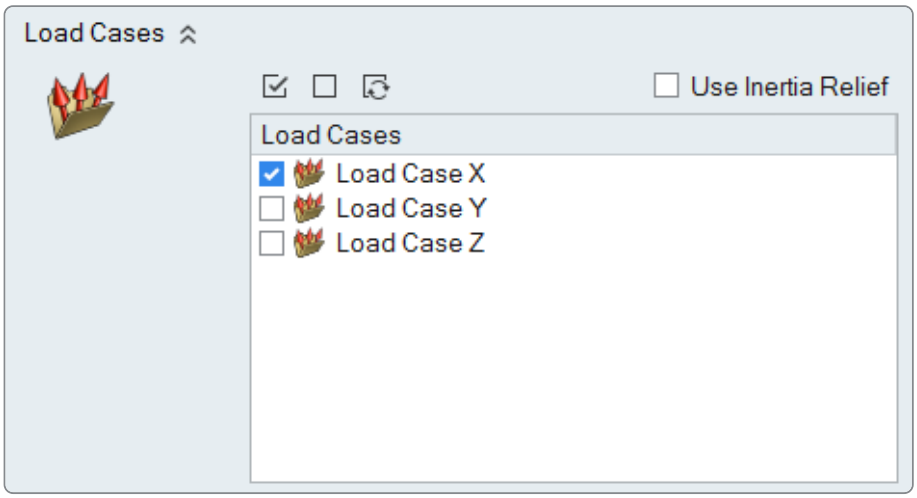
2. Select **Maximize Stiffness** for the optimization Objective.
3. Under Mass targets, select **% of Total Design Space Volume** from the drop-down menu and choose **30** percent.



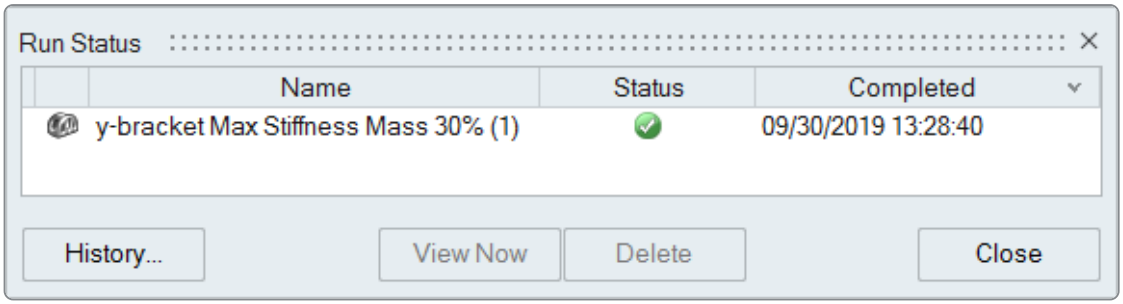
4. Under Thickness constraints, increase the **Minimum** to `0.015 m`. (This will speed up the optimization.)



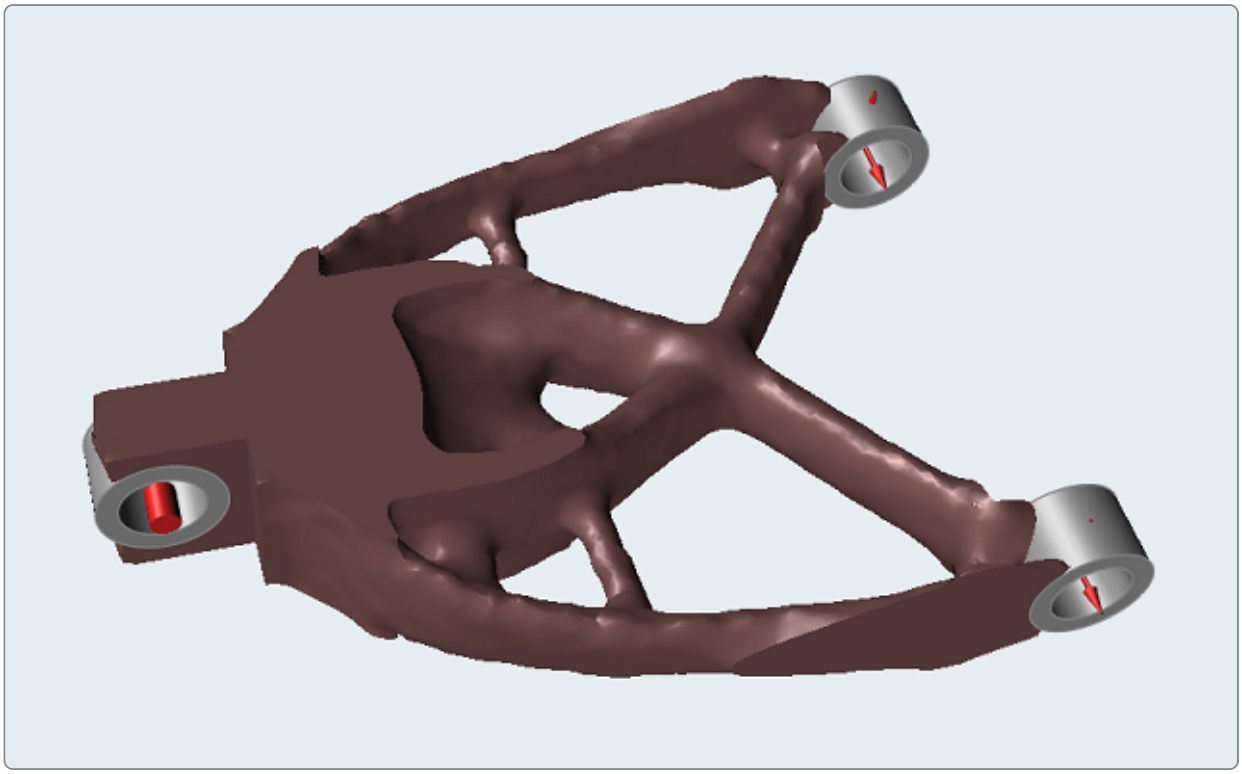
5. Under Load Cases, deselect **Load Case Y** and **Load Case Z**.
   This will run the optimization with only Load Case X applied.



6. Click **Run**.
   The Run Status window appears. A green check mark will appear when the optimization is complete.

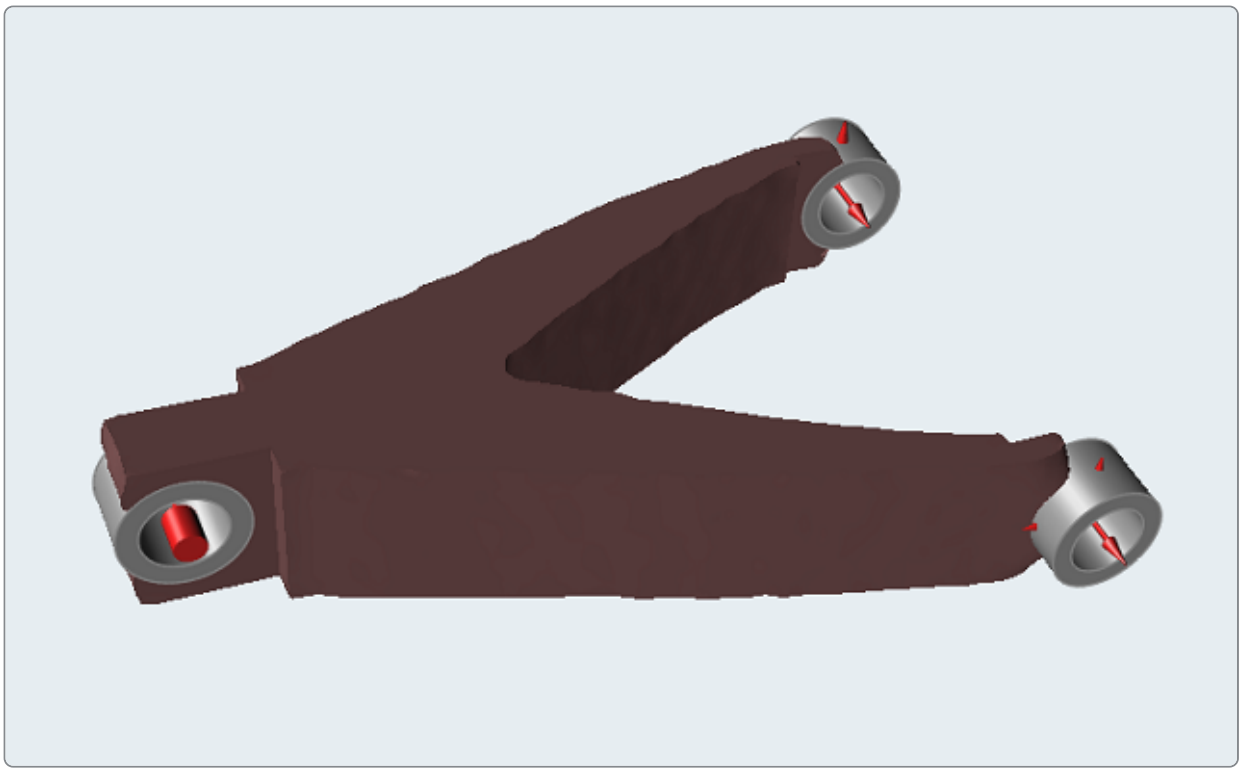


7. Double-click on the name of the run to view the results.
   The optimized shape is displayed in the modeling window and is listed as an alternative in the Shape Explorer.

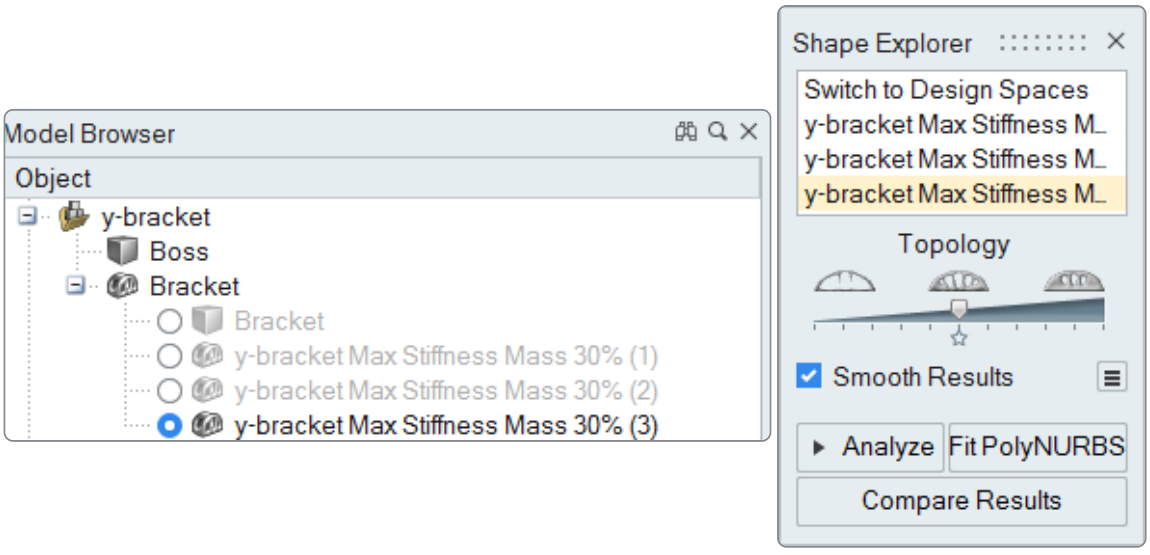


8. Repeat the above procedure to run an optimization for Load Case Y.

9. Repeat the above procedure to run an optimization for Load Case Z.

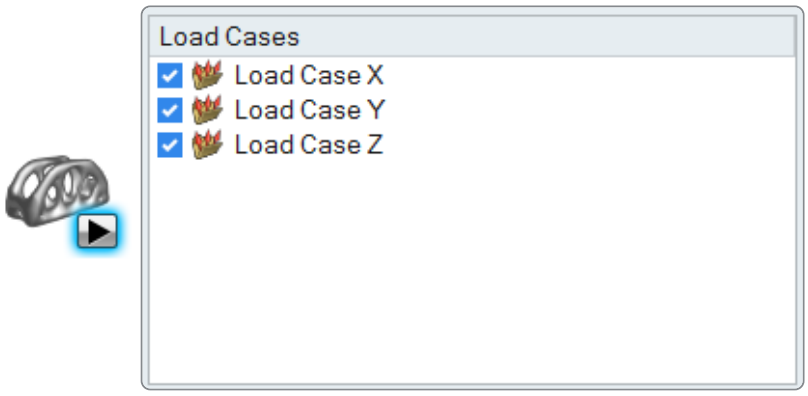


10. The optimization runs for Load Case Y and Load Case Z appear as additional alternatives in the Model Browser and the Shape Explorer.

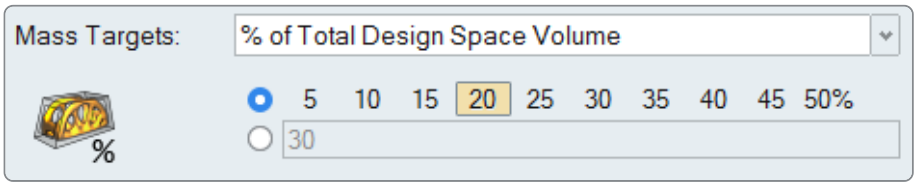


Step 10. Explore Optimized Shapes

1. Now run the optimization one more time using all three load cases simultaneously. Click the **Run Optimization** icon to open the Run Optimization window, and select all three load cases.



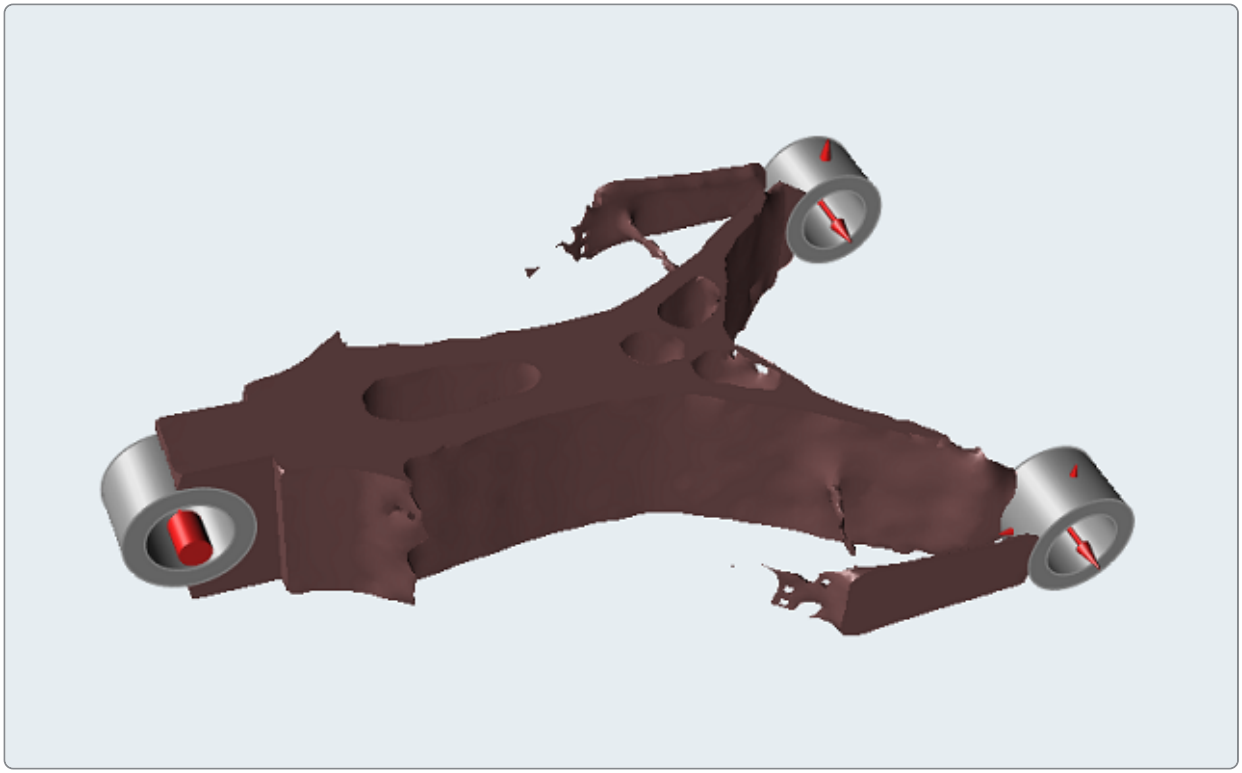
2. Change the **Mass target** to **20** percent of the total design space volume.



3. Click the **Run**.

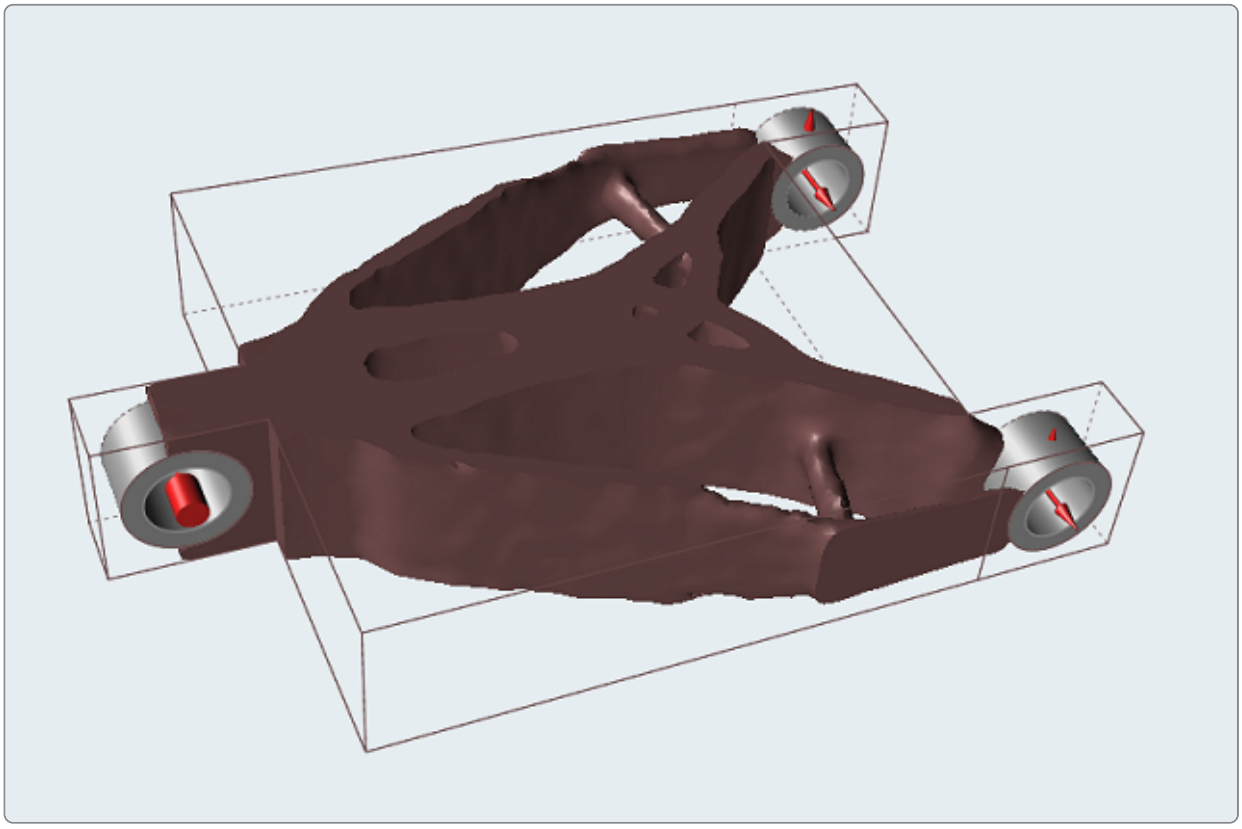
   When the optimization is complete, a green flag appears above the Optimize icon group, indicating that the run completed successfully.

4. Click the green flag to view the optimized shape.



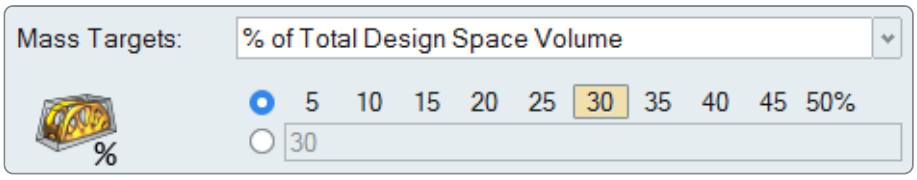
5. Drag the **Topology** slider in the Shape Explorer to explore the optimized shape. Changing the topology adds and subtracts material, giving you an idea of how this impacts the shape.

   Note: Notice that as you drag the slider to the right, additional structures emerge. This indicates that you need to rerun the optimization with a higher percentage of material.



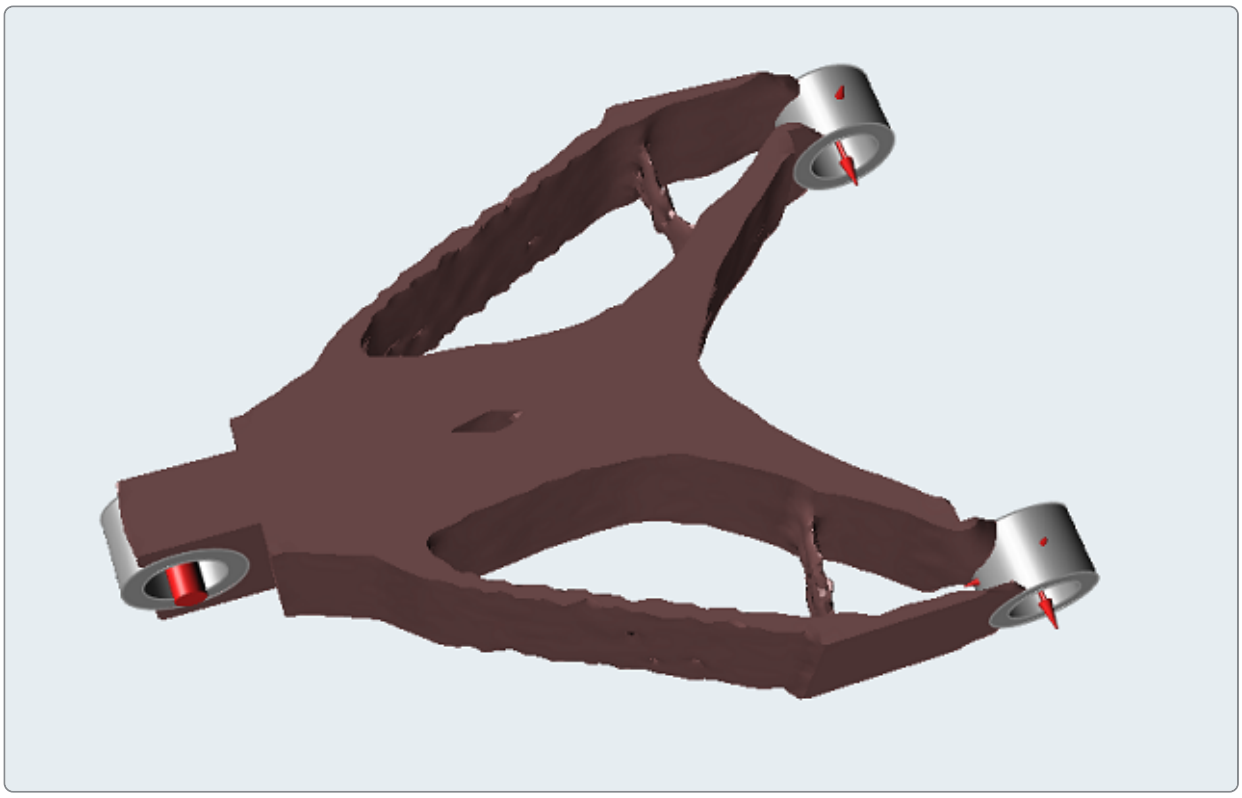
6. Click the **Run Optimization** icon to open the Run Optimization window.

7. Under Mass targets, change the **% of Total Design Space Volume** to **30** percent.



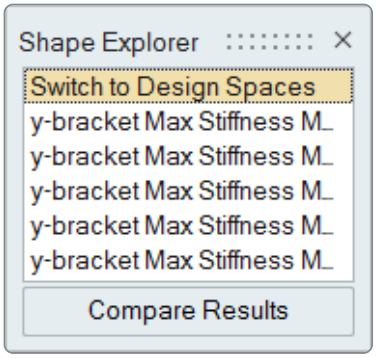
8. Click **Run**. When the optimization is complete, click the green flag to view the optimized shape.

9. This should produce a result similar to the one below:



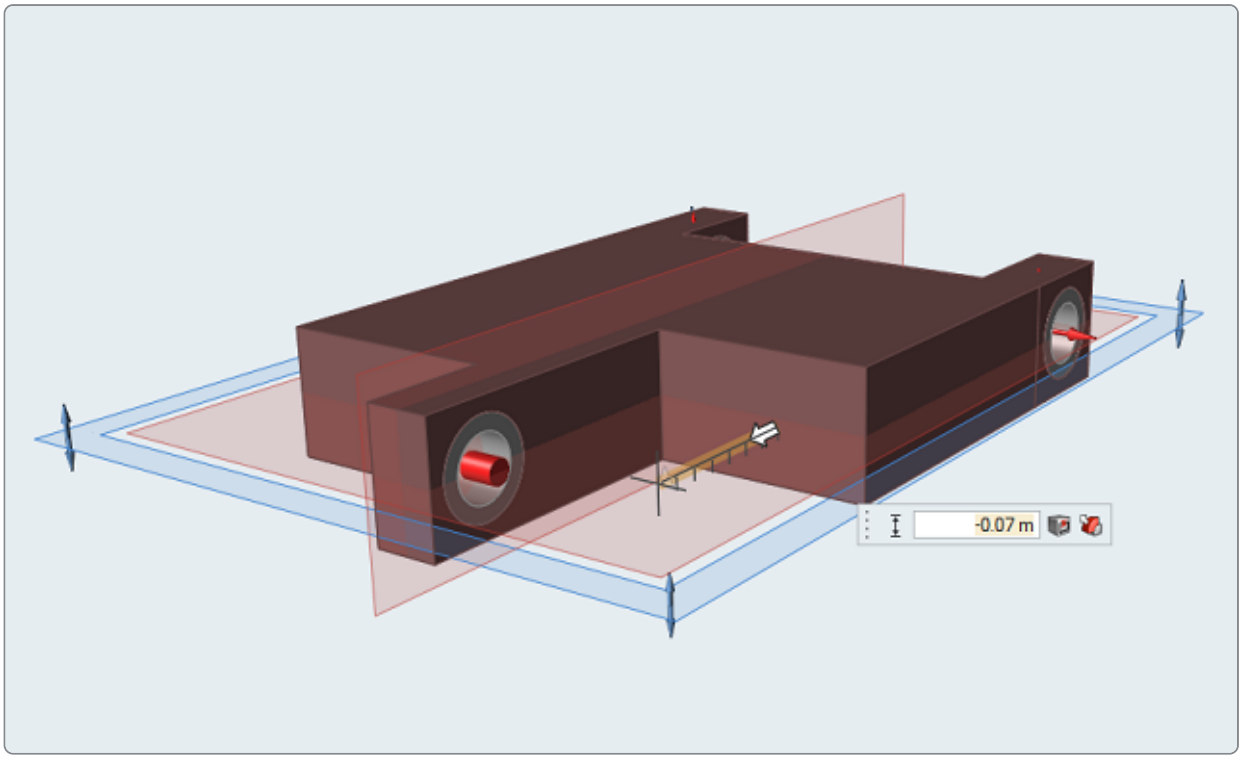
Step 11. Change the Design Space and Rerun Optimization

1. Select **Switch to Design Spaces** in the Shape Explorer to switch back to the original design space.

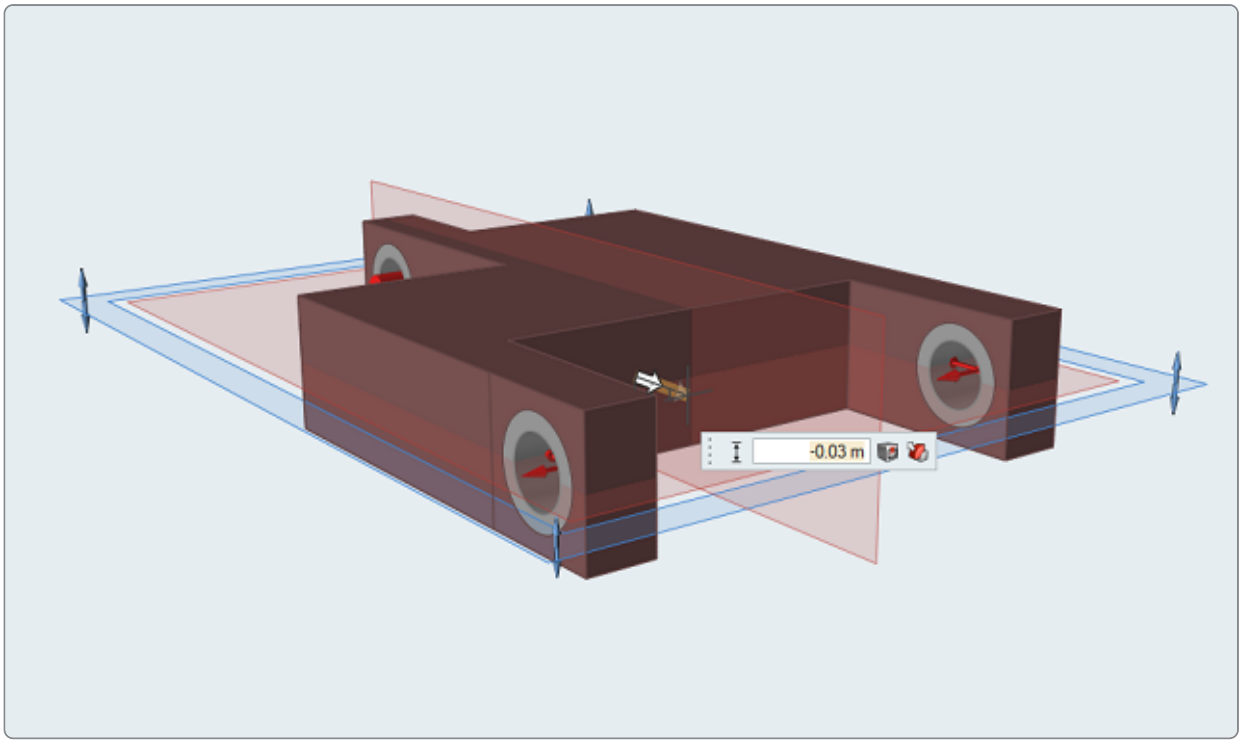


2. Select the **Push/Pull Faces** tool.

3. Reposition the model as shown below, then left-click on the right front face and push it inward `0.07 m` to make the design space asymmetric.

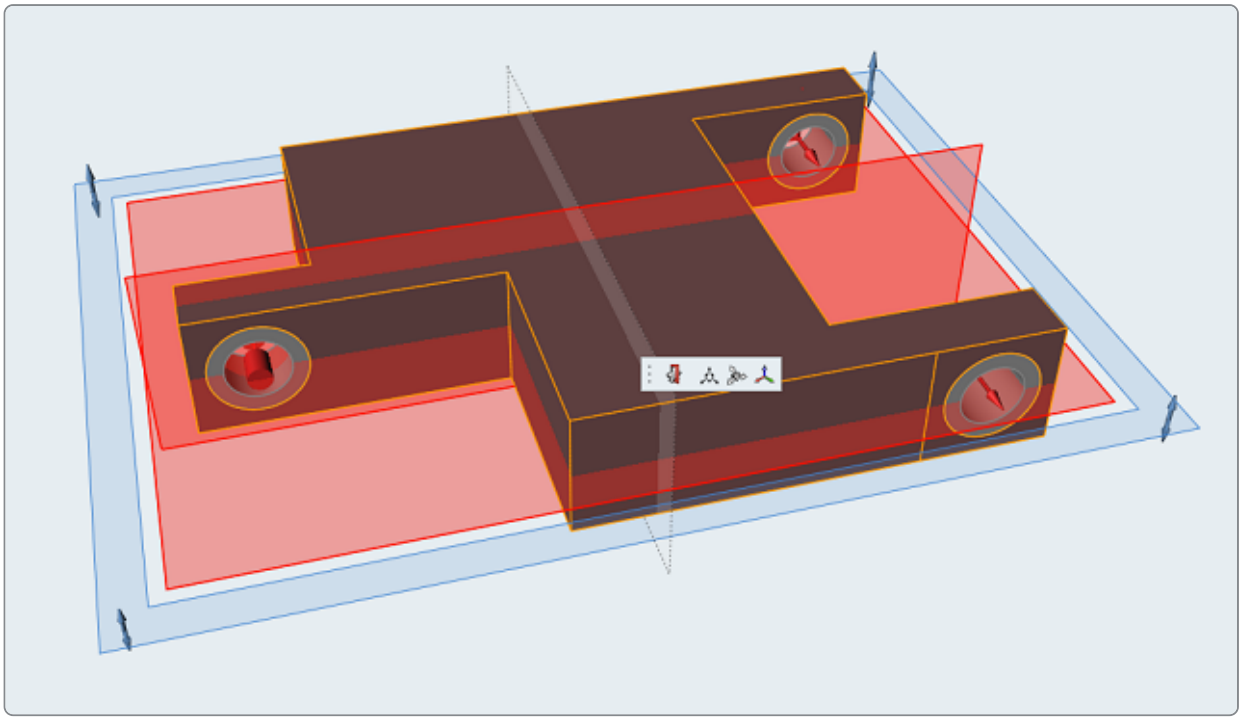


4. Rotate the model, then left-click on the rear face and push it inward `0.03 m` .

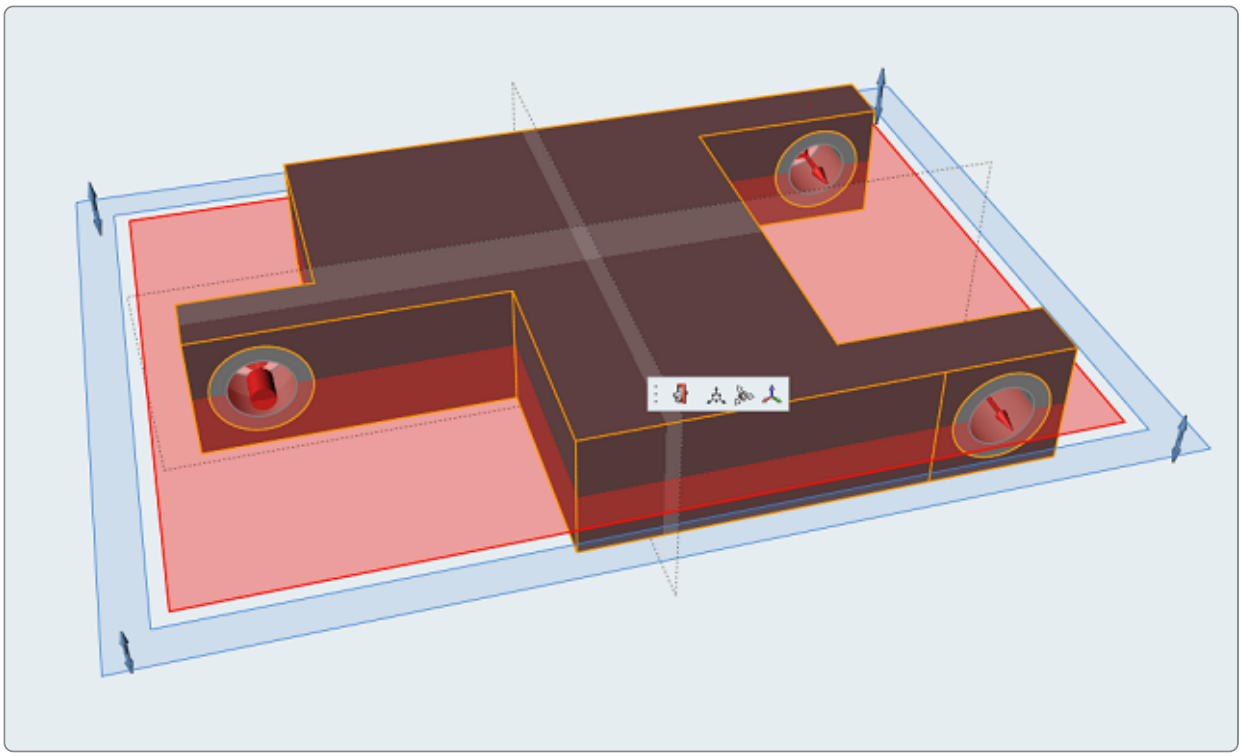


5. Double right-click to exit the Push/Pull Faces tool.

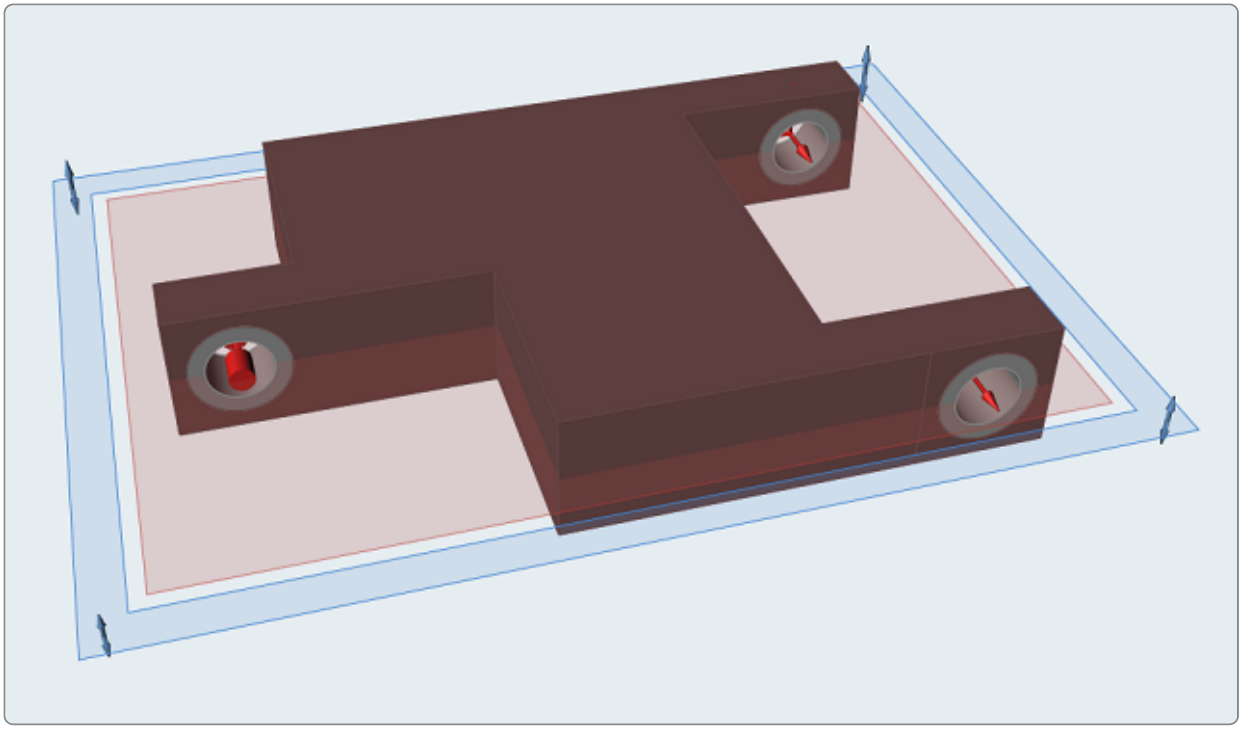
6. Reposition the model as shown, then double-click one of the red symmetry planes to activate the Symmetric tool.



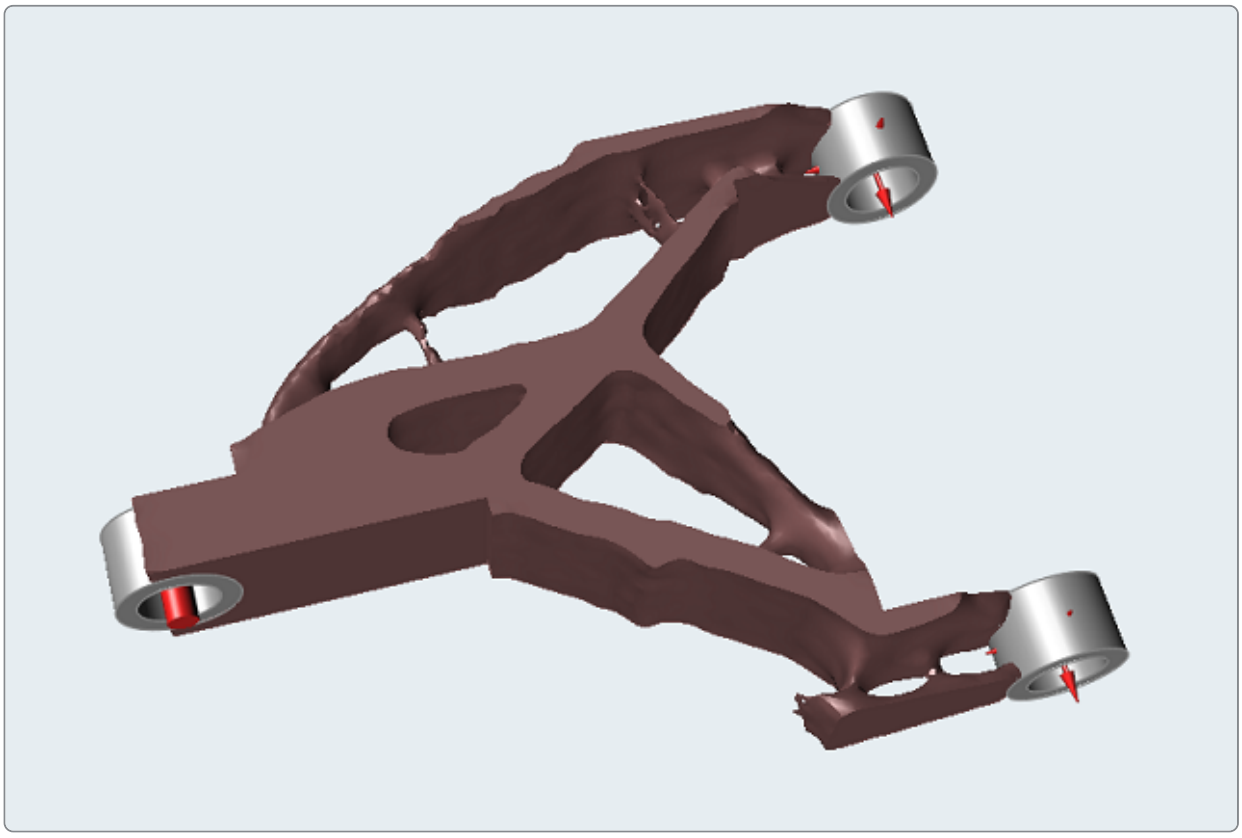
7. Click on the red plane in the z direction to deactivate it.



8. Double right-click to exit the tool.



9. Rerun optimization with all three load cases active and a mass target with **% of Total Design Space Volume** set to **30** percent.
   This should produce a result similar to the one below:



10. Adjust the **Topology** slider in the Shape Explorer to explore the optimized shape.

11. To save and export the optimized shape, refer to the Saving and Exporting Files tutorial.