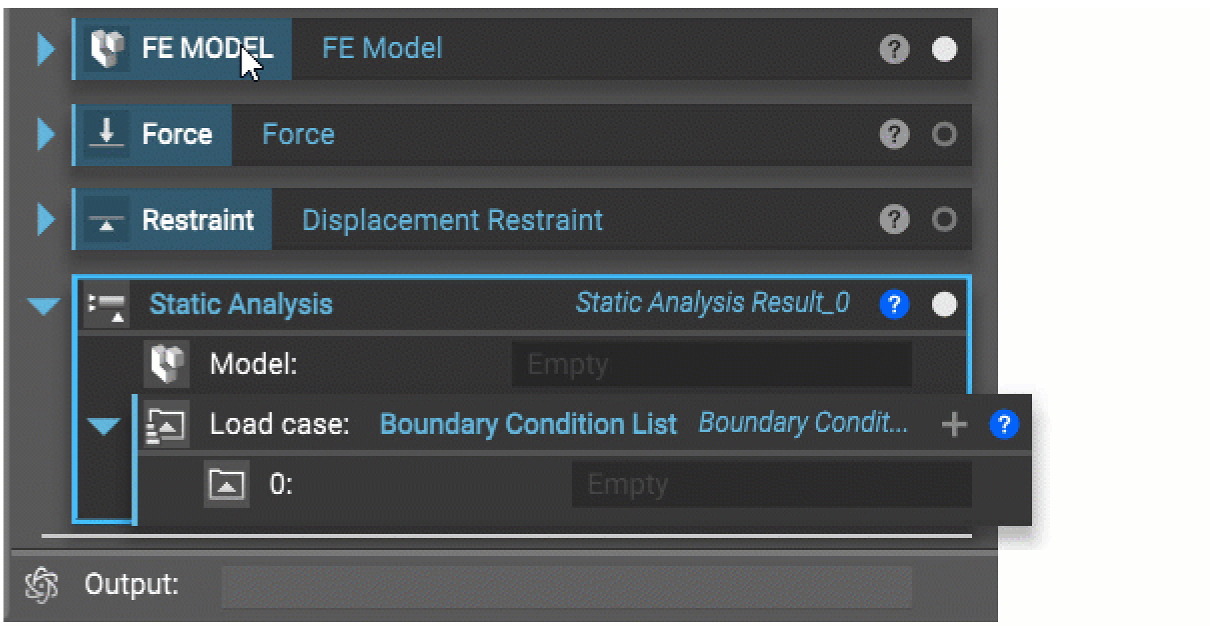
Follow Along: Running a Static Analysis

In this lesson, we will run the simulation of the brake pedal from the previous two Follow Along Videos.

Please use the completed [Boundary Conditions](#)to follow along with the tutorial.

Step 1: We will begin from scratch by first adding a static analysis block from the simulation tab to our notebook. Insert the **FE Model** variable into the Model input (found in the FE Model section of the notebook).

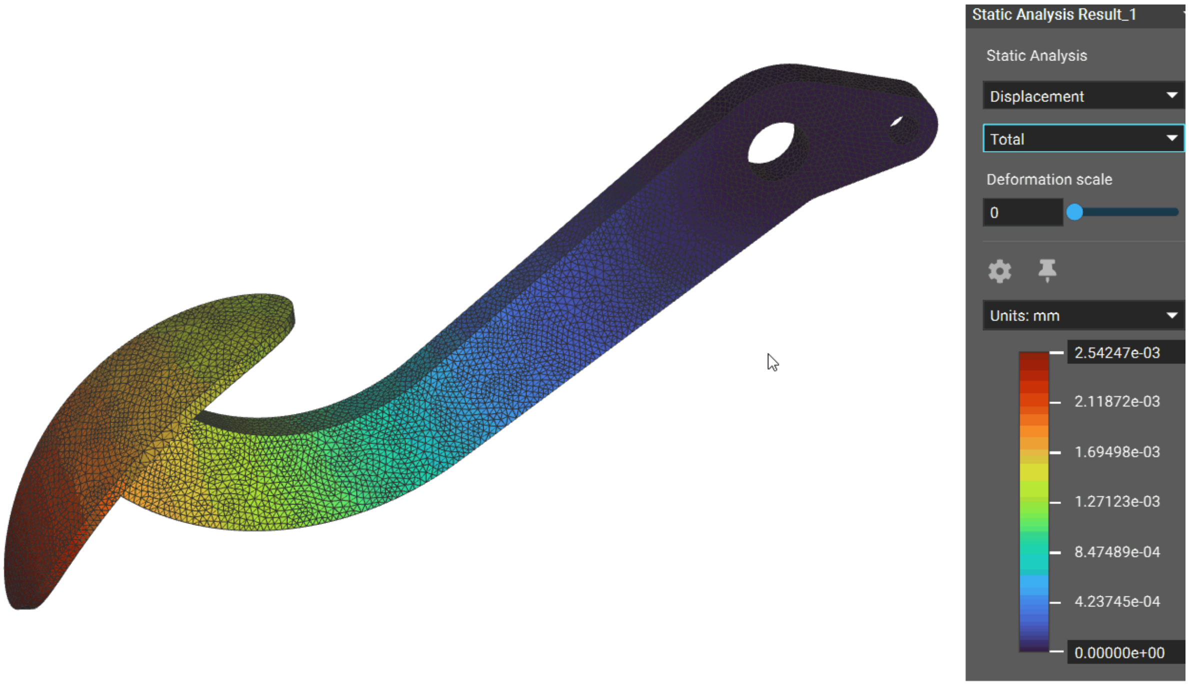
Step 2: Add another item in for the Boundary Conditions List and add the Force Variable and the Restraint variable (found in the Boundary Conditions Section of the notebook).



Step 3: Once you run an analysis, a Heads Up Display (HUD) will appear showing the results. The HUD allows you to toggle through several different types of results and allows you to edit the upper and lower bounds of each.

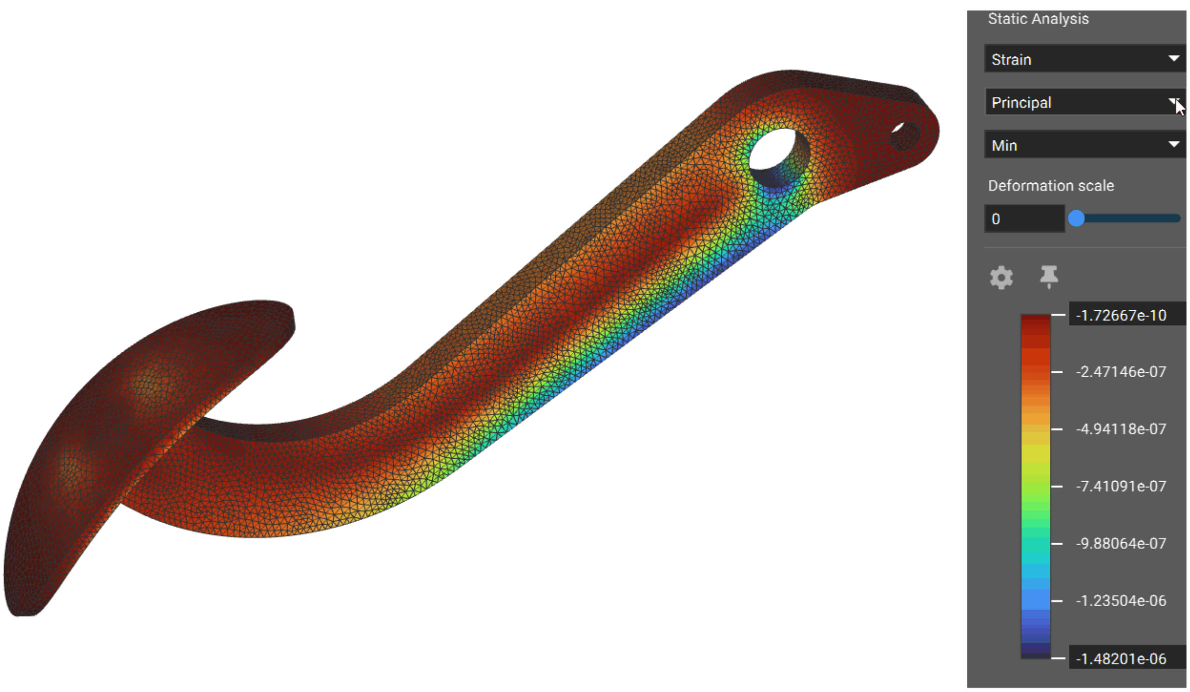
Displacement

The Displacement results show you how much your model moves under the given boundary conditions. You can see a scaled version of the displacement by clicking the Displacement checkmark and sliding the Scale Factor up. You can view either the Total or Directional displacement by changing the second combo box from the top of the HUD.



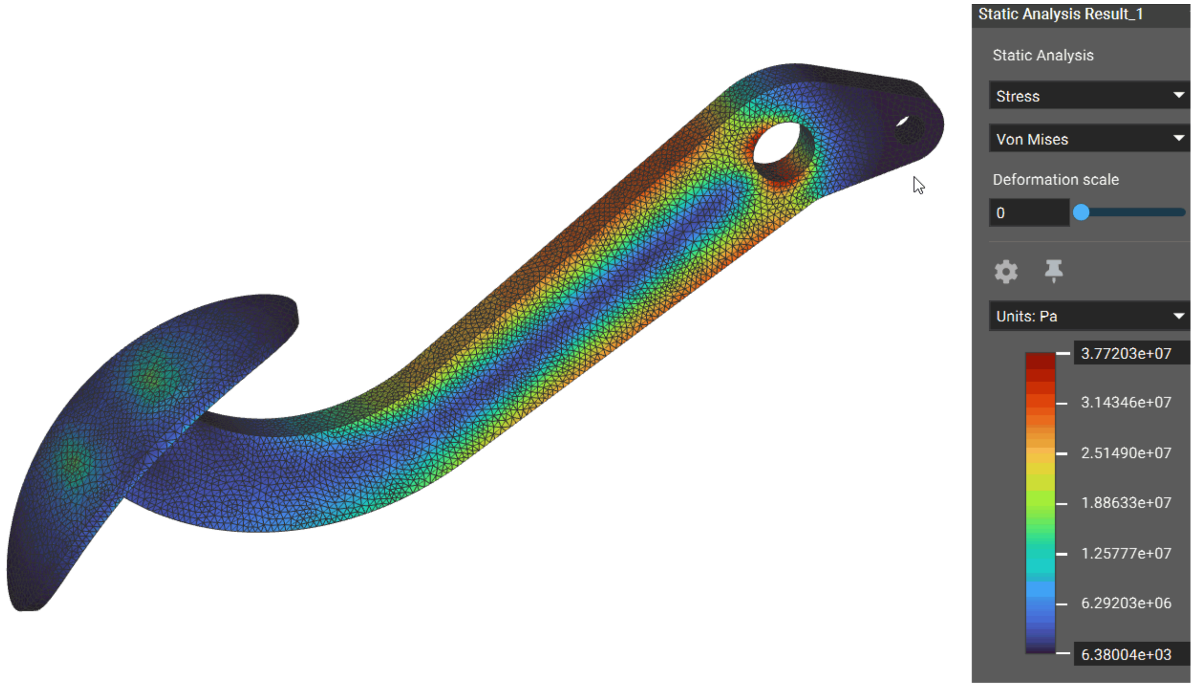
Strain

With the Strain option, you can view the Principal Strain for Min, Mid, and Max, or the Strain of Components.



Stress

With Stress, you can view the Von Mises Stress, the Principle Stress, or the Components Stress.

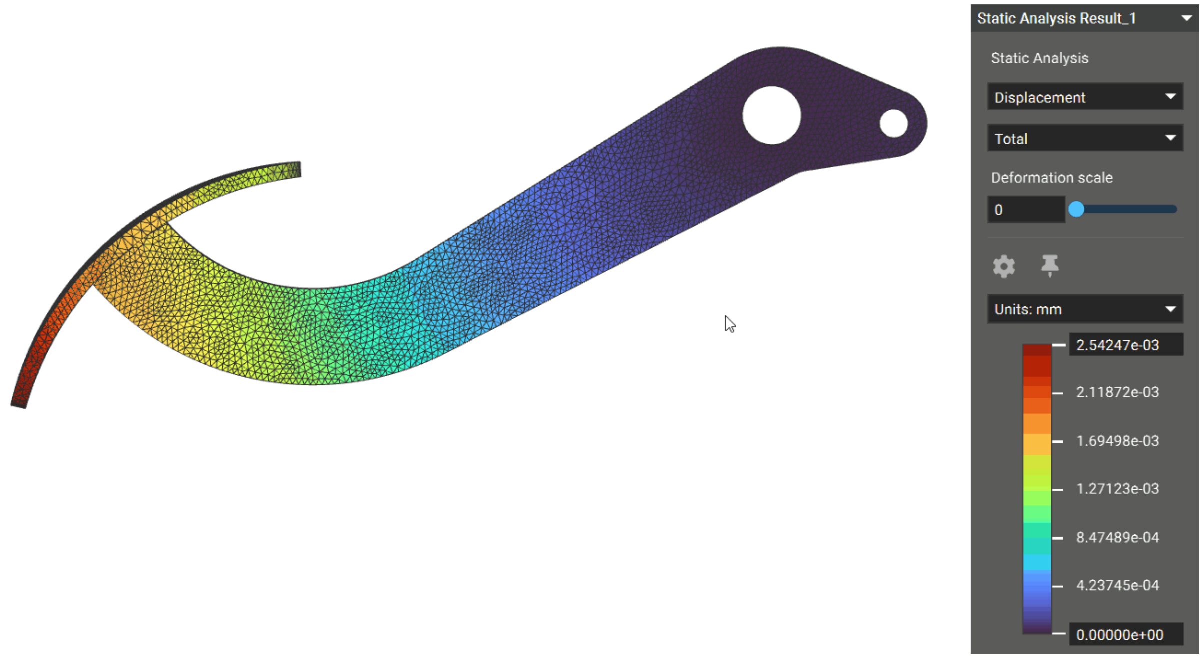


Reaction Forces

With Reaction Forces, you can see the Total or Directional forces within the model.

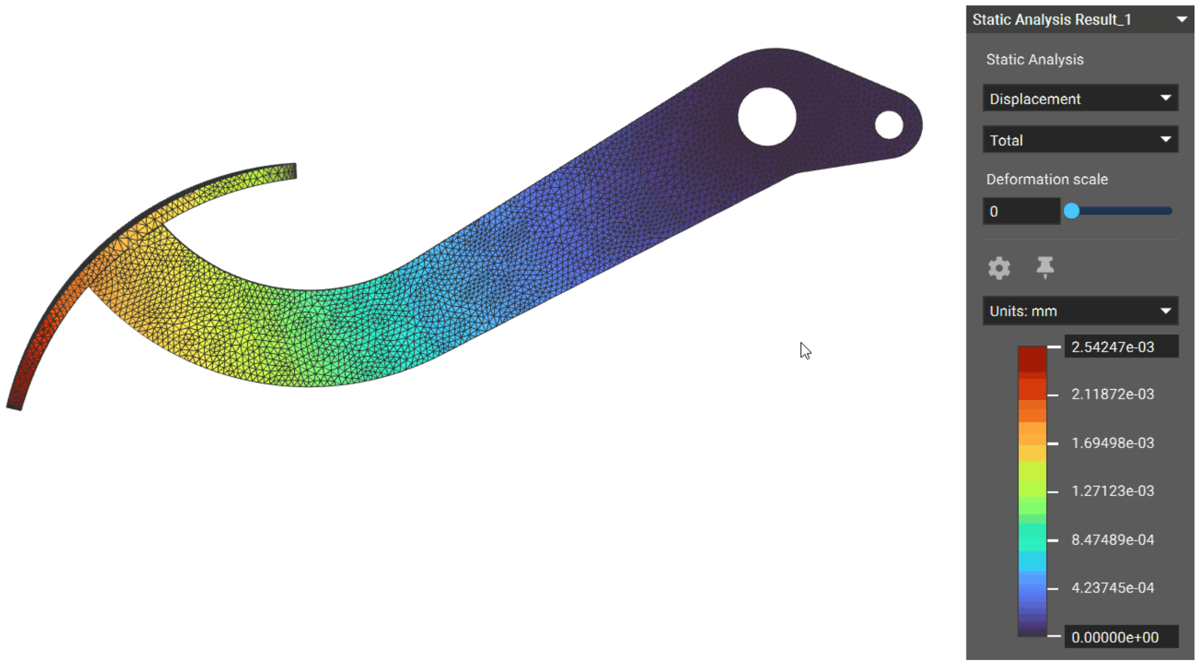
Color Settings

Edit the color palette as needed. You can choose between continuous and banded color transitions to better visualize and understand your simulation results.



Probe Tool

Use the Probe Tool to view simulation results at the vertices of individual mesh elements. Section cut the part and check “Show volume mesh elements” to probe elements inside the FE Mesh.



Simulation Result Fields

Open the Properties in the Block Details to view the generated scalar fields from your simulation results. These fields can be used to drive further design (see our [Intro to Field Driven Design]() course).