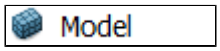
Modal Analysis of a Satellite - Mesh

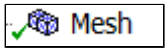
Author: Robert McBride, Cornell University

Mesh

Generating the Mesh

To open the mesher, double click the Model tab 

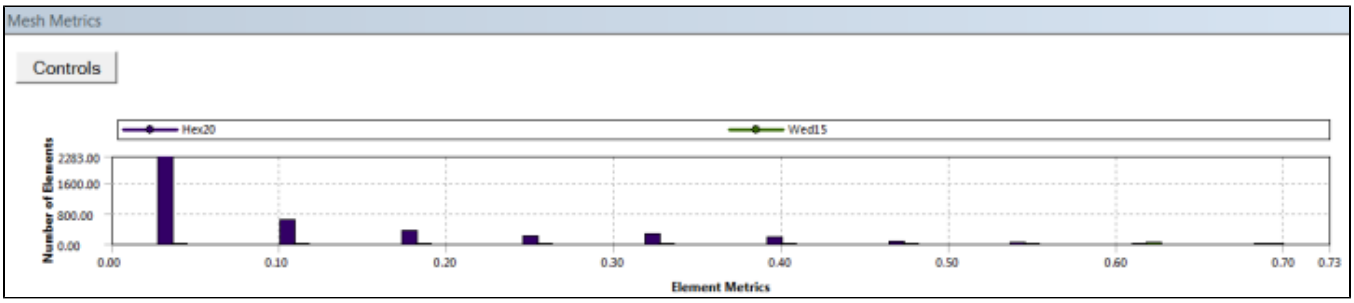
in the *Workbench* window. This will open up the *Design Modeler*.

From here, left click on the Mesh tab 

and maximize the **Sizing** tab in the detail window. Change the Element Size to 0.005m. To generate the mesh, right click on the **Mesh tab > Generate Mesh**.

I found this mesh size to give a relatively low skewness, implying more accurate element types on which theory is applied.

The skewness of a mesh can be viewed by selecting **Mesh,** maximizing the **Statistics** tab in the detail window, and selecting **Skewness** in the drop-down menu for **Mesh Metric**. The graph displays the number of elements and their skewness (0 corresponds to a perfect element type while 1 corresponds to a deformed element). From the skewness graph below, it can be seen that the majority of elements are of Hex20 type and low skewness.



Notes

This CubeSat's geometry has been greatly simplified to reduce the complexity of the mesh and thus reducing the run-time of the modal analysis. While this model only requires a reduction in element size, a more realistic CubeSat model would require mesh refinements and the application of different meshing methods. In particular, when dealing with a larger, more complex isogrid (face of triangular stiffeners), the Sweep Method is a great tool for reducing skewness.

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules