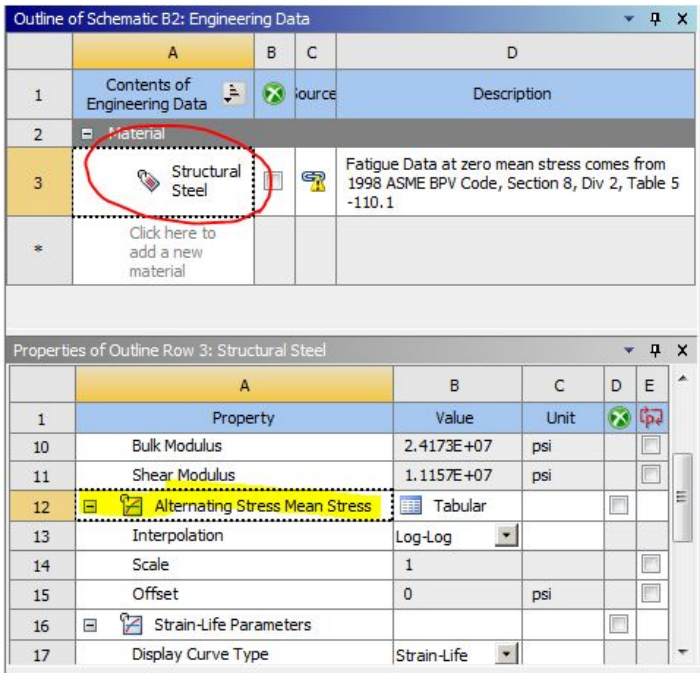
ANSYS - Fatigue Analysis

This tip will use the Tensile Bar tutorial to demonstrate fatigue analysis in Mechanical. It is recommended to become familiar with the tensile bar tutorial before you proceed. Please download the workbench model here.

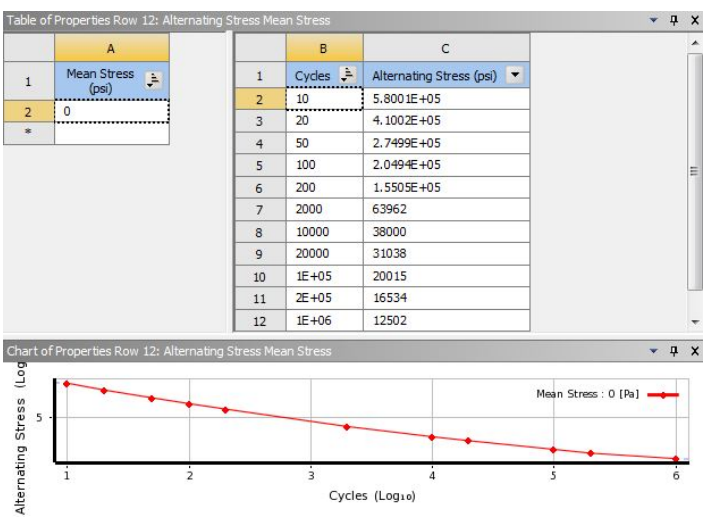
Please also see related discussion in this document.

Procedure

The provided workbench file already has the solution for the problem. Restore the downloaded archive and double click on ***Engineering Data***. In Engineering Data, click on Structural Steel and click on Alternating Stress Mean Stress.

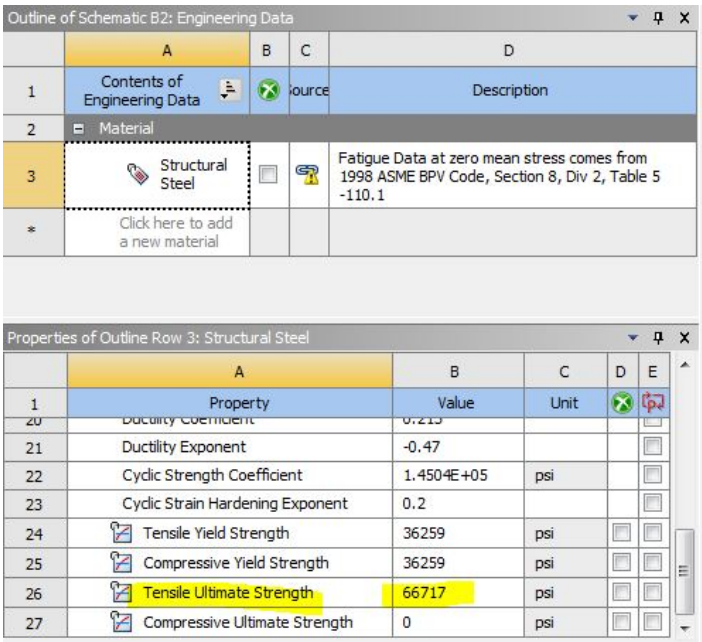


The window on the right shows tabulated alternating stress for steel and its mean stress.

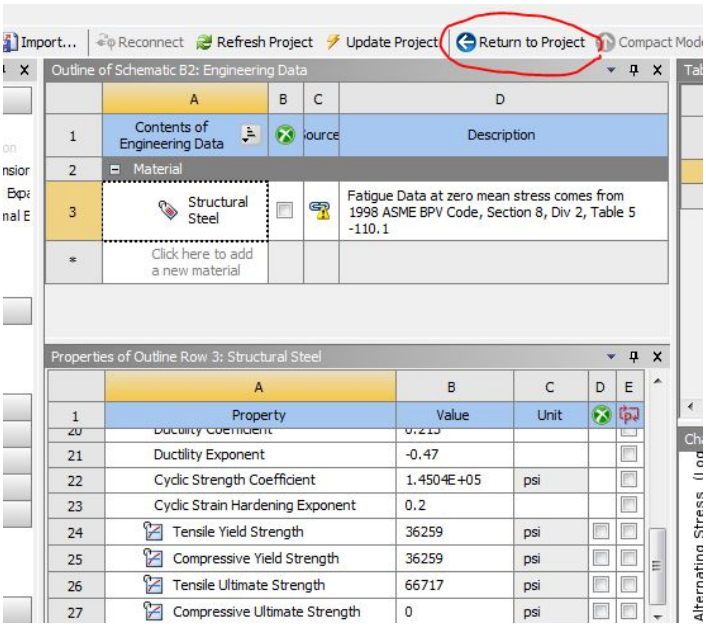


> **Tip**
>
> If you wish to use a new material to perform fatigue analysis, you must enter its alternating stresses at specific cycles before you specify its isotropic properties. The data for alternating stresses for various materials can be found in most strength of materials textbooks.

Verify that the Tensile Ultimate Strength is specified:

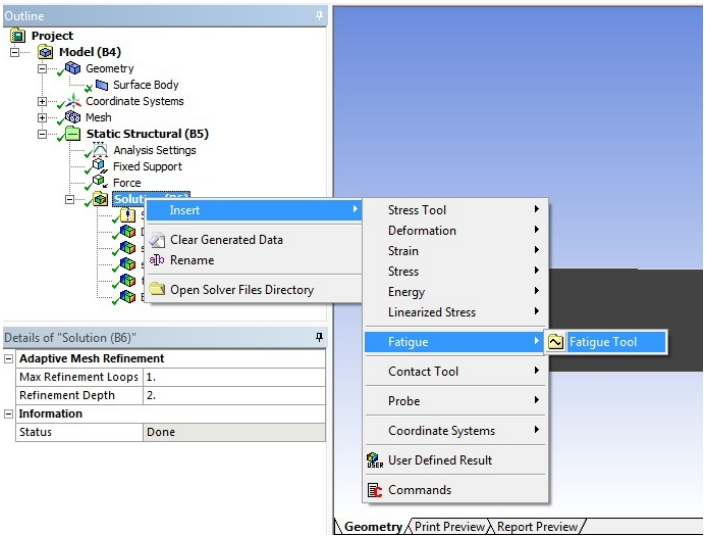


Click on Return to Project.

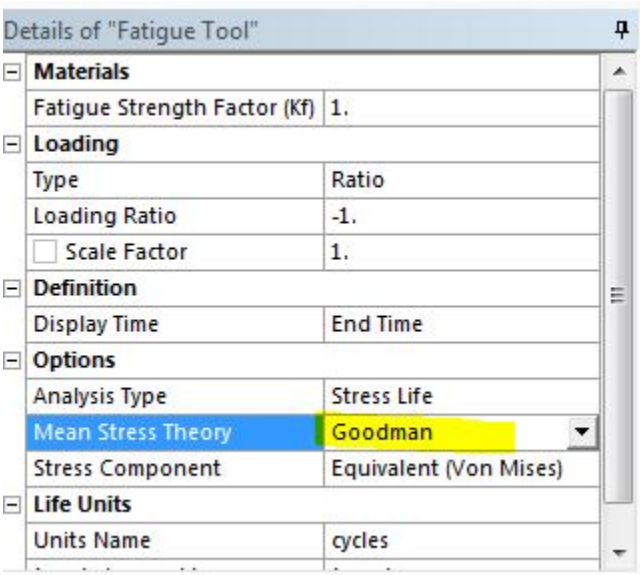


Double click on Model to launch ANSYS Mechanical. The boundary conditions have already been setup and the solution is provided.

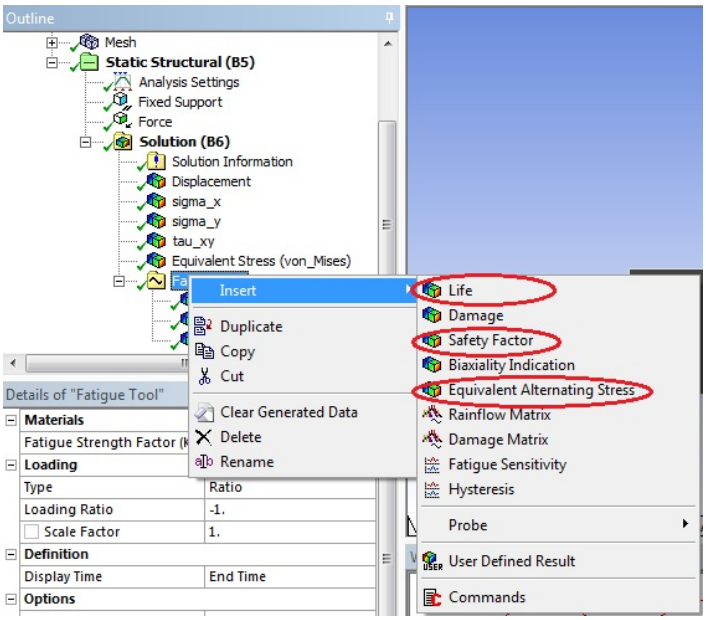
Right click on Solution and insert ***Fatigue Tool***.



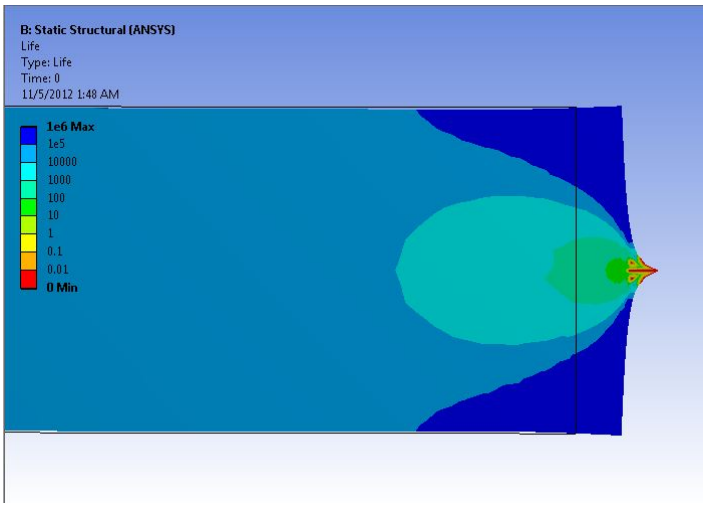
Highlight Fatigue Tool and select ***Goodman*** for Mean Stress Theory.

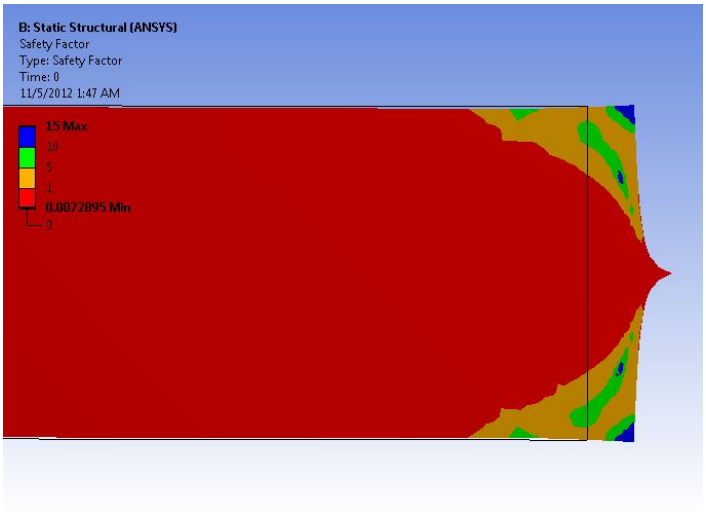


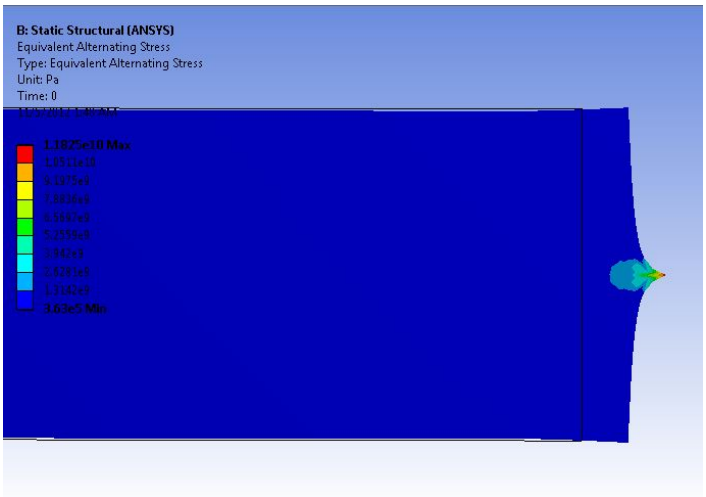
There are a number of options from the fatigue tool you can choose to include in the analysis, choose them accordingly. For demonstration, only life, safety factor, and alternating stress will be evaluated. Right click on Fatigue tool and insert Life, Safety Factor, and Equivalent Alternating Stress.



Click on Solve to generate new results. Zoom in on the right side of the bar by right click and drag. The fatigue analysis of a tensile bar subjected to a point load is shown below:







Specifying the loading ratio to get a loading scheme in ANSYS

Consider the case where the load varies smoothly between 500N and -1000N on each cycle. You first perform static loading at -1000N. For the fatigue portion, you must specify the loading ratio in the Fatigue tool (under "Loading Type", select "Ratio") to let ANSYS know what your load cycle will be. This is where ANSYS will know the ratio of your loading scheme (which is varying from 500N to -1000N). This value should be set to -0.5. The resulting sinusoidal graph produced on the right, should vary from an amplitude of 1 to -0.5. ANSYS uses the -1000N in the static loading as a starting point. What this means in ANSYS is that the static loading boundary condition of -1000N is going to be applied as the "max" (ie -1000*1 = -1000N) and then for the "min" it will be 500N (ie -1000*-0.5 =500N).