

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](#).

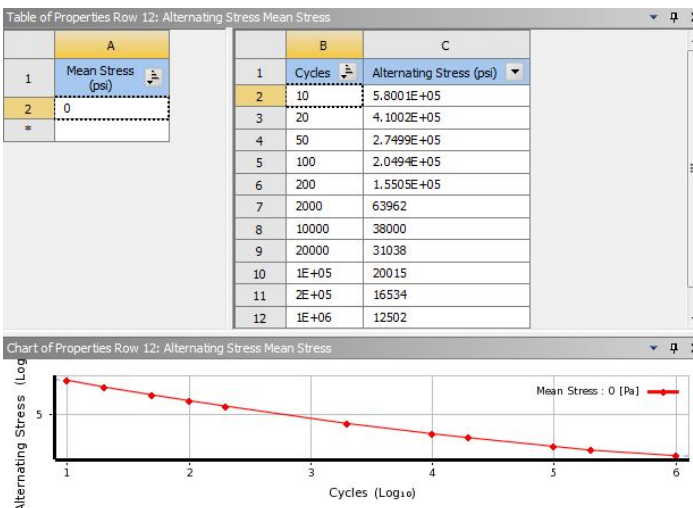
Procedure

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 screenshot shows the ANSYS Engineering Data interface. The top window, 'Outline of Schematic B2: Engineering Data', has a table with columns A, B, C, and D. Row 3, 'Structural Steel', is highlighted with a red circle. Below it, the 'Properties of Outline Row 3: Structural Steel' window is open, showing a table with columns A, B, C, D, and E. Row 12, 'Alternating Stress Mean Stress', is highlighted in yellow. The 'Tabular' option is selected for the 'Interpolation' property.

	A	B	C	D	E
1	Property	Value	Unit		
10	Bulk Modulus	2.4173E+07	psi		
11	Shear Modulus	1.1157E+07	psi		
12	Alternating Stress Mean Stress	Tabular			
13	Interpolation	Log-Log			
14	Scale	1			
15	Offset	0	psi		
16	Strain-Life Parameters				
17	Display Curve Type	Strain-Life			

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:

Outline of Schematic B2: Engineering Data				
	A	B	C	D
1	Contents of Engineering Data	Source	Description	
2	Material			
3	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
*	Click here to add a new material			

Properties of Outline Row 3: Structural Steel				
	A	B	C	D
1	Property	Value	Unit	
20	Ductility Coefficient	0.215		
21	Ductility Exponent	-0.47		
22	Cyclic Strength Coefficient	1.4504E+05	psi	
23	Cyclic Strain Hardening Exponent	0.2		
24	Tensile Yield Strength	36259	psi	
25	Compressive Yield Strength	36259	psi	
26	Tensile Ultimate Strength	66717	psi	
27	Compressive Ultimate Strength	0	psi	

Click on Return to Project.

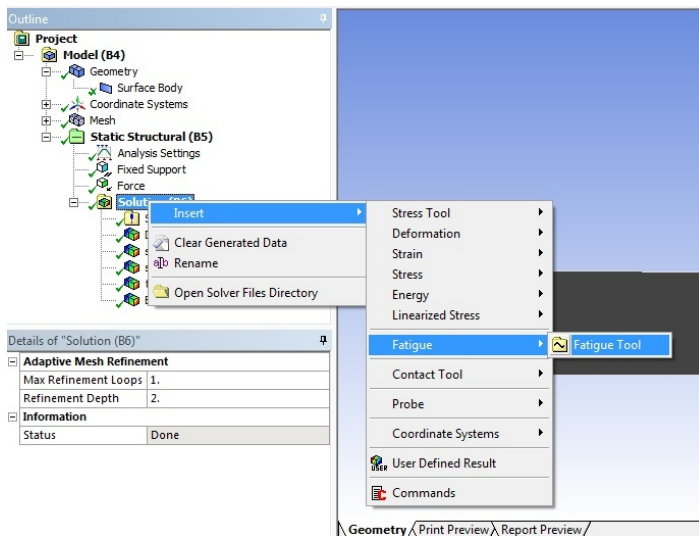
Import... Reconnect Refresh Project Update Project **Return to Project** Compact Mode

Outline of Schematic B2: Engineering Data				
	A	B	C	D
1	Contents of Engineering Data	Source	Description	
2	Material			
3	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
*	Click here to add a new material			

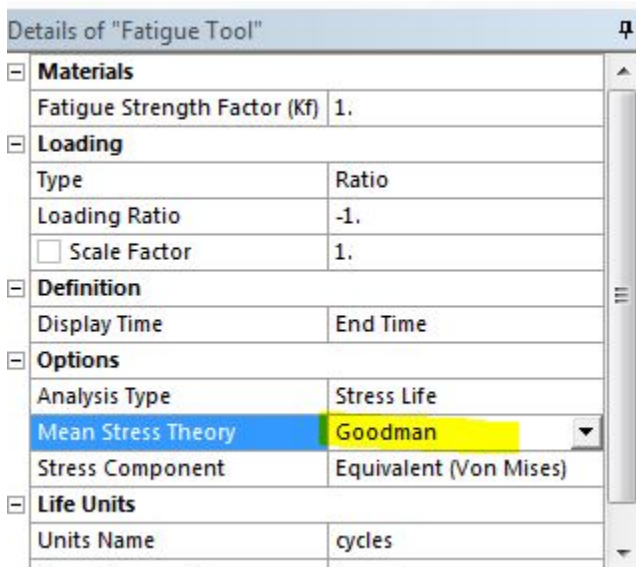
Properties of Outline Row 3: Structural Steel				
	A	B	C	D
1	Property	Value	Unit	
20	Ductility Coefficient	0.215		
21	Ductility Exponent	-0.47		
22	Cyclic Strength Coefficient	1.4504E+05	psi	
23	Cyclic Strain Hardening Exponent	0.2		
24	Tensile Yield Strength	36259	psi	
25	Compressive Yield Strength	36259	psi	
26	Tensile Ultimate Strength	66717	psi	
27	Compressive Ultimate Strength	0	psi	

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

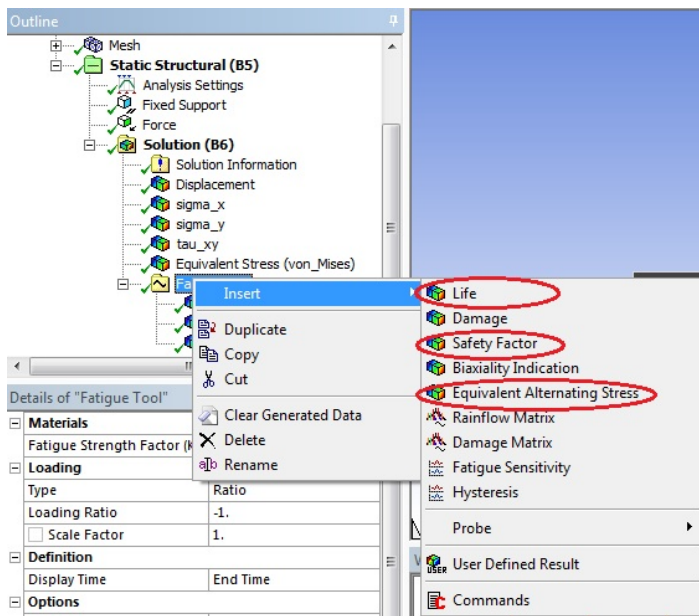
Right click on Solution and insert *Fatigue Tool*.



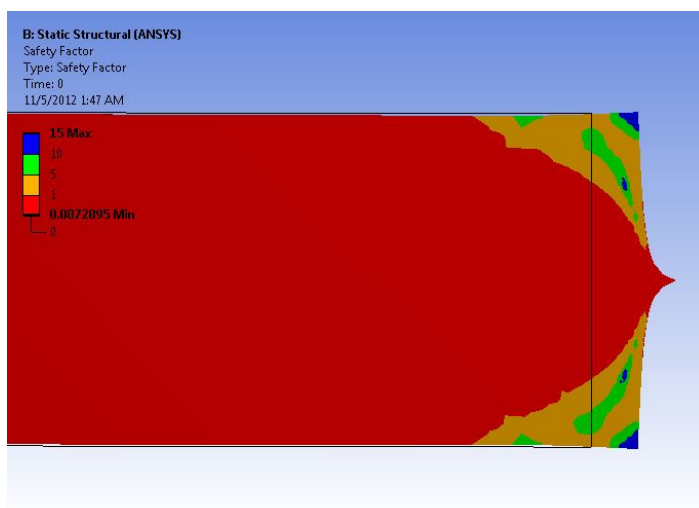
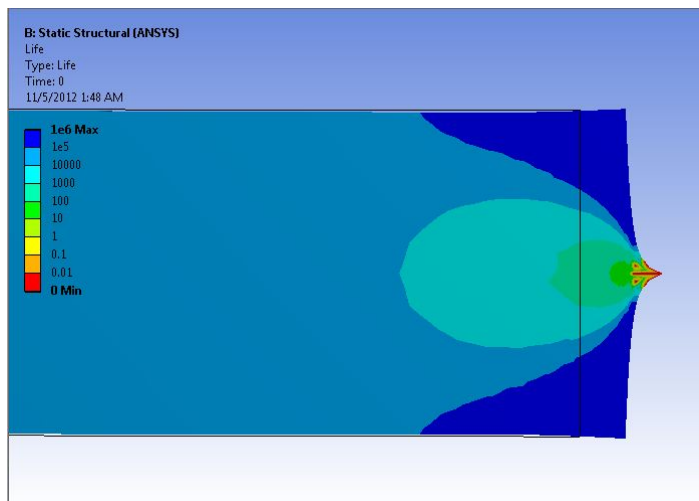
Highlight Fatigue Tool and select **Goodman** for Mean Stress Theory.

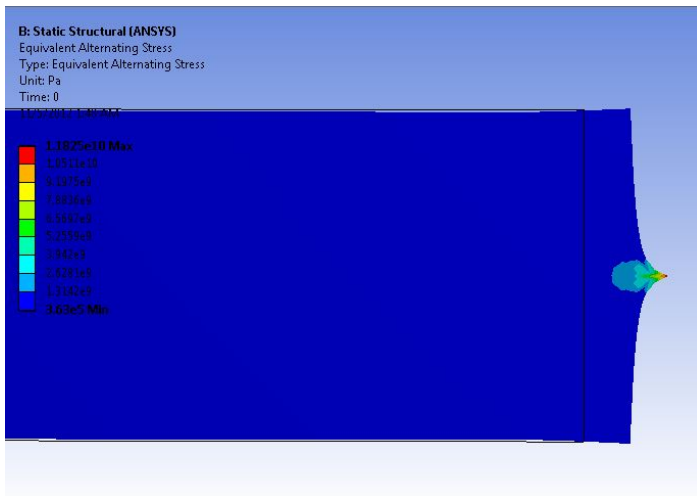


There are a number of options from the fatigue tool you can choose to include in the analysis, choose them accordingly. For demonstration, only life, safety factor, and alternating stress will be evaluated. Right click on Fatigue tool and insert Life, Safety Factor, and Equivalent Alternating Stress.



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 $-1000 \times 1 = -1000\text{N}$) and then for the "min" it will be 500N (ie $-1000 \times -0.5 = 500\text{N}$).