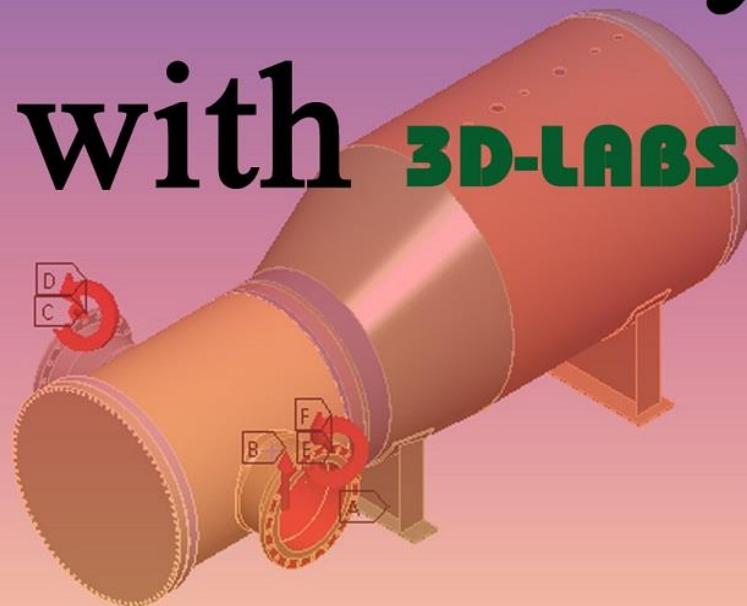


Basics of Finite Element Analysis with 3D-LABS

B: Transient Structural
Transient
Time: 33000 s

- A** Fixed Support
- B** Pressure: 0.245 MPa
- C** Force: 27872 N
- D** Moment: 5.0965e+007 N-mm
- E** Force 2: 46274 N
- F** Moment 2: 9.0539e+007 N-mm



0.00 1000.00 2000.00 (mm)
500.00 1500.00



- Introduction
- Why Should we do Analysis
- Elastic Stress Analysis Method
- Static Structural Analysis
 - Engineering Data
 - Geometry
 - Model
 - Setup
 - Solution
 - Results
- Fatigue Analysis
 - Fatigue Tool
 - Solution
 - Results

Prepared by: Gokul

Company: 3D-LABS

Approved by: Abdul Khader

Date: 30-09-2017



1. INTRODUCTION

Ansys is basically an analysis software, which allows to analyse the CAD model of the product before producing. It is used to test its strength and test of what would it behave in its environment. All types of analysis test Such as thermal, fatigue, structural, CFD etc... can be conducted. It is basically a simulation software to visualize working model without actually making it in the real world.

2. WHY SHOULD WE DO ANALYSIS

Let's consider you are manufacturing a product. In the first place you need to manufacture some prototypes of the product and test it for all kinds of worse situation that could happen to damage the product. In some cases, you need to conduct few tests to optimize the material required, the design, the loads if any acting on the product, the natural frequency or its temperature controlling mechanism, and many more. In order to know the every detail of your product you need to manufacture prototypes. This would eventually lead to increase the cost to company.

In that case, we use the analysis software's like ANSYS. ANSYS uses the certain inputs and evaluates the product behaviour in their environment. This would certainly reduce the cost of producing prototypes and mainly the time. In this competitive world the accuracy and time are the most deciding factors for the company or the organization to sustain. ANSYS helps in increasing the accuracy and decreasing the time of outcome of the final product. ANSYS can import all kinds of CAD geometries (3D and 2D) from different CAD software's and perform simulations. ANSYS has inbuilt CAD developing software's like Design Modeler and Space Claim which makes the work flow even smoother.

3. ELASTIC STRESS ANALYSIS METHOD

Here, we going to study about the stress analysis of vessel. Stress analysis is the determination of the relationship between external forces applied to a vessel and the corresponding stresses. The starting place for stress analysis is to determine all the design condition for a given problem and then determine all the related external forces. We must then relate these external forces to the vessel parts which must resist them to find the corresponding stresses.

Refer ASME_BPVC_VIII-2_2015 part-5 for design by analysis requirements. To evaluate protection against plastic collapse, the results from an elastic stress analysis of the component subject to defined loading conditions are compared to an associated limiting value.

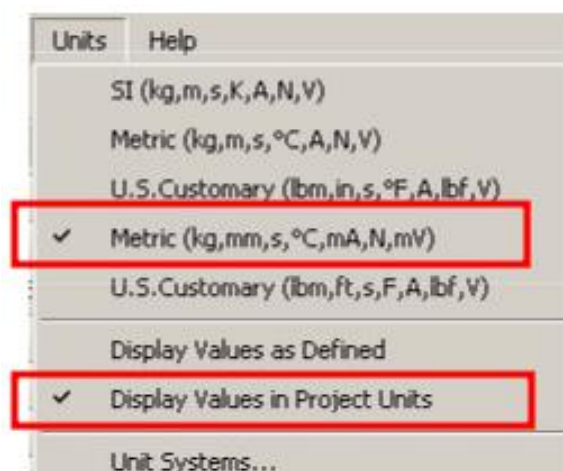
(1) A quantity known as the equivalent stress is computed at locations in the component and compared to an allowable value of equivalent stress to determine if the component is suitable for the intended design conditions. The equivalent stress at a point in a component is a measure of stress, calculated from stress components utilizing a yield criterion, which is used for comparison with the mechanical strength properties of the material obtained in tests under uniaxial load.

(2) The maximum distortion energy yield criterion shall be used to establish the equivalent stress. In this case, the equivalent stress is equal to the von Mises equivalent stress given by equation below

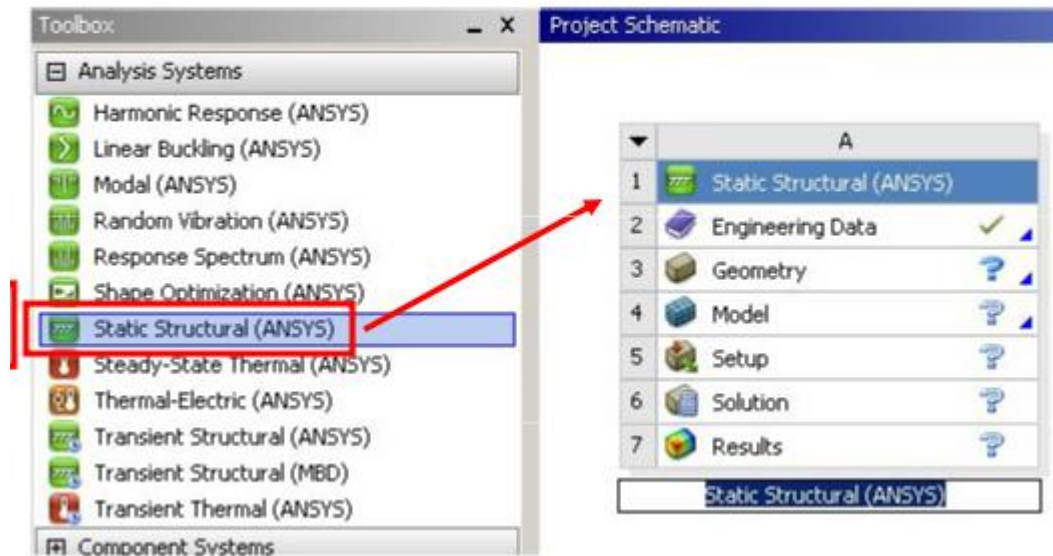
$$S_e = \sigma_e = \frac{1}{\sqrt{2}} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]^{0.5}$$

4. STATIC STRUCTURAL ANALYSIS

The model consists of a IGES file. Here, our goal is to verify the model will function in its intended environment. Open the Ansys workbench and set the units in “**Metric** (kg, mm, s, °C, mA, N, mV).



From the Toolbox, choose **Static Structural system** (drag/drop or double click).

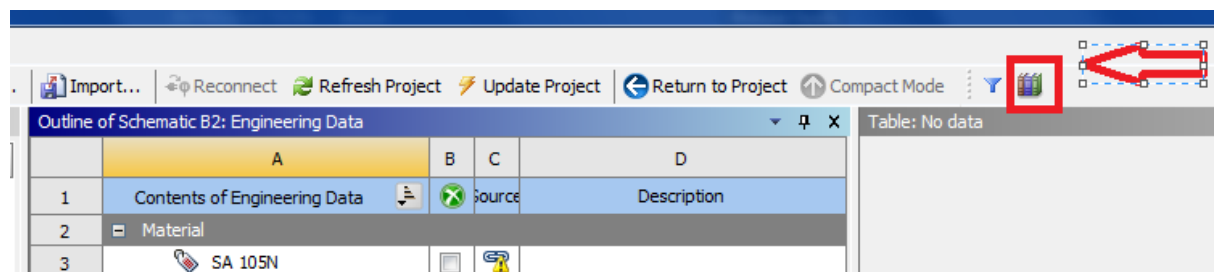


steps in static structural analysis,

1. Engineering data
2. Geometry
3. Model
4. Setup
5. Solution
6. Results

4.1. ENGINEERING DATA

- In the static structural workspace, double click the engineering data.
- If you need to take the standard material which are already placed in the engineering data.
- Go to engineering data sources in the toolbar



- Then select type of material and what material that you needed and ADD it .

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

Engineering Data Sources

	A	B	C	D
1	Data Source		Location	Description
3	General Materials	<input type="checkbox"/>		General use material samples for use in various analyses.
4	General Non-linear Materials	<input type="checkbox"/>		General use material samples for use in non-linear analyses.
5	Explicit Materials	<input type="checkbox"/>		Material samples for use in an explicit analysis.

Outline of General Materials

	A	B	C	D	E
1	Contents of General Materials		Add	Source	Description
2	Material				
3	Air				General properties for air.
4	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
5	Aluminum Alloy				General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
6	Concrete				
7	Copper Alloy				
8	Gray Cast Iron				
9	Magnesium Alloy				

Table of Properties Row

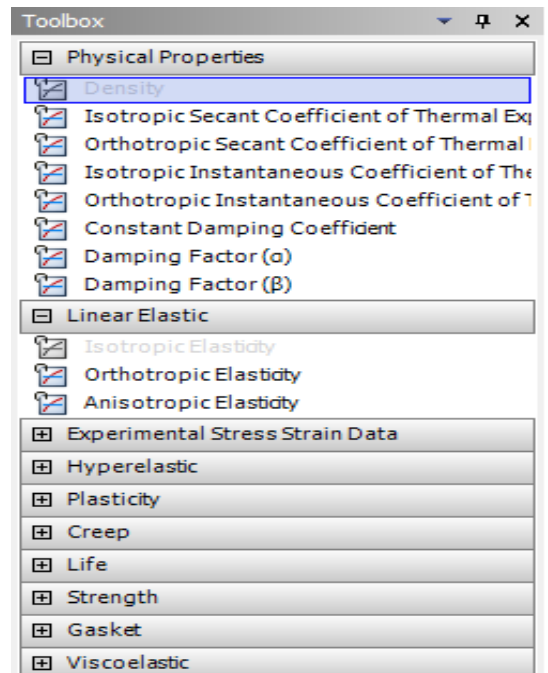
Chart of Properties Row

- To return to the workspace, again click engineering data sources.
- You can also add new material which are not present in the engineering data.
- In the engineering data toolbar, click to add a new material and give the name for new material (like SA 105N, SA 106 gr.B)

Outline of Schematic A2: Engineering Data

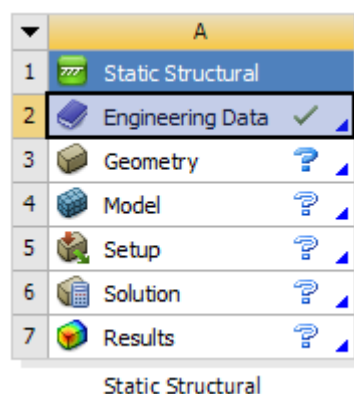
	A	B	C	D
1	Contents of Engineering Data		Source	Description
2	Material			
3	Structural Steel	<input type="checkbox"/>		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
4	SA 105N	<input type="checkbox"/>		
*	Click here to add a new material			

- In the toolbox, select physical properties and double click the density. Then, select the linear elastic and double click the isotropic elasticity
- In the properties of outline row. Give the value for each property to the corresponding units.
- The values are taken from ASME_BPVC_IID-M_2015 for the corresponding materials.
- Select **UPDATE PROJECT** for updating the data. Then, select **RETURN TO PROJECT**.



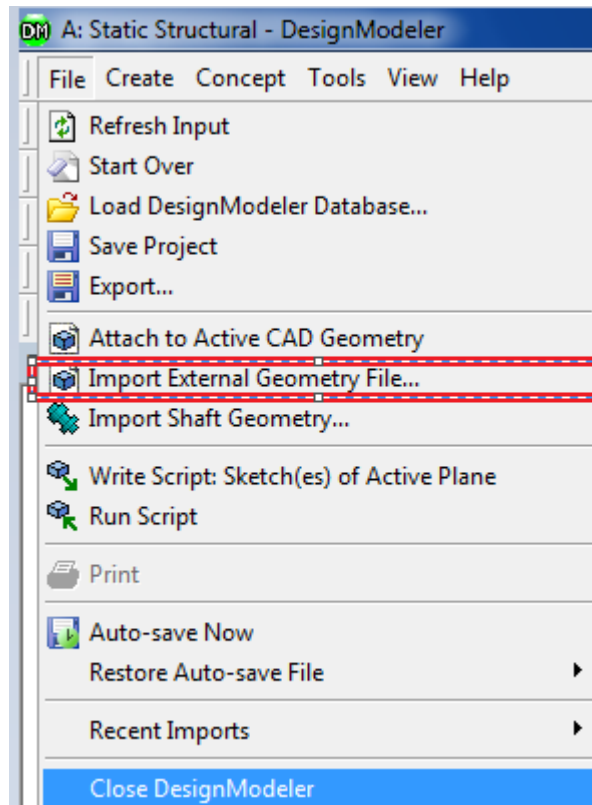
Properties of Outline Row 3: SA 105N					
	A	B	C	D	E
1	Property	Value	Unit		
2	Density		kg m ⁻³		
3	Isotropic Elasticity				
4	Derive from	Young's...			
5	Young's Modulus		Pa		
6	Poisson's Ratio				
7	Bulk Modulus		Pa		
8	Shear Modulus		Pa		

- Then, you will see the tick mark as shown in below figure. This Tick mark tells you that you have updated the engineering data & question mark in Geometry, Model, and Setup indicates such data still needs to be updated.

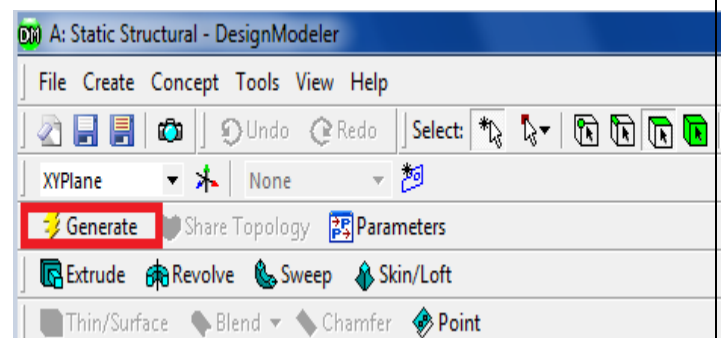


4.2. GEOMETRY

- In the static structural, double click the geometry. A **design modeller** will open to draw (or) import the geometry.
- To import the geometry, select the File —————> Import external geometry file.

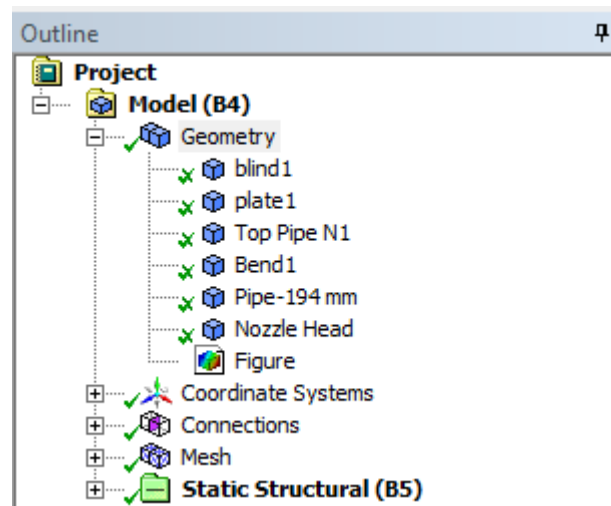


- Select the source file and then open it.
- Select the generate option in the workspace to generate the geometry.
- Close the design modeller and save it.
- Then, you will see the tick mark in the geometry box.

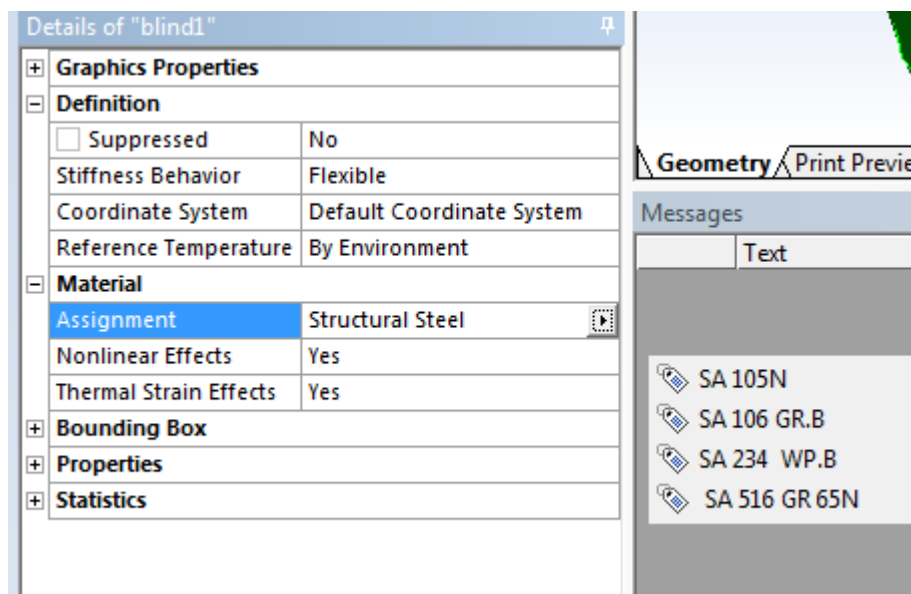


4.3.MODEL

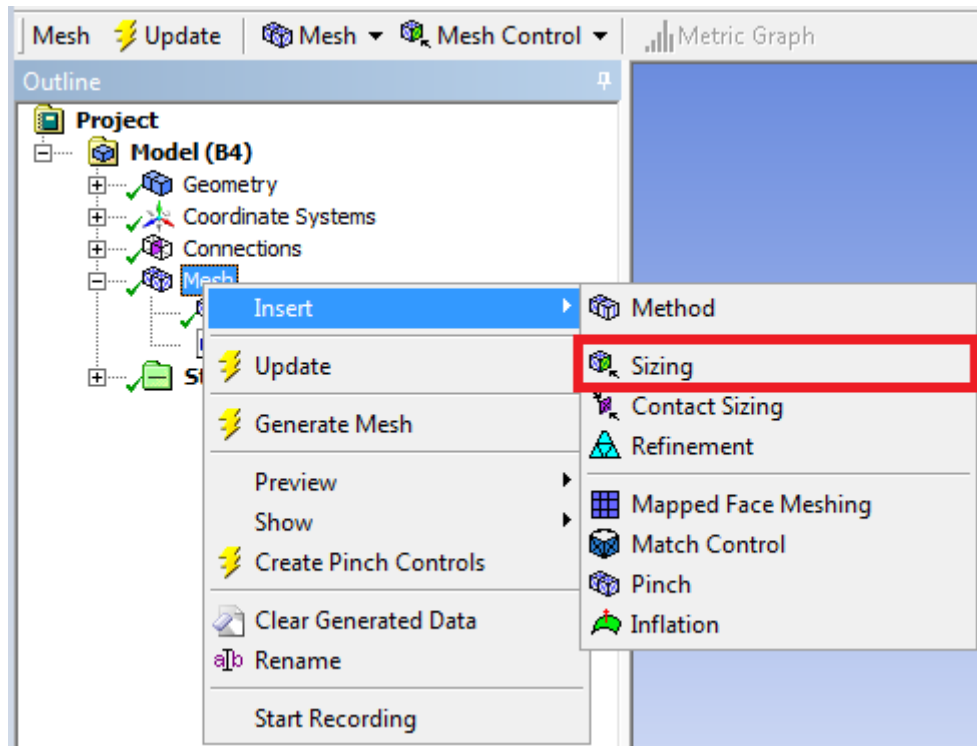
- In the static structural, double click the model. A **static structural - Mechanical** will open to analyse the model.
- In the toolbar, select geometry and then give name for each parts.



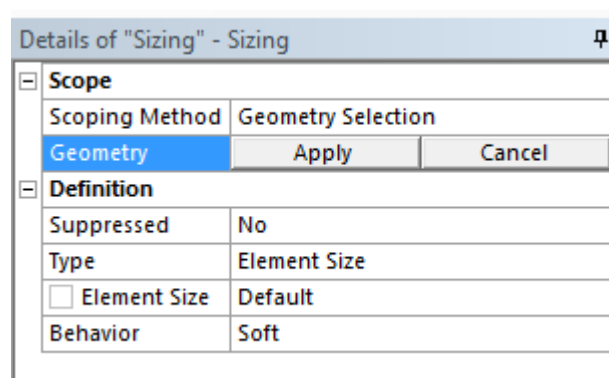
- In the lowest part of the toolbar, select material option and then select corresponding material instead of structural steel.



- Then, mesh the model automatically by **update** option in the toolbar. For accurate results, Increase the nodes and elements of the model by Mesh **sizing** option.
- Select the Mesh option → Insert → Sizing option.



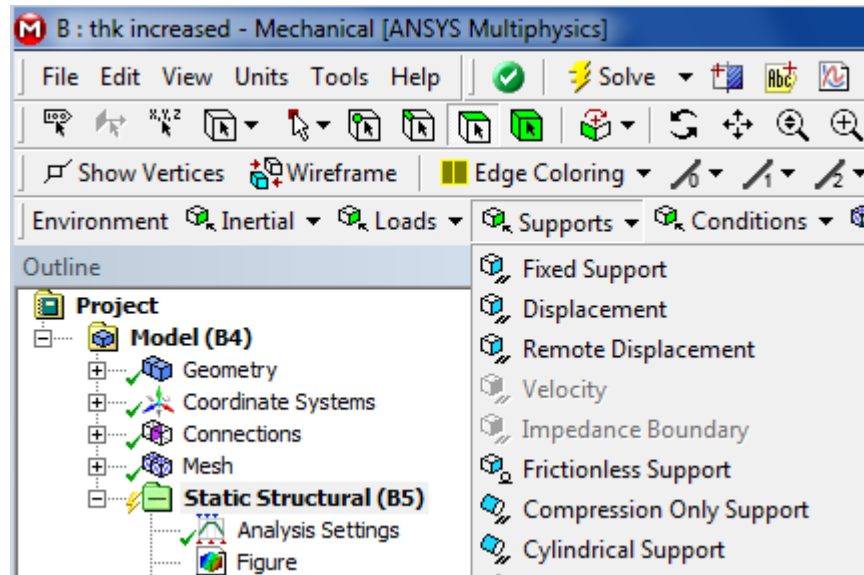
- Select the body (or) face for meshing and then apply.
- Provide a suitable value in element size instead of default value.



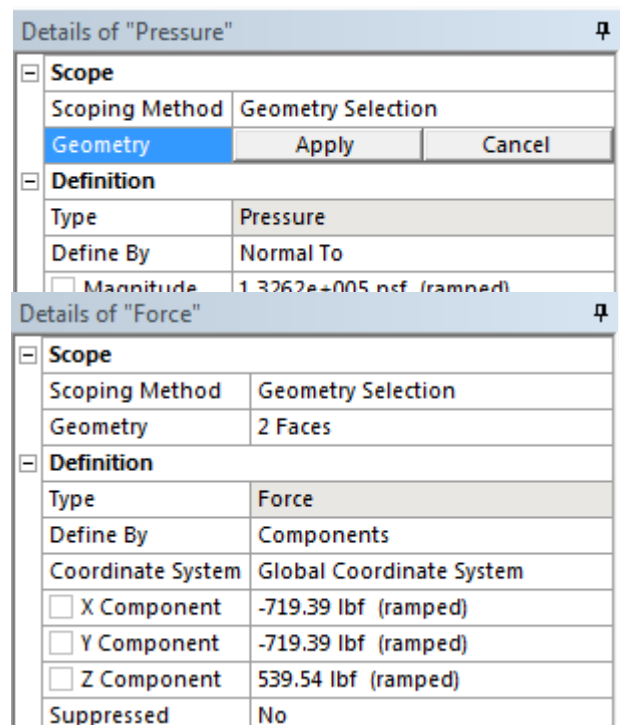
- Meshing should be done appropriately to get the accurate results.

4.4. SETUP

- Select the static structural and apply the boundary condition (like force, moment, pressure, supports etc).
- To assign a support, select the support in the toolbar and then select the appropriate support depending upon the requirements.

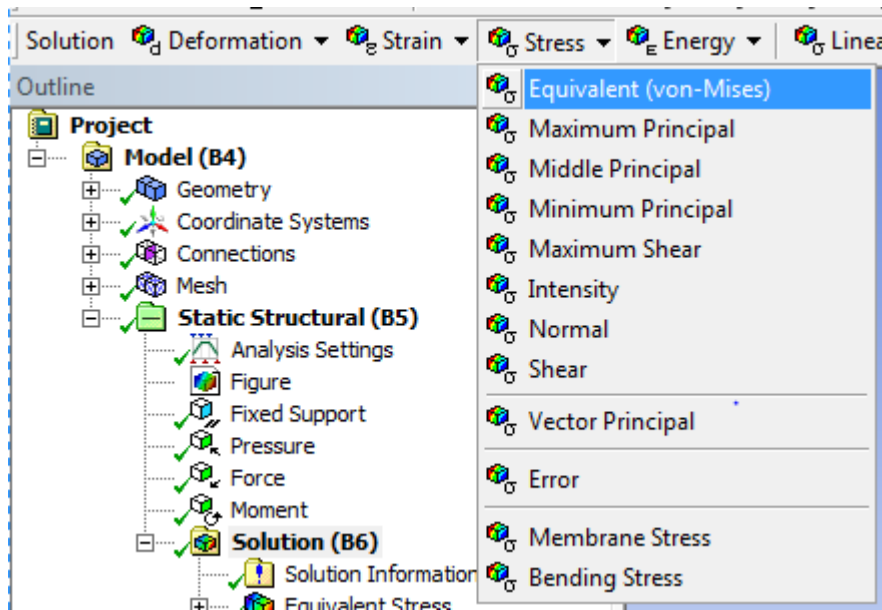


- After selecting the support, select the required face to be applied.
- To apply pressure, select loads → pressure.
- In the pressure dialog box, select the required face to be apply and give the value of pressure.
- To apply loads and moment, select Loads → Force (or) moments.
- In the force dialog box, select the required face to apply and provide the values in their respective directions.
- Likewise in the moment dialog box, select the required face to apply and provide the values in their respective directions.

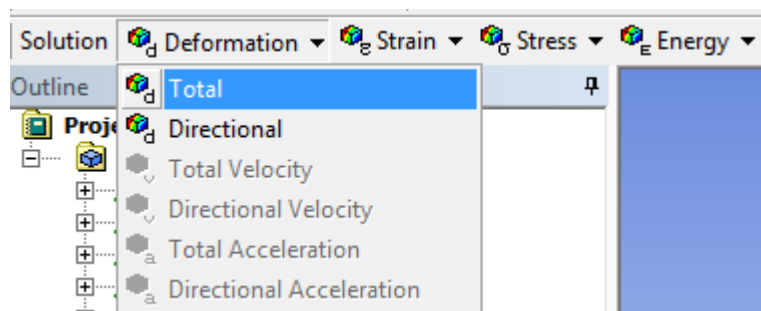


4.5. SOLUTION

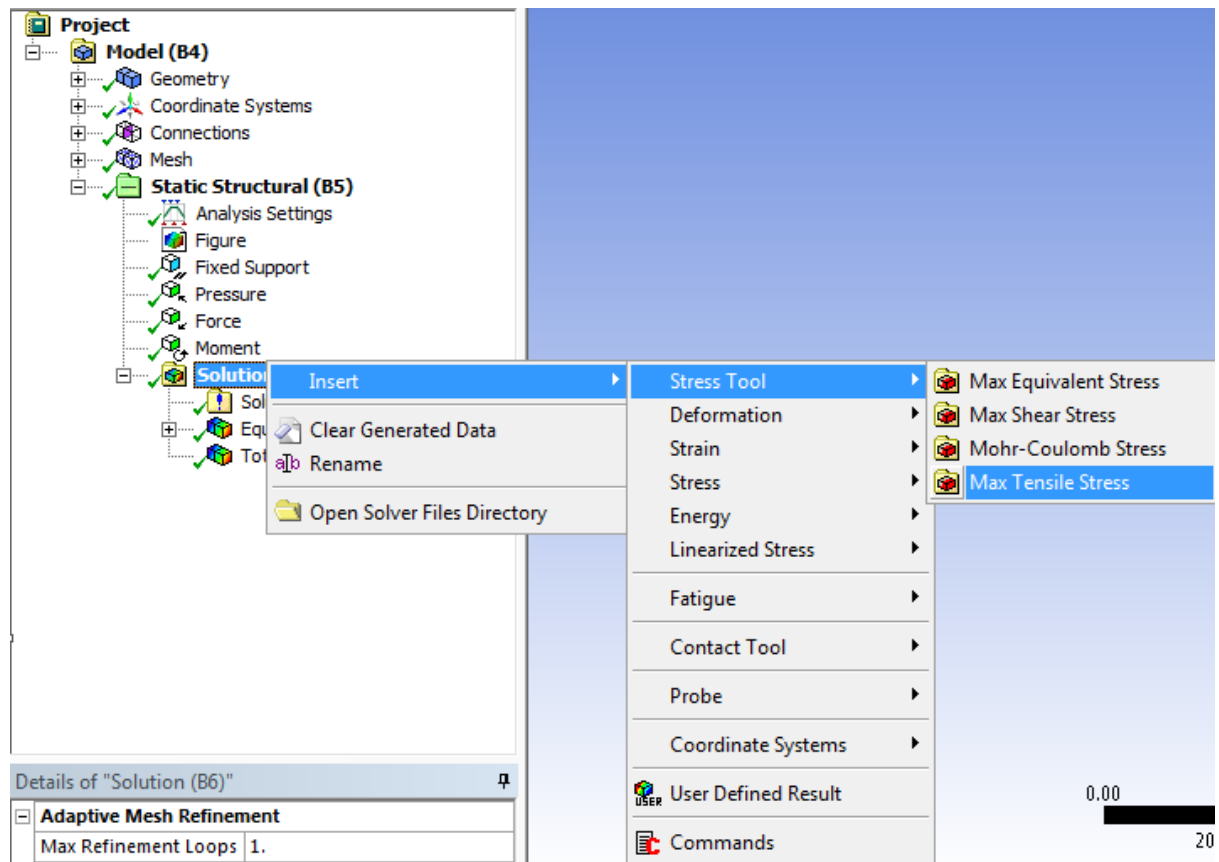
- Select the solution option and then select the required stress to be viewed.
- For equivalent stresses, select stress in the toolbar and then select equivalent (von-Mises) stress.
- For other stresses, select stress → maximum shear (or) normal (or) maximum principal



- If you need stress for a specified face (or) body, Select the appropriate face (or) body to view the required stress.
- For total displacement, select deformation → total (or) directional. If you want deformation for a specified face (or) body, Select the appropriate face (or) body to view the required deformation.



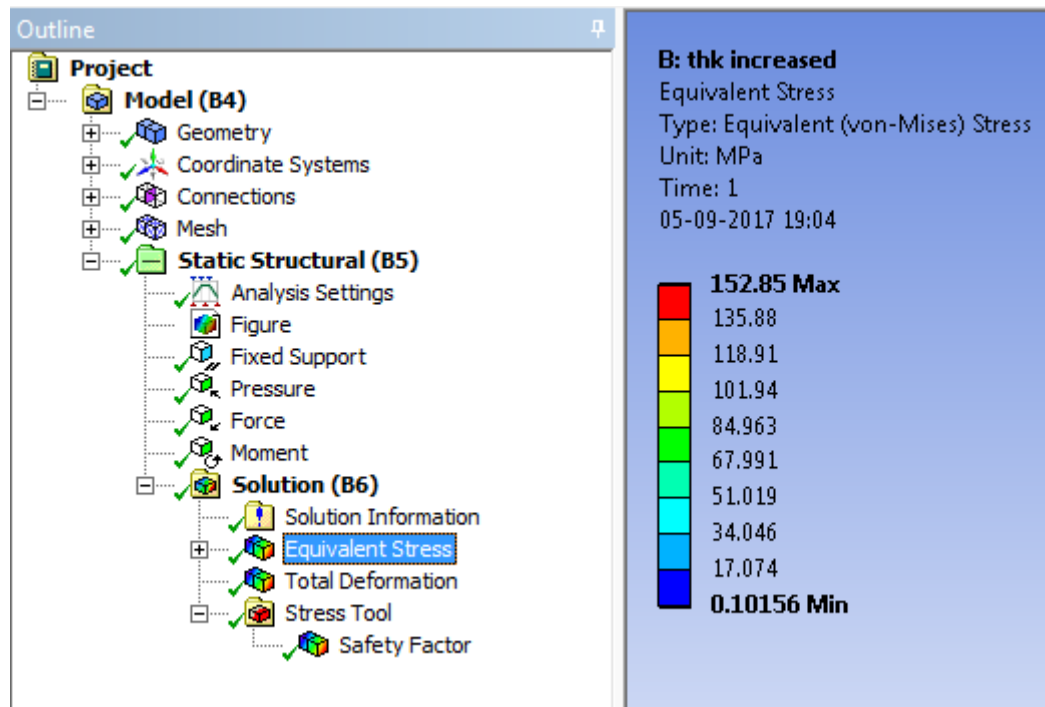
- For safety factor, Solution → Stress tool → Max tensile stress.
- You will get safety factor for your model in its intended environment.



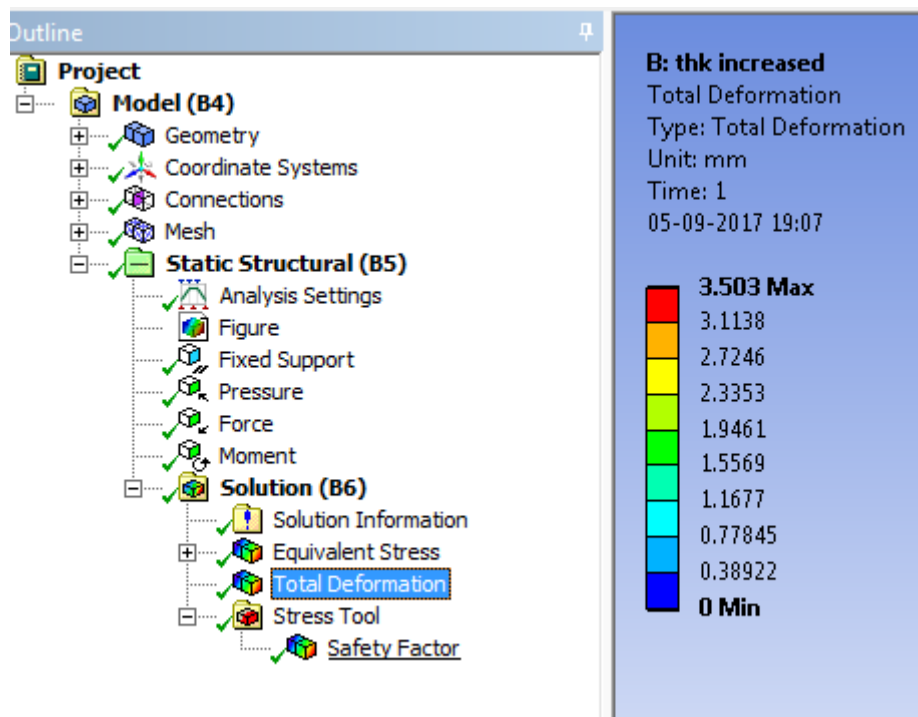
- Using the **Solve** command, we can run the analysis and the output results can be obtained.

4.6. RESULTS

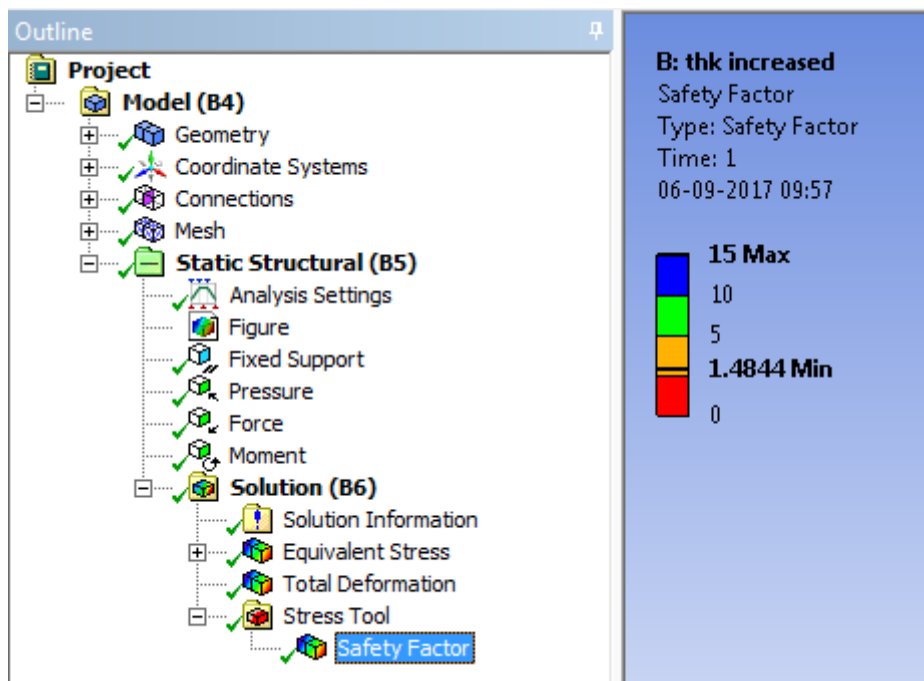
- Here, the equivalent stress results shows the values of variation of equivalent stress across the model. So that maximum and minimum equivalent stress can be determined.



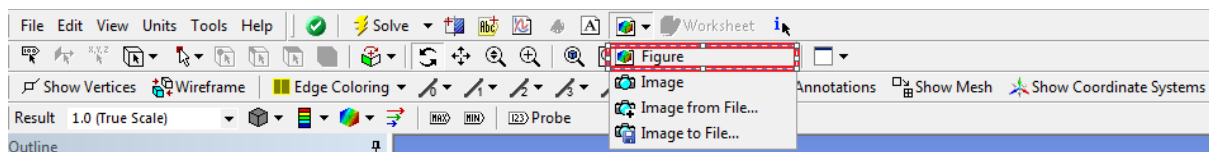
- Here, the total deformation results shows the value of variation of total deformation across the model. So that max and min total deformation can be determined.



- Here, the safety factor results shows the value of safety factor.



- We can also take the figure of the workspace by the **figure** command.



- Compare the results of equivalent stress with the allowable stress. If the equivalent stress is within the allowable stress then the design is safe.

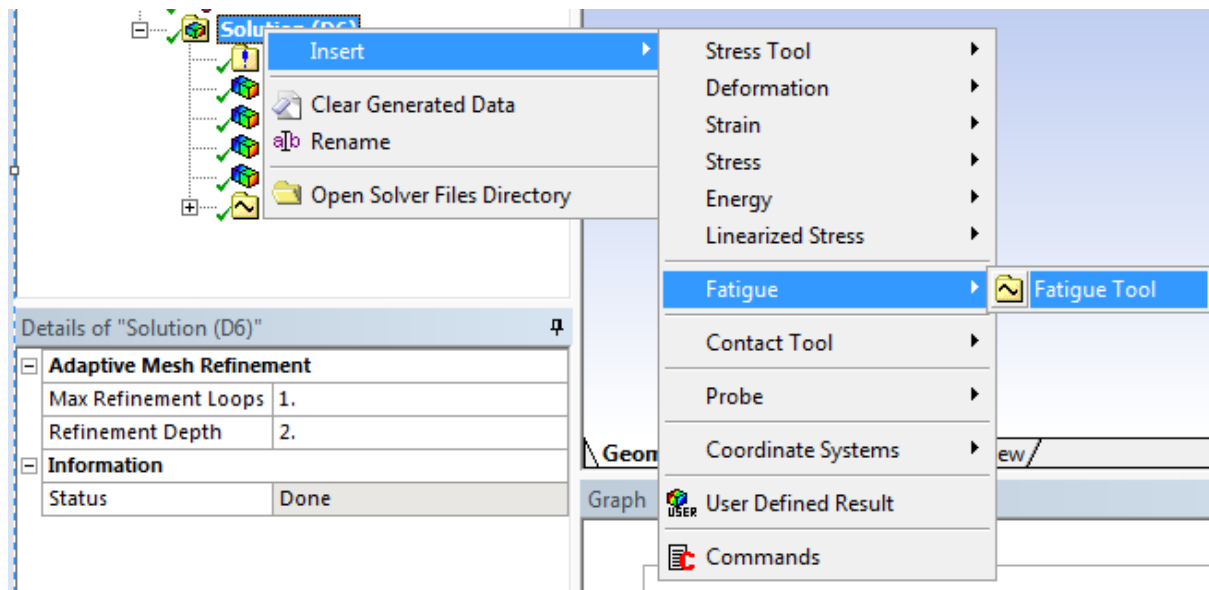
5. FATIGUE ANALYSIS

This analysis is used to determine the capability of the material to survive under the repeated loading. While many parts may work well initially, they often fail in service due to fatigue failure caused by repeated cyclic loading.

5.1. FATIGUE TOOL

Determines life, damage, and factor of safety information using a stress-life or strain-life approach. The **Fatigue Tool** is available only for static structural and transient structural analyses. Fatigue results can be added before or after a stress solution has been performed. To create fatigue results, a fatigue tool must first be inserted into the tree. Steps involved in fatigue analysis.

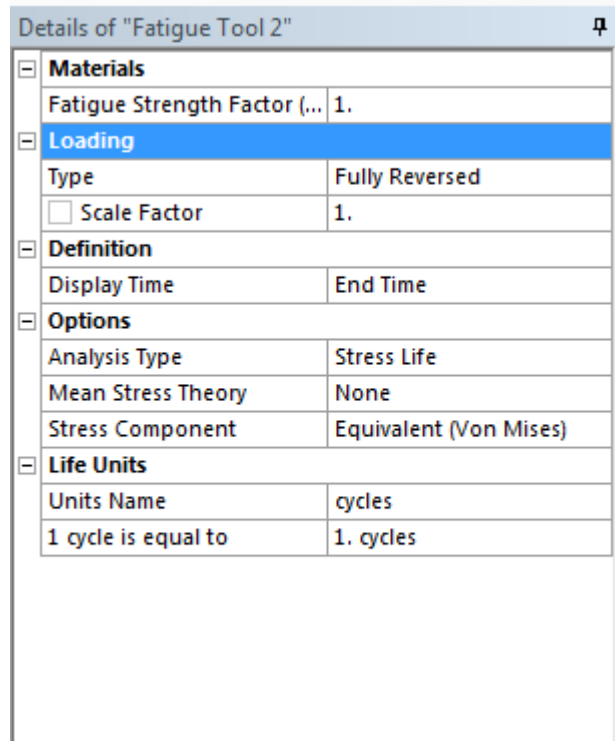
- To insert the fatigue tool, select Solution → Insert → Fatigue → Fatigue tool.



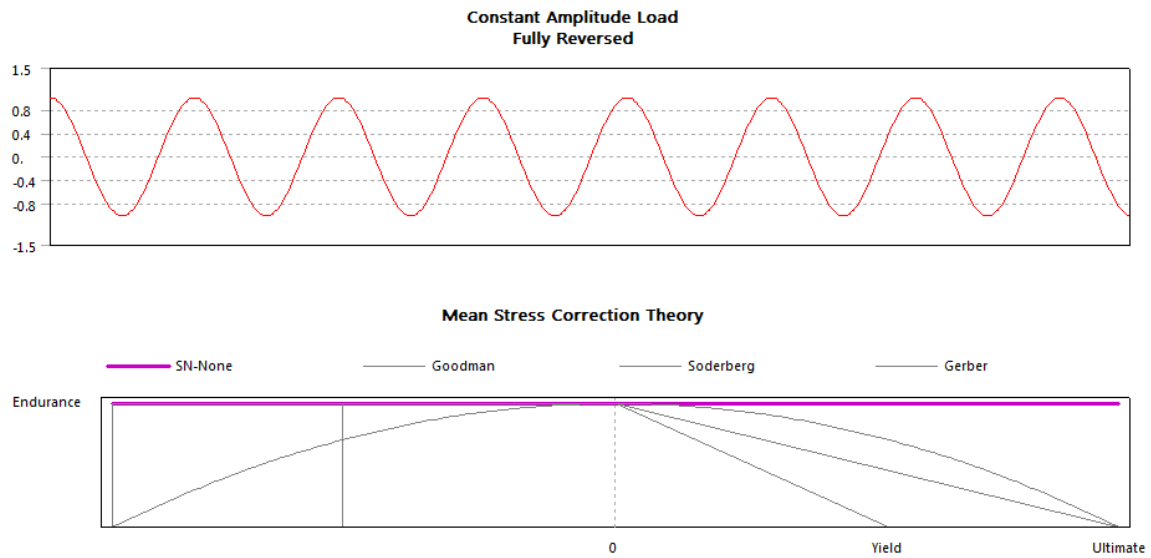
- A worksheet will open like the fig shown below.
- In the fatigue dialog box, give the value as 1 for fatigue strength factor.
- You can give loading type depending on the requirements. Normally, we use fully reversed.

Types of loading

- ✓ Zero based
- ✓ Fully reversed
- ✓ Ratio
- ✓ History data
- ✓ Non-proportional loading



- We can choose the analysis type as stress life (or) strain life. Stress life is concerned with total life and strain life is concerned with crack initiation.



- We can choose the mean stress theory depending upon the requirements. Types of mean stress theory for stress life analysis type
 - ✓ S-N Curve
 - ✓ Goodman
 - ✓ Gerberg
 - ✓ Soderberg
- Types of mean stress theory for strain life analysis type
 - ✓ None
 - ✓ Marrow
 - ✓ SWT(Smith-watson-topper)
- Generally, we use the equivalent(von-Mises) as the stress component.

UNITS NAME

This field allows you to specify the name for the **Life Units**. The unit options include:

- **cycles**
- **blocks**
- **seconds**
- **minutes**
- **hours**
- **days**
- **months**
- **User Defined**



User Defined

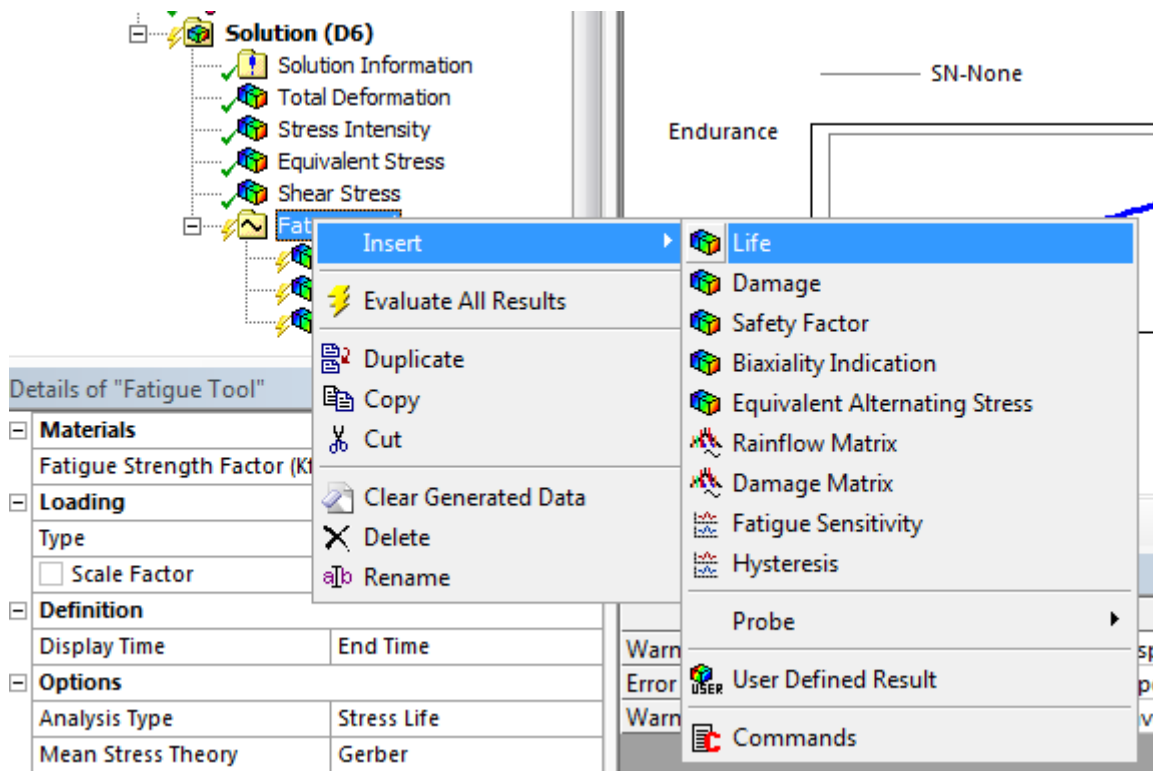
Selecting the **User Defined** option displays the **Custom Units Name** field. Enter the name for your customized unit name in this field. The specified unit is reflected in the Details view for all applicable fatigue settings.

1 “UNIT” IS EQUAL TO

Where "unit" is either cycle or block based on the **Units Name** selection. Modify the field's value based on the desired number of cycles or blocks for the units.

5.2. SOLUTION

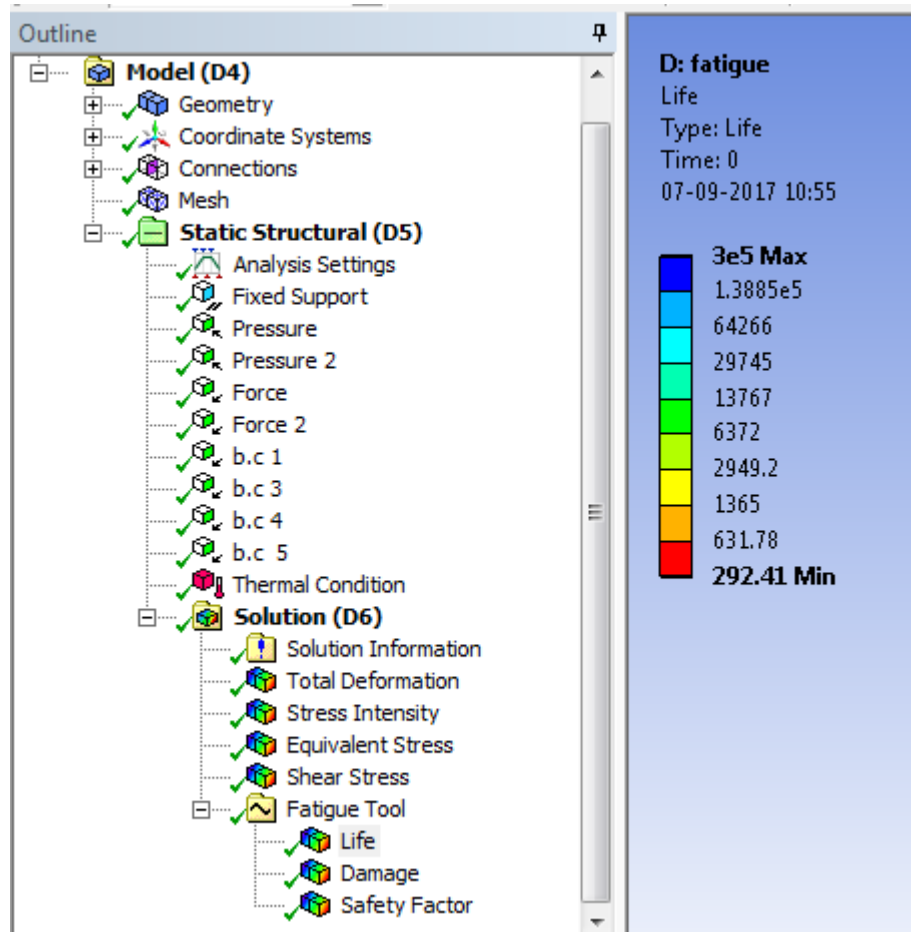
- Select the solution option and then select Fatigue  Insert  Life to determine the available life for the given fatigue analysis.

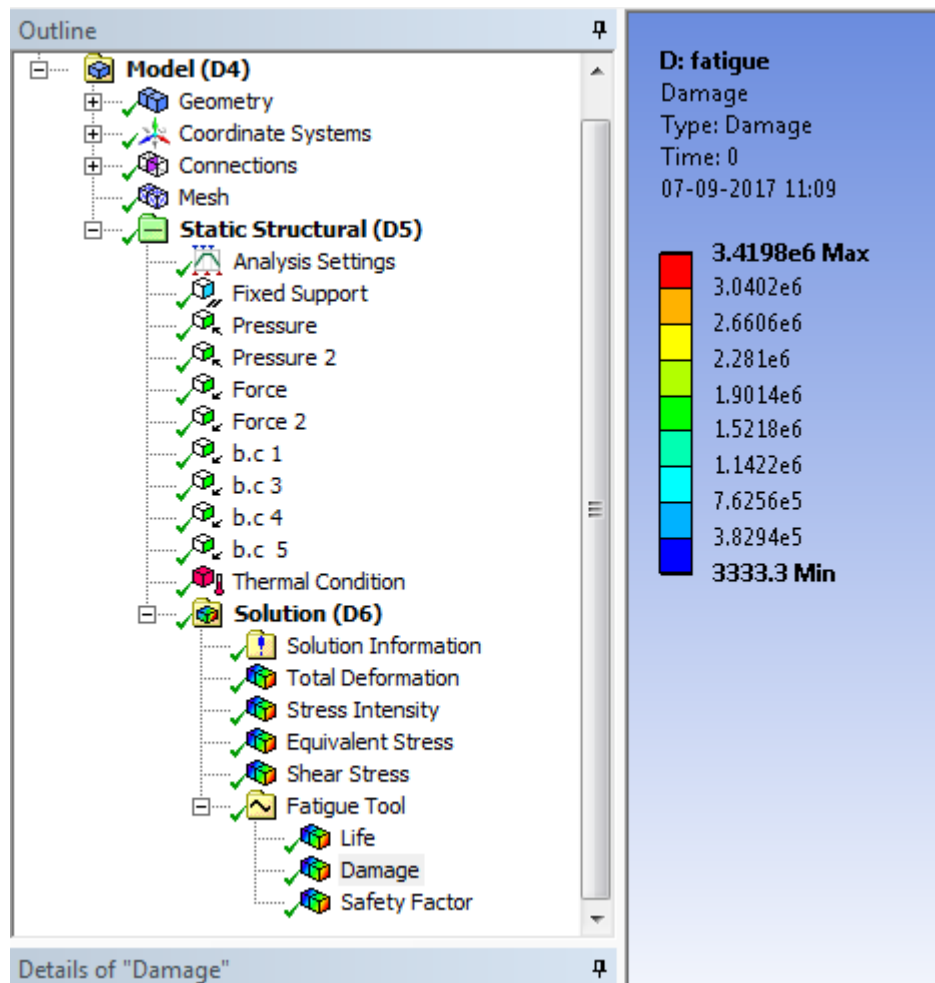


- Similarly, for damage and safety factor. Select the solution option and select Fatigue → Insert → damage (or) safety factor to determine the available life for the given fatigue analysis.
- Fatigue damage is defined as the ratio of design life to available life.
- The result is a contour plot of the safety of factor (FS) with respect to a fatigue failure at a given design life. Maximum FS reported is 15.

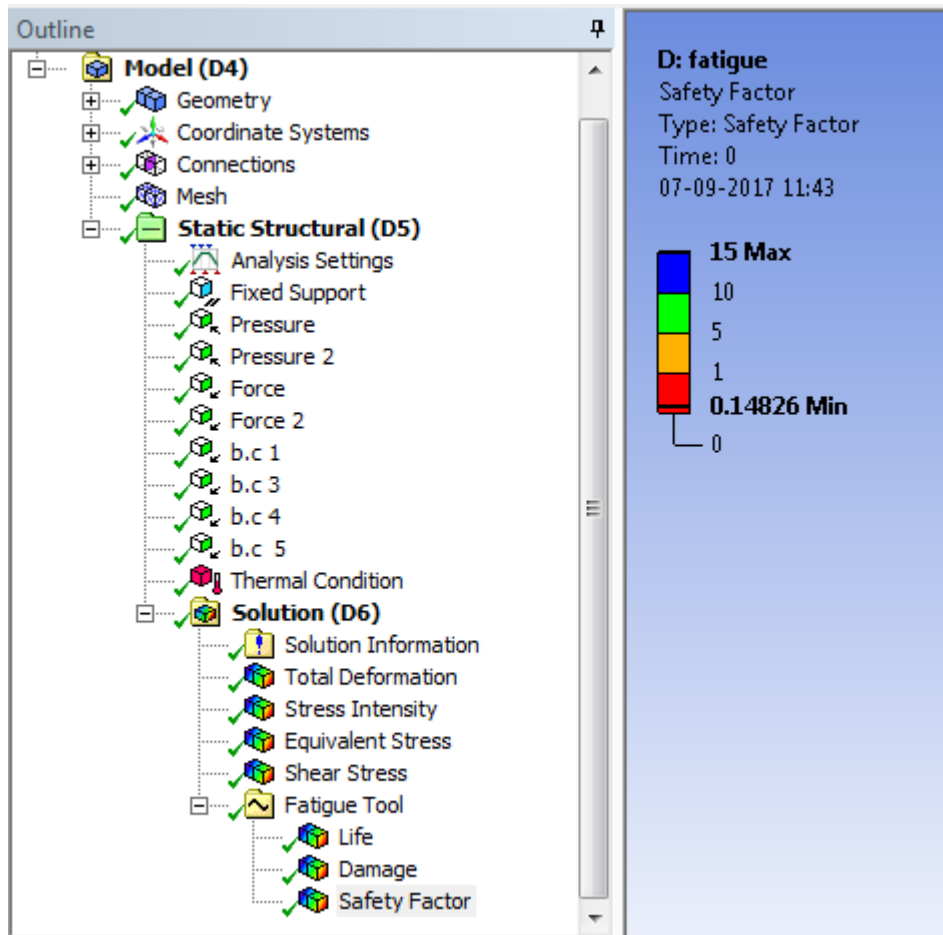
5.3. RESULTS

- Here, the results of life represent the number of cycles until the part will fail due to fatigue. Thus, if the given load history represents one month of loading and the life was found to be 120, the expected model life would be 120 months.





- Fatigue damage is defined as the ratio of design life to available life. The default design life may be set through the option dialog box. A damage of greater than 1 indicates that the part will fail before the design life due to fatigue.



- Here, the safety factor results shows the value of safety factor for the given fatigue analysis.