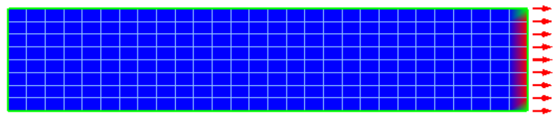
4-2 Quasi-Static Fatigue

Case Description: Composite coupon subject to tensile cyclic loading

Example Location: **Tutorials > Fatigue > Quasi Static Fatigue**

Model Description:



Nodes: 261; Elements: 224

Length: 1.0” (1.013091”); Width: 0.1964” (0.196248”); and Thickness: 0.10198”

Material Description: Fiber/Matrix (FVR = 55%), with nonlinear matrix stress-strain response

Layup: [0/90/0/90/0/90/0] woven

Objective of Analysis: Predict the fatigue life of the coupon

ASTM Number: -

Control Type: Load Control

Analysis Type: Quasi-Static Fatigue

Solution: ***Fatigue 10**

(See Section 5 of User Manual)

Input Requirements: GENOA data bank (including experimental stress-strain and S-N curves)

GENOA model files

FEA Solver: **MHOST**

(Use the keyword **SOLVER* as described in Section 5 of User Manual to invoke other FEA solver options such as NASTRAN, ANSYS and ABAQUS)

Output from Analysis: Fatigue life (number of cycles to failure) for three constant load (stress) levels, ply stresses and strains, failure modes at various amounts of cycles

Summary of Results:

(a) At the load level of 30% of the ultimate load the fatigue life is 12,500 cycles;
(b) At the load level of 50% of the ultimate load the fatigue life is 3,900 cycles;
(c) At the load level of 70% of the ultimate load the fatigue life is 373.5 cycles

Introduction

This tutorial demonstrates how to use GENOA-PFA to estimate the fatigue life of a composite coupon subject to cyclic tension. For details on the technical approach and general features of the code please refer to the GENOA PFA Quick Reference and Theoretical Manuals.

In the example herein, a [0/90/0/90/0/90/0] cross-ply coupon, which is made of the 96-oz 3TEX 3Weave E-glass/Dion 9800™ composite system, is used. The coupon is 1.0 inch long by 0.1964 inch wide by 0.102 inch thick. The analysis is based on the fiber/matrix properties of the composite system. These properties, together with the composite matrix nonlinear stress-strain curve and the S-N degradation curves for the fibers and matrix, were obtained from experiments.

The fatigue life is predicted for the three load (stress) levels with the amplitudes corresponding to approximately 30, 50 and 75 percents of the ultimate static load, which can be determined by running static GENOA-PFA analysis. The simulated results are compared with experimental data.

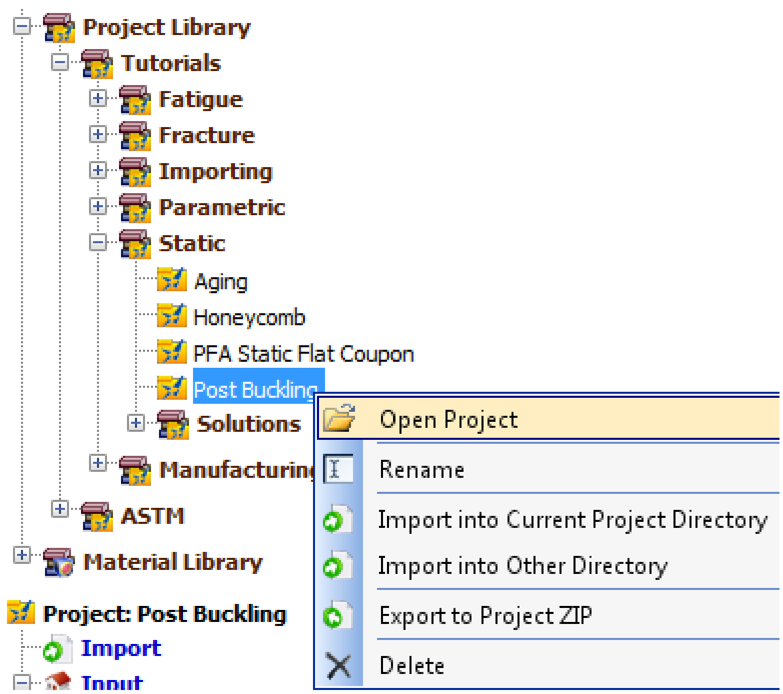
Launching GENOA

1. Start GENOA by executing it from the desktop or typing **genoa** in the command prompt.

Importing GENOA Model File

The geometry has already been created in GENOA format as a '.dat' input file.

2. Make sure the **Unit System** in the upper right corner is set to **Inch-Second-Pound**.
3. Right click on **Quasi Static** node under the **Fatigue** node and select **Open Project** (see Figure below).



Replace this figure

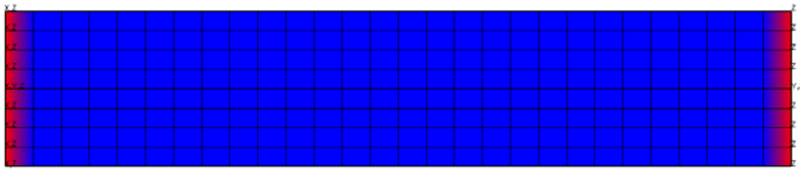
4. Navigate to the **42ksi** directory under **Fatigue** node under **Quasi Static** node in the tree.

Note: The FE Model will load in the Mesh view window.

Boundary Conditions

5. Click on the **Boundary Conditions** ( ) icon on the left of the Mesh Setup window to invoke the **Boundary Conditions** panel.

6. To view the applied boundary conditions, simultaneously highlight the items **Boundary X**, **Boundary Y**, and **Boundary Z** from the list by holding the Ctrl key as you click the left mouse button (see Figure below).



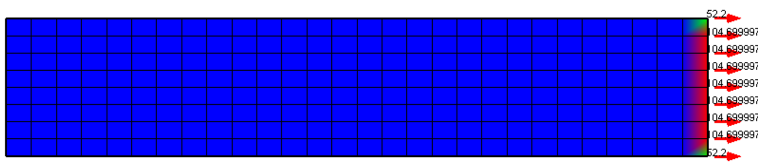
Boundary Conditions

Loading conditions

7. Click on the **Force** ( ) icon on the left of the **Mesh Setup** window to invoke the **Force** panel.

8. Select **Force X**. Figure below shows that the tensile forces are applied to the model at the right edge.

   Caution: The forces that are applied to the coupon are for fatigue loading is obtained from maximum stress loading.

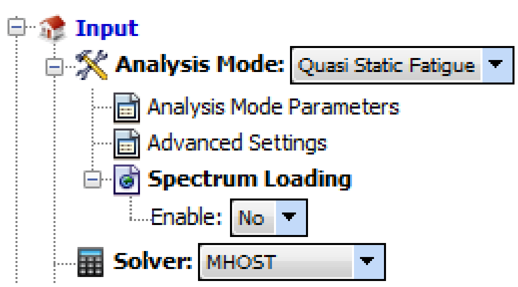


Tensile load acting on the coupon

Analysis Mode Parameters

9. Change the **Analysis Mode** to **Quasi Static Fatigue**.

10. Be sure and verify that **No** is selected for **Enable** field under **Spectrum Loading** node.



11. Double click on **Analysis Mode Parameters** node under **Analysis Mode** node in the tree.

12. Enter **10** for **Incremental Step for Material Non-Linearity**.

    Note: This parameter increases the accuracy. The higher the value, the lower the load increment becomes; however, it is time consuming and should be used when nonlinearity of the material is to be considered. In case of linear elastic assumption, set this parameter to 1.

13. Set **Starting Cycle** value to **10000**.

Note: If you expect your model to initiate damage at much higher cycles, then you can enter higher starting cycle value. The analysis will attempt to skip the analysis for specified cycles unless there is damage in the FE model. If you are not sure of the starting cycle number, then you are advised to start with Starting Cycle value of 1. The analysis will take longer to run in this condition.

14. Enter **0.1** for **Stress Ratio**.

    Note: If the material databank contains multiple S-N curves for different Stress Ratios, the user can specify which S-N to be used for this analysis. If the Stress Ratio value is different from that available in the databank and is in between the range in the databank, then GENOA will interpolate between the two S-N curves. We have only 0.1 available in the databank.

15. Switch **Use S-N Curves in Materials** to **true** (see Figure below).

Description	Value
Number of Nodes/Elements Allowed to Damage	500
Number of Nodes/Elements Allowed to Fracture	12
Number of Iterations to Run	1000
Incremental Step for Material Non-Linearity	10
Quasi-Static Fatigue Parameters	
Starting Cycle	1
Stress Ratio	0.1
Use S-N Curves In Materials	true

Analysis Settings

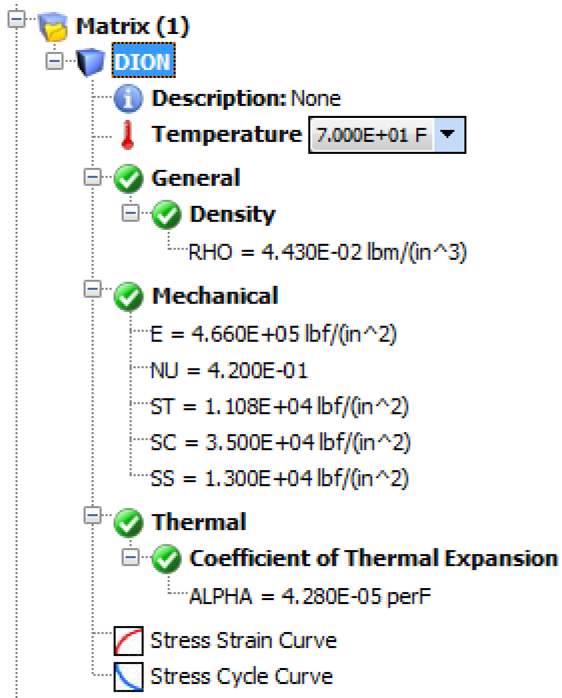
16. Double click on **Advanced Settings** node.

17. Make sure that under the **Post Damage Degradation** section, the **Damage Force Locations** and **Damage Boundary Locations** parameters are set to **true**.

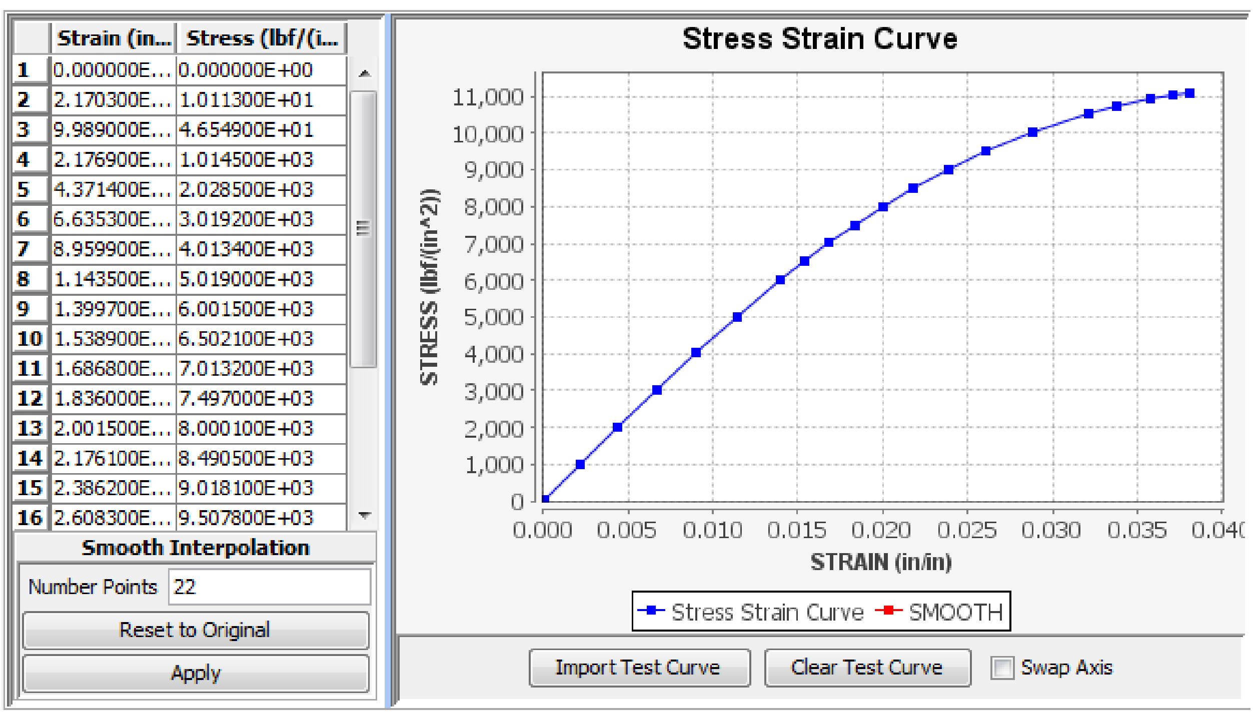
    Note: Enabling the Damage Force Locations and Damage Boundary Locations to true will enable GENOA to remove those elements where either load or boundary conditions may be defined and where the stresses exceed the failure loads.

Viewing & Editing the Materials

18. Expand **DION** node under **Matrix** node under **Material** node in the tree.

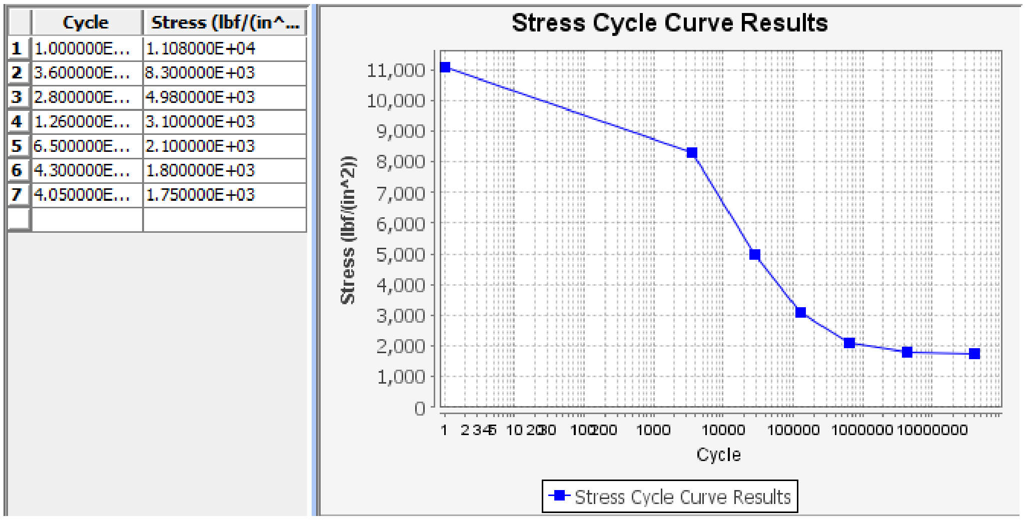


19. Double click on **Stress Strain Curve** node to view the matrix nonlinear stress strain curve.



20. Double click on **Stress Cycle Curve** node to view matrix degradation curve (SN curve).

Note: Stress ratio value of 0.1 is specified under Stress Cycle Curve node in the tree.



	Cycle	Stress (lbf/(in^...
1	1.000000E...	1.108000E+04
2	3.600000E...	8.300000E+03
3	2.800000E...	4.980000E+03
4	1.260000E...	3.100000E+03
5	6.500000E...	2.100000E+03
6	4.300000E...	1.800000E+03
7	4.050000E...	1.750000E+03

Note:

- The S-N data in the databank is entered from lower number of cycles to higher number of cycles.
- In fatigue simulations, materials performance is commonly characterized by a S-N curve (Wöhler curve). The S-N curve is a graph of the magnitude of a cyclical stress (S) against the number of cycles to failure (N). It shows how many cycles are required to cause a fatigue failure for a given nominal stress.
- The S-N relationship is determined for a specified value of stress ratio ($R=\sigma_{min}/\sigma_{max}$).
- The S-N curve is relevant for fatigue failure at high number of cycles (N > 105 cycles).
- If the experimental S-N curve is not available, the bilinear degradation curve can be used instead. In this case, the value of the Use S-N Curves In Databank parameter will be set to false, and the user will be required to input two pairs of coefficients a and b for the expression:

$$\frac{S}{S_0} = a - b\log N$$

21. Repeat the above steps for **EGKG** glass fiber material.

    Note: usually carbon/graphite/ceramic/aramid fibers show linear behavior; therefore, the SS and S-N curve tabs are left empty. You can create a SN curve if you see degradation in S-N curve for 0 degree ply data subjected to tension-tension axial loading. The S-N curve in this case will represent the reduction in fiber tensile strength due to breakage in the fiber tows.

Laminate Editor

22. Right click on **Laminate_1** node under **Laminate** node in the tree and select **Edit** in the popup menu.

    Note: For this exercise will not modify the laminate.

	Material Type	Fiber	Matrix	Braid	Temperature (F)	Thickness (in)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.204600E-02	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
2	Braid/Matrix	NONE	DION	Braid	7.000000E+01	6.240000E-04	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
3	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.953200E-02	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
4	Braid/Matrix	NONE	DION	Braid_1	7.000000E+01	1.028000E-03	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
5	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.204600E-02	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
6	Braid/Matrix	NONE	DION	Braid	7.000000E+01	6.240000E-04	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
7	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	9.633000E-03	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
8	Braid/Matrix	NONE	DION	Braid_1	7.000000E+01	5.070000E-04	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
9	Braid/Matrix	NONE	DION	Braid	7.000000E+01	6.240000E-04	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
10	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.204600E-02	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
11	Braid/Matrix	NONE	DION	Braid_1	7.000000E+01	1.028000E-03	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
12	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.953200E-02	9.000000E+01	5.500000E-01	1.000000E-01	FailCrit_1	NONE
13	Braid/Matrix	NONE	DION	Braid	7.000000E+01	6.240000E-04	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE
14	Fiber/Matrix	EGKG	DION	NONE	7.000000E+01	1.204600E-02	0.000000E+00	5.500000E-01	1.000000E-01	FailCrit_1	NONE

Laminate Editor

Note: The laminate Editor shows that the coupon lay-up consists of 14 composite (EGKGDION) different thickness plies with E-glass fibers arranged in the cross-ply pattern. Also, braid cards are defined which is explained in detail in another tutorial example.

23. Double click on **Braid** node under **Braid (2)** node under **Materials (4)** node in the tree. You will see the vectors defined for up and down going fibers (see Figure below).

    Note: the vectors are used to define the out-of-plane orientation of fibers along the x-axis of the ply. Please consult Braid and Woven step-by-step exercise under Static directory for more explanation.

Braid Entry Setup				
Fiber	Fiber Volume Ratio	X Angle Vector	Y Angle Vector	Z Angle Vector
EGKG	5.000000E-01	5.000000E-01	0.000000E+00	8.660250E-01
EGKG	5.000000E-01	5.000000E-01	0.000000E+00	-8.660250E-01

24. Similarly, double click on **Braid_1** node under **Braid (2)** node in the tree.

    Note: the vectors are used to define the out-of-plane orientation of fibers along the y-axis of the ply (see Figure below).

Braid Entry Setup				
Fiber	Fiber Volume Ratio	X Angle Vector	Y Angle Vector	Z Angle Vector
EGKG	5.000000E-01	0.000000E+00	5.000000E-01	8.660250E-01
EGKG	5.000000E-01	0.000000E+00	5.000000E-01	-8.660250E-01

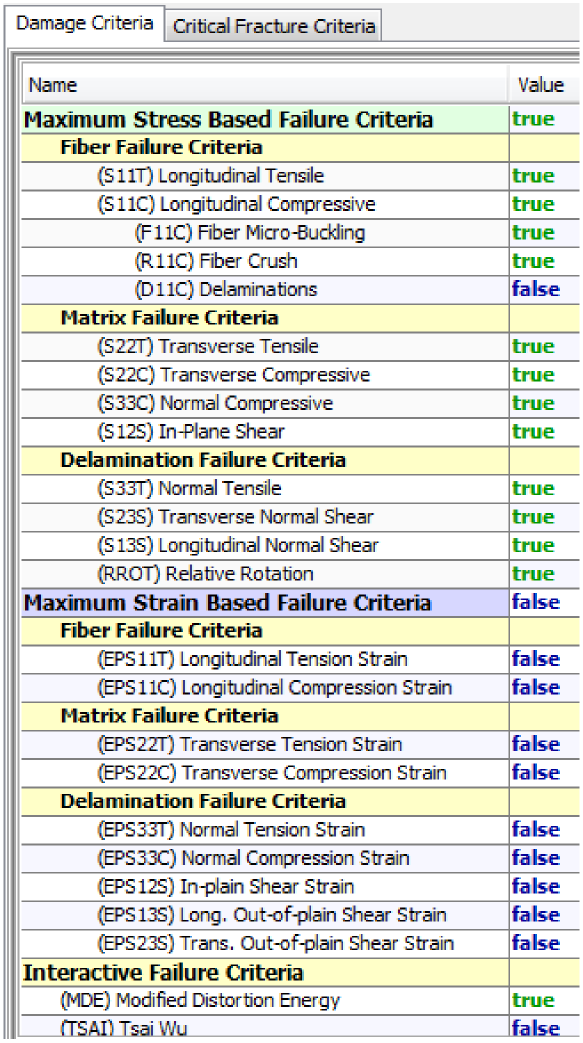
25. Click on the **Boundary Conditions** icon to invoke the **Boundary Conditions** panel.

Failure Criteria

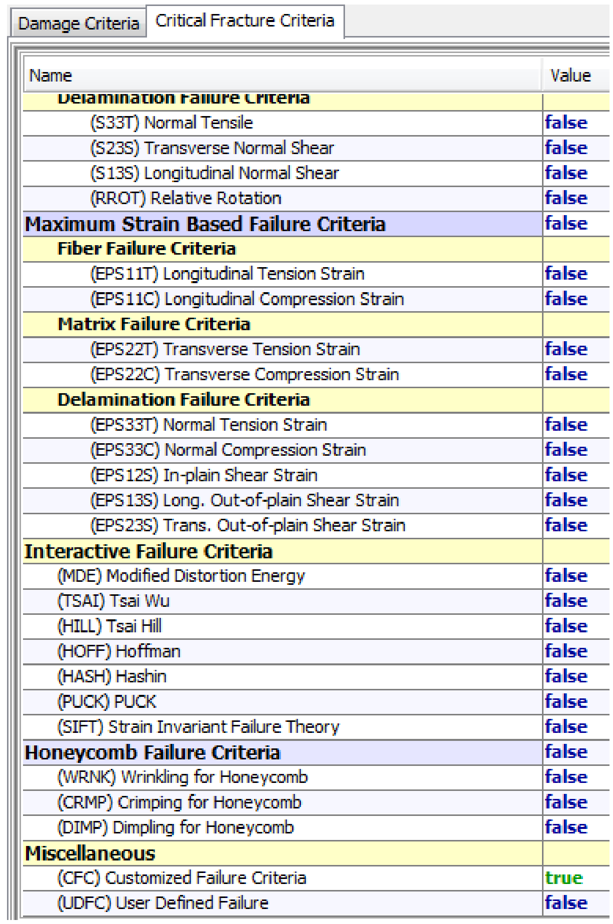
26. Under the **Failure** node, double click on **FailCrit_1** node to review the damage and failure criteria assigned to the laminates.

27. Click on **Composite Default** button underneath the **Damage Criteria** and **Critical Failure Criteria** tabs.

    Note: For this exercise will not modify the Failure Criteria.



Name	Value
Maximum Stress Based Failure Criteria	true
Fiber Failure Criteria	
(S11T) Longitudinal Tensile	true
(S11C) Longitudinal Compressive	true
(F11C) Fiber Micro-Buckling	true
(R11C) Fiber Crush	true
(D11C) Delaminations	false
Matrix Failure Criteria	
(S22T) Transverse Tensile	true
(S22C) Transverse Compressive	true
(S33C) Normal Compressive	true
(S12S) In-Plane Shear	true
Delamination Failure Criteria	
(S33T) Normal Tensile	true
(S23S) Transverse Normal Shear	true
(S13S) Longitudinal Normal Shear	true
(RROT) Relative Rotation	true
Maximum Strain Based Failure Criteria	false
Fiber Failure Criteria	
(EPS11T) Longitudinal Tension Strain	false
(EPS11C) Longitudinal Compression Strain	false
Matrix Failure Criteria	
(EPS22T) Transverse Tension Strain	false
(EPS22C) Transverse Compression Strain	false
Delamination Failure Criteria	
(EPS33T) Normal Tension Strain	false
(EPS33C) Normal Compression Strain	false
(EPS12S) In-plain Shear Strain	false
(EPS13S) Long. Out-of-plain Shear Strain	false
(EPS23S) Trans. Out-of-plain Shear Strain	false
Interactive Failure Criteria	
(MDE) Modified Distortion Energy	true
(TSAI) Tsai Wu	false



Name	Value
Delamination Failure Criteria	
(S33T) Normal Tensile	false
(S23S) Transverse Normal Shear	false
(S13S) Longitudinal Normal Shear	false
(RROT) Relative Rotation	false
Maximum Strain Based Failure Criteria	false
Fiber Failure Criteria	
(EPS11T) Longitudinal Tension Strain	false
(EPS11C) Longitudinal Compression Strain	false
Matrix Failure Criteria	
(EPS22T) Transverse Tension Strain	false
(EPS22C) Transverse Compression Strain	false
Delamination Failure Criteria	
(EPS33T) Normal Tension Strain	false
(EPS33C) Normal Compression Strain	false
(EPS12S) In-plain Shear Strain	false
(EPS13S) Long. Out-of-plain Shear Strain	false
(EPS23S) Trans. Out-of-plain Shear Strain	false
Interactive Failure Criteria	
(MDE) Modified Distortion Energy	false
(TSAI) Tsai Wu	false
(HILL) Tsai Hill	false
(HOFF) Hoffman	false
(HASH) Hashin	false
(PUCK) PUCK	false
(SIFT) Strain Invariant Failure Theory	false
Honeycomb Failure Criteria	false
(WRNK) Wrinkling for Honeycomb	false
(CRMP) Crimping for Honeycomb	false
(DIMP) Dimpling for Honeycomb	false
Miscellaneous	
(CFC) Customized Failure Criteria	true
(UDFC) User Defined Failure	false

28. Select **Save** under **Project** menu (or press Ctrl and S on the keyboard).

Progressive Failure Analysis

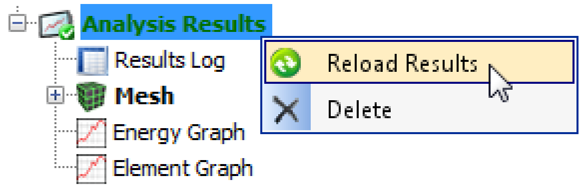
29. Right click on **Analysis** node and select **Progressive Failure Analysis** option in the **Add** popup menu.

30. Right click on **Progressive Failure Analysis** node and select **Run Analysis**.

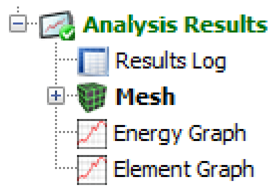
Progressive Failure Analysis Results

Note: After the analysis is completed, the program will automatically switch to the Results Log screen. But if you wish to load the current results during the analysis, then you may choose the

Reload Results menu item under the popup menu for the **Analysis Results** node. You may reload the results at any time if you believe that the results are not current or updated correctly.



When there are results to be loaded, there will be additional nodes under the Analysis Results node as shown below.



Results Log

31. Double click on **Results Log** node to view the iteration log if not already there after the simulation is complete.

    Note: The **Results Log** table provides information about the amplitude of the applied cyclic load and the number of damaged and fractured nodes corresponding to the number of fatigue cycles. The fatigue cycles corresponding to stable equilibrium are highlighted in green.

32. **Click** on **View All Iterations** button. The log will show all the iteration rows (default).

Iteration	Elements	Nodes	Force X	Force Y	Force Z	Mome...	Mome...	Mome...	Pressure	Cycle	Damag...	Fractu...	Status
1	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	1.95312...	261	0	Damage...
2	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	1.95312...	261	0	Damage...
3	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	1.95312...	261	0	Equilibri...
4	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	3.90625...	261	0	Equilibri...
5	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	7.81250...	261	0	Equilibri...
6	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	1.56250...	261	0	Equilibri...
7	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	2.34375...	261	0	Equilibri...
8	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	2.73437...	261	0	Equilibri...
9	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	2.92968...	261	0	Equilibri...
10	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	3.02734...	261	0	Equilibri...
11	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	3.05175...	261	0	Equilibri...
12	224	261	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	3.07617...	261	4	Damage...
13	224	263	8.37500...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	0.00000...	3.07617...	263	442	Damage...

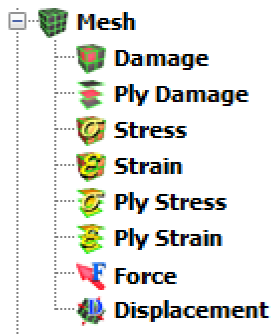
Complete Results Log table

The log indicates that the last equilibrium iteration before fracture corresponds to **~305** cycles, which is the fatigue life of the composite coupon for the stress ratio $R = 0.1$ and the load amplitude ~838 lbs (nominal stress amplitude ~42 ksi).

Results Mesh

33. Double click on **Mesh** node in the tree under **Analysis Results** node.

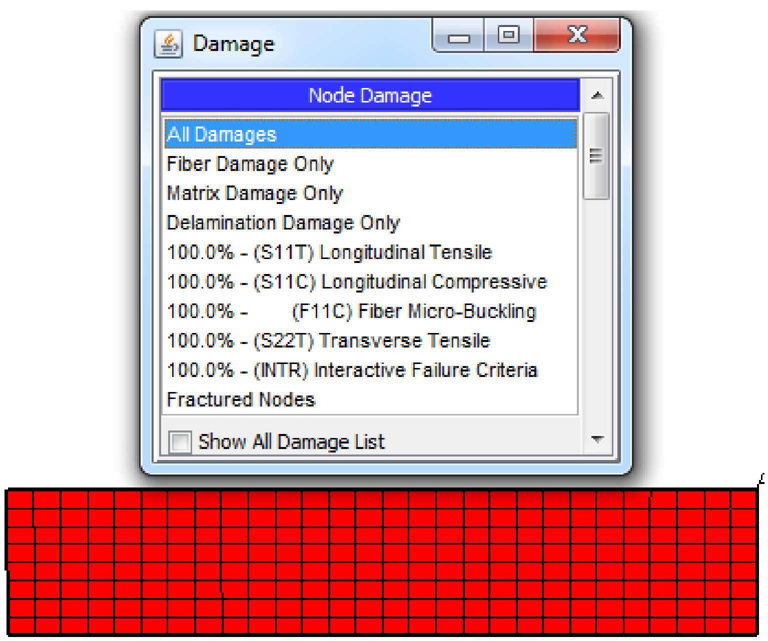
    Note: You will see more nodes under Mesh node in the tree (as shown below).



Node Damage

34. Double click on **Damage** node under **Mesh** node in the tree.
35. Drag the slider to **iteration 5** or enter **5** in the iteration text box.
36. Select **Cycles** in the drop down list in the player next to iteration text box, as shown in the following Figure.

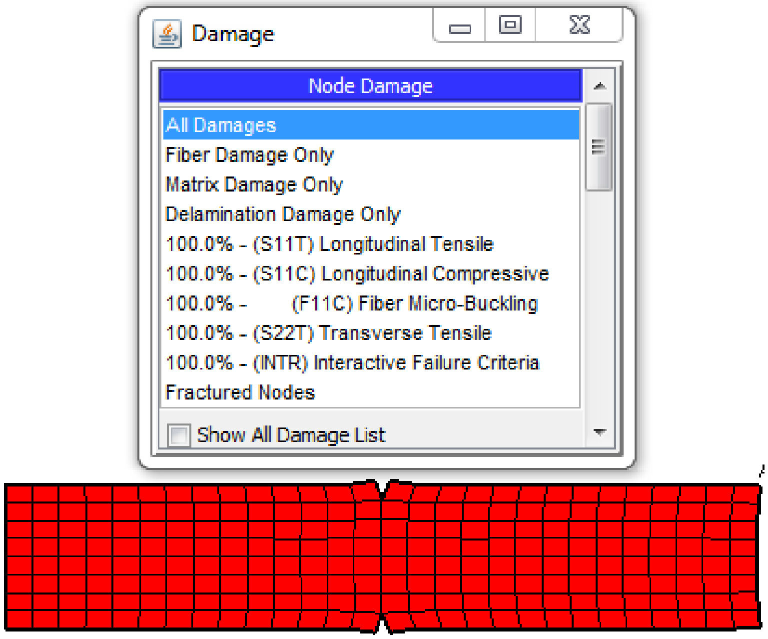
Note: You will see the following damage modes.



Damage results mesh at Iteration 5 (~78 cycles)

Note: The element switches to red color even if one ply out of 14 plies is damaged.

37. Advance the iterations further until you reach the last iteration.

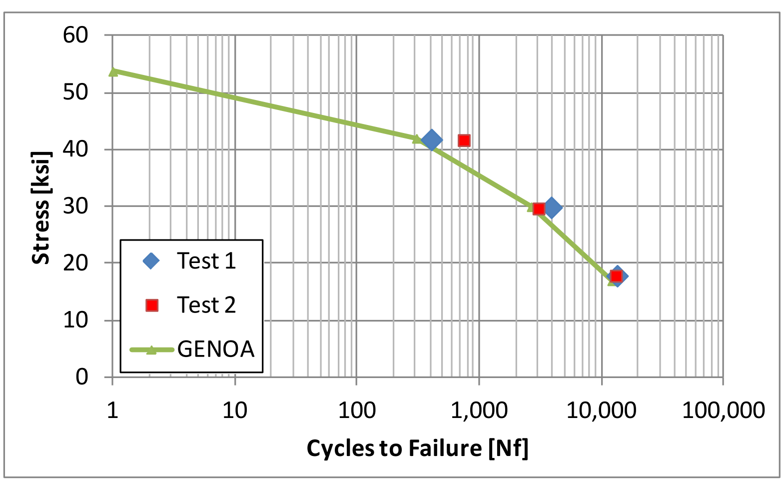


Damage results mesh at Iteration 13 (308 cycles)

Note: The damage panel shows Longitudinal Tension failure that corresponds to fiber failure for 0 degree plies in the coupon. The Longitudinal Compression appears because the fibers in the 90 degree plies are assumed to have buckled during the analysis after matrix failure.

Note: Figures above show that after ~308 cycles of cyclic tensile loading, crack propagation initiation takes place in the composite coupon.

Repeat above steps to predict the fatigue life of the coupon for the load amplitudes ~600 lbs (nominal stress amplitude ~30 ksi) and ~345 lbs (nominal stress amplitude ~17 ksi).



Fatigue life of the woven composite coupon subject to cyclic tensile loading

You have finished the example demonstrating how to use GENOA to predict the fatigue life of a composite structure.