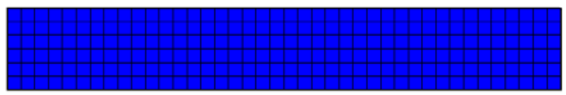
PFA Static Un-notched Coupon

Case Description: Composite coupon subjected to a tensile loading in dry conditions.

Example Location: **Tutorials > Static > PFA Static**

Model Description:



Nodes: 287; Elements: 240

Plate length: 10”, Plate width: 1.5” and Plate thickness: 0.033”

Material Description: Graphite Epoxy Ply properties are used, lay-up of [0/90/0]s and 0.0055” ply thickness.

Objective of Analysis: Determine the ultimate (failure) load and associated failure modes.

ASTM Number: ASTM D3039

Control Type: Load Control

Analysis Type: Static

Unit System: Inch Second Pound

Solution: ***STATIC** UPDATE GEOMETRY TRUE (Updated Lagrangian)

See PFA Case Control Keywords Section 3-2 of User Manual.

Input Requirements:

1. Import FEM Model (.dat) GENOA file
2. Assign/Edit boundary and loading conditions
3. Assign/Edit and verify material properties (fiber/matrix properties)
4. Create Ply Schedule

FEA Solver: **MHOST**

See PFA Case Control Keywords (Section 3-2 of User Manual).

Output from Analysis: Structural deformations, ply stresses and strains, load-displacement curve, active failure modes at various load increments including damage initiation and progression, fracture initiation, and failure loads.

Summary of Results:

Damage Initiation:

Load 4314.0 lbf, Active damage modes: Longitudinal tensile, transverse tensile and modified distortion energy.

Final Failure:

Load ~10161.4 lbf, Active failure modes: Longitudinal tensile and compressive, transverse tensile, and modified distortion energy.

Introduction

This tutorial demonstrates how to modify and edit a previously translated FE model in GENOA format and set if for performing the Static Progressive Failure Analysis using the default MHOST FE solver.

This example contains a standard un-notched coupon and was originally prepared in PATRAN. The model does not contain any boundary and loading conditions and any ply schedule information.

Note: To see the solution and results for this tutorial, please open the project from the 'Solutions' library.

Launch GENOA

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

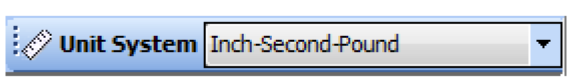
Setup

The first step is to import the model file which is already in GENOA format.

2. Make sure the **Unit System** in the upper right corner is set to **Inch-Second-Pound**.

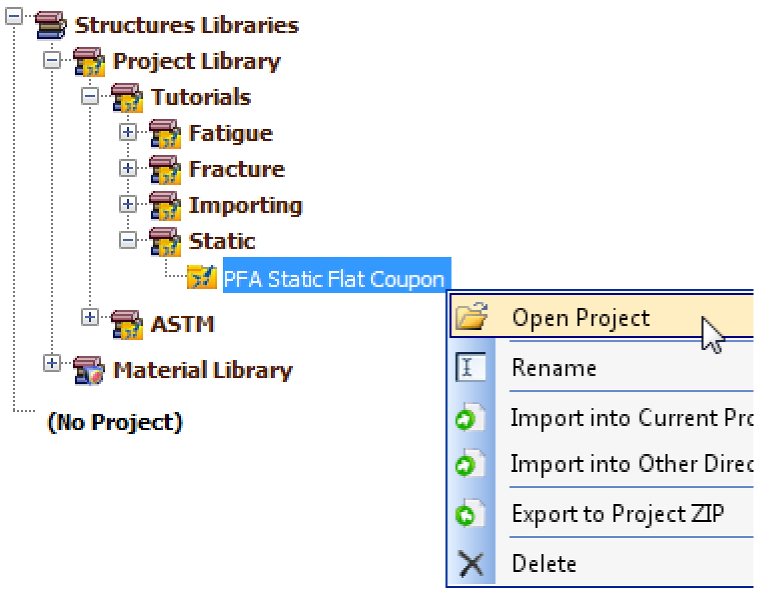
   Note: Depending upon the units that have been used to create FE models, the units should be changed before preparing or importing the model if necessary.

   Note: The imported GENOA input file contained '*UNIT' keyword to indicate the units



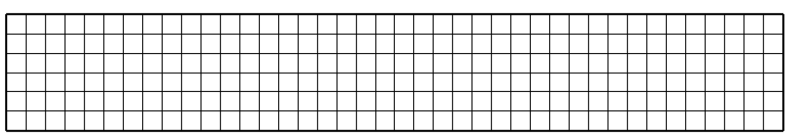
3. Right click on **PFA Static Flat Coupon** item under the **Static** tutorials and select **Open Project**.

Note: GENOA recognizes the filename extension of ".dat" as a GENOA format and automatically reads the file without any translation.



Mesh View and Setup

4. Right click on the **Mesh** node in the tree under **Input** node and select **Edit**. You may double click on the node to view and edit it. You should see the FE model as shown below.



Imported FE Model made of 4-node shell elements.

Use the **Left** mouse button for rotation of the model.

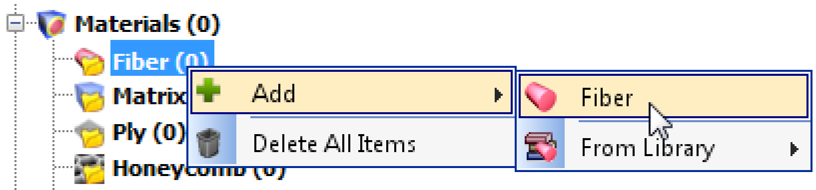
Use the **Right** mouse button for translation of the model.

Use the **Middle** mouse button for zooming in and out. You can click both left and right button and press forward or back arrow keys on the keyboard.

Roll the mouse wheel for fast zoom in and out.

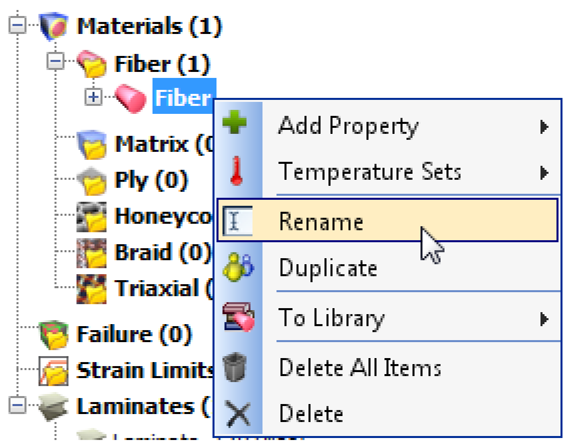
Adding a Fiber Material

5. Right click on the **Fiber** node under the **Materials** tree node. A Fiber material will be added to the fiber list.



6. Right click on the new **Fiber** material and select **Rename**. Or you may rename it by pressing **F2** key while the **Fiber** tree node is selected. Enter the new name **CARB.**

   Note: The material name can be any length, lower and upper case. You can also use numerals with the alphabets.

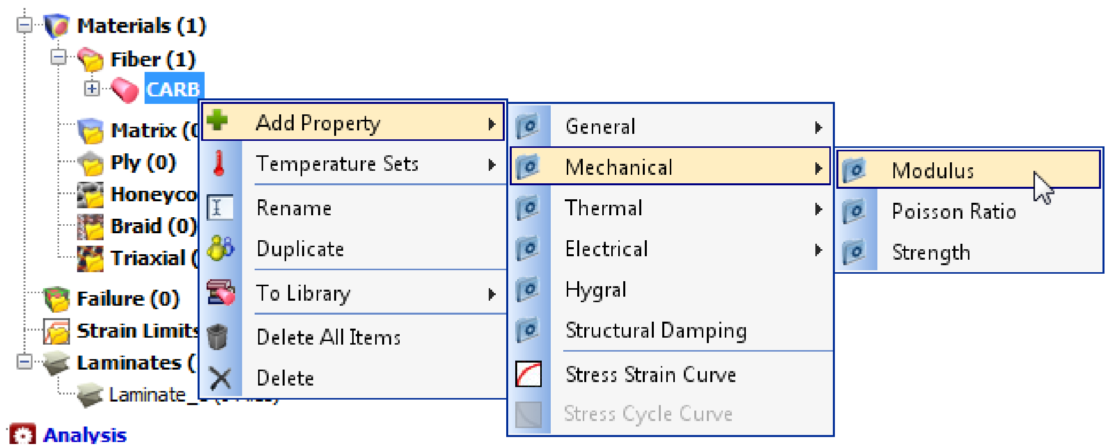


   Note: In this exercise we will use effective fiber/matrix properties as input obtained from unidirectional ply properties. Please refer to MCQ-Composites to learn how to generate effective fiber/matrix properties from unidirectional ply properties (usually obtained from test or literature).

   Note: For this exercise, the material properties are assumed to be valid for 70̊ F (room temp.).

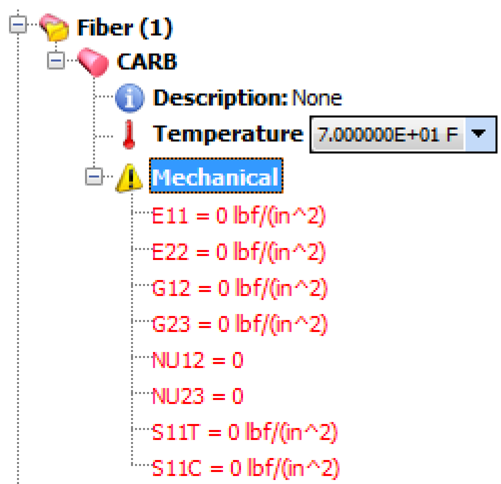
7. To enter the fiber material properties, right click on the **CARB** node and select **Add Property** from the popup menu.

8. Navigate and select **Mechanical** option to add Modulus, Poisson’s Ratio, and Strength, as shown below.



Note: the present tutorial is for PFA static analysis in dry condition, only mechanical properties are considered and other hygral-thermal-electrical properties can be ignored.

9. You should see the following after adding the **Mechanical** option.

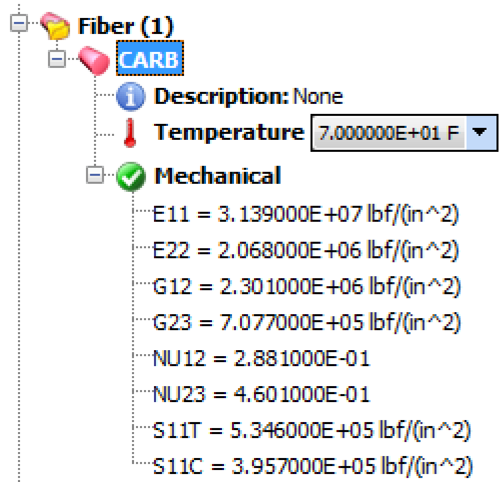


Note: Fiber properties are assumed to be transversely isotropic.

10. Double click on a single entry (shown in red in the Figure above) to edit the value.

Note: You may right click on the node and select Edit from the popup menu. To edit all the category values together in a table format, you may right click on the category, such as Modulus, and select Edit to modify the values in table format.

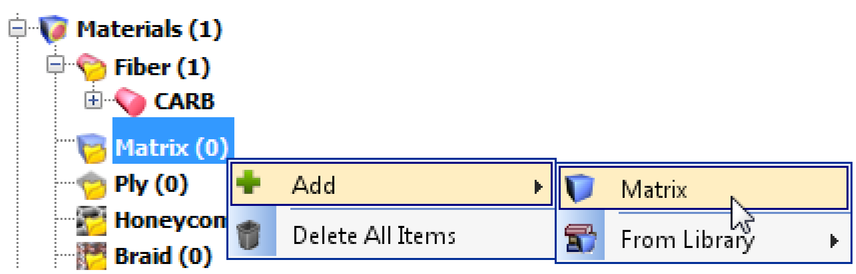
11. Edit the values to match the Figure below.



CARB Fiber Material properties

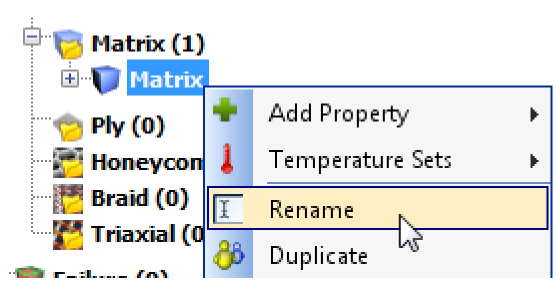
Adding a Matrix Material

12. Right click on the **Matrix** node under the **Materials** tree node. A Matrix material will be added to the matrix list.



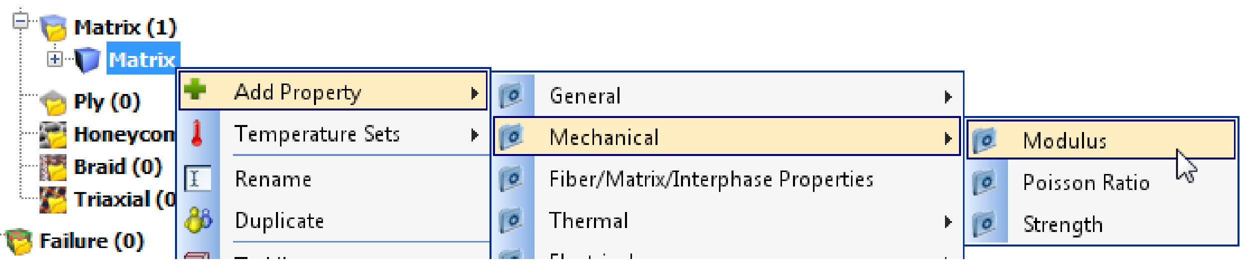
13. Right click on the new **Matrix** material and select **Rename**. Or you may rename it by pressing **F2** key while the **Matrix** tree node is selected. Enter the new name **EPOX.**

Note: The material name can be any length, lower and upper case. You can also use numerals with the alphabets.



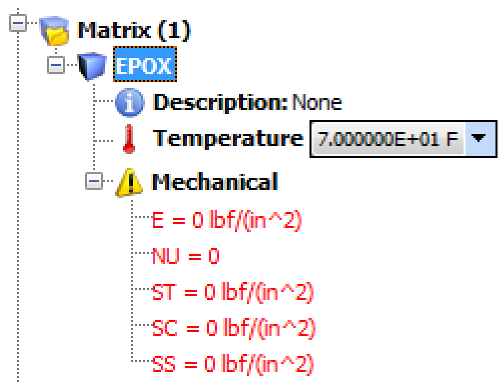
14. To enter the matrix material properties, right click on the **EPOX** node and select **Add Property** from the popup menu.

15. Navigate and select **Mechanical** option to add Modulus, Poisson's Ratio, and Strength, as shown below.



Note: the present tutorial is for PFA static analysis in dry condition, only mechanical properties are considered and other hygral-thermal-electrical properties can be ignored.

16. You should see the following after adding the **Mechanical** option.

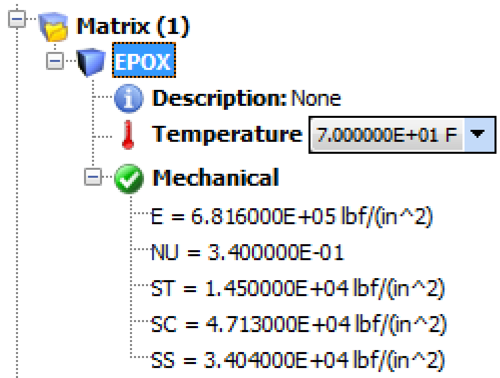


Note: Matrix properties are assumed to be isotropic.

17. Double click on a single entry (shown in red in the Figure above) to edit the value.

Note: You may right click on the node and select Edit from the popup menu. To edit all the category values together in a table format, you may right click on the category, such as Modulus, and select Edit to modify the values in table format.

18. Edit the values to match the Figure below.

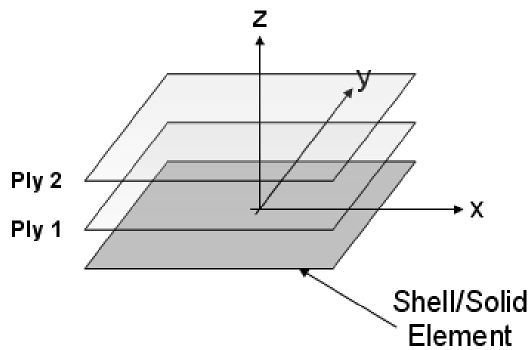


EPOX Matrix Material properties

Adding a Laminate

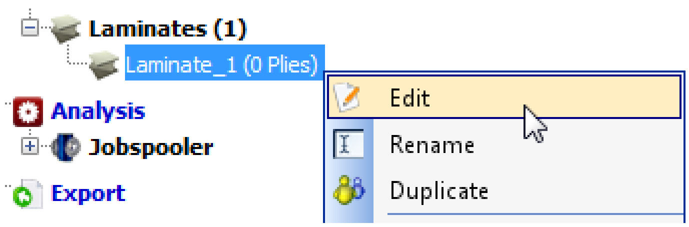
Laminates show all the laminate information in the model including the laminate sequence, the use temperature, orientation angles, material type and thickness of each ply. Since the material properties are entered at constituent level (fiber and matrix), the laminate definition contains both ply definition to identify the mixture content of fiber, matrix and void as well as lamination stacking sequence and ply thickness.

Note: Ply 1 is at the bottom and as the ply number increases it is stacked along the positive z-direction of the elements in the FE model, shown in the Figure below.

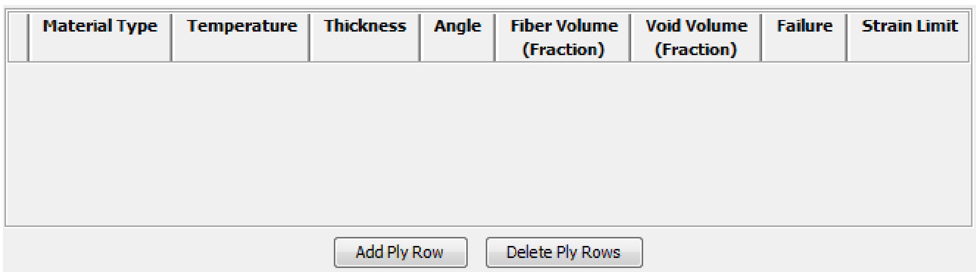


Note: By default a single laminate is created as shown in the **Laminates** tree as **Laminate_1**. To add more laminates, you would right click on the **Laminates** node and select **Add** from the popup menu. For this tutorial we will edit the current default laminate.

19. Right click on the **Laminate_1** node and select **Edit** in the popup menu. You may double click on the Laminate_1 node to view and edit it.



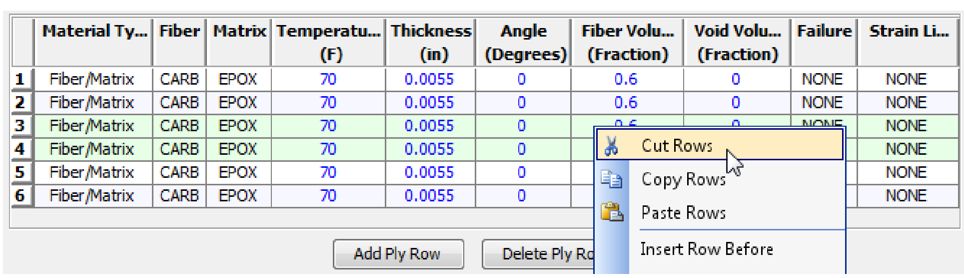
Note: The laminate editor will be displayed as shown below.



20. Click on **Add Ply Row** button six times at the bottom of the **Laminate Editor** panel to add six layers in the laminate.

    Note: You may add, delete, cut, copy, insert, and paste more plies in any configuration. You may use the buttons at the bottom to add and delete ply rows or use the popup menu as shown below by right clicking on any selected row(s).

    Note: Laminate Editor automatically determines the Fiber and Matrix names for each ply. The user needs to update the Material Type, Fiber and Matrix entries if wishes to have different material assigned to each ply.



21. Change the **Fiber Volume (Fraction)** to **0.57** (57%) by clicking and editing the cell of each six rows.

22. Similarly update the **Void Volume (Fraction)** to **0.02** (2%) by clicking and editing the related cells.

23. Now change the **Angle (Degrees)** entry of each ply to represent **[0/90/0]s** layup.

	Material Ty...	Fiber	Matrix	Temperatu... (F)	Thickness (in)	Angle (Degrees)	Fiber Volu... (Fraction)	Void Volu... (Fraction)	Failure	Strain Li...
1	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE
2	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	NONE	NONE
3	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE
4	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE
5	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	NONE	NONE
6	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE

Add Ply Row | Delete Ply Rows

Final Ply Schedule after updating the angle, thickness and material assignment for each ply.

Note: You may choose to define un-symmetric ply-lay.

Note: You may mix different Material Types: Ply, Fiber/Matrix, or Matrix in one laminate layup definition.

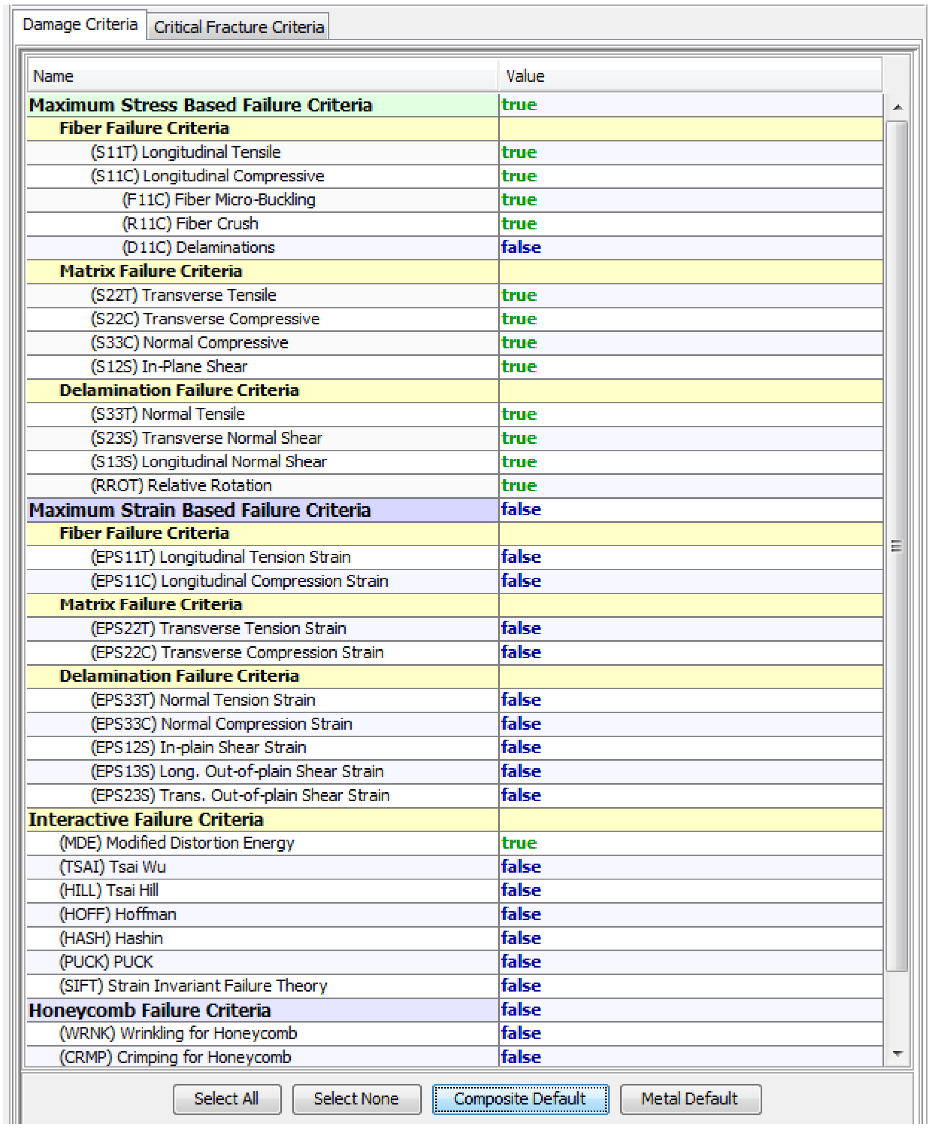
Adding a Failure Criteria

24. Right click on the Failure tree node and select Add and then Failure option, as shown in the following Figure.

Note: This will add failure criteria, FailCrit_1, as shown below.

25. Right click on the **FailCrit_1** node and select **Edit** to review the damage and failure criteria assigned to the laminates.

26. In the **Damage Criteria** tab, click on the **Composite Default** button at the bottom of the panel to select the set the default recommended criteria.



27. Similarly, in the **Critical Fracture Criteria** tab, click on the **Composite Default** button to select the default recommended failure criteria. This will set the CFC (Customized Failure Criteria) to true.

    Note: A CFC criterion (if activated) automatically determines the primary failure criteria based on the loading direction in each ply. When all the plies in the laminate meet this criteria, the element consisting of this laminate is considered broken.

    Now we will assign the failure criteria to the laminate ply layup that we were defining earlier.

28. Choose **FailCrit_1** from the drop down box in the **Failure** column for each ply.

	Material Type	Fiber	Matrix	Temperature (F)	Thickness (in)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE
2	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	NONE	NONE
3	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	FailCrit_1	NONE
4	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE
5	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	NONE	NONE
6	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	NONE	NONE

29. Make sure all of the plies are assigned **FailCrit_1** in the **Failure** column.

	Material Type	Fiber	Matrix	Temperature (F)	Thickness (in)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	FailCrit_1	NONE
2	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	FailCrit_1	NONE
3	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	FailCrit_1	NONE
4	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	FailCrit_1	NONE
5	Fiber/Matrix	CARB	EPOX	70	0.0055	90	0.57	0.02	FailCrit_1	NONE
6	Fiber/Matrix	CARB	EPOX	70	0.0055	0	0.57	0.02	FailCrit_1	NONE

30. Save the project by selecting **Save** option under **Project** menu (or press Control-S) which you may be prompted to save it in another location since the current project is in a temporary directory. This will ensure that no work is lost.

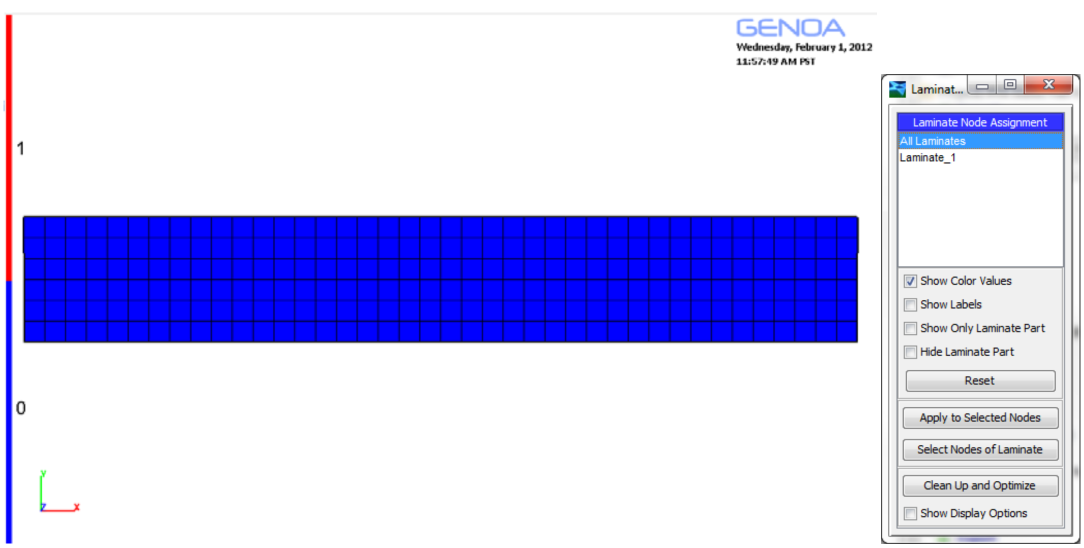
Laminate Assignments

We will now associate the finite element mesh with laminate created above.

31. Right click on the **Mesh** node and select **Edit**.

32. Click on the **Laminates Assignment** icon on the left toolbar.

Note: You can also double click on **Laminate Assignment** node under **Mesh** node in the tree.



33. Select the **Laminates_1** entry from the list in the **Laminate Assignment** Panel.

34. Click on the **Nodes** icon on the left toolbar, or select **Nodes** option under **Display** menu (press the **spacebar** on the 3D model view).

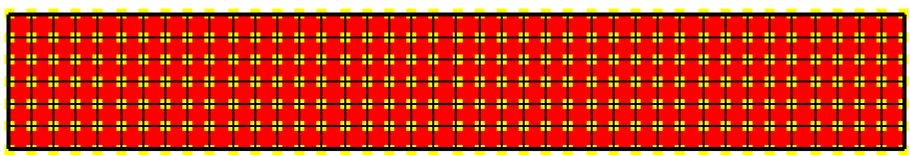
    Note: Since MHOST is our Solver for this exercise, you selected nodes in the FE Model. For all other FE Solvers, you will select the elements for assigning the Laminate card.

35. Expand the **Node Selection** arrow icon on **Nodes** panel to show the node selection panel.

36. Hold the **Shift** key and press the **left mouse button** to drag a box to select all the nodes in the model. The list of selected nodes is shown under the node selection list in the Nodes panel. (e.g.,1-287). You may also select all the nodes by pressing the **Select All Nodes** button in the **Nodes** panel.

37. With **Laminate_1** selected in the **Laminate Assignment** panel, click on **Apply to Selected Nodes** button in the **Nodes** panel. This will assign the laminate to the selection of nodes.

    Note: Blue color will switch to red color indicating that laminate assignment process was successful.



38. Click on **Select None** button under the **Nodes** panel to unselect the selected nodes.

39. Close the Laminate Assignment panel.

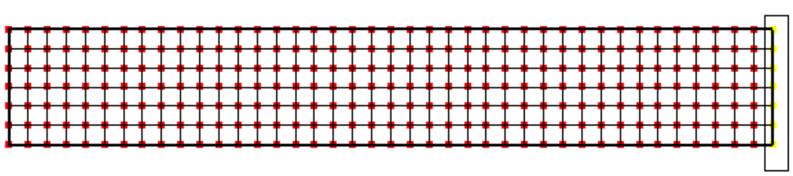
    Notes: We will keep the Nodes panel open for the force loading discussed next.

Force Loading

40. Click on the **Force** icon () on the left toolbar.

    Note: you can also access the **Force** panel by right clicking on **Mesh** node and selecting **Nodal Force** under **Load** and **Add** popup menus.

41. While holding the **Shift** key on the keyboard, click on the left mouse button and drag a selection box around the right edge nodes of the mesh.



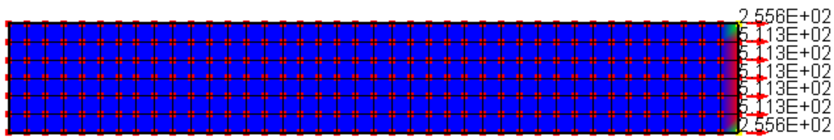
    Note: The right edge of the model, consisting of 7 nodes will be highlighted yellow while the rest of the model nodes will be red.

42. From the **Force** panel, select the **Force X** item in the force type list.

43. In the **Enter New Value** field, type in **255.6333** lbf and click on the **Apply to Selected Nodes**. The FE model will show directional arrows and a red gradient color at the right edge.

44. To apply uniform loading, we will assign double the value of 255.6333 lbf to the middle nodes of the edge. Using the mouse and Shift key, select the middle five nodes. Type in **511.2667** lbf in the **Enter New Value** field in the **Force** panel and click on **Apply to Selected Nodes** button.

    Note: The scale bar located left side of the model view shows the values of the color in the coupons discretely for further verification.



Note: The user may have to define a smaller value as in GENOA this corresponds to incremental load and not the final load as is usually the case in other commercial FE solvers.

The FE model is now ready to have boundary conditions defined.

Boundary Conditions

45. Right click on **Mesh** node and select **Boundary Conditions** under **Constraints** and **Add** popup menus.

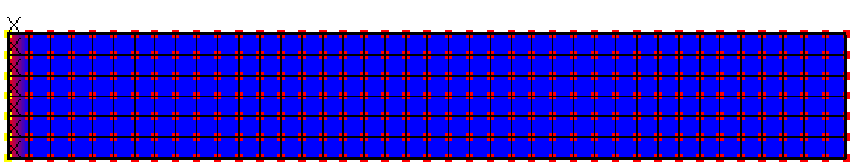
    Note: You may access the same panel by clicking on the Boundary Conditions ( ) Icon on left of the model view window.

    Notes: The Force panel will switch to the Boundary Conditions panel. The Nodes panel should still be active since we want to continue selecting nodes. The FE model should be filled with a solid blue color.

46. Press and hold the **Shift** key on the keyboard, then click left mouse button to select the left edge nodes of the model by dragging a selection box, as was done earlier to assign force.

    Note: The left edge of the model, consisting of 7 nodes, will be highlighted yellow while the rest of the model nodes will be red.

47. From the **Boundary Conditions** panel, select the **Boundary X** item in the type list and click the **Apply to Selected Nodes**. The left edge will be fixed in the X direction.



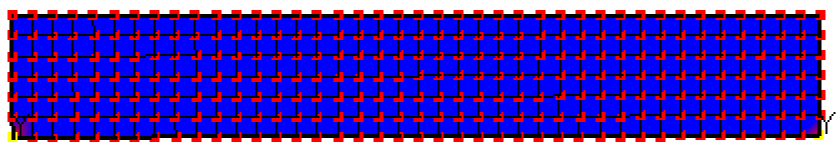
Boundary Condition X direction on the left edge

48. Similarly, select the bottom corner nodes on the left and right edges of the FE model.

    Note: You can use CTRL key on the keyboard to add nodes to existing selection. However, the Shift key is for making new selection and ignoring any existing selection.

49. From the **Boundary Conditions** panel, select the **Boundary Y** item in the type list and click the **Apply to Selected Nodes**. The node will be fixed in the Y direction, which is sufficient for this direction on this model.

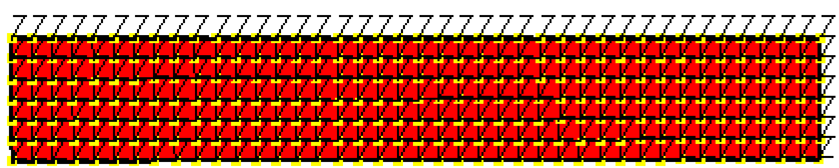
    Note: The intention here is to subject the coupon to uniform stresses and remove any possible free body motion in the FE Model.



Boundary Condition Y direction

50. Similarly, select all nodes in the whole model.

51. From the **Boundary Conditions** panel, select the **Boundary Z** item in the type list and click the **Apply to Selected Nodes**.



Boundary Condition Z direction

52. Close the **Boundary Conditions** panel by toggling the menu item under the **Constraints** submenu under **Setup** menu.

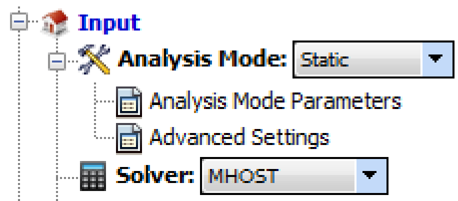
53. Unselect the selected nodes by clicking on **Select None** button under **Node Selection** sub-panel within **Nodes** panel.

54. Close the **Nodes** panel by clicking on the Nodes icon ( ) next to the model view window.

    The FE Model setup is now complete. We can now proceed to the Progressive Failure Analysis.

Analysis Type and Settings

55. Be sure that the **Analysis Mode** to **Static**.



56. Be sure that the **Solver** is set to **MHOST**.

57. Double click on **Analysis Mode Parameters** node under **Analysis Mode**, or right click it and select Edit.

58. Make sure the **Number of Nodes/Elements Allowed to Damage** is set to **24**.

59. Make sure the **Number of Nodes/Elements Allowed to Fracture** is set to **4**.

    Note: The Number of Nodes/Elements Allowed to Damage is set to 10% of the total nodes while the Number of Nodes/Elements Allowed to Fracture is set to 2%. The user should not change these parameters. They should be used by advance GENOA users only to overcome convergence issues during the simulation.

Description	Value
Number of Nodes/Elements Allowed to Damage	24
Number of Nodes/Elements Allowed to Fracture	4
Number of Iterations to Run	1000
Static Parameters	
Post Buckling Enabled	false
Unloading Enabled	false

Analysis Mode Parameters Settings

60. Double click on **Advanced Settings** node under **Analysis Mode** node, or right click it and select Edit.

61. Under **Advanced Settings** panel, set **True** for **Print Running Stress**, **Print Running Strain** and **Print Displacement** by clicking the left mouse button on the true/false cell only once.

62. Select **All Plies** next **Ply Stress** and **Ply Strain** options.

Print Options Section	
Print Multiple Large Files	false
Print Running Stress	true
Print Running Strain	true
Print Displacement	true
Print Reaction	false
Print Crack Density	false
Print Degradation for Crack Density	false
Print Margin of Safety	false
Print Miner Damage	false
Print Updated Model File	false
Print Velocity	false
Print Acceleration	false
Ply Stress	ALL PLIES
Ply Strain	ALL PLIES
Fiber Stress	NONE
Matrix Stress	NONE
Ply Principal Stress	NONE

Note: GENOA always sets these basic print options in previous two instructions to true by default and they were intentionally set to false as part of this exercise.

Note: You can request to print stress-strain data for a specific node/element in the FE Model and print it in a separate text file.

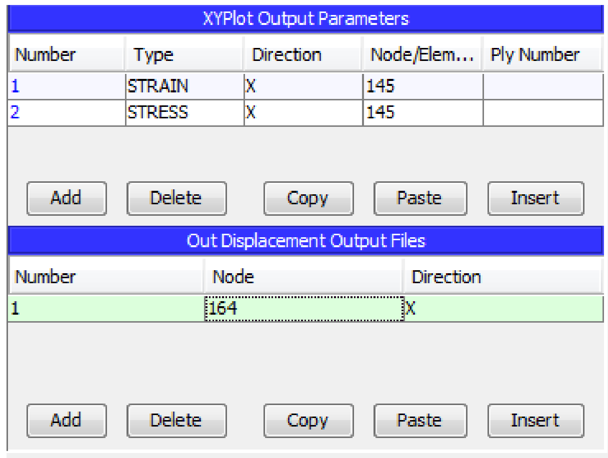
63. Set **true** for **Enable XYPlot** option under **Advanced Settings** panel.

64. Set **STRAIN** and **STRESS** for **X** direction for node **145**, as shown in the following Figure. GENOA will save the data in a separate text file during the simulation (*project_name*.sig).

    Note: You can print stress strain data for a specific ply by entering ply number in the Ply Number column. If left empty, GENOA will print the laminate level stress and strain data in the text file.

65. Similarly set **true** for **Enable Out Displacement** option under **Advanced Settings** below Enable XYPlot option. This option will allow you to print the total load, nodal reaction load and nodal displacement for the selected nodes and directions.

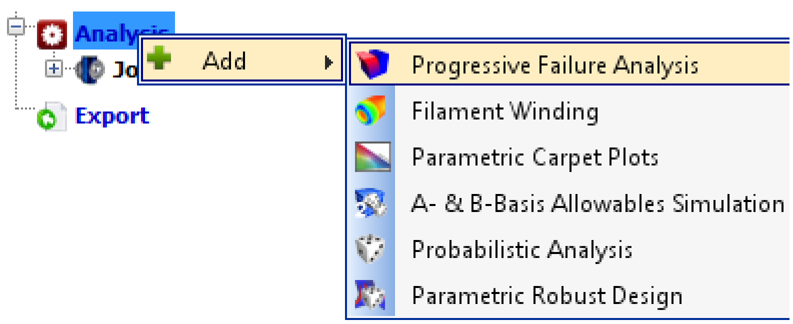
66. Set **164** and **X** node number and direction, as shown in the following Figure. Node 164 is located on the right edge of the FE model. GENOA will save the data in a separate text file during the simulation (*project_name*.dis).



Requesting output in text files during simulation

Progressive Failure Analysis

67. Right click on **Analysis** node and select **Progressive Failure Analysis** option under **Add**, as shown in the Figure below.



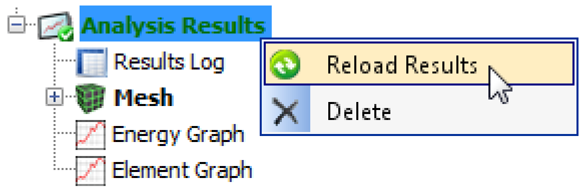
68. Right click on the **Progressive Failure Analysis** node and select **Run Analysis**.

Saving the Project

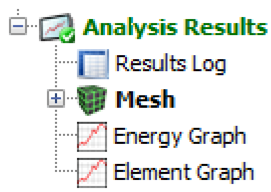
69. Save the current project and all of the changes in the memory. Choose **Save** from the **Project** menu.

Analysis Results

After the analysis is complete, the program will automatically show the **Results Log** screen. But if you wish to load the current results during the analysis, then you may press the **Reload Results** menu item under the popup menu for the **Analysis Results** node in the tree. 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 following Figure.



Results Log

70. Double click on the **Results Log** node to view the iteration log.

Note: The **Results Log** will display iteration of the analysis with the current state of the number of nodes, elements, force loading, moment, pressure, equilibrium status, number of damaged nodes, and number of fracture nodes. The rows highlighted in green are iterations that have reach equilibrium state.

Iteration	Elements	Nodes	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z	Pressure	Load	Damage...	Fracture...	Status
1	240	287	3.068E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	3.068E+03	0	0	Equilibrium...
2	240	287	3.835E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	3.835E+03	0	0	Equilibrium...
3	240	287	4.218E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.218E+03	0	0	Equilibrium...
4	240	287	4.314E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.314E+03	0	0	Equilibrium...
5	240	287	4.338E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.338E+03	287	0	Damaged/...
6	240	287	4.338E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.338E+03	287	0	Damaged/...
7	240	287	4.338E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.338E+03	287	0	Equilibrium...
8	240	287	4.362E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.362E+03	287	0	Equilibrium...
9	240	287	4.410E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.410E+03	287	0	Equilibrium...
10	240	287	4.601E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	4.601E+03	287	0	Equilibrium...
11	240	287	6.135E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	6.135E+03	287	0	Equilibrium...
12	240	287	9.203E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	9.203E+03	287	0	Equilibrium...
13	240	287	9.970E+03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	9.970E+03	287	0	Equilibrium...
14	240	287	1.016E+04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	1.016E+04	287	0	Equilibrium...
15	240	287	1.026E+04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	1.026E+04	287	574	Damaged/...

71. The log indicates that **damage** in the coupon initiates at **4314.0 lbf** and **final failure** happens at approximately **10161.4 lbf**.

    Note: the damage initiation and final failure load values should be read for equilibrium iteration.

72. **Click** on the **View Equilibrium Iterations Only** button. The log will show only the equilibrium iteration rows (highlighted in green).

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

Results Mesh

Now we will look at the 3D model deformation and results.

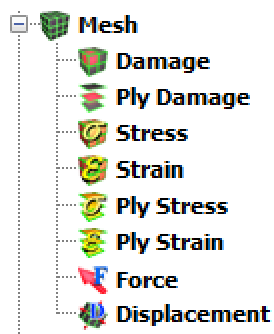
74. Double click on the **Mesh** node under **Analysis Results** node in the tree to expand the tree node.

75. Right click **Mesh** node and choose **View**.

76. **Forward** the iteration counter by scrolling the bottom slider to the far right or by clicking on the last iteration icon below.

77. Rewind the model to the first iteration by clicking the button or dragging the slider at the bottom back to the beginning.

78. The **Mesh** node under **Analysis Results** node will also show available properties to view under the Mesh node as shown below which we will examine in the next sections.



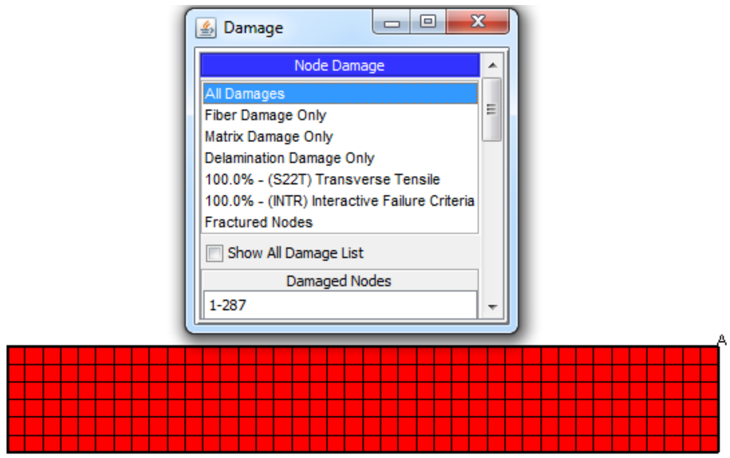
Damage

79. Double click on **Damage** node under the **Mesh** node to view the node damages. You may click on **Damage** icon ( ) located to the left of the model view window.

Note: Another option will be to press the space bar on the model view window to show the popup menu and choose **Damage** from the **Results** menu to view the node/element damages.

Note: By default, **All Damages** item is selected to show every damage criteria that has occurred in the model at the current iteration. During the initial loading, no damage has occurred yet until the model reaches iteration **5**.

80. Drag the slider to iteration **5** or enter **5** into the Iteration text box.

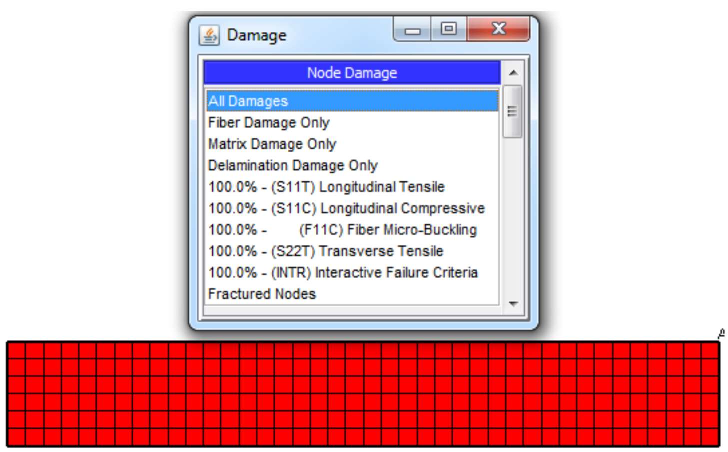


The Damage results model at Iteration 5

Note: Transverse Tensile and **Modified Distortion Energy** damage criterion have occurred within the model at the highlighted elements marked in red. This indicates that the 90 deg plies are failed due to matrix failure.

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

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



The final failure results at the last Iteration

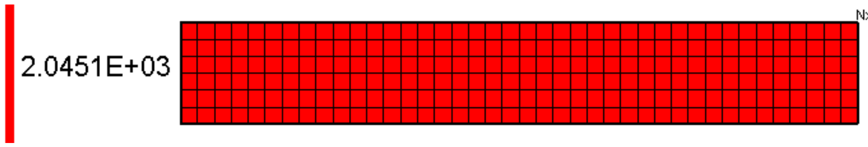
Note: The damage panel shows Longitudinal Tension failure that corresponds to fiber failure for 0 degree plies in the coupon. The Longitudinal Compression appears because the fibers in the 90 degree plies are assumed to have buckled during the analysis after matrix failure.

Element Stress

82. Select **Stress** ( ) node under Analysis Results node in the tree.

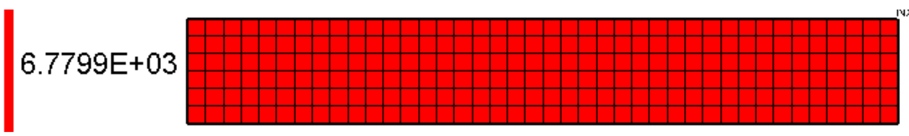
83. Select **Equilibrium Iterations** from the drop-down list box located near the bottom right of the screen.

84. Rewind the iteration to the beginning to see the initial stress.



The initial stress (Nx) at Iteration 1

85. Drag through the iterations, or press the play button , and watch the stress change with respect to the color scale on the left.



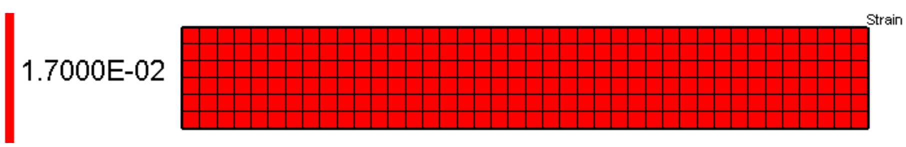
Stress (Nx) at iteration 14 stress during playback

86. Click on different stress types one-by-one **Nx**, **Ny**, and **Nxy**. You may also change the current iteration and play back the iteration animation while a particular stress type is selected.

Element Strain

87. Select **Strain** ( ) node under Analysis Results node in the tree.

88. Choose **Strain** from the **Results** menu to view the different strain results.



Strain (Strain in mid-plane Ex) mode display at last equilibrium (14) iteration

Ply Damage

89. Select **Ply Damage** ( ) node under Analysis Results node in the tree.

90. Select **All Iterations** from the drop-down list box located near the bottom right of the screen.

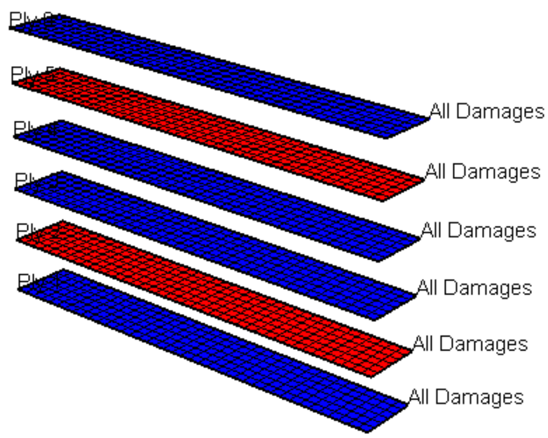
91. Change **the curren**t iteration to **5** where the first damage occurred.

92. Select **Laminate_1** in under **Select Laminate** subpanel.

93. Click and highlight all six plies in the ply window (by using the **Ctrl** or **Shift** key) to show all six plies stacked in the default Z direction. By default the **All Damages** item is selected.

94. Enter **2.0** for **Offset Distance**.

95. Rotate the mesh if necessary to view the plies from top to bottom, as shown below.



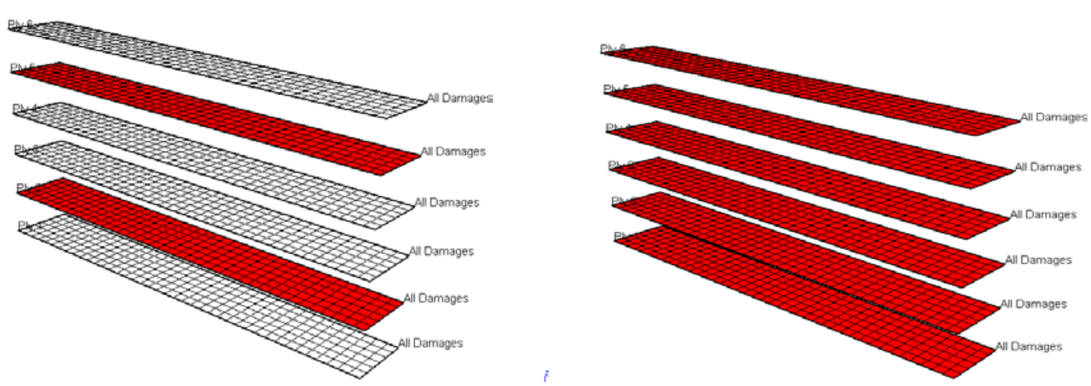
Ply-by-Ply Damage Initiation results at iteration 5

Note: Here you can examine which ply damage occurred first and the ply damage growth throughout the analysis iterations.

Note: The 90 degree plies failed due to matrix failure criteria (transverse tension).

96. Turn on the **Isolate Damage Areas** checkbox to make all the non-damaged areas transparent. This is useful for examining more complex solid models where damage may occur internally.

97. You may also change the **Offset Distance** proportion to **0.5** to reduce the spacing further.



(a) Iteration 5 (damage initiation) **(b) Iteration 15 (Final Failure)**

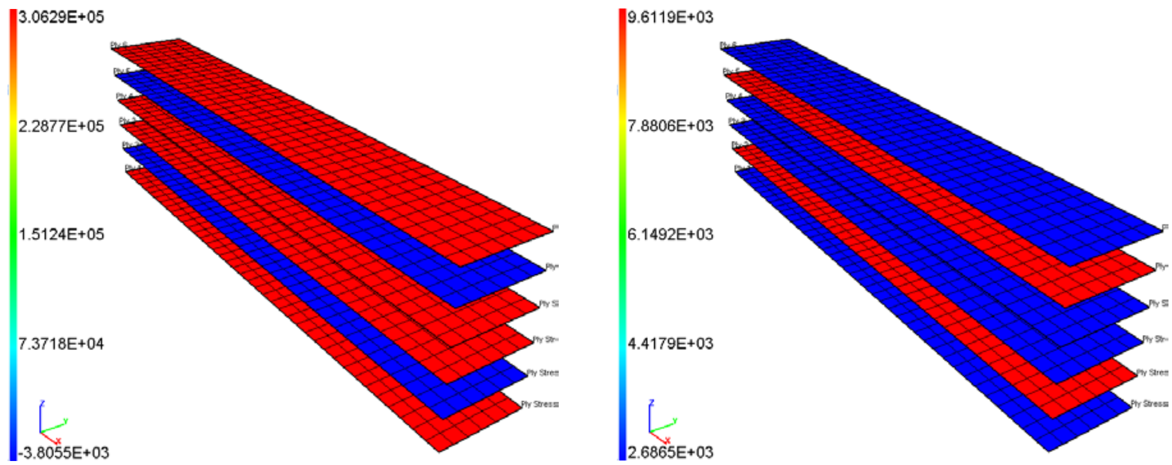
Ply Damage with Isolate Damage Areas enabled to show transparency

Ply Stress

98. Select **Ply Stress** ( ) node under Analysis Results node in the tree.

99. Click and highlight the all six plies in the ply window (by using the **Ctrl** or **Shift** key) to show all six plies stacked in the default Z direction. By default the **Ply Stress S11** item is selected.

100. **Click** and **highlight Ply Stress S11** and then **Ply Stress S22**.



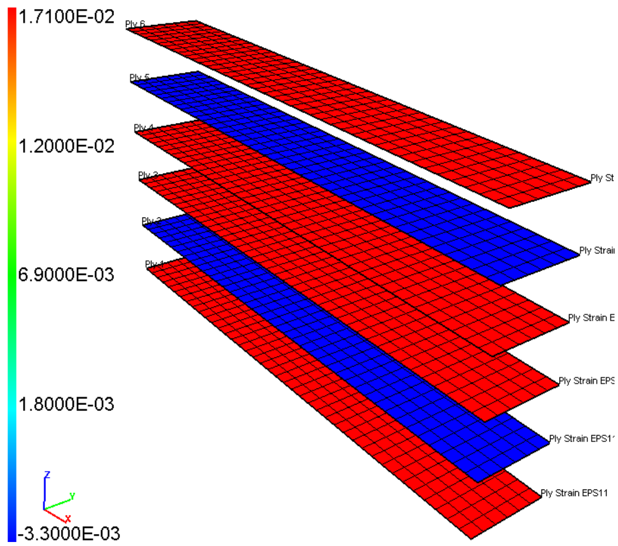
(a) Ply Stress S11 **(b) Ply Stress S22**

Ply Stress type results at last iteration 15

Ply Strain

101. Select **Ply Strain** ( ) node under Analysis Results node in the tree.

102. Click and highlight the all six plies in the ply window (by using the **Ctrl** or **Shift** key) to show all six plies stacked in the default Z direction. By default the **Ply Strain EPS11** item is selected.



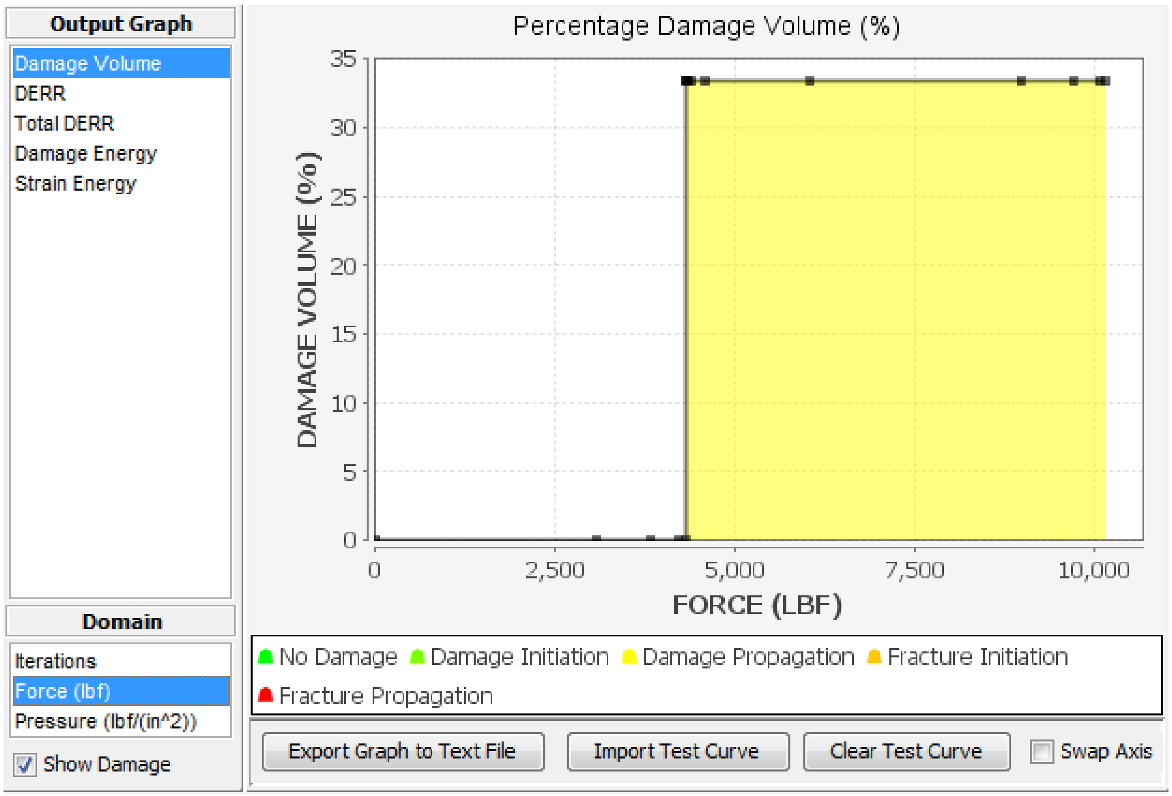
Ply Strain (Ply Strain EPS 11) results at last iteration 15

Results Graphs

103. Let's examine the results data more closely. Double click on the **Energy Graph** node under the **Analysis Results** node.

104. Click on the **Damage Volume** item for **Percent Damage Volume** under **Output Graph** Section and **Force (lbf)** under **Domain** Section.

     Note: This Percent Damage Volume is the ratio of the damaged elements in the FE model to the total number of elements times 100.



Percent Damage Volume graph

Note: The x-axis domain represents the iterations (default) while the y-axis range shows the current item selected, Damage Volume. Here the graph shows a significant rise in damage increase around **4314.0 lbf**.

Node Graph

105. Click on the **Node Graph** node.

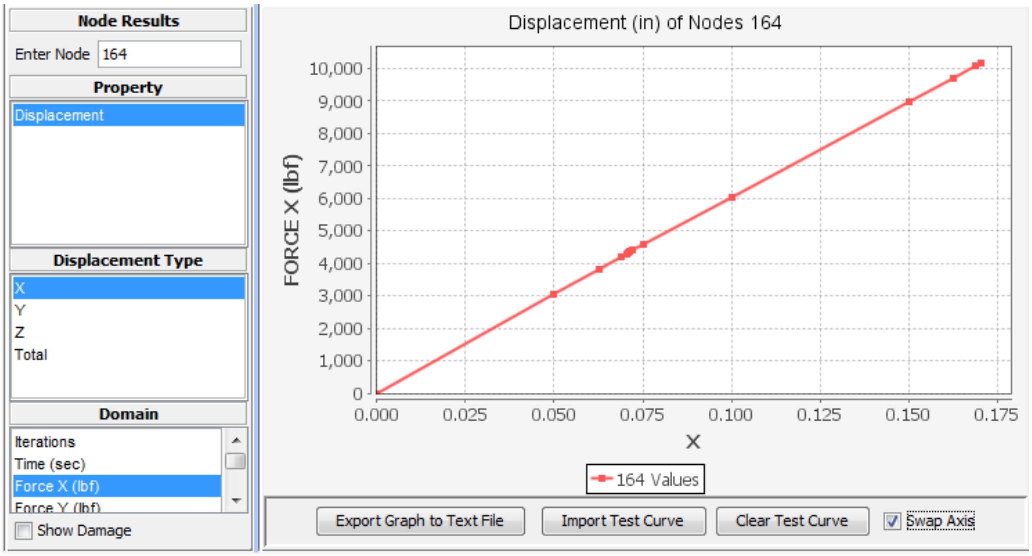
106. Type in one of the edge nodes that the force was being applied to, such as node **164**.

107. Select **Force X (lbf)** under **Domain**.

    Note: The forces shown are total forces in the x-direction in the whole FE model and not the nodal reaction forces.

    Note: The **Displacement** is default selection under **Property** panel and **X** direction under **Displacement Type**.

108. Check **Swap Axis** option near lower right corner



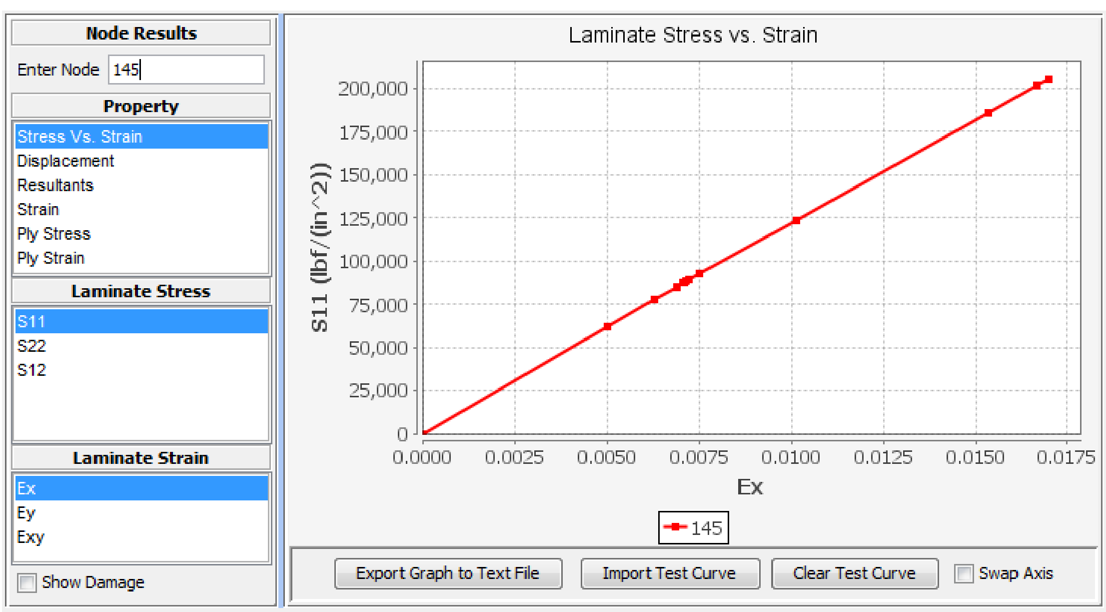
Individual node look-up of node 164 with X-Displacement

Here we will generate stress strain data for element 22 in the FE model.

109. Uncheck **Swap Axis**.

110. Change the node number to **145**.

111. Select **Stress Vs. Strain** option under **Property**. Leave the default **S11** under **Laminate Stress** and **Ex** under **Laminate Strain** options.

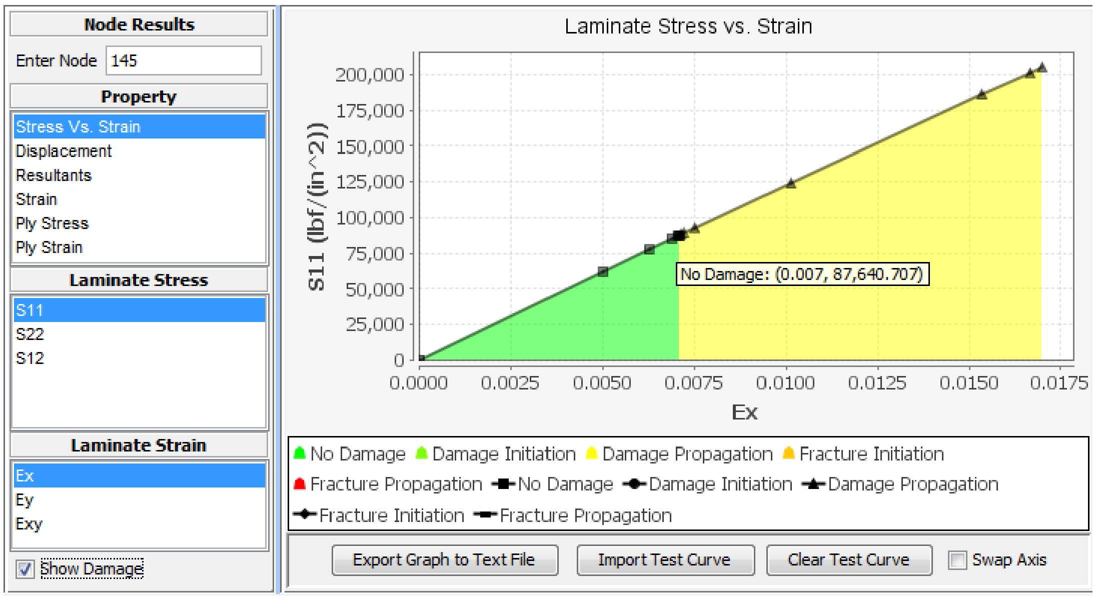


Individual element look-up of node 145: Stress-strain data

112. Select **Show Damage** option near lower left corner of the screen.

Note: This will allow you to monitor at what stress level damage initiations, propagates and then ultimately fail.

Note: Move the cursor and hold onto the point of interest in the graph and you will see the corresponding x and y values for the point on the graph.



Individual node look-up of node 145: Stress-strain data

Exporting Graph Data

You may export data throughout the GUI into different formats. For any graph in any module of GENOA, you may export the graph data into a text file (or import data depending on some modules) by clicking on the **Export Graph to Text File** button at the bottom or by selecting **Export to Text File** from the **Graph** menu.

113. Unselect **Show Damage** option near lower left corner of the screen.

114. Click on the **Export Graph to Text File** button at the bottom.

115. Type in a filename in the file dialog such as “Stress_Strain_n145.txt” and press **Export Data.**

The text file data will look similar to below.

Ex	S11_(lbf/(in^2))
0.0000000000E+00	0.0000000000E+00
4.9985982478E-03	6.1971720713E+04
6.2486343086E-03	7.7469438452E+04
6.8737007678E-03	8.5218898426E+04
7.0299734361E-03	8.7156339251E+04
7.0707253180E-03	8.7640706855E+04
7.1114776656E-03	8.8125074460E+04
7.1929832920E-03	8.9093824465E+04

Stress_Strain_n145.txt file generated on clicking Export Data button

Saving the Project

116. Click **Save** item under the **Project** menu or you may select the same function by right clicking on the Project node in the tree to show the popup menu.

117. **Exit** the Project by selecting **Exit** under **Project** menu and then confirm by clicking on **Save** button.

Check Previously Requested Stress-Strain and Displacement Files

Recall that we requested GENOA to print stress-strain for node 145 and load-displacement for node 165 in separate ASCII files (project_name.sig and *project_name*.dis), respectively. You should see these additional files in your working directory (**PFA Static Flat Coupon_PFA**) where this simulation was run on your computer.

FEM NO	145 STRAIN_11	145 STRESS_11
0	0.0000E+00	0.0000E+00
1	4.9986E-03	6.1972E+04
2	6.2486E-03	7.7469E+04
3	6.8737E-03	8.5219E+04
4	7.0300E-03	8.7156E+04
7	7.0707E-03	8.7641E+04
8	7.1115E-03	8.8125E+04
9	7.1930E-03	8.9094E+04
10	7.5192E-03	9.2969E+04
11	1.0129E-02	1.2397E+05
12	1.5352E-02	1.8603E+05
13	1.6660E-02	2.0157E+05
14	1.6987E-02	2.0546E+05

Stress_Strain data for node 145 from PFA Static Flat Coupon.sig file

DISPLACEMENTS FOR USER SELECTED NODES

INTER.	FORCE_X	FORCE_Y	FORCE_Z	PRESSURE	DISPLACE 164_1	FORCE 164_1
0	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	3.0676E+03	0.0000E+00	0.0000E+00	0.0000E+00	4.9986E-02	5.1127E+02
2	3.8345E+03	0.0000E+00	0.0000E+00	0.0000E+00	6.2549E-02	6.3908E+02
3	4.2179E+03	0.0000E+00	0.0000E+00	0.0000E+00	6.8839E-02	7.0299E+02
4	4.3138E+03	0.0000E+00	0.0000E+00	0.0000E+00	7.0412E-02	7.1897E+02
7	4.3378E+03	0.0000E+00	0.0000E+00	0.0000E+00	7.0823E-02	7.2296E+02
8	4.3617E+03	0.0000E+00	0.0000E+00	0.0000E+00	7.1233E-02	7.2696E+02
9	4.4097E+03	0.0000E+00	0.0000E+00	0.0000E+00	7.2054E-02	7.3495E+02
10	4.6014E+03	0.0000E+00	0.0000E+00	0.0000E+00	7.5339E-02	7.6690E+02
11	6.1352E+03	0.0000E+00	0.0000E+00	0.0000E+00	1.0163E-01	1.0225E+03
12	9.2028E+03	0.0000E+00	0.0000E+00	0.0000E+00	1.5440E-01	1.5338E+03
13	9.9697E+03	0.0000E+00	0.0000E+00	0.0000E+00	1.6768E-01	1.6616E+03
14	1.0161E+04	0.0000E+00	0.0000E+00	0.0000E+00	1.7100E-01	1.6936E+03

Load Displacement data for node 165 from PFA Static Flat Coupon.dis file

Edit System Configurations to Change Format Settings

If you wish to change the font size, type or precision values that you see through GENOA GUI, then do the following.

118. Under the **Project** menu, select **Advanced System Settings**.

119. Change the entries under **3D Color Scale Number Format**, as you see fit.

120. Exit and **Save** the project as explained earlier.

You have finished this tutorial of importing and editing GENOA model file, running the Progressive Failure Analysis with default MHOST FE Solver, and viewing the results.