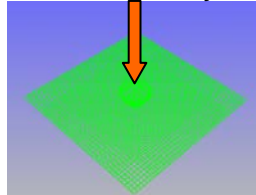


## Progressive Failure Dynamic Analysis (PFDA) of Impact Problem

**Case Description:** Composite plate subject to an impact from a rigid ball.

**Example Location:** Tutorials > Impact > Progressive Failure Dynamic Analysis  
Rigid Impactor Initial Velocity: -65.04 in/sec



4-node shell elements

**Model Description:**

Nodes: 3810; Element: 3712

Actual Plate length: 11.0 in, width: 10.0 in, thickness: 0.112 in

Impacted Plate Section: Length=5.0 in, Width=5.0 in

Impactor Diameter: 1.0 in; Impact Energy (E) = 302.57 in-lbf

**Material Description:** Graphite Fiber and Epoxy Matrix properties are used with 60% Fiber Volume Ratio, lay-up of [0/90/45/-45]<sub>s</sub> and 0.014 in ply thickness.

**Objective of Analysis:** Determine the contact force vs. time, and associated damage pattern.

**Control Type:** Initial velocity of 65.04 in/sec

**Analysis Type:** Explicit

**Solution:** **\*Explicit**  
[See PFA Case Control Keywords Section 5-2 of User Manual.](#)

Verify material properties (fiber and matrix) and ply schedule

Define contact surfaces and contact pair

**Input Requirements:** Define rigid part and assign initial velocity and boundary conditions on it  
Setup termination time and time step

**FEA Solver:** LS-DYNA

**Output from Analysis:** Structural deformations, ply stresses and strains, load-displacement curve, active failure modes at various time including damage initiation and progression, contact force, impactor velocity and energy.

**Damage Pattern:**

Longitudinal tensile/compressive (fiber crushing), transverse tensile/compressive, in-plane/out-plane shear.

**Summary of Results:** **Contact Force vs. Time:**

Contact force reaches the peak (3137.9 lbf) at around 9.60E-3 sec, and reduces afterwards because the rigid ball is bounced back.

## Introduction

This tutorial uses the composite plate impact example in the Example Directory of GENOA. The purpose of this tutorial is to help user to learn how to use the progressive failure dynamic analysis (PFDA) capability of GENOA. In this example, the rigid ball (impactor) moves towards the composite plate with initial velocity 65.04 in/sec, and crashes the composite plate. The impactor is bounced back afterwards. During the impact process, the composite plate is damaged due to various criteria, such as longitudinal compression/tension, in-plane shear, etc. The amplitude of contact force with respect to time increases at beginning and reaches the peak before dropping again due to bounce-back.

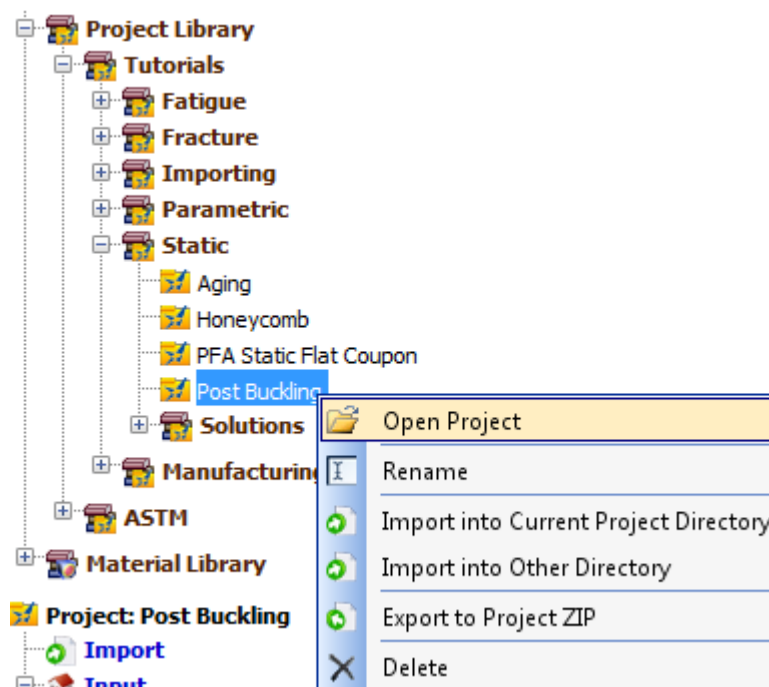
## Launch GENOA

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.

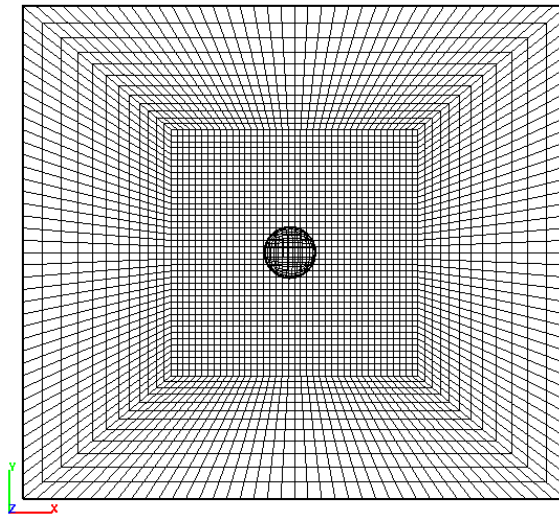
2. Make sure the **Unit System** in the upper right corner is set to **Inch-Second-Pound**.
3. Right click on **Progressive Failure Dynamic Analysis** node under the **Impact** node and select **Open Project** (see Figure below).



Replace this figure

**Note:** GENOA will read the file and will automatically import the FE model, ply schedule and any material properties defined in it.

**Note:** The FE model was originally generated in PATRAN; therefore, the Solver Type is set as MSC\_NASTRAN.



**Main GENOA screen after importing**

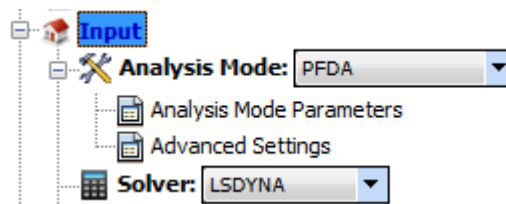
**Note:** By default the model is displayed at the +Z direction. For a better view of the model, select the **–Y** direction icon located above the Mesh Setup view below the tab. You may also access the option under the **Display** menu (click on spacebar on the keyboard when Mesh Setup view is active) and **Orientation** submenu. You will see a composite plate and a spherical impactor displayed on the screen.



**Model view from the –Y direction**

### Analysis Mode Parameters

4. Switch **Static** to **PFDA** next to the **Analysis Mode** node in the tree.
5. Switch **MSC\_NASTRAN** to **LSDYNA** next to the **Solver** node in the tree (see Figure below).



6. Double click on **Advanced Settings** node under **Analysis Mode** node in the tree.
7. Under **Print Options Section** set **Print Velocity** and **Print Acceleration** options to **true**.

**Note:** This option will allow the user to monitor the change in the velocity and the acceleration once the simulation is over and under Results Mesh view.

8. Under **Post Damage Degradation** category, make sure **Element Failure Factor** is set to **0**.

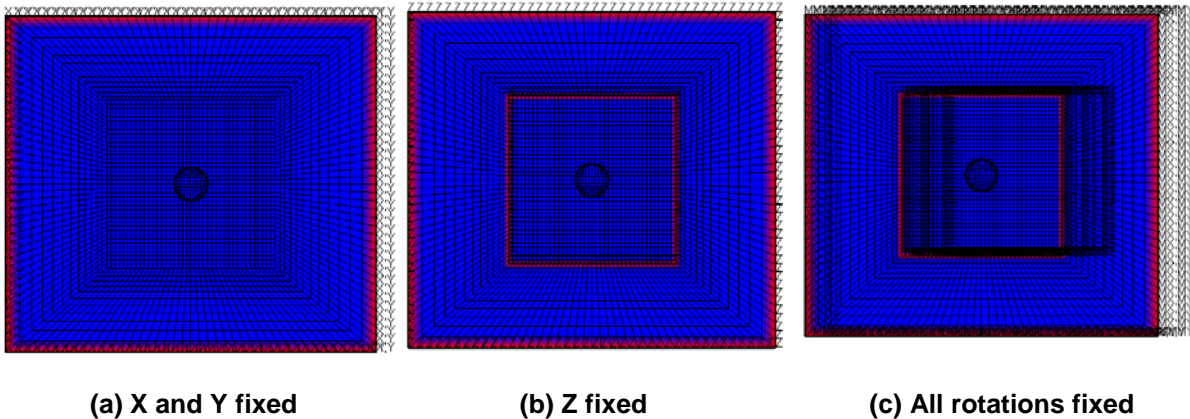
**Note:** When a user sets the Element Failure Factor to other than 0, the code doesn't remove the elements upon failure and assigns the entered residual stiffness to the broken elements. This is different from Fiber/Matrix and Ply failure Tension/Compression Factors in the sense that a ply

residual stiffness in the tension, shear and compression will be assigned based on defined values once the ply in an element is damaged while other plies in the element are not damaged.

## Boundary Conditions

We will review the Boundary Conditions assigned to the model.

9. Click on **Boundary Conditions** icon to display the boundary conditions assignment to the model.
10. **Holding the CTRL key**, select **Boundary Y to MZ** to multiple select five items. The labels will show **X, Y, Z, MX, MY, and MZ** at the nodes with each of the **Boundary** constraints (same as Figure below).



Boundary Conditions

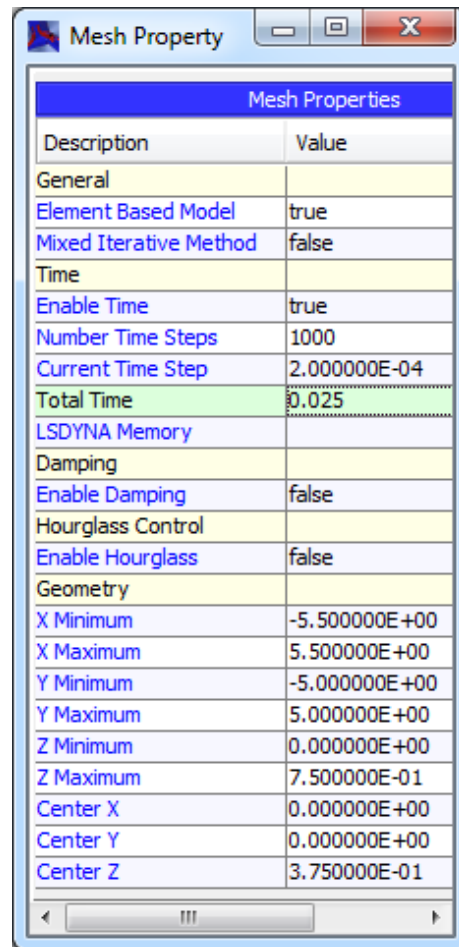
**Note:** For this example, the boundary conditions are assigned for impact loading. The model can be modified later to switch to static type loading for simulating Compression After Impact (CAI), where the boundary conditions can be defined on the edge of the panel rather than near the center area. Multiple boundary conditions can be viewed at the same time by holding the **CTRL** key down and choosing **Boundary X**, **Boundary Y**, and **Boundary Z** from the list.

## Time definition for PFDA

11. Click on **Mesh Properties** (M) icon to define time increments and total time for this analysis.
12. Set **Enable Time** item to **true**. Four more options will appear.
13. Leave the default **Number Time Steps** to **1000**.
14. Enter the **Current Time Step** to be **0.0002**.
15. Enter the **Total Time** to be **0.025** (see Figure below).

**Note:** This will setup the time interval for PFDA to update the damages and the total simulation time. The smaller the **Current Time Step** is the more data points will be generated during the simulation.

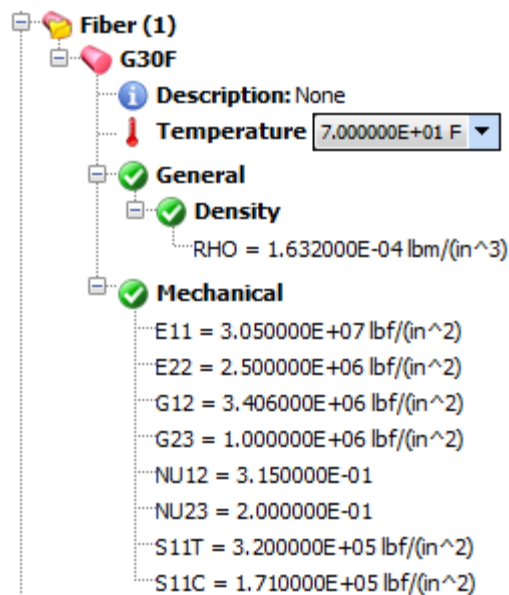
16. Close the **Mesh Properties** panel.



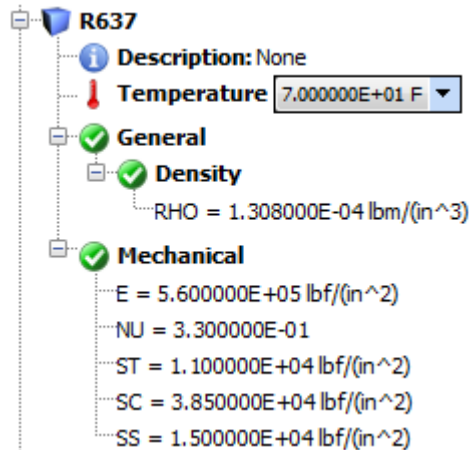
Modified Mesh Properties panel for time

### Material Properties

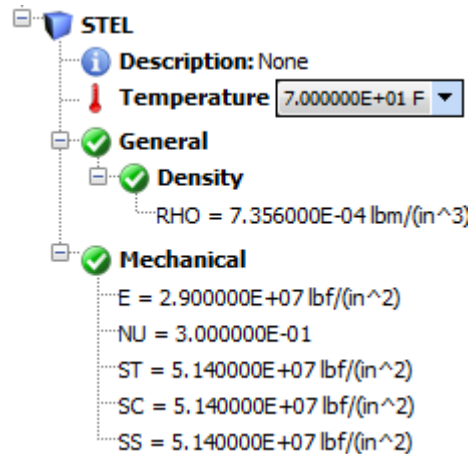
17. Expand the **Materials (3)** node in the tree.
18. Expand the **G30F** node under **Fiber (1)** node and verify if the input matches as shown below.



19. Expand the **R637** node under **Matrix (2)** node and verify if the input matches as shown below.



20. Expand the **STEL** node under **Matrix (2)** node and verify if the input matches as shown below.



**Note:** We will not modify any values for this exercise.

**Note:** For explicit problems make sure you define the density value.

### Laminate Editor

21. Expand the **Laminates** nodes till you see **Laminate\_1** node and then double click to view the individual plies in laminate 1 (see Figure below).

**Note:** Be sure that each ply thickness equals 0.014 inch.

	Material Type	Fiber	Matrix	Temperature (F)	Thickness (in)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	0.000000E+00	6.000000E-01	2.000000E-02	FailCrit_1	NONE
2	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	9.000000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
3	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	4.500000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
4	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	-4.500000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
5	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	-4.500000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
6	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	4.500000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
7	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	9.000000E+01	6.000000E-01	2.000000E-02	FailCrit_1	NONE
8	Fiber/Matrix	G30F	R637	7.000000E+01	1.400000E-02	0.000000E+00	6.000000E-01	2.000000E-02	FailCrit_1	NONE

Laminate Editor for viewing the ply angles and Laminate\_1 information [0/90/45/-45]s

22. Similarly double click on **Laminate\_2** node in the tree to view the individual plies in laminate 2 (see Figure below).


**Note:** Laminate 2 is assigned to the composite panel.

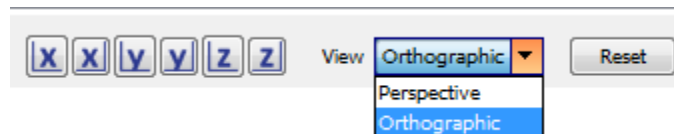
	Material Type	Matrix	Temperature (F)	Thickness (in)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Matrix (Isotropic)	STEL	7.000000E+01	5.000000E-02	0.000000E+00	0.000000E+00	0.000000E+00	FailCrit_1	NONE
2	Matrix (Isotropic)	STEL	7.000000E+01	5.000000E-02	0.000000E+00	0.000000E+00	0.000000E+00	FailCrit_1	NONE

#### Laminate Editor for viewing the ply angles and Laminate\_2 information

**Note:** Laminate 1 is assigned to the Impactor and is just a place holder as it will be replaced by a Rigid Part definition discussed next.

#### Define Rigid Part

23. Click on **Rigid Part** (  ) icon to launch the **Rigid Part** panel.
24. Click on **Add** button in the **Rigid Part** panel at the top.
25. The **Rigid Part Group 1** will be displayed and selected in the **Rigid Part Setup** list box.
26. In the **Editor** subpanel below, change the mass **Density** in the factor box to be **0.9107 lbm/in<sup>3</sup>**.  
**Note:** The density here refers to the value obtained by dividing the actual mass ( $2E/v_o^2=0.143052$  lbm) of the real impactor divided by the volume of the meshed impactor ( $0.15708 \text{ in}^3$ ). The density unit should be consistent with the density unit in the material.
27. Change the **Modulus** to be **560.0E+06** psi
28. Change the **Poisson** to be **0.3**
29. Change the **Thickness** to be **0.1** inch.
30. Change **Constraint** to be **Global Coordinate System**. Two more items for constraint will appear.
31. Choose **XY** in the **Translation** box. The program will fix the x and y direction of this rigid part.
32. Chose **XYZ** in the **Rotation** box.
33. Be sure to orient the FE Model as shown below using the orientation **–Y** icon above the Model view window (see Figure below).
34. Select **Orthographic** option from the drop down list next to view (see Figure below).



35. Launch **Elements** panel.
36. Drag a box and **select** the **elements** of the **Impactor** in the FE model.

**Note:** All the selected elements will turn yellow.

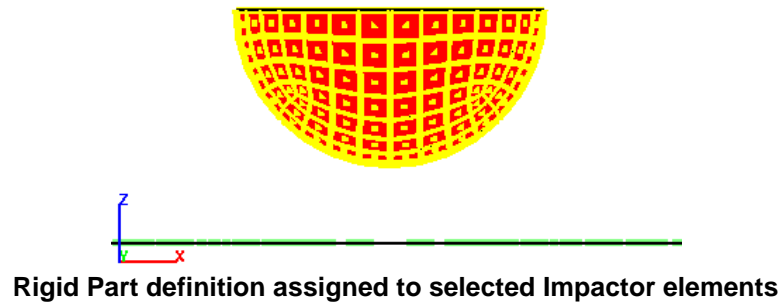


#### Selected Impactor elements

37. Click on **Apply to Selected Elements** button.

**Note:** The above definition will be applied to the selected impactor elements. The impactor elements should turn red to indicate assignment to the Rigid Part Group 1 with the above definitions (see Figure below).





38. Close **Rigid Part** panel.

### Creating Node Based Surfaces


39. Click on **space bar** when active in **Mesh Setup** view mode.
40. Select **Node Surfaces** option under **Setup** popup menu.
41. Click on **Add** button to define a new surface under **NSURFACE** panel. An item **New** will appear in the list box.
42. Click on **Rename** button and type in the new name **IMPACTOR** for your surface.
43. Click on **Apply to Elements** button to assign the impactor elements to **IMPACTOR**.
44. Under the **Elements** display panel, expand the **Element Selection** submenu by **clicking** on the arrow icon to the right of the label.
45. Click on **Select None** tab.

**Note:** The assigned **IMPACTOR** surface will now be highlighted in red.

We will now assign the other elements of the model that consist of the plate to a new surface too.

46. Drag a box and **select** the **elements** of the **Panel** in the FE model.
47. Under **NSURFACE** panel, click on **Add** button to define a new surface. An item **New** will appear in the list box.
48. Click on **Rename** button and type in the new name **PANEL** for your surface.
49. Click on **Apply to Elements** button to assign the plate elements to **PANEL**.
- Note:** The assigned **PANEL** surface will now be highlighted in red.
50. Close the **NSURFACE** panel.
51. Expand **Element Selection** subpanel under **Elements** panel and click on **Select None** button.
52. Close **Elements** panel.

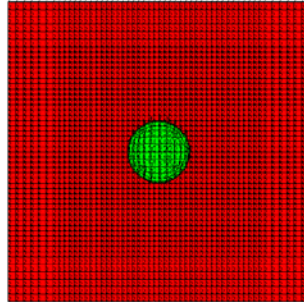
### Assign Contact Pairs

53. Click on **Contact** (  ) icon to choose and define contact surfaces and conditions.
54. Click on **Add** button which **Contact Group 1** will appear in the list box.
55. Select **SURFACE TO SURFACE** as **Type**.
56. Make sure **Type Value** is set to 3.
57. **Set** the **Static Coefficient Friction**, **Dynamic Coefficient Friction**, **Exponential Decay Coefficient**, **Tolerance**, and **PENCHK** to **0.01**, **0.008**, **0.0**, **0.0** and **0**, respectively.



**Note:** the user can consult LS-DYNA User Manual for details about these keywords.

58. Select **IMPACTOR** in the **Select Master Surface** list box.
59. Select **PANEL** in the **Select Slave Surface** list box.
60. Close the **Contact** panel.

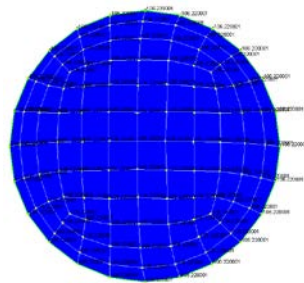


**Editing Contact Pairs (Green and Red indicate Master and Slave Surfaces, respectively)**

### Define Initial Velocity

We will now assign them with initial velocity.

61. Launch **Nodes** panel.
62. Drag a box and **select** the **nodes** of the **Impactor** in the FE model.  
**Note:** All the selected nodes will turn yellow.
63. Click on **Velocity** (V) icon to launch the **Rigid Part** panel.
64. Select **Translation Z** in the **Velocity Properties** list box.
65. In the **Enter New Value** field, enter **-65.04** in/s for the initial velocity.
66. Click on **Apply to Selected Nodes** button in the **Velocity** panel.
67. The newly added velocity will appear in the model as a blue color along with labels of magnitude. The blue indicates on the color scale to the left that the value is negative while zero is red.



**Setting Initial Velocity to the Impactor**

68. Click on **Select None** button under **Node Selection** subpanel inside **Nodes** panel.
69. Close the **Nodes** and **Velocity** panels.

### Failure Criteria

70. Under the **Failure** node, double click on **FailCrit\_1** node to review the damage and failure criteria assigned to the laminates.

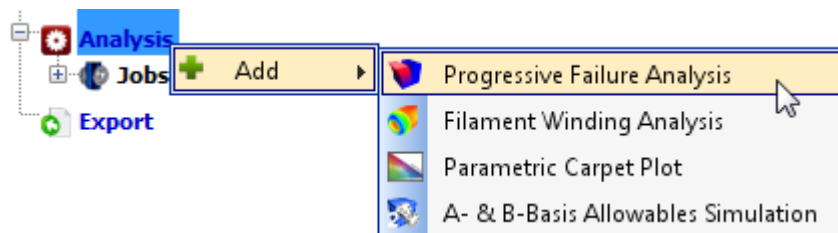
**Note:** For this exercise will not modify the Failure Criteria.

Damage Criteria	Critical Fracture Criteria
Name	Value
<b>Maximum Stress Based Failure Criteria</b>	<b>true</b>
<b>Fiber Failure Criteria</b>	
(S11T) Longitudinal Tensile	true
(S11C) Longitudinal Compressive	true
(F11C) Fiber Micro-Buckling	false
(R11C) Fiber Crush	true
(D11C) Delaminations	false
<b>Matrix Failure Criteria</b>	
(S22T) Transverse Tensile	true
(S22C) Transverse Compressive	true
(S33C) Normal Compressive	true
(S12S) In-Plane Shear	true
<b>Delamination Failure Criteria</b>	
(S33T) Normal Tensile	true
(S23S) Transverse Normal Shear	true
(S13S) Longitudinal Normal Shear	true
(RR0T) Relative Rotation	true
<b>Maximum Strain Based Failure Criteria</b>	<b>false</b>
<b>Fiber Failure Criteria</b>	
(EPS11T) Longitudinal Tension Strain	false
(EPS11C) Longitudinal Compression Strain	false
<b>Matrix Failure Criteria</b>	
(EPS22T) Transverse Tension Strain	false
(EPS22C) Transverse Compression Strain	false
<b>Delamination Failure Criteria</b>	
(EPS33T) Normal Tension Strain	false
(EPS33C) Normal Compression Strain	false
(EPS12S) In-plane Shear Strain	false
(EPS13S) Long. Out-of-plane Shear Strain	false
(EPS23S) Trans. Out-of-plane Shear Strain	false
<b>Interactive Failure Criteria</b>	
(MDE) Modified Distortion Energy	true
(TSAI) Tsai Wu	false
(HILL) Tsai Hill	false

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

### Progressive Failure Analysis

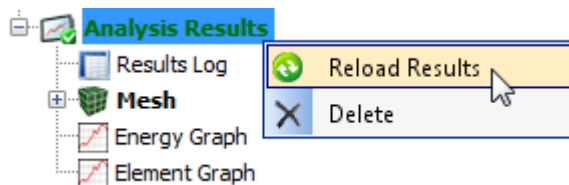
72. Right click on **Analysis** node and select **Progressive Failure Analysis** option in the **Add** popup menu.



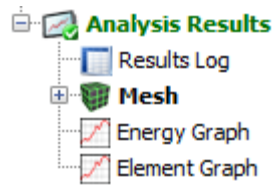
73. Right click on **Progressive Failure Analysis** node and select **Run Analysis**.

### 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.



**Note:** the results can slightly be different using different version of LS-DYNA in this example LSDYNA971R5 has been used.

## Results Log

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

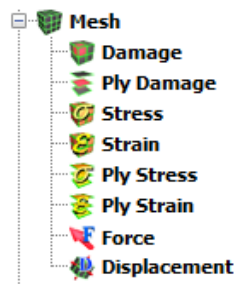
Iteration	Elements	Nodes	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z	Pressure	Time	Damage...	Fractur...	Status
1	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.000000E-04	0	0	Equilibr...
2	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	4.000000E-04	0	0	Equilibr...
3	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	6.000000E-04	0	0	Equilibr...
4	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	8.000000E-04	0	0	Equilibr...
5	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	9.999999E-04	0	0	Equilibr...
6	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	1.200000E-03	0	0	Equilibr...
7	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	1.400000E-03	0	0	Equilibr...
8	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	1.600000E-03	0	0	Equilibr...
9	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	1.800000E-03	0	0	Equilibr...
10	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.000000E-03	0	0	Equilibr...
11	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.200000E-03	0	0	Equilibr...
12	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.400000E-03	0	0	Equilibr...
13	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.600000E-03	0	0	Equilibr...
14	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	2.800000E-03	0	0	Equilibr...
15	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	3.000000E-03	0	0	Equilibr...
16	3712	3810	0.000000...	0.000000...	0.000000...	0.000000...	0.00000...	0.000000...	0.00000...	3.200000E-03	0	0	Equilibr...

Partial Results Log table

## Results Mesh

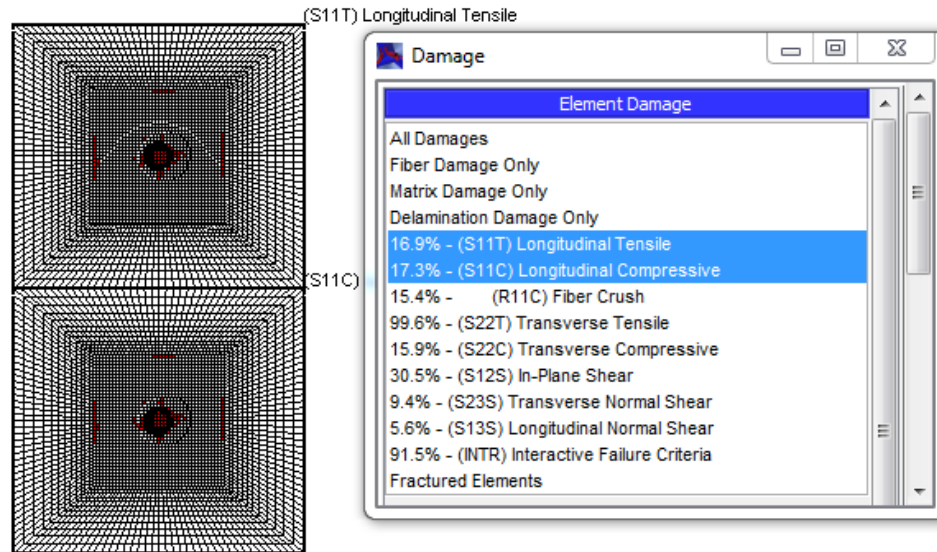
75. Double click on **Mesh** node in the tree under **Analysis Results** node.

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



## Laminate Damage

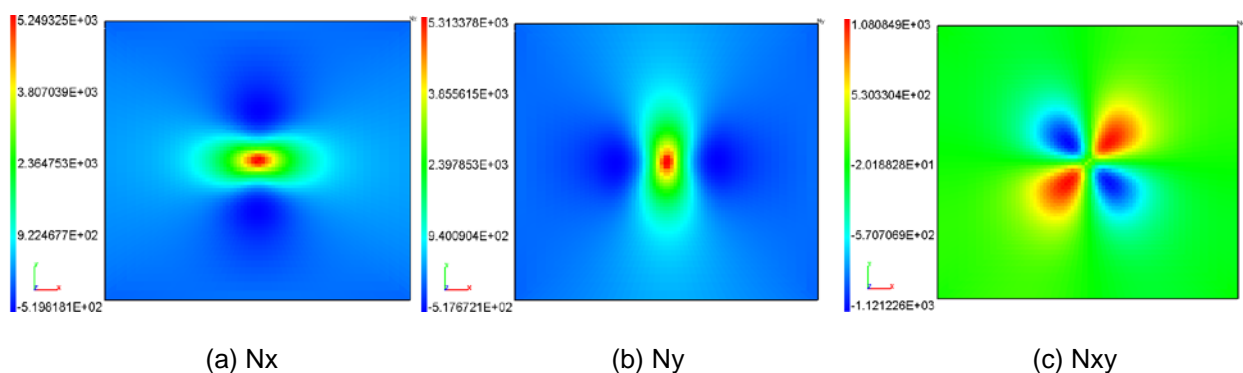
76. Double click on **Damage** node under **Mesh** node in the tree.
77. Because the impactor sphere might hide any damage to the plate underneath, **enable** the **Isolate Damage Areas** from the **Damage** panel for better visibility.
78. Multiple select the damage criteria **Longitudinal Tensile** and **Longitudinal Compressive** modes.
79. You may select multiple damages, etc. in the property window on the right by using the **CTRL** key and selecting multiple items as shown below.



Damage view of model with Isolate Damage feature at peak load (9.6 ms)

### Laminate Stress

80. Click on **Laminate** icon.
81. Click on **Show Only Laminate Part** toggle button and select **Laminate\_1** to isolate the spherical impactor.
82. Close **Laminate** panel.
- Note:** We will hide the mesh for better viewing before stresses are shown on the panel.
83. Select **Wireframe** under **Display** menu.
84. Uncheck **Show Inner Mesh** option and then **close** the **Wireframe** panel.
85. Click on **Stress** (🔍) icon to view the different stress results.



Viewing the Stress types of the model at peak load (9.6 ms)

### Ply Damage

86. Click on **Ply Damage** icon or double click on **Ply Damage** node under **Analysis Results** node in the tree to view the different ply results.
- Note:** We will isolate the plate by choosing only the Laminate of the plate to view.
87. Check the **Isolate Laminate Part** checkbox under **Ply Damage** panel.

88. Click on **Laminate\_1** in the **Select Laminate** list box to isolate the plate.

**Note:** We will now view the damages in the plies.

89. Multiple select all **8 plies** in the **Select Ply(s)** list box, as shown in the Figure below.

90. To view the damage areas better, check **Isolate Damage Areas** to make all of the non-damaged areas transparent.

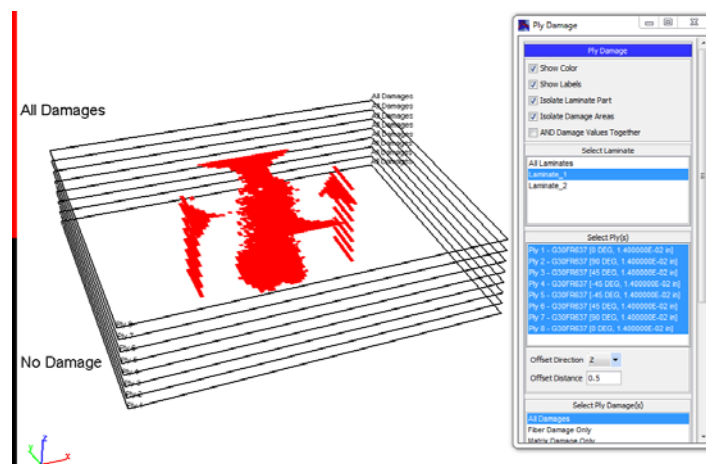
**Note:** We will hide the mesh again for better viewing before stresses are shown on the panel.

91. Select **Wireframe** under **Display** menu.

**Note:** You can access the **Display** menu by clicking on **Spacebar** on the keyboard when **PFA Mesh Results** window is the active window.

92. Uncheck **Show Inner Mesh** option and then **close** the **Wireframe** panel.

93. **Change** the **Offset Direction** to **Z** and set **Offset Distance** to **0.5**.



Viewing the separate Ply Damage of multiple plies at peak load (9.6 ms)

## Ply Stress

94. Double click on **Ply Stress** node under **Analysis Results** node in the tree.

95. **Check** the **Isolate Laminate Part** checkbox.

96. Select **Laminate\_1** from the list box.

97. Select first **three** plies in the **Select Ply(s)** box.

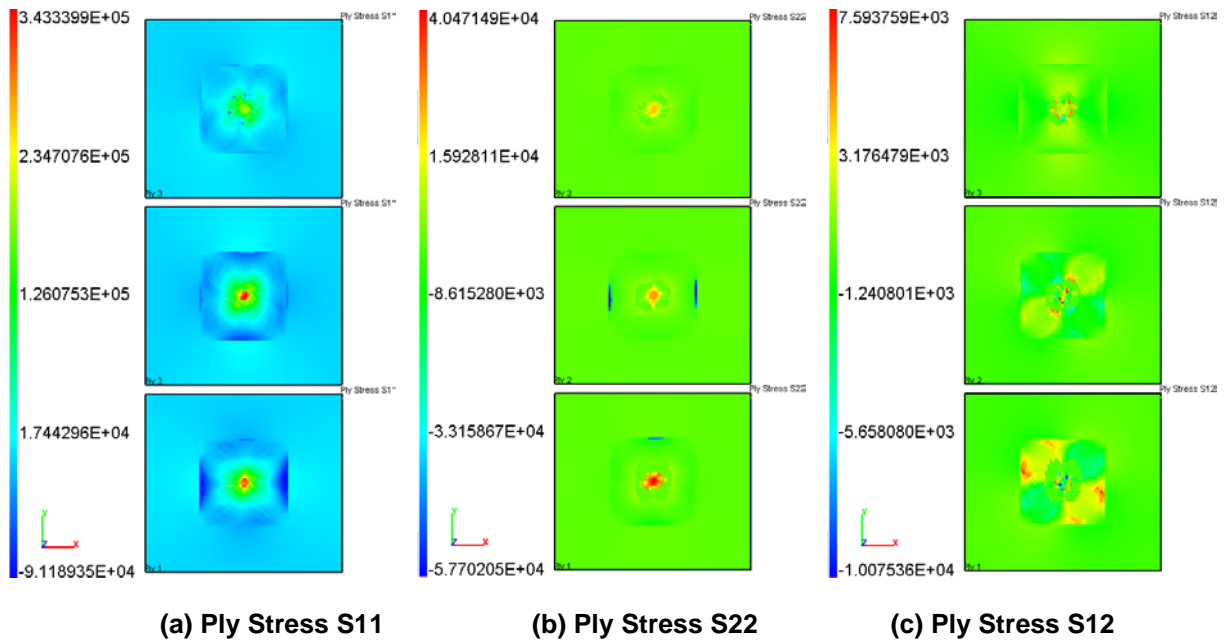
98. **Change** the **Offset Direction** to the **Y** direction.

99. Go to **Wireframe** under **Display** menu and uncheck **Show Inner Mesh** option when a dialog box appears.

100. Select **+Z** option under **Orientation** submenu under **Display** menu for orienting the FE model.

101. Select **Ply Stress S11**, **Ply Stress S22**, and **Ply Stress S12** one-by-one.

**Note:** Figures below show each of the plies and selected stress types.



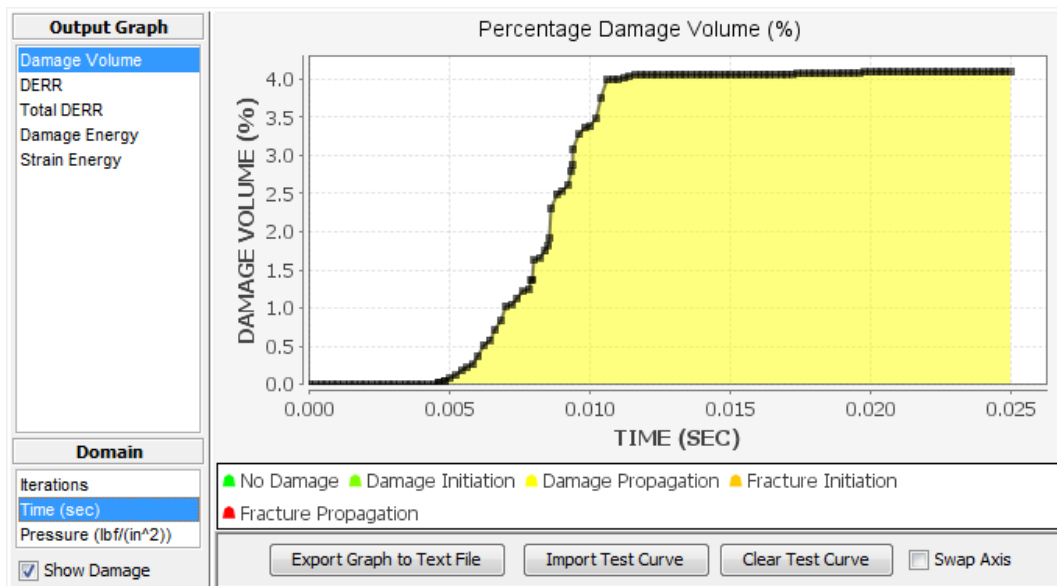
Viewing simultaneous plies and ply stress types at peak load (9.6 ms)

102. Close **Ply Stress** panel.

## Results Graphs

103. Double click on **Energy Graph** node under **Analysis Results** node under the tree.

104. Click on **Damage Volume** (under **Output Graph** panel) and **Time (sec)** under **Domain** panel in the default **Energy** graph section to show the percentage of damage volume occurring throughout the analysis.



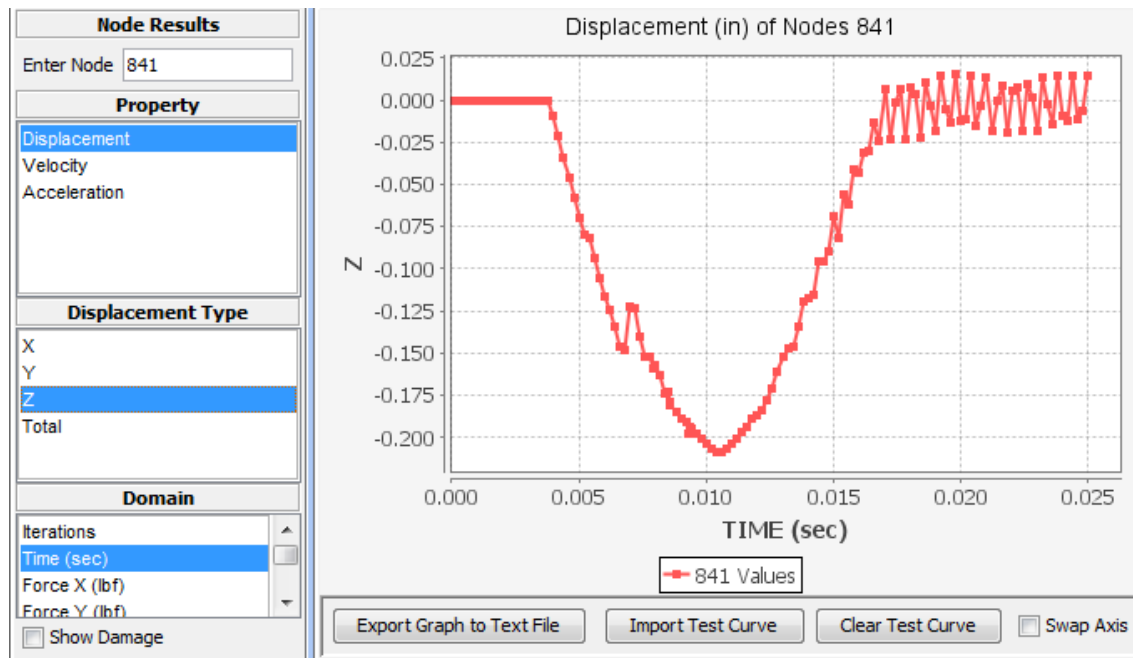
Damage Volume versus time graph of the PFA results

## Node Graph

105. Double click on **Node Results** node under **Analysis Results** node in the tree.

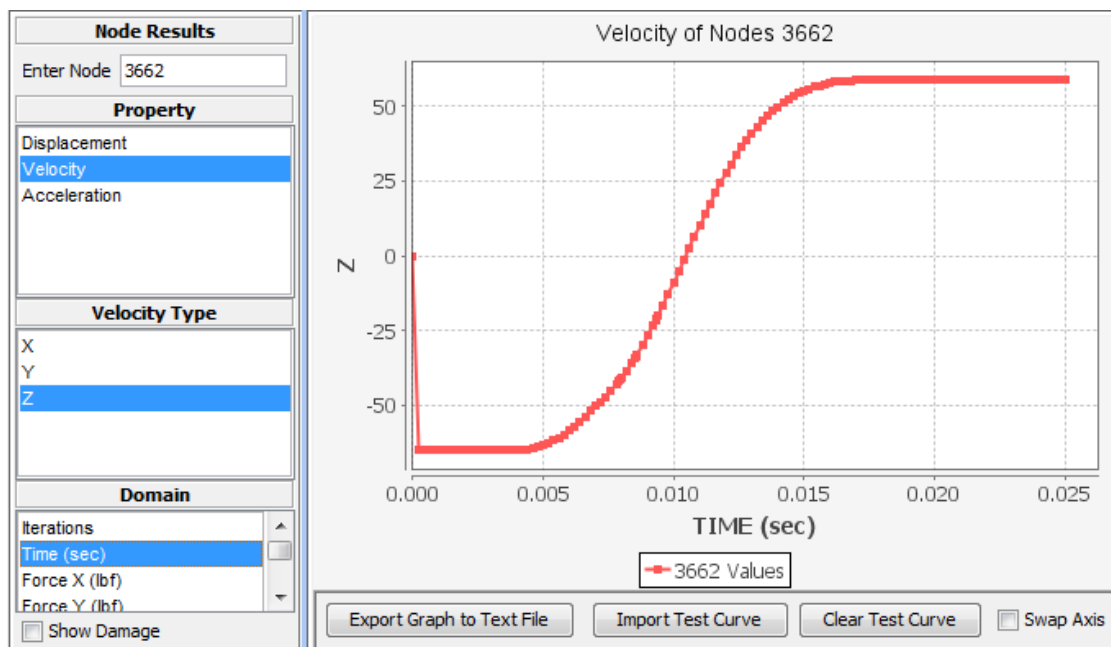


106. Enter **841** in the **Enter Node** field to display the results for Node 841 which in the center of the composite plate.
107. Under **Property** window, **Displacement** will appear. Select **Z** under **Displacement Type** window and **Time (sec)** under **Domain** window.



Displacement versus time of Node 841 (center of the panel) in the Z direction

108. Enter **3662** in the **Enter Node** field to display the results for Node 3662 which in the center of the impactor.
109. Under **Property** window, select **Velocity** option. Select **Z** under **Velocity Type** window and **Time (sec)** under **Domain** window.

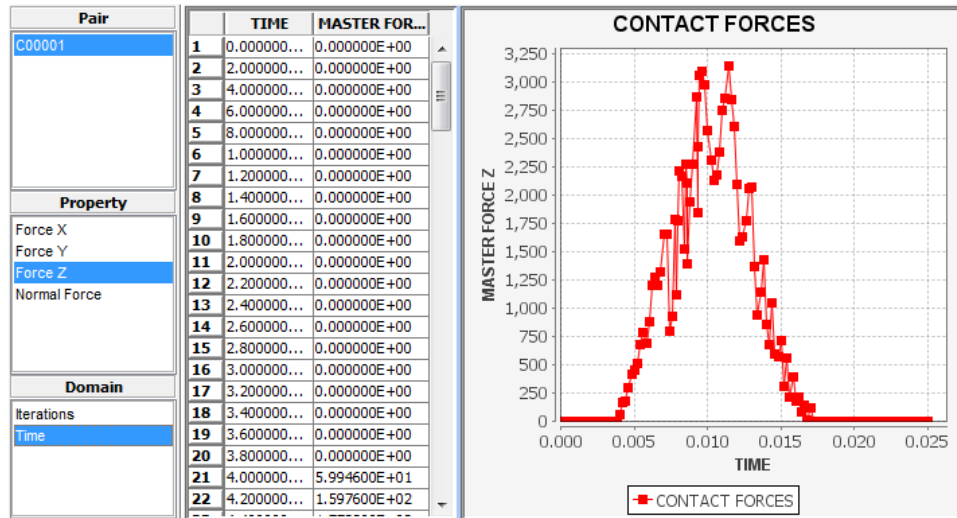


Impactor velocity versus time of Node 3662 (impactor) in the Z direction



### Contact Force Graph

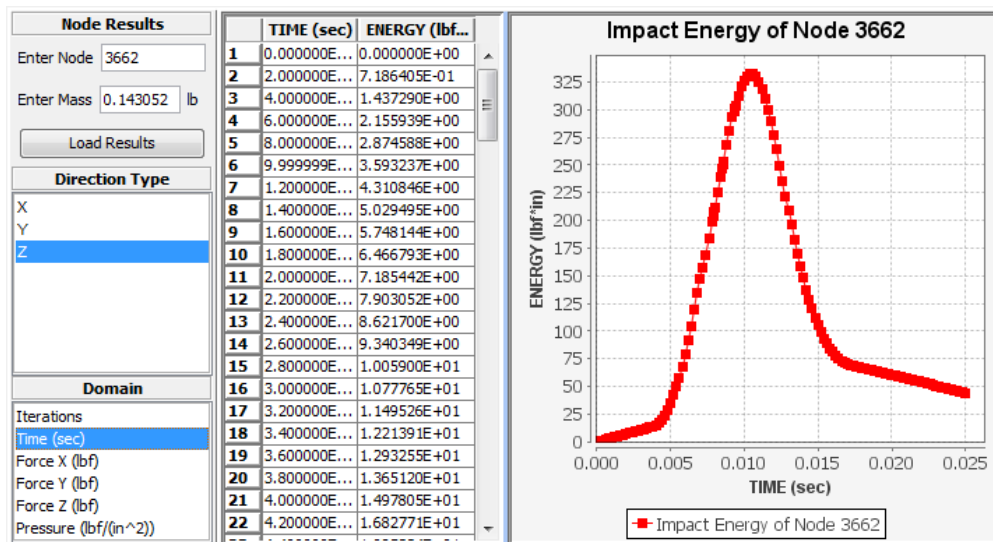
110. Double click on **Contact Force Graph** node under **Analysis Results** node in the tree.
111. Select **C00001** under **Pair** window.
112. Select **Force Z** under **Property** window.
113. Under **Domain** window, choose **Time** as the x-axis.



Contact Force Plot

### Impact Energy Graph

114. Double click on **Impact Energy Graph** node under **Analysis Results** node in the tree.
115. Enter **3662** in the **Enter Node** field—center of the impactor
116. Enter **0.143052** lbm for **Enter Mass** field.
117. Click on **Load Results** button.
118. Select **Z** as **Direction Type** and **Time (sec)** under **Domain** window.



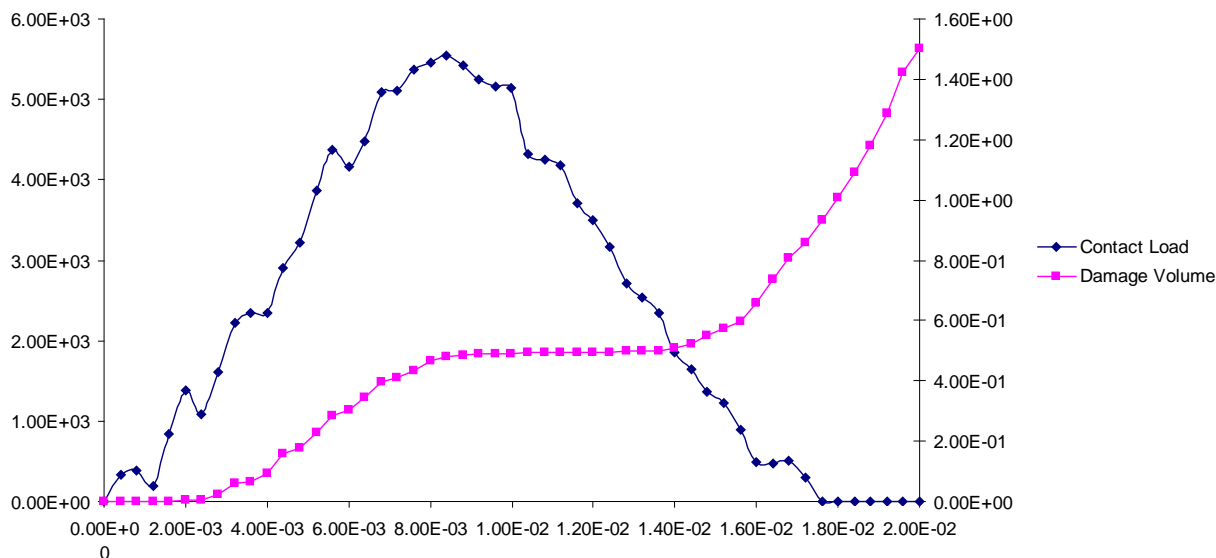
Impact Energy Plot

### Code Limitation

The combination of Explicit Dynamic analysis and local degradation of the material could cause instable behavior for large velocities. At this point it is important to understand the physics behind this instability. This section explains how to cope with these problems.

Due to impact, the material is degraded locally. This means that stiffness properties will be reduced at the damaged nodes. The degraded nodes however still have their original masses. The combination of low stiffness and significant mass in the plate will result in dynamic loads in the plate, degrading even more nodes. This results in unstable damage propagation in the plate. To account for this behavior, the effects of this behavior must be recognized and ignored. An example shows how this is done for a plate impacted with a velocity of 78.7 inches/sec.

Figure below shows both the contact load as well as the damage volume. From this figure the instability behavior can be detected. It can be seen that after the contact load has reached its maximum load, the damage volume stabilizes. With the reduction of the contact load, the plate loses its support and becomes more dynamic, introducing additional loads in the plate. This is shown in this figure by the increase of the damage volume. It should also be noted that after the impactor has lost its contact with the plate, the damage still keeps growing exponentially indicating instable behavior.



**Contact Load and Damage volume showing instable behavior**

Performing an impact analysis, it is important to recognize the stabilized damage volume, the damage corresponding to this state should be used as the final damage.

You have finished this tutorial of setting up the FE model for Dynamic Impact Analysis using an Explicit FE solver.