

# HELIUS:MCT User Experience

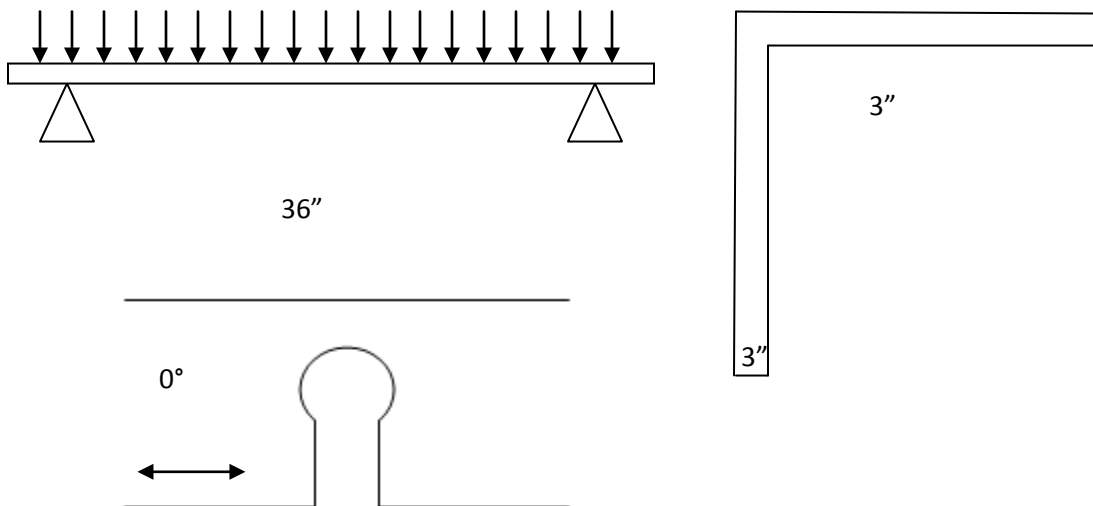
## A LOOK AT COMPOSITE FAILURE SIMULATION WITH HELIUS:MCT AND ANSYS MECHANICAL APDL (ANSYS CLASSIC)

### Helius:MCT Features Demonstrated

- **Progressive Failure** – How the failure in the composite material will initiate and propagate through a structure.
- **Multi-Scale Failure** – The composite ply stresses will be separated into stresses in the fiber and matrix. Using the stresses in the matrix, failure in the matrix will be predicted. Similarly, using the stresses in the fiber, failure in the fiber will be predicted.

### Problem Description

Let's examine how the failure of the composite materials of a notched L-Beam in bending initiate and propagate through the beam until ultimate failure is achieved. Our model will be a simply supported 3"x3" L-Beam that is 36" long placed under uniform pressure distribution.



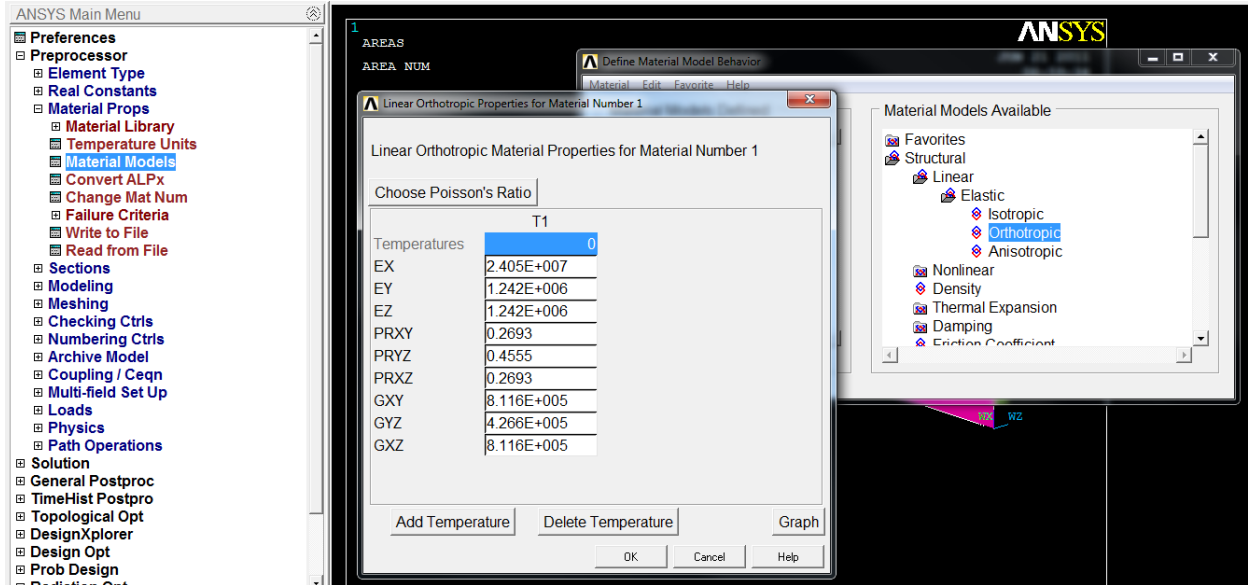
The layup in this example is an 8 ply symmetric, quasi-isotropic layup,  $[45/0/-45/90]_s$ , composed of a unidirectional carbon/epoxy composite (IM7/8551).

# How Using Helius:MCT Differs From A Conventional ANSYS Mechanical Analysis

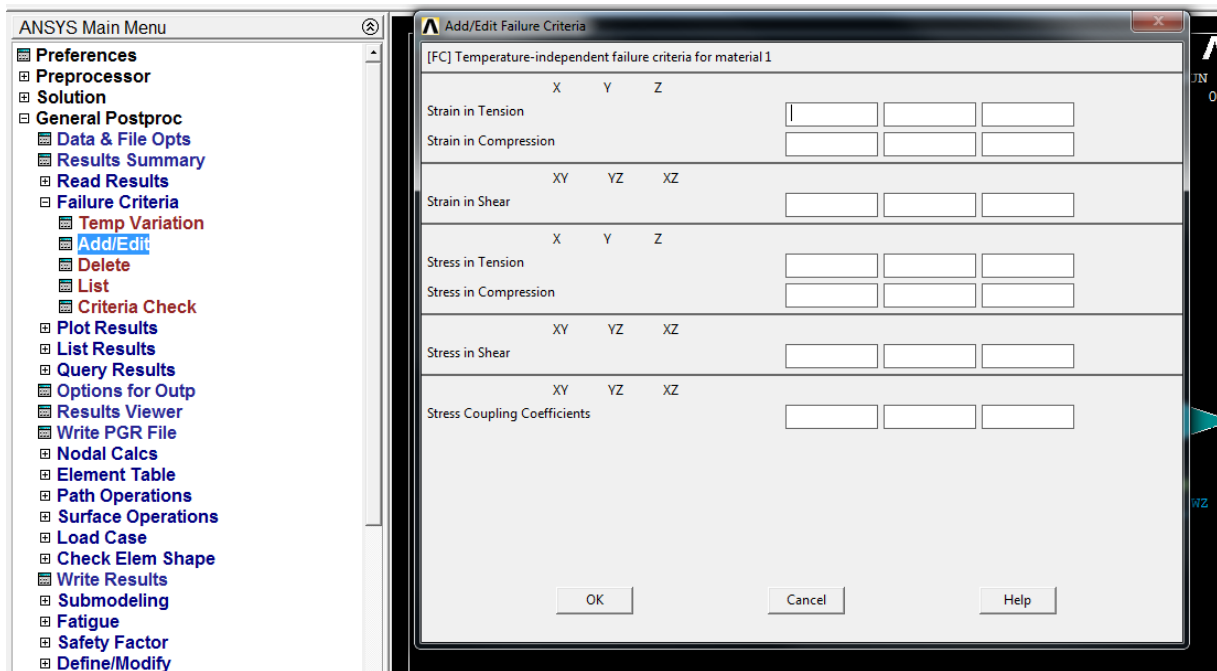
## Material Definition

### Conventional Method:

Create orthotropic material using lamina material properties provided through a combination of testing and material data sheets.



Specify material stresses/strains at failure to use various failure criteria to provide a failure index.



### Helius:MCT Method:

A material is created outside of ANSYS Mechanical using Helius Material Manager. The newly created material is stored in a material library that is set up during install and can be any folder on a computer or a shared folder on a network. To run a multi-scale, progressive failure analysis, the only additional material input required beyond the elastic constants and strength values of the lamina is the fiber volume fraction of the lamina. Matrix and fiber material constants are required for a multi-scale analysis but default properties are provided and are adjusted when the material is saved so that the combination of fiber and matrix material constants provide the same material constants as the composite lamina when assembled together into a composite. The reason this extra level of detail is required is so that the Helius:MCT software can separate **composite ply stresses** into **fiber and matrix stresses** to provide more accurate failure determination and subsequent material property reduction for a progressive failure simulation.

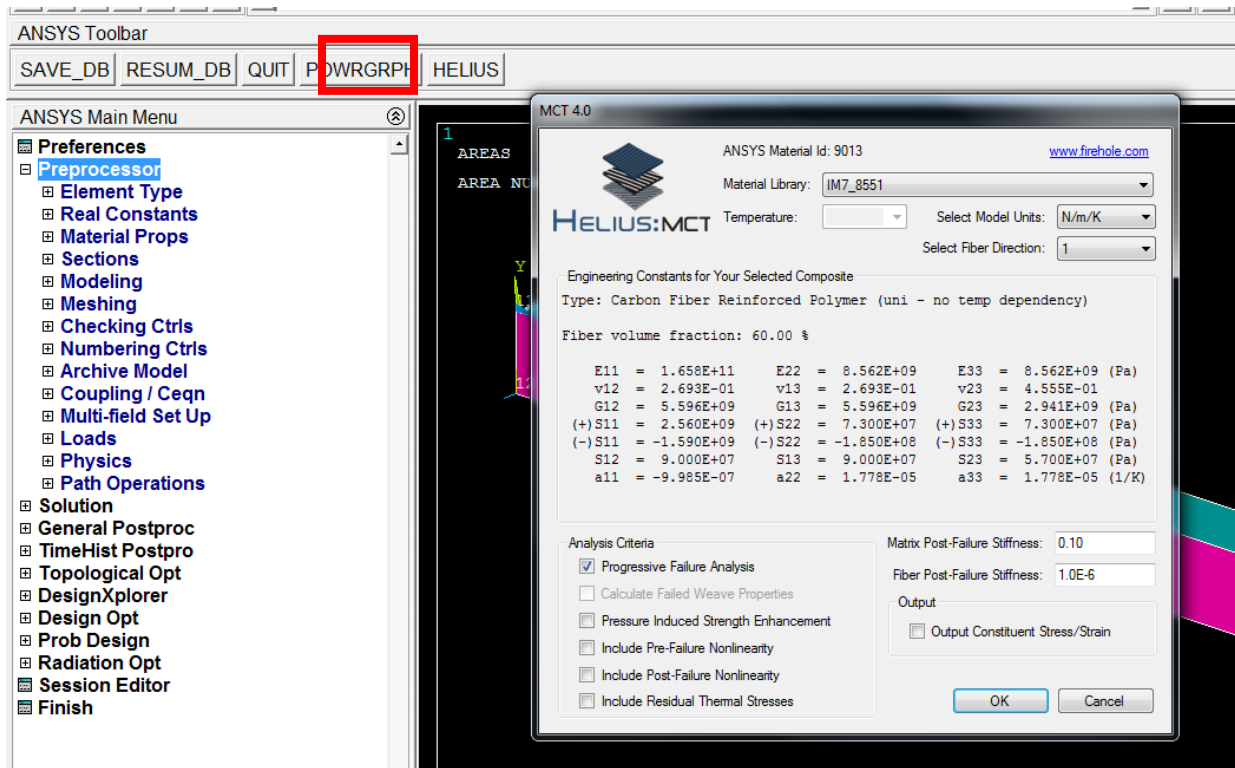
The screenshot shows the Helius Material Manager window for material IM7\_8551. The interface includes a menu bar (File, Help) and several input fields for material properties. The 'Material Name' is IM7\_8551, 'Choose Units' is lb/in/R, and 'Material Type' is Carbon/Epoxy. The 'Fib. Vol. Frac.' is 0.60. The 'Density' is in lbm/in³. There are tabs for 'General', 'Constants', and 'Non-Linear'. The 'Constants' tab is active, showing three sections of elastic constants: Lamina, Fiber, and Matrix. Each section has input fields for E<sub>11</sub>, E<sub>22</sub>, E<sub>33</sub>, ν<sub>12</sub>, ν<sub>13</sub>, ν<sub>23</sub>, G<sub>12</sub>, G<sub>13</sub>, G<sub>23</sub>, α<sub>11</sub>, α<sub>22</sub>, and α<sub>33</sub>. A checkbox 'Keep matrix properties isotropic' is checked. A footer note says 'Use the File menu to save your material file.'

Lamina Elastic Constants			
E <sub>11</sub>	2.405E+07	E <sub>22</sub>	1.242E+06
E <sub>33</sub>	1.242E+06	ν <sub>12</sub>	2.693E-01
ν <sub>13</sub>	2.693E-01	ν <sub>23</sub>	4.555E-01
G <sub>12</sub>	8.116E+05	G <sub>13</sub>	8.116E+05
G <sub>23</sub>	4.266E+05	α <sub>11</sub>	-5.547E-07
α <sub>22</sub>	9.878E-06	α <sub>33</sub>	9.878E-06
1/Δ R			

Fiber Elastic Constants			
E <sub>11</sub>	3.989E+07	E <sub>22</sub>	1.973E+06
E <sub>33</sub>	1.973E+06	ν <sub>12</sub>	2.000E-01
ν <sub>13</sub>	2.000E-01	ν <sub>23</sub>	3.000E-01
G <sub>12</sub>	1.378E+07	G <sub>13</sub>	1.378E+07
G <sub>23</sub>	7.587E+05	α <sub>11</sub>	-8.650E-07
α <sub>22</sub>	6.807E-06	α <sub>33</sub>	6.807E-06
1/Δ R			

Matrix Elastic Constants			
E <sub>11</sub>	5.918E+05	E <sub>22</sub>	5.918E+05
E <sub>33</sub>	5.918E+05	ν <sub>12</sub>	3.800E-01
ν <sub>13</sub>	3.800E-01	ν <sub>23</sub>	3.800E-01
G <sub>12</sub>	2.144E+05	G <sub>13</sub>	2.144E+05
G <sub>23</sub>	2.144E+05	α <sub>11</sub>	2.934E-05
α <sub>22</sub>	2.934E-05	α <sub>33</sub>	2.934E-05
1/Δ R			

Once a Helius:MCT ready material has been created using Helius Material Manager, the material is brought into the ANSYS Mechanical environment by using the HELIUS Macro that is created and placed in the ANSYS Toolbar during install.



There are a few options that are important for conducting a Helius:MCT progressive failure analysis that must be input using the HELIUS macro:

- Matrix Post-Failure Stiffness – A default value of 0.1 is provided and this value indicates how much of a fraction to reduce the matrix material constants ( $E_{11}$ ,  $G_{12}$ , etc.) to after a failure has been determined in the matrix material. For example, if the default value of 0.1 is used, a matrix  $E_{11}$  equal to 10% of the pristine matrix  $E_{11}$  will be used as the stiffness value during continued loading after failure is determined in the matrix material.
- Fiber Post-Failure Stiffness – Similar to the matrix PFS, a default value of 1E-06 is provided and this value indicates how much of a fraction to reduce the fiber material constants to after a failure has been determined in the fiber material.

Once the “OK” button is clicked at the bottom of the HELIUS macro, a new material is created inside of the ANSYS environment and is assigned the Material ID specified at the top of the macro window. From this point forward, the material is ready to be used throughout the rest of the model building process in the exact same manner that a material created the conventional way would be used.

## Analysis Execution

### Conventional Method:

The conventional way to go about predicting failure in a composite part is to run what is typically referred to as a “linear static analysis”. What this means is a load is applied to a composite structure all at once and the stresses and strains that result are computed. The analyst can look at these stresses and strains and make a determination if they suspect failure or they can use various failure criteria that take the composite ply stresses and/or strains and provide a failure index. Typically a failure index value greater than or equal to 1.0 means that failure has occurred in a composite ply based upon the particular failure criterion used.

### Helius:MCT Method:

When conducting a progressive failure analysis, one of the main points to remember is that an analysis needs to be executed that allows the structure to redistribute load around failed regions so that an entire cascade of failure from initiation until ultimate structural failure can be simulated. To do this, a linear analysis that applies the entire load at once won't take advantage of the load redistribution aspect. What is typically referred to as a “non-linear static analysis” is now required. A non-linear analysis applies the load gradually over a series of discrete sub-steps. This allows failure to occur at a stress concentration region and then gives the structure the chance to degrade a failed region and distribute a higher percentage of the load into the adjacent, un-failed material. By applying the load gradually, the structure can react, redistribute load, and continue to accumulate a higher load level which in turn will fail additional regions of the structure until there is no load carrying capability left.

There are only a few changes that need to be made to an ANSYS model to allow for a non-linear analysis to occur. The following commands would be entered into the command prompt:

- `NROPT,FULL,,OFF` This command instructs ANSYS to use the ‘Full’ Newton Raphson algorithm and prevents ANSYS from using the ‘Adaptive Decent’ algorithm. This will allow the Helius:MCT code to accurately and efficiently redistribute load during a progressive failure analysis.
- `PRED,OFF,,OFF` This command prevents ANSYS from using the converged solution at the last sub-step to estimate the solution for the current sub-step. This will allow the Helius:MCT code to accurately and efficiently redistribute load during a progressive failure analysis.
- `NSUBST,NSBSTP,NSBMX,NSBMN` This command is used to specify the number of sub-steps to be used in the analysis. For example, to break the analysis down into 100 sub-steps and force ANSYS to make each load sub-step equivalent, `NSUST,100,100,100` would be used.
- `NEQIT,NEQIT` This command specifies the number of equilibrium iterations that must be performed before ANSYS evaluates the need to reduce the sub-step size. This number should be intentionally large, e.g. 1000, so ANSYS is forced to find an equilibrium solution for each sub-step.
- `TIME,1` This specifies a step size equal to 1. A value other than 1 can be used but it is useful to use 1 in a non-linear analysis so that when the user sees that the sub-step they are examining has a time of 0.73, that 73% of the load has been applied.
- `CNVTOL,F,,,0` This command is used to define the convergence tolerance for residual node forces. This will allow the Helius:MCT code to accurately and efficiently redistribute load during a progressive failure analysis.

## Results

### Conventional Method:

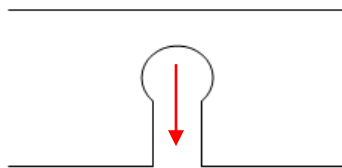
The analyst can view stress and strain contours to examine how the load is distributed through the composite part. If a failure criterion is used, a failure index contour plot can be viewed to identify “hot spots” or regions where failure is likely to initiate.

### Helius:MCT Method:

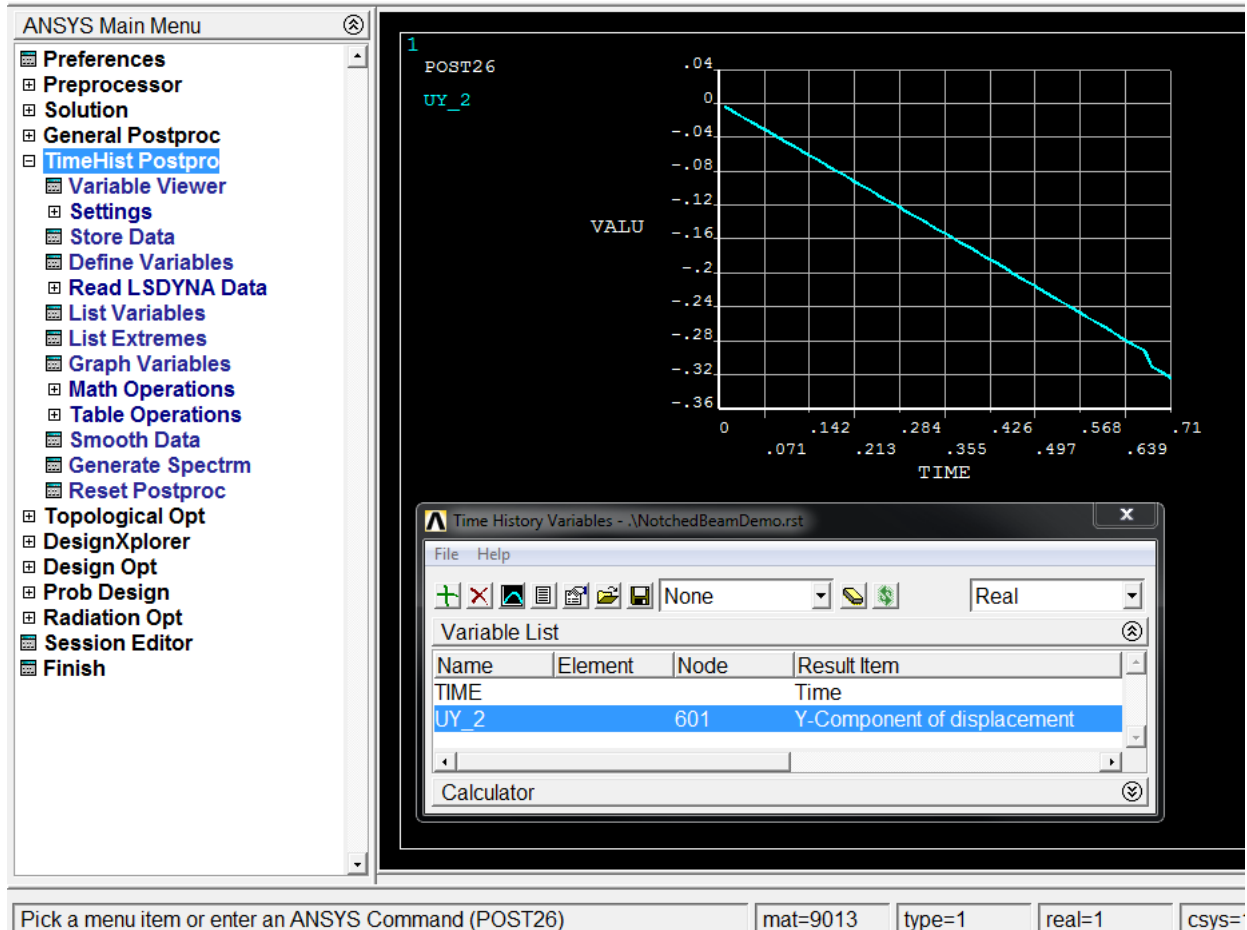
There are two methods for viewing results that are critical for forming a complete opinion about the ultimate failure of a composite structure:

#### *Load-Displacement Curve*

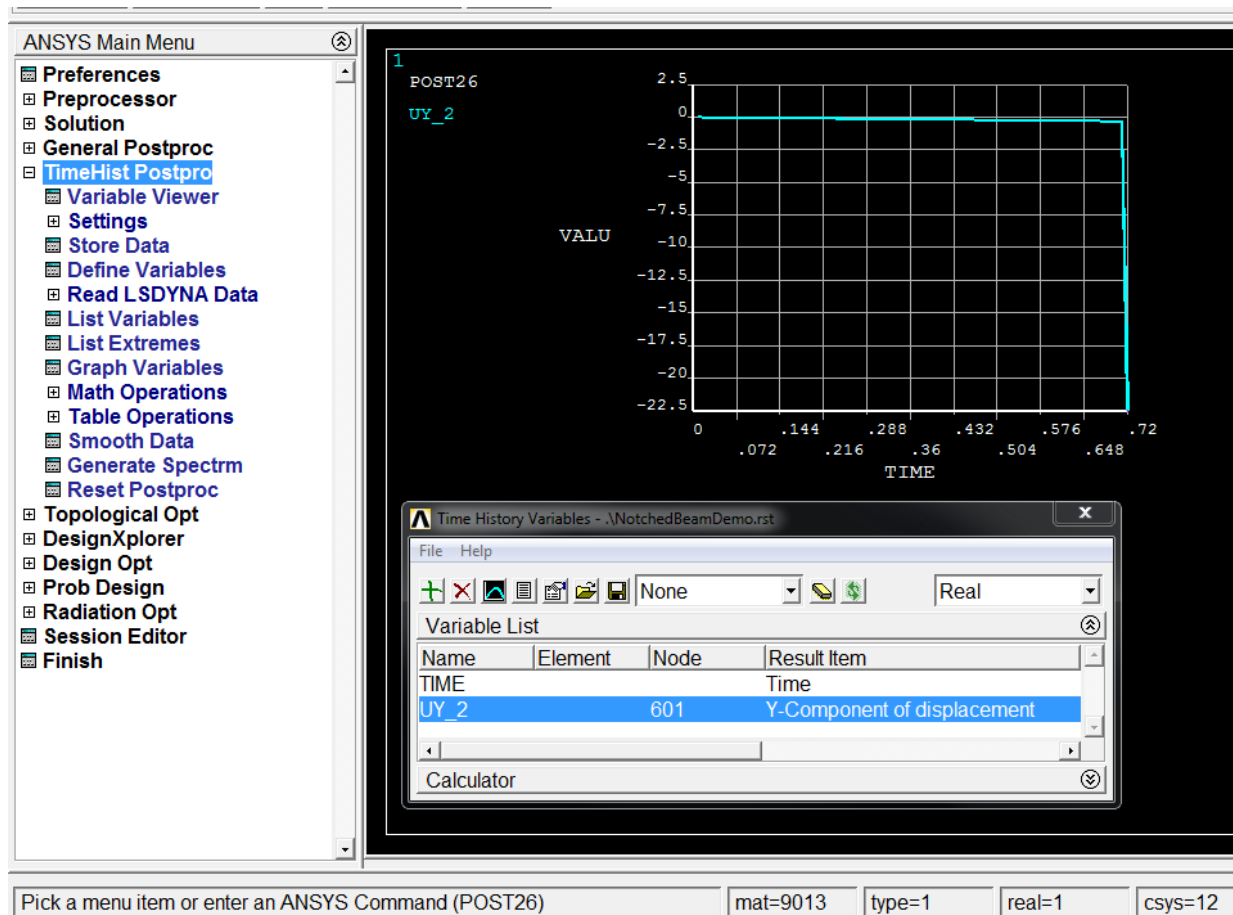
In our example problem, a uniform pressure distribution is being applied to the top, horizontal flange of the L-Beam. 360 lbs. is being distributed over ( $36'' \times 3'' = 108 \text{ in.}^2$ ) for a pressure load of 3.3 psi. To gain an understanding of the ultimate load carrying capability of the L-Beam, we can run a non-linear analysis using the techniques described in the previous section, and using the Helius:MCT progressive failure software, determine when the failure accumulation leads to an ultimate failure. To pinpoint an ultimate failure in the composite beam requires the use of a Load-Displacement curve. We know the load that is being applied to the structure, so we just need to choose an appropriate point (node) in which to track displacement to provide an understanding of the response of the structure. An appropriate point for our example would be a node along the center plane of the beam, as we would expect this point to deflect the most. Using the Time History Postprocessor (POST26) command in ANSYS, we can plot the vertical displacement of a center plane node.



Plotting the displacement of a node on the center plane up to a time of 0.71 (remember we are doing a non-linear analysis so this mean 71% of the load - 256 lbs. - has been applied) shows that a small jump in displacement occurs around 0.64 (230 lbs.) when the node has moved about 0.3 in. downward. This could indicate that a small failure cascade has begun at the tip of the notch in the beam but because there isn't a large discontinuity, ultimate failure has not occurred yet.



If we plot the response out to a time value of 0.72, we see the scale of the plot has dramatically shifted and we see a huge displacement jump to a value of about 22.5 in. This large displacement jump is a great indication of ultimate failure of the beam.

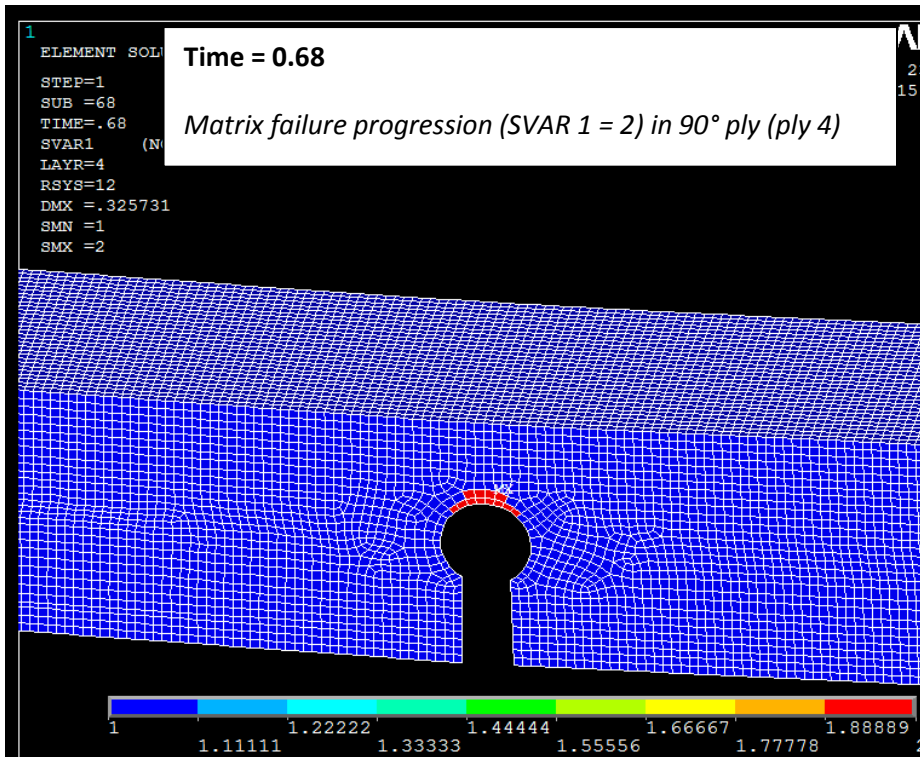
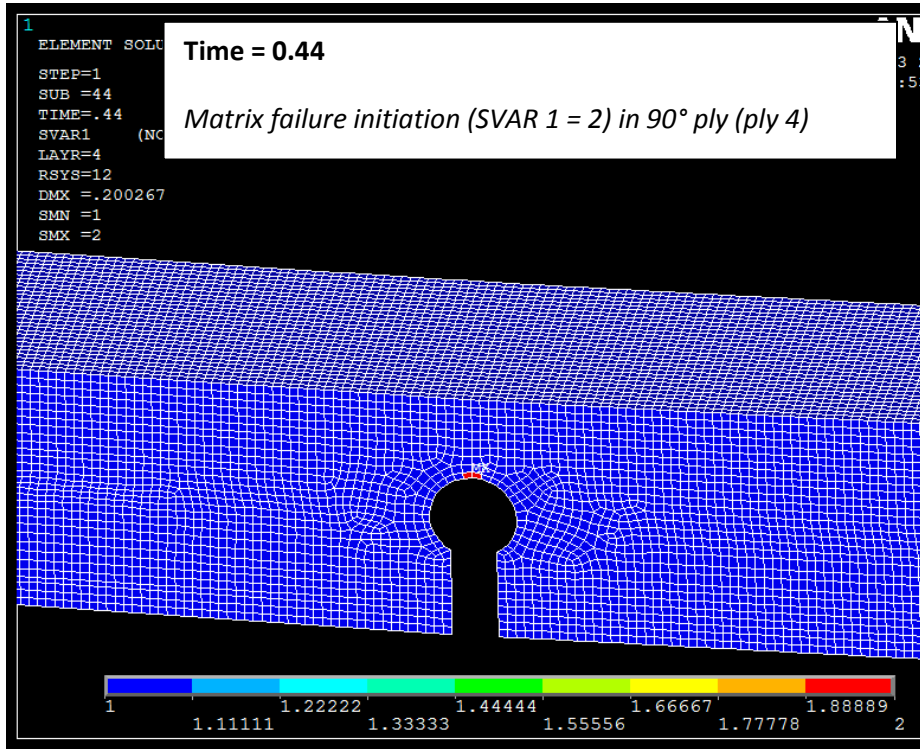


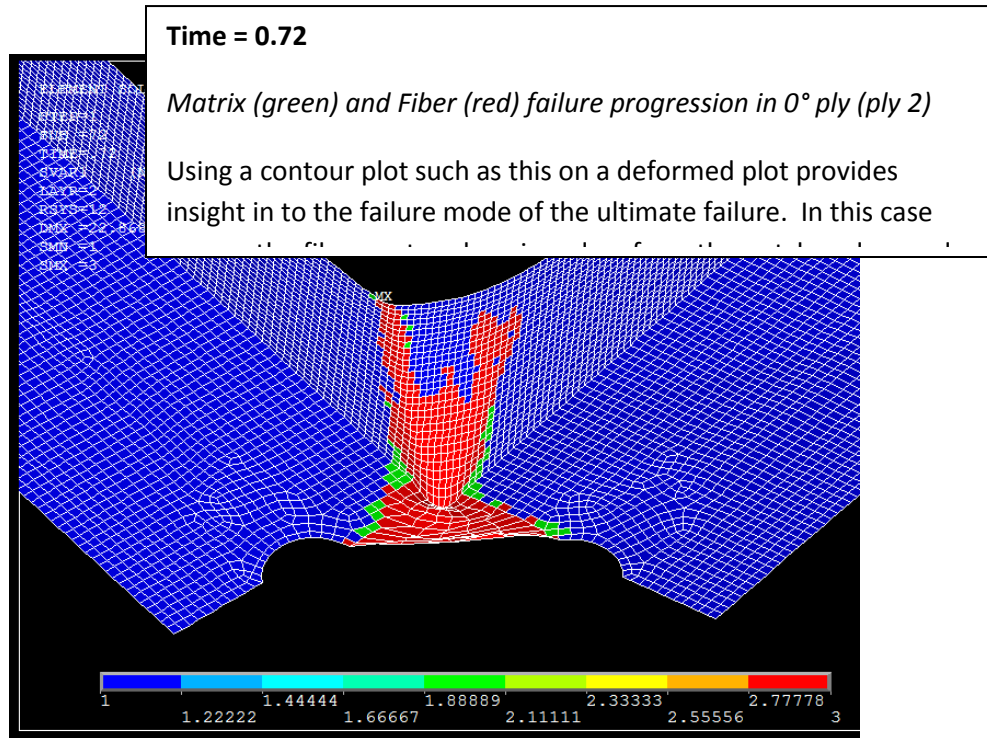
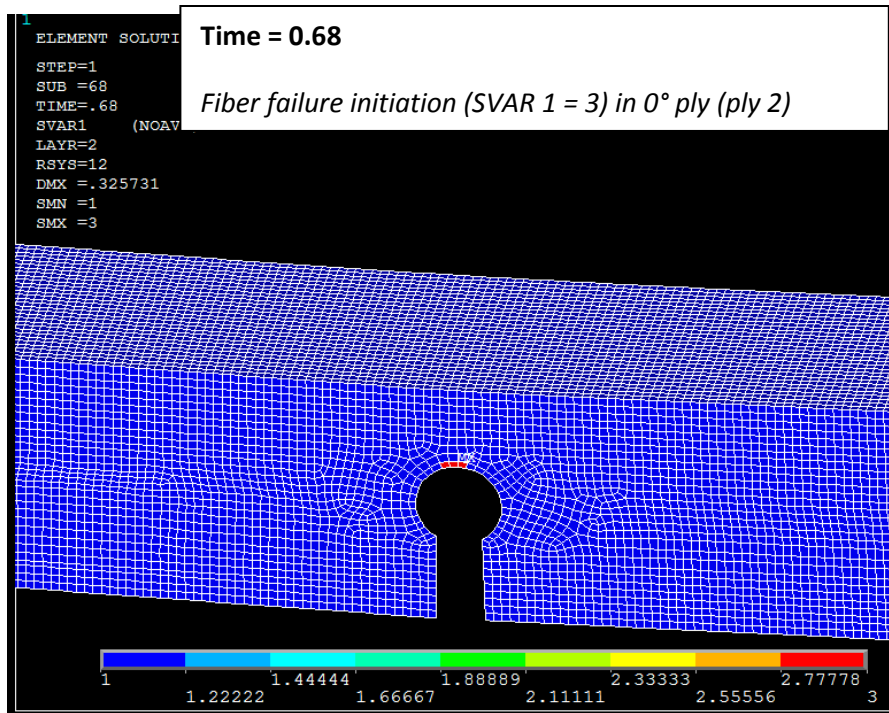
### Contour Plot of Helius:MCT Output Variables

The second method of viewing results for a Helius:MCT progressive failure analysis is to view a contour plot of the unique Helius:MCT output variables. The most important variables to look at are:

- SVAR1 – This variable indicates the level of damage at an integration point in an element. SVAR1 can assume 3 values:
  - 1 – No failure
  - 2 – Matrix failure
  - 3 - Fiber failure
- SVAR2 – This variable is a value between 0 and 1 that indicates the matrix failure index (1.0 indicates the matrix failure criterion has been achieved)
- SVAR 3 – This variable is a value between 0 and 1 that indicates the fiber failure index (1.0 indicates the fiber failure criterion has been achieved)

We can step through the solution and using the SVAR1 variable, view the initiation and progression of failure through different composite layers:





Using Helius:MCT, the actual failure of composite structures can be simulated with accuracy, taking the finite element analysis of composite structures to the next level. With very few modifications to the model building process, a wealth of information can be gained about a composite part and cost savings can be realized early on in the design process.