

HELIUS:MCT

ADVANCED COMPOSITES SIMULATION | FIREHOLE COMPOSITES

Tutorial 1

Helius:MCT™ Version 4.0 for ANSYS
February 1, 2011

Abstract

This document provides a step-by-step tutorial that demonstrates the use of Helius:MCT. The primary emphasis is the creation of ANSYS models that are compatible with Helius:MCT and the viewing of special solution variables that are computed by Helius:MCT. Tutorial 1 demonstrates the use of ANSYS Mechanical APDL in building an ANSYS finite element model.

For questions, comments or further information, contact Firehole Composites at support@firehole.com.

Legal Notices

Copyright 2011, Firehole Technologies, Inc.

Helius:MCT is a trademark of Firehole Technologies, Inc. Any use of the Helius:MCT trademark requires the prior written consent of Firehole Technologies, Inc.

ANSYS Mechanical is a trademark of ANSYS, Inc.

Table of Contents

1	Introduction	3
2	Tutorial Steps	4
2.1	Creating the part.....	4
2.2	Define an element type	6
2.3	Characterizing a user material with Helius Material Manager	6
2.4	Creating a user material with Helius:MCT	8
2.5	Define and assign the element coordinate system.....	10
2.6	Mesh the part	11
2.7	Defining a composite layup	13
2.8	Applying boundary conditions	14
2.9	Defining the load	15
2.10	Specify solution controls.....	16
2.11	Defining solution output to the results file.....	17
2.12	Solve the model	17
2.13	Viewing and interpreting the results	17
2.13.1	<i>Generating a load-displacement plot</i>	<i>20</i>
3	Summary.....	22
4	References.....	22

Table of Figures

Figure 1.	Dimensions of composite plate.....	3
Figure 2.	Ansys-MCT-Launcher interface.....	4
Figure 3.	Plate geometry.....	5
Figure 4.	Completed General tab.....	8
Figure 5.	Completed Constants tab.....	8
Figure 6.	Helius:MCT GUI.....	10
Figure 7.	Local coordinate system.....	11
Figure 8.	Areas to concatenate for 1 of the volumes that surround the hole.....	12
Figure 9.	Through-thickness lines picked for manual seeding.....	12
Figure 10.	Plate mesh.....	13
Figure 11.	Shell section dialog box.....	14
Figure 12.	Boundary conditions.....	15
Figure 13.	Coupling constraint on top surface of plate.....	16
Figure 14.	PLESOL contour plot of ply 1 at the end of the step.....	18
Figure 15.	PLESOL contour plot of ply 2 at the end of the step.....	19
Figure 16.	Progression of failure in ply-2 (0°).....	20
Figure 17.	Time History Variables GUI.....	21
Figure 18.	Load-displacement plot.....	21

Helius:MCT Tutorial 1

1 Introduction

The following tutorial provides step-by-step instructions to create and analyze a composite plate using the Helius:MCT GUI in ANSYS Mechanical™. The model is based on coupon level test results from the National Institute for Aviation Research (NIAR). Specifically, an open-hole tension (OHT) plate made from T700/2510 is built and compared to the experimental results¹ determined by NIAR. The plate layup is $[45/0/-45/90]_{3S}$ and the plate dimensions are given in Figure 1.

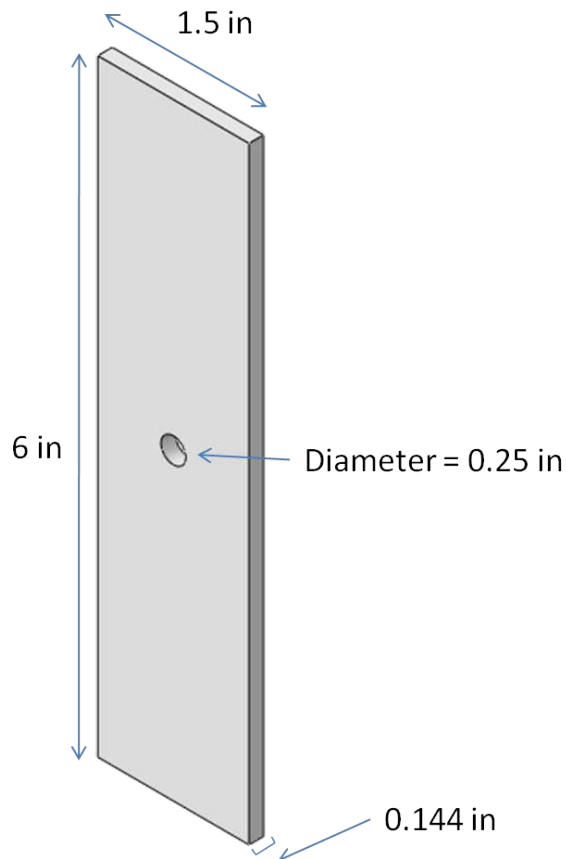


Figure 1. Dimensions of composite plate.

If one is interested only in learning how to view results generated by Helius:MCT, an Ansys input file (HeliusMCT_Tutorial_1_Ansys.ans) is available for download from www.firehole.com that can be used to generate an Ansys results (*.rst) file. In this case, follow the steps in section 2.3, then run the input file and refer to section 2.13. Before submitting the file for analysis, ensure that the matID values in the input file match the matID for Tutorial_1 (the name assigned to the material created in this tutorial) on your workstation. The matID for Tutorial_1 can be determined by selecting Tutorial_1 in the Helius GUI within ANSYS as detailed in section 2.4.

2 Tutorial Steps

In the following tutorial, elementary modeling details are omitted as it is assumed that the user has previous experience in the ANSYS Mechanical APDL environment. Please refer to the ANSYS documentation before completing this tutorial if you are unfamiliar with the ANSYS GUI.

2.1 Creating the part

Defining the part geometry is generally the first step in the development of a finite element model. Here, the plate geometry is defined to generate a 3-dimensional part.

1. Open ANSYS Mechanical APDL using the **ansys-MCT-launcher** tool (Figure 2).
 - a. This tool is installed with Helius:MCT in the Firehole directory on your computer and starts the ANSYS Launcher with the necessary environment variables to run Helius:MCT with ANSYS.
2. Click the **Run** button in the ANSYS Mechanical APDL Product Launcher. The ANSYS GUI will appear.

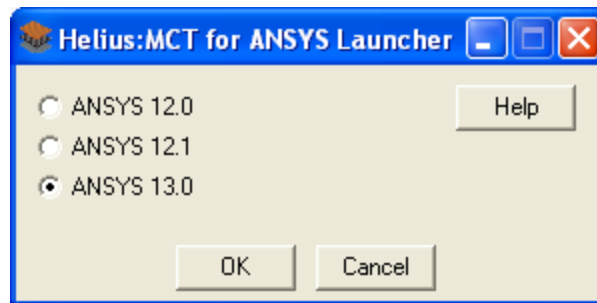


Figure 2. Ansys-MCT-Launcher interface.

3. In order to generate a uniform, mapped mesh, the plate will be divided into 8 volumes.
4. Click **Main Menu → Preprocessor → Modeling → Create → Volumes → Block → By 2 Corners & Z**.
5. In the dialog box that appears, enter the following values and click the **Apply** button:
 - a. **WP X = 0**
 - b. **WP Y = 0**
 - c. **Width = 0.75**
 - d. **Height = 2.25**
 - e. **Depth = 0.144**
6. Repeat step 4 entering the following values for WP X, WP Y, Width, Height, and Depth each time:
 - a. **0.75, 0, 0.75, 2.25, 0.144**

- b. **0, 2.25, 0.75, 0.75, 0.144**
 - c. **0.75, 2.25, 0.75, 0.75, 0.144**
 - d. **0, 3, 0.75, 0.75, 0.144**
 - e. **0.75, 3, 0.75, 0.75, 0.144**
 - f. **0, 3.75, 0.75, 2.25, 0.144**
 - g. **0.75, 3.75, 0.75, 2.25, 0.144**
7. Click **Main Menu** → **Preprocessor** → **Modeling** → **Create** → **Volumes** → **Cylinder** → **Solid Cylinder**.
8. In the dialog box that appears, enter the following values then click **OK**:
- a. **WP X = 0.75**
 - b. **WP Y = 3.0**
 - c. **Radius = 0.125**
 - d. **Depth = 0.144**
9. Click **Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Subtract** → **Volumes**.
10. Select the 8 blocks that were created in steps 4 and 5 and click **OK**. When prompted to select the volumes to be subtracted, pick the cylinder and click **OK**. The plate should appear as shown in Figure 3.
11. To complete the geometry, the 8 volumes must be merged. Click **Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Glue** → **Volumes**. Select the 8 volumes and click **OK**.

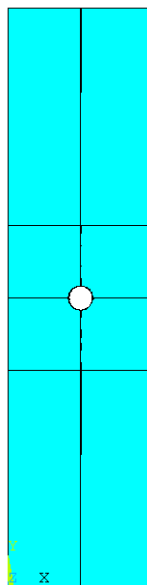


Figure 3. Plate geometry.

2.2 Define an element type

There are several ANSYS elements that have layered functionality. For this tutorial, element type SOLSH190 is used. SOLSH190 is a 3-dimensional, 8-node layered solid shell. This element is commonly used to model plate-like structures.

1. Click **Main Menu** → **Preprocessor** → **Element Type** → **Add/Edit/Delete**.
2. In the dialog box that appears, click **Add**.
3. Select **Solid-Shell** in the Library of Element Types dialog box. Click **OK**.
4. Click **Close** in the Element Types dialog box.

In order to store top and bottom section point data for all layers within each element, KEYOPT(8) is used.

5. Type **KEYOPT,1,8,1** into the command prompt and press **Enter**.

2.3 Characterizing a user material with Helius Material Manager

A unique material characterization must be performed for each material that will be used in a Helius:MCT analysis. A stand-alone graphical-user-interface program called Helius Material Manager is installed with Helius:MCT and is used to characterize composite materials for use with Helius:MCT. In this section, the steps to create a new composite material using Material Manager are described.

The properties² for the material used in this Tutorial, T700/2510, are given in Table 1. We will be comparing OHT data at 75°F so it is appropriate to use the RTD properties. Further, since the plate is loaded in tension, it is appropriate to use the tensile moduli for the characterization of the material.

It is common for datasheets to be missing a few of the properties required by Helius:MCT. In this case, our source of data is missing the interlaminar stiffness (ν_{23} or G_{23}) and interlaminar strength (S_{23}). S_{23} is not required but should be included in analyses where significant interlaminar stresses are present. When properties are missing they should be estimated based on similar materials and engineering judgment. Common values^{3,4} for ν_{23} and S_{23} for carbon/epoxy unidirectional materials are 0.4 and 7.25 ksi.

Since the Helius:MCT algorithm decomposes the lamina-level stresses into fiber and matrix stresses, the fiber and matrix stiffnesses must also be provided. Accurate fiber and matrix properties are not as important as accurate lamina properties because they are iteratively tuned to match the specified lamina properties and volume fraction. Ideally, properties for the fiber and matrix would be known and entered. However, it is most often the case that the properties are not known and default properties should be entered. Material Manager comes pre-populated with default fiber and matrix values for carbon/epoxy and glass/epoxy materials.

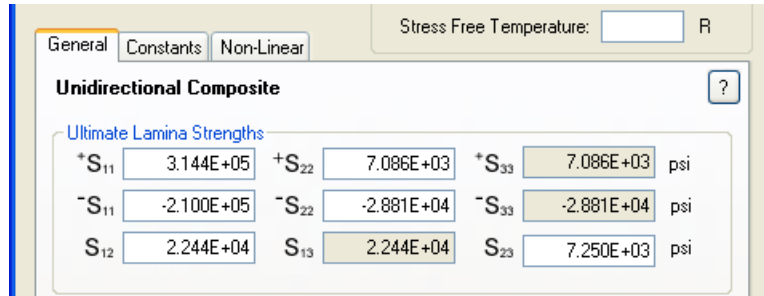
Table 1. T700/2510 lamina properties.

		CTD		RTD		ETD		ETW	
		B-Basis	Mean	B-Basis	Mean	B-Basis	Mean	B-Basis	Mean
F_1^{tu}	(ksi)	210.579 (216.021)	240.912 (243.955)	272.894 (277.268)	314.387 (315.086)	276.967 (281.634)	319.079 (320.048)	282.985 (288.420)	326.012 (327.759)
E_1^t	(Msi)	---	18.422 (18.529)	---	18.104 (18.209)	---	17.710 (17.812)	---	17.636 (17.745)
ν_{12}^{tu}		---	0.350	---	0.309	---	0.309	---	0.323
F_2^{tu}	(ksi)	6.389	7.683	6.207	7.086	5.488	6.415	3.291	3.757
E_2^t	(Msi)	---	1.313	---	1.219	---	1.083	---	0.920
F_1^{cu}	(ksi)	174.601 (175.305)	202.546 (202.546)	184.914 (185.786)	209.999 (210.272)	179.882 (180.730)	204.285 (204.549)	156.723 (154.388)	177.255 (174.041)
E_1^c	(Msi)	---	16.618 (16.512)	---	16.347 (16.284)	---	17.192 (17.188)	---	16.981 (16.916)
F_2^{cu}	(ksi)	36.594	40.964	26.149	28.814	19.443	21.425	15.324	16.886
E_2^c	(Msi)	---	2.043	---	1.471	---	1.231	---	1.153
F_{12}^{su}	(ksi)	21.546	23.136	21.106	22.443	17.501	18.610	12.988	13.811
G_{12}^s	(Msi)	---	0.757	---	0.613	---	0.509	---	0.453
F_{13}^{su**}	(ksi)	---	---	11.142	12.489	---	---	---	---

** *Apparent* interlaminar shear strength

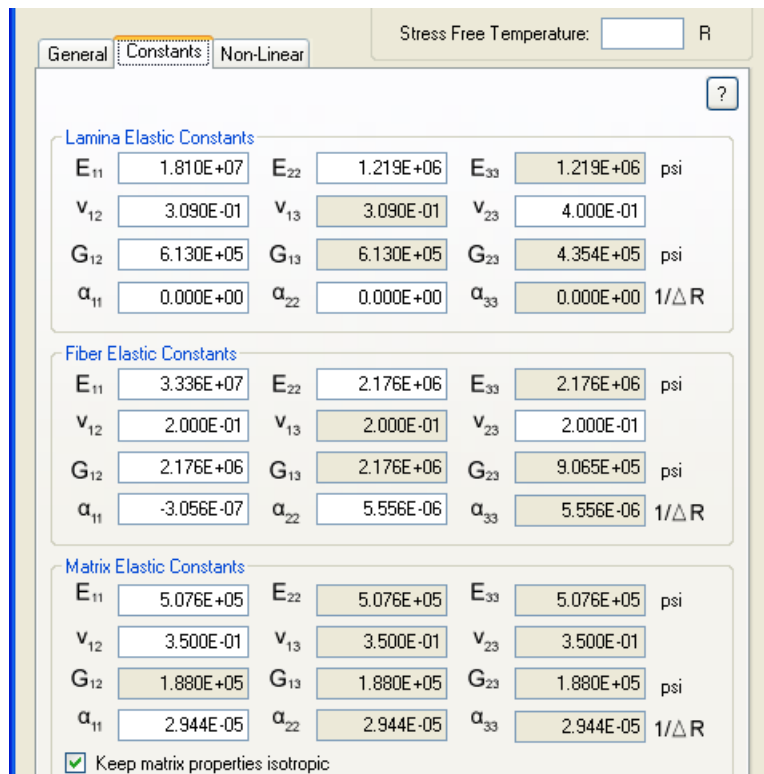
1. Open the Helius Material Manager GUI (accessible from either the Firehole directory or the Windows Start menu) and select **File → New → Unidirectional**.
2. Enter the material name as **Tutorial_1**.
3. Select **lb/in/R** as the unit system, **Carbon/Epoxy** as the Material Type, and enter the fiber volume fraction, **0.55**.
4. Fill out the **General tab** by entering the RTD mean lamina strengths given in Table 1. It is not necessary to enter the compressive strengths with a “-“ sign. The GUI should appear as shown in Figure 4.
5. Fill out the **Constants tab** by entering the RTD mean tensile lamina stiffnesses given in Table 1. Click the **Apply Material Type Characteristics** button to apply the default carbon/epoxy fiber and matrix stiffnesses. The GUI should appear as shown in Figure 5.
6. Select **File → Save As** and browse to the location of the Firehole Materials directory. Unless you selected an alternate directory during installation, this directory is located at *C:\Firehole\Materials*.

- After the material is saved, you may open the material file using Material Manager and review the converged material properties.



Property	Value	Unit
+S ₁₁	3.144E+05	psi
+S ₂₂	7.086E+03	psi
+S ₃₃	7.086E+03	psi
-S ₁₁	-2.100E+05	psi
-S ₂₂	-2.881E+04	psi
-S ₃₃	-2.881E+04	psi
S ₁₂	2.244E+04	psi
S ₁₃	2.244E+04	psi
S ₂₃	7.250E+03	psi

Figure 4. Completed General tab.



Property	Value	Unit
E ₁₁	1.810E+07	psi
E ₂₂	1.219E+06	psi
E ₃₃	1.219E+06	psi
ν ₁₂	3.090E-01	
ν ₁₃	3.090E-01	
ν ₂₃	4.000E-01	
G ₁₂	6.130E+05	psi
G ₁₃	6.130E+05	psi
G ₂₃	4.354E+05	psi
α ₁₁	0.000E+00	1/Δ R
α ₂₂	0.000E+00	1/Δ R
α ₃₃	0.000E+00	1/Δ R

Property	Value	Unit
E ₁₁	3.336E+07	psi
E ₂₂	2.176E+06	psi
E ₃₃	2.176E+06	psi
ν ₁₂	2.000E-01	
ν ₁₃	2.000E-01	
ν ₂₃	2.000E-01	
G ₁₂	2.176E+06	psi
G ₁₃	2.176E+06	psi
G ₂₃	9.065E+05	psi
α ₁₁	-3.056E-07	1/Δ R
α ₂₂	5.556E-06	1/Δ R
α ₃₃	5.556E-06	1/Δ R

Property	Value	Unit
E ₁₁	5.076E+05	psi
E ₂₂	5.076E+05	psi
E ₃₃	5.076E+05	psi
ν ₁₂	3.500E-01	
ν ₁₃	3.500E-01	
ν ₂₃	3.500E-01	
G ₁₂	1.880E+05	psi
G ₁₃	1.880E+05	psi
G ₂₃	1.880E+05	psi
α ₁₁	2.944E-05	1/Δ R
α ₂₂	2.944E-05	1/Δ R
α ₃₃	2.944E-05	1/Δ R

Keep matrix properties isotropic

Figure 5. Completed Constants tab.

2.4 Creating a user material with Helius:MCT

The Helius:MCT GUI is the central interface between the user and Helius:MCT. It allows the user to choose from a variety of material and analysis options including the following:

- Selection of a composite material to use in the analysis (Helius:MCT compatible material definitions must be created outside of ANSYS using the Helius Material Manager.)

- The choice of four unit systems
- Selection of the principle material direction
- Inclusion of temperature dependence
- Inclusion of progressive failure in the analysis
- Method of calculation of the failed plain weave properties
- Inclusion of hydrostatic pressure induced material strength enhancement
- Inclusion of pre failure and/or post failure nonlinearity
- Accounting for thermal residual stresses caused by cooling of the composite during the cure cycle
- Setting the matrix and fiber constituent stiffness post-failure degradation ratios
- Creating additional output variables that provide fiber and matrix constituent stresses and strains

Using these options, the user can tailor his or her analysis to the requirements of the problem. For a detailed discussion of the options available, refer to Section 3.1 of the Helius:MCT User's Manual.

In the following steps, a user-material is created for the plate, and progressive failure analysis is requested using the Helius:MCT GUI.

1. Click on the **HELIUS** button from the ANSYS Toolbar. The MCT 4.0 GUI appears.

Note that the HELIUS button only works in the model creation preprocessor (/PREP7).

2. From the Material Library list, select the material that you created in the previous section, Tutorial_1.

The unit dependent Engineering Constants specific to this material are listed in the dialog box for the user to review. Note that the ANSYS Material ID (matID) that corresponds to this material is also shown in the dialog box. This value may vary from computer to computer. It is useful to know the Material ID when working with input files since ANSYS uses the matID to identify Helius materials.

3. Since this model uses inches and pounds as base units, select **lb/in/R** from the Select Model Units list.

There are 4 unit systems to choose from. The default unit system is N/m/K.

4. Select **1** as the fiber direction.

2 can also be used as the fiber direction, but it would require a different composite layup orientation than the 1 direction. As a general rule, it is recommended that 1 be used as the fiber direction to maintain consistency from model to model. On occasion, however, it will not be possible to create an orientation in ANSYS that allows for the 1 direction to be the fiber direction due to the combination of complex model geometry and orientation limitations. In such cases, it may be necessary to use the 2 direction as the fiber direction.

5. Select Progressive Failure Analysis, the foundation component of Helius:MCT.
6. Set the Matrix Post-Failure Stiffness value to **0.10** and the Fiber Post-Failure Stiffness value to **0.01**. The dialog box should appear as shown in Figure 6.

These values specify the ratio of damaged to undamaged material properties for the material. For example, a value of 0.01 for the Fiber Post-Failure Stiffness means that when an integration point undergoes a fiber failure event, the material properties at that integration point will degrade to 1% of the undamaged properties.

7. Click **OK**.

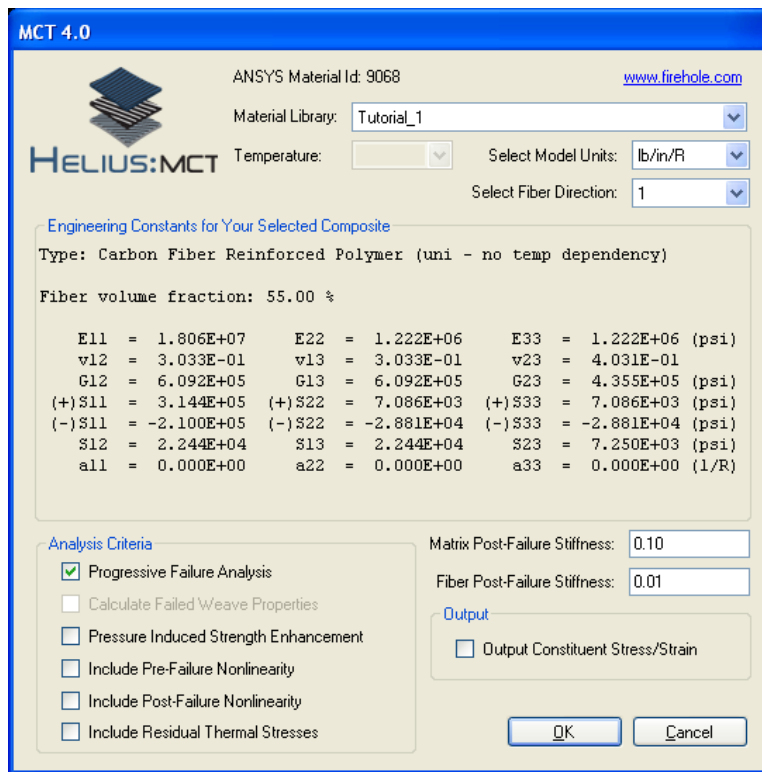


Figure 6. Helius:MCT GUI.

2.5 Define and assign the element coordinate system

In order to align the 0 degree plies with the loading direction, a local coordinate system needs to be defined and assigned to the elements.

1. To define a local, rectangular coordinate system with the x-axis aligned with the global loading direction (the global y-axis), enter the following into the command prompt:

LOCAL,11,CART,0,0,0,90,0,0

- To assign the local coordinate system to the elements, enter **ESYS,11** into the command prompt. Figure 7 shows the local coordinate system relative to the base of the part.



Figure 7. Local coordinate system.

2.6 Mesh the part

In meshing the composite plate, a global seed size of 0.075 is used to ensure that the mesh is fine enough to capture the location and propagation of failure, but also coarse enough to allow for a moderate solution time. Manual edge seeding is employed to ensure that there is a single element in the thickness direction and the area concatenation tool is used to allow for a mapped mesh around the hole.

- From the menu at the top of the GUI, click **Select → Entities...**. In the dialog box that appears, select **Volumes** from the drop-down list and select the **Reselect** radio button. Click **OK**.
- Select the 4 volumes that surround the hole and click **OK**.
- From the top menu, click **Plot → Replot**. The 4 volumes that surround the hole should be the only volumes displayed.

To use the mapped mesh algorithm, the volume must be defined by 6 areas. Since each of the 4 volumes that surround the hole is defined by 7 areas, two areas per volume must be concatenated.

- Click **Main Menu → Preprocessor → Meshing → Concatenate → Areas**.
- For *each* of the 4 volumes, select the two areas as shown in Figure 8 and click the **Apply** button.
 - Do not select all 8 areas and then click the Apply button. Simply select the two appropriate areas per volume and click the Apply button.

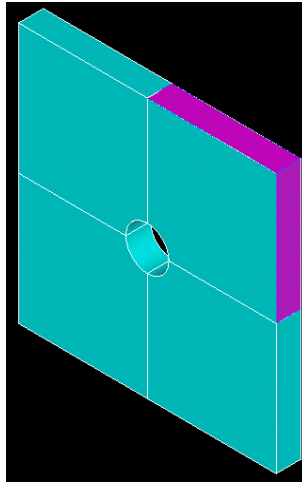


Figure 8. Areas to concatenate for 1 of the volumes that surround the hole.

Since a single element will be used to define the entire 24 ply layup, manual seeds must be specified on the lines that are parallel to the global Z-direction.

6. From the top menu, click **Select → Everything**. Then click **Plot → Replot**.
7. Click **Preprocessor → Meshing → Size Cntrls → ManualSize → Lines → Picked Lines**.
8. Select all of the lines that are parallel to the global Z-direction as shown in Figure 9. Click **OK**.
9. In the dialog box that appears, enter a value of **1** in the NDIV entry box. Click **OK**.

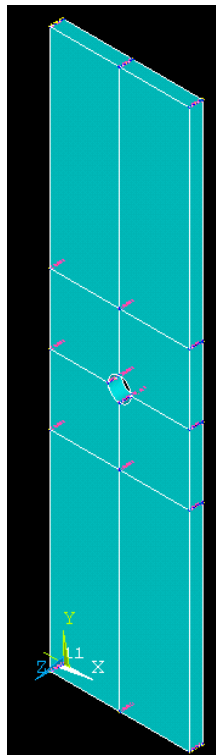


Figure 9. Through-thickness lines picked for manual seeding.

10. Click **Preprocessor** → **Meshing** → **MeshTool**.
11. In the dialog box that appears, click the **Set** button next to the Global option under Size Controls, enter **0.075** in the **SIZE** entry box, and click **OK**.
12. Select the **Hex** and **Mapped** radio buttons and click the **Mesh** button.
13. Select all 8 volumes from the Graphics Window and click **OK**. The part is now meshed and should be similar to the mesh shown in Figure 10.
14. Since a mapped mesh was used, the element normals may be inconsistent. To reorient the element normals, select **Preprocessor** → **Modeling** → **Move / Modify** → **Elements** → **Orient Normal**. Select **Positive z-axis** from the DIR list and click **OK**.

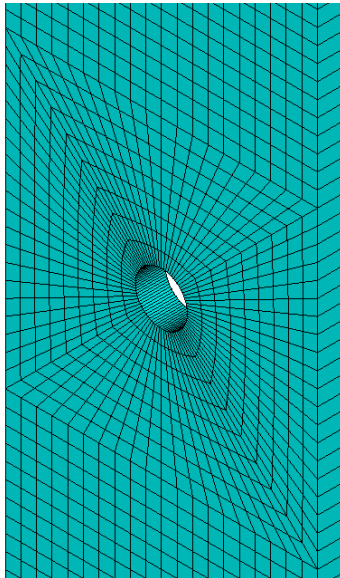


Figure 10. Plate mesh

2.7 Defining a composite layup

After the part and the material have been created, a composite layup section can be created. The composite layup editor is used to create plies and to assign materials and orientations to these plies. In this step, a composite layup that represents the plate layup is created and defined.

1. Select **Main Menu** → **Preprocessor** → **Sections** → **Shell** → **Lay-up** → **Add/Edit**.
2. In the GUI the dialog box that appears, click the **Add Layer** button until there are 12 layers.
3. Enter **0.006** for all thickness values.
4. Enter the orientations for plies 1 thru 12 as **45, 0, -45, 90, 45, 0, -45, 90, 45, 0, -45, and 90**, respectively.
5. Select **11** from the KCN or Node drop-down menu.

- Click **Tools** → **Add Symmetry** to get the full number of plies. The section editor should appear as shown in Figure 11. Note that the Material ID values may differ depending on the materials in your Material Library. Click **OK**.

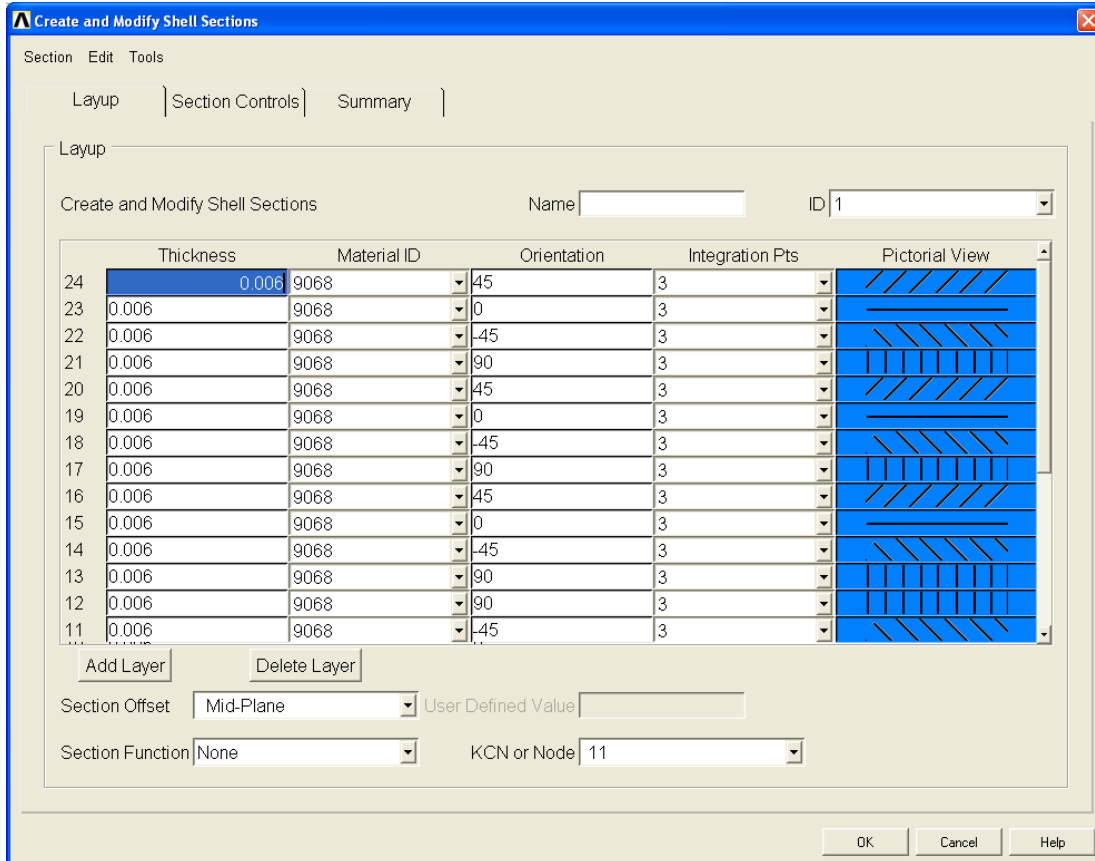


Figure 11. Shell section dialog box.

2.8 Applying boundary conditions

The following steps create boundary conditions that minimally constrain the movement of the bottom surface of the plate and prevent rigid body motion. The boundary conditions are shown in Figure 12.

- Select **Preprocessor** → **Loads** → **Define Loads** → **Apply** → **Structural** → **Displacement** → **On Areas**.
- From the Graphics Window, select the 2 bottom surfaces of the plate and click **OK** in the Apply U,ROT on Areas dialog box.
- In the dialog box that appears, enter a Displacement value of **0** for **UY** and click **OK**.
- Select **Preprocessor** → **Loads** → **Define Loads** → **Apply** → **Structural** → **Displacements** → **On Keypoints**.
- Select point “A” and set **UX** and **UZ** equal to **0**.

- Repeat steps 4 and 5, selecting point “B” and setting just **UX** to **0**.

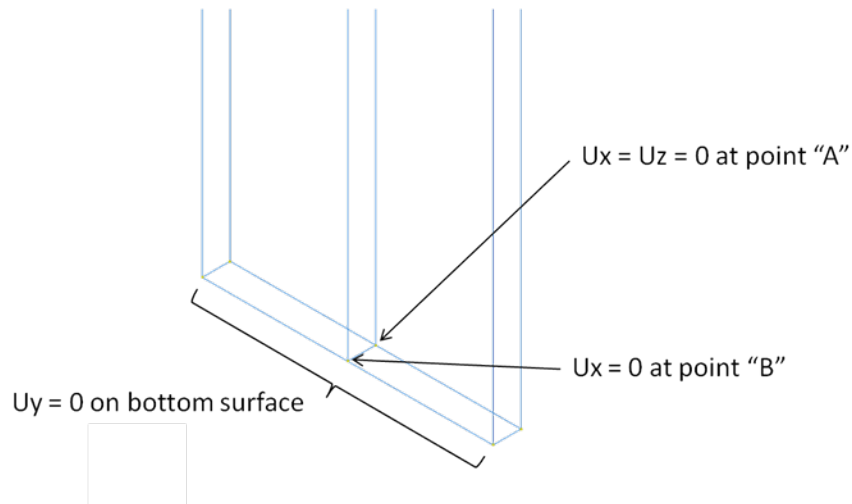


Figure 12. Boundary conditions.

2.9 Defining the load

A second boundary condition is created to impose a vertical displacement along the top surface of the plate. The plate is loaded by imposed displacements because it results in a much more gradual failure process than a comparable loading by applied forces. When a simple structure, such as this composite plate, begins to fail under the action of applied forces, the structure fails very rapidly because the load continues to increase as the load carrying capacity of the structure decreases. With displacement controlled loading, the load carried by the structure decreases as the structure fails which allows for a slower rate of failure.

First, a coupling constraint is applied to the nodes on the top surface that allow for a simple determination of the total reaction force during post-processing.

- Select **Preprocessor** → **Coupling** / **Ceqn** → **Cupl DOFs w/Mstr**.
- Pick all of the nodes on the top surface of the plate, making sure that the first node picked is the node indicated in Figure 13. Click **OK**.
- Set the reference number to **20** and the degree-of-freedom label to **UY**. Click **OK** to complete the coupling definition.

As a consequence of the coupling constraint, a displacement in the y-direction applied to the node highlighted in Figure 13 will result in the same y-displacement applied to the remaining nodes in the constraint definition.

- Select **Preprocessor** → **Loads** → **Define Loads** → **Apply** → **Structural** → **Displacement** → **On Nodes** and select the node indicated in Figure 13.
- Apply a **UY** value of **0.06** and click **OK**.

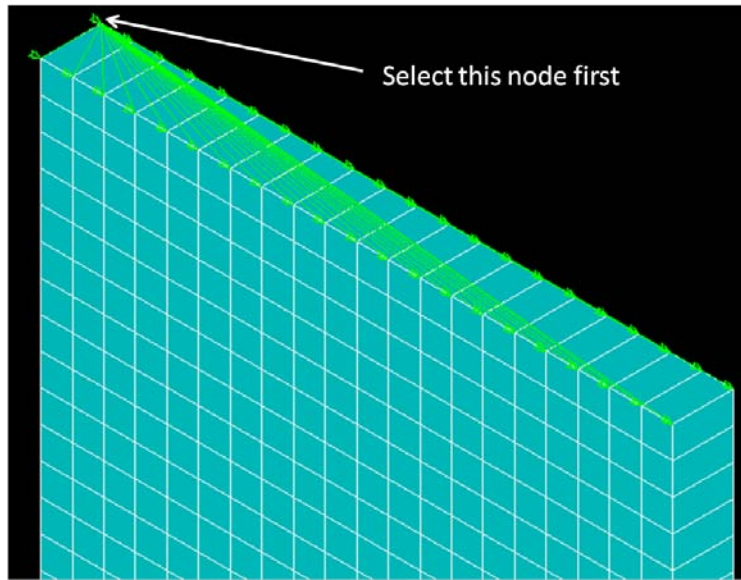


Figure 13. Coupling constraint on top surface of plate.

2.10 Specify solution controls

The default solution controls in ANSYS typically do not allow for a robust and efficient solution in a progressive failure simulation of a composite structure. There are specific solution control parameters that, when used in conjunction with Helius:MCT, allow for a solution with a decreased tendency for time increment cutbacks and result in a faster overall solution time. These solution controls are defined in the following steps. For further information regarding each command, refer to the ANSYS documentation and the Helius:MCT User's Guide.

1. Enter the following commands into the command prompt:
 - a. **NROPT, FULL, , OFF** – This command instructs ANSYS to use the 'Full' Newton Raphson algorithm and prevents ANSYS from using the 'Adaptive Descent' algorithm.
 - b. **PRED, OFF, , OFF** – This command prevents ANSYS from using the converged solution at the last substep to estimate the solution for the current substep.
 - c. **NSUBST, 50, 50, 50** – This command is used to specify the number of substeps to be used in the analysis. Since the step time is 1, each substep time increment will be 0.02.
 - d. **NEQIT, 1000** – This command specifies the number of equilibrium iterations that must be performed before ANSYS evaluates the need to reduce the time increment size. It is intentionally large in order to force ANSYS to converge at each of the specified time steps.
 - e. **TIME, 1** – Specifies a step size equal to 1.

2. Switch to the solution processor by clicking **Main Menu → Solution**. Enter the following command into the command prompt:
 - a. **CNVTOL, F, , , 0** – This command is used to define the convergence tolerance for residual node forces.

2.11 Defining solution output to the results file

In order to view the fiber and matrix failure states generated by Helius:MCT, the user must request that the state variables (SVAR) be written in the ANSYS results (.rst) file.

1. Enter **OUTRES, SVAR, ALL** in the command prompt. This command specifies that all state variables are to be written to the results file at every substep.
2. Enter **OUTRES, NSOL, ALL**
3. Enter **OUTRES, NLOAD, ALL**
4. Enter **OUTRES, RSOL, ALL**

2.12 Solve the model

The model is now defined and ready to be solved.

1. To solve the model, click **Main Menu → Solution → Solve → Current LS.**
 - a. Disregard any warnings that may be presented

2.13 Viewing and interpreting the results

Helius:MCT generates several state variable outputs. In this step, a key variable (SVAR1) is viewed and discussed. SVAR1 is the state variable that keeps track of fiber and matrix failure within an element. Also, a load-displacement curve is generated and the simulated strength is compared to the experimental strength.

1. After the solution has completed, click **Main Menu → General Postproc.**
2. To view the true deformation scale, click **PlotCtrls → Style → Displacement Scaling.** Select the **1.0 (true scale)** radio button in the dialog box that appears and click **OK.**

State variable SVAR1 is used to identify the discrete damage state of the composite material. To plot this variable, either the PLESOL or the PLNSOL commands may be used. The PLESOL contour plot does not average values across element boundaries while the PLNSOL contour plot does average values across element boundaries. Since SVAR1 has discrete values, the PLESOL plot is the most useful. Note that in order to view SVARs greater than 11, power graphics must be turned off using the /GRAPHICS, FULL command.

3. To plot SVAR1 using a PLESOL contour plot, enter **PLESOL, SVAR, 1** into the command prompt. The default plot will plot the bottom section point of the bottom layer and the top section point of the top layer.

- Since there are 3 possible values of SVAR1 for unidirectional Heliuss:MCT materials, it is useful to adjust the contour color values. Click **PlotCtrls** → **Style** → **Contours** → **Non-uniform Contours**. In the dialog box that appears, set **V1 = 1**, **V2 = 2**, and **V3 = 3** and click **OK**. The contour plot should be similar to the plot shown in Figure 14. Minor differences are usually the result of mesh variations.

The following table lists the three possible values for SVAR1 and the corresponding failure states and contour colors.

Value of SVAR1	Failure State	Color
1	No failure	Blue
2	Matrix failure	Green
3	Fiber and matrix failure	Red

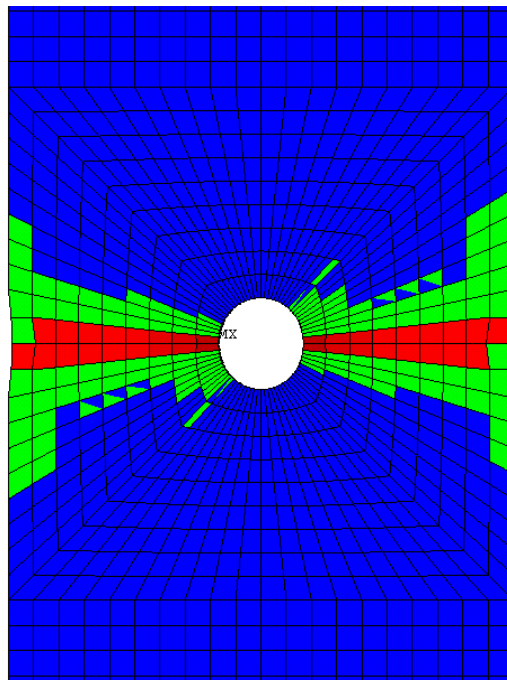


Figure 14. PLESOL contour plot of ply 1 at the end of the step.

To view results in a specific ply (or layer) of the composite plate, the LAYER command can be used.

- Enter **LAYER, 2** in the command prompt to view results in ply 2 (a 0° ply). Note that it may be necessary to click **Plot** → **Replot** to refresh the Graphics Window. The contour plot of SDV1 for ply 2 at the end of the load step is shown in Figure 15.

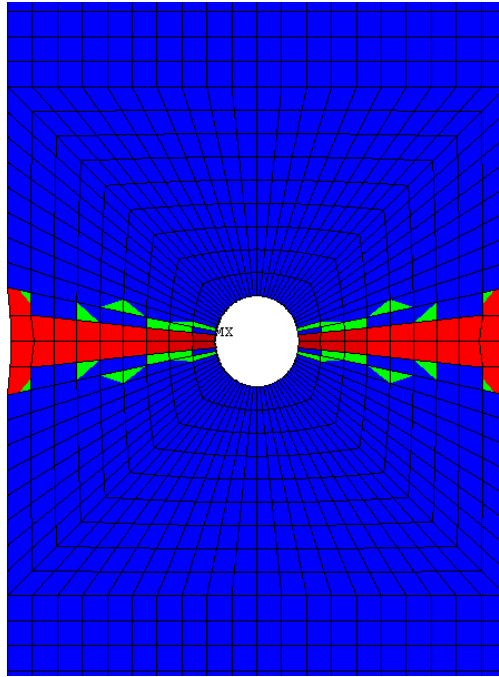


Figure 15. PLESOL contour plot of ply 2 at the end of the step.

Viewing the progression of failure is often useful for visualizing the way a structure fails. Commands to view results at specific points in time can be accessed from **Main Menu → General Postproc → Read Results** or using the **SET** command.

6. Starting at Substep 1, progress through the step while refreshing the Graphics Window to determine when failure initiates and how failure propagates in ply 2.
 - a. Enter the commands, **SET, 1, substep#** or use **Main Menu → General Postproc → Read Results → By Load Step**.
 - b. Failure should initiate at Time = 0.68 (substep 34).
 - c. Fiber failure should extend laterally from the hole to the plate edges at Time = 0.78 (substep 39).

The progression of failure in ply-2 is shown in Figure 16. As expected, failure initiates at the corners of the notch and progresses laterally towards the edge of the plate until ultimate failure.

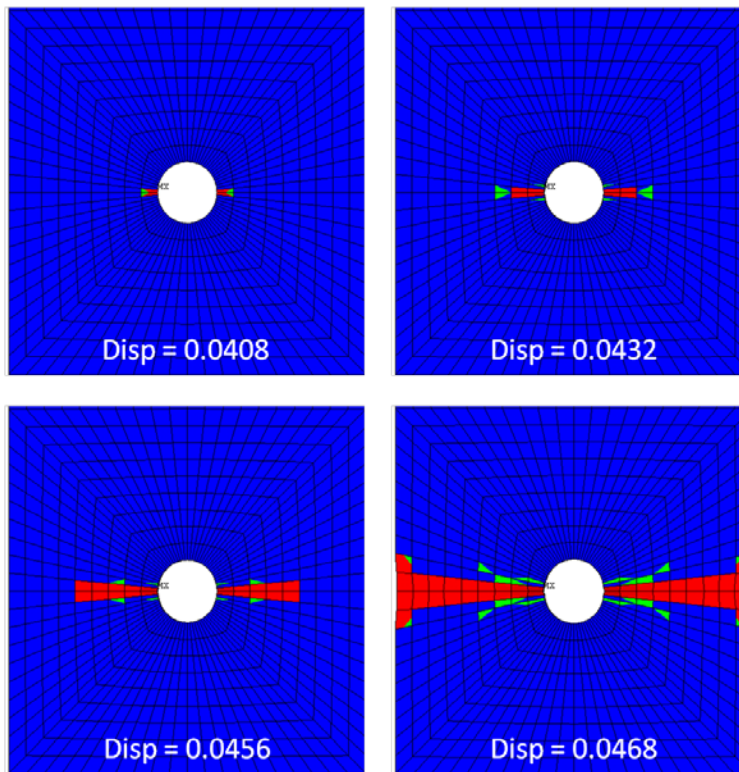


Figure 16. Progression of failure in ply-2 (0°).

2.13.1 Generating a load-displacement plot

Load-displacement plots are commonly used as a tool to determine the global stiffness response of a structure. They are particularly useful for progressive failure analyses as they are a simple means of determining how the structure behaves as failure initiates and progresses.

1. Click **Main Menu** → **TimeHist Postpro**.
2. In the GUI that appears, click the **Add Data** icon (the green plus sign).
3. Select **Reaction Forces** → **Structural Forces** → **Y-Component of force**. Click **OK**.
4. From the graphics window, select the node that had the specified displacement (refer to Figure 13). Click **OK**.
5. Repeat steps 2-4, but instead, select **Nodal Solution** → **DOF Solution** → **Y-Component of displacement**.
6. To plot the reaction force vs. displacement, set the X-Axis to **UY_3**, select **FY_2** and **UY_3**, and click the **Graph Data** icon. Refer to Figure 17.

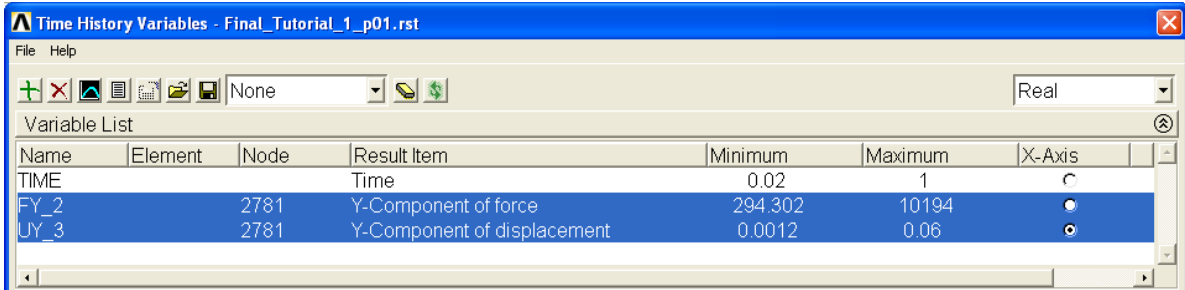


Figure 17. Time History Variables GUI.

The load-displacement plot is shown in Figure 18. The sharp drop in stiffness at a displacement of 0.0468 corresponds to ultimate failure of the plate. The simulated ultimate load is 10,194 lbs and the strength is 47.2 ksi. The experimental¹ strength of the plate is given in Table 2 as 50.5 ksi. The ultimate strength predicted by Helius:MCT and the experimental strength differ by 6.5%.

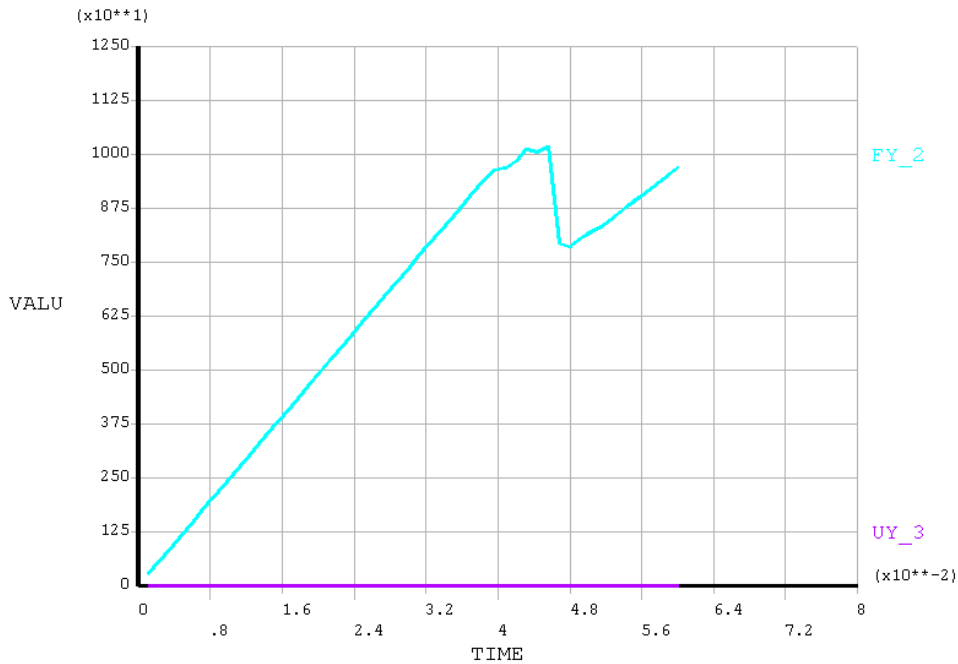


Figure 18. Load-displacement plot.

Table 2. Experimental OHT strength.

Test Property	Layup:		50/40/10		25/50/25		10/80/10	
	Test Condition	Unit	B-Value	Mean	B-Value	Mean	B-Value	Mean
No Hole Tension								
Ult. Strength	75°F/Dry	ksi	124	157	77.3	98.2	42.1	53.5
Modulus		msi	-	10.8	-	6.92	-	4.42
Filled Hole Tension								
Ult. Strength	-65°F/Dry	ksi	47.1	60.3	35.1	45.0	34.9	44.7
	75°F/Dry		57.3	71.2	41.3	51.3	34.9	43.4
Open Hole Tension								
Ult. Strength	-65°F/Dry	ksi	-	-	32.9	42.2	30.8	39.4
	75°F/Dry		-	-	40.6	50.5	30.9	38.4
	180°F/Wet		-	-	51.6	64.1	29.1	36.2

3 Summary

A composite plate with a hole was modeled, and Helius:MCT was used to predict progressive failure caused by a displacement controlled load. Three different procedures specific to Helius:MCT were used:

1. In section 2.3, Helius Material Manager was used to characterize the material using experimental lamina properties.
2. In section 2.4, the Helius:MCT GUI was used to define the material with ANSYS.
3. In section 2.10, solution controls were defined.
4. In section 2.11, state variable (SVAR) output to the results file (.rst) was requested.

After the solution was completed, ANSYS was used to plot fiber and matrix failure in individual plies and to view the progression of failure within ply 2. A load-displacement plot was also generated.

4 References

1. Toray Composites (America), INC. AGATE Laminate Material Qualification of T700G/#2510, 150 g/m², Unidirectional Tape. Test Report No. TCQAL-T-1026, December, 4, 2002.
2. Tomblin J., et al., "A-Basis and B-Basis Design Allowables for Epoxy-Based Prepreg TORAY T700GC-12K-31E/#2510 Unidirectional Tape." AGATE-WP3.3-033051-132, November 2002.
3. Soden, P.D., Hinton, M.J. and Kaddour, A.S., "Lamina Properties, Lay-up Configurations and Loading Conditions for a Range of Fibre-Reinforced Composite Laminates." *Composites Science and Technology*, Vol. 58, 1011-1022, 1998.
4. Kaddour, A.S. and Hinton, M.J., "Instructions to Contributors of the Second World-Wide Failure Exercise (WWFE-II): Part (A)." Data available upon request.