

# HELIUS:MCT

ADVANCED COMPOSITES SIMULATION | FIREHOLE COMPOSITES

## Tutorial 2

**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 input files that are compatible with Helius:MCT and the viewing of special solution variables that are computed by Helius:MCT. Tutorial 2 demonstrates the process of manually converting an existing ANSYS input file to achieve compatibility with Helius:MCT.

For questions, comments or further information, contact Firehole Composites at [support@firehole.com](mailto: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

|          |  |          |
|----------|--|----------|
| <b>1</b> | <b>Introduction</b> .....                    | <b>3</b> |
| <b>2</b> | <b>Tutorial Steps</b> .....                  | <b>4</b> |
| 2.1      | Modifying the material definition .....      | 4        |
| 2.2      | Adding solution controls .....               | 6        |
| 2.3      | Requesting state variable output .....       | 7        |
| 2.4      | Submitting the input file for analysis ..... | 7        |
| <b>3</b> | <b>Summary</b> .....                         | <b>8</b> |

## Table of Figures

|           |   |   |
|-----------|---|---|
| Figure 1. | Dimensions of composite plate. ....     | 3 |
| Figure 2. | Location of the ANSYS Material ID. .... | 5 |
| Figure 3. | Command prompt. ....                    | 8 |

## Helius:MCT Tutorial 2: Modifying an Input File for use with Helius:MCT

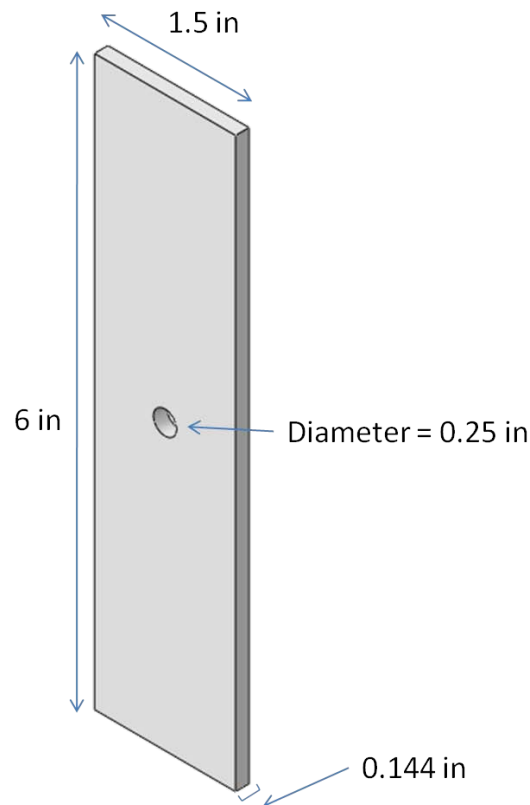
### 1 Introduction

In the first tutorial, a finite element model of a composite plate was created in ANSYS and analyzed using Helius:MCT to predict the progressive failure response of the plate. This tutorial provides instructions for modifying an *existing* ANSYS input file to achieve compatibility with Helius:MCT. The ANSYS input file is based on the problem from Tutorial 1, except that all Helius:MCT specific modifications have been left out. Therefore, the mesh, layup, dimensions, etc. for this model are the same as the mesh, layup, dimensions, etc. for the model from Tutorial 1.

The following sections of the ANSYS input file will be modified in this tutorial to accommodate Helius:MCT:

1. Material definition
2. Solution controls
3. Output definition

The model is based on coupon level test results from the National Institute for Aviation Research (NIAR). Specifically, it is based on an open-hole tension (OHT) plate made from T700/2510 that is compared to the experimental results 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.**

The unmodified ANSYS input file (HeliusMCT\_Tutorial\_2\_ANSYS.ans) for this tutorial can be downloaded from the Customer Login at [www.firehole.com](http://www.firehole.com).

## 2 Tutorial Steps

### 2.1 Modifying the material definition

The materials that are processed by Helius:MCT are considered by ANSYS to be *user-defined* material types, as opposed to the standard material types that are recognized by ANSYS. The unmodified input file for this Tutorial uses MP commands to define an orthotropic elastic material. Hence, the material definition needs to be changed from an elastic material type to a *user-defined* material type.

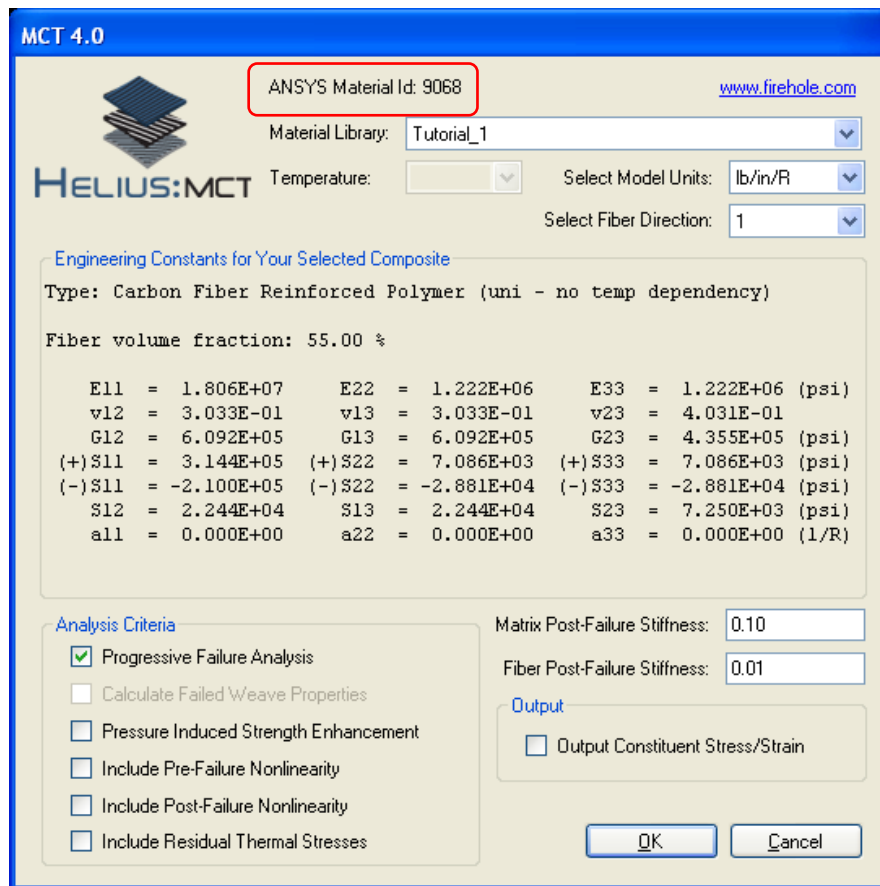
The specific format of a *Helius user-defined material* command is given by:

**HELIUSMCT**, *MATID*, *NSTATV*, *UNITS*, *PFIB\_DIR*, *PFA*, *PREFAIL*, *POSTFAIL*, *PRESS*, *TEMPDEPEND*, *unused*, *unused*, *unused*, *unused*, *MDEG*, *FDEG*, *FCONV* (*optional*), *LCONV* (*optional*), *TCONV* (*optional*)

For detailed information regarding each argument in the HELIUSMCT command, refer to Appendix A of the Helius:MCT User's Guide.

The following steps will modify the HELIUSMCT command for this exercise.

1. Open Tutorial 1 and create the user material (which is named Tutorial\_1) using the instructions provided in section 2.3 of Tutorial 1.
2. Determine the ANSYS Material ID (*matID*) number for Tutorial\_1 by opening the Helius GUI from within ANSYS and selecting Tutorial\_1 from the Material Library list. The location of the Material ID number is highlighted in Figure 2. Alternatively, the *matID* can be determined by opening the HeliusMCT\_matDB.xml file, which is located in the Helius Material directory. The particular value of the Material ID may differ depending on the number of materials in your Helius Material directory.



**Figure 2. Location of the ANSYS Material ID.**

3. Within HeliusMCT\_Tutorial\_2\_ANSYS.ans, locate the section containing the MP commands.
4. Replace the set of MP commands with the following, ensuring that matID is replaced with the value determined in step 1:

```
HELIUSMCT, matID, 6,3,1,1,0,0,0,0,0,0,0,0,0.1,0.01
```

The first item in the above command instructs ANSYS to call the Helius:MCT macro and the second item identifies the Helius:MCT material. In order, the third through sixteenth items specify the number of state variables, system of units, fiber direction, failure analysis on/off, pre-fail non-linearity on/off, post-fail non-linearity on/off, pressure modifications on/off, thermal residual stresses on/off, (items 11-14 are unused), matrix post-failure stiffness fraction, and fiber post-failure stiffness fraction. For this problem, there are six state variables, the unit system is in/lb/°R, 1 is the fiber direction, failure analysis is turned on, pre-failure is off, post-failure is off, pressure modifications are off, thermal residual stresses are off, the matrix post-failure stiffnesses are 10% of the

undamaged matrix stiffnesses, and the fiber post-failure stiffnesses are 1% of the undamaged fiber stiffnesses.

5. If necessary, change the matID in the section definition (secdata commands) to match the matID determined in step 1.

## 2.2 Adding 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 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. Within HeliusMCT\_Tutorial\_2\_ANSYS.ans, locate the command:

```
D,P51X, ,0.06, , , ,UY, , , , , .
```

2. Directly below the above command, insert the following:

```
NROPT,FULL, ,OFF
PRED,OFF, ,OFF
NSUBST,50,50,50
NEQIT,1000
TIME,1
FINISH
/SOL
CNVTOL,F, , ,0
```

The NROPT command instructs ANSYS to use the 'Full' Newton Raphson algorithm and prevents ANSYS from using the 'Adaptive Descent' algorithm.

The PRED command prevents ANSYS from using the converged solution at the last substep to estimate the solution for the current substep.

The NSUBST command is used to specify the number of substeps to be used in the analysis. For this Tutorial, 50 substeps will be imposed. Multiple substeps help to identify when failure initiates and how it progresses as the load increases.

The NEQIT 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.

The TIME command simply specifies a step size equal to 1.

The CNVTOL command is used to define the convergence tolerance for residual node forces.

## 2.3 Requesting state variable output

Helius:MCT uses ANSYS state variables to store information about the fiber and matrix at each integration point in the model. To view the state variables generated by Helius:MCT in ANSYS, the state variables (SVAR) must be written to the results (.rst) file.

1. Directly below the CNVTOL command, enter the following command:

```
OUTRES , SVAR , ALL
```

This command specifies that all state variables are to be written to the results file at every substep.

2. Below the above command, enter the following commands. These commands allow for the reaction force to be computed.

```
OUTRES , NSOL , ALL
```

```
OUTRES , NLOAD , ALL
```

```
OUTRES , RSOL , ALL
```

3. Save the file.

Note: Within HeliusMCT\_Tutorial\_2\_ANSYS.ans, the command **KEYOPT, 1, 8, 1** is used to store integration point data at the top and bottom of all layers in the section. The default is to only store integration point data for the bottom of the bottom layer and the top of the top layer.

## 2.4 Submitting the input file for analysis

The input file is now ready to be submitted for analysis through a command prompt. A batch file named *ansys-mct.bat* is included with the Helius:MCT installation that simplifies the submission of Helius:MCT jobs. This batch file is installed in the Firehole directory on your computer. The use of this batch file is demonstrated here.

1. Open a command prompt window.
2. Change the directory to the directory where the input file is located by entering:

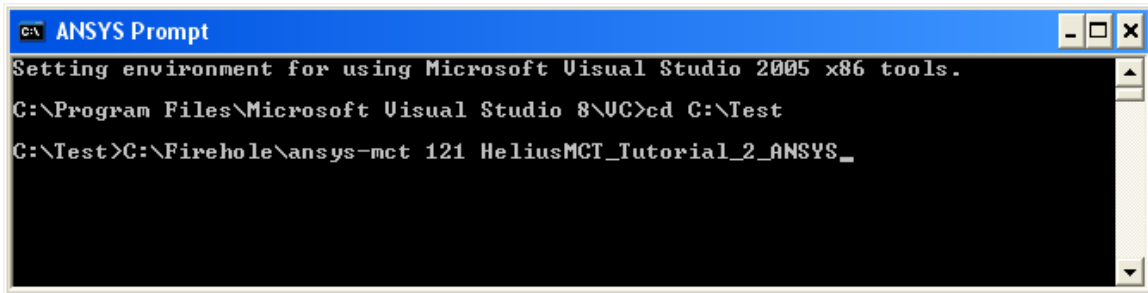
```
cd [directory address]
```

For example, if the input file is located in “C:\Test”, enter “cd C:\Test” as shown in Figure 3.

3. Submit the job by entering following command as shown in Figure 3:

```
C:\Firehole\ansys-mct 121 HeliusMCT_Tutorial_2_ANSYS
```

In the above, `C:\Firehole\ansys-mct` calls the batch file that submits jobs using the custom ANSYS executable, `121` is the version of ANSYS (120 and 130 are the other valid options), and `HeliusMCT_Tutorial_2_ANSYS_` is the name of the input file.



```

ANSYS Prompt
Setting environment for using Microsoft Visual Studio 2005 x86 tools.
C:\Program Files\Microsoft Visual Studio 8\VC>cd C:\Test
C:\Test>C:\Firehole\ansys-mct 121 HeliusMCT_Tutorial_2_ANSYS_
  
```

**Figure 3. Command prompt.**

4. After the finite element simulation is complete, refer to section 2.13 of Tutorial 1 for instructions to view and interpret the results.
  - a. To open the results file, start ANSYS APDL and open `HeliusMCT_Tutorial_2_ANSYS.db`. Then click **General Postproc → Data & File Opts** and select `HeliusMCT_Tutorial_2_ANSYS.rst`. To read the last set of results, click **General Postproc → Read Results → Last Set**.
  - b. Note, that in order to view SVARs greater than 11, power graphics must be turned off by issuing the `/GRAPHICS, FULL` command.

### 3 Summary

In this 2<sup>nd</sup> tutorial, an ANSYS input file was modified to make it compatible with Helius:MCT. Three sections of the input file required modification:

1. Material definition
  - a. Replaced elastic material definition (MP commands) with the `HELIUSMCT` command
2. Solution controls
  - a. Added `NROPT`, `PRED`, `NSUBST`, `NEQIT`, `TIME`, and `CNVTOL` commands to make the non-linear solution process more efficient and robust
3. Output requests
  - a. Specified `SVAR` output for every substep