



# Tutorial 2

**Heliuss:MCT™ Version 2.0 for ANSYS**  
**July, 2009**

## **Abstract**

This document provides a step-by-step tutorial that demonstrates the use of Heliuss:MCT. The primary emphasis is the creation of ANSYS input files that are compatible with Heliuss:MCT and the viewing of special solution variables that are computed by Heliuss:MCT. Tutorial 2 demonstrates the process of manually converting an existing ANSYS input file to achieve compatibility with Heliuss:MCT.

For questions, comments or further information, contact Firehole Technologies at [support@fireholetech.com](mailto:support@fireholetech.com)

## **Legal Notices**

Copyright 2009, Firehole Technologies, Inc.

Heliuss:MCT is a trademark of Firehole Technologies, Inc. Any use of the Heliuss: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 .....	8
<b>3</b>	<b>SUMMARY.....</b>	<b>9</b>

## Table of Figures

Figure 1:	Dimensions and loading of composite plate .....	4
Figure 2:	Location of the ANSYS Material ID.....	5
Figure 3:	Command prompt .....	9

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

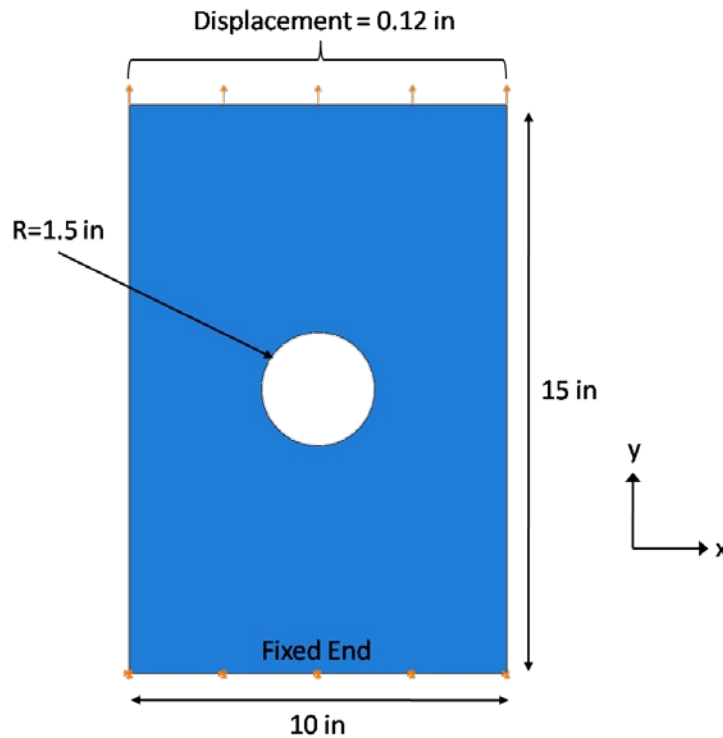
### 1 Introduction

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 example problem consists of a flat plate with a hole in the center subject to fixed boundary conditions on one end and imposed displacements on the other end (see Figure 1). The material is IM7/8552; the layup is  $[0/\pm 45/90]_s$ , and the ply thickness is 0.005" resulting in a plate thickness of 0.04". The mesh was generated using 8-node layered solid shell elements (SOLSH190) and a global seed size of 0.2.



**Figure 1: Dimensions and loading of composite plate**

The unmodified ANSYS input file (HeliuSMCT\_Tutorial\_2\_ANSYS\_v2.inp) for this tutorial is provided on Firehole's internet based User Portal.

## 2 Tutorial Steps

### 2.1 Modifying the material definition

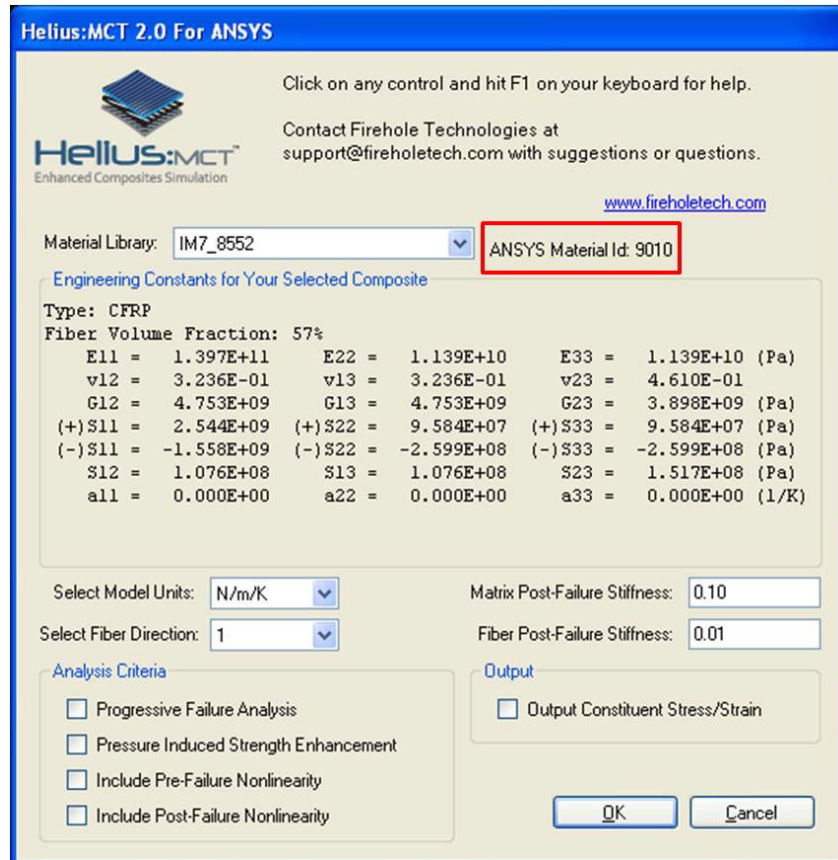
The materials that are processed by HeliuSMCT are considered by ANSYS to be *user-defined* material types, as opposed to the standard material types that are recognized by ANSYS. The unmodified input file for this Tutorial uses MP commands to define an orthotropic elastic material. Hence, the material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

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

*HELIOUSMCT, MATID, NSTATV, UNITS, PFIB\_DIR, PFA, PREFAIL, POSTFAIL, PRESS, MDEG, FDEG, FCON (optional), LCONV (optional), TCONV (optional).*

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

1. Determine the ANSYS Material ID (*matID*) number for IM7\_8552 by opening the Helius GUI from within ANSYS and selecting IM7\_8552 from the Material Library list. The location of the Material ID number is highlighted in Figure 2. 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**

2. Within HeliusMCT\_Tutorial\_2\_ANSYS.inp, locate the section containing the MP commands.
3. 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, 0.1, 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 eleventh

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, 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, 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.

4. If you are using Ansys 11, enter the following command directly below the HELIUSMCT command:

```
MP , EX , matID , 1e7
```

The above command defines a longitudinal elastic modulus definition for the Helius:MCT material. This value is used to fill a place holder required by ANSYS when using layered elements with a user-defined material. The value of EX does not affect the results. This value is not required in Ansys 12.

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.inp, locate the command `DA , P51X , UY , 0 . 1 2 .`
2. Directly below the above command, insert the following:

```
NROPT , FULL , , OFF  
PRED , OFF , , OFF  
NSUBST , 10 , 10 , 10  
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, 10 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 , ESOL , NONE
```

This command suppresses writing all element solutions to the results file and is used to decrease the size of the results file. Depending on the model size, writing all element solutions to the results file will slow down the solution and can create very large files.

2. Below the above command, enter:

```
OUTRES , SVAR , ALL
```

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

3. Below the above command, enter the following command if you wish to view element nodal stresses:  
`OUTRES , STRS , ALL`
4. Save the file.

Note: Within HeliumMCT\_Tutorial\_2\_ANSYS.inp, 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 using batch mode.

1. Open a command prompt window.
2. Change the directory to the directory where the input file is located by entering:  
`cd [directory address]`
  - a. For example, if the input file is located in “C:\Firehole”, enter “cd C:\Firehole” as shown in Figure 3.
3. Submit the job by entering one of the following suggested commands:

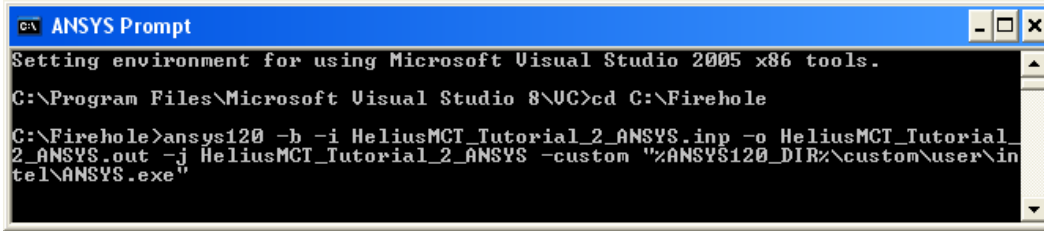
- a. For 32-bit machines:

```
ansys120 -b -i HeliumMCT_Tutorial_2_ANSYS.inp -o HeliumMCT_Tutorial_2_ANSYS.out -j  
HeliumMCT_Tutorial_2_ANSYS -custom "%ANSYS120_DIR%\custom\user\intel\ANSYS.exe"
```

- b. For 64-bit machines:

```
ansys120 -b -i HeliumMCT_Tutorial_2_ANSYS.inp -o HeliumMCT_Tutorial_2_ANSYS.out -j  
HeliumMCT_Tutorial_2_ANSYS -custom "%ANSYS120_DIR%\custom\user\winx64\ANSYS.exe"
```

In the above, the `-b` option tells ANSYS to run in batch mode, the `-i` option specifies the name of the file to be read, the `-o` option writes the procedures to an output file instead of the command prompt, the `-j` option specifies the name for files generated during analysis, and the `-custom` option specifies the location of the custom ANSYS executable.



```
ANSYS Prompt
Setting environment for using Microsoft Visual Studio 2005 x86 tools.
C:\Program Files\Microsoft Visual Studio 8\VC>cd C:\Firehole
C:\Firehole>ansys120 -b -i HeliusMCT_Tutorial_2_ANSYS.inp -o HeliusMCT_Tutorial_2_ANSYS.out -j HeliusMCT_Tutorial_2_ANSYS -custom "%ANSYS120_DIR%\custom\user\intel\ANSYS.exe"
```

**Figure 3: Command prompt**

4. After the finite element simulation is complete, refer to Section 2.12 of Tutorial 1 for instructions to view and interpret the results.

### 3 Summary

In the 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.
  - b. Added a longitudinal modulus definition to satisfy ANSYS 11 requirements for using layered elements with user-defined materials.
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.