

# HELIUS:MCT

ADVANCED COMPOSITES SIMULATION | FIREHOLE COMPOSITES

## Tutorial 2

**HeliuS:MCT™ Version 4.0 for Abaqus**  
**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 Abaqus 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 Abaqus 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.

Abaqus/Standard is a trademark of Dassault Systemes S.A. and Dassault Systemes SIMULIA Corp.

## 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	Modifying the section parameters.....	5
2.3	Modify the step definition.....	6
2.4	Modify the output requests .....	7
2.5	Submitting the input file for analysis.....	8
<b>3</b>	<b>Summary.....</b>	<b>9</b>

## Table of Figures

Figure 1:	Dimensions of composite plate .....	3
Figure 2:	Command prompt .....	9

## Helius:MCT Tutorial 2

### Modifying an Abaqus Input File for use with Helius:MCT

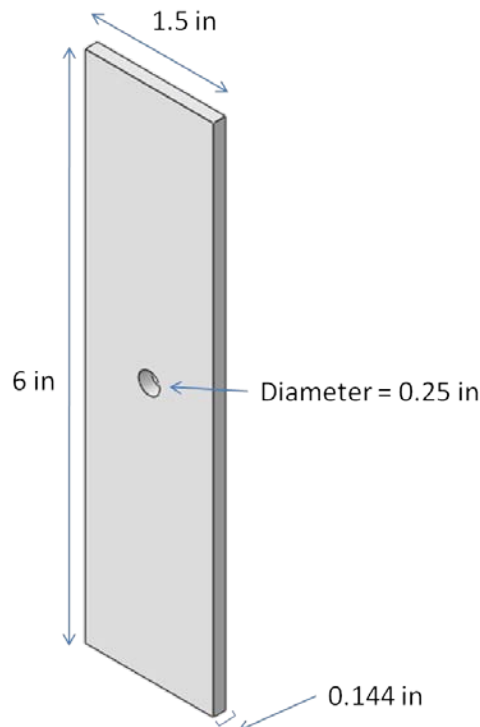
#### 1 Introduction

In the first tutorial, a finite element model of a composite plate was created in Abaqus/CAE and analyzed using Helius:MCT to simulate the progressive failure response of the plate. This tutorial provides instructions for modifying an *existing* Abaqus input file to achieve compatibility with Helius:MCT. The Abaqus input file is based on the problem from Tutorial 1, except 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 Abaqus input file will be modified in this tutorial to accommodate Helius:MCT:

1. Material definition
2. Section definition
3. Step definition
4. 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 Abaqus input file (HeliusMCT\_Tutorial\_2\_Abaqus.inp) used for this tutorial is available for download from the Customer Login at [www.firehole.com](http://www.firehole.com).

## 2 Tutorial Steps

### 2.1 Modifying the material definition

Helius:MCT uses a \*User Material definition instead of conventional Abaqus material definitions to define materials in Abaqus. In this step, the \*Elastic material definition will be replaced with a \*User Material definition that is formatted for Helius:MCT.

1. Open Tutorial 1 and create the user material using the instructions provided in Section 2.2 of Tutorial 1.
2. Open HeliusMCT\_Tutorial\_2\_Abaqus.inp in a text editor.
3. Locate the beginning of the material definition by searching for the **\*Material** keyword statement.
4. The entire material definition includes the following:
 

```
*Material, name=ExampleMaterial
*Elastic, type=LAMINA
  1.5e+07, 2e+06, 0.3, 800000., 800000., 450000.
```
5. In the \*Material keyword statement, change the name of the material from **ExampleMaterial** to **Tutorial\_1**. Note that the new material name must exactly match the name of a composite material that is stored in the Helius:MCT Composite Material Database created in Section 2.2 of Tutorial 1. For detailed information on creating and managing Helius:MCT materials, refer to the Helius Material Manger User's Guide.
6. Delete the **\*Elastic** keyword statement and the single data line that follows the keyword statement.
7. Add the keyword **\*DepVar** after the **\*Material, name=Tutorial\_1** line. The \*DepVar keyword allocates memory for Helius:MCT solution-dependent state variables at each integration point.
8. Add **6** after the **\*DepVar** line. The value of 6 indicates that 6 solution-dependant state variables must be tracked throughout the solution process. Six is the default number of solution-dependant variables required by MCT.
9. Add the line **\*User Material, constants=13** after the state variable name assignments. The parameter "constants=13" indicates that there are 13 user material constants for this user material.
10. After the **\*User Material** line, add:
 

```
3, 1, 1, 0, 0, 0, 0
0, 0, 0, 0.1, 1E-06
```

- a. Two lines of user material constants are required since Abaqus permits a maximum of 8 user material constants per line. These thirteen numbers are *user material constants* that are used by Helius:MCT to determine the precise form of multiscale constitutive relations to use for the composite material. In order, they specify the system of units, fiber direction, progressive failure analysis on/off, pre-fail non-linearity, post-fail non-linearity/energy-based degradation, pressure modifications on/off, thermal residual stresses, (constants 8-11 are unused), matrix post-failure stiffness, and fiber post-failure stiffness. For this problem, the unit system is in/lb/R; 1 is the fiber direction; failure analysis is turned on, pre-failure nonlinearity, post-failure nonlinearity, energy-based degradation and pressure modifications are turned off, thermal residual stresses are not included, matrix post-failure stiffness is set to 10% of the unfailed stiffness, and fiber post-failure stiffness is set to 1E-04 % of the unfailed stiffness. For detailed information on the user material constants, refer to Appendix A of the Helius:MCT User's Guide.

11. The new material definition should now look like the following:

```
*MATERIAL, NAME=Tutorial_1
*DEPVAR
6
*USERMATERIAL, CONSTANTS=13
3, 1, 1, 0, 0, 0, 0, 0
0, 0, 0, 0.1, 1E-06
```

## 2.2 Modifying the section parameters

When a \*User Material definition is used, Abaqus is unable to compute certain section parameters because the elastic constants necessary to compute these section parameters are not available. The user must define these parameters in order for an analysis to run successfully.

Firehole has developed a command line program called xSTIFF that reads an Abaqus input file and *automatically* computes and inserts all of the *extraneous stiffness parameters* that are required by any reduced integration elements that utilize Helius:MCT composite materials. This auxiliary program significantly improves the speed and accuracy of the model building process.

1. Locate the section definition by searching for **\*SolidSection**.
2. The section definition will look like this:

```
*SOLIDSECTION, ELSET=PLATE_LAYUP-1, COMPOSITE, ORIENTATION=ORI-1,
STACKDIRECTION=3, LAYUP=PLATE_LAYUP
1., 1, EXAMPLEMATERIAL, 45., PLY-1
1., 1, EXAMPLEMATERIAL, 0., PLY-2
1., 1, EXAMPLEMATERIAL, -45., PLY-3
1., 1, EXAMPLEMATERIAL, 90., PLY-4
```

```
1., 1, EXAMPLEMATERIAL, 45., PLY-5
1., 1, EXAMPLEMATERIAL, 0., PLY-6
1., 1, EXAMPLEMATERIAL, -45., PLY-7
1., 1, EXAMPLEMATERIAL, 90., PLY-8
1., 1, EXAMPLEMATERIAL, 45., PLY-9
1., 1, EXAMPLEMATERIAL, 0., PLY-10
1., 1, EXAMPLEMATERIAL, -45., PLY-11
1., 1, EXAMPLEMATERIAL, 90., PLY-12
1., 1, EXAMPLEMATERIAL, 90., PLY-13
1., 1, EXAMPLEMATERIAL, -45., PLY-14
1., 1, EXAMPLEMATERIAL, 0., PLY-15
1., 1, EXAMPLEMATERIAL, 45., PLY-16
1., 1, EXAMPLEMATERIAL, 90., PLY-17
1., 1, EXAMPLEMATERIAL, -45., PLY-18
1., 1, EXAMPLEMATERIAL, 0., PLY-19
1., 1, EXAMPLEMATERIAL, 45., PLY-20
1., 1, EXAMPLEMATERIAL, 90., PLY-21
1., 1, EXAMPLEMATERIAL, -45., PLY-22
1., 1, EXAMPLEMATERIAL, 0., PLY-23
1., 1, EXAMPLEMATERIAL, 45., PLY-24
```

3. Replace all instances of **ExampleMaterial** with the name of the HeliumMCT material created in Section 2.1.
4. Save the input file.
5. Open an Abaqus command prompt and change the directory to the location where HeliumMCT\_Tutorial\_2\_Abaqus.inp is saved.

```
>>cd <directory containing HeliumMCT_Tutorial_2_Abaqus.inp>
```

6. Issue the following command:
 

```
>>xstf /i HeliumMCT_Tutorial_2_Abaqus
```
7. The xSTIFF utility will copy the input file, calculate and insert all necessary extraneous stiffness parameters, and save the file as HeliumMCT\_Tutorial\_2\_Abaqus\_xs.inp.
8. Open the modified input file and locate the section definition. Verify that the \*HourglassStiffness keyword was added to the end of the section definition.

## 2.3 Modify the step definition

Many linear elastic analyses require only a single increment per step. A single increment is inadequate, however, when progressive failure is modeled because multiple increments are required to visualize the initiation and propagation of failure. Default nonlinear solution controls, as recommended in Section 3.4 of the Helium:MCT User's Guide, are also modified.

1. If it is not already open, open HeliumMCT\_Tutorial\_2\_Abaqus\_xs.inp.
2. Locate the step definition by searching for **\*Step**.

- The step definition should now appear as:

```
*STEP, NAME=LOAD_STEP
*STATIC
1., 1., 1E-05, 1.
```

- Replace the third line with **0.01, 1, 1e-05, 0.01**

- The first term is the initial increment, the second is the time period of the step, the third is the minimum time increment allowed, and the fourth term is the maximum time increment allowed.

- On the following line, add the following solution controls statement:

```
*Controls, parameters=time incrementation
1000,1000,1000,1000,1000,,1000,,,10,
```

These parameters are included to allow Abaqus to take advantage of the enhanced convergence characteristics provided by Helius:MCT.

- The modified step definition should look like the following:

```
*STEP, NAME=APPLYLOAD
*STATIC
0.01, 1., 1E-05, 0.01
*Controls, parameters=time incrementation
1000,1000,1000,1000,1000,,1000,,,10,
```

## 2.4 Modify the output requests

To view the solution-dependant MCT state variables in Abaqus Viewer, the state variables must be written to the .odb file. The state variables are only written to the .odb file if explicitly requested in the Abaqus input file. This request is accomplished by adding SDV to the list of element output variables. By default, a preselected list of output variables is written to the output database. This default list will be modified as well.

- Locate the output definition by searching for **\*Output**.

- The definition should appear as:

```
*OUTPUT, FIELD, VARIABLE=PRESELECT
```

- Delete **, variable=PRESELECT**. This parameter is used to indicate that the default list of output variables will be written to the output database. State variables are not included in this list, so each output variable (represented by an identifying key) must be individually requested. This is done in step 5.
- Following the \*Output line, add the following lines:

```
*Node Output
RF, U
```

These lines write the nodal displacements (U) and reaction forces (RF) to the database.

5. Following the RF, U line, add the following lines:

```
*Element Output, directions=YES
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16
S, E, SDV
*Element Output, directions=YES
17, 18, 19, 20, 21, 22, 23, 24
S, E, SDV
```

- a. The first line, *\*Element Output, directions=YES*, is used to write element variables and section points to the output database. The *directions=YES* parameter indicates that the element material directions should be written to the database.
  - b. The second line, 1, 2, 3, ... is used to indicate specific section points in the layered element where the output variables are computed. Two instances of these commands are required since the maximum number of section points allowed per line is 16 (the 4<sup>th</sup>, 5<sup>th</sup> and 6<sup>th</sup> lines).
  - c. The third line, *S, E, SDV*, is a list of the variable identifying keys. *S* represents stress components, *E* represents total strain components, and *SDV* represents state variables.
6. The definition should now read:

```
*OUTPUT, FIELD
*NODEOUTPUT
RF, U
*ELEMENTOUTPUT, DIRECTIONS=YES
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16
E, S, SDV
*ELEMENTOUTPUT, DIRECTIONS=YES
17, 18, 19, 20, 21, 22, 23, 24
E, S, SDV
```

7. Save the file

The above is a simple output definition. Other input files may be different and more complicated. Generally speaking, there is only one modification to the output definition that is Helius:MCT specific: the addition of SDV to the list of element output variable identifying keys.

## 2.5 Submitting the input file for analysis

The input file is now ready to be submitted for analysis. The method used to submit a Helius:MCT job is the same as the standard method of submitting a job and there is no additional syntax required.

1. Open an Abaqus Command prompt.
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:\ABAQUS\Helius Tutorials”, enter “cd C:\ABAQUS\Helius Tutorials” as shown in Figure 2.

3. Submit the job. The format of the job submission depends on an option selected during the installation of Helius:MCT:

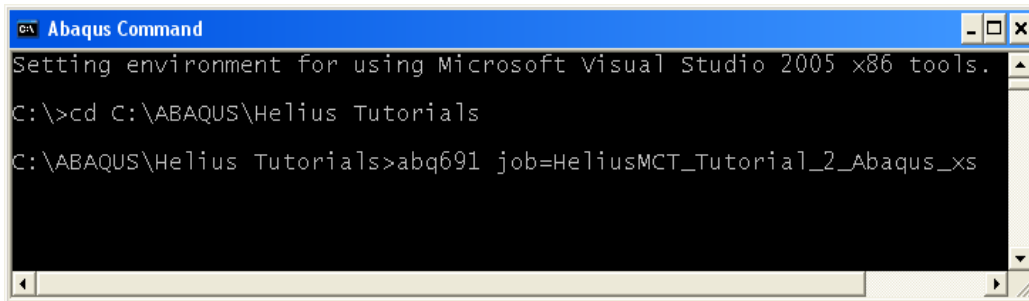
- a. If 'Yes' was selected when asked if you use user subroutines, the following line must be added to the Abaqus environment file:

```
usub_lib_dir='C:/Firehole/hmct/4.0'
```

Here, C:\Firehole is the root directory for Helius:MCT files. If your Helius:MCT root directory is different, simply replace C:\Firehole with the path to the root directory.

The job can then be submitted by typing “**abq691 job=[name of input file]**” as shown in Figure 2.

- b. If 'No' was selected when asked if you use user subroutines, then the job can be submitted by typing “**abq691 job=[name of input file]**” as shown in Figure 2.



```
Setting environment for using Microsoft Visual Studio 2005 x86 tools.  
C:\>cd C:\ABAQUS\Helius Tutorials  
C:\ABAQUS\Helius Tutorials>abq691 job=HeliusMCT_Tutorial_2_Abaqus_xs
```

**Figure 2: 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 Abaqus input file was modified to make it compatible with Helius:MCT. Four sections of the input file required modification:

1. \*Material
  - a. Replaced the elastic material definition with a user-defined material definition
  - b. Specified the user material constants and the number of state variables
2. \*Solid Section
  - a. Changed the material names to match the name of the Helius:MCT material
  - b. Ran the xSTIFF utility to automate the calculation and insertion of the extraneous stiffness parameters
3. \*Step
  - a. Adjusted the time incrementation
  - b. Modified the default nonlinear solution controls

4. \*Element Output  
Requested state variable output to the database