



Tutorial 2

**HeliuS:MCT™ Version 2.0 for Abaqus
July, 2009**

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 Technologies at support@fireholetech.com

Legal Notices

Copyright 2009, 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 is a trademark of ANSYS, Inc.

Table of Contents

1	INTRODUCTION.....	3
2	TUTORIAL STEPS	4
2.1	MODIFYING THE SECTION PARAMETERS	4
2.2	MODIFYING THE MATERIAL DEFINITION	5
2.3	MODIFY THE STEP DEFINITION	6
2.4	MODIFY THE OUTPUT REQUESTS.....	7
2.5	SUBMITTING THE INPUT FILE FOR ANALYSIS	8
3	SUMMARY.....	9

Table of Figures

Figure 1.	Dimensions and loading of composite plate	3
Figure 2.	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 Abaqus/CAE and analyzed using Helius:MCT to predict 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 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 ABAQUS input file will be modified in this tutorial to accommodate Helius:MCT:

1. Section definition
2. Material definition
3. Step definition
4. 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 continuum shell elements (SC8R) and a global seed size of 0.2.

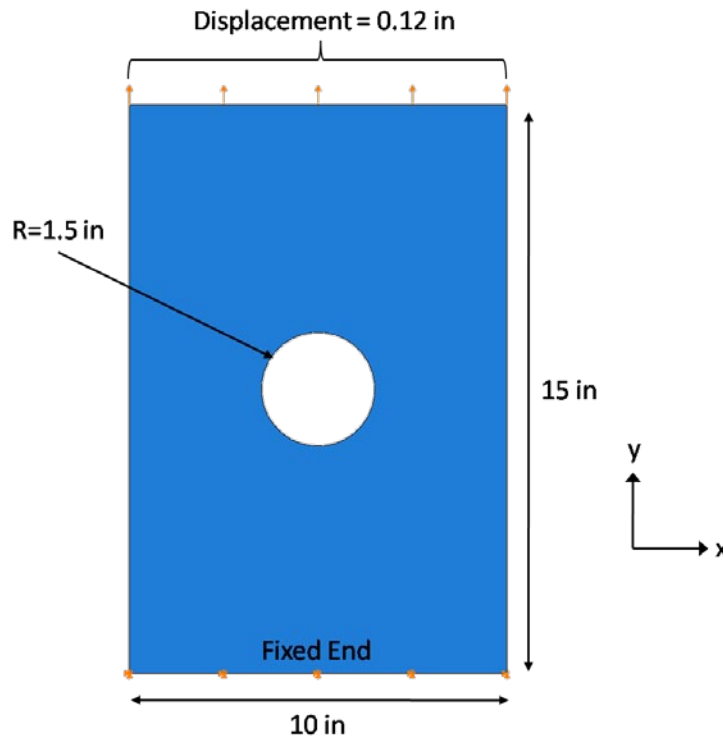


Figure 1: Dimensions and loading of composite plate

The unmodified ABAQUS input file (Helius_Tutorial_2_Abaqus_v2.inp) for this tutorial is provided on Firehole's internet based User Portal.

2 Tutorial Steps

2.1 Modifying the section parameters

The materials that are processed by Helius:MCT are considered by ABAQUS to be *user-defined* material types, as opposed to the standard material types that are recognized by ABAQUS. As explained in Appendix B of the Helius:MCT User's Guide, the use of user-defined material types places additional requirements on the section definitions that appear in the input file. According to Appendix B.1 of the Helius:MCT User's Guide, shell sections that are referenced by continuum shell elements require the section Poisson ratio, section thickness modulus, hourglass stiffness control parameters and transverse shear stiffnesses to be added to the section definition.

1. Open the input file provided above.
2. Locate the section definition by searching for ***Shell Section**.
3. The section definition will look like this:

```
*Shell Section, elset=PlateLayup-1, composite, orientation=Ori-1,
stack direction=3, layup=PlateLayup
1., 3, ExampleMaterial, 0., Ply-1
1., 3, ExampleMaterial, 45., Ply-2
1., 3, ExampleMaterial, -45., Ply-3
1., 3, ExampleMaterial, 90., Ply-4
1., 3, ExampleMaterial, 90., Ply-5
1., 3, ExampleMaterial, -45., Ply-6
1., 3, ExampleMaterial, 45., Ply-7
1., 3, ExampleMaterial, 0., Ply-8
```

The following seven steps are used to modify the shell section definition to achieve compatibility with Helius:MCT. These steps involve changing the name of the composite material and adding the required extraneous stiffness parameters to the section definition. The method used to determine the values of the extraneous stiffness parameters is discussed in Appendix B.3 of the Helius:MCT User's Guide. For expediency, these values are simply provided below in steps 2, 4 and 6.

1. Change the material name **ExampleMaterial** to the name of the user material, **IM7_8552**.
 Tip: To reduce user time, particularly on input files with many sections, use the Replace tool in the text editor.
2. Add **poisson=0.5** and **thickness modulus=1.38e+06** to the first line in the section definition.
3. Add ***Transverse Shear** after the last line.
4. Add **4.42e+06, 3.62e+06, 63300** after the line containing the ***Transverse Shear** keyword.
5. Add ***Hourglass Stiffness** after the last line.
6. Add **3445,,2592**, after the line containing the ***Hourglass Stiffness** keyword.
7. The shell section definition should now look like the following:

```
*Shell Section, elset=PlateLayup-1, composite, orientation=Ori-1,
poisson=0.5, thickness modulus=1.38e+06, stack direction=3,
layup=PlateLayup
1., 3, IM7_8552, 0., Ply-1
1., 3, IM7_8552, 45., Ply-2
1., 3, IM7_8552, -45., Ply-3
1., 3, IM7_8552, 90., Ply-4
1., 3, IM7_8552, 90., Ply-5
1., 3, IM7_8552, -45., Ply-6
1., 3, IM7_8552, 45., Ply-7
1., 3, IM7_8552, 0., Ply-8
*Transverse Shear
4.42e+06, 3.62e+06, 63300
*Hourglass Stiffness
3445,,2592,
```

If a particular input file has multiple section definitions, the above procedure needs to be repeated for each section definition.

2.2 Modifying the material definition

The material definition needs to be changed from an *elastic* material type to a *user-defined* material type.

1. Locate the beginning of the material definition by searching for the ***Material** keyword statement.
2. The entire material definition includes the following:

```
*Material, name=ExampleMaterial
*Elastic, type=LAMINA
1.5e+07, 2e+06, 0.3, 800000., 800000., 450000.
```
3. In the ***Material** keyword statement, change the name of the material from **ExampleMaterial** to **IM7_8552**. 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.
4. Delete the ***Elastic** keyword statement and the single data line that follows the keyword statement.
5. Add the keyword ***DepVar** after the ***Material** line.
6. Add **6** after the ***DepVar** line.
7. Add the line ***User Material, constants=8** after the state variable name assignments.
8. After the ***User** line, add **3, 1, 1, 0, 0, 0, 0.01, 0.01**

These eight 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, failure analysis on/off,

pre-fail non-linearity, post-fail non-linearity, pressure modifications on/off, 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- and post-failure nonlinearity and pressure modifications are turned off, and both matrix and fiber post-failure stiffnesses are set to 0.01. For further information on the user material constants, refer to Appendix A of the Helius:MCT User's Guide.

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

```
*Material, name=IM7_8552
*DepVar
6
*User Material, constants=8
3, 1, 1, 0, 0, 0, 0.01, 0.01
```

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 3.3 of the User's Guide, are also modified.

1. Locate the step definition by searching for ***Step**.
2. The step definition should look like this:

```
*Step, name=ApplyLoad
*Static
1., 1., 1e-05, 1.
```

3. In the ***Step** line, add **extrapolation=NO**.

The extrapolation feature is not appropriate for non-smooth nonlinear response that is characteristic of material failure.

4. Replace the third line with **0.05, 1, 1e-10, 0.05**

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.

5. Following the line added in step 4., add the following:

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

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

- The modified step definition should look as following:

```
*Step, name=ApplyLoad, extrapolation=NO
*Static
0.05, 1, 1e-10, 0.05
*Controls, parameters=time incrementation
1000,1000,1000,1000,1000,,1000,,,10,
```

2.4 Modify the output requests

To view the 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 look like this:
*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.

- Following the RF, U line, add the following lines:

```
*Element Output, directions=YES
2, 5, 8, 11, 14, 17, 20, 23
S, E, SDV
```

- The first line, *Element Output, direction=YES, is used to write element variables and section points to the output database. The direction=YES parameter indicates that the element material directions should be written to the database.
- The second line, 2, 5, 8, 11, 14, 17, 20, 23, is used to indicate specific section points in the layered element where the output variables are computed. In this case, the selected sections represent the mid-surface of each material ply.
- The third line, S, E, SDV, is a list of the variable identifying keys. S represents stresses; E represents strains, and SDV represents state variables.

6. The definition should now look like:

```
*Output, field
*Node Output
RF, U
*Element Output, directions=YES
2, 5, 8, 11, 14, 17, 20, 23
S, E, SDV
```

7. Save the file

The above is a simple output section definition. Other input files will 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 the 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.

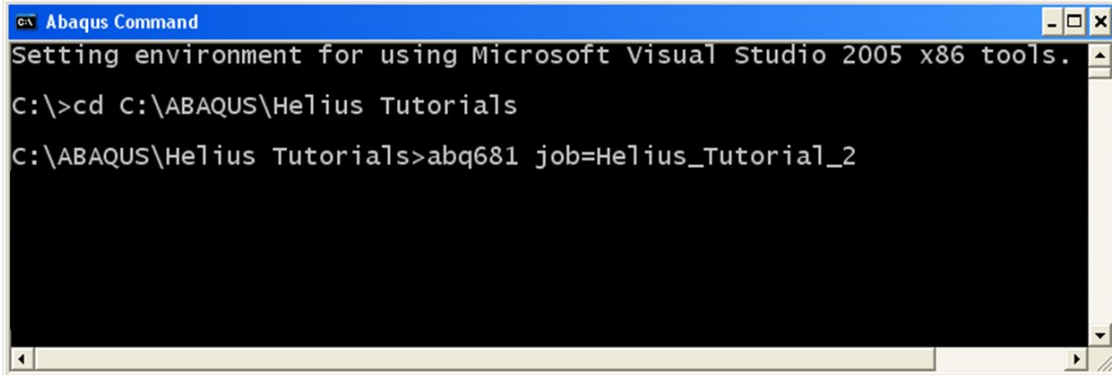
3. Submit the job. The format of the job submission depends on an option selected during the installation of Helius:MCT:
 - a. If ‘Yes’ was selected when asked if you use user subroutines, the following line must be added to the Abaqus environment file:

```
usub_lib_dir='C:\ABAQUS\Firehole\bin_68' (for v6.8 users)
```

```
usub_lib_dir='C:\ABAQUS\Firehole\bin_67' (for v6.7 users)
```

The job can then be submitted by typing “**abq681 job=[name of input file]**” as shown in Figure 2 where “C:\ABAQUS” is the ABAQUS root directory.

- b. If ‘No’ was selected when asked if you use user subroutines, then the job can be submitted by typing “**abq681 job=[name of input file]**” as shown in Figure 2.



```
Abaqus Command
Setting environment for using Microsoft Visual Studio 2005 x86 tools.
C:\>cd C:\ABAQUS\Helius Tutorial1
C:\ABAQUS\Helius Tutorial1>abq681 job=Helius_Tutorial_2
```

Figure 2: Command prompt

4. After the finite element simulation is complete, refer to section 2.11 of Tutorial 1 for instructions to view and interpret the results.

3 Summary

In the 2nd tutorial, an ABAQUS input file was modified to make it compatible with Helius:MCT. Four sections of the input file required modification:

1. *Shell Section
 - a. Specified poisson, thickness modulus, transverse shear, and hourglass stiffness parameters
 - b. Changed the material names to match the name of the Helius:MCT material
2. *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
3. *Step
 - a. Turned extrapolation off
 - b. Adjusted the time incrementation
 - c. Modified the default nonlinear solution controls
4. *Element Output
 - Requested state variable output to the database