How to get the orientation tensors in .xml or .ele format from Moldflow results.

Software *: DIGIMAT*
Release *: 3.1*
Keywords *: Orientation tensors, Moldflow*

Moldflow orientation tensors (OT) can be used in DIGIMAT to CAE (ABAQUS, ANSYS, LS-DYNA) analysis to provide information on the fiber orientation in each element of the structural mesh. The properties of the composite dramatically depend on this fiber orientation.

For each mesh element, DIGIMAT to CAE will find the corresponding OT in the results file of Moldflow and will use it in the homogenization process to compute the stress/strain response of the composite in the element.

Two kinds of OT can be distinguished:

- *the OT computed in solid injection mesh :* the format of OT file supported by DIGIMAT to CAE for solid element is *.xml*. This file contains one OT by element mesh. Be careful, Moldflow 3D can provide *.xml* files that contain the full OT for each element or only the principal directions of the OT. DIGIMAT needs the full definition of the OT and not only the principal directions!
- *the OT computed for shell injection mesh (Moldflow/Midplane) :* DIGIMAT to CAE supported format is the *.ele*. A shell element is composed of several layers (default in Moldflow is 12 but can be increased up to 20) and Moldflow provides one OT file (that contains one OT by mesh element) at each interface between two layers, one file at the top of the shell and one at the bottom of the shell. For 12 layers, you will then have 13 files with *.ele.xxx* as extension where *xxx* goes from 1 to 13.

In the next section we detail all the steps needed to get the *.ele.xxx* and *.xml* files from Moldflow computation.

<u>Moldflow / Midplane (*.ele.xxx*)</u>

1. Run a Moldflow/Midplane analysis with the ABAQUS interface option (see Figure 1):
    - Open the 'Process Settings Wizard".
    - Click on "Advanced options".
    - Edit "Solver parameters" options.
    - Go to the "Interface" tab.
    - Select the "MPI-ABAQUS 6.3" interface.

e-Xstream engineering
e-mail : info@e-Xstream.com

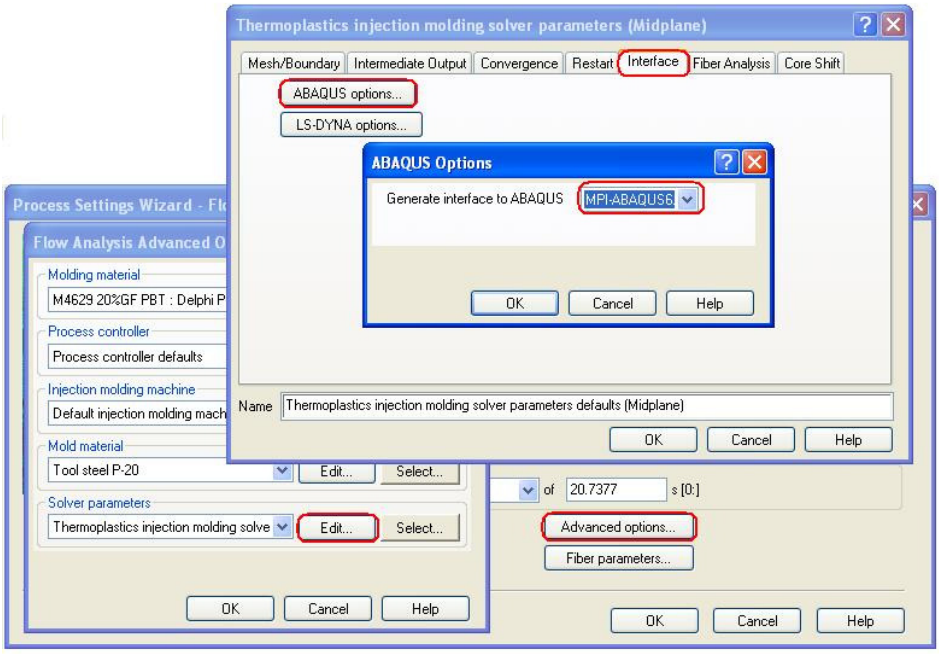

Figure 1: Select the ABAQUS interface before running the Moldflow/Midplane analysis.

After running the Moldflow/Midplane analysis, you should have a *JobName.osp* file in your results. This file contains the thermo-mechanical properties of the Moldlfow material and the residual stresses computed by Moldflow. But other files are needed in order to run the coupled analysis in ABAQUS.

2. Export the mesh in Patran format (needed by ABAQUS) :
   - *File Menu -> Export.*
   - Choose the Patran format and save the file *: JobName.pat.*

3. Export the fiber orientation tensor results into 'ele' files (needed by DIGIMAT) :
   - Select : *Fiber orientation tensor result.*
   - *Results Menu -> Save Plot Data in Patran Format.*

MOLDFLOW (*.xml*)

1. Run a Moldflow/3D analysis
2. Export the Moldflow results to ABAQUS: run the macro *stinft3d.vbs*
   - *Tools -> Play Macro:* choose strinft3d.vbs.

   By running this macro Moldflow will create the following files:

   - *JobName.pat* : mesh in PATRAN format.
   - ***JobName_fiber_tensor.xmli*: fiber orientations expressed in global axes.**
   - *JobName_E11.xml, ... JobName_G23.xml* (9 files): mechanical properties based on the orthotropic assumption.
   - *JobName_ltec_1.xml, ... JobName_ltec_3.xml* (3 files): thermal expansion coefficients based on the orthotropic assumption.

e-Xstream engineering
e-mail : info@e-Xstream.com

- *JobName_volumetric_shrinkage.xml* : volumetric shrinkage used to compute initial stresses.

For a Moldflow/3D - DIGIMAT - ABAQUS/Standard analysis, only 3 files are useful, the PATRAN file, the orientation file and the volumetric shrinkage file.

Moldflow 6 :

When using Moldflow 6 you can run another Moldflow macro: *mpi2abq.vbs.*

- *Tools -> Play Macro*: choose mpi2abq.vbs.

This macro will create 2 extra files:

- *JobName_principal_directions.xml* : principal directions of the orientation tensor, used to define a user orientation in ABAQUS.

  !! Not usefull for DIGIMAT to CAE !!

- *JobName_initial_stress.xml* : initial stress for the ABAQUS analysis.

3. Export the mesh in Patran format (needed by ABAQUS) :
   - *File Menu -> Export.*
   - Choose the Patran format and save the file *: JobName.pat.*

e-Xstream engineering
e-mail : info@e-Xstream.com