

## Tutorial: Converse + OptiStruct

This tutorial was written and developed by Altair Brazil.



The purpose of the Converse software is to map the equivalent properties of the fibers to a finite element model (eg: RADIOSS\OptiStruct), using the fiber orientation generated by an injection analysis (eg: Moldflow, Moldex3D).

This tutorial shows how to convert results from Moldflow (.pat and .ele files) to the RADIOSS\OptiStruct (.fem). Finally, for each element, a property with equivalent characteristics of fiber orientation is going to be generated.

### Mapping Capabilities (OptiStruct)

Injection Molding Solver => Structural Solver

Solids => Solids (tets, hex)

Shells => Solids (tets, hex)

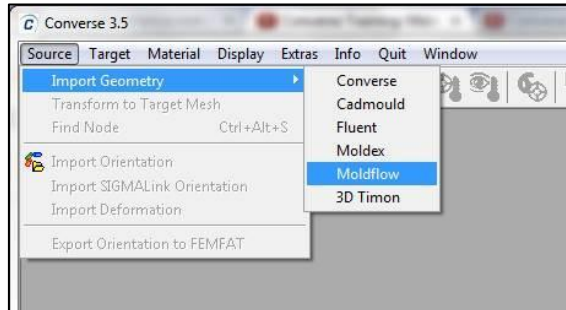
Both, first and second order elements are supported.

	Cadmould	Moldex	Moldflow	Sigma	Simpo	3D Timon	Fluent	Converse Data Exchange Format
Fiber Orientation	x	x	x	x	x	x		x
Shrinkage & Warp	x		x					x

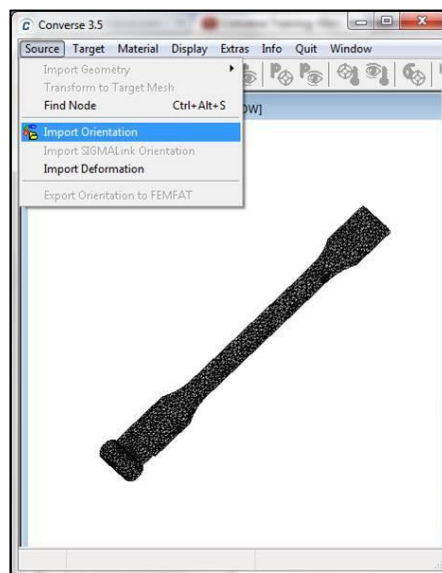
Required number of HWUs: 50HWUs

- **Converse:**

1. Import the CFD simulation file:  
➔ Source > Import Geometry > Moldflow  
(File: *Tensile\_Bar.pat*)

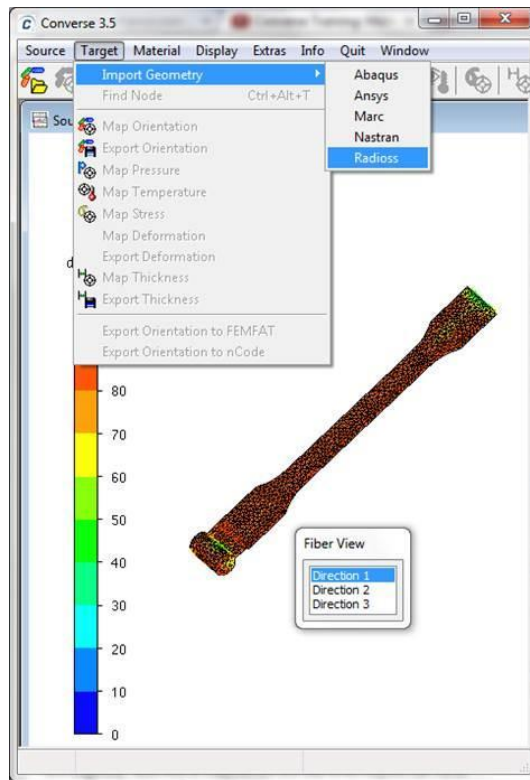


2. Import the fiber orientation file:  
➔ Source > Import Orientation.  
(File: *Tensile\_Bar\_orientation.ele*)



3. Import the target model in which the fiber orientations have to be transferred :  
➔ Target > Import Geometry > Radioss.  
(File: *Tensile\_Bar\_OS.fem*)

NOTE: The model can be imported with the boundary conditions already set, so when export the final file, the analysis will be ready to run.

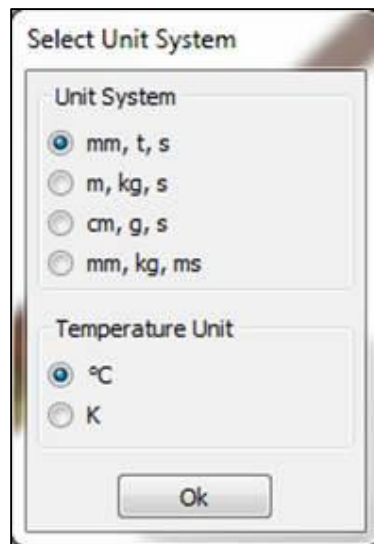


NOTE: If the elements are not displayed:

➔ Display > Geometry > Show element Edges.

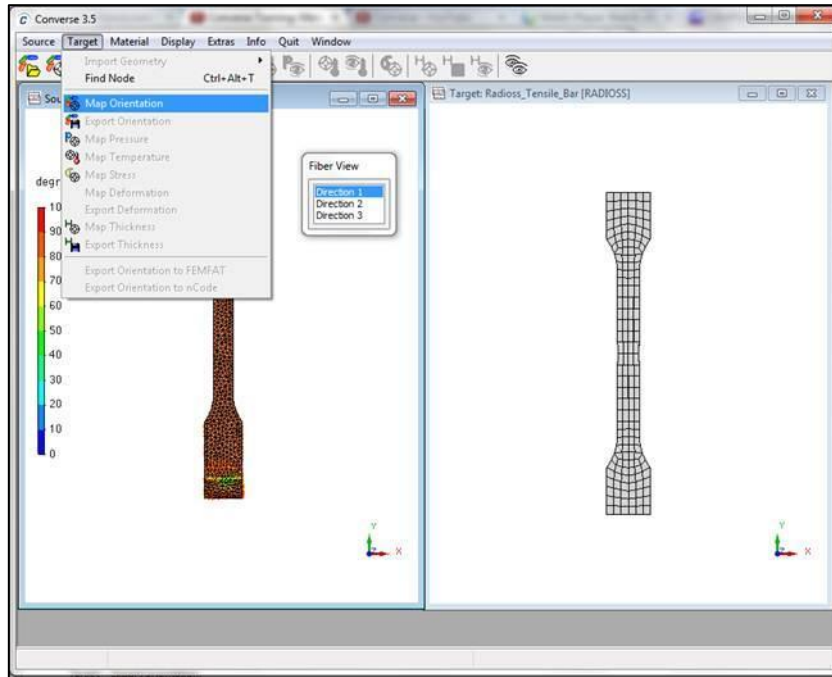
4. Automatically a window to select length and temperature units will be open.

➔ Set Unit System: mm and Temperature Unit: ° C.



5. Map the fiber orientations:

➔ Target > Map Orientation.

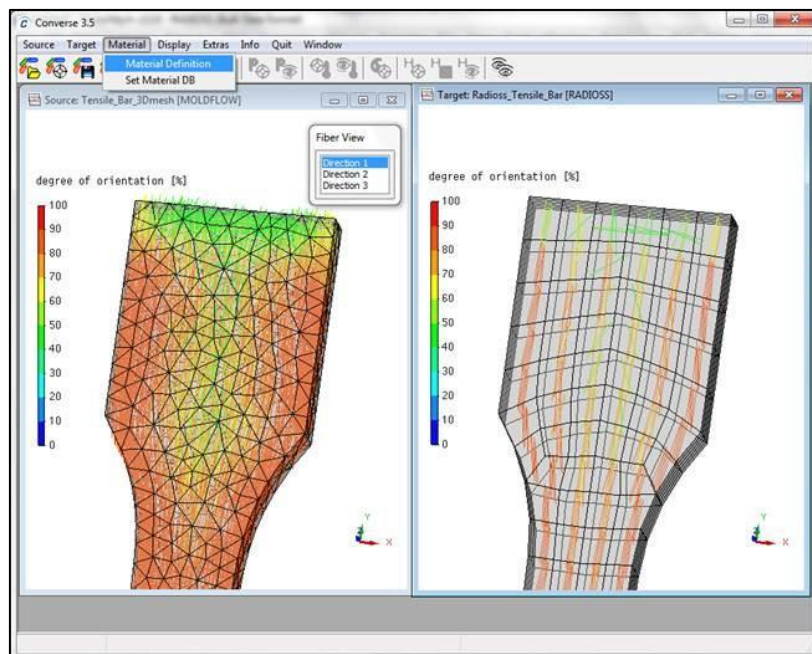


NOTE: If the orientation is not displayed:

➔ Display > Show orientation.

6. Set the material properties for the plastic components in the target model:

➔ Material > Material definition.

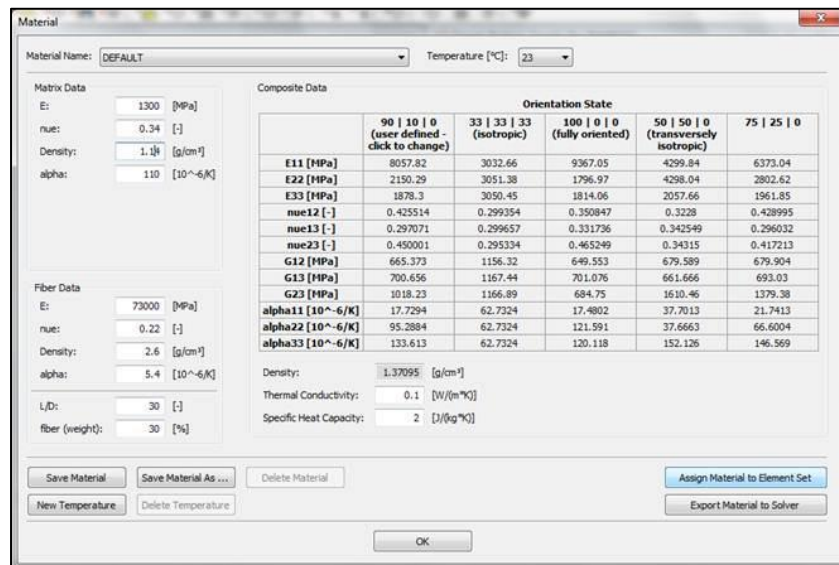


7. Modify the material settings and assign material to element set :

➔ change E: 1300 and density: 1.14

➔ To assign the material defined click “Assign Material to Element Set” and select the set of elements.

NOTE: You can save the properties defined in 'Save the Material'.



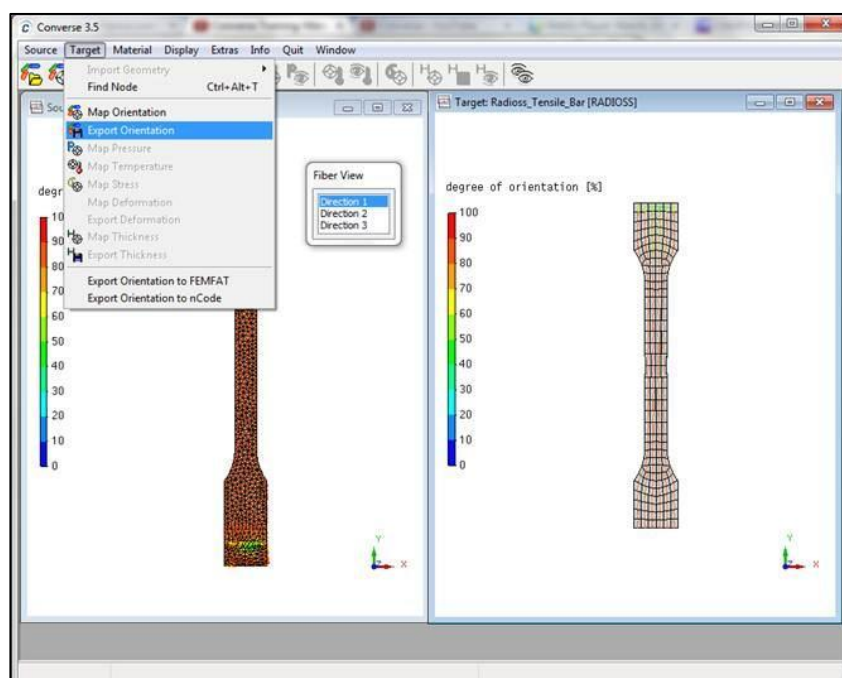
NOTE: You can also select the material in a database of materials, if it is already set:

➔ Material – Set Material DB.

8. Export to solver:

➔ Target – Export orientation

➔ The file *tensile\_bar\_cof.fem* will be exported



Note: This exported file is a complete file for RADIOSS\OptiStruct which contains all the properties of existing elements related reinforced components and properties of anisotropic materials. It contains the anisotropic materials settings and local coordinate systems for each element.

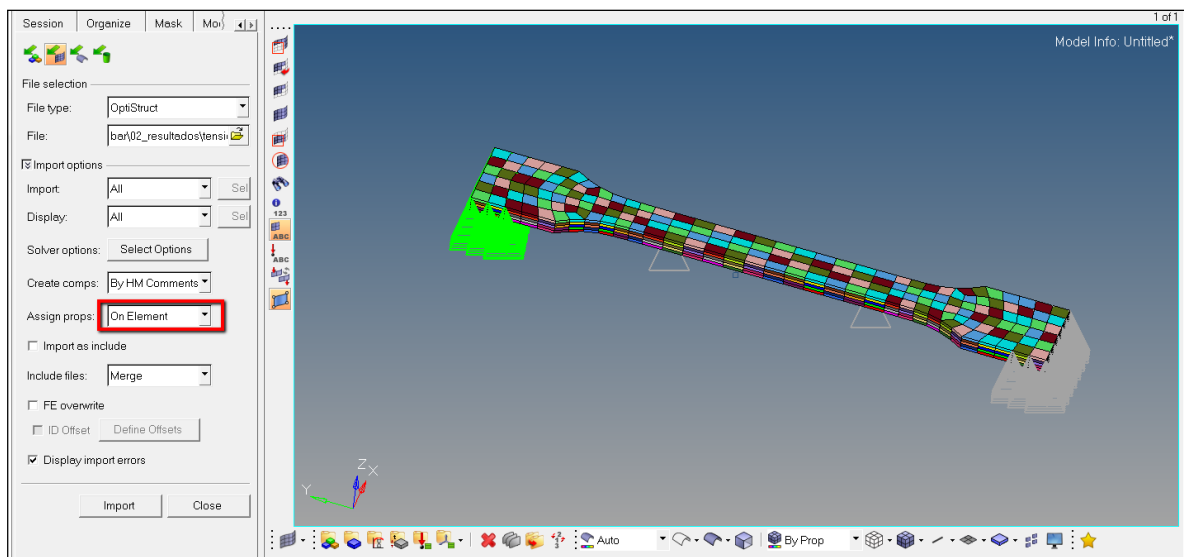
- **HyperMesh\OptiStruct:**

1. Import the model:

(File: *tensile\_bar\_cof.fem*)

- ➔ File Import > Solver Deck
- ➔ In Assign props select 'On Element' option.
- ➔ Click Import

Note: To view the model according to the elements property, change the 'element color mode' to 'by prop'.



2. Run the analysis and see the results:

- ➔ Click Analysis > OptiStruct  
(export options: all – run options: Analysis – Memory options: Default)
- ➔ Click OptiStruct

NOTE: Remember that the model used for mapping the fibers was already configured to analyze. If it is not already set, would be necessary to set the load collectors and load steps before performing the analysis.

→ At the end of analysis, click HyperView to plot the results:  
(File: tensile\_bar\_cof.h3d)

