OpenFOAM® and Tecplot Post-Processing

| **Dipl.-Phys. Jens Warkall**, | Genias Graphics GmbH & Co. KG |
| jens.warkall@genias-graphics.de | DE-93128, Regenstauf, Germany |

Abstract

Tecplot, Inc., in conjunction with OpenCFD®, has created a foamToTecplot360 converter for OpenFOAM® v1.6 simulation results. Users report that the converter works with OpenFOAM® v1.5-dev results as well.

The converter will work with the following OpenFOAM® results:
- static mesh, multiple time steps
- moving mesh (geometry only changes)
- changing mesh (geometry and topology changes)
- polyhedral cell support
- point values (internal fields)
- point values (boundary fields)
- cell values (internal fields)
- cell values (boundary fields)
- multiple sets of lagrangian data ('clouds') with fields
- outputting a mesh with multiple regions

Access to the converter is located at the OpenFOAM® 1.6.x git repository:

http://www.opencfd.co.uk/openfoam/download.html

Tecplot 360 gets you answers faster, helping you to quickly plot and animate your data exactly the way you want. Using just one tool, you can analyze complex data, arrange multiple layouts, and communicate your results with professional images and animations. Of all the CFD visualization tools available, Tecplot 360 is easiest to use and offers the most visually powerful output. http://www.tecplot.com/products/360/index.aspx

The OpenFOAM® community reports that typical CFD simulations are run on large parallelized machines resulting in multiple solutions (from multiple nodes). Tecplot 360 and the foamToTecplot360 converter can be easily scripted to combine these multiple solutions into one application for visualization and post-processing.

Please contact us to learn more about this process, and how it is accomplished.

Key words: OpenFOAM®, Workshop, Tecplot, Converter, Post-Processing, foamToTecplot360