FocusOn

Showing Helpful and Unique Features in Pointwise®

Summer 2009

Pointwise Flexibility Saves Time

Pointwise writes native files for ANSYS FLUENT®, ANSYS CFX®, STAR-CCM+®, OpenFOAM® and many other CFD packages. This includes both grid and boundary conditions. You can build a mesh and identify boundary condition regions in Pointwise, then easily export it to many different CFD solvers.

Using one preprocessor that can work with many different solvers saves you time. You only have to go through the learning curve one time. Once you have learned the preprocessor, you can keep using it for all your flow solvers. Maybe you have one flow solver that is particularly good for multi-phase flows or combustion, but prefer to use a different flow solver for general fluid flow and heat transfer problems. No problem. Once the grid is made in Pointwise, you can export it in the native format of each flow solver in a matter of minutes.

This flexibility can be useful to you in several ways, but before we get to them let's discuss the CFD analysis process in general. There are three main steps in a CFD analysis: preprocessing, solving, and post-processing. People seem to spend most of their time in the preprocessing stage, and for good reason since grid quality is the single biggest influence on the results over which you have control. You normally don't change the numerics or models included in the flow solver, and once you get to the post-processing stage the solution is determined already. But the quality of the grid directly affects the accuracy of your CFD solution, and it affects how quickly the CFD solution will converge. In fact, if grid quality is bad enough it can keep the CFD solver from converging at all. So, it makes sense to emphasize grid quality.

Pointwise writes the native files of more than 30 CFD solver packages, as shown in Table 1. It also supports several de facto standard grid formats for interfacing with a variety of CAE tools as shown in Table 2.

In its next release, Pointwise will support CAE interface plug-ins that will let you develop your own CAE interfaces and have them appear in Pointwise. This includes both boundary conditions and mesh. In fact, the OpenFOAM, AcuSolve and USM3D interfaces have been implemented using the plug-in approach. Details of the plug-in development process, including a complete application programming interface (API) definition, will be made available to users at that time. The plug-in approach is ideal for engineers, researchers and academics using custom in-house flow solvers.

Pointwise gives the flexibility to use any flow solver without having to learn how to use multiple preprocessors. In this way, users can easily run the most physics-appropriate CFD package

Table 1: Pointwise writes native files for more than 30 CFD solvers

- AcuSolve
- CGNS
- FDNS/UNIC
- OpenFOAM
- STAR-CCM+
- VSAERO
- ADPAC
- CNSFV
- FrontFlow
- Overflow
- STAR-CD
- XPATCH
- ANSYS CFX
- COMO
- GUST
- PHOENICS
- TACOMA
- ANSYS FLUENT
- CRUNCH
- INCA V2
- SCRYU
- TEAM
- CFD++
- FALCON
- NCC
- SCRYU/Tetra
- TETREX
- CFDShip-Iowa
- FANS
- NPARC
- Splitflow
- USM3D

213 South Jennings Avenue Fort Worth, Texas 76104-1107 Toll-free 800-4PTWISE
Tel (817) 377-2807 Fax (817) 377-2799 sales@pointwise.com www.pointwise.com

POINTWISE®

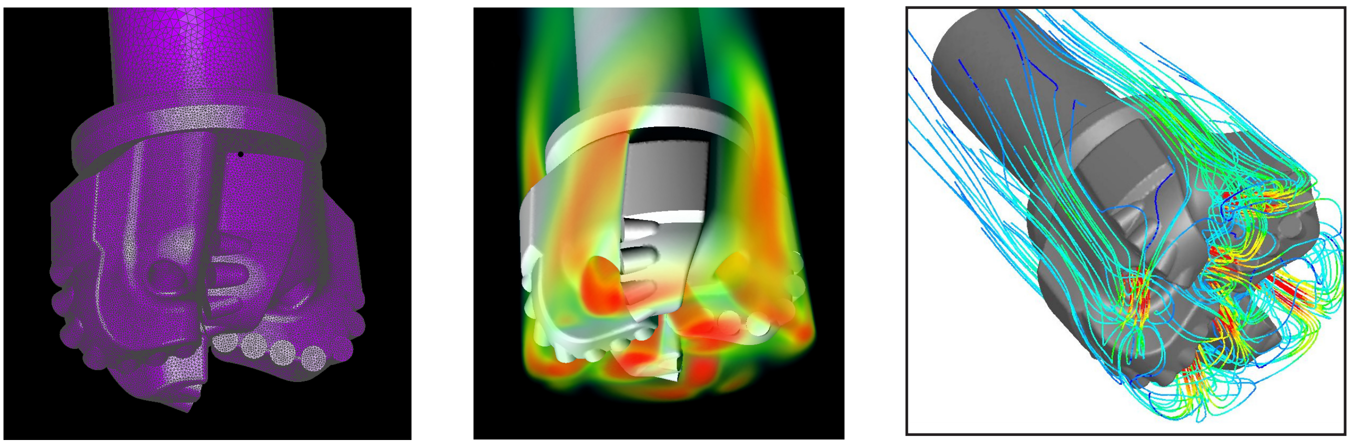
for each problem, run CFD solutions in multiple solvers for benchmarks, and run one-of-a-kind solutions in a different flow solver.

In addition to the time saving advantages of using the same grid and the same preprocessor for CFD analysis, Pointwise produces high quality grids that improve flow solver accuracy and convergence. That is what Reliable CFD Meshing is all about.

Check out www.pointwise.com/pw/ for the latest information about Pointwise and the currently supported flow solvers.

Table 2: Standard formats provide grid interfaces to additional CAE tools

- FIELDVIEW UNS
- STL
- NASTRAN
- UCD
- PATRAN
- VRML
- PLOT3D



Pointwise was used for the mesh, left, in this drill bit simulation. ANSYS® FLUENT®, center, and OpenFOAM® were used for the flow.

Toll Free (800) 4PTWISE www.pointwise.com

Reliable CFD Meshing

RETURN SERVICE REQUESTED

First Class
Presort
US Postage
PAID
Arlington, TX
Permit 419

213 South Jennings Avenue
Fort Worth, Texas 76104-1107