

April, 2010: DHCAE Tools announced that the latest version of the company's meshing and preprocessing tool CastNet 2.2 is available now. The current release supports new features, which increase the functionality and usability for OpenFOAM®.

CastNet is a preprocessing tool using CAD kernel geometry (Parasolid, Acis or Granite) and discrete mesh data for input. CastNet provides various direct case definitions for typical CFD and FEA solvers and focuses on the model setup for the open source CFD tool box OpenFOAM®: CastNet enables the user to define the complete model input for more than 15 OpenFOAM® solvers at the graphical user interface. Solver settings, fluid properties and boundary conditions (including interior zones such as porous and MRF regions) are defined in association with CAD geometry. CastNet generates anisotropic hybrid meshes (including extrude and prism layer regions) on the basis of the advanced Simmetrix Inc. meshing and abstract modelling technology. Also, CastNet supports OpenFOAM® meshing features, whereas the complete parameter definition remains associated with CAD geometry entities.

CastNet 2.2 includes new functions to increase usability and simplify the case setup for OpenFOAM® 1.6: A new warning section is implemented giving warnings during the model export in case the latter is incomplete or, with regard to the chosen OpenFOAM® solver, not well-defined. Furthermore, version 2.2 contains an additional graphical user interface. This new application completes the graphical process control and model setup for OpenFOAM®: The new GUI allows a detailed job control during OpenFOAM® calculations. This includes plotting residuals, changing time control settings, switching matrix solvers or discretisation schemes.

Moreover, CastNet 2.2 supports new solvers and provides additional features for OpenFOAM® 1.6.(x).

With CastNet 2.2 DHCAE Tools constitutes an essential step forward providing access to the strong solution capabilities of OpenFOAM® without the need of defining the model setup or controlling the operation via text interfaces.

The GUI approach presents an additional option to set up models for users without expert knowledge of the command and keyword structure in OpenFOAM®.

For more information, please go to:

[www.dhcae-tools.com](http://www.dhcae-tools.com)

OPENFOAM® is a registered trademark of OpenCFD Limited

*This offering is not approved or endorsed by OpenCFD Limited, the producer of OpenFOAM software and owner of OPENFOAM® and OpenCFD® trademarks.*