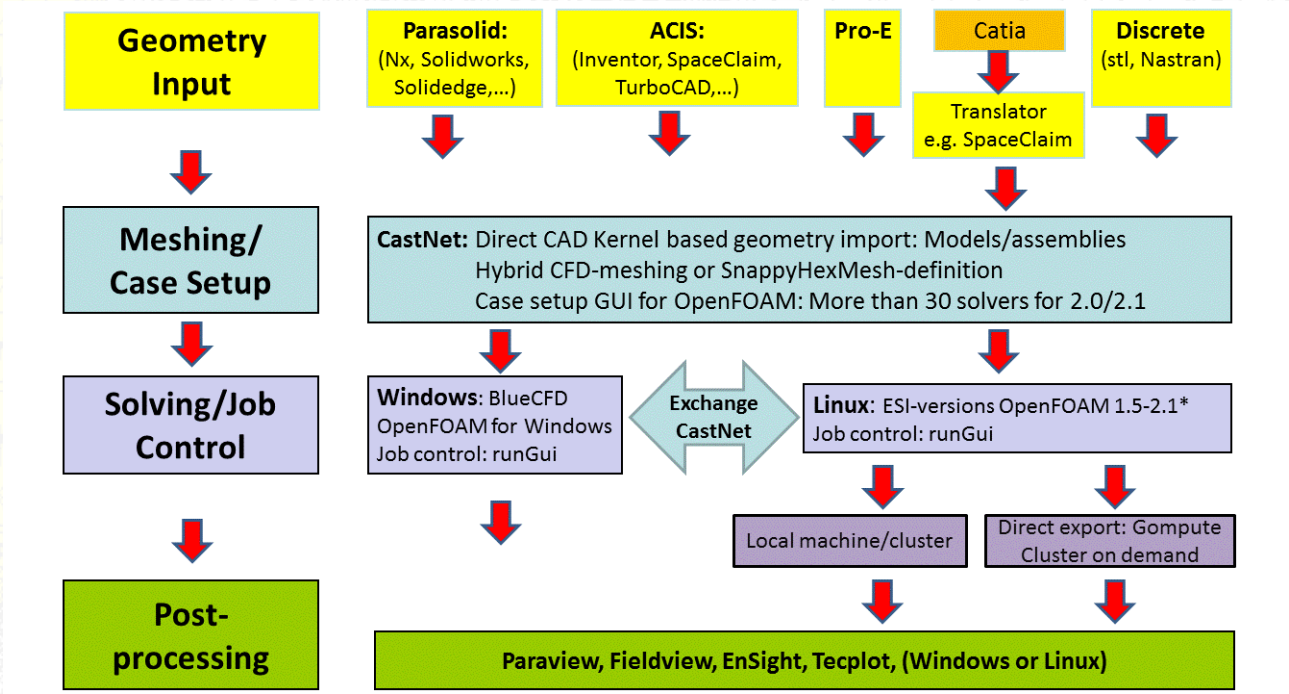
CastNet: GUI environment for OpenFOAM®

CastNet is a preprocessing system and job-control system for OpenFOAM®. CastNet works with the standard OpenFOAM® releases provided by ESI Group as well as ports for Windows based on OpenFOAM® technology.

The outstanding features of the CastNet modeling environment are:

- Providing access to reliable, robust and high quality CFD-meshing based on CAD geometry
- Establishing a complete GUI based environment: Access to strong OpenFOAM®-solution capabilities without editing text files or detailed knowledge of keyword-structure
- Reducing the time from CAD model to OpenFOAM® ready-to-run case
- Allowing a reliable and stable CFD analysis by detail job control
- Supporting a flexible complete CFD environment for Windows and Linux including preprocessing, solving and post-processing

The following chart demonstrates the typical workflow from CAD to results with OpenFOAM® using the CastNet modeling environment:



CAD model based input

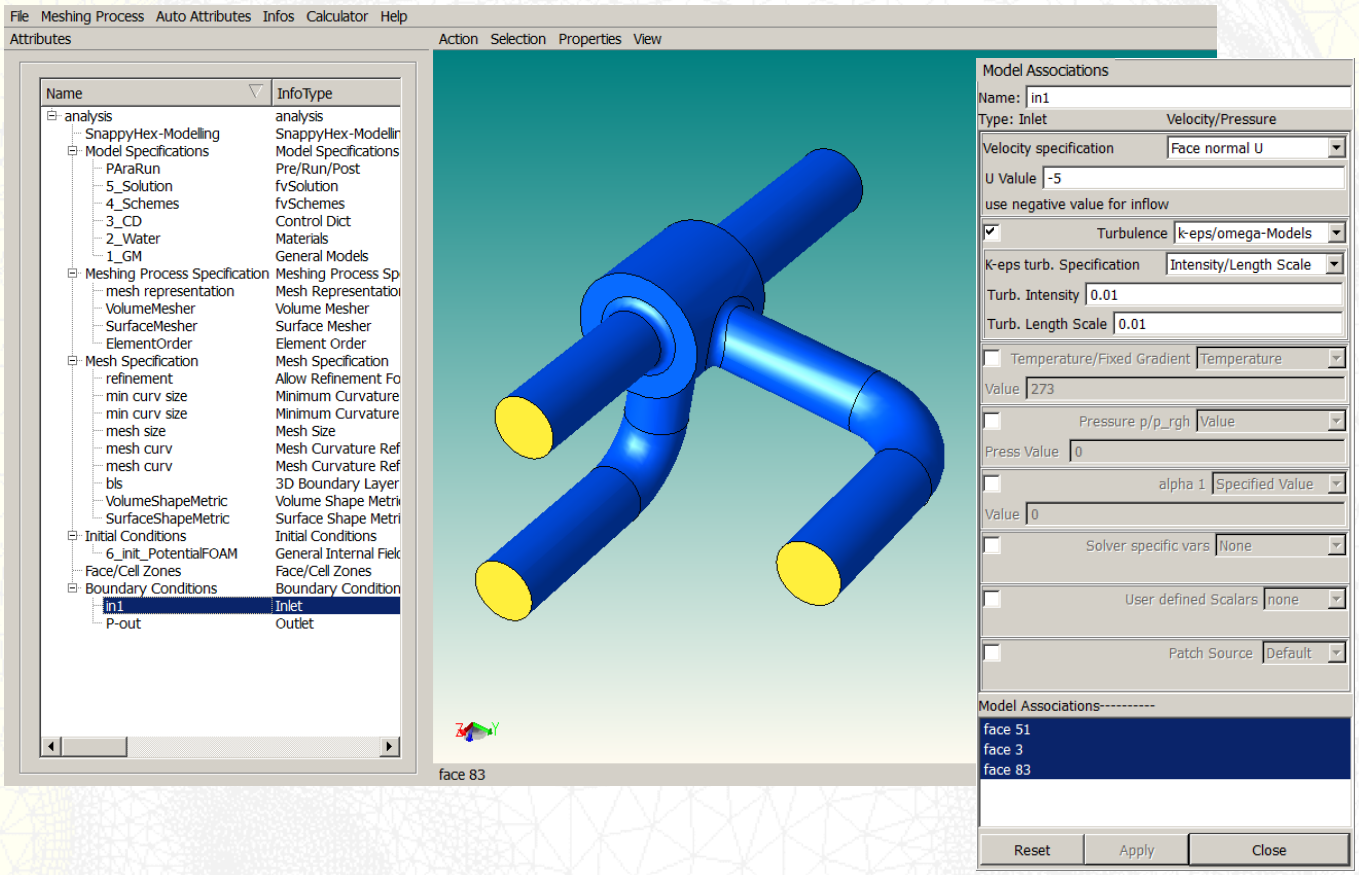
CastNet allows a high quality geometry import based on CAD kernel data. There is no need to repair the CAD geometry and volume information is preserved. The CAD import includes:

- Direct access to Parasolid, ACIS and Granite (ProE) parts and assemblies.
- CastNet generates internally a non-manifold model of complex assemblies resulting in conformal grids between the various regions (parts). This feature simplifies the case setup for region models, e.g. Porous, MRF, AMI rotating, CHT. Also different meshing approaches (e.g. free or extrude meshes) can be setup in the regions.
- Automated geometry improvement for meshing by suppression of sliver faces if needed.

The meshing and case setup GUI

The complete case setup including mesh definition, OpenFOAM®-solver setup and boundary conditions is done in a single graphical user interface.

Mesh specifications and OpenFOAM® settings (e.g. the velocity inlet conditions in the figure below) are defined with associations to the CAD geometry. The complete case definition is stored in a single data-file. Settings can be modified easily by re-opening the attributes and defining new parameters. Changing a flow rate for OpenFOAM® or refining the mesh locally is carried out with few clicks.

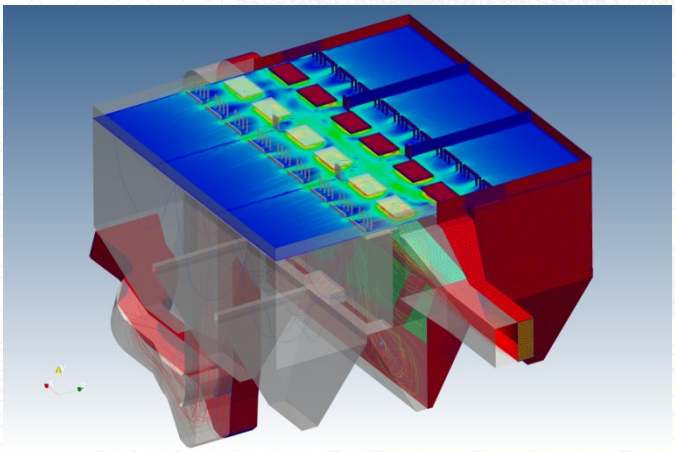


CFD-Meshing

CastNet supports two meshing approaches:

- The internal hybrid meshing with extrude meshes, free meshes and strong boundary layer capabilities and
- the external meshing with snappyHexMesh.

Both meshing approaches complement each other perfectly: The hybrid meshing is preferred if near wall effects are very important or a good geometry resolution is required. Stretched domains can be effectively meshed using extrude grids in CastNet's hybrid meshing approach. The snappyHexMesh meshes with polyhedral cells are preferred for free surface flows, time critical transient applications or very bad CAD input.

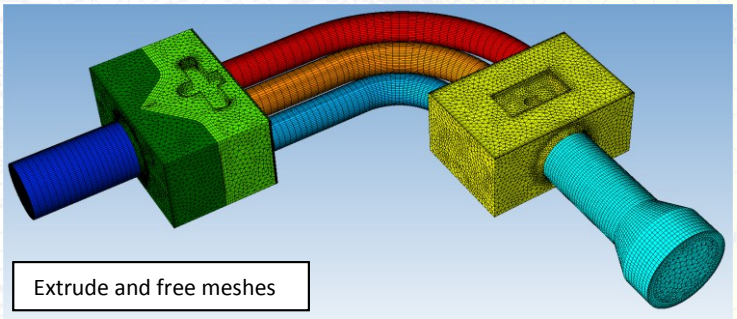


CAD model based meshing and case setup: Reliable CFD with CastNet and OpenFOAM® for complex geometry

CastNet hybrid meshing

In the hybrid meshing CastNet generates tetrahedral cells, combined with prisms in boundary layer regions and hexahedral cells (e.g. in extrude regions). The particular features in this meshing approach are:

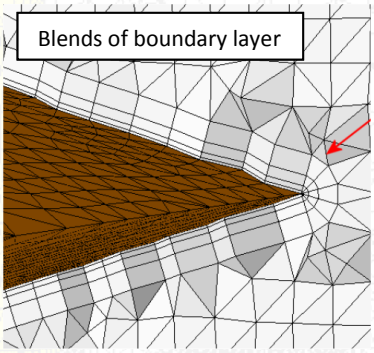
- Fast and high-quality mesh generation using relative element sizes and curvature control parameters
- Efficient mesh setup by combinations of extrude and free meshes.
- Full mesh control by means of local meshing parameters (element sizes, refinement boxes, boundary layer sizes)

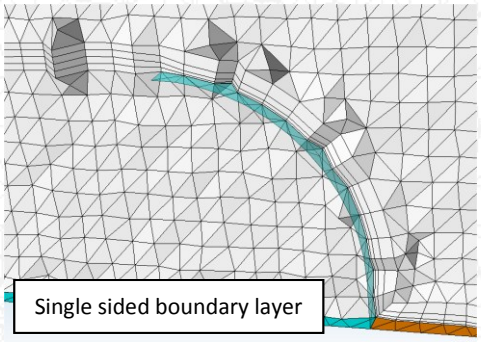


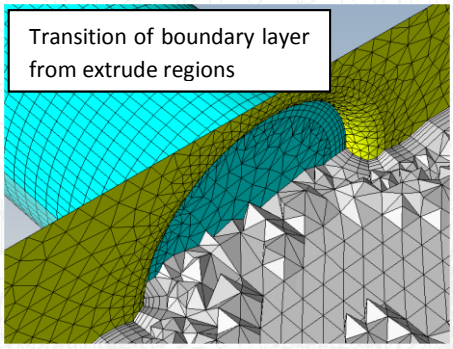
Mesh changes are quickly carried out: Changing the boundary layer thickness or setting refinement zones takes just one click.

Additionally CastNet provides advanced boundary layer features:

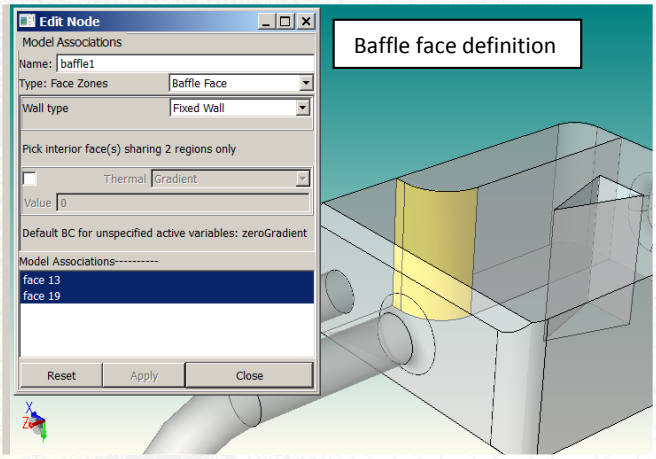
- Different types of boundary layer, e.g. first layer thickness, number of layer relative size …
- Blend options
- Single sided boundary layer for interior meshes (baffle faces, solid zones in CHT analysis)

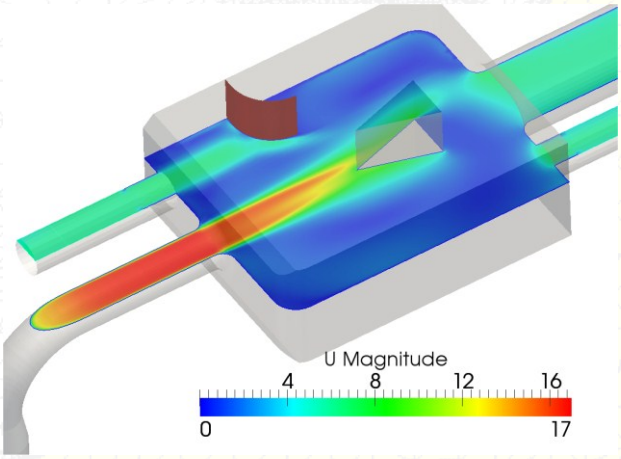






The CastNet meshing features allow an easy access to advanced OpenFOAM® functionality. These are in particular: Fan and baffle faces, regions (porous, MRF, AMI, CHT) and cyclic patches. Furthermore CastNet provides a special mesh improver for OpenFOAM® quality criteria. This results in an improved non-orthogonality of hybrid meshes.

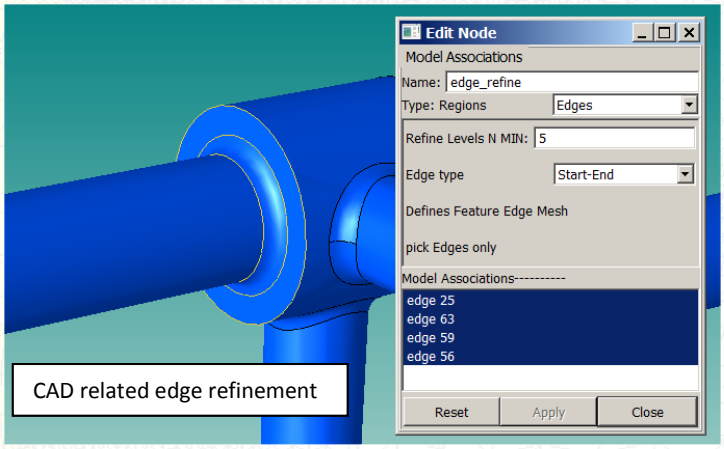




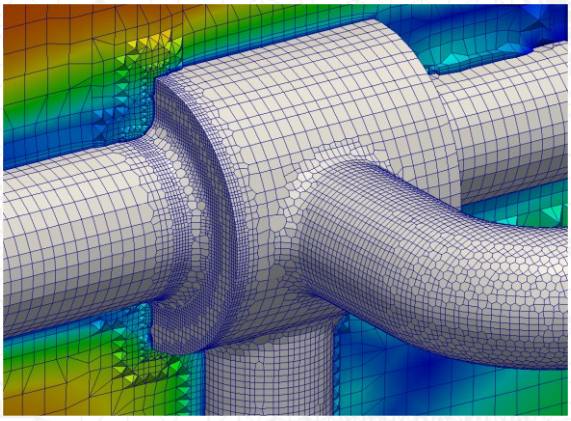
SnappyHexMesh support in CastNet

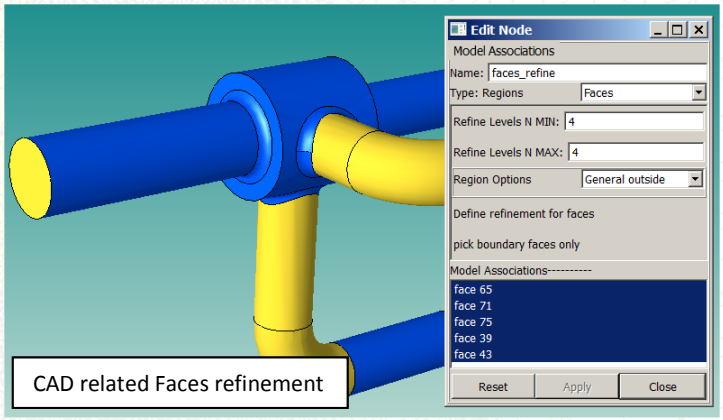
Additionally to the hybrid meshing approach CastNet supports snappyHexMesh, the script based meshing system in OpenFOAM®. The meshing setup for SnappyHexMesh is strongly simplified compared to manually editing text files. CastNet exports directly the snappyHexMesh-dictionary based on the GUI settings and CAD model selections. The meshing process is incorporated into the automated case setup of the complete OpenFOAM® case. The entire meshing and case setup is conducted in background. Following features are supported:

- Region support (e.g. porous or MRF) based on CAD geometry
- AMI support for sliding meshes (e.g. for mixer or turbo machinery)
- Feature edges with automated feature extraction
- Face and region refinement
- Boundary layer meshing
- CFD domain generation based on solid region input

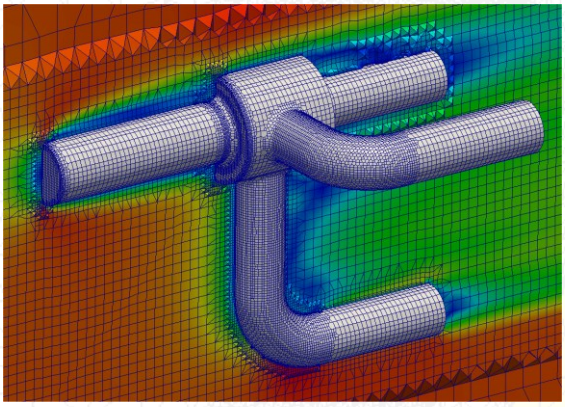


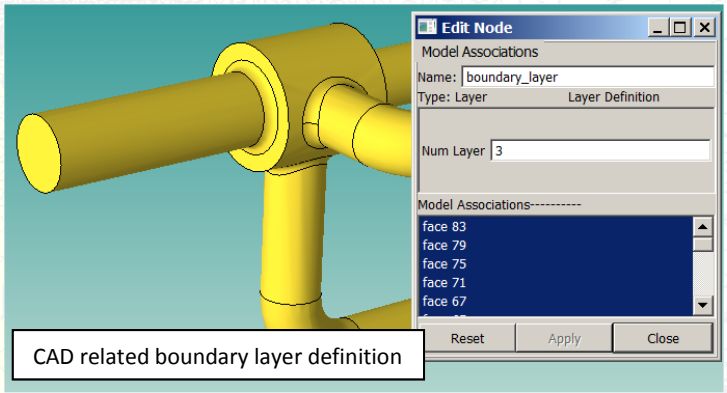
CAD related edge refinement



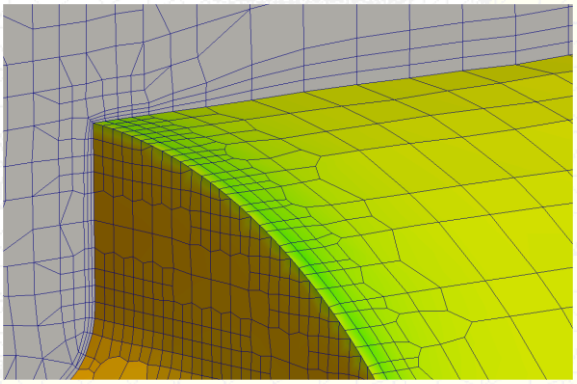


CAD related Faces refinement





CAD related boundary layer definition



OpenFOAM® support in CastNet

CastNet supports OpenFOAM® 1.5 - 2.1 ESI-Linux versions and the BlueCFD Windows version.

CastNet offers access to the following solver directly:

Incompressible solvers: simpleFoam, MRFSimpleFoam, porousSimpleFoam, pisoFoam, pimpleFoam, pimpleDyMFoam, SRFSimpleFoam, SRFPimpleFoam, icoFoam, nonNewtonianIcoFoam

Compressible solvers: rhoSimpleFoam, rhoSimplecFoam, rhoPorousMRFPimpleFoam, rhoPorousMRFSimpleFoam. rhoPorousMRFLTSPimpleFoam, rhoPimpleFoam, sonicFoam,

Heat transfer: buoyantSimpleFoam, buoyantPimpleFoam, buoyantSimpleRadiationFoam, buoyantBoussinesqSimpleFoam, buoyantBoussinesqPimpleFoam, chtMultiRegionSimpleFoam, chtMultiRegionFoam

Multiphase: interFoam, interDyMFoam, compressibleInterFoam, MRFInterFoam, porousInterFoam, LTSInterFoam, twoLiquidMixingFoam, bubbleFoam

Basic: laplacianFoam

Lagrangian: uncoupledKinematicParcelFoam, icoUncoupledKinematicParcelFoam

These solvers include the following modeling capabilities:

- Steady-state/transient analysis
- compressible or incompressible flows and media
- With or without heat transfer including radiation
- Multiphase flow (VOF/Euler-Euler/Lagrangian particle tracking)
- Dynamic meshes (sliding meshes or sloshing applications)
- Conjugate heat transfer analysis

CastNet provides a strong user support for OpenFOAM® during the case setup:

- CastNet gives warnings in case of certain setting are missing or do not agree with the standard OpenFOAM® solver requirements. E.g. if gravity is required by the selected solver but the definition is missing a warning will appear.
- CastNet defines the boundary by patch, not by field. There is no need for the user to define zero gradient conditions or special wall conditions for all fields explicitly.
- CastNet helps the user with many defaults in the background. E.g. CastNet selects the thermo-class automatically or sets up valid time loop parameter for the chosen solver. These defaults can be modified by the user in a configuration file.

Beside the standard solvers the user can also define solution settings and boundary conditions for custom solver and custom fields. The CastNet output is fully compatible to standard OpenFOAM® releases. Therefore the user can extend the output manually for own developments or not supported features.

Further CastNet functionalities for OpenFOAM® usage are:

- Simple configuration of settings for parallel runs.
- Probes definition and post processing output options.
- Fully automated setup for conjugate heat transfer and sliding mesh applications.
- Convenient definition of boundary conditions using derived OpenFOAM® patches.
- Easy and stable initialization with potentialFoam before the final run.
- Definition of local initial conditions, e.g. specification of volume fractions in particular regions.
- Support of baffle faces, fan faces, cyclic boundary conditions and two-dimensional modeling.
- Time dependent boundary conditions and support of groovyBC.
- Definition for incompressible (Newtonian/Non-Newtonian), multiphase or thermophysical materials.
- Second solver run for particle tracking.

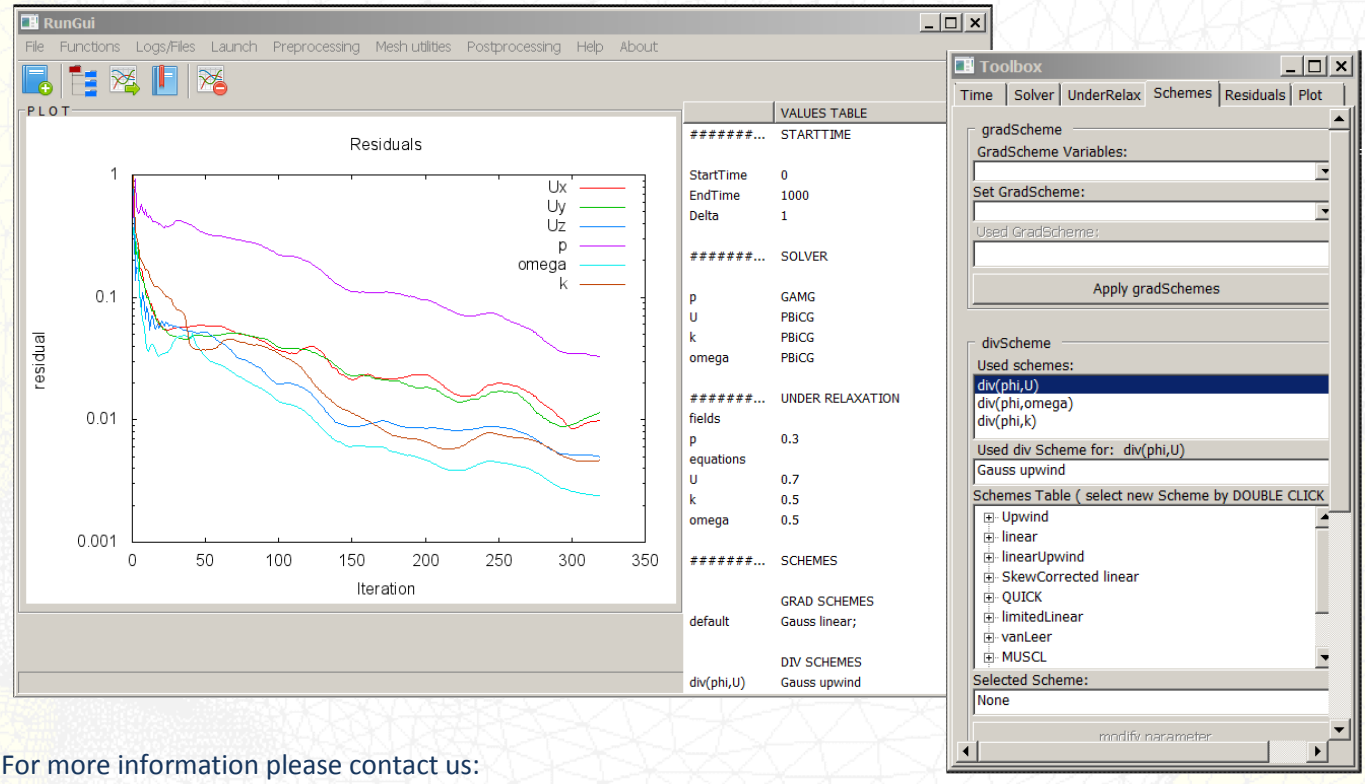
Job-Control System for OpenFOAM®: runGui

The runGui is a job control system for OpenFOAM®. It completes the GUI based environment for settings during runtime. The major goals for the runGui are:

- To enable a reliable CFD analysis by a detailed job control
- To facilitate the usage of OpenFOAM® features such as modifying solution settings during runtime
- To provide an easy access to OpenFOAM® meshing, pre- and post-processing features.

These goals are reached by following outstanding features:

- One click button for case setup from the CastNet output
- Job control by automated residual and probes plots controlled by CastNet settings
- Modifying solver settings during the run, e.g. by changing schemes and solution settings without stopping the solver
- Selectable automated stabilization of runs for rho-solvers
- Selectable automated *stable start* and *accurate end* by scheme modifications during the run
- GUI based OpenFOAM feature:
    - Mesh manipulations such as mesh extrusion, mesh scaling and wall refinement
    - MapField: Mapping of results from one case to another
    - Wall results such as wall shear stress, YPlus-values etc.



For more information please contact us:

DHCAE Tools GmbH

Address: Friedrich-Ebert-Str. 368, 47800 Krefeld Germany
Phone +49 2151 821493 Fax: +49 2151 821494
Website: www.dhcae-tools.com email: info@dhcae-tools.de