

## OpenFOAM® support in CastNet

CastNet supports OpenFOAM® ESI-Linux versions and BlueCFD Windows versions.

CastNet offers access to the following solver directly:

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

**Compressible solvers:** rhoSimpleFoam, rhoSimplecFoam, rhoPimpleFoam, rhoPorousMRFPimpleFoam, rhoPorousMRFSimpleFoam, rhoPorousMRFLTSPimpleFoam, rhoPimplecFoam, rhoPimpleDyMFoam, 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 modelling 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:

- The case setup is strongly simplified by solver templates to each application.
- CastNet gives warnings if certain settings are missing or do not agree with the standard OpenFOAM® solver requirements. If e.g. 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. CastNet selects e.g. the thermo-class automatically or sets up valid time loop parameters 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 solvers and custom fields. The CastNet output is fully compatible with 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