

## Release Notes for CastNet 2.3

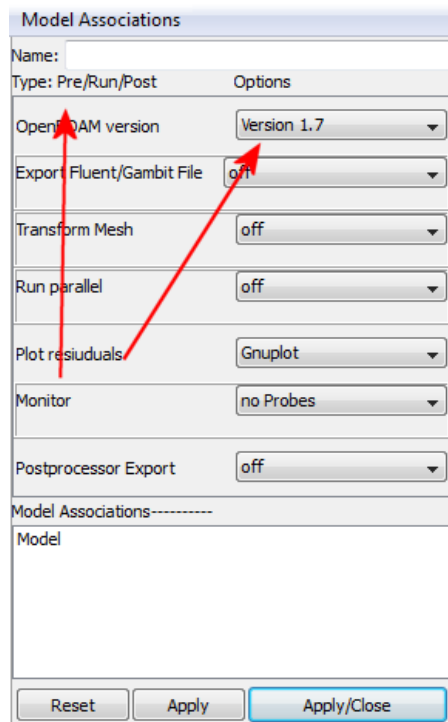
OpenFOAM® 1.7 .....	1
Working with references: .....	2
Generation of references .....	2
Usage of references: .....	5
Model generation of the conjugate heat transfer solver .....	5
Definition of a general face and cell zone set .....	6
User defined scalar fields.....	6
Generate a neutral gambit -out: .....	6

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.*

### OpenFOAM® 1.7

CastNet 2.3 supports OpenFOAM® 1.7: To allow an easy version change, OF 1.6 and 1.7 have been implemented into the OpenFOAM® 1.6-task:



By default, CastNet exports files for OpenFOAM® 1.6.x.

The user can switch to OpenFOAM® 1.7 export in the pre-run post options:

*Model/Specifications/PreRunPost-*

## Working with references:

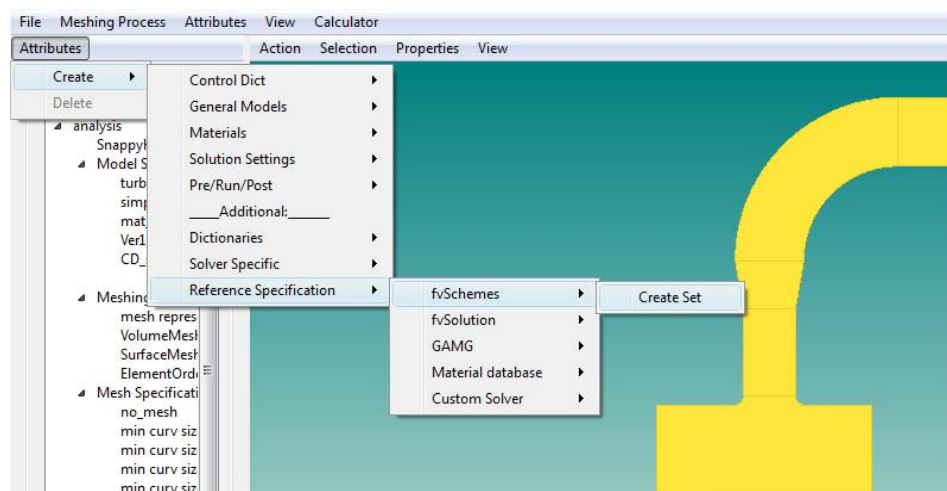
CastNet versions 2.3 and higher allow working with references. Generally, references are attribute definitions, which are not used directly, but can be assigned to other solution settings. At present, references can be defined for the following OpenFOAM® settings:

- fvSchemes
- fvSolution
- GAMG-settings
- Material data
- Custom-Solver

Typical applications for references are:

- By using the CHT solver, the user would like to assign solution settings locally for each region (e.g. fvSchemes, fvSolution). For this purpose, various fvSchemes and fvSolution can be defined in the reference settings and assigned to the CHT region in the course of the CHT zone definition.
- In general, references are suitable, if more than a single solution setup shall be stored in the same model-file (smf-file). The user can, for example, define fvSchemes sets like "accurate" or "stable" and activate them by selecting these schemes sets in the solution settings before meshing and exporting the case.
- Furthermore, references often allow much more entries than the „hard-coded“ settings in the solution setup. This, for example, gives the user access to considerably more schemes and scheme variants (e.g. strictly bounded schemes for scalars) than the solution settings.
- Working with references will offer particular advantages, if abstract modeling is used. When working with this technique, special reference attributes can be defined in a solver template (e.g. material data, fvSchemes, fvSolution etc.) and applied to multiple simulation cases.

## Generation of references



Selection-  
Select Model  
Model-  
specification  
needs to  
be active

*Attributes*  
*Create ->*  
*Reference*  
*Specification*  
*e.g.*  
*fvSchemes -*  
*Create Set*

**Model Associations**

Name: ref\_my\_schemes

Type: fvSchemes Create Set

Time scheme: Steady State

Grad scheme: Default Gauss Linear

Default Div Scheme: linear

Momentum Define Div Scheme

Div Momentum Name: div(phi,U)

Div Momentum Method: PreDefined Div Schemes

Div scheme: MUSLC

Vector formulation V: Use if possible

Turbulence k-eps/omega-Models

Div Scheme: Turbulence: k

Div k Name: div(phi,k)

Div k Method: PreDefined Div Schemes

Div scheme: LinearUpwind

Gradient Limiter: Cell limited Gauss Linear

Limiter: 1

Div Scheme: Turbulence: Epsilon/Omega

Div Eps/Omega Name: div(phi,epsilon/omega)

Div Eps/Omega Method: PreDefined Div Schemes

Epsilon/omega div scheme: Upwind (1.Order)

Model Associations-----

Model

Reset Apply **Apply/Close**

Reference for fvSchemes:  
Allows detail selection of fvSchemes

**Model Associations**

Name:

Type: fvSolution Simple

Use for Simple Solvers

Pressure solver p : GAMG

1. Abs. Tol. 2. Rel Tol. 3. Under Relax: 1e-06 0 0.3

U solver: Smooth solver

Abs./Rel./Relax: 1e-05 0 0.7

Turb solver: PBICG precond DILU

Abs./Rel./Relax: 1e-05 0 0.7

Heat Trans solver: Smooth solver

Abs./Rel./Relax: 1e-05 0 0.9

Heat Trans solver: Solver Name: H

Rho solver: None

Non orthogonal correctors: 0

N U Correctors: 0

P ref Cell / Value: 0 0

Limits: none

Options for 1.7: Pressure: p\_rgh

Model Associations-----

Model

Reset Apply **Apply/Close**

Reference for fvSolution:  
Allows detail selection of solution settings

**Model Associations**

Name:

Type: GAMG Specific settings

Please enter a name for the GAMG settings

Smoother Gauss Seidel

n Pre Sweeps

n Post Sweeps

n V Cycles

cache agglomeration true

n Cells In Coarsest Level

merge Levels

Model Associations-----

Model

Settings for GAMG can be defined and applied to the model (e.g. different GAMG settings for the various pressure solvers in Pimple-Foam or InterFOAM)

**Model Associations**

Name:

Type: Material database Material definition

Database Specification Thermophysical

Base type hPsiThermo

hPsiThermo is used by: rho\*, cht\*, buoyantSimpleFoam

Name

Number of Moles

Molecular Weight

hConstThermo janafThermo

Lower Temp Limit Tl

Upper Temp Limit Th

Common Temp Tc

High T a0..a1

High T a2..a3

High T a4

High T Enthalpy Offset a5

High T Entropy Offset a6

Low T a0..a1

Low T a2..a3

Low T a4

Low T Enthalpy Offset a5

Low T Entropy Offset a6

Transport Properties Sutherland Transport

As

Ts

P Ref

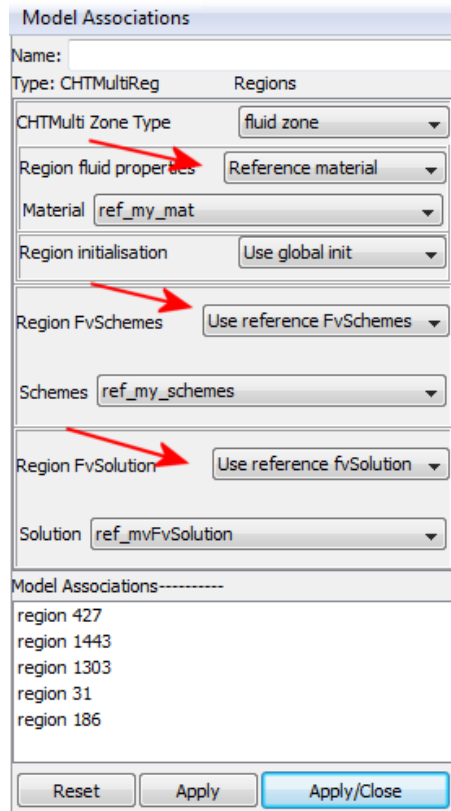
Default Air mixture

Model Associations-----

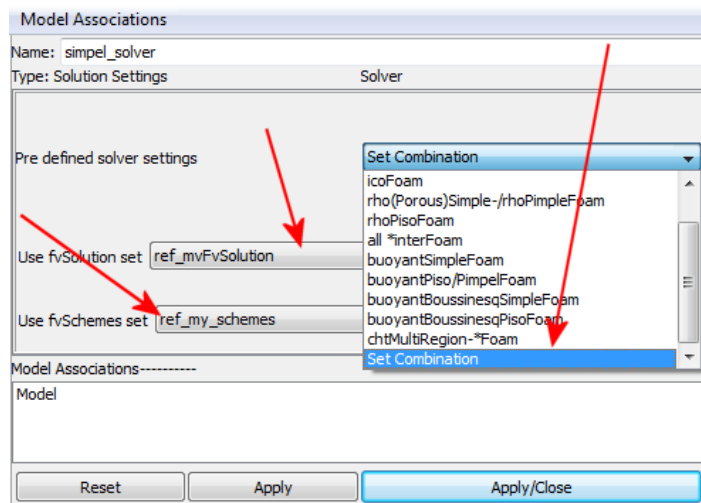
Model

Reference for material data:  
Allows more detail entries of material properties (e.g. temperature dependant viscosity (Sutherland parameter or heat capacity - Janaf parameter)

## Usage of references:



Defined reference attributes can be used in the CHT multi-region solver settings. The detail attribute settings for material properties can be assigned to different regions.

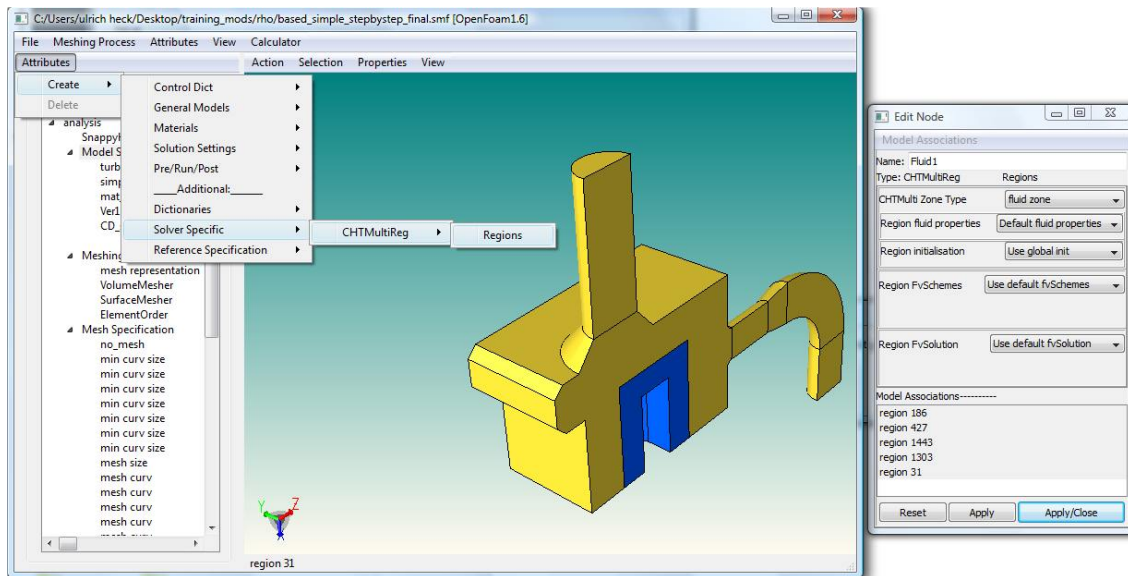


Under “solution settings” solution settings can be used for a set combination:

## Model generation of the conjugate heat transfer solver

Compared to the manually case setup, the setup for conjugate heat transfer problems is considerably simplified: Parts of a CAD assembly can be picked directly in CastNet and the fluid and solid region settings can be assigned directly. Local schemes, solution or material settings can be associated with the regions. The complete case and directory setup are conducted automatically.

The picture below shows a fluid region (named fluid 1) that consists of multiple geometry regions (see *Model Associations*).



OpenFOAM® CHT solvers allow you to assign individual solution settings to the regions. On the one hand, this makes the selected settings highly flexible with regard to the particular regions. However, on the other hand, a higher effort is needed to define these settings for each region. The principal modeling approach for CHT cases in CastNet tries to meet both of these requirements: A fast model generation with standard settings or a detail model setup where settings are individually assigned to the regions.

### Definition of a general face and cell zone set

Besides porous and MRF zones, it is also possible to define a general zone set, which can then be used for other purposes. An interior face set can be defined for particular needs.

### User defined scalar fields

The user can define own fields, which will be exported as files in the startTime-directory (0). Among other things, this will makes the setup for solvers with specific fields easier. For reactingFoam, for example, the user can set fields like H2 or O2 and then has to modify only a few boundary settings manually.

### Generate a neutral gambit -out:

Neutral file export is implemented in the OpenFOAM® 1.6 solver task. The user can export a Neutral file additionally or export the Neutral file only.

Note that Gambit Neutral file is a mesh import option for many CFD solvers (e.g. also Open source CFD solvers such as Code Saturne). This option gives the opportunity to run a problem with other CFD (open source) solvers, too, without needing to redefine the mesh.