Ulrich Heck, DHCAE-Tools UG

CastNet: GUI based modeling and solution control environment for OpenFOAM®

CastNet: CAD-based Pre-Processor for OpenFOAM®

Attributes: Concept of CAD associated mesh and solution set-up

Example: Model setup for CHT-Solver

OpenSource CFD Conference 2010
Munich, Germany

CastNet CAD input and meshing

- Modeling and meshing environment for CFD and FEA

- CAD-Model based (reads Parasolid -xt, Acis –sat or Granite Pro-E parts) or discrete mesh data (e.g. stl)

- Builds internally non-manifold model based on CAD assemblies with conformal mesh transition between parts

- Generates hybrid meshes

- CFD specific features (boundary-layer-meshing, curvature controlled meshing,…)

- CastNet is based on commercial meshing, CAD import and abstract modeling technology (not open source)

Meshing:

Curvature controlled
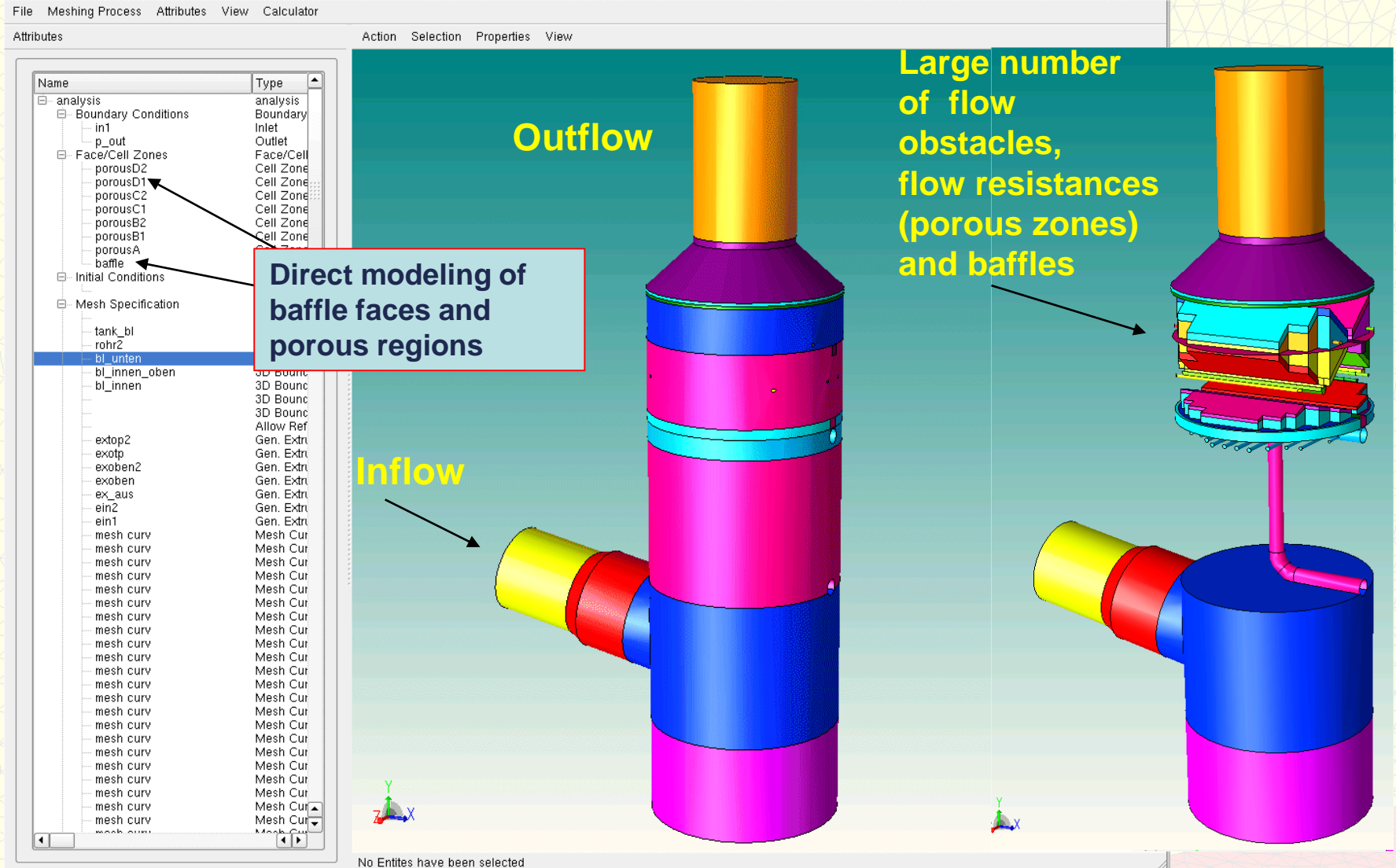
meshing with boundary

layer
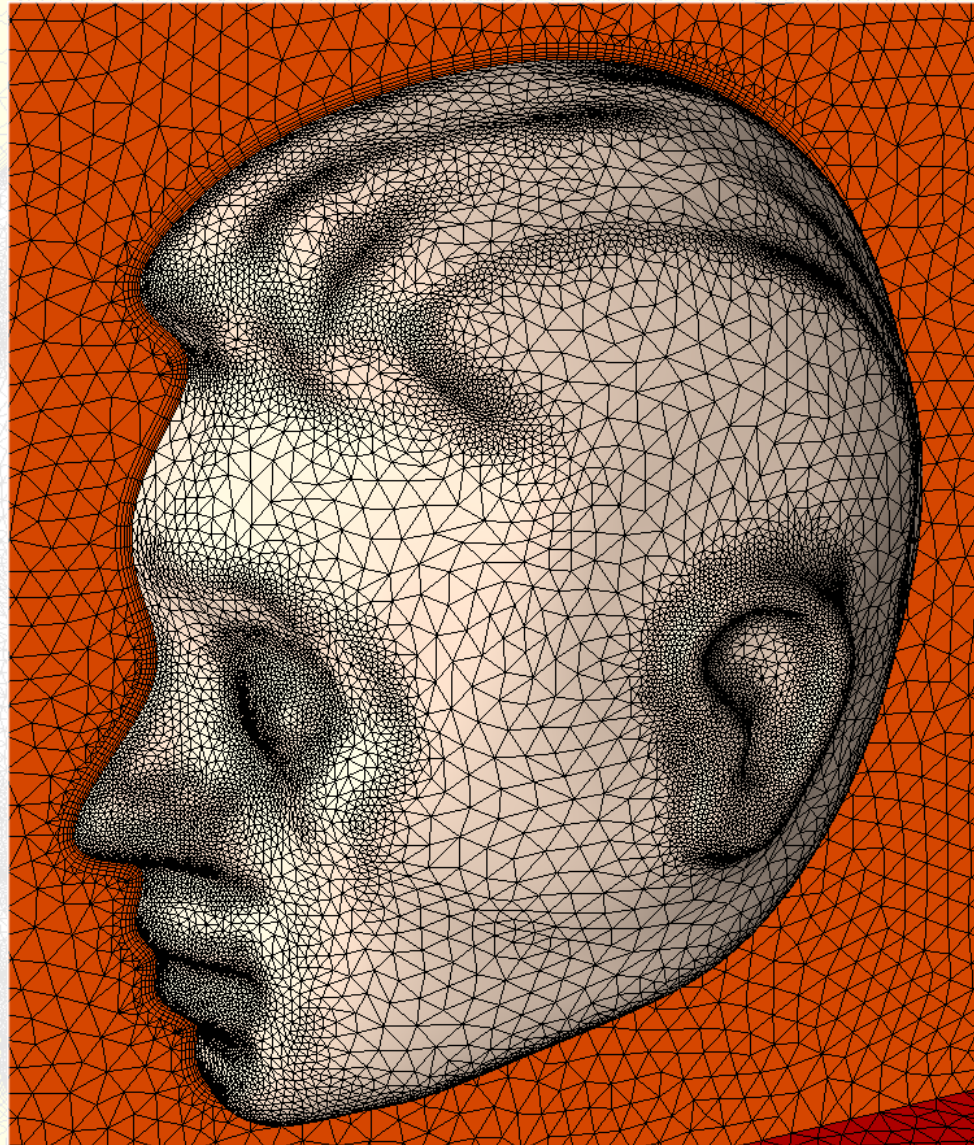
Major goals:

• Providing access to reliable, robust and high quality CFD-meshing based on CAD geometry

• Establishing complete GUI based environment: Access to strong OpenFOAM® -solution capabilities without editing text files or detail knowledge of keyword-structure (e.g. "turbulentMixingLengthFrequencyInlet")

• Reducing the time from CAD model to OpenFOAM® run-ready case

Fully compatible: User can switch anytime from GUIs to text-based model setup
User can extend the CastNet model output for specific needs

CastNet for OpenFOAM®

Major features for OpenFOAM:

• Support of OpenFOAM® 1.5, OpenFOAM® 1.6.(x) and OpenFOAM® 1.7

• Direct generation of complete models (variables, controlDict, solver settings, materials etc.) for more than 17 solvers in 1.6/1.7

• Convenient definition of boundary conditions using derived OpenFOAM® patches

• Definition of local, initial conditions e.g. specification of volume fractions in particular regions

• Face and cell zones (e.g. baffle or fan faces, porous or MRF-zones)

• User support by "defaults" e.g. walls with zero-gradient or wall function selection

• SnappyHexMesh in case hex-dominant meshes are required

• Additional GUI for job-control during the run: Switching solvers, changing schemes, modifying underrelaxtion factors, plotting residuals,…
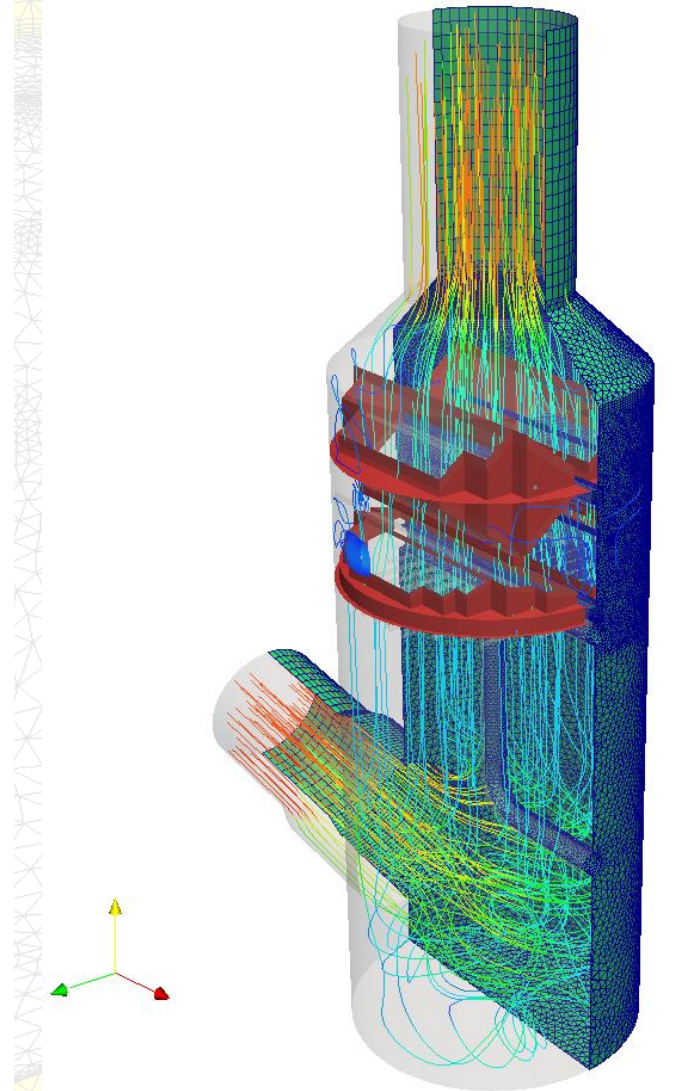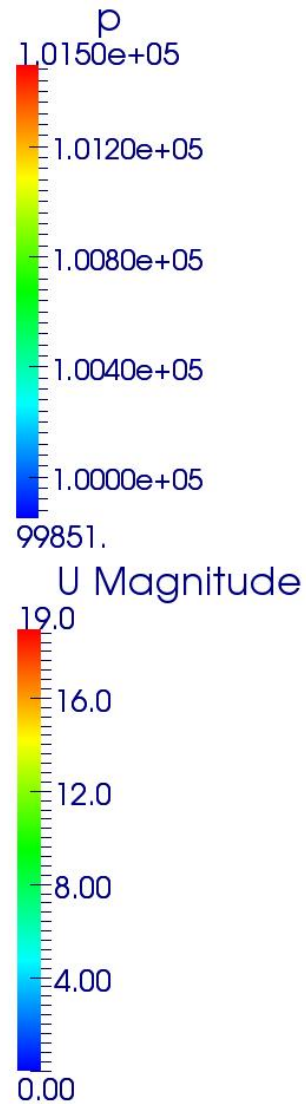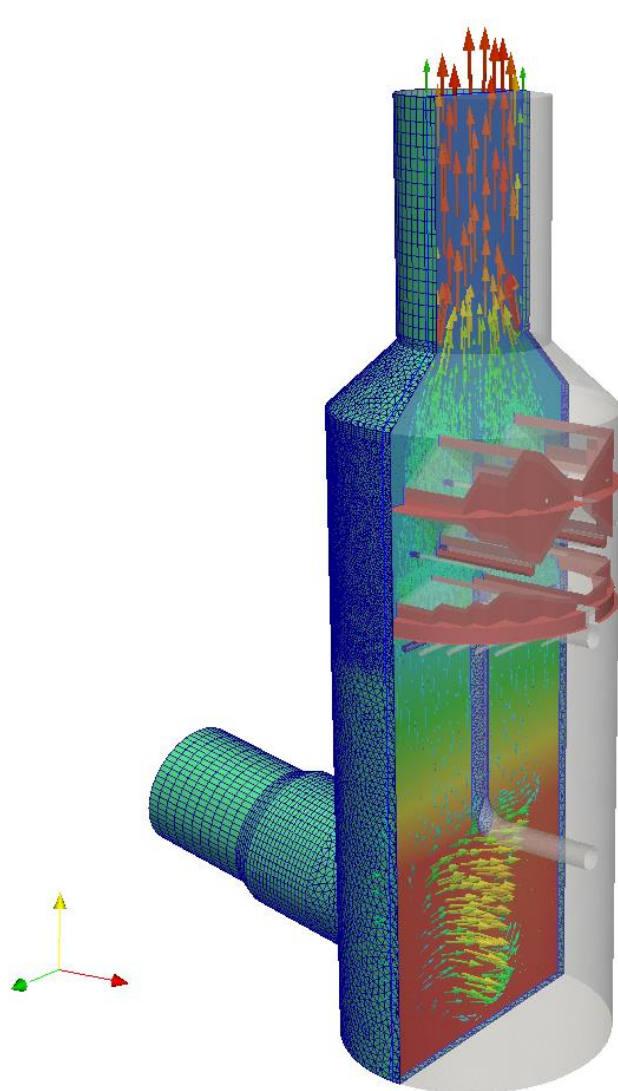
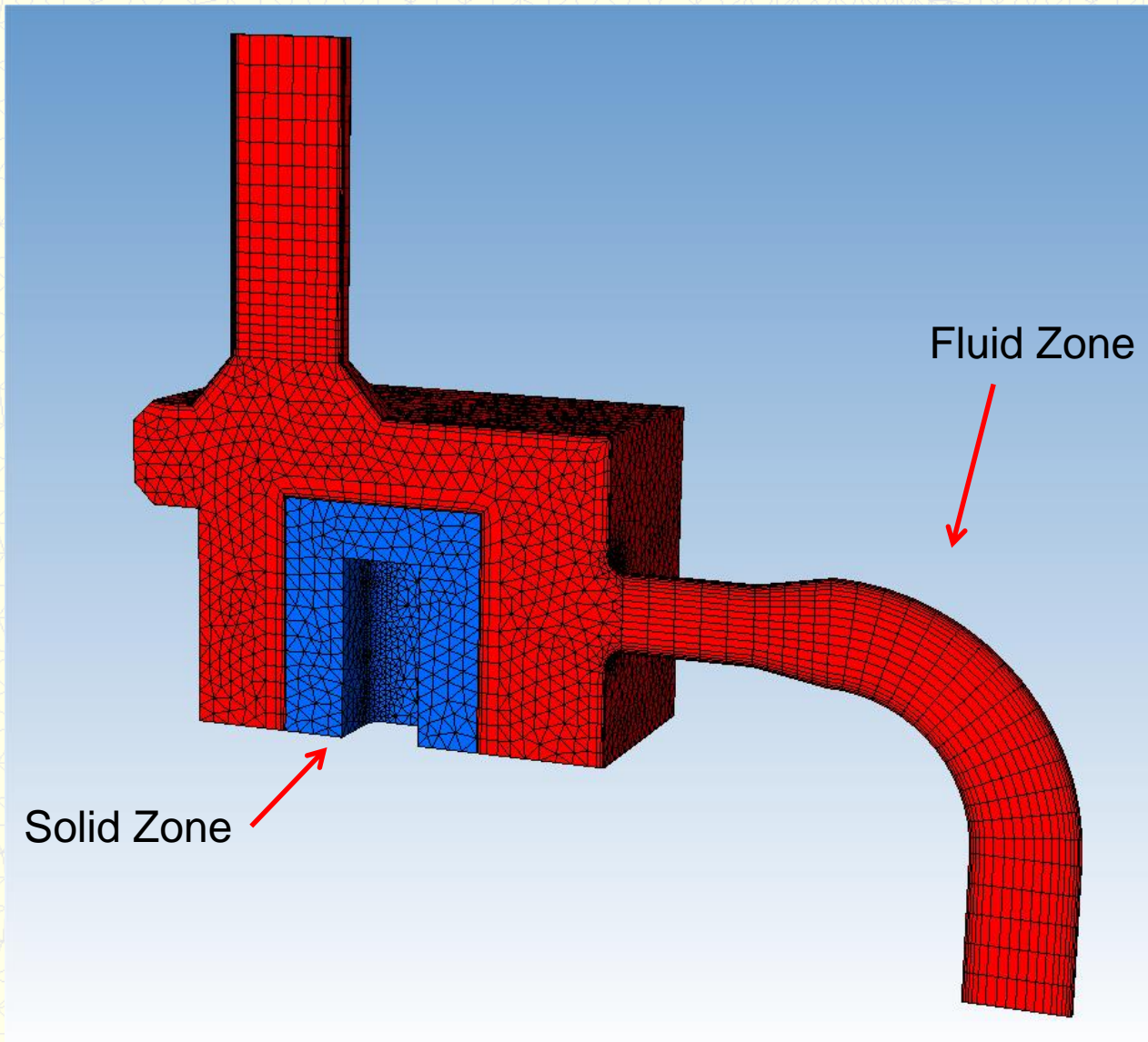Example

Free-Mesh and extrude regions
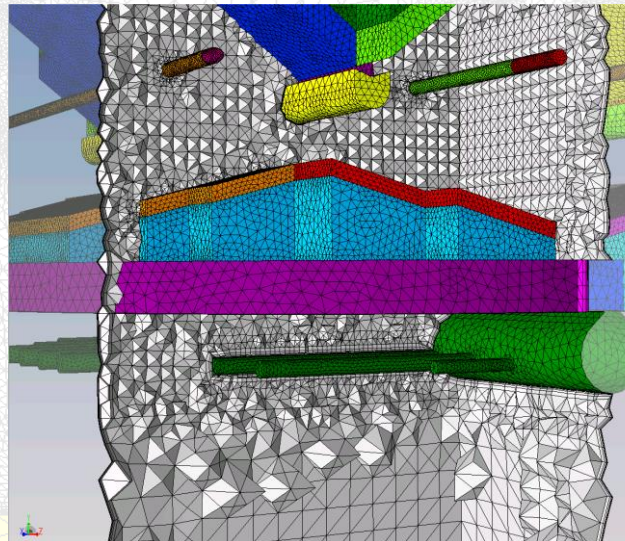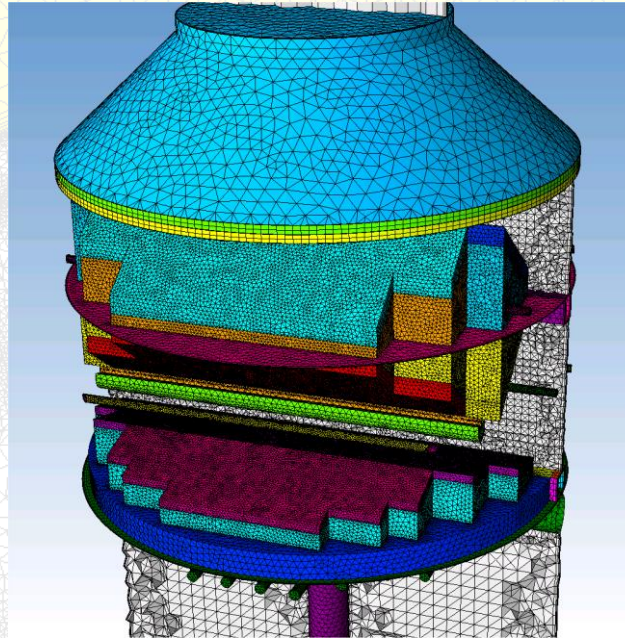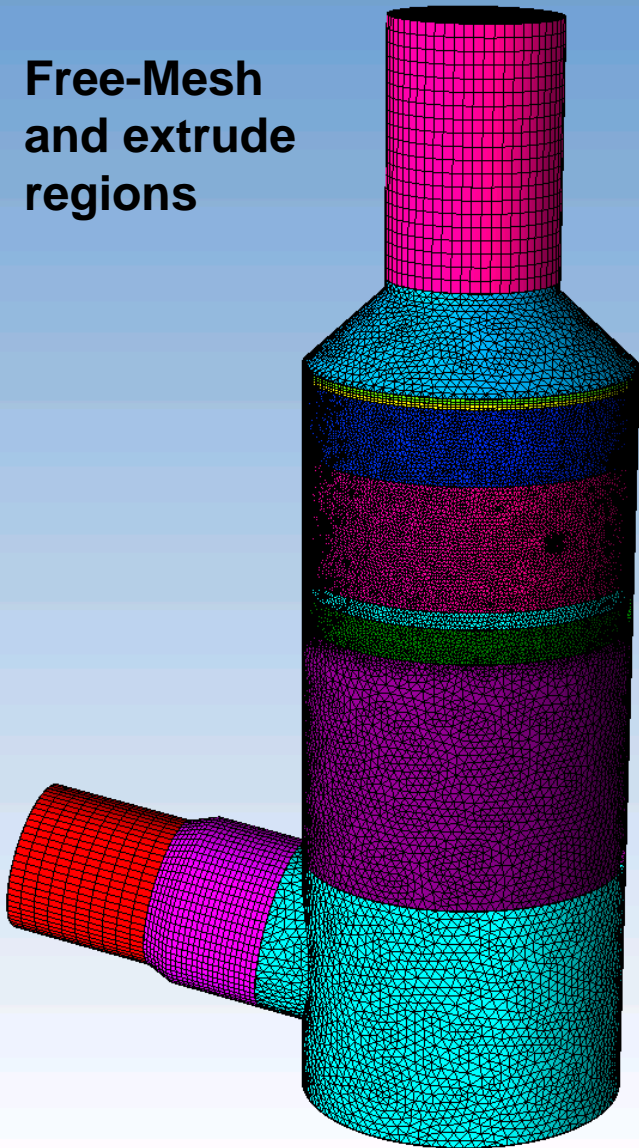
Boundary-Layer

Note: More complex meshing examples of customer cases presented in Gothenburg

Visualized with Paraview

Fluid Zone

Solid Zone

General characteristic of the cht-Solvers

• Each region: subdirectory in time-directories and system-directory

    • e.g. 0/fluid1 system/fluid1 and 0/solid1 system/solid1

• Region allow local solution settings:

    • e.g. 0/fluid1/fvSchemes and 0/fluid1/fvSolution

Result: Extremely flexible

                    But high effort for case setup

**Model approach in CastNet:**

  Geometry:

  • Transfer of parts of CAD assembly into geometry regions

  • Geometry regions can be assigned to CHT-regions

  • CastNet detects interfaces between CHT-regions automatically
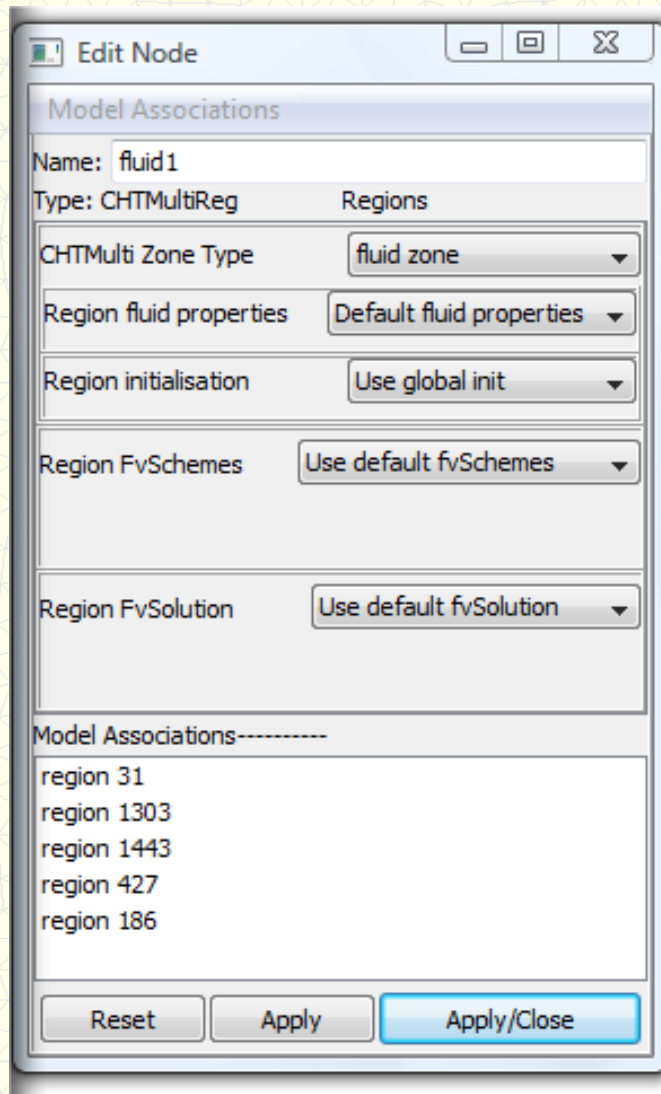
  Solution parameter:

  • Can be assigned locally to each individual CHT-region (high flexibility)

  • Or globally defined settings can be used (reduced definition effort)

Example: Model setup for CHT-Solvers
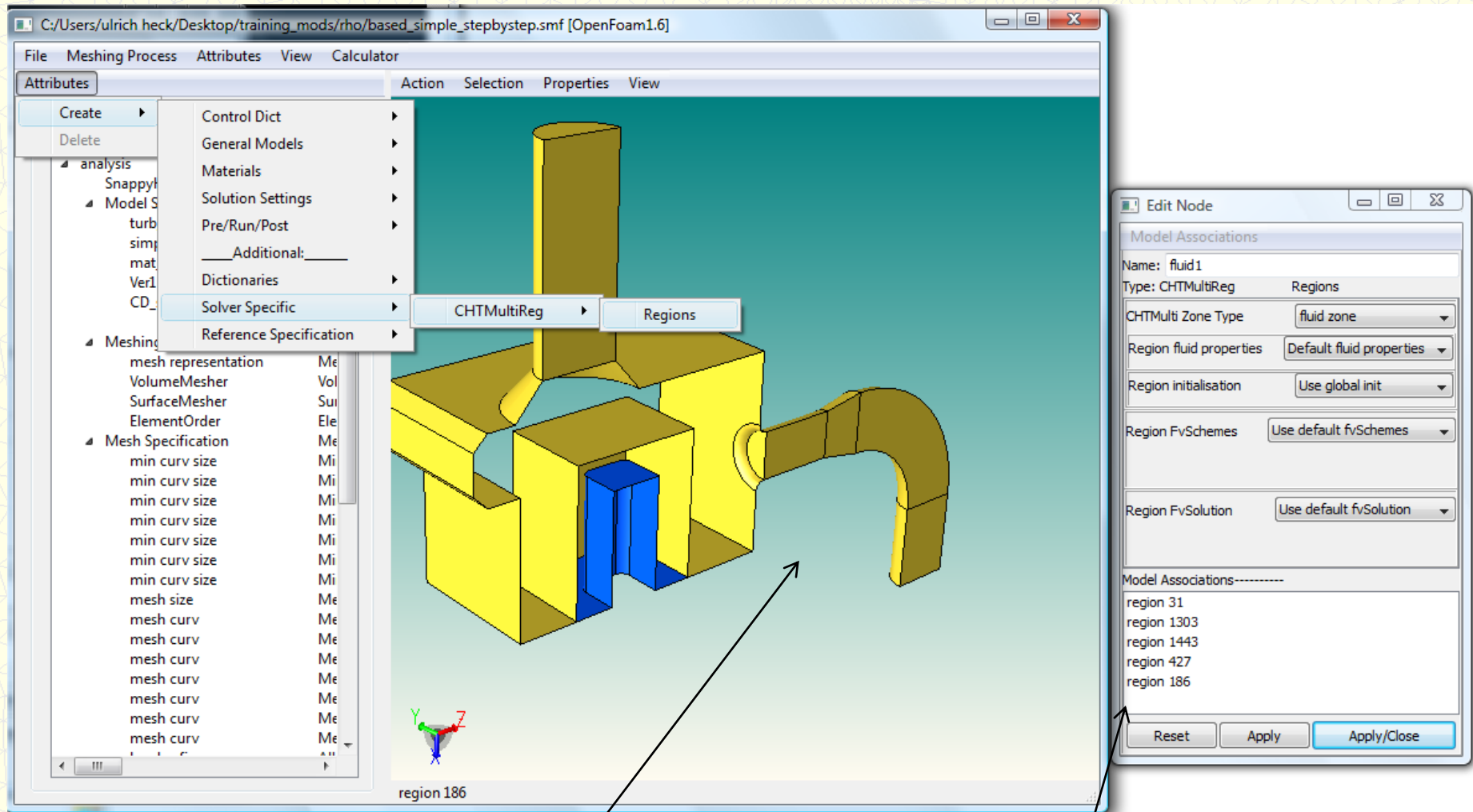
Parts of a CAD assembly can be picked and assigned to CHT-Zones

Solution and properties settings for CHT-region can be defined locally or global

E.G: Entries for Control Dict and properties

Model Associations

Name: CD_simple
Type: Control Dict   CD Definition

Application	chtMultiRegionSimpleFoam ▼

steady-state; turbulent; compr. fluids; gravity
with heat tranfer and solid conduction
Vars: p, U, T, Turbulent RANS
Use: Real abs. pressure, g, cht-Mat.

Start from	Start time ▼
Start time	0
Stop at	End time ▼
End time	1000
Delta time	1
Write control	timeStep ▼
Write interval	250
Purge write	1
Write format	Ascii ▼
Write precision	9
Write compression	Uncompressed ▼
Time format	General ▼
Time precision	6
Run time modifiable	Yes ▼
☐	Adjust Time Step Yes ▼
☐ max Courant Number	0.2
☐ max delta T	1
Solver specific	none ▼

Model Associations----------
Model

[Reset] [Apply] [Apply/Close]

Model Associations

Name: turb_ke
Type: General Models   Models

Pressure Unit System	Solver specific defaults ▼
Turbulence Model	k-omega SST ▼
Gravity	On ▼
Gravity X Y Z	0 0 0
Buoyant/Inter solver: wall p	use buoyantPressure ▼
Value for p buoyantPressure	use internal field ▼
Default wall treatments	Use defaults ▼

Model Associations----------
Model

[Reset] [Apply] [Apply/Close]

Definition of boundary condition
Associated with CAD geometry
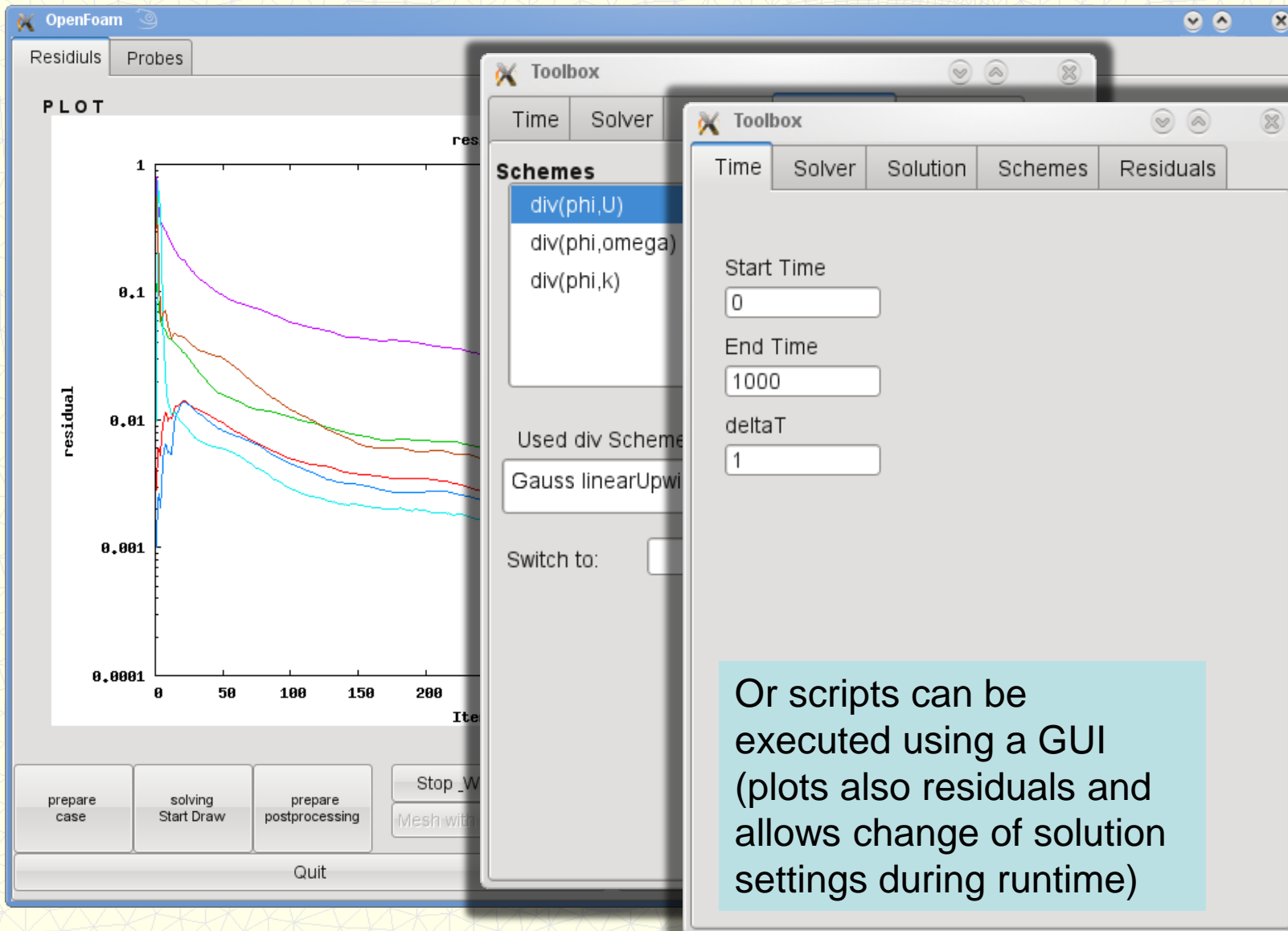
Example: Model setup for CHT-Solvers

```
cp system/fvSolution_2 system/fvSolution
mv rho_solid 0/rho
mv cp_solid 0/cp
mv K_solid 0/K
splitMeshRegions -cellZones -overwrite >logs/splitMeshRegions.log
mv regionProperties constant/
rm 0/fluid1/K
rm 0/fluid1/rho
rm 0/fluid1/cp
mkdir constant/fluid1
mv thermophysicalProperties_defaultfluid constant/fluid1/thermophysicalProperties
cp constant/RASProperties constant/fluid1/.
cp constant/turbulenceProperties constant/fluid1/.
cp constant/g  constant/fluid1/.
cp fvSchemes_defaultfluid system/fluid1/fvSchemes
cp fvSolution_defaultfluid system/fluid1/fvSolution
# Fixes bug in splitMeshRegion for omega turb-inlet and wall
sed -i 's/value           uniform 0;/value           $internalField;/g'  0/fluid1/omega
# Fixes bug in splitMeshRegion for p in OF 1.7
sed -i 's/value           uniform 0;/value           $internalField;/g'  0/fluid1/p
mv changeDictionaryDict_fluid1 system/fluid1/changeDictionaryDict
changeDictionary -region fluid1 >logs/changeDictionary_fluid1.log
cp fvSchemes_defaultsolid system/solid1/fvSchemes
cp fvSolution_defaultsolid system/solid1/fvSolution
# Fixes bug in splitMeshRegion for rho zero values
sed -i 's/value           uniform 0;/value           $internalField;/g'  0/solid1/rho
# Fixes bug in splitMeshRegion for T zero values
sed -i 's/value           uniform 0;/value           $internalField;/g'  0/solid1/T
rm 0/solid1/p
rm 0/solid1/p_rgh
rm 0/solid1/U
rm 0/solid1/k
rm 0/solid1/omega
rm 0/solid1/mut
rm 0/solid1/alphat
mv changeDictionaryDict_solid1 system/solid1/changeDictionaryDict
changeDictionary -region solid1 >logs/changeDictionary_solid1.log
mv fvSolution_gobal system/fvSolution
mv fvSchemes_defaultfluid system/
mv fvSchemes_defaultsolid system/
mv fvSolution_defaultfluid system/
mv fvSolution_defaultsolid system/
```

Script export
Case is generated with a single command:
Fully automatic case generation depending on the settings in CastNet

Example: Model setup for CHT-Solvers

Or scripts can be executed using a GUI (plots also residuals and allows change of solution settings during runtime)

CastNet for OpenFOAM®

• Further support of CastNet features: Easy identification of cell and faces zones for multi-region applications

• Adding solvers and OpenFOAM functionality:
    Target solvers: reactingFoam, rhoReatingFoam, fireFoam,
    rhoPorousMRFPimpleFoam, simpleWindFoam

• Extending the runGui: Allowing a detail job control and manipulation (e.g. plotting probes, switching more schemes….)

• Support further features for 1.6-extended as soon as this version is available
        • GGI
        • Sliding meshes
        • Windows Version
        • more schemes, patches and solver

DHCAE
Tools

CastNet for OpenFOAM®

• Reliable CAD-based CFD meshing

• Reduces the set up time for run ready OpenFOAM case significantly

• Helps to learn "OpenFOAM"

• Provides a modeling and solution control environment:

    o Creating OpenFOAM input comparable to commercial systems

    o Without restricting OpenFOAM functionality

Thank you for your attention !

For more information (e.g. movies „CastNet for OpenFOAM®") please visit:

http://www.dhcae-tools.com/OpenFOAM.htm