Ulrich Heck, Martin Becker,
Bruno Santos and Nelson Marques

CastNet and blueCFD: A GUI controlled Windows environment for CFD based on OpenFOAM® technology

Introduction
CAD model based workflow
CastNet
BlueCFD
Demo
Conclusion

DHCAE
Tools

DHCAE Tools:

Engineering company in Krefeld, Germany

Offer services to bring OpenFOAM in industries:

- GUI-development (CastNet)
- OpenFOAM development
- Training/Support

Our philosophy: More pragmatic then dogmatic

We have to accept the reality of our customers. Not always but often:

- Commercial CAD systems with proprietary kernels
- Windows systems (IT says "NO" to Linux)
- New software must be productive from the first second

Can not be covered with OpenFOAM directly
Combine proprietary solutions with OpenSource
Try to share open source part of work (e.g. OpenFOAM-extensions)

Workflow

DHCAE Tools

Parasolid: (Nx, Solidworks, Solidedge,…)	**ACIS:** (Inventor, SpaceClaim, TurboCAD,…)	**ProE**	Catia, other	**Discrete** (stl, Nastran)

Translator SpaceClaim

CastNet: CAD Kernel based geometry import: Models/ Assemblies
Hybrid meshing or SnappyHexMesh-Definition
Case setup OpenFOAM: 30 solvers for 2.0/2.1

Windows: BlueCFD
Job control: runGui

Exchange
CastNet

Linux: OpenFOAM 1.5- 2.1
Job control: runGui

Paraview, EnSight…

Major goals for CastNet:

• 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

Helps to learn the usage/keywords of OpenFOAM

OpenFOAM Setup in CastNet for Chemical Engineering

DHCAE Tools

File Meshing Process Auto Attributes Infos Calculator Help

Attributes

Action Selection Properties View

Name	InfoType
analysis	analysis
SnappyHex-Modelling	SnappyHex-Mode
Model Specifications	Model Specificatio
	Control Dict
	General Models
	Materials
	Solution Settings
	Pre/Run/Post
Meshing Process Specification	Meshing Process
mesh representation	Mesh Representat
VolumeMesher	Volume Mesher
SurfaceMesher	Surface Mesher
ElementOrder	Element Order
Mesh Specification	Mesh Specificatio
top_ground	3D Boundary Lay
top_durchgang	3D Boundary Lay
rohre	3D Boundary Lay
regs	Mesh Size
refine1	Refinement Sourc
refine1	Refinement Sourc
out	Classic Extrusion
no_mesh	No Mesh
mesh size	Mesh Size
kanten_hauben	Mesh Size
ex_kanal	Gen. Extrusion M
ex5	Gen. Extrusion M
ex4	Gen. Extrusion M
ex3	Gen. Extrusion M
ex2	Gen. Extrusion M
ex1	Gen. Extrusion M
deckel_hauben_rand	Mesh Size
VolumeShapeMetric	Volume Shape Me
SurfaceShapeMetric	Surface Shape Me
3d_blPorb	3D Boundary Lay
	Gen. Extrusion M
	Allow Refinement
	3D Boundary Lay
Initial Conditions	Initial Conditions
	General Internal F
Face/Cell Zones	Face/Cell Zones
rotor	Cell Zones
lochblech	Cell Zones
filter	Cell Zones
baffle_zustroem	Face Zones
baffle_sammelrunpf	Face Zones
baffle_hauben	Face Zones
baffle_filter	Face Zones
Boundary Conditions	Boundary Conditi
in1	Inlet
P-out	Outlet

Chemical plant engineering
Meshing setup and OpenFOAM
case definition

← Model setup parameter
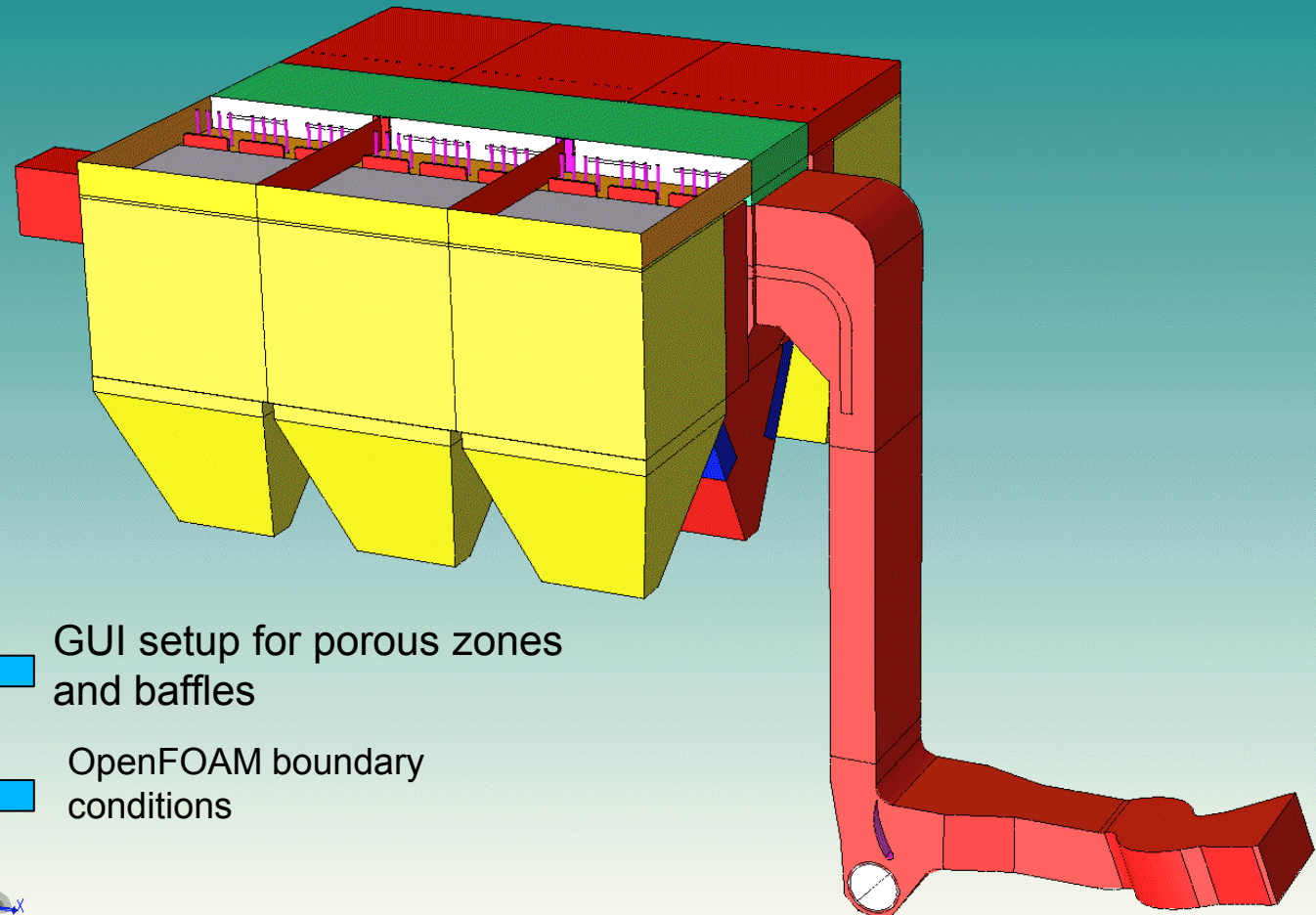
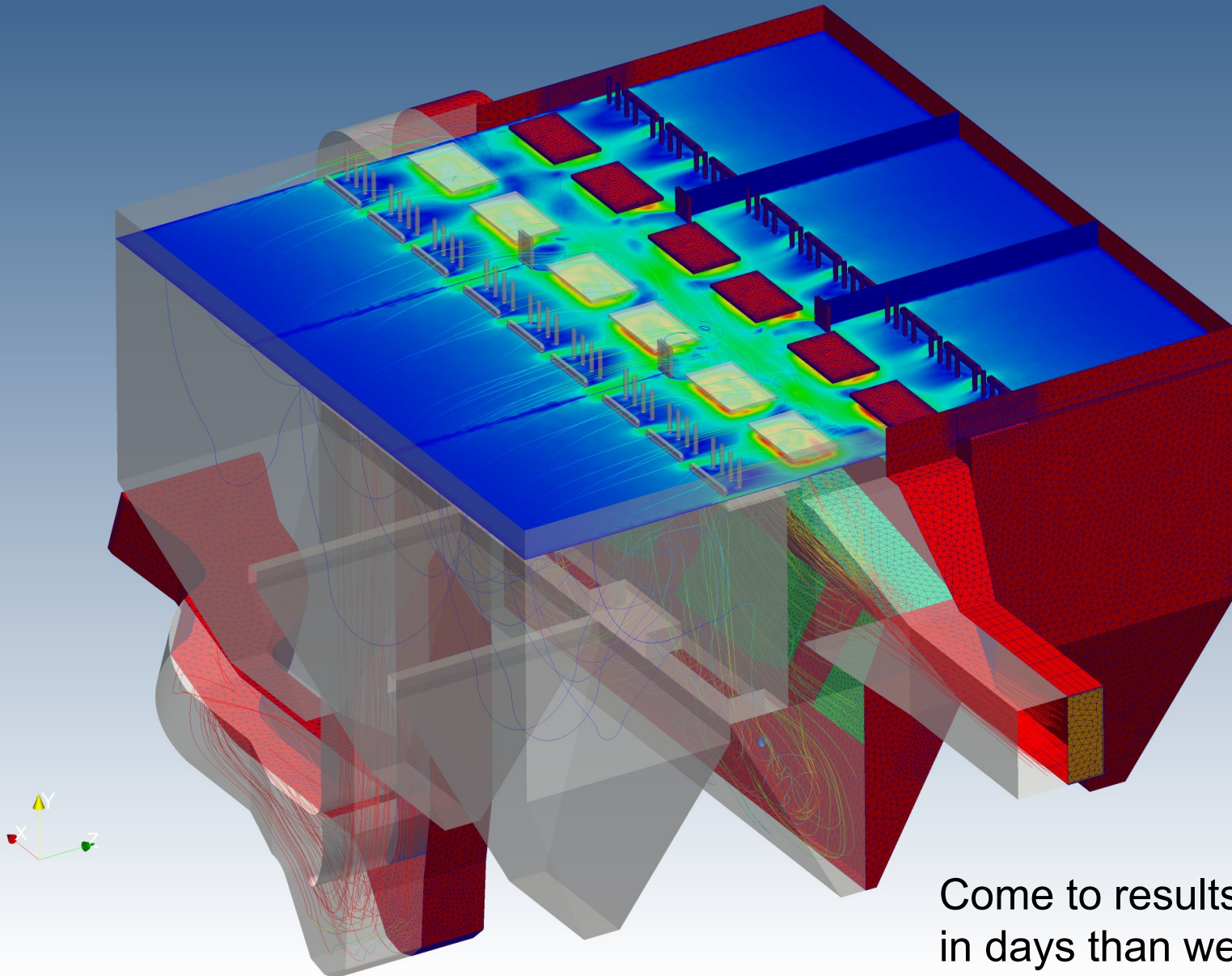← GUI setup for porous zones and baffles

← OpenFOAM boundary conditions

face 30848
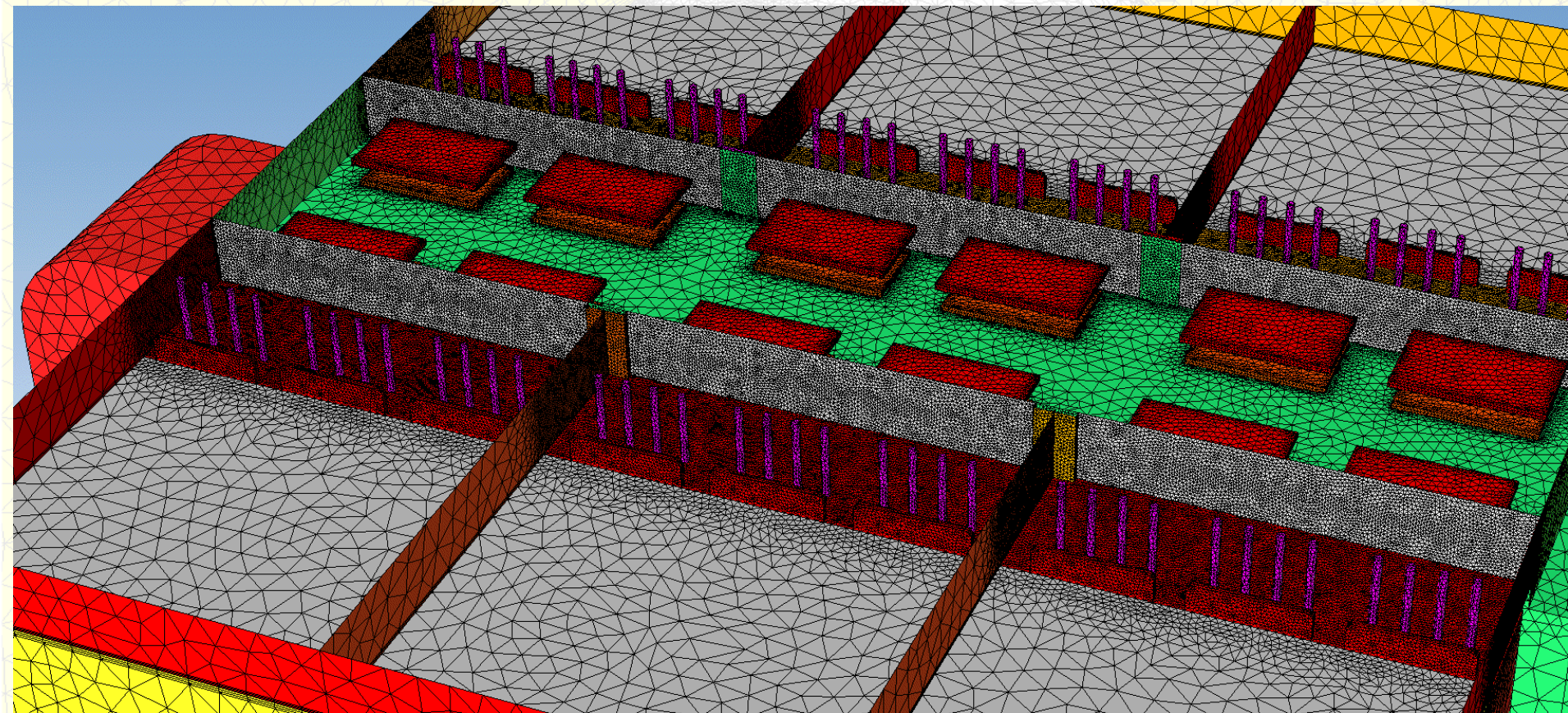
Roof section

Come to results rather in days than weeks

blueCAPE's blueCFD

History and Capabilities

blueCAPE Lda
Rua Fonte dos Corvos, n.29
Casais da Serra
2665-305 Milharado
PORTUGAL
www.bluecape.com.pt
info@bluecape.com.pt

What is blueCFD:

- A Product/Service of a high quality cross-compiled build version of OpenFOAM® for Windows XP/Vista/Seven, 32 and 64bit.

- Releases are synched with the official releases when possible or requested.

- Tries to bring to Windows open-source applications that are mostly used with OpenFOAM on Linux.

- Modifications to the OpenFOAM source are based on Symscape's modifications.

- Not endorsed by SGI/OpenCFD.

11th November 2009 – blueCFD 1.6-1 is released as a Product/Service of «a high quality cross-compiled build version of OpenFOAM® for Windows XP and Vista, 32 and 64bit».

Featured:

- First 64bit build of OpenFOAM 1.6.0 for Windows.

- Parallel capabilities with MPICH2.

- Decomposition with Metis.

- Highly detailed wiki page on how to cross-compile.

- Updates/fixes for the builds are provided on request

16th of August of 2010 – blueCFD 1.7-1 (OpenFOAM 1.7.0)

15th of December 2011 – blueCFD 2.0-1 (OpenFOAM 2.0.x)

27th of February 2012 – blueCFD 2.0-2 and 2.1-1 (2.1.x)

Latest Features:

- Increased focus on blueCFD documentation.
- More Third-Party applications, all open-source
    - E.g.: enGrid, swak4Foam, PyFoam (partial)
- More MPI support: Open-MPI, MPICH2 and MS-MPI.
- OpenFOAM code compilable directly on Windows.
    - Including #codeStream!
- High focus on having all tutorials working.

All of the customized OpenFOAM binaries should work as well on Windows as the original version does on Linux.

Original scripts have been tweaked for working in Windows through Msys.

Windows Command Line can also be used, but shell scripts will only work if MSys is installed.

Building OpenFOAM *centric* source code directly on Windows, with minimum modifications. (Note: GCC only.)

Numerical results differ between Linux versus Windows, but are within an acceptable margin of error.

- Reason: the *usual* compiler and operating system differences. E.g.: Intel Compiler vs GCC in CFD.

- Report of specific differences in results is currently unavailable and has not yet been requested.

- Anyone can test this by using the binaries present in the online blueCFD-SingleCore project.

Improve:
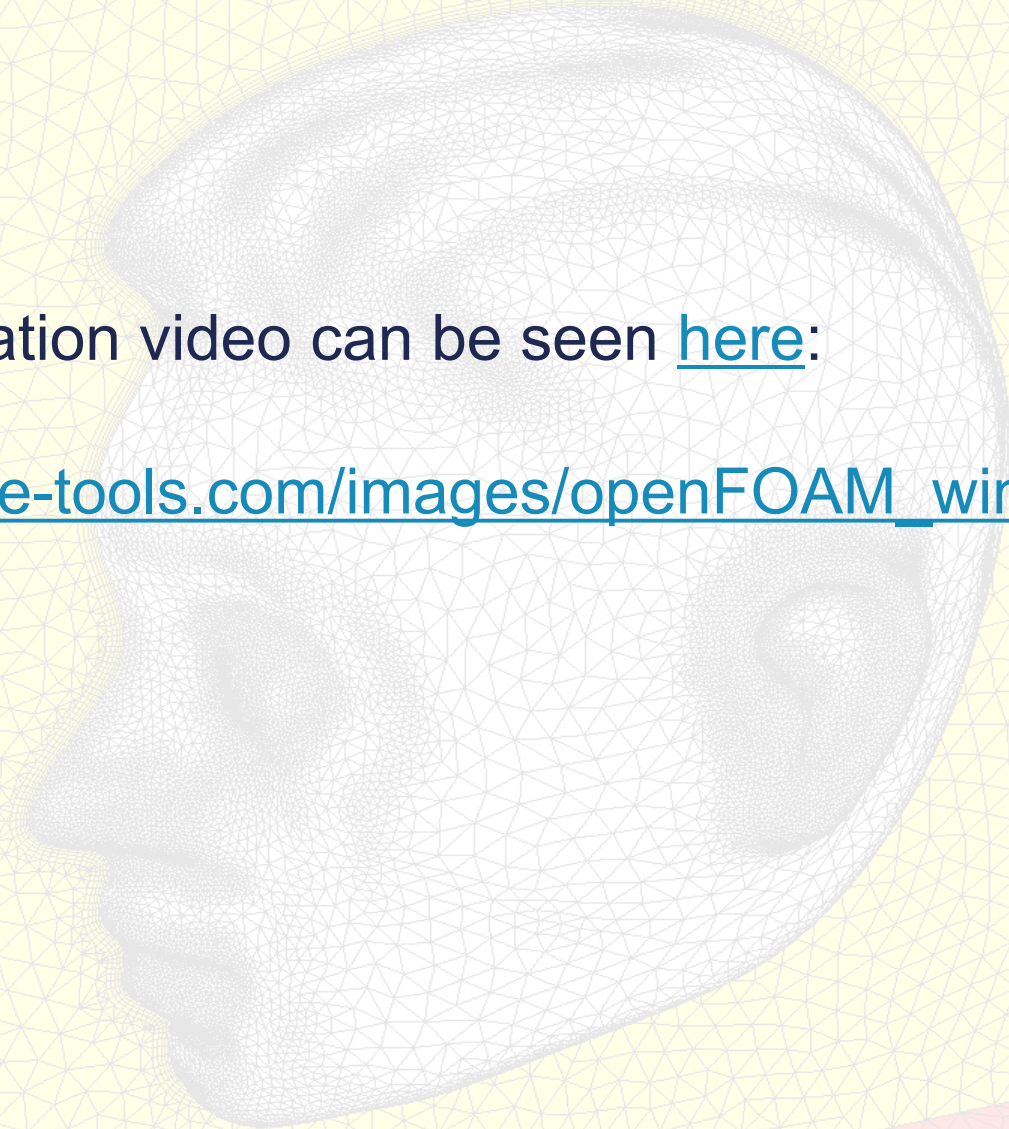
- Performance on Windows

- Documentation

- Test loop

- Reports on quality

- Building capabilities on Windows

- Third-Party applications:

  – PyFoam – enable full capabilities on Windows

  – enGrid – GUI capabilities for OpenFOAM > 1.6.x

Short demonstration video can be seen here:

http://www.dhcae-tools.com/images/openFOAM_windows.htm

**DHCAE
Tools**

CastNet and blueCFD: Complete GUI based workflow of OpenFOAM in a Windows environment:

- CAD model based case setup,

- CFD meshing,

- solving,

- graphical job control and

- post processing.

Thank you for your attention !

The workflow coincides with commercial CFD systems

Possible bridge to OpenFOAM

For more information please visit www.dhcae-tools.com