2018 *Code_Saturne* User Meeting

5 April 2018, Version 1.0

Cooling and ventilation applications and fundamental validations of *Code_Saturne* on different mesh types

Nicolas Tonello
Director

Renuda, 329-339 Putney Bridge Road, London, SW15 2PG, UK

Contact

For further information, please contact:

Nicolas Tonello
Director
RENUDA

nicolas.tonello@renuda.com

T: +44 (0)20 3371 1709

Outline

1. Renuda and *Code_Saturne* Projects
2. **Cooling Towers**
3. **CFD on non-Hexahedral Meshes**
4. Conclusions

1. Renuda and *Code_Saturne* Projects

Renuda

- **CFD Specialists**
  - Consulting, Software development, Training
  - Fully independent
  - UK, France, Germany
- **Blue Chip Clients**
  - Applications from single phase pipe flow to turbomachinery, multiphase flow, coupled heat transfer, mechanical calculations
  - Industries: transport, automotive, processing, nuclear, power generation, civil engineering
- **Compete on**
  - Skills
  - Difficult problems

Research Partnership And Collaborations

- Research and development is very important
- Collaborative research relationship with EDF R&D on the development of *Code_Saturne*
- Collaboration with the SALOME teams:
  - Development of GUI for specialised steam turbine code
    - From CAD to Analysis
  - Beta testers for the parametric design module SHAPER
- Part of the UK Consortium on Turbulent Reactive Flow
- Collaboration with different universities and research labs
  - University of Manchester
  - Daresbury Laboratory (Science and Technology Facilities Council) – HPC research and application
  - University of Edinburgh (software parallelisation)

Challenges

- To be competitive, tools and their users must be fast and methodologies must be robust and reliable. This means being able to generate models and meshes on any geometry, being able to control and modify models and meshes, sometimes in fine detail, and having access to compatible tools
- **Today, our go-to CAD-to-Analysis set of tools is:**
  - **SALOME**
  - *Code_Saturne*
  - **Paraview**

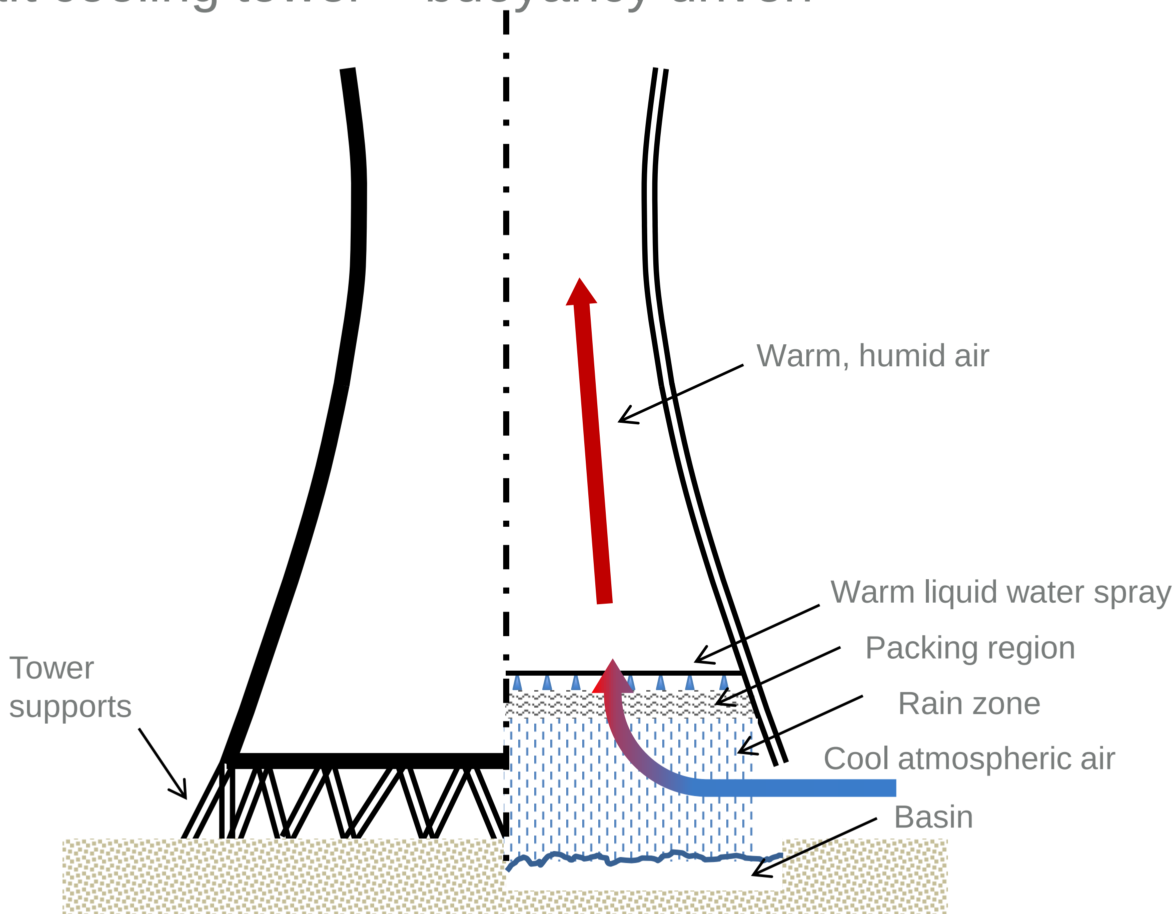
Illustrative *Code_Saturne* Projects

- Different applications to a wide range of industries
  - Waste water treatment
  - Machine and plant engineering
  - Auto industry
  - Process industry
  - Energy: Nuclear, Hydro
  - Turbomachines
- Mesh motion
- Multiphase
  - Lagrangian
  - Drift flux
  - VOF
- Application and development

2. Cooling Towers Modelling

Wet Draft Cooling Towers

- Natural draft cooling tower – buoyancy driven



Modelling

- Drift flux model
  - Bulk phase
  - Dry air
  - Water vapour and condensate
  - Rain drops
  - IAHR 2017*

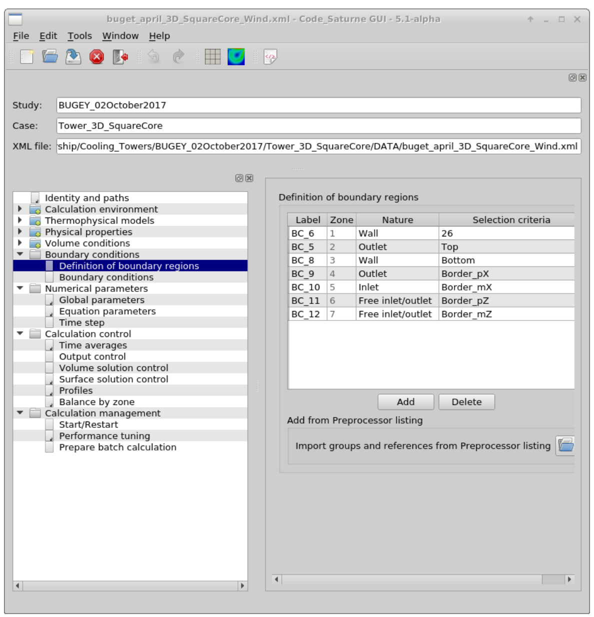
- Extended validation based on
  - MISTRAL test bench
  - Existing sites where available: Dampierre, Golfech

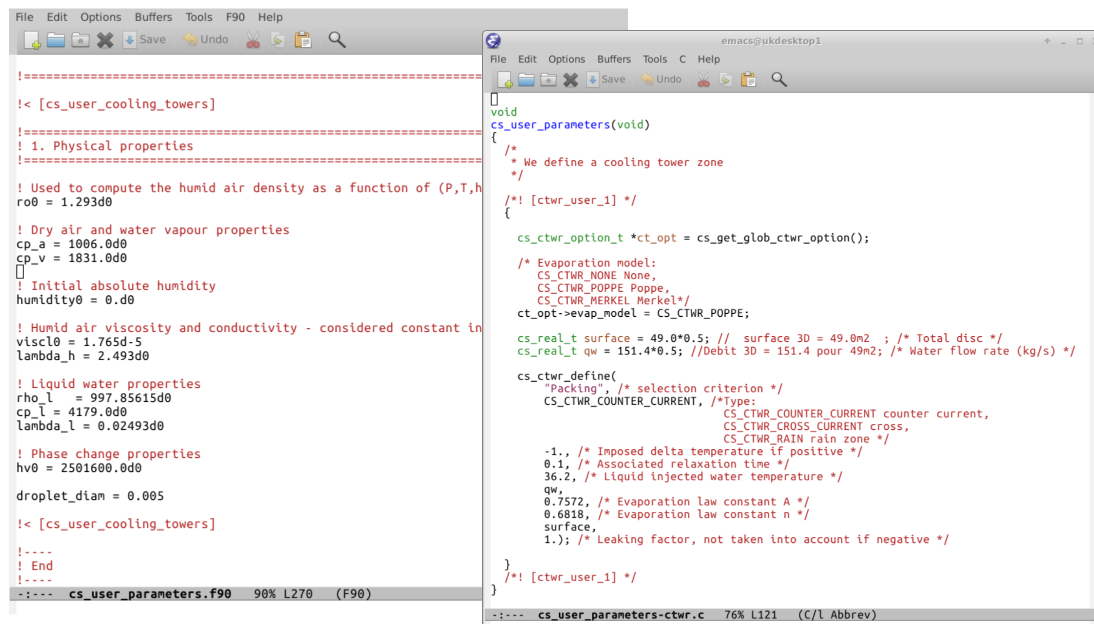
- Upcoming: connections to the Atmospheric Module for large scale calculations including both the towers and the atmospheric dispersion

*Tonello, Ferrand, Fournier, *Implementation of a drift flux multiphase model for 3D draft cooling tower simulations in Code_Saturne*, 18th IAHR International Conference on Cooling Tower and Air Cooled Heat Exchanger, Lyon, France 17 October 2017

Implementation

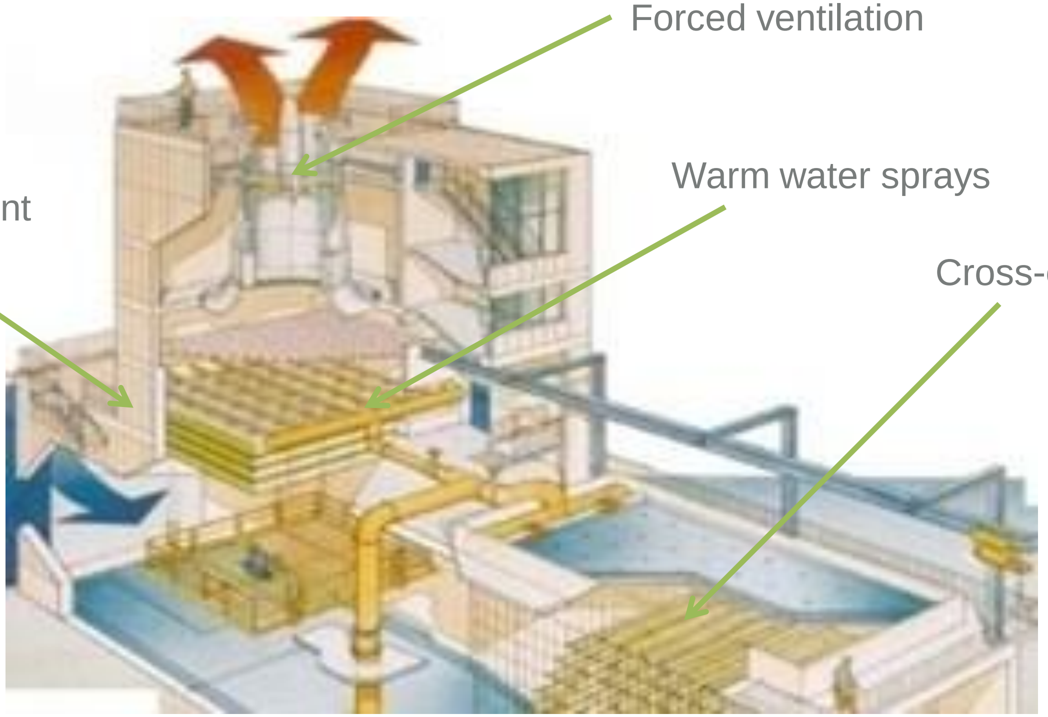
- Implemented in the code kernel
  - When the module is activated, the solution of the set of equations is activated automatically
  - GUI specification of the boundaries and numerical setup
  - Zones defined in the geometry
  - Sub-models parameters and physical properties specified in two user subroutines available in template format





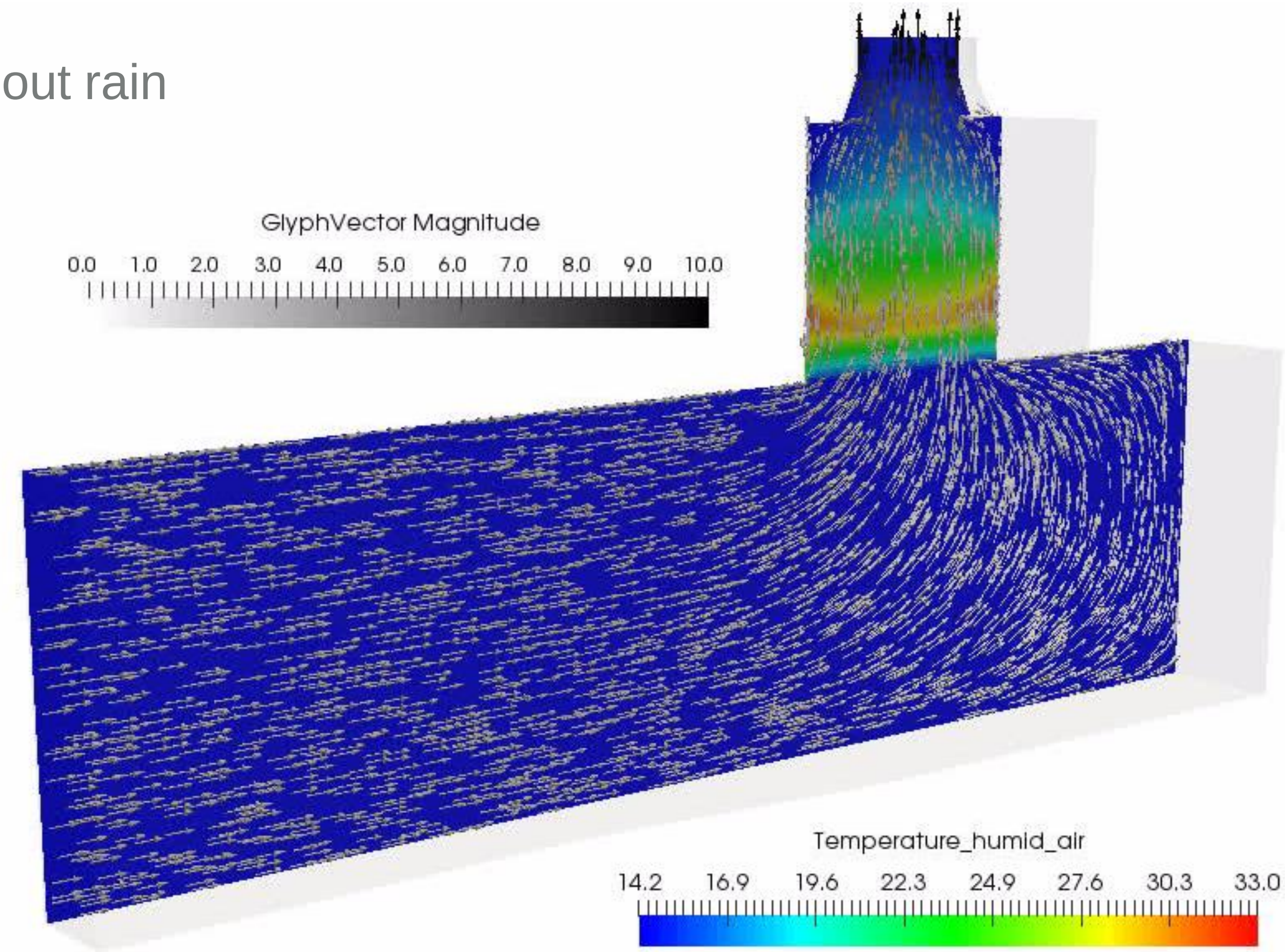
MISTRAL

- Experimental test bed which is part of the Bugey electricity production plant – test and validate real size equipment such as packing material



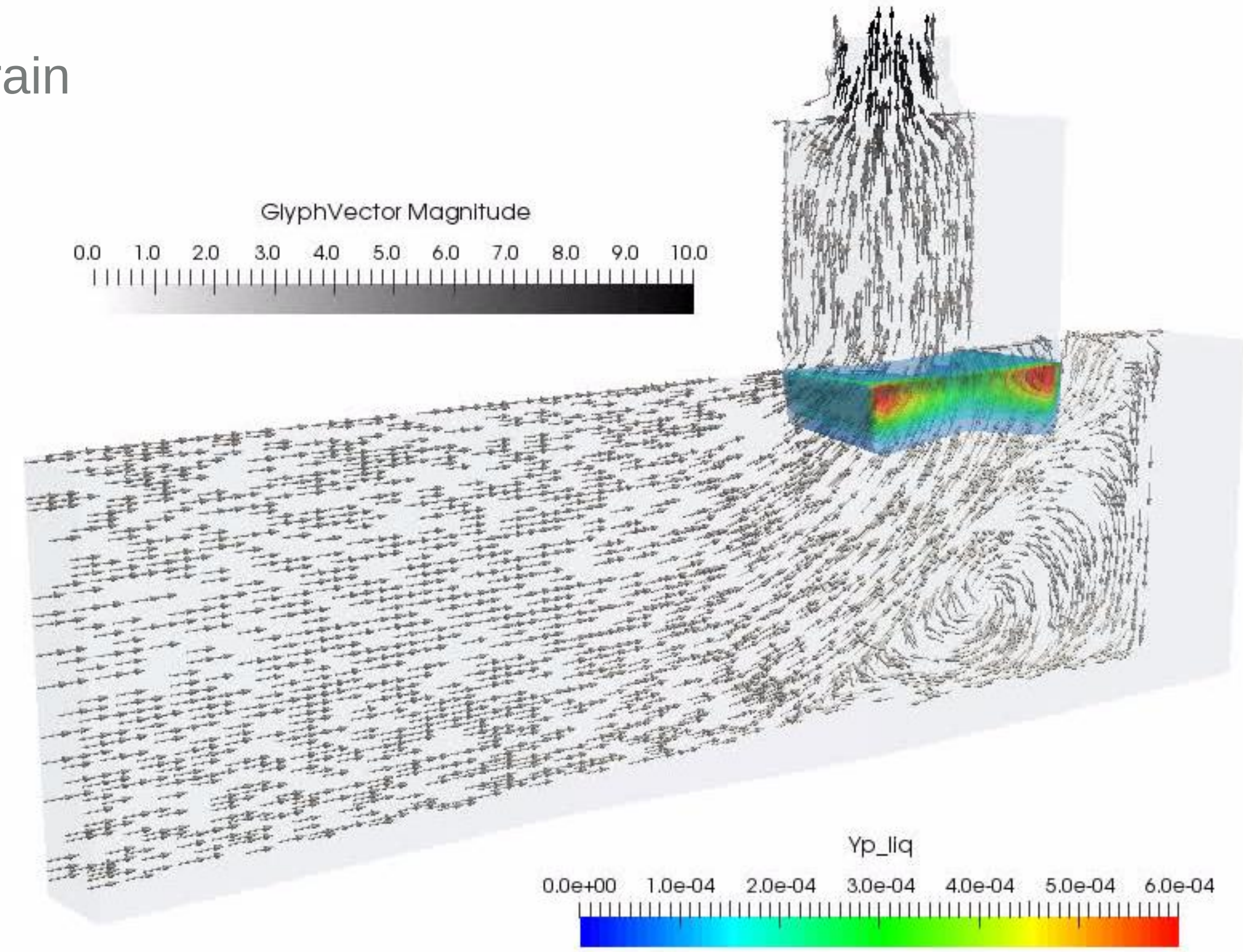
MISTRAL results

- Without rain

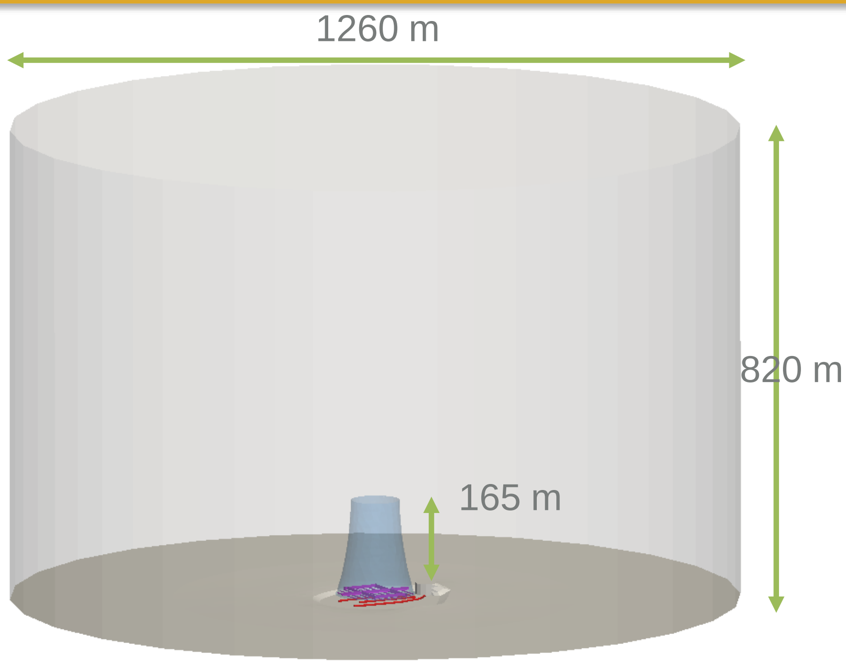


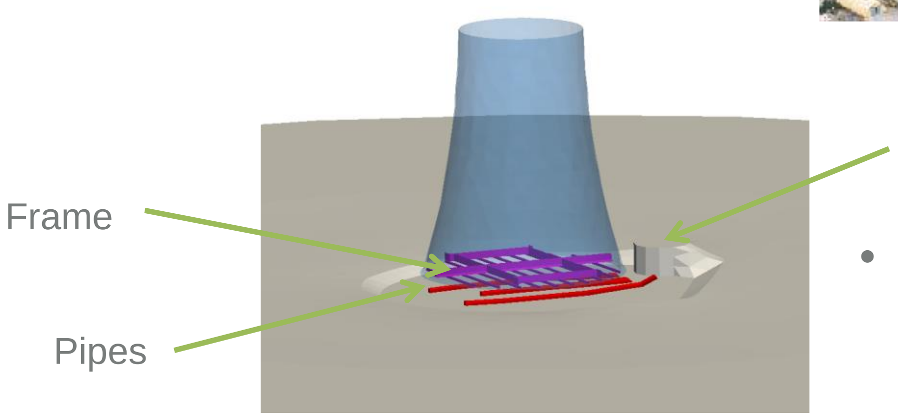
MISTRAL results

- With rain



Golfech



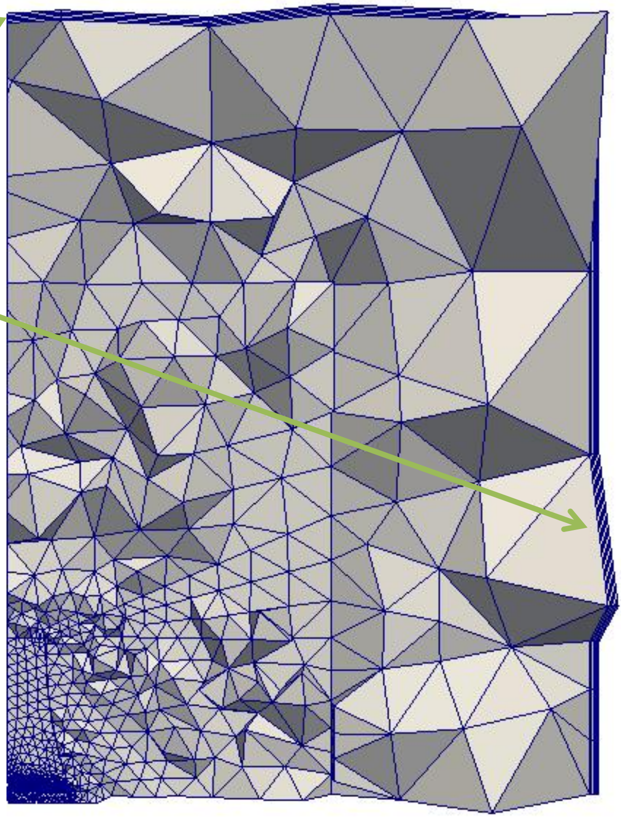


- Tower supports represented by pressure losses

Tetrahedral Mesh

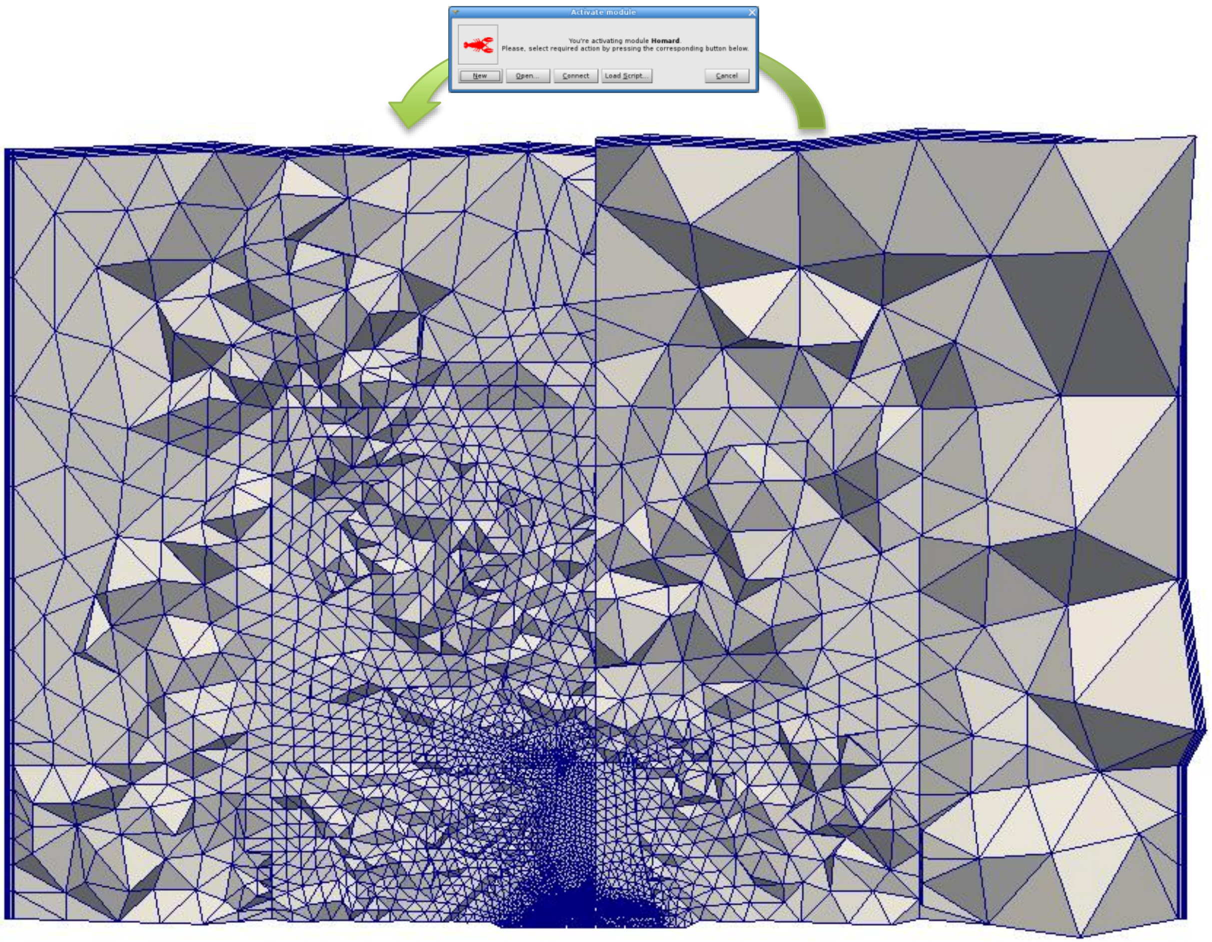
- Existing, very coarse mesh, rehabilitated for v5, including extrusion layers

Extrusion layers build internally with *Code_Saturne*

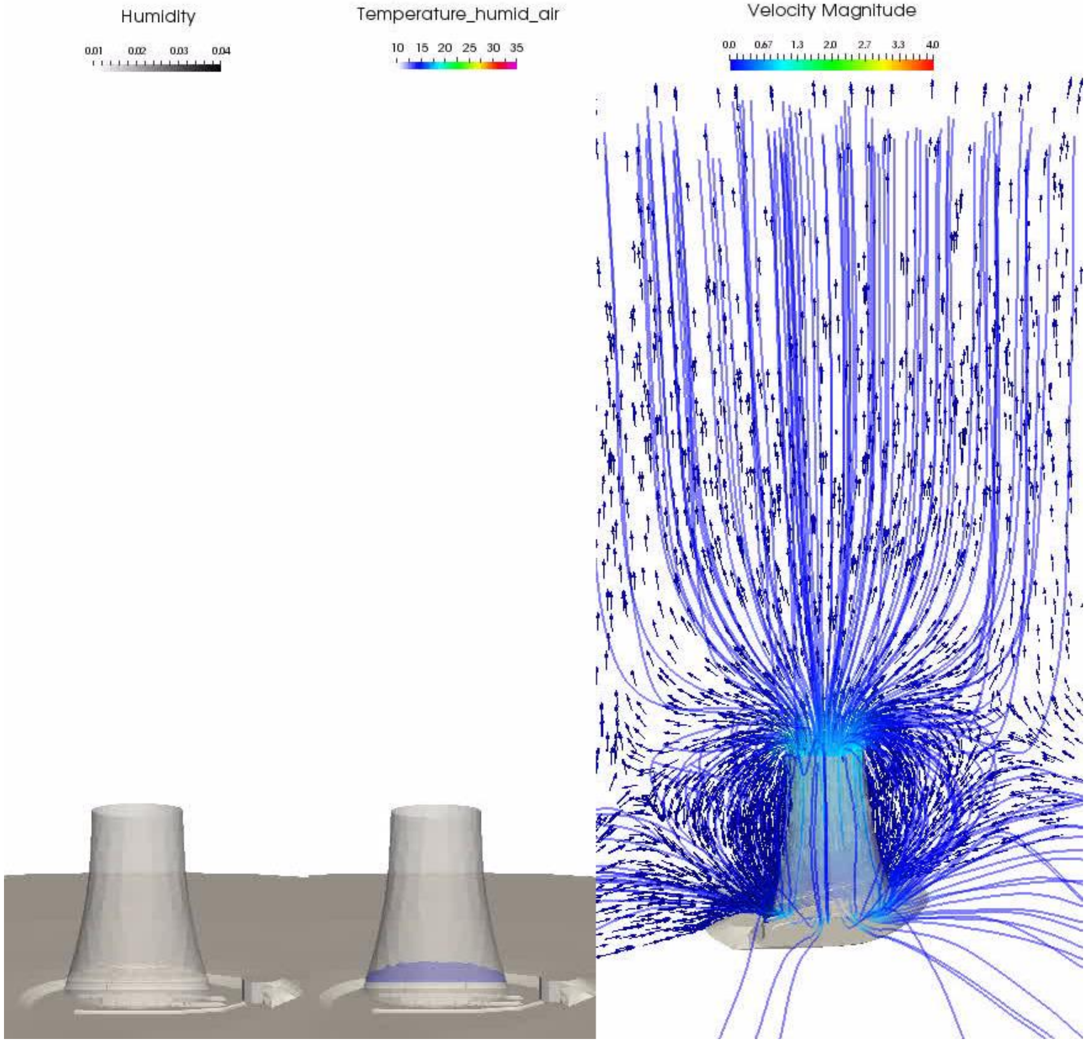


Tetrahedral Mesh

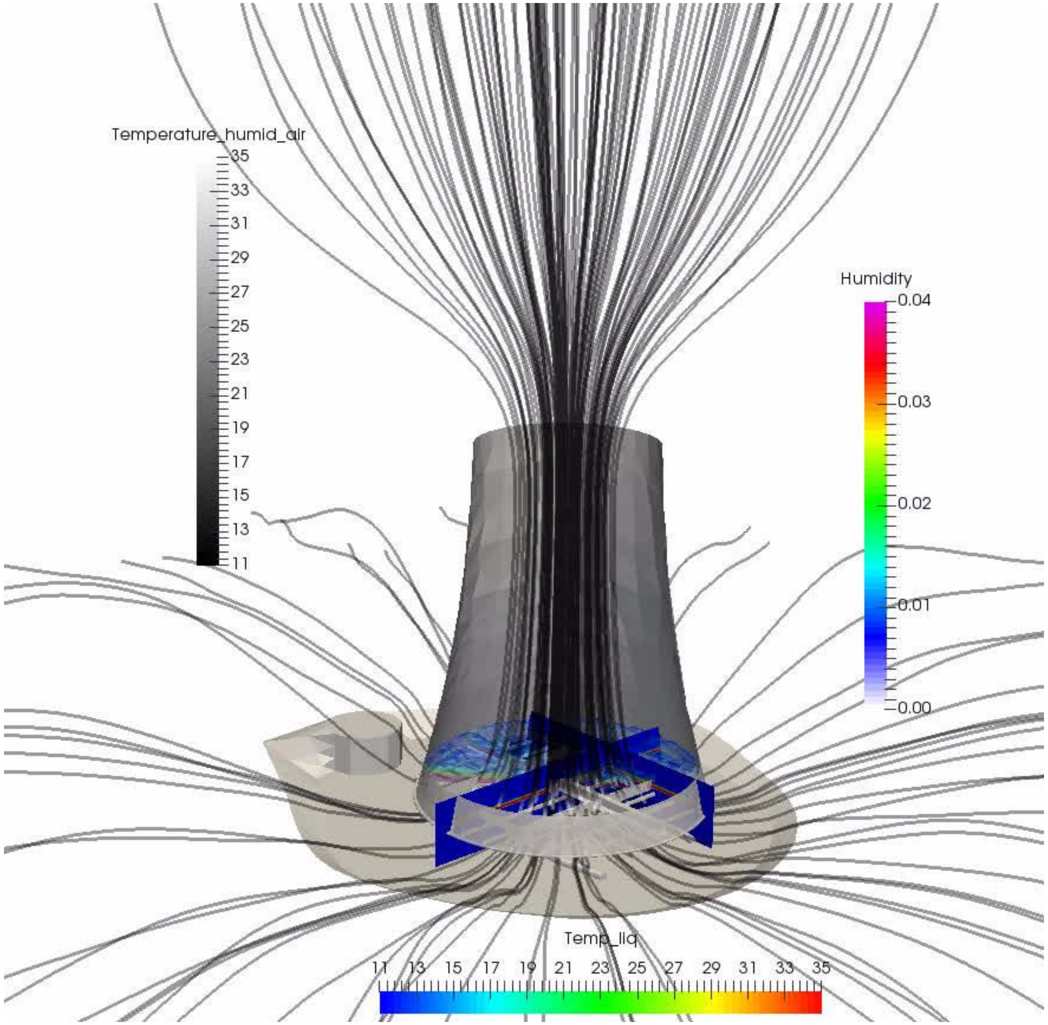
- Then refined using SALOME's Homard module



Golfech Results

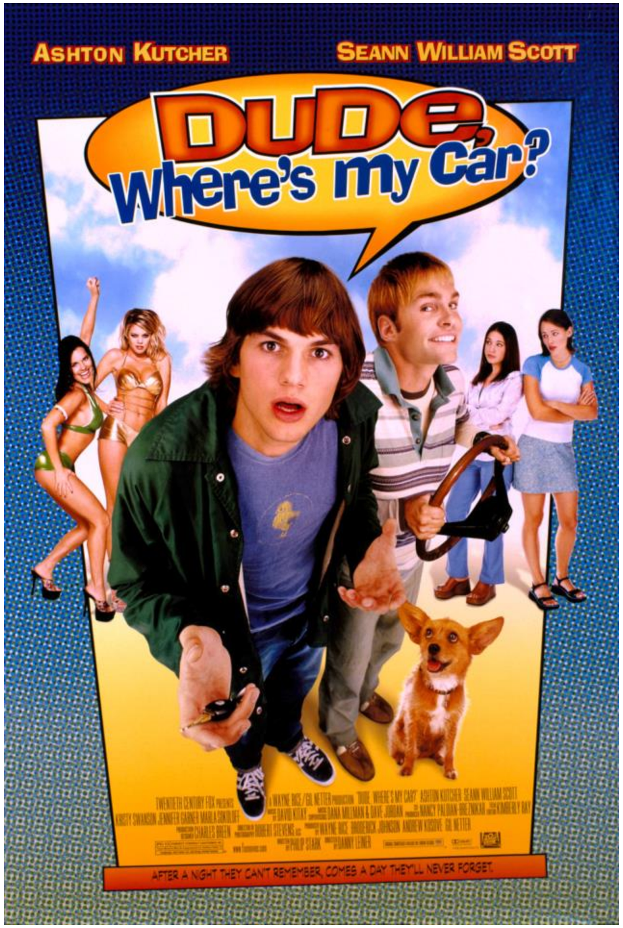


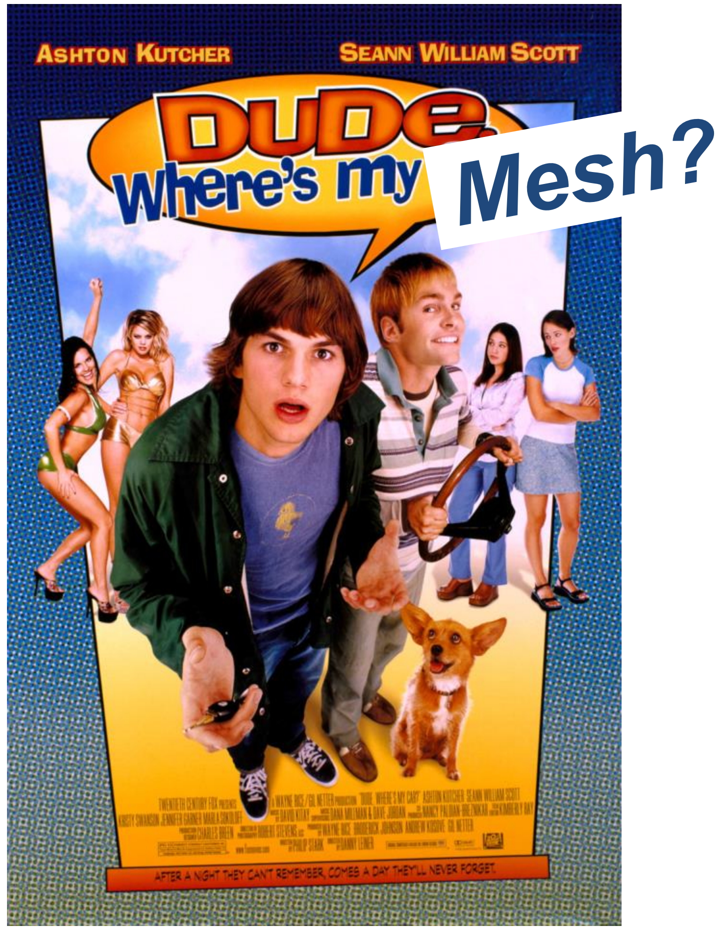
Golfech Results



3. CFD on non-Hexahedral Meshes

Fundamentals

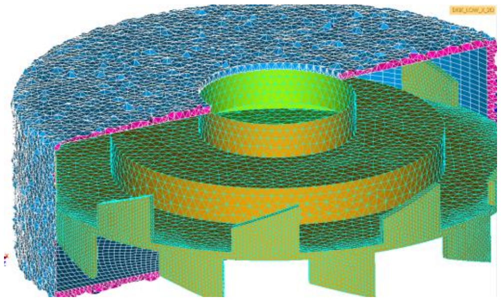


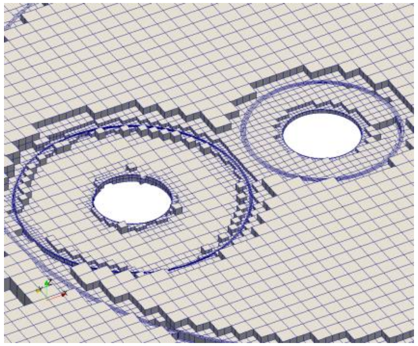

ASHTON KUTCHER
SEANN WILLIAM SCOTT
DUDE,
Where's my
Mesh?
AFTER A NIGHT THEY CAN'T REMEMBER, COMES A DAY THEY'LL NEVER FORGET.

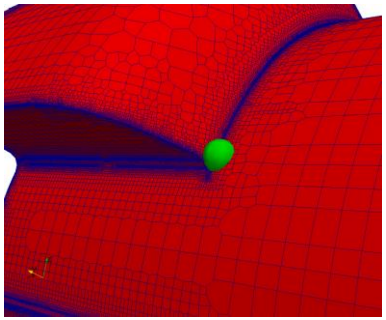

Mesh?
Solver
Model
Meshing Tools
User

High Fidelity Simulations

- Accurate simulations of turbulent flow fields, particularly for more direct methods such as Large Eddy Simulations (LES) often are best conducted on conformal, purely hexahedral mesh topologies
- However, in general,
  - Industrial geometries have complex shapes
  - Large disparity of scales, details which might be significant, objects separated by gaps, interfaces, etc.







High Fidelity Turbulence Simulations

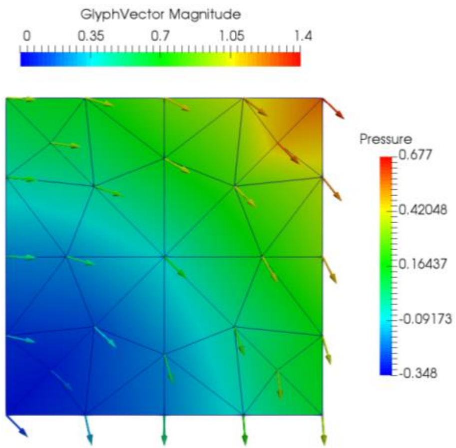
- This is very challenging for meshing methodologies
  → Tetrahedral and hybrid meshes
  → Solver

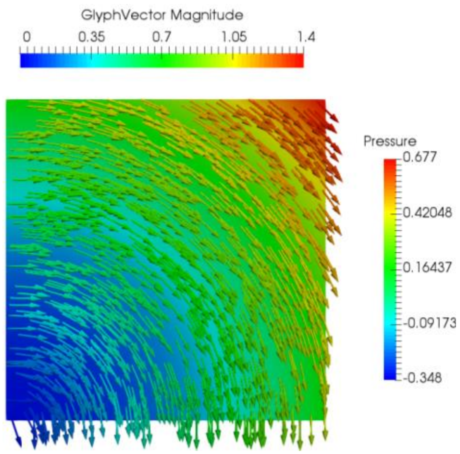
- On-going research and development to validate such simulations on tetrahedral and hybrid meshes
  - Meshing tools: Netgen and MGTetra within SALOME's SMESH module
  - *Code_Saturne* internal meshing capabilities
  - Solver

- Validation and evaluation
  - Benchmarks – e.g. FVCA8
  - Fundamental cases – e.g. orifice flow
  - Geometries of industrial relevance – nuclear reactors
    - Modelling of experiments
    - Modelling of real systems

FVCA8 Conference

- Finite Volumes for Complex Applications 8*
- Benchmark case 3.1 – simple vortex, steady
  - Vary viscosity
  - 6 meshes of increasing density





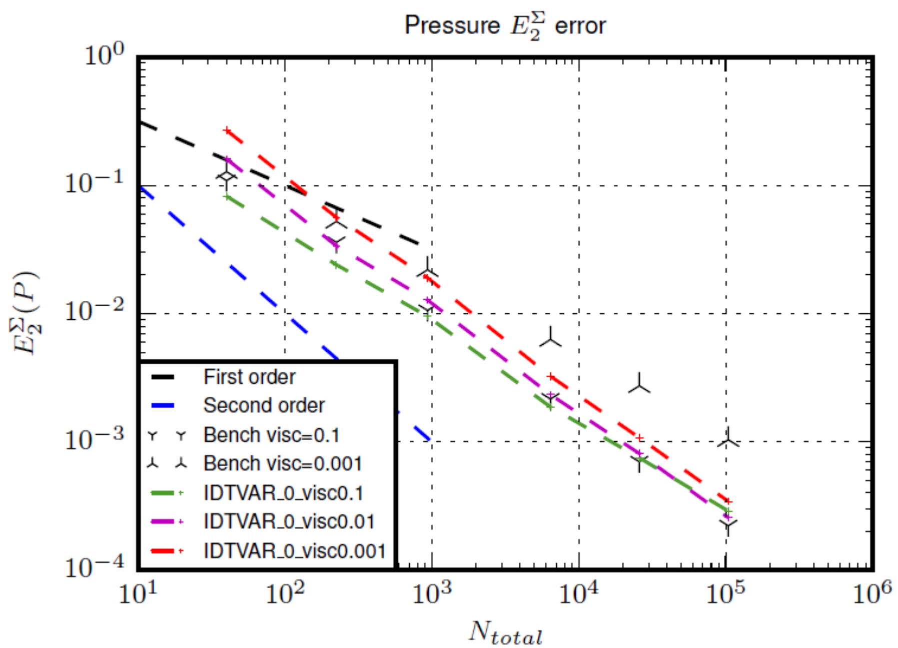
- Systematic variations of numerical parameters

*https://indico.math.cnrs.fr/event/1299/page/5, 12-16 June 2017, Lille, France

Benchmarks

- Convergence graph for FVCA8

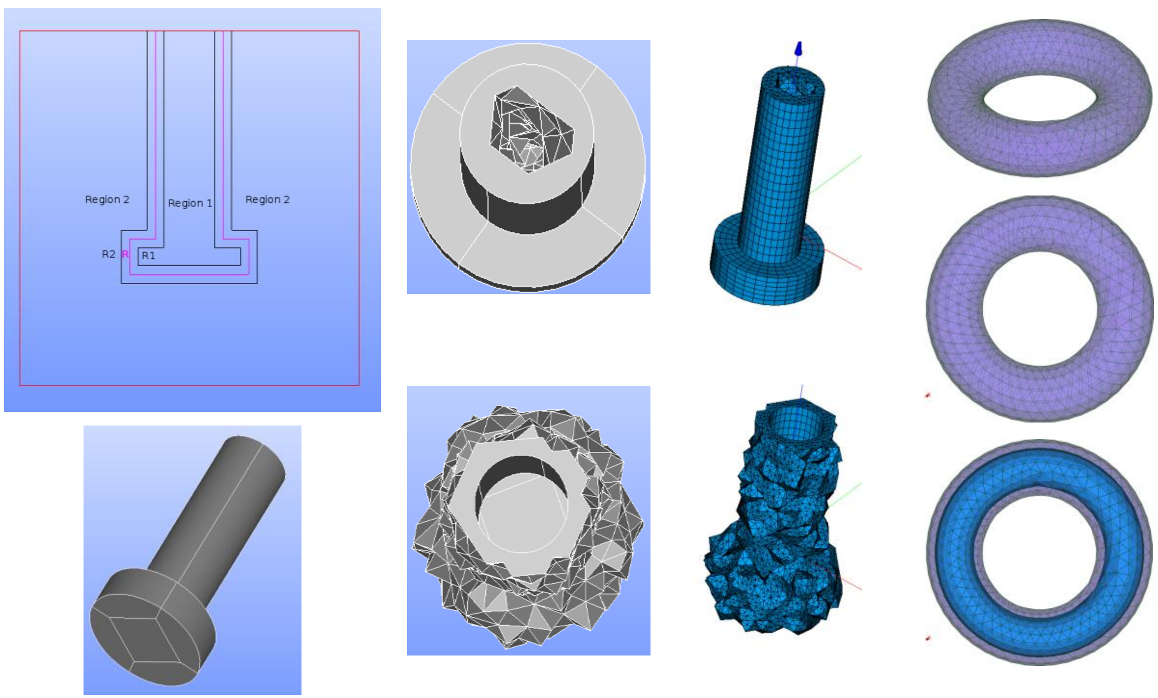
Compared with Angeli et al.'s data

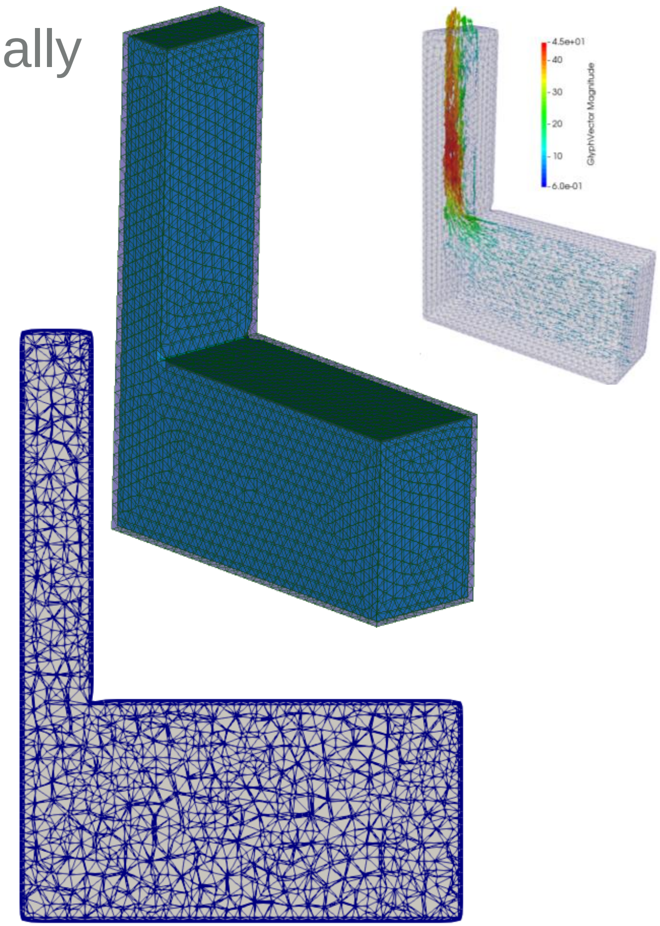


- 211 cases are being evaluated – some of them several times to adapt the parameters

Internal Meshing

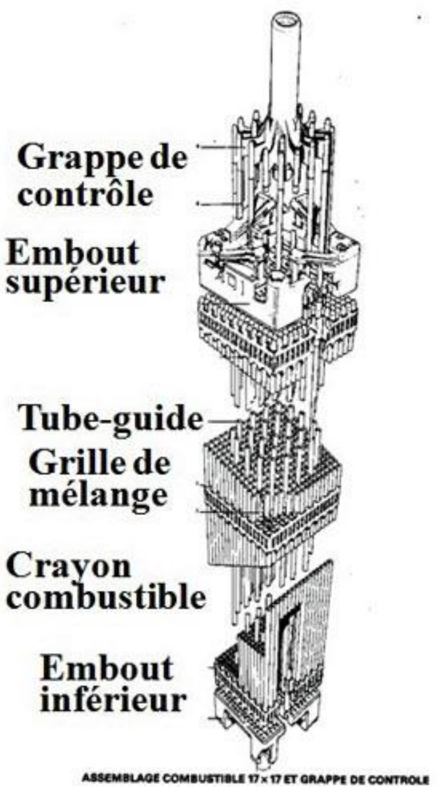
- One of the key elements remains the wall layers
- Collaborative effort to build wall layers internally using node motion algorithms

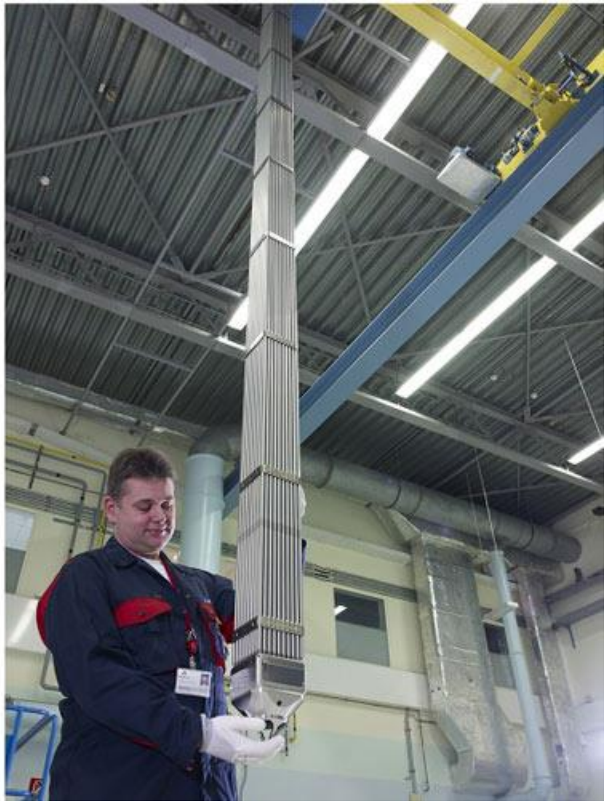




Industrial Cases

- Where it (will) all come(s) together



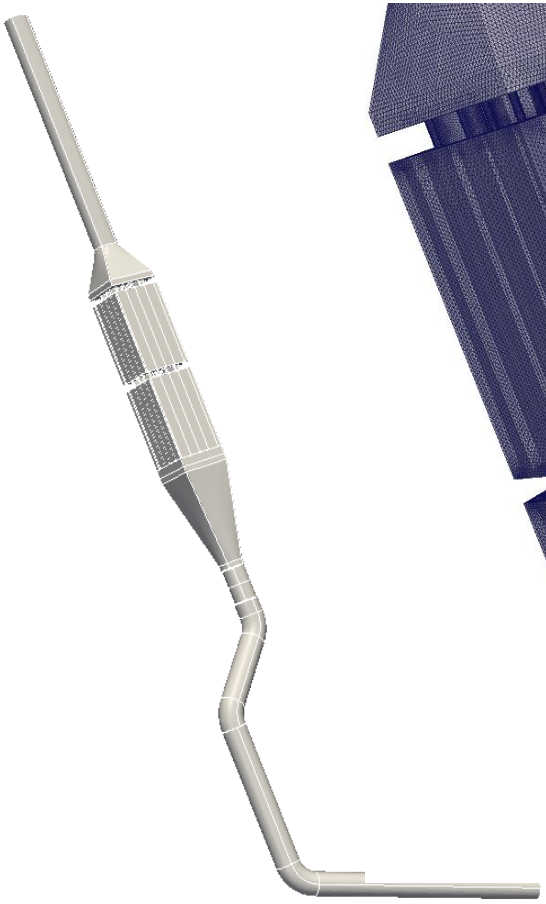


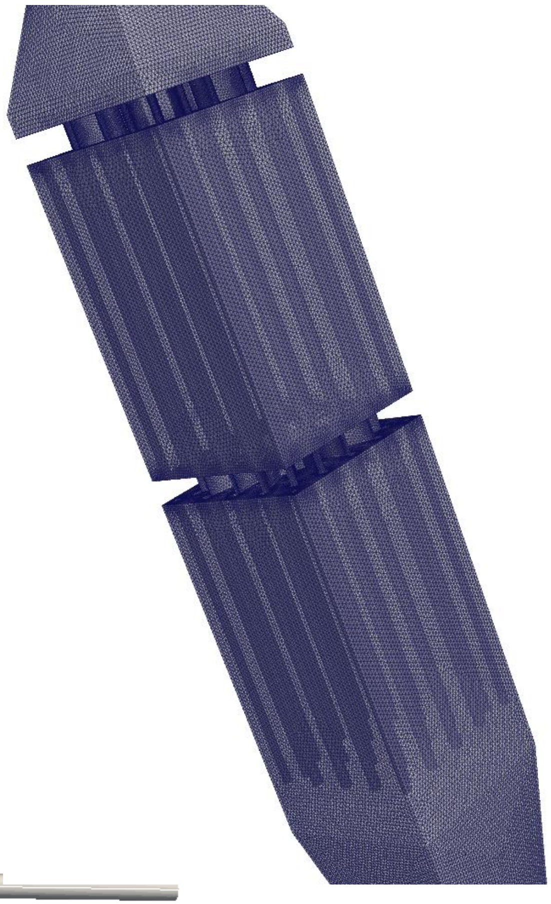
(Some of the) Challenges:
- Turbulence
- Vibration
- Deformation
- Multiple assemblies
- Etc.

Illustration only: "Assemblage du combustible nucléaire dans un réacteur à eau pressurisée" ©AREVA www.encyclopedie-environnement.org/physique/l-energie-nucleaire/

Control Rods Guide Plates

- Modelling of an experimental setup

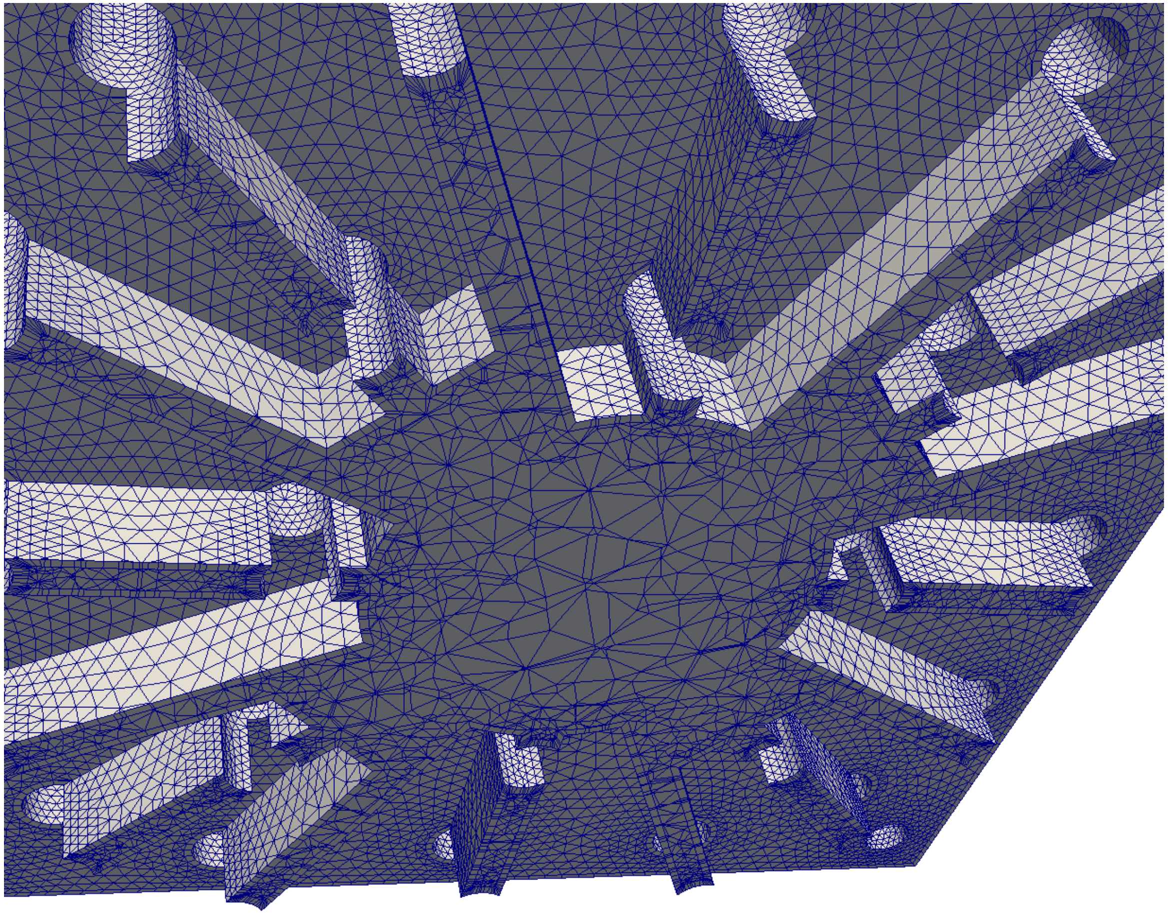


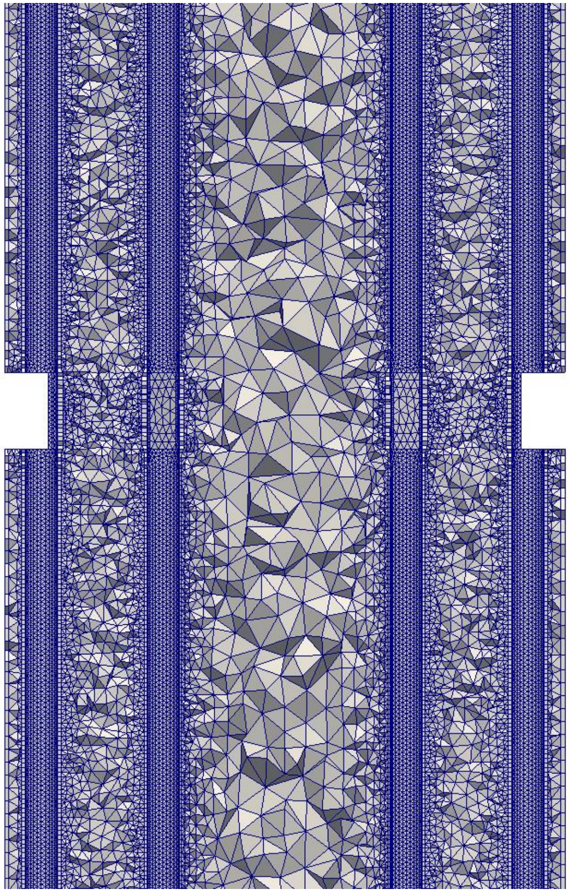


- CAD rebuilt from original
- Cleaning
- Joining
- Separation of the different surfaces and volumes for analysis

Control Rods Guide Plates

- Very detailed tube surfaces and guide plates

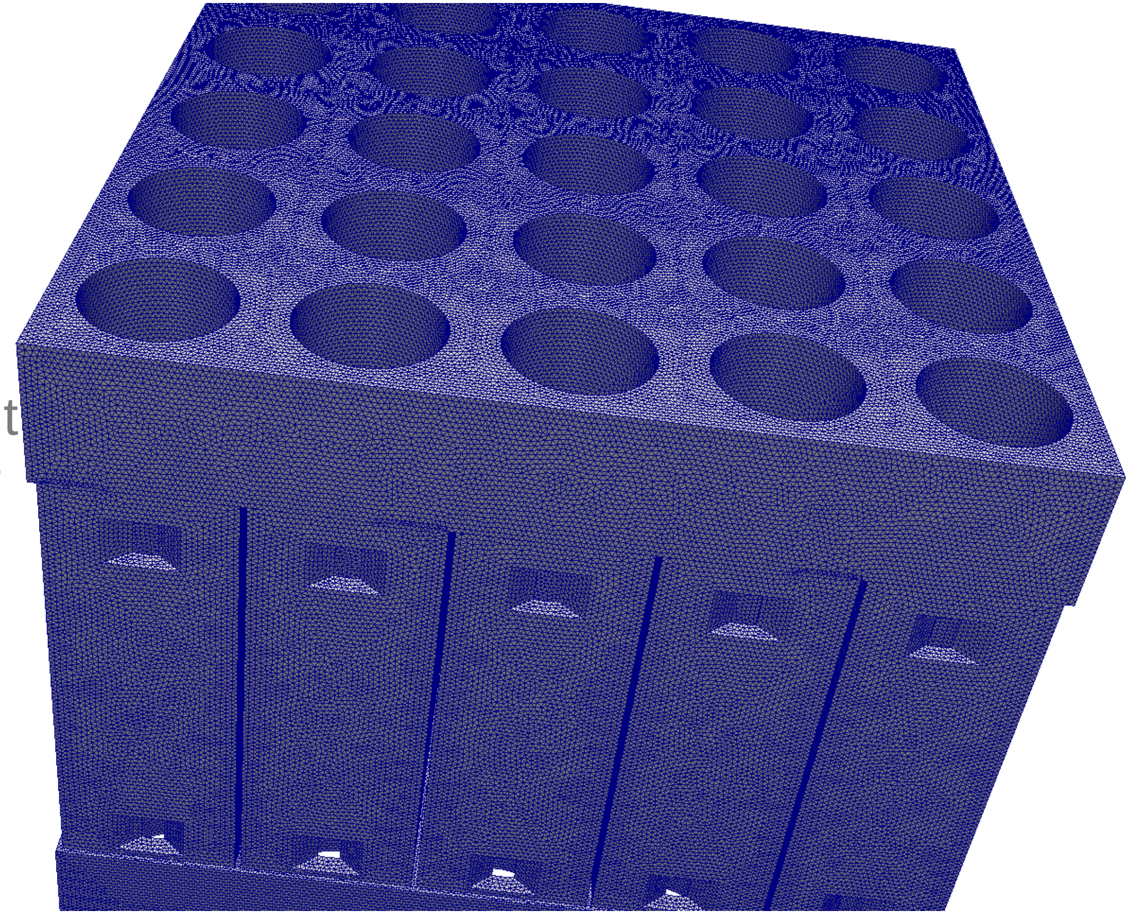




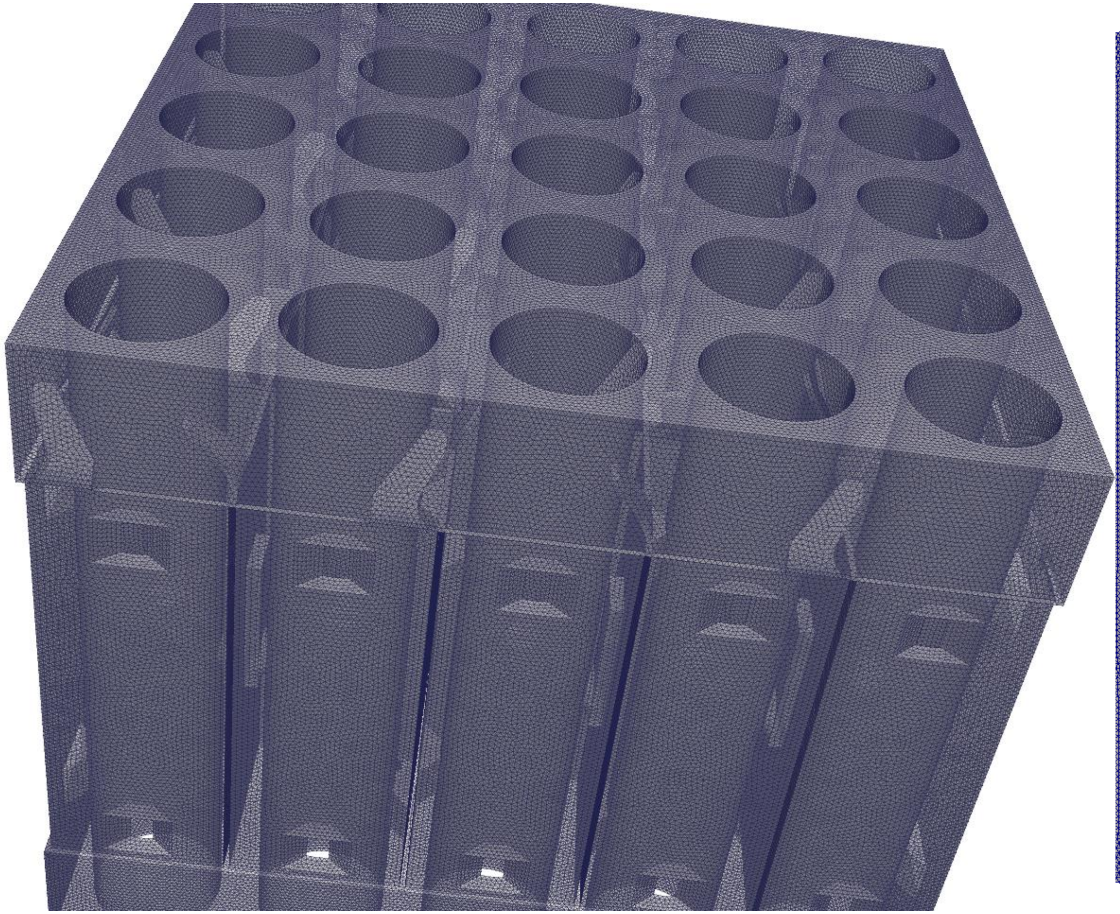
CALIFS

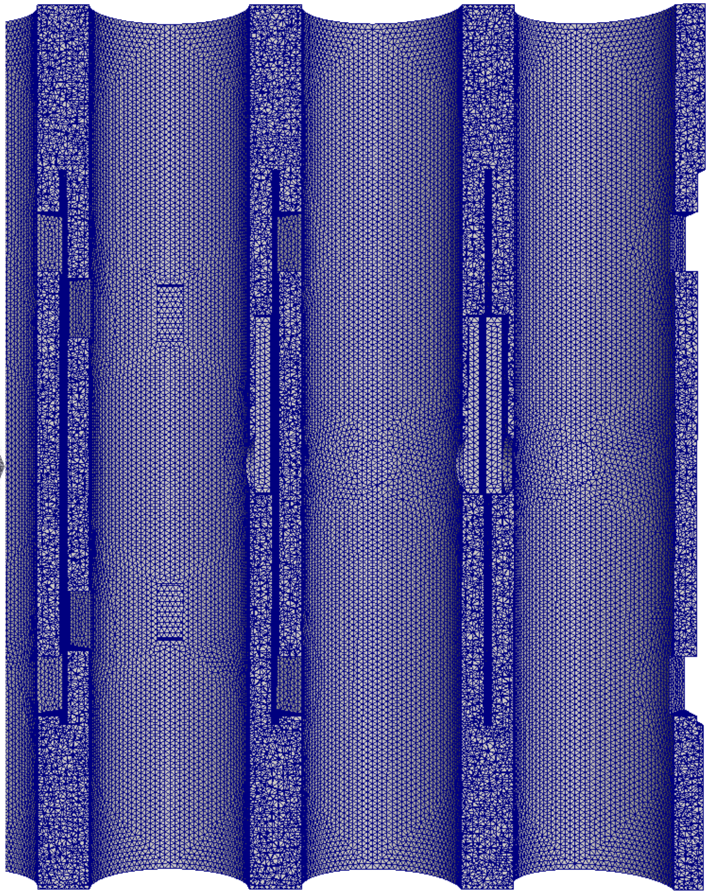
- Modelling of an experimental setup

- CAD rebuilt from original
- Cleaning
- Joining
- Separation of the different surfaces and volumes for analysis



- Modelling of an experimental setup





4. Conclusions and Future Work

Conclusions and Future Work

- *Code_Saturne* and SALOME form a very powerful combination, which Renuda is applying in very different industrial settings and purposes. Renuda is also participating in its development
- The integration within SALOME's set of tools gives a lot of flexibility to prepare and modify models and to mesh them
- Work is on-going on model development and general meshes CFD
- It is extremely good to see and collaborate on the all-encompassing research and development work taking place which takes into account the entire chain of CFD requirements, and including the combination of meshing approaches
  - This will deliver unique and empowering capabilities based on a solid scientific and validated foundation → not just features but also best practices

Manchester 5^{th} and 6^{th} of June 2018

CODE SATURNE

SALOME

eDF

RENUDA FLUID SOLUTIONS

5^{th} of June: *Code_Saturne* UK User Conference

6^{th} of June: Introduction to CFD analysis using *Code_Saturne* and SALOME