

#### **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

Siempelkamp

www.renuda.com

- Compete on
  - Skills
  - Difficult problems

FLUID SOLUTIONS © Renuda ® 2018



#### **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



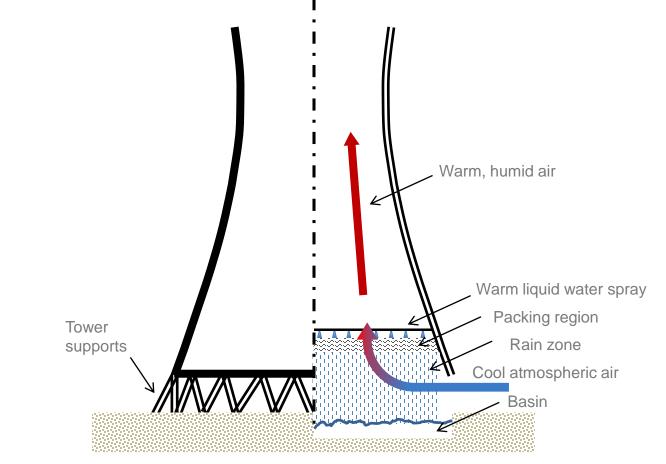
#### 2. Cooling Towers Modelling





## 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*, 18<sup>th</sup> 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

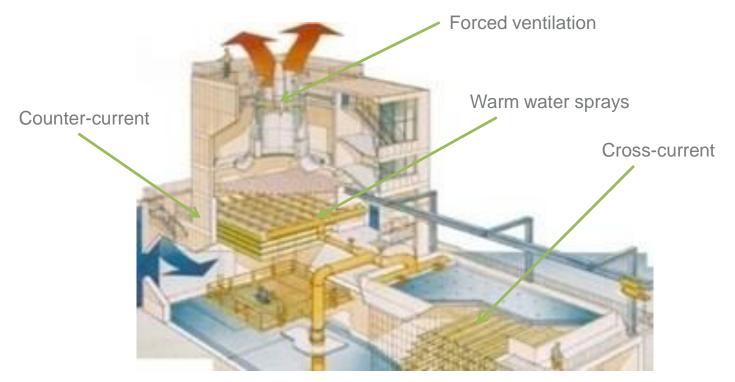
buget_april_3D_SquareCore_Wind.xml - Code_Saturne GUI - 5.1-alpha 🔶 🗉 X		File Edit Options Buffers Tools F90 Help	
<u>File Edit Tools Window Help</u>		🔽 🚞 🐹 🗸 Save 🧠 Undo 🕌 💿 🔐 🔍	emacs@ukdesktop1 + _ □ ×
📑 🚍 🖎 🚺 🚯 🕐 🔠 🗾 💀			File Edit Options Buffers Tools C Help
	8 10	!	🔒 🚍 📰 🗱 👵 Save 👒 Undo 🕌 💿 😭 🔍
Study: BUGEY_02October2017		<pre>!&lt; [cs_user_cooling_towers]</pre>	U void
Case: Tower_3D_SquareCore		1	cs_user_parameters(void)
XML file: ship/Cooling_Towers/BUGEY_02October2017/Tower_3D_SquareCore/DATA/buget_april_3D_SquareCore_Wind.xml		1. Physical properties	1 /*
08		!	* We define a cooling tower zone */
Identity and paths     Galculation environment		! Used to compute the humid air density as a function of (P,T,h ro0 = 1.293d0	/*! [ctwr_user_1] */
Volume conditions BC_5 2 Outlet	Тор	! Dry air and water vapour properties cp_a = 1006.0d0	<pre>cs_ctwr_option_t *ct_opt = cs_get_glob_ctwr_option();</pre>
Definition of boundary regions     Boundary conditions     Boundary conditions     BC_8 3     Wall     BC_9 4     Outlet	Border_pX Border_mX	cp_v = 1831.0d0 ☐ ! Initial absolute humidity	/* Evaporation model: CS_CTWR_NONE None, CS_CTWR_POPPE Poppe,
Global parameters BC_11 6 Free inlet/outlet BC_12 7 Free inlet/outlet Time step	Border_mZ	humidity0 = 0.d0 ! Humid air viscosity and conductivity - considered constant in	CS_CTWR_MERKEL Merkel*/ ct_opt->evap_model = CS_CTWR_POPPE;
Calculation control     Time averages     Output control		visclo = 1.7650-5 lambda_h = 2.493d0	cs_real_t surface = 49.0*0.5; // surface 3D = 49.0#2 ; /* Total disc */ cs_real_t qw = 151.4*0.5; //Debit 3D = 151.4 pour 49#2; /* Water flow rate (kg/s) */
Volume solution control , Surface solution control Profiles Balance by zone Add		! Liquid water properties rho_l = 997.85615d0	cs_ctwr_define(
Calculation management     Start/Restart     Add from Preprocessor listing	belete	cp_l = 4179.0d0 lambda_l = 0.02493d0	CS_CTWR_COUNTER_CURRENT counter current, CS_CTWR_CROSS_CURRENT cross, CS_CTWR_CROSS_CURRENT cross,
Prepare batch calculation Import groups and references f		! Phase change properties hv0 = 2501600.0d0	-1., /* Imposed delta temperature if positive */ 0.1, /* Associated relaxation time */ 36.2, /* Liquid injected water temperature */
		droplet_diam = 0.005	qw, 0.7572, /* Evaporation law constant A */ 0.6818, /* Evaporation law constant n */
		!< [cs_user_cooling_towers] !	<pre>surface, 1.); /* Leaking factor, not taken into account if negative */</pre>
1		! End	} /*! [ctwr_user_1] */
		-: cs_user_parameters.f90 90% L270 (F90)	}
			-: cs_user_parameters-ctwr.c 76% L121 (C/l Abbrev)

FLUID SOLUTIONS © Renuda ® 2018



#### MISTRAL

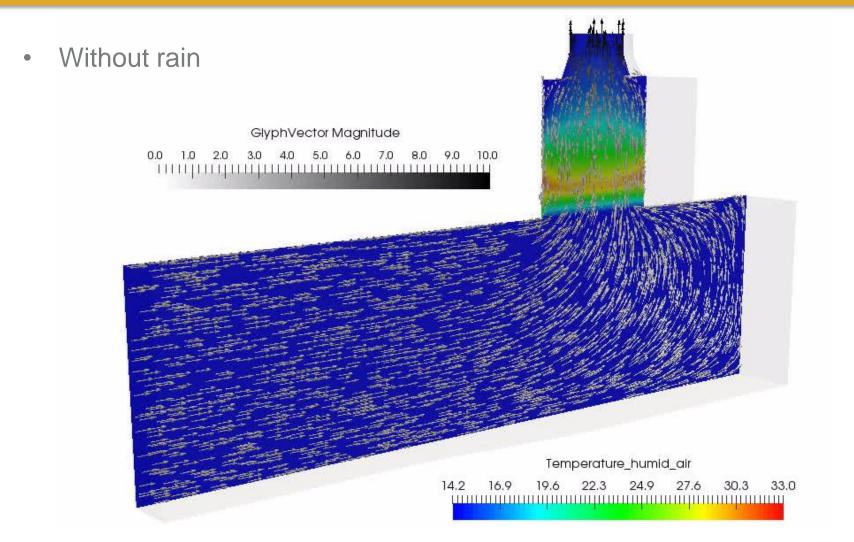
 Experimental test bed which is part of the Bugey electricity production plant – test and validate real size equipment such as packing material







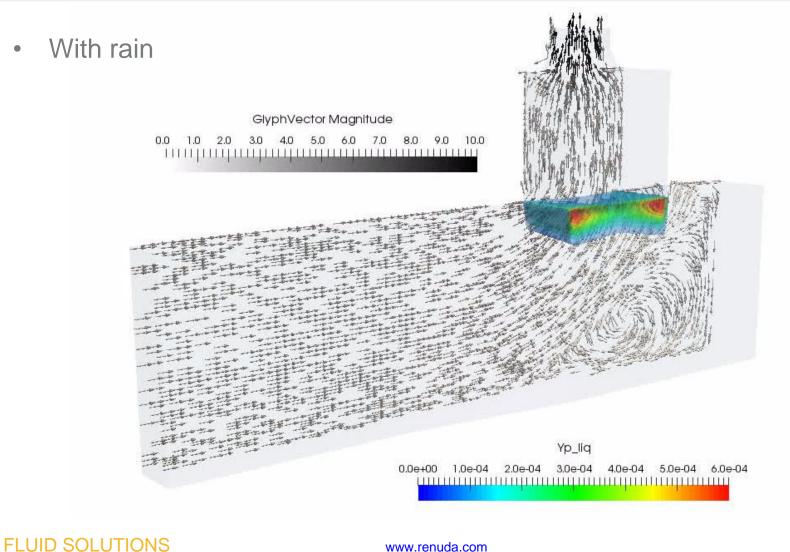
#### **MISTRAL** results



FLUID SOLUTIONS © Renuda ® 2018



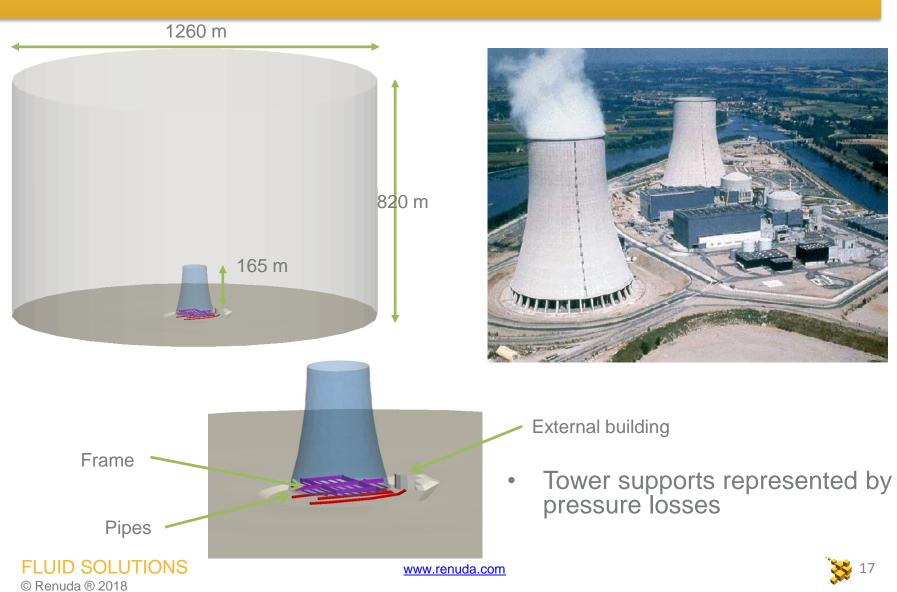
#### **MISTRAL** results





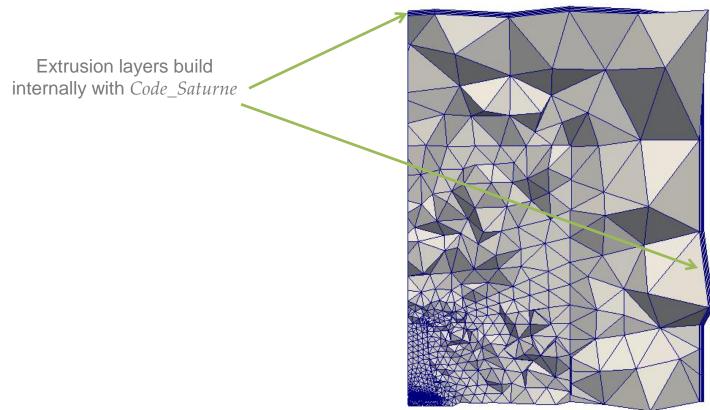
© Renuda ® 2018

#### Golfech



#### **Tetrahedral Mesh**

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



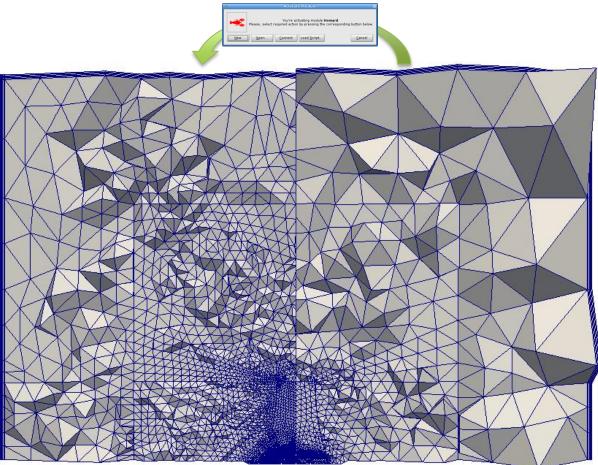






#### **Tetrahedral Mesh**

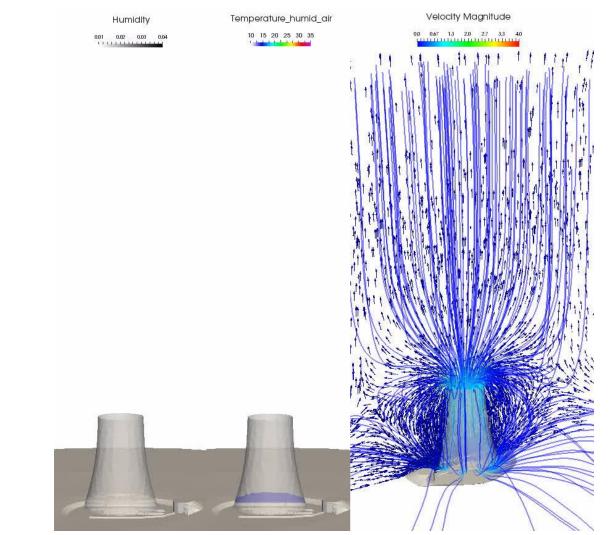
• Then refined using SALOME's Homard module







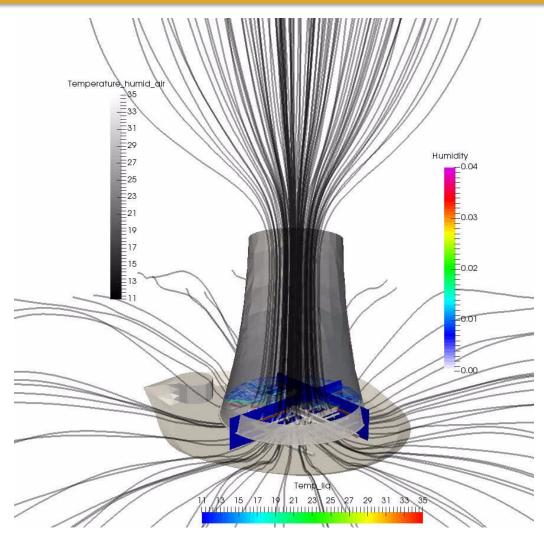
#### **Golfech Results**



FLUID SOLUTIONS © Renuda ® 2018



#### **Golfech Results**



FLUID SOLUTIONS © Renuda ® 2018



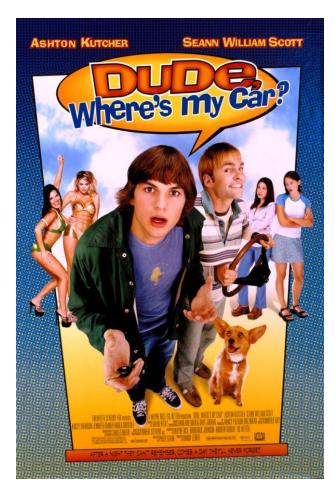
#### 3. CFD on non-Hexahedral Meshes







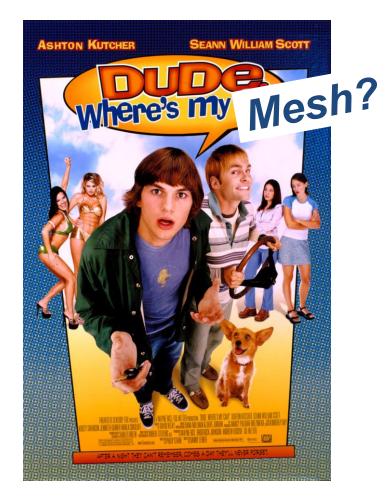
#### **Fundamentals**







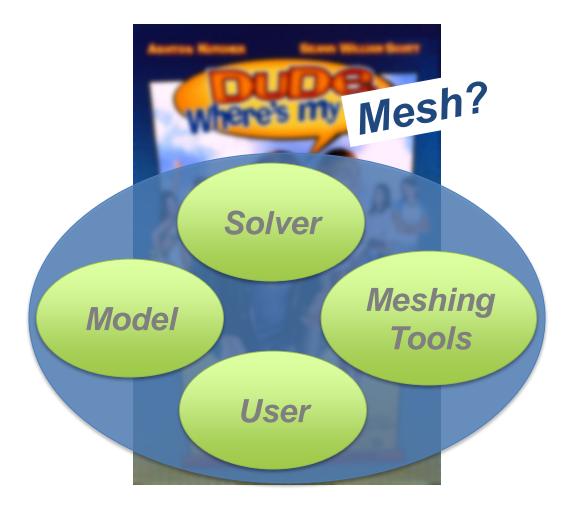
#### **CFD** Fundamentals







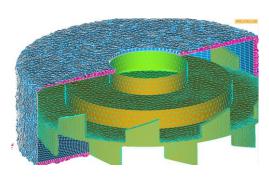
#### **CFD** Fundamentals



FLUID SOLUTIONS © Renuda ® 2018

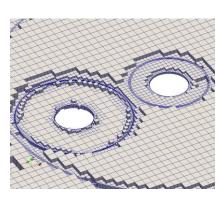


- Accurate simulations of turbulent flow fields, particularly for more direct methods such as Large Eddy Simulations (LES) often are best conducted on conformal, purely hexahedral mesh topologies
- However, in general,
  - Industrial geometries have complex shapes
  - Large disparity of scales, details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

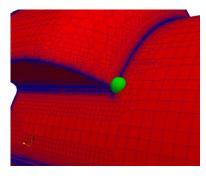


FLUID SOLUTIONS

© Renuda ® 2018









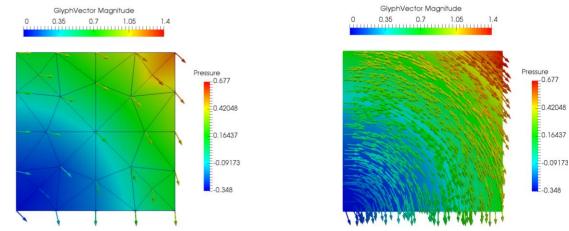
## High Fidelity Turbulence Simulations

- 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

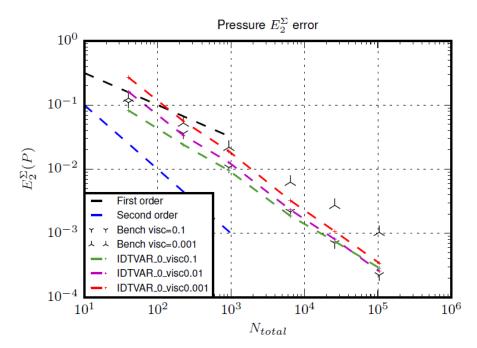
\*https://indico.math.cnrs.fr/event/1299/page/5, 12-16 June 2017, Lille, France



#### **Benchmarks**

• Convergence graph for FVCA8





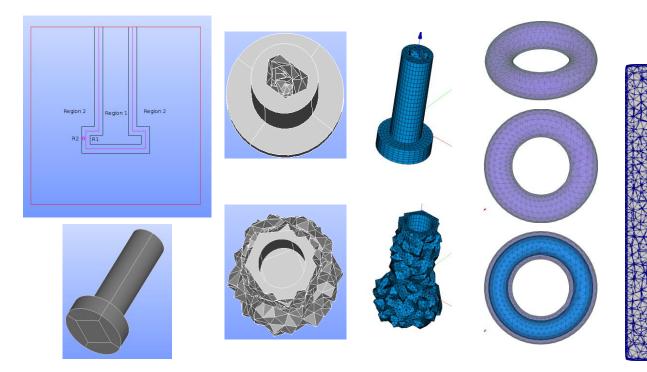
 211 cases are being evaluated – some of them several times to adapt the parameters

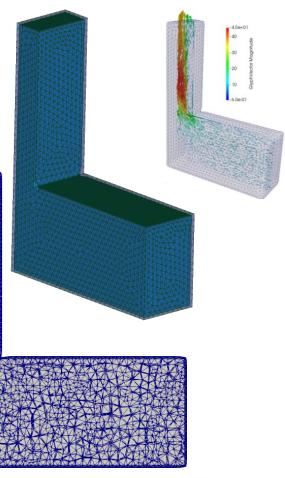




## **Internal Meshing**

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



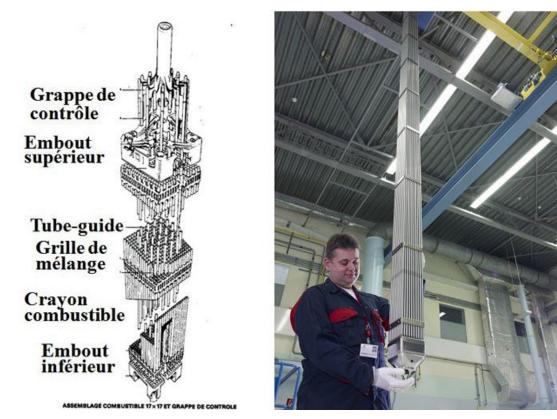


FLUID SOLUTIONS © Renuda ® 2018



#### **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/

FLUID SOLUTIONS © Renuda ® 2018



### **Control Rods Guide Plates**

• Modelling of an experimental setup

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

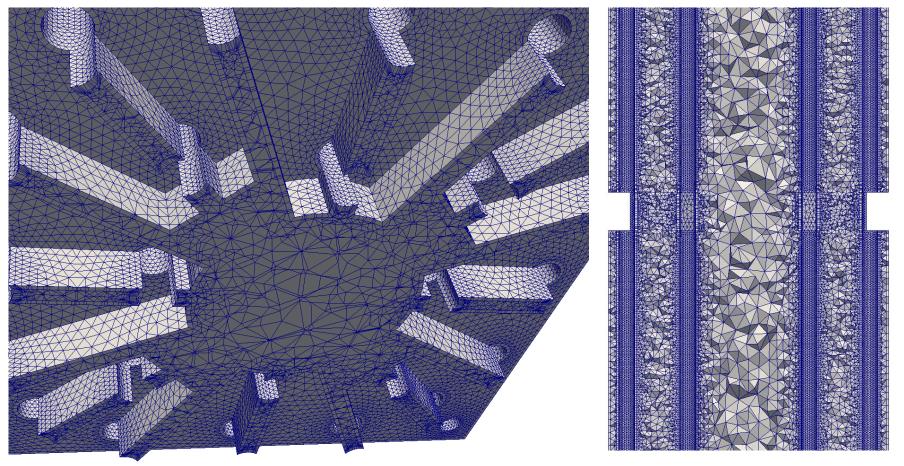






#### **Control Rods Guide Plates**

• Very detailed tube surfaces and guide plates



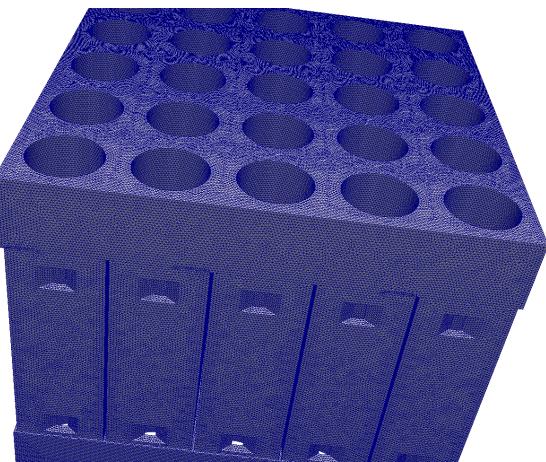
FLUID SOLUTIONS © Renuda ® 2018



#### CALIFS

• Modelling of an experimental setup

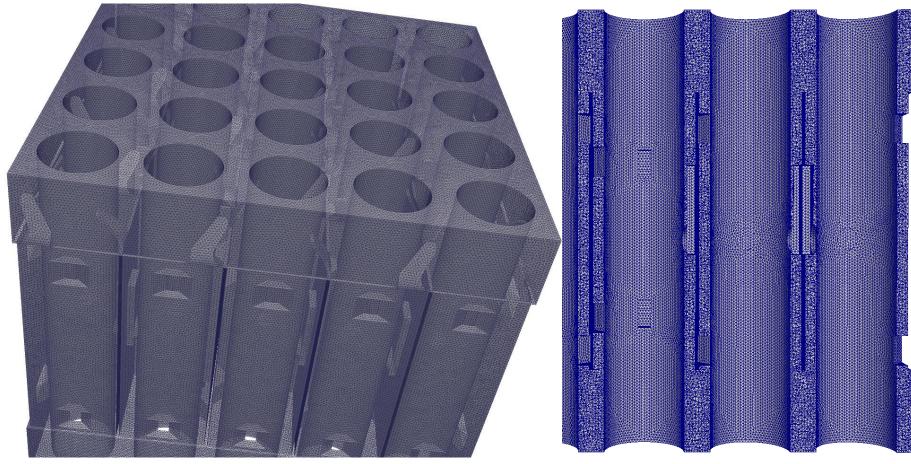
- CAD rebuilt from original
- Cleaning
- Joining
- Separation of the different surfaces and volumes for analysis





#### CALIFS

😽 35



• Modelling of an experimental setup

FLUID SOLUTIONS © Renuda ® 2018

#### 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<sup>th</sup> and 6<sup>th</sup> of June 2018











5<sup>th</sup> of June: *Code\_Saturne* UK User Conference 6<sup>th</sup> of June: Introduction to CFD analysis using *Code\_Saturne* and SALOME

#### FLUID SOLUTIONS © Renuda ® 2018



