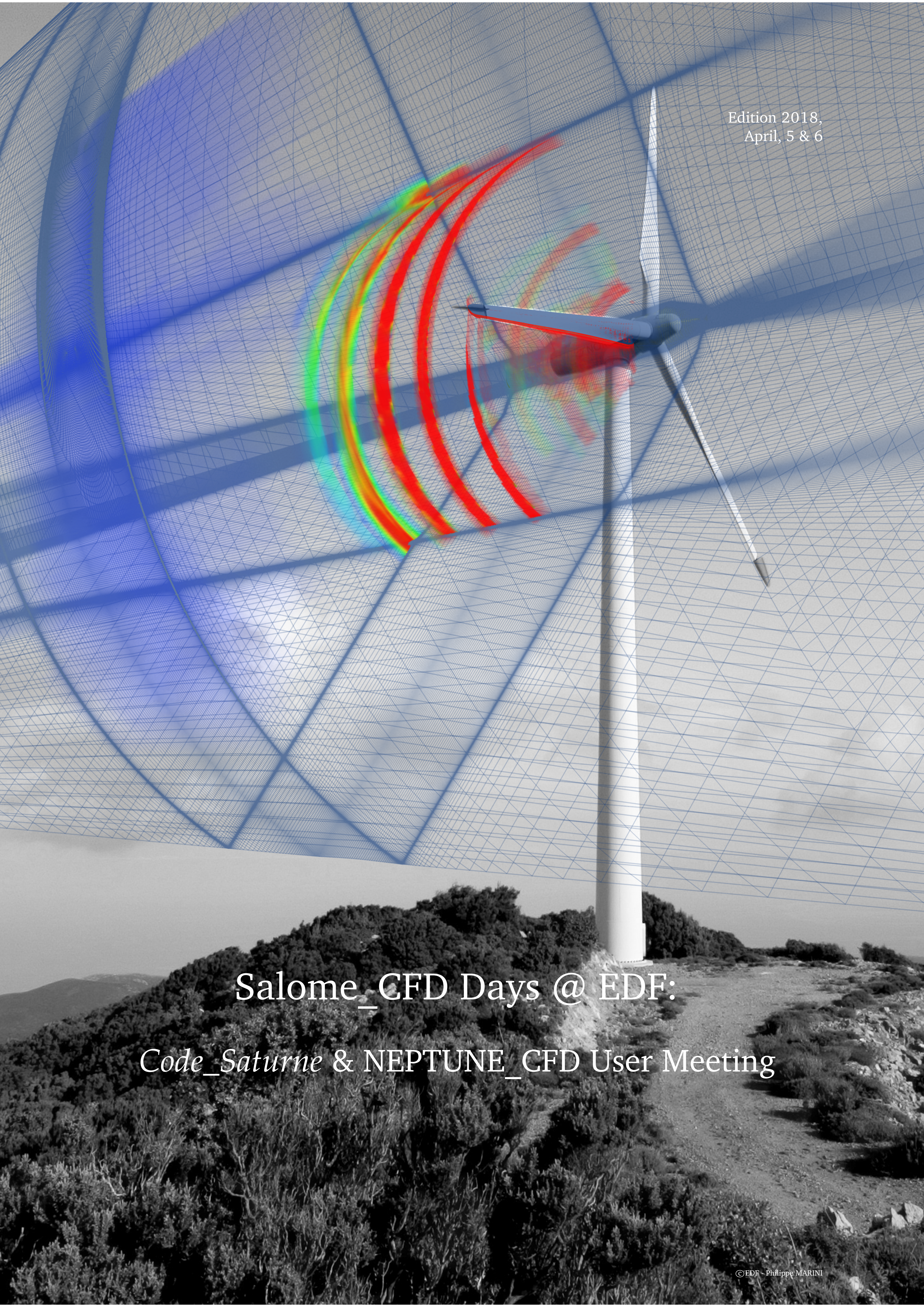


Edition 2018,
April, 5 & 6

Salome_CFD Days @ EDF:

Code_Saturne & NEPTUNE_CFD User Meeting

Code_Saturne user day

Program

WebSite *BugTracker* *Forum*

8:00	**Welcome – Breakfast**	
9:00	Foreword	M. Ferrand *Code_Saturne* project leader
9:05	Introduction	M. Boucker Deputy Head of Nuclear Future Initiatives
9:20	Latest news and prospects in *Code_Saturne* and Salome_CFD	*Code_Saturne* Dev. Team EDF R&D - MFEE
9:50	CFD-Uncertainty Quantification benchmark on Cold Leg Mixing from OECD	R. Camy EDF DT - THL
10:10	Wall-Modelled Large Eddy Simulation of the flow through PWR fuel assemblies at $Re = 66000$ – Validation on CALIFS experimental setup	M.-C. Gauffre EDF R&D - MFEE
10:30	**Break**	
11:00	Hot Gas Release in a Gas Circulator Hall	L. Rouault EDF R&D UK Centre
11:20	*Code_Saturne* in China: 2017-2018 activity overview	T. Xu EDF China R&D Center
11:40	Overtopping flows simulations with the volume of fluid module of *Code_Saturne*	Y. Bercovitz EDF R&D - LNHE
12:00	**Lunch**	
13:30	Cooling and ventilation applications and fundamental validations of *Code_Saturne* on different mesh types	N. Tonello Renuda
13:50	Development of an accurate and efficient compressible algorithm in *Code_Saturne* for the modelling of high-speed compressors	A. Heffron Queen Mary Uni.
14:10	Heat exchanger multilevel modelling and optimization using *Code_Saturne*	F. Mastrippolito CEA Tech LITEN, LMFA ECL Lyon, Valeo & EDF
14:30	**Break - Poster and live demonstration session**	
16:00	Review of 2017/2018 *Code_Saturne* Atmospheric Module Applications at ARIA Tech.	L. Makke & M. Nibart ARIA Technologies
16:20	WRAPP - A comprehensive methodology to estimate the wind resource, production and wake effects	G. Angot EDF R&D - MFEE
16:40	Modelling frozen salt walls in molten salt fast reactors	S. Rolfo STFC, UK
16:30	Closure	D. Banner Head of the EDF R&D simulation program
	Closing Reception	

CFD-Uncertainty Quantification benchmark on Cold Leg Mixing from OECD

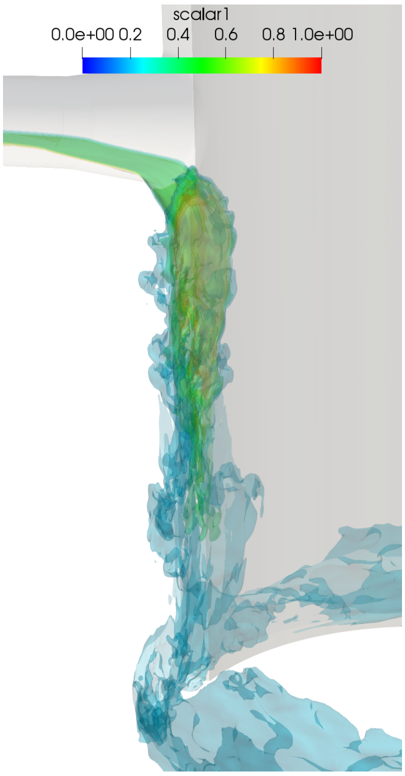
by R. Camy – EDF DT/THL

In 2015-2016 a first benchmark on Uncertainty Quantification of CFD simulations was organized by OECD/NEA. The experiment for this benchmark was called GEMIX. In 2017-2018 a new benchmark has started. It is called “Cold Leg Mixing CFD-UQ” and the experiment is located at Texas A&M University in the USA.

Briefly, the experiment consists in a barrel connected to an annular space by a tube initially closed with a valve. The barrel side is filled with saline water and the other side with a mixture of water and ethanol. At the valve opening, the density difference between the two fluids leads to the propagation of a heavy fluid front. First in the tube then in the annular space. This propagation is meant to be representative of what would happen in a nuclear transient of Pressurized Thermal Shock (PTS) with Emergency Core Cooling (ECC) injecting cold fluid in the hot water in the vessel.

The facility is a 1:12.5 scaled-down model made of transparent acrylic for flow and passive scalar visualization. The geometry of the junction between the tube and the annular space is representative of an actual Pressurized Water Reactor (PWR). The small scale leads to $Re < 2500$ everywhere in the tube. The participants to the benchmark will have access to open results from a test at a limited Froude number and will be compared on a blind test with a higher Froude Number.

The methodology used by EDF to participate to the benchmark involves 2 steps calculations. A preliminary calculation on a tetrahedral grid (with prism layers) and a RANS model aimed at verifying the general behaviour of the facility. Then, more advanced calculations that rely on information from previous step, with Large Eddy Simulation (LES) performed on fully hexahedral and conform grids to get results with an expected higher fidelity. The presentation highlights the grid sensitivity and the solving of the density in the loop on Navier-Stokes.

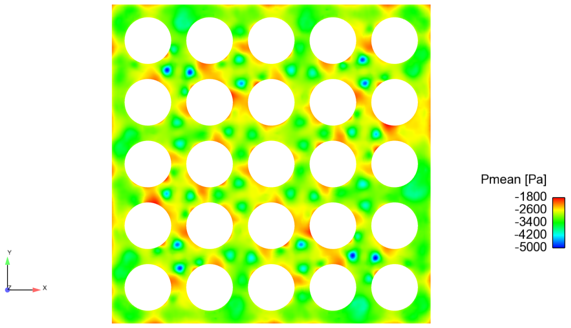


Wall-Modelled Large Eddy Simulation of the flow through PWR fuel assemblies at $Re = 66000$ – Validation on CALIFS experimental setup

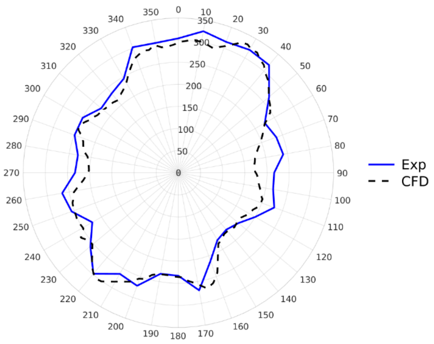
by S. Benhamadouche, M-C. Gauffre & P. Badel – EDF R&D / MFEE, ERMES

EDF aims at identifying what causes fuel assembly vibrations downstream mixing vane grids. Reasonable results have been obtained using wall-modeled LES computations on a 2×2 tri-periodic domain chained with deformation predictions [1], [2]. Indeed, the deformations were of the order of few microns, what is compatible with experimental observations. However, no validation of the wall-modeled LES approach was available. We started by a hydraulic validation on Matis-H experiment, a 5×5 Kaeri mixing vane grid [3], and we showed that the mean velocity and the Reynolds stresses are well predicted. The present work focuses on the validation of pressure fluctuations along the central rod of a 5×5 configuration.

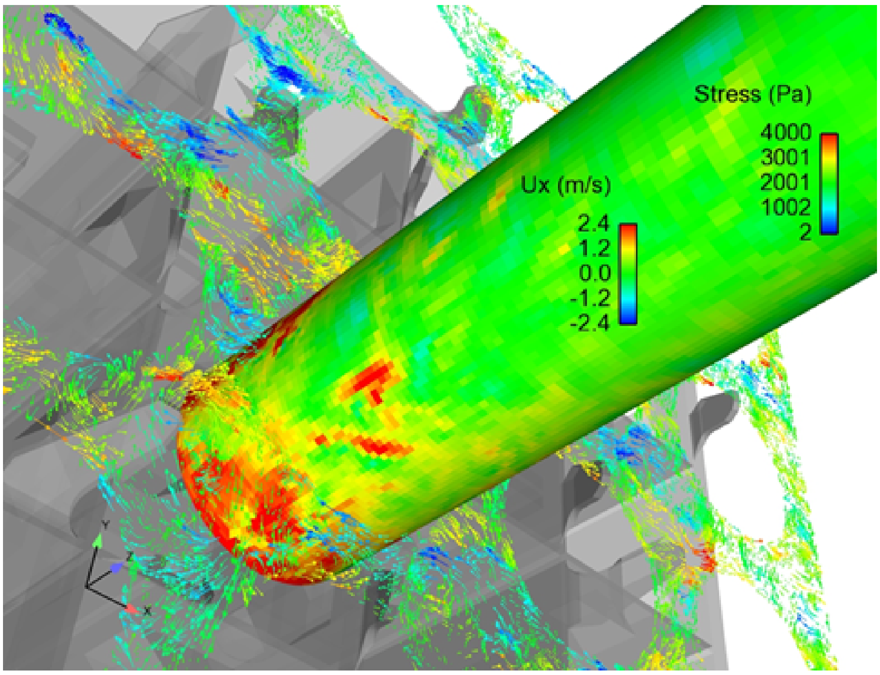
In the framework of Fuel Assembly EDF/CEA/AREVA I3P project, new experiments have been carried out by CEA (Atomic Energy Commission in Cadarache, France) on CALIFS configuration, a 5×5 mixing vane grid. In addition to pressure drop and velocity measurements using PIV, pressure measurements have been performed along the central rod. The computational domain is representative of a span of the experimental mock-up, composed of a 5×5 rod bundle with a grid with or without split-type mixing vanes (see Fig. *Pressure field one hydraulic diameter downstream a mixing vane grid*). Periodic boundary condition are used in the streamwise direction. Only conformal hexahedral meshes are used. The hydraulic Reynolds number is equal to 66000. A sensitivity study to the convection scheme and to the sub-grid scale model has been carried out. Most of the computations give very satisfactory results for the pressure drop, the mean velocity and the Reynolds stresses at different locations. The standard deviation of the pressure along the central rod is also compared to experimental data (see Fig. *Pressure standard deviation two hydraulic diameters downstream a mixing vane grid*). The behaviour is in very good agreement up to 5 hydraulic diameters downstream the mixing vane grid. Further downstream, more than half of the fluctuations seems to be filtered. There is no clear explanation for that at the moment.



Pressure field one hydraulic diameter downstream a mixing vane grid



Pressure standard deviation two hydraulic diameters downstream a mixing vane grid

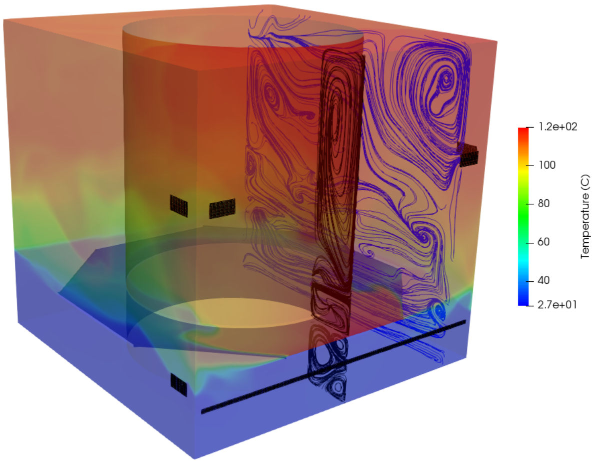


[1] S. Benhamadouche, P. Moussou, C. Le-Maître. "CFD estimation of the flow-induced vibrations of a fuel rod downstream a mixing grid", Proceedings of PVP 2009 ASME Pressure Vessels and Piping 2009 / Creep 8 Conference, July 22-26, Prague, Czech Republic (2009).
[2] P. Moussou, S. Benhamadouche, C. Bodel. "CFD estimation of the unsteady fluid force along a fuel rod downstream a mixing grid", Proceedings of PVP 2011 ASME Pressure Vessels and Piping 2011, July 17-21, 2011 Baltimore, Maryland (2011)
[3] L. Capone, Y. Hassan and S. Benhamadouche. "Large Eddy Simulation for 5x5 MaTis-H fuel bundle configuration using split vanes with *Code_Saturne*", The 15th International Topical Meeting on Nuclear Reactor Thermalhydraulics, NURETH-15, NURETH 15, May 12-17 Convenction Hall, Pisa Italy (2013).

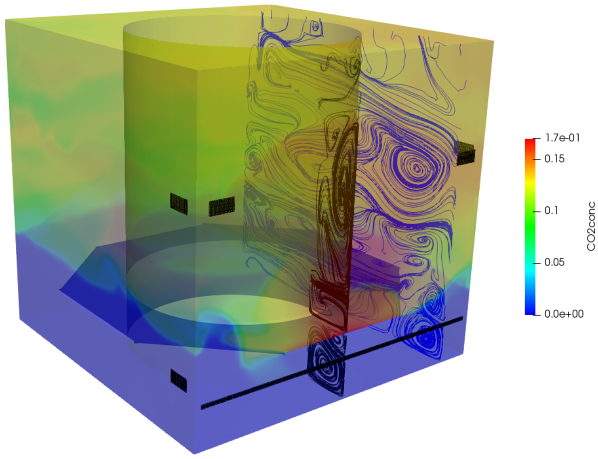
Hot Gas Release in a Gas Circulator Hall

by L. Rouault – EDF R&D UK Centre

The project presents the computation of a hot gas/steam release in a gas circulator hall using the CFD code *Code_Saturne*. The aim of the modelling is the study of accidental scenario of Loss of Coolant Accident (LOCA) in an Advanced Gas-cooled Reactor (AGR), where hot CO_2 or Steam leak from reactor's concrete pressure vessel into the reactor building. The idea being is to forecast the evolution of different parameters in the gas circulator hall as the pressure, the temperature and the CO_2 mass fraction. The main objective is to be able to give relevant information to operators in that specific case, so that, they would be able to enter in the room by respecting guidelines concerning CO_2 level of exposure. It is also important to ensure that the temperature limits of the components in the room are not reached. The presentation will focus on the physical hypothesis that have been made to take in account the leak and the properties of the room.



Temperature in the gas circulator after 30 minutes



CO_2 mass fraction in the gas circulator after one hour

Code_Saturne in China: 2017-2018 activity overview

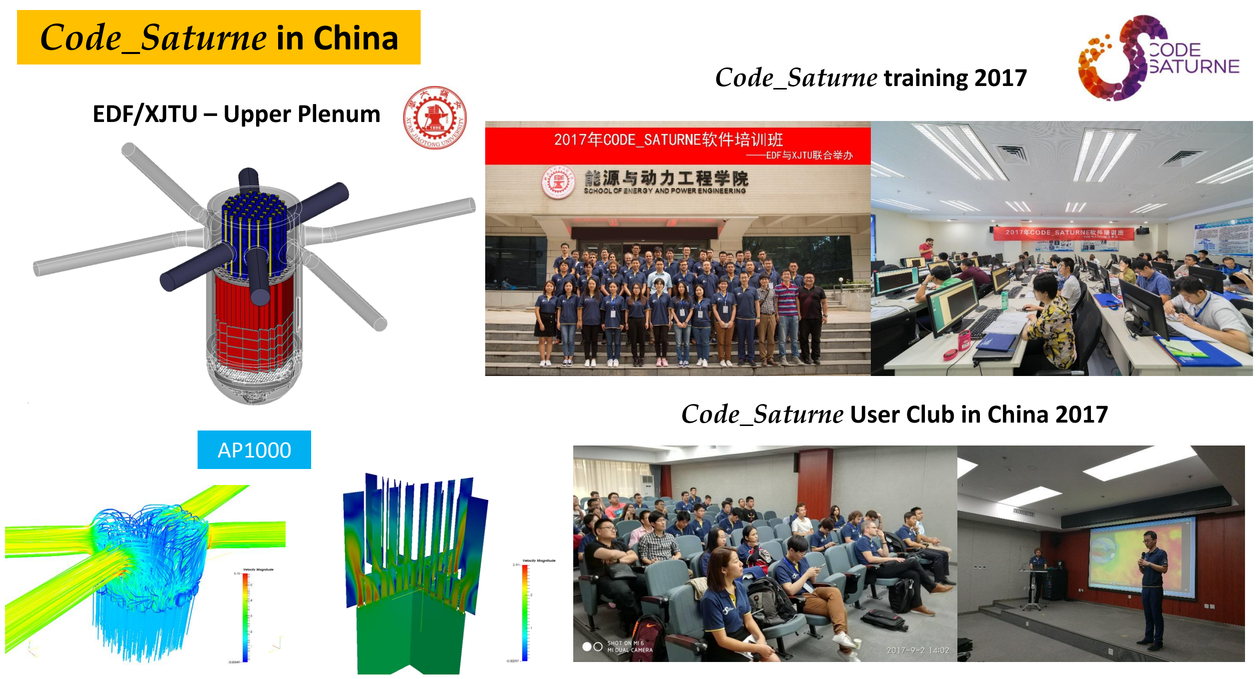
by T. Xu – EDF China R&D Center

Code_Saturne, as a generic CFD software, has been known and recognized by more and more Chinese users. Since 2011, EDF R&D China disseminates *Code_Saturne* in the research and industry domain through the cooperation with Chinese partners and the organization of open-session training. Last year, a four-day *Code_Saturne* training was organized in Xi'An with a mini user club. This training caused strong interest of Chinese partners. The participants gave good feedback and demanded on advanced session in the future.

Besides the training organization, R&D China continues technical research works on application domain with *Code_Saturne*.

In the basis of the previous study on the comparison of different typical flow diffuser, EDF gains knowledge and experience on flow patterns in lower plenum and the flow rate distribution at core inlet. In order to obtain the synthetic understanding on the thermal hydraulic behaviour in reactor vessel, R&D China extends the research domain to upper plenum. The thermal-hydraulic behaviour in upper plenum of a Pressurized Water Reactor (PWR) concerning the flow distribution at core outlet under nominal operation conditions is an important point in reactor functional margin increase and Rod Cluster Control Assembly (RCCA) guide tube wear decrease. This difficulty is due to the presence and absorption effect of hot legs at the reactor upper plenum. Due to the changes of flow direction and the complex geometries in the upper plenum internal region of PWR, the flow patterns are very complicated in three dimensions everywhere. Therefore, it is of importance to study the thermal hydraulic behaviour in upper plenum.

In another under-going project with Chinese partner, *Code_Saturne* is used to simulate the steam dispersion and condensation experiments. The primary results proved the good performance of the condensation module in *Code_Saturne*. In the meantime, a new function (steam condensation in turbulence natural convection) is trying to be added in the condensation module.



Overtopping flows simulations with the volume of fluid module of *Code_Saturne*

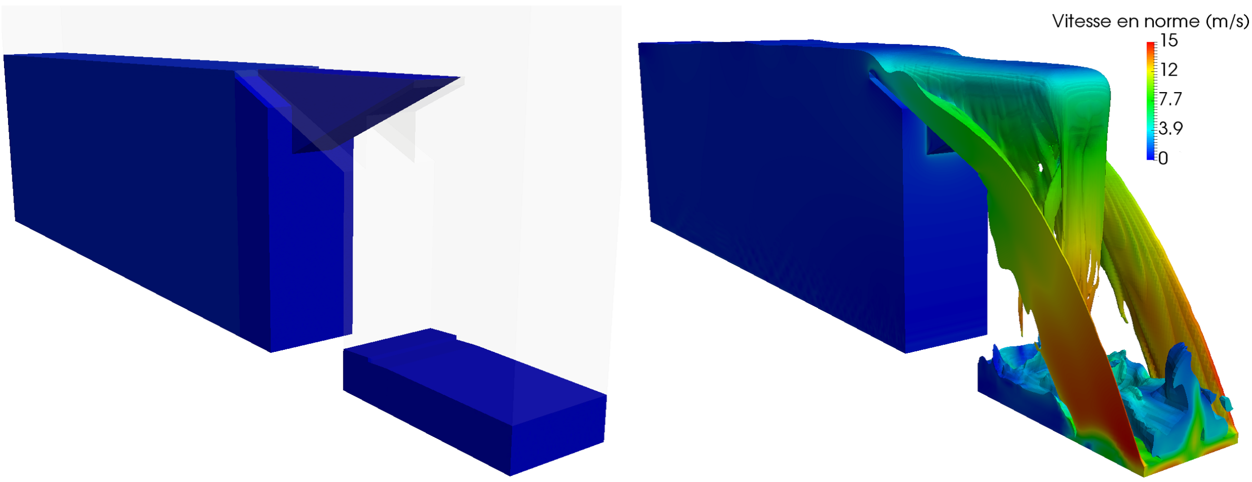
by Y. Bercovitz, N. Aymerich, E. Le Coupanec – EDF R&D LNHE, MFEE - ESPCI

EDF operates 250 dams higher than 20 meters in France. For safety reasons, especially during floods, a very good understanding of physical phenomena potentially appearing along the spillway is required. Detailed information about the flow provided by CFD simulations can help engineers size projects.

A Volume of Fluid module is available in *Code_Saturne* since version 5.0, released last year. In order to validate the module for industrial applications such as those encountered on EDF hydroelectric facilities, we have simulated overtopping flows for four different kinds of weirs: thin crest, Creager 2D, Creager 3D and Piano Key Weir (PKW). For each kind of weir, we compare head to flow rate laws obtained numerically with experimental data [1], [2], [3]. For the thin crest, we compare the trajectory of the jet obtained with *Code_Saturne* to Simemi and De Marchi empirical expressions [4]. For the Creager spillway we compare the pressure distribution on the weir to USACE [2] data and 3D simulations permit to observe the fluid vein detachment.

Except for the PK Weir, where we observe gap bigger than 10%, all other comparison give very coherent results with experimental data. To finish, the validation, VOF module should be used on a real site.

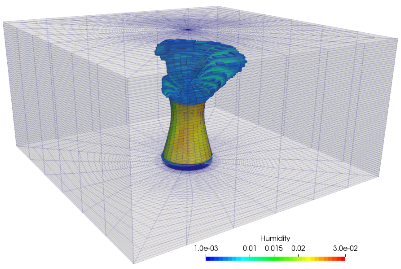
[1] Carlier M. Hydraulique gÃĺnérale et appliquée. Editions Eyrolles, 1986.
[2] CIH, rapport technique. Modélisation en 2D d'un seuil de type Creager : validation de la débitance, de la forme de la surface libre et de la pression Ãă l'interface solide/fluide, Juillet 2011.
[3] LempériÃĺre F. and Ouamane A. The piano keys weir: a new cost-effective solution for spillways. Hydropower & Dams, Issue Five, 2003.
[4] Sentürk. Hydraulics of Dams and Reservoirs. Water Resources Publications, 1994.



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

by N. TONELLO – RENUDA

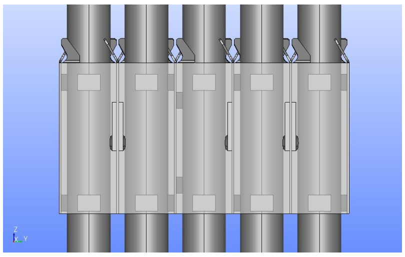
Renuda will present some of its recent applied, research and development work carried out with *Code_Saturne* in collaboration with EDF R&D, with a particular focus on energy production applications. In plants, controlling the temperature inside the facilities is of prime importance to ensure safe working conditions and to protect components. *Code_Saturne* has been applied to model and simulate the ventilation of entire nuclear reactor buildings in order to verify the air flow in the buildings for different configurations.



Test cooling tower

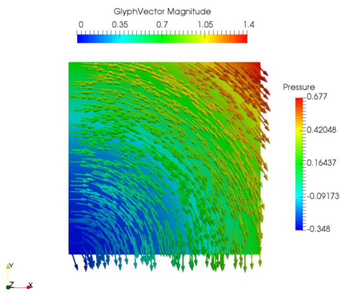
Industrial examples will be shown where possible.

Further evolutions of the cooling tower module have also been implemented, notably to add a functionality to model the rain zones underneath the packing. Details of the model will be presented as well as results from the on-going validation against experimental bench and full size data.



Industrial case: Fuel rods

As industrial geometries can often be complex, it may be very difficult to mesh them using exclusively hexahedral elements and necessary to use hybrid or fully tetrahedral meshes. In order to assess the quality of the tetrahedral meshes which may be obtained on industrial geometries and, in parallel, the impact of tetrahedral meshes on results for turbulent flow, fundamental work is presently on-going using both FVCA8 [1] benchmark cases for which solutions are known and actual industrial cases. Some of the test and industrial cases will also be presented where possible.



Verification case: FVCA8 vortex

[1] https://github.com/FranckBoyer/FVCA8_Benchmark.

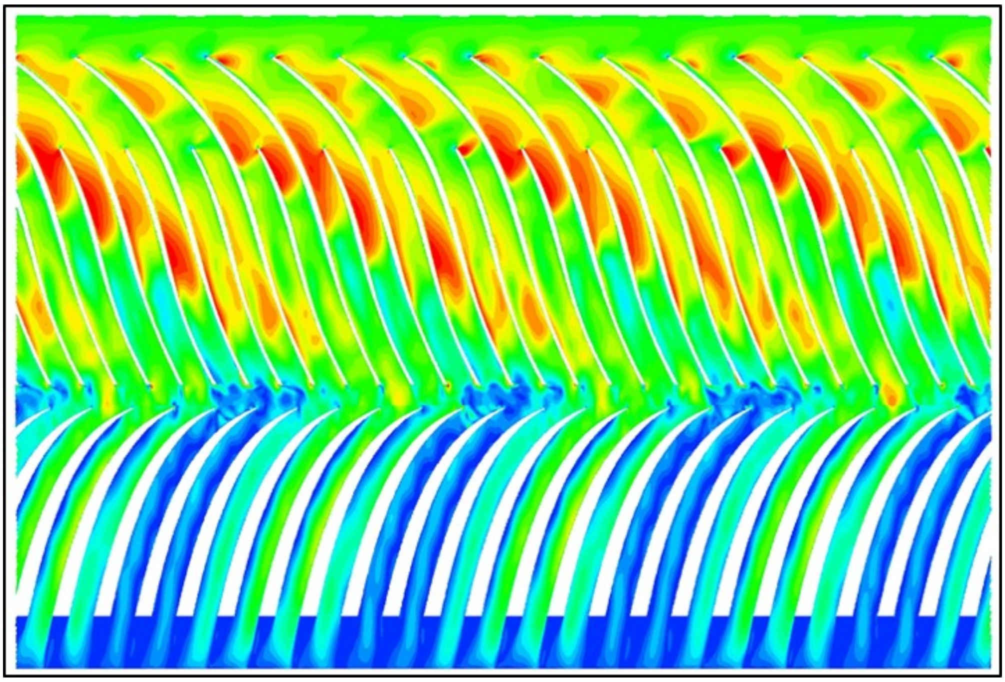
Development of an accurate and efficient compressible algorithm in *Code_Saturne* for the modelling of high-speed compressors

by A. HEFFRON – QUEEN MARY, UNIVERSITY OF LONDON

The operating range and efficiency of compressors are limited by the development of rotating stall and surge at low mass flow rates. Rotating stall is a macroscopic problem that affects the whole compressor, requiring a numerical solver that is efficient and accurate to resolve such a large-scale problem while capturing small flow features that initiate the rotating stall. Traditionally, RANS has been used to study turbomachinery, but continuing advancement of computational power and numerical methods has allowed large eddy simulation to become a possibility, allowing for a more detailed study of such machines.

As previously presented, a compressible SIMPLE algorithm was written in the framework of *Code_Saturne* that was compatible with the turbomachinery module. The algorithm incorporated a 2nd-order MUSCL scheme for convective terms and AUSM+-up for mass flux computation. This algorithm has been further refined and tested with a 2nd-order time scheme and with LES. A 3^{rd}-order MUSCL scheme has also been introduced to further improve accuracy. These new improvements have been tested on several different test cases as presented.

The SIMPLE algorithm has been tested on Rotor 37 and 67. The NASA CC3 high-speed centrifugal compressor has also been successfully modelled for a single, vaneless passageway and for the whole compressor with a vaned diffuser. The centrifugal compressor has been tested from choke to stall with rotating stall replicated at low mass flow rates. The numerical results of these simulations are presented as the first steps towards using *Code_Saturne* to model a high-speed compressor with LES.



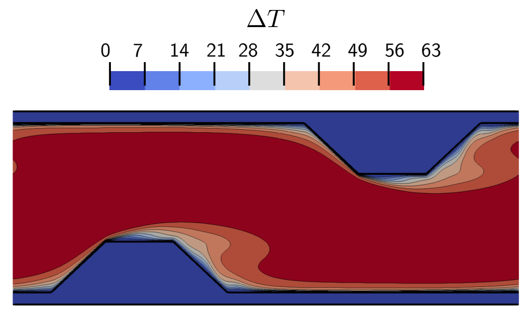
Heat exchanger multilevel modelling and optimization using *Code_Saturne*

by F. MASTRIPPOLITO, S. AUBERT, F. DUCROS & J.-F. FOURMIGUÉ – CEA TECH LITEN, LMFA, ECL LYON, VALEO & EDF

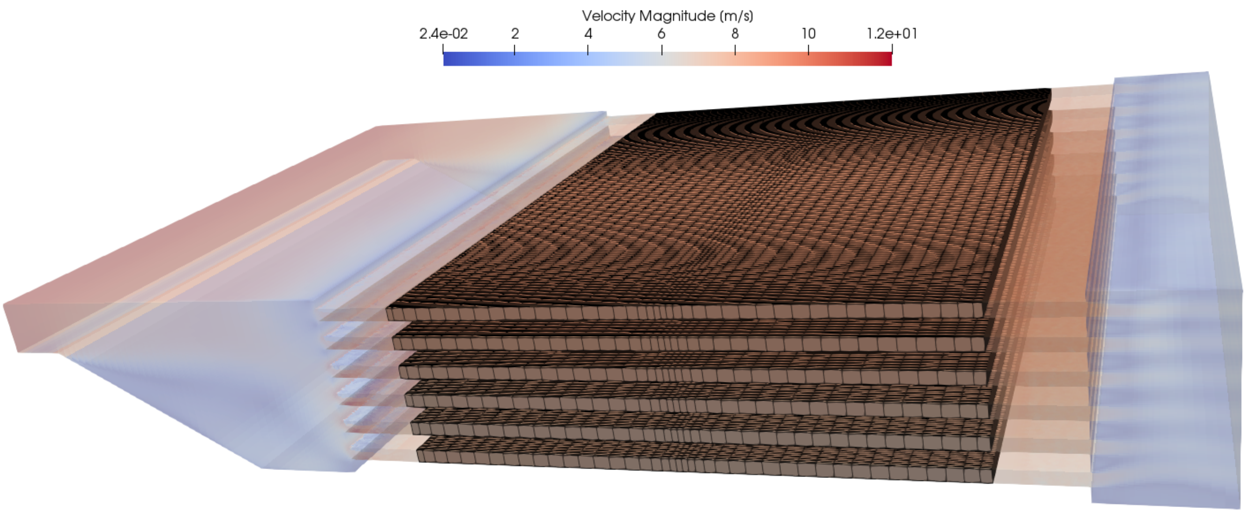
Heat exchangers (HEX) are used in many industrial applications, as power engineering, chemical industry, transport and spatial to transfer thermal energy. Heat transfer enhancement and HEX design are still investigated in order to improve energy efficiency. For this purpose, computational fluid dynamics (CFD) simulations and efficient optimization tools are more and more used. However, heat exchanger physics is a multi-scale issue where small scale enhancement mechanisms coexist with macro-scale distribution ones. Consequently, it is difficult to perform an accurate numerical simulation of the complete system with both scales. This work proposes a multilevel HEX modelling which couple accurate small scale CFD simulations to ones of the entire HEX through kriging based metamodels. Thus, micro-scale enhancement and macro-scale distribution phenomenons are treated simultaneously. This approach is then used to perform a four parameters HEX shape optimization.

A common way to enhance heat transfer in plate HEX is to use obstacles among which periodic ribs are one of the most popular solution (Fig. *Periodic rib modelling*). The rib shape has a strong influence on heat transfer and head losses. Diffusers are frequently used as manifolds. The height (h_{rib}), the base width (E), the top width (e) of the rib and the diffuser angle α are chosen as parameters for the optimization.

Code_Saturne simulations of the periodic rib are used to build two metamodels for the Nusselt number Nu and the head loss coefficient C_f based on the three rib shape parameters and the Reynolds number. Then, the active part of the HEX is modelled using only one cell on the channels height, connected to the diffuser (Fig. *Heat exchanger modelling*), and the metamodels are used to prescribe the heat transfer and pressure drop. A genetic algorithm is used to maximize the HEX effectiveness and to minimize the pressure drop by varying the four parameters h_{rib}, e, E and α.



Periodic rib modelling – Temperature variation



Heat exchanger modelling – Velocity

List of posters & demonstrations

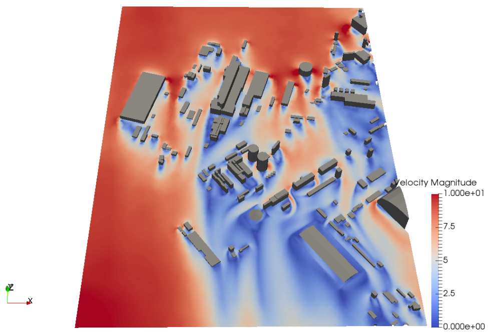
Vote for the best one on `www.code-saturne.org`!

Melissa: Large Scale In Transit Sensitivity Analysis of Model Outputs Avoiding Intermediate Files	by A. Ribes EDF R&D - PERICLES
A time implicit colocated finite volume method for incompressible fluid flows in obstructed media	by C. Colas, M. Ferrand, J.-M. Hérard & E. Le Coupanec EDF R&D - MFEE & I2M
Hybrid RANS/LES modelling of unsteady turbulent loads in hydraulic pumps	by V. Duffal EDF R&D - MFEE
Second-moment closure model for the dissipation rate of temperature variance	by G. Mangeon, R. Manceau, S. Benhamadouche & J.-F. Wald EDF R&D - MFEE, Uni. Pau
Performance at Scale of CDO numerical schemes to solve the Richards Equation for Groundwater Flows	by C. Moulinec, J. Bonelle, Y. Fournier & D. Emerson STFC, EDF R&D - MFEE
Modification of a Wake model for hydrodynamic forces on submarine cables with a rough seabed	by K. Kuznetsov, J. Harris, N. Germain & F. Aristodemo LHSV, Ecole des Ponts ParisTech, CEREMA, France Énergies Marines, Università della Calabria
Ventilation System to reduce pollutant concentration in a Waste Warehouse	by V. Michaud and L. Makke EGIS, ARIA
Salome_CFD, a demonstration of live visualisation & coprocessing using ParaView	by M. Paolillo and E. Le Coupannec EDF R&D - PERICLES, MFEE
Salome_CFD, a demonstration of Uncertainty Quantification using OPENTURNS	by C. Koren – EDF R&D - MFEE
A correlation for the discontinuity of ε_θ at the fluid-solid interface in turbulent channel flows	by C. Flageul Jozef Stefan Institude
NEPTUNE_CFD: the Salome_CFD multiphase solver powered by *Code_Saturne* HPC capabilities	by M. Guingo, C. Koren, J. Lavieville, N. Mérigoux & S. Mimouni EDF R&D - MFEE
Access your simulation software from anywhere without installtion	by N. TRIPATHI EDF R&D
Taking advantage of CDO schemes' robustness: examples of two hydrogeological applications	by R. Lamouroux & V. Loizeau EDF R&D - LNHE
Coupling the neutronics code DYN3D-MG to *Code_Saturne* to model molten salt fast reactors	by G. M. Cartland-Glover, A. Skillen, D. Litskevich, S. Rolfo, D. Emerson, B. Merk & C. Moulinec STFC
Arc cathode coupling and boundary conditions on predicted plasma flow field in a DC plasma torch	by R. Zhukovskii, A. Vardelle, C. Chazelas & V. Rat Limoges Uni.
The Artificial Compressibility Method for Velocity-Pressure coupling	by R. Milani, J. Bonelle & A. Ern EDF R&D MFEE, CERMICS
Comparison of numerical schemes for two-phase flow simulations	by L. Quibel, O. Hurisse EDF R&D - MFEE, Strasbourg Uni.
Fire modelling with *Code_Saturne*	by F. Nmira, A. Amokrane, B. Sapa EDF R&D - MFEE
Evaluation and benchmark of *Code_Saturne* capabilities for external aerodynamics	by T. Donze SOGETI

Review of 2017/2018 *Code_Saturne* Atmospheric Module Applications at ARIA Tech.

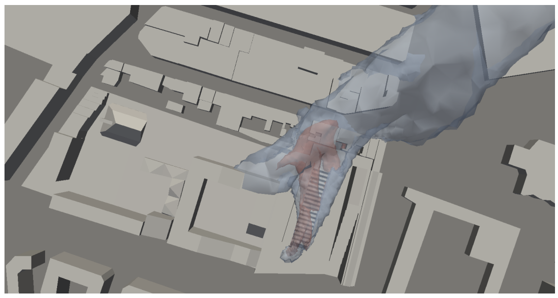
by L. Makke, M. Nibart, X. Wei & V. Michaud – ARIA Technologies, EGIS

ARIA Technologies is a French SME specialized in atmospheric environment modelling. In this field ARIA Technologies edits several computer programs to compute indoor and atmospheric flows and also uses *Code_Saturne* CFD model, and especially its atmospheric module, since 2009 to perform studies with various objectives. ARIA Technologies is also supporting some other companies or Institutes like EGIS and CNES to use *Code_Saturne*, as a standalone tool or call in the framework of specific software like the ARIACity. Besides, ARIA Technologies develops some features in atmospheric model in collaboration with *Code_Saturne* development team.



Highlights on different studies will be presented:

- A new radiative properties model suited for 3-D radiative atmospheric transfer model in *Code_Saturne* V5.1 (*Code_Saturne* development team and ARIA Technologies)
- Ventilation System to reduce pollutant concentration in a Waste Warehouse. (EGIS)
- Atmospheric study of mitigation potential of windbreak barriers / meshes around ILVA mineral deposits, Taranto (ERM)
- Atmospheric dispersion study of pollutants emitted by a bus maintenance workshop in an dense urban area



Feedback about CAD and mesh generation for Indoor and atmospheric modelling will be also discussed, mainly based on two solutions: Salome platform used for complex cases and a specific tool, starting from standard GIS data for topography and buildings, used for simple cases.

WRAPP - A comprehensive methodology to estimate the wind resource, production and wake effects

by G. Angot – EDF R&D MFEE - CEREA

A precise assessment of the wind flow over a site is a crucial information to have when tackling the question of the potential development of a wind farm. Indeed, the estimation of the annual energy production of the wind farm directly derives from the wind flow assessment. Besides, wind farm projects can have very large funding plans (up to hundreds of millions of dollars, or even billions of dollars), and the financing costs vary especially according to the precision of the annual energy production assessment.

EDF-R&D has developed an ambitious methodology to evaluate the wind resource, the energy production and the wake effect for both onshore and offshore wind farms. It is called Wind Resource Assessment and Power Production (WRAPP). This methodology is based on a chain from a regional scale meteorological model (WRF) to a local scale with the CFD code *Code_Saturne*.

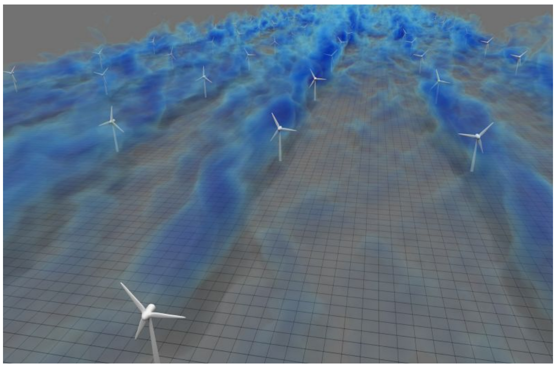
The main steps of the methodology are as follows:

1. Long-term mesoscale simulations;
2. Clustering of the output situations (from 1.) to identify a subset of situations representative of the long-term dataset;
3. Interpolation to chain the two codes;
4. CFD simulations;
5. Post-processing and analysis of the results.

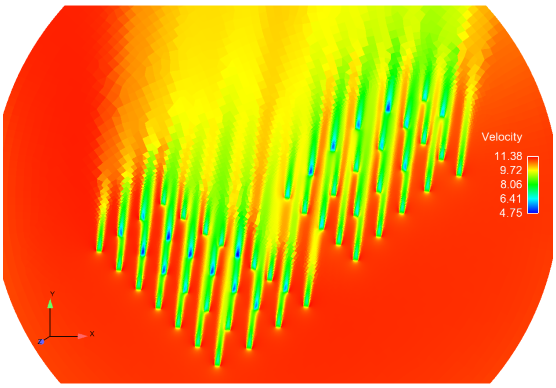
Compared to less sophisticated methods, this methodology is designed to lead to better results, especially on:

- Onshore complex terrain sites;
- Offshore (or onshore) wind farms with significant wake effects;
- Large sites or sites where meteorological fields present a significant horizontal variability.

In this oral presentation, the WRAPP methodology is described, different studies are shown and some promising results are delivered.



The wake effect affects the wind farm production

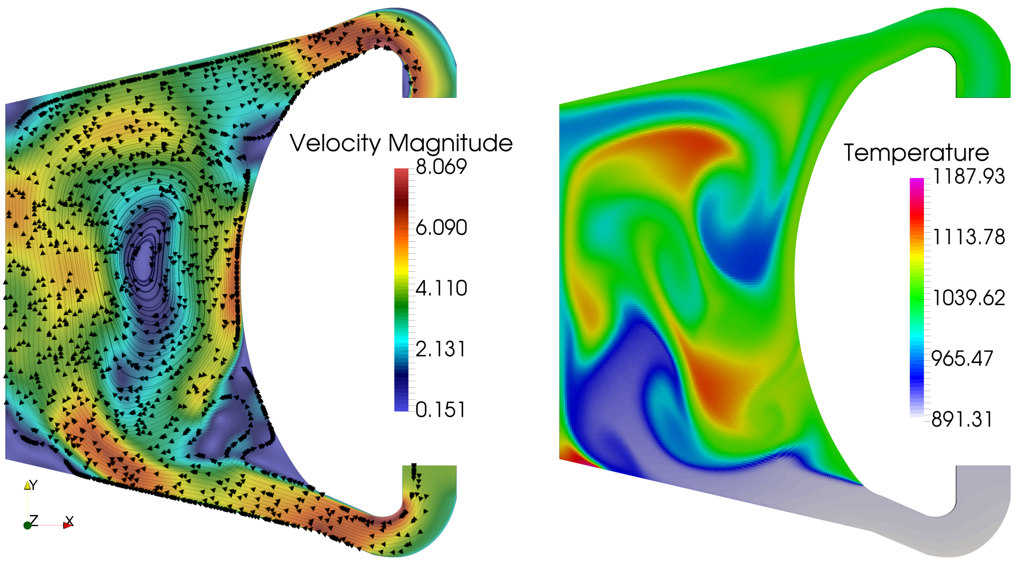


Wind field over an offshore wind farm

Modelling frozen salt walls in molten salt fast reactors

by S. ROLFO, G.M. CARTLAND-GLOVER, A. SKILLEN, D. EMERSON, C. MOULINEC, D. LITSKEVICH & B. MERK – STFC

Molten salt reactors are a type of nuclear reactor concept under development as a Generation-IV reactor. In these concepts nuclear fuel is dissolved in fluoride salts and it is pumped through a core region and heat is extracted via a number of heat exchangers. The reactors can be operated in the thermal, thermal-epithermal or fast modes with changes to the configuration of the core region, which can moderate the nuclear reactions. In fast reactor concepts no core internals are specified. This leads to a strong neutron flux in the vessel. The neutrons may react with material in the vessel wall and embrittle it. The salts used are highly corrosive and this may reduce the safe operational lifetime of the reactors. A feasibility study is being performed to investigate if we can freeze the salt at the vessel walls. The frozen salt may provide an additional layer of protection to the vessel wall and enable longer operational lifetimes. To do this *Code_Saturne* was coupled to the neutronics code DYN3D-MG in order to capture the interactions between the heat and mass transfer in the fluid salt with the core neutronics. A temperature dependent porosity was also implemented to model the freezing of the salt in colder regions in the reactor and the heat exchangers. The domain coupling method was also employed in order to model the heat flux through the wall and determine whether the heat can feasibly be removed from the wall.



Velocity and temperature field after $100s$

NEPTUNE_CFD user day

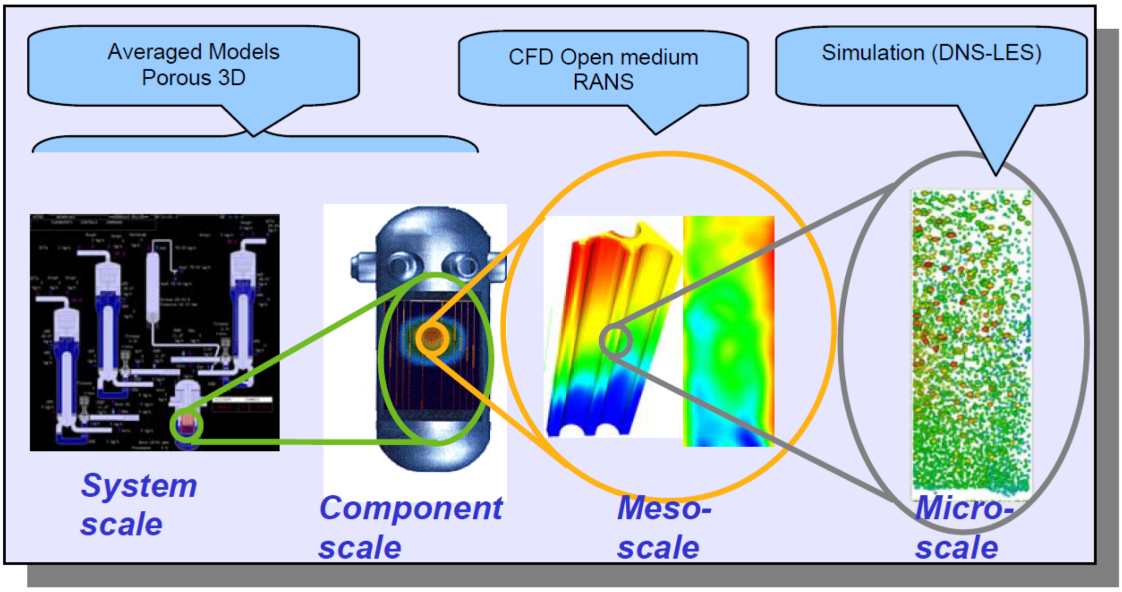
Program

8:30	**Welcome – Breakfast**	
9:00	Foreword	M. Guingo EDF R&D - MFEE
9:15	Nuclear Reactor Thermalhydraulics and the role of CFD	D. Bestion CEA
9:45	Some recent advances in two-phase flow modelling and applications	S. Mimouni EDF R&D - MFEE
10:15	Overview of multiphase industrial applications with NEPTUNE_CFD	N. Mérigoux EDF R&D - MFEE
10:45	**Coffee Break**	
11:15	Simulation of external reactor vessel cooling phenomenon for in-vessel retention using NEPTUNE_CFD - a collaboration between EDF and CGN	L. Zhou EDF R&D China
11:45	Simulating the emptying of a water bottle with a multi-scale two-fluid approach	S. Mer IMFT
12:15	CEA needs and contributions to two-phase flows: models and applications	G. Bois & M.-G. Rodio CEA
12:45	**Lunch break**	
14:00	NEPTUNE_CFD fluidized bed simulations from laboratory to industrial scales for the developement of new energy systems: chemical looping combustor and concentrating solar power plants	R. Ansart ENSIACET
14:30	Time and space dependent porosity method: two-phase flow application cases	W. Benguigui EDF R&D - MFEE
15:00	Towards a multiscale multiphase numerical coupling of NEPTUNE_CFD and CATHARE	C. Koren EDF R&D - MFEE
15:30	Closing remarks	J.-P. Chabard EDF R&D - Scientific director
15:45	**End of Salome_CFD days**	

Nuclear Reactor Thermalhydraulics and the role of CFD

by D. Bestion – CEA

Nuclear reactor thermalhydraulics uses various simulation tools for design and safety studies. System codes remain by far the most widely used codes for safety analyses and licensing since they are able to model the whole reactor circuits in single-phase and two-phase flow conditions encountered in a large variety of accidental transients. Porous-3D models are used in component codes or in 3D modules of system codes for core, for the whole pressure vessel, and for heat exchangers including steam generators. Single-phase CFD is being used mainly for turbulent mixing problems and fluid-structure interaction. Two-phase CFD -or CMFD- started to be developed in the early 2000 to answer specific needs, for two-phase flow in large 3D volumes like a containment with spray cooling, the external pressure vessel cooling, pool heat exchangers, or some passive systems, for flow in complex geometry with a possible strong influence of small geometrical details like the spacer grid effects on DNB, for revisiting old issues with a local analysis for a better understanding of the physics and more accurate modelling like the core reflooding with possible fuel rod ballooning, or when a very fine space and time resolution is necessary for the pressurized thermal shock, the thermal fatigue, or SG tube vibrations. New applications may arise with CFD being used as a support to the system code modelling and validation in a multi-scale approach. The various applications of two-phase CFD to reactor thermalhydraulics will be presented with the various model options and a short review of the state of the art. Some perspective for future applications will conclude this general overview.

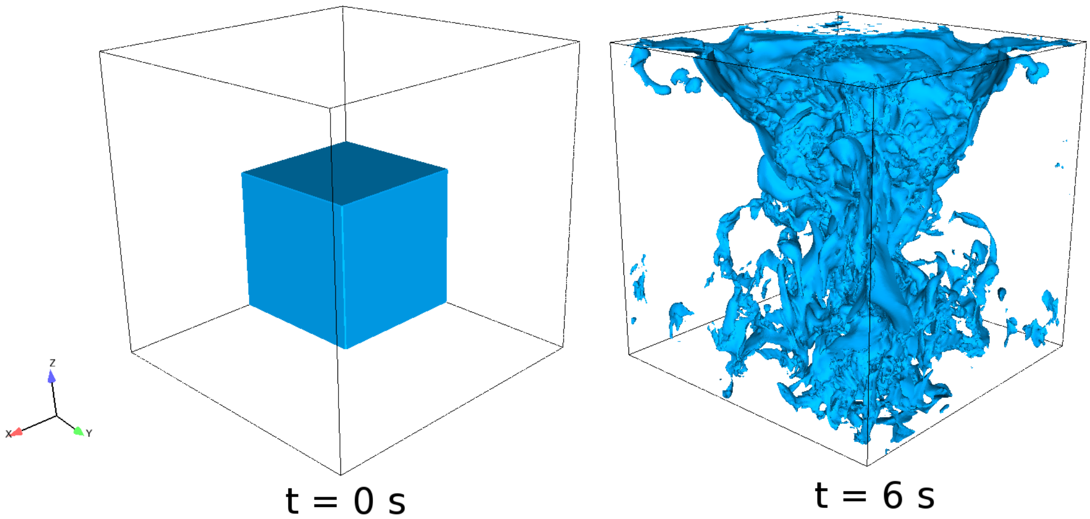


Schematic view of the different modelling scales of nuclear thermal-hydraulics

Some recent advances in two-phase flow modelling and applications

by S. MIMOUNI – EDF R&D

Boiling crisis and flows occurring in a steam generator or a heat exchanger remain a major limiting phenomenon for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems. Firstly, the choice is made to investigate a hybrid modelling of the flow. In so doing, the small and spherical bubbles are modelled through a dispersed approach within the two-fluid model, and the distorted or large bubbles are simulated with an interface locating method. The simulation of these interfaces between two continuous fields includes the definition of a surface tension model, a drag force expression to couple the velocity of the two fields at the interface and a numerical strategy to limit the numerical interface smearing. Mowever, the coupling between condensation, coalescence, break-up, relative velocity, forces exerted on bubbles, and turbulence makes the modelling task difficult. Conclusions drawn from agreement between experiments and computations could be biased because of discrepancies on turbulent variables. As a consequence, the important topic of turbulence modelling for two-phase flows including large interfaces is of relevant interest and is adressed. Finally, in order to tackle non isothermal flows occurring in industrial studies, a new heat transfer model is implemented and validated to deal with phase change at large interfaces. This heat transfer model is particularly adapted to the simulation of the vapor expansion observed experimentally in sodium boiling flows in the frame of Sodium-cooled Fast Reactors.

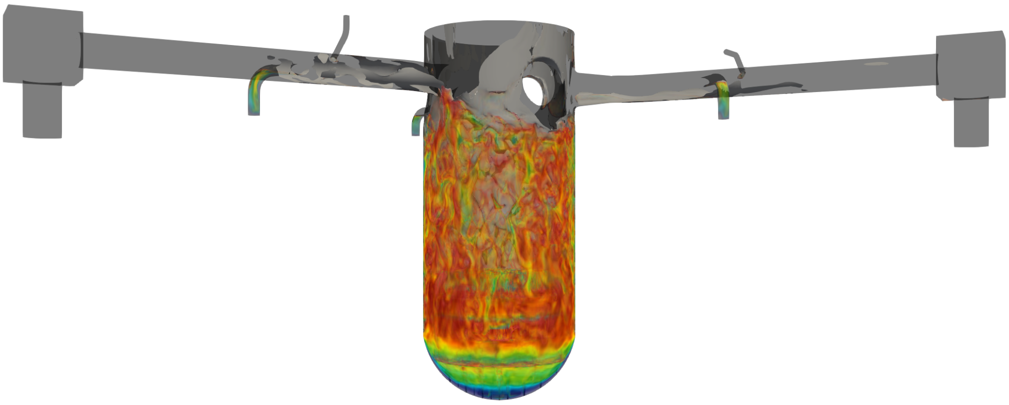


Simulation of the phase inversion benchmark

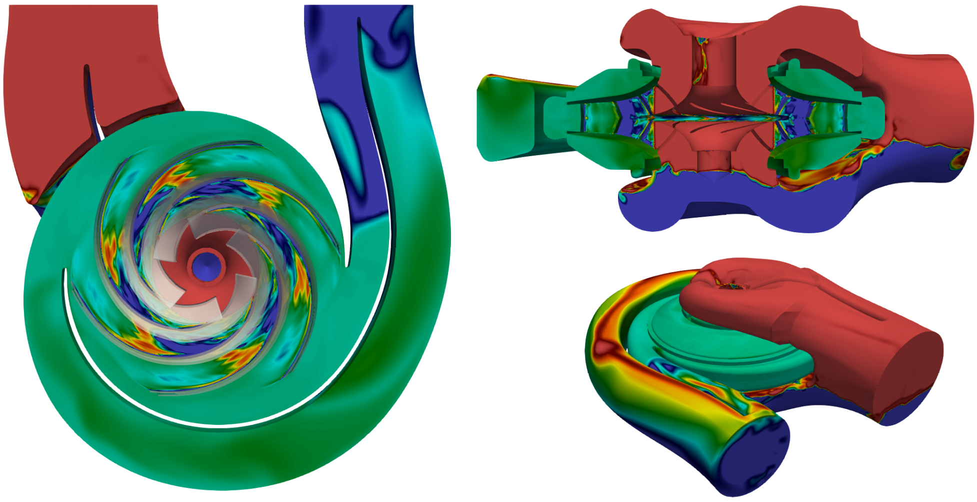
Overview of multiphase industrial applications with NEPTUNE_CFD

by N. Mérigoux – EDF R&D

NEPTUNE_CFD development started in 2001 with two main targeted applications: Departure from Nucleate Boiling (DNB) and Pressurized Thermal Shock (PTS). It led to the development of two main physical approaches which are the Dispersed Bubbly Flow and Free Surface Flow models. Nowadays, these models have become mature and new industrial applications are growing up, including the one with all-flow regimes transitions. This presentation will give a brief overview of these multiphase industrial cases modelled with NEPTUNE_CFD, including recent advances on Natural Circulation, Flow-Regime Transitions, Turbomachinery, Time and Space Variable Porosity, Free Surface Flows or Sodium modelling.



Simulation of a IB-LOCA scenario with NEPTUNE_CFD



Two-phase flow in a pump with NEPTUNE_CFD using the turbomachinery module

Simulation of external reactor vessel cooling phenomenon for in-vessel retention using NEPTUNE_CFD - a collaboration between EDF and CGN

by L. Zhou, J. Laviéville – EDF R&D China Center / MFEE

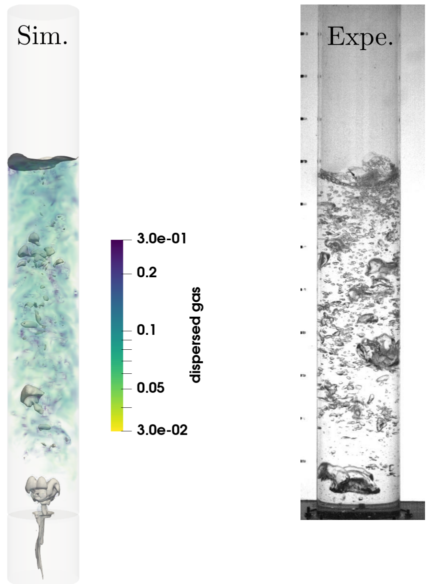
In order to do in-depth study of various engineering problems in the aspect of severe accidents, an EDF-CGN R&D benchmark project was launched and the numerical and experimental study of the external reactor vessel cooling (ERVC) loop is one of the topics of this project. The ERVC is a key severe accident management strategy (SAMS) used in AP1000, APR1400 and CAP1400. Under severe accident conditions, water is released to the cavity and the annular channel formed between the insulation structure and the outer surface of the reactor vessel (RPV). Thus, the lower head of the RPV is cooled by the water circulation driven by the boiling and siphon phenomena to retain and confine the corium in the RPV. A preliminary simulation of the ERVC loop with NEPTUNE_CFD was conducted and the case was defined by the 2D test facility of CGN based on CPR 1000+. A set of sensitivity analysis is now being conducted.

External view of the ERVC experimental loop

Simulating the emptying of a water bottle with a multi-scale two-fluid approach

by S. Mer, O. Praud, H. Neau, N. Mérigoux, J. Magnaudet & V. Roig – IMFT, EDF R&D

Many industrial processes involve gas-liquid flows characterized by a wide range of spatial and temporal scales. Simulating such flows remains a major challenge nowadays, as the computational cost associated with Direct Numerical Simulations still makes them unaffordable. An interesting alternative to DNS is the use of Euler-Euler solvers based on the two-fluid model. This approach has been widely employed and validated to simulate dispersed two-phase flows involving a carrier phase and a dispersed phase, such as small bubbles transported in a liquid. In this configuration, assuming non-deformable and mono-disperse bubbles, the momentum interfacial transfer terms are usually reduced to drag, lift and added-mass forces. When dealing with multi-scale flows, these assumptions are no longer valid and the interfacial momentum transfer terms need to be tailored to the local flow configuration. In the most recent approaches, large enough bubbles are fully resolved and may deform over time, while smaller bubbles are modelled as a dispersed phase. However, the closure models still involved in these approaches and the influence of the cut-off length separating the resolved and modelled bubbles definitely need to be validated against detailed experiments. In order to assess the validity of these models, we present a one-on-one comparison between experiments performed in a simple configuration, namely the emptying of a water bottle, and numerical simulations. This flow configuration is particularly relevant for checking such modelling approaches, as it exhibits a wide range of temporal and spatial scales. Large air bubbles with diameters of the order of the bottle neck are periodically generated and rise within the bottle until the free surface. While ascending, these large bubbles undergo break-up, resulting in a swarm of smaller bubbles, part of which may coalesce again and participate into the reconfiguration of the large bubble population. Different multi-field approaches along with models of interfacial momentum exchange will be presented and discussed, based on comparisons of simulations with experimental results. Furthermore, influence of compressibility effects on bubble formation and pressure oscillations at the top of the bottle will be highlighted.

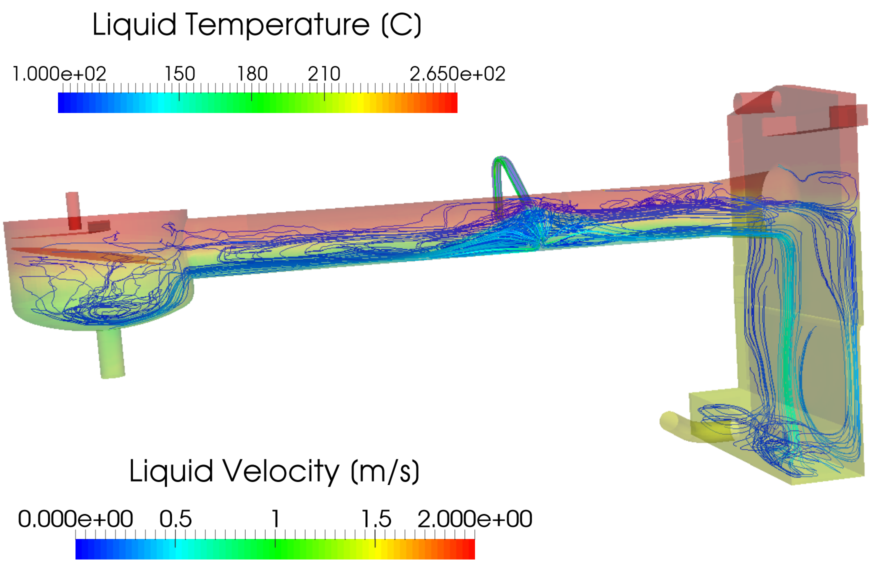


Experiments vs. Simulations of the emptying bottle

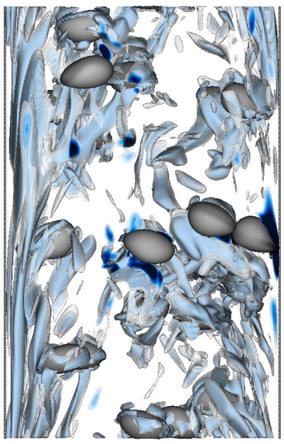
CEA needs and contributions to two-phase flows: models and applications

by G. Bois, M.-G. Rodio – CEA

The CEA research is focused on the simulation of nuclear reactor accidents in order to predict the global behavior of the system. Historically, the computations were based mainly on system and component codes that rely on strong modelling hypotheses and large scale averaging. In more recent years, computations are more and more evolving towards a more intensive use of simulation tools at smaller scales, typically CFD or DNS scales. In this context, the assessment of the prediction capabilities of CFD and DNS codes is crucial. For this reason, important modelling efforts have been concentrated towards the improvement of two-phase flows simulations. In this presentation, at first, the context of the multiscale approach will be rapidly introduced, then the most recent model developments at CEA will be presented and finally, a real application case will be shown.



The TOPFLOW-PTS validation test-case

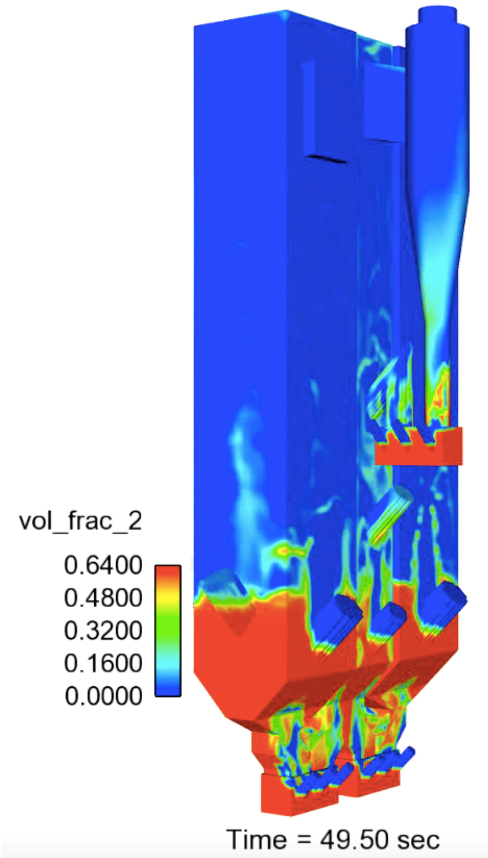


DNS simulation of a two-phase bubbly channel flow

NEPTUNE_CFD fluidized bed simulations from laboratory to industrial scales for the developement of new energy systems: chemical looping combustor and concentrating solar power plants

by R. Ansart, P. Fede, E. Masi, H. Neau, O. Simonin – ENSIACET, INPT

GasâĂŞsolid fluidized bed reactors have a wide application in industries for energy conversion processes. NEPTUNE_CFD code, using Euler-Euler approach for dense particulate flows, is a powerful numerical tool to design industrial energy power plants. This presentation will focus on two industrial processes developed in the framework of European projects: Chemical Looping Combustion (European Project SUCCESS 2013-2017) and Concentrated Solar Power plants (European Projects CSP2 2011-2015 and NEXT-CSP 2016-2020). The combustion of fossil fuels in nearly pure oxygen, rather than air, presents an opportunity to simplify carbon dioxide (CO2) capture in power plant applications. Chemical looping systems provide oxygen internal to the process via oxidation-reduction cycling of an oxygen carrier, eliminating the large capital, operating, and energy costs associated with oxygen separation. SUCCESS European project (Scale-Up of oxygen Carrier for Chemical-looping combustion using Environmentally SuStainable materials) aims at defining oxygen carrier and production techniques for its use at industrial scale 10 MWth fuel power. In the frame of the project, IMFT carried out simulations coupling hydrodynamics, reaction and heat transfer of chemical looping combustion system at pilot scale. The predictions were compared with experimental measurements. Then, numerical simulations at industrial scale were realized to design to design the commercial plant. In order to increase Concentrated Solar Power (CSP) plants efficiency, current research aims at increasing operating temperatures of the heat transfer fluid, and consequently increasing the efficiency of the power cycle. In addition, heat transfer fluids classically used for CSP have several limitations: restricted range of operating temperature, safety issues, corrosion and maintenance costs. Since solid particles are not subject to such limitations, provide a good thermal capacity, and a low-cost heat transfer and storage; an innovative alternative is to use an air-fluidized dense particle suspension as the heat transfer fluid, also called Upflow Bubbling Fluidized Bed. The Next-CSP European project, of which EDF is a partner, aims at improving the reliability and performance of CSP plants with this new technology, by demonstrating the concept feasibility with a 4MWth working pilot, and upscale it to the industrial scale by designing a 150MWel plant. IMFT and LGC carried out numerical simulations for a lab-scale set-up. The numerical results are compared with experimental measurements to enable numerical simulations at larger scale for the design and the optimization of the multi-megawatt particle solar receiver.

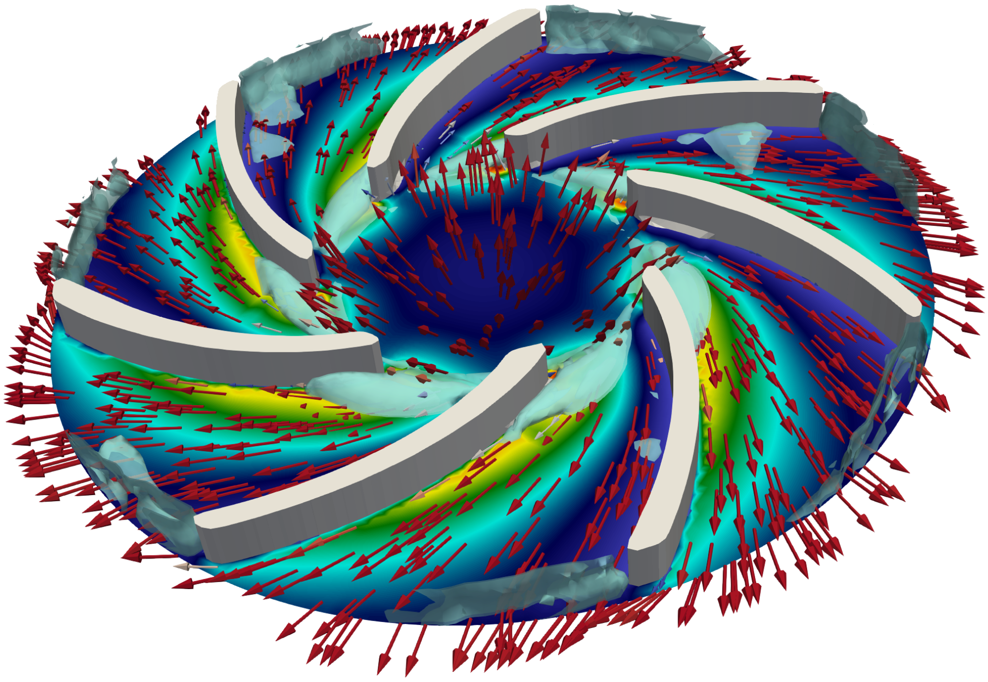


View of the particle volume fraction

Time and Space Dependent Porosity Method : two-phase flow Application Cases

by W. Benguigui – EDF R&D

The numerical simulation of interaction between structure and two-phase flow is a major concern for many industrial applications. To address this challenge, the motion of structures has to be tracked accurately. In the present work, a discrete forcing method based on a porous medium approach is proposed to follow non-deformable rigid body by using NEPTUNE_CFD. To deal with the action reaction principle at the solid wall interfaces in a conservative way, a porosity is introduced to locate the solid insuring no diffusion of the fluid-structure interface. The volumetric fraction equilibrium is adapted to this novelty. Then, mass and momentum balance equations are re-formulated on a fixed cartesian grid by taking into account immersed solid boundaries. This so-called Time and Space Dependent Porosity (TSDP) method allows to perform simulation of complex geometry on a very simple mesh with an imposed or a free solid motion in two-phase flow. For this presentation, the capacities of the method will be illustrated with two applications: two-phase flow in a centrifugal pump, and the free-fall of a spherical solid on a free surface.

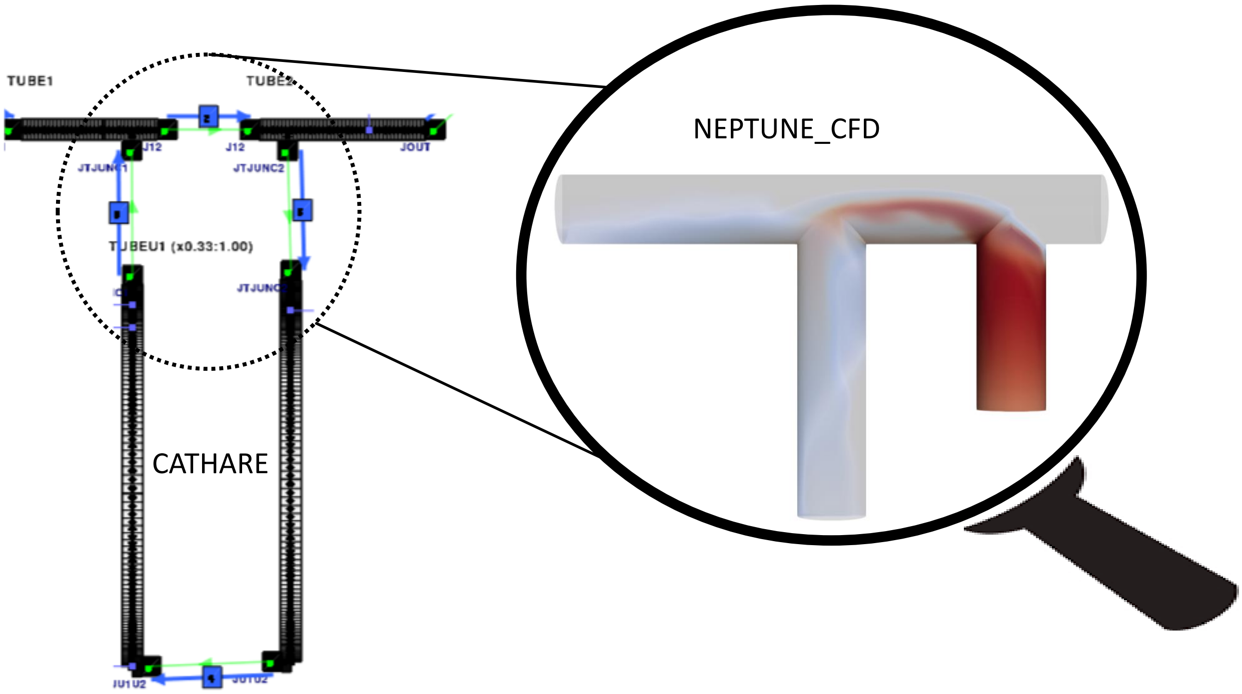


Two-phase flow in a pump with the porosity method

Towards a multiscale multiphase numerical coupling of NEPTUNE_CFD and CATHARE

by C. Koren, C. Geffray – EDF R&D, CEA

System Thermal Hydraulic (STH) codes are used for Pressurized Water Reactors (PWR) safety analysis since they allow a global evaluation within a reasonable computational time thanks to 1D-lumped formulations. Nevertheless, when strong multidimensional effects occur in a flow, this approach can be limited and affect the accuracy of the code predictions. Coupling a STH code with a Computational (Multiphase)Fluid Dynamics (CMFD) code is therefore a means of improving the physical modelling while keeping the increase of the computational cost to a reasonable level. Indeed, the CMFD code is only used in the parts of the domain where 3D effects are expected whereas the STH is used elsewhere. In the work reported here, the coupling between the codes CATHARE and NEPTUNE_CFD is presented in the context of single-phase flows. A verification test case is first provided to showcase the correct implementation of the coupling. Finally a validation test case based on a double T-junction experimental facility is studied, where strong mixing effects are encountered.



Schematic view of the multiscale coupling between CATHARE and NEPTUNE_CFD

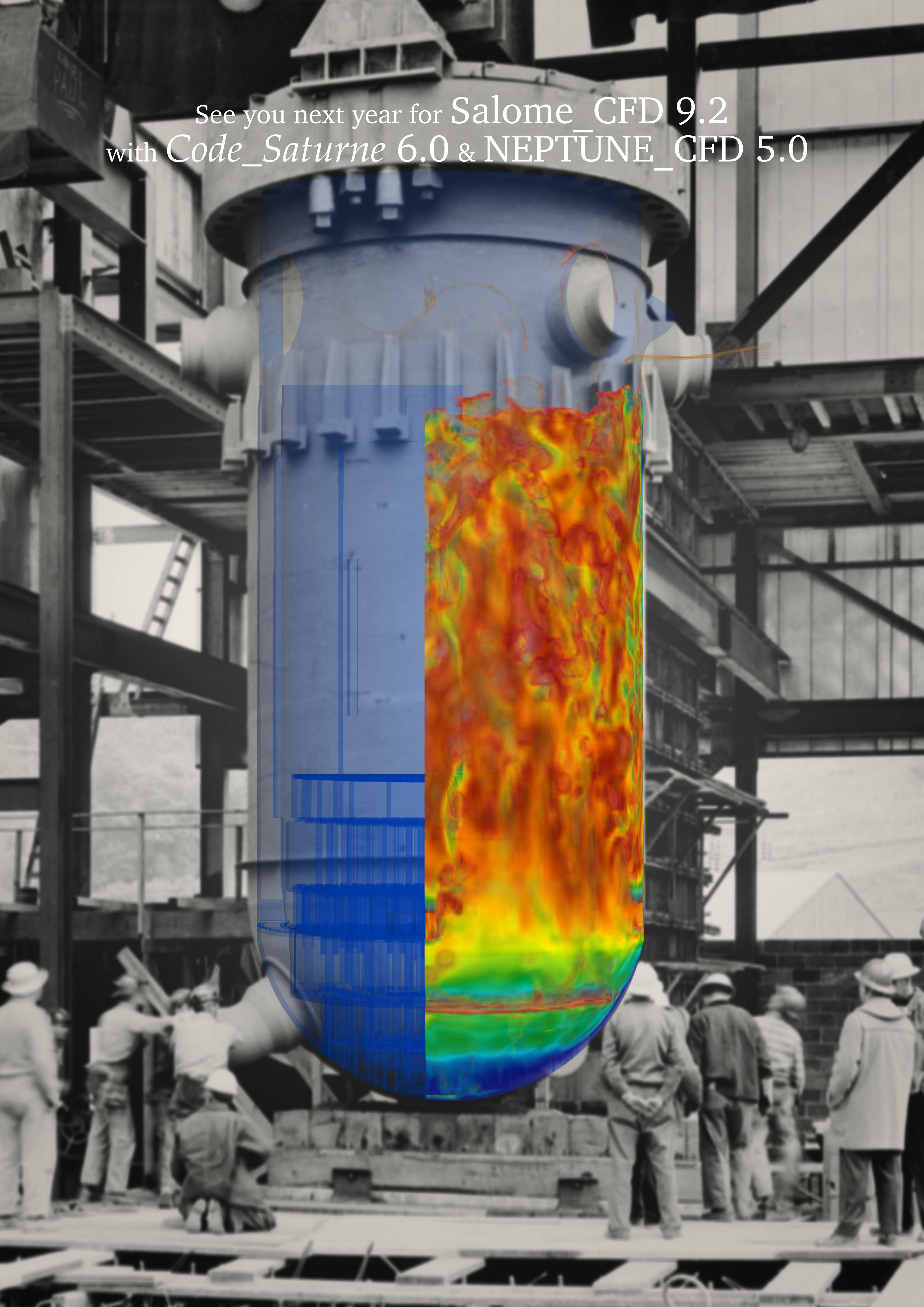
List of participants

ABSI	Rafik	EBI	France	r.absi@hubebi.com
AGBALESSI	Mocia	EDF R&D	France	mocia.agbalessi@edf.fr
AMOKRANE	Abdenour	EDF R&D	France	abdenour.amokrane@edf.fr
ANGOT	Guillaume	EDF R&D	France	guillaume-g.angot@edf.fr
ANSART	Renaud	LGC/INP	France	renaud.ansart@ensiacet.fr
ANTOLINOS	André	AKKA	France	andre.antolinos@akka.eu
ASMAR	Lea	Sorbonne university/ EDF	France	leaasmar@gmail.com
ASPROULIS	Panos	Renuda Ltd.	Germany	panos.asproulis@gmail.com
AUDOUIN	Yoann	EDF R&D	France	yoann.audouin@edf.fr
BAHLALI	Meissam	EDF R&D	France	meissam.bahlali@edf.fr
BANNER	Didier	EDF R&D	France	didier.banner@edf.fr
BAUDRY	Cyril	EDF R&D	France	Cyril.baudry@edf.fr
BELOUAH	Salma	EDF R&D	France	salma.belouah@edf.fr
BENGUIGUI	William	EDF R&D	France	william.benguigui@edf.fr
BENHAMADOUCHE	Sofiane	EDF R&D	France	sofiane.benhamadouche@edf.fr
BERCOVITZ	Yvan	EDF R&D	France	yvan.bercovitz@edf.fr
BESTION	Dominique	CEA	France	dominique.bestion@cea.fr
BIEDER	Ulrich	CEA	France	ulrich.bieder@cea.fr
BOIS	Guillaume	CEA Saclay	France	guillaume.bois@cea.fr
BONELLE	Jerome	EDF R&D	France	jerome.bonelle@edf.fr
BOUCKER	Marc	EDF R&D	France	marc.boucker@edf.fr
BURBEAU	Anne	CEA	France	anne.burbeau@cea.fr
BURKHART	Stephane	DGA	France	stephane.burkhart@intradef.gouv.fr
CAMY	Romain	EDF R&D	France	romain.camy@edf.fr
CARISSIMO	Bertrand	CEREA - Ecole des Ponts - EDF R&D	France	Bertrand.Carissimo@enps.fr
CARTLAND-GLOVER	Gregory	Science and Technology Facilities Council	United Kingdom	greg.glover@stfc.ac.uk
CARUYER	Céline	EDF R&D	France	celine.caruyer@edf.fr
CHARBONNEL	Sophie	ENSEEIHT / EDF R&D	France	sophie.charbonnel@edf.fr
CHARMEAU	Anne	CEA	France	anne.charmeau@cea.fr
CHAZELAS	Christophe	University of Limoges	France	chazelas@ensil.unilim.fr
CHEN	Zehao	CentraleSupelec	China	13220168499@163.com
CLERC	Thomas	Edvance (EDF Group)	France	thomas.clerc@edf.fr
COHN	Florian	NUMTECH	France	florian.cohn@numtech.fr
COLAS	Clément	EDF R&D	France	clement.colas@edf.fr
CROUZET	Fabien	EDF R&D	France	fabien.crouzet@edf.fr
CROUZET	Fabien	EDF R&D	France	fabien.crouzet@edf.fr
DABBENE	Frederic	CEA	France	frederic.dabbene@cea.fr
DBJAY	Sylvain	Akka technologies	France	sylvain.dbjay@akka.eu
DE AMICIS	Jacopo	EDF Energy	United Kingdom	jacopo.deamicis@edfenergy.com
DE LAAGE DE MEUX	Benoît	EDF R&D	France	benoit.de-laage-de-meux@edf.fr
DE SANTIS	Andrea	Nuclear Research and Consultancy Group (NRG)	Netherlands	a.desantis@nrg.eu
DE SOZA	Thomas	EDF R&D	France	thomas.de-soza@edf.fr
DELMAS	Josselin	EDF R&D	France	josselin.delmas@edf.fr
DEMAY	Charles	EDF R&D	France	charles.demay@edf.fr
DENEFLE	Romain	EDF R&D	France	romain.denefle@edf.fr
DORADOUX	Adrien	SIREHNA	France	adrien.doradoux@sirehna.com
DUFEIL	Philippe	CEA	France	philippe.dufeil@cea.fr
DUFFAL	Vladimir	EDF R&D	France	vladimir.duffal@edf.fr
DUMOND	Julien	New NP Gmbh	Germany	julien.dumond@areva.com
DUPONT	Eric	EDF-R&D	France	eric.dupont@edf.fr
DYAN	Anthony	EDF R&D	France	anthony.dyan@edf.fr
EMERSON	David	STFC Daresbury Laboratory	United Kingdom	david.emerson@stfc.ac.uk
EUDE	Yohann	Renuda	France	yohann.eude@renuda.com
FAYARD	Jean-Luc	CEA	France	jean-luc.fayard@cea.fr
FAYDIDE	Bernard	CEA	France	bernard.faydide@cea.fr
FERRAND	Martin	EDF R&D	France	martin.ferrand@edf.fr
FERRARI	Jerome	EDF R&D	France	jerome.ferrari@edf.fr
FERRARI	Jerome	EDF R&D	France	jerome.ferrari@edf.fr
FLAGEUL	Cedric	Institut Jozef Stefan	Slovenia	cedric.flageul@ijs.si
FLOUR	Isabelle	EDf R&D - MFEE	France	isabelle.flour@edf.fr
FONTAINE	Jacques	EDF R&D	France	jacques-j.fontaine@edf.fr
FOURNIER	Yvan	EDF R&D	France	yvan.fournier@edf.fr
FREYDIER	Philippe	EDF DT	France	philippe.freydier@edf.fr
FREYDIER	Philippe	EDF DT	France	philippe.freydier@edf.fr
FRUNGIERI	Graziano	Politecnico di Torino	Italy	graziano.frungieri@polito.it
GAUFFRE	Marie-Charlotte	EDF R&D	France	marie-charlotte.gauffre@edf.fr
GEFFRAY	Clotaire	CEA	France	clotaire.geffray@cea.fr
GUILBERT	Niels	EDF R&D	France	niels.guilbert@edf.fr

HARRIS	Jeffrey	Ecole des Ponts ParisTech	France	jeffrey.harris@enpc.fr
HEFFRON	Andrew	Queen Mary, University of London	United Kingdom	a.p.heffron@qmul.ac.uk
HILLHOUSE	Robert	Renuda Fluid Solutions	United Kingdom	robert.hillhouse@renuda.com
HÜLSEMANN	Frank	EDF R&D	France	frank.hulsemann@edf.fr
HURISSE	Olivier	EDF R&D	France	olivier.hurisse@edf.fr
IMEN	Hassen	National Engineering school of Tunis	Tunisia	imen.hassen@hotmail.fr
JAMELOT	Erell	France	France	erell.jamelot@cea.fr
JONCHIERE	Romain	EDF	France	romain.jonchiere@edf.fr
KOREN	Chaï	EDF R&D	France	chai.koren@edf.fr
KUZNETSOV	Konstantin	Laboratoire d'Hydraulique Saint-Venant Univ. Paris-Est (EDF R&D, Cerema, ENPC)	France	konstantin.kuznetsov@enpc.fr
LACAZEDIEU	Elisabeth	EDF R&D	France	elisabeth.lacazedieu@edf.fr
LACOURT	Frederic	Dow Chemical company	France	flacourt@dow.com
LAERA	Sergio	EDF R&D	Paris	sergio.laera@edf.fr
LAMOUROUX	Raphaël	EDF R&D	France	raphael.lamouroux@edf.fr
LANCIAL	Nicolas	EDF R&D	France	nicolas.lancial@edf.fr
LAUPSIEN	David	INSA de Toulouse / LISBP	France	laupsien@insa-toulouse.fr
LAVAL	Damien	Naval Group	France	damien.laval@naval-group.com
LAVIÉVILLE	Jerôme	EDF R&D	France	jerome-marcel.lavieville@edf.fr
LE COUPANEC	Erwan	EDF R&D	France	erwan.lecoupanec@edf.fr
LEFEBVRE	Vincent	EDF R&D	France	vincent.lefebvre@edf.fr
LELONG	Franck	EDF DT	France	franck-f.lelong@edf.fr
LINÉ	Alain	INSA-LISBP	France	alain.line@insa-toulouse.fr
LOIZEAU	Vincent	EDF R&D	France	vincent.loizeau@edf.fr
LOMBARD	Virginie	EDF R&D	France	virginie.lombard@edf.fr
LORENTZ	Eric	EDF R&D	France	eric.lorentz@edf.fr
MAGNE	Johan	EDF	France	johan.magne@edf.fr
MAKKE	Laurent	ARIA Technologies	France	lmakke@aria.fr
MALARTIC	Quentin	École Normale Supérieure Paris-Saclay	France	qmalarti@ens-paris-saclay.fr
MANGEON	Gaëtan	EDF R&D / Université de Pau et Pays de l'Adour	France	gaetan.mangeon@edf.fr
MARC	Raphaël	EDF R&D / PERICLES	France	raphael.marc@edf.fr
MASTRIPPOLITO	Franck	CEA Grenoble	France	franck.mastrippolito@cea.fr
MER	Samuel	IMFT	France	samuel.mer@imft.fr
MERIGOUX	Nicolas	EDF R&D	France	nicolas.merigoux@edf.fr
MIKCHEVITCH	Alexei	EDF R&D	France	alexei.mikchevitch@edf.fr
MILANI	Riccardo	Ecole des Ponts et Chaussées	France	riccardo.milani@edf.fr
MIMOUNI	Stéphane	EDF R&D	France	stephane.mimouni@edf.fr
MINIER	Jean-Pierre	EDF R&D	France	jean-pierre.minier@edf.fr
MOULINEC	Charles	STFC Daresbury Laboratory	United Kingdom	charles.moulinec@stfc.ac.uk
NAURY	Sylvie	CEA	France	sylvie.naury@cea.fr
NEAU	Herve	IMFT / CNRS	France	neau@imft.fr
NISTOR	Ionel	EDF R&D	France	ionel.nistor@edf.fr
OUBANBAL	Jamal	ENSEEIHT - EDF R&D	France	jamal.oubanbal@edf.fr
OURAOU	Mehdi	FRAMATOME	France	mehdi.ouraou@areva.com
PAJAUD	Valentin	EDF R&D	France	valentin.pajaud@edf.fr
PAOLILLO	Martine	EDF R&D	France	martine.paolillo@edf.fr
PATIL	Aakash	Ecole Centrale de Lille	France	aakash.patil@master.centralelille.fr
PEYRARD	Chrisotphe	EDF R&D	France	christophe.peyrard@edf.fr
PORCHERON	Lynda	EDF R&D	France	lynda.porcheron@edf.fr
PRODANOVIC	Pat	ENPC	France	pprodano@gmail.com
RAYNAUD	Christelle	EDF R&D	France	christelle.raynaud@edf.fr
RIBES	Alejandro	EDF R&D	France	alejandro.ribes@edf.fr
RIGALL	Tommy	IMSIA/Ensta ParisTech	France	tommy.rigall@ensta.fr
RODIO	Maria Giovanna	CEA	France	mariagiovanna.rodio@cea.fr
ROLFO	Stefano	STFC Daresbury Laboratory	United Kingdom	stefano.rolfo@stfc.ac.uk
ROUAULT	Laurent	EDF R&D UK Centre	United Kingdom	laurent.rouault@manchester.ac.uk
SALVATORE	Patricia	CEA SACLAY	France	patricia.salvatore@cea.fr
SAPA	Bertrand	EDF R&D	France	bertrand.sapa@edf.fr
SEGRÉ	Jacques	CEA	France	jacques.segre@cea.fr
SENECHAL	Dorothee	EDF R&D	France	dorothee.senechal@edf.fr
SHAMS	Afaque	Nuclear Research and Consultancy Group (NRG)	Netherlands	shams@nrg.eu
STANCIU	Mugurel	EDF R&D	France	mugurel.stanciu@edf.fr
TANIGASSALAME	Subashiny	EDF R&D PERICLES	France	subashiny.tanigassalame@edf.fr
TELMEN	Baris	Universite Paris-Sud	France	baristelmen@gmail.com

TONELLO	Nicolas	Renuda	United Kingdom	nicolas.tonello@renuda.com
TOTI	Antonio	EDF Energy R&D UK Centre	Uk	toti.antonio@outlook.com
TRIPATHI	Nikhil	EDF	France	nikhil-externe.tripathi@edf.fr
TRUONG	Ngoc Minh	AKKA	France	ngocminh.truong@akka.eu
URIBE	Juan	EDF energy	United Kingdom	juan.uribe@edfenergy.com
VARDELLE	Armelle	University of Limoges	France	armelle@ensil.unilim.fr
VIVALDI	Daniele	IRSN	France	daniele.vivaldi@irsn.fr
WALD	Jean-Franọis	EDF R&D	France	jean-francois-j.wald@edf.fr
WENDUM	Denis	EDF R&D	France	denis.wendum@edf.fr
XU	Tingting	EDF R&D China	China	tingting.xu@edf.fr
ZANCHETTI	Alexandre	EDF DT	France	alexandre.zanchetti@edf.fr
ZHOU	Lu	EDF R&D China	China	lu.zhou@edf.fr
ZHUKOVSKII	Rodion	University of Limoges	France	rodion.zhukovskii@etu.unilim.fr

Here for the two days


See you next year for Salome_CFD 9.2
with Code_Saturne 6.0 & NEPTUNE_CFD 5.0