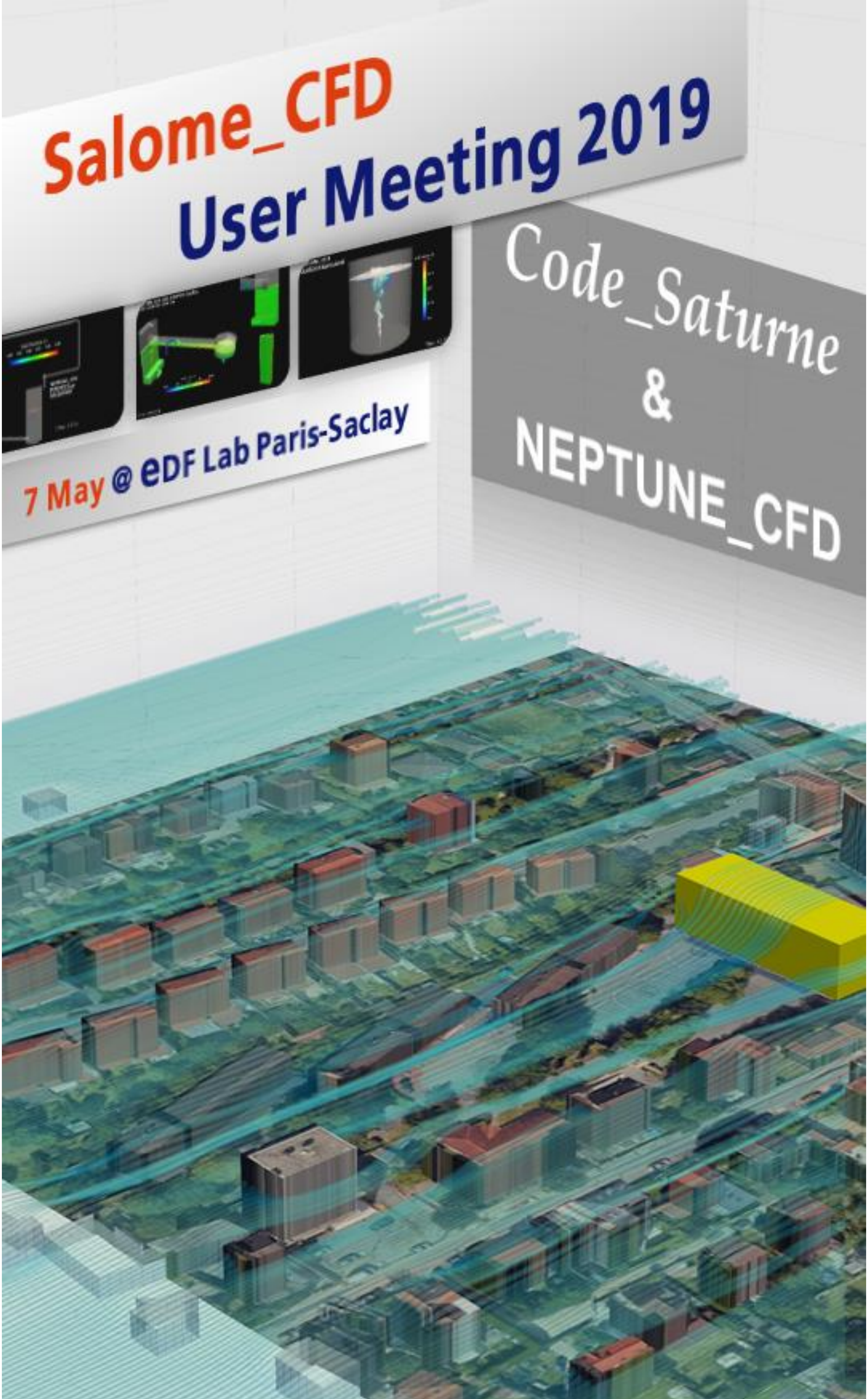

Salome_CFD
User Meeting 2019
7 May @ eDF Lab Paris-Saclay
Code_Saturne
&
NEPTUNE_CFD

Program of Salome_CFD User Day 2019
7 May – EDF Lab Paris-Saclay - Auditorium

From 8h: Welcoming breakfast

9h: Start of the User Meeting

Introduction - Didier Banner (EDF R&D – Head of simulation program)

Selected Salome_CFD evolutions	Development Team

9h50: *Code_Saturne* in the sky and underground

Code_Saturne: quality is in the air	Martin Ferrand (EDF R&D/CEREA), Cédric Flageul (ENPC/CEREA)
Code_Saturne for groundwater flow applications: Case of long-term safety studies	Jérôme Bonelle (EDF R&D/MFEE), Raphaël Lamouroux (EDF R&D/LNHE)

10h30: Coffee Break
11h: EDF engineering applications & foreign centers

Code_Saturne and NEPTUNE_CFD activities in China	Tingting Xu (EDF R&D – China Center)
Recent applications of Code_Saturne at EDF R&D UK Centre	Juan Uribe (EDF R&D – UK Center)
Measurement of primary coolant flow rate from a pressure difference	Romain Camy (EDF – DIPNN/DT)
Design of submerged thermal diffusers by numerical modelling	Olivier Bertrand (Artelia)

12h20: Lunch
13h30: Opening a new phase

Avoiding segregation with Code_Saturne	Charles Demay (EDF R&D/PRISME)
Bayesian calibration of a two-group IATE model for air-water flows	Nicolas Leoni (CEA)
Towards a numerical reactor for LOCA with NEPTUNE_CFD	Nicolas Mérigoux (EDF R&D/MFEE)
Moving solids every way in multiphase flows with NEPTUNE_CFD	William Benguigui, Jérôme Laviéville (EDF R&D/MFEE)

14h50: Coffee break, with posters and demo session
15h50: Opening the sources of *Code_Saturne*

Development of Code_Saturne for Rarefied Gas Flow	Dave Emerson (STFC)
Applications of the Arbitrary Lagrangian-Eulerian approach for modeling water waves	Jeffrey Harris (ENPC)
Cleaning up with Code_Saturne and SALOME	Nicolas Tonello (RENUDA)

16h50: Conclusion - Isabelle Flour (EDF R&D – Head of the MFEE department)

17h: Closing reception

Posters

L. Quibel, P. Helluy, O. Hurisse	Out-of-equilibrium two-phase flow model- Coupling with complex equations of state.
R. Milani, J. Bonelle, A. Ern	The incompressible Navier-Stokes equations with Compatible Discrete Operator schemes
G. Mangeon, S. Benhamadouche, J.-F. Wald, R. Manceau	Conjugate heat transfer: a big challenge for RANS modeling
C. Colas, M. Ferrand, J.-M. Hérard, E. Le Coupanec	Integral formulation for fluid flow in congested media modelling
L. Makke, X. Wei, M. Nibart, V. Michaud	Review of atmospheric case studies with *Code_Saturne*
K.I. Kuznetsov, J.C. Harris, N. Germain	Prediction of hydrodynamic loads on a cable near a rough seabed
F. Oukaili, Y. Bercovitz, C. Goeury, F. Zaoui, E. Le Coupanec	Optimum design of spillways using a metaheuristic algorithm
P. Landesman, C. Peyrard, J.C. Harris, M. Benoît	Absorption of regular water waves by a sponge layer in the Volume Of Fluid module of Code_Saturne
M. Tekavcic, M. Ursic	NEPTUNE_CFD simulations of heat transfer around cylindrical core fragment in sodium coolant
V. Duffal, R. Manceau, B. de Laage de Meux	Hybrid RANS-LES modeling of unsteady turbulent loads in hydraulic pumps
R. Zhukovskii, C. Chazelas, A. Vardelle, V. Rat	Control of the movement of the arc in a cascaded-anode plasma torch
S. Rolfo, C. Moulinec, D. R. Emerson	Transitional flow around a heated cylinder
C. Defforge, B. Carissimo, M. Bocquet, R. Bresson	Improving local wind simulations with Code_Saturne using data assimilation

Abstracts

Posters

Out-of-equilibrium two-phase flow model - Coupling with complex equations of state

L. Quibel (EDF R&D, UNISTRA), P. Helluy (UNISTRA), O. Hurisse (EDF)

Safety issues require reliable numerical simulations of two-phase flow phenomena which may occur within a pressurized water nuclear reactor. Some accidental scenarios, like vapor explosion, LOCA or RIA, involve rapid transients with important mass transfer. A model able to account for strong thermodynamical disequilibria may thus be needed to get more accurate results. For that purpose, a six-equation homogeneous model has been implemented in *Code_Saturne*. This model accounts for the full thermodynamical disequilibrium - i.e. in terms of the pressures, temperatures and chemical potentials. In order to couple this homogeneous model with realistic (complex) equations of state, robust and accurate numerical schemes are required. In this work, the homogeneous model is coupled with complex equations of state: look-up tables built on the IAPWS-97 formulation. This coupling is first verified on the basis of analytical Riemann problems, with initial states at thermodynamical equilibrium or out-of- equilibrium. Two numerical schemes are compared: a classical Rusanov scheme, very robust but diffusive, and a relaxation scheme proposed by Chalons and Coulombel, which is more accurate and remains robust. Then, the influence of the relaxation time describing the return towards thermodynamical equilibrium is highlighted by using simple relaxation time-scale models built on the basis of the classical nucleation theory.

The incompressible Navier-Stokes equations with Compatible Discrete Operator schemes

R. Milani (EDF, ENPC), J. Bonelle (EDF), A. Ern (ENPC)

We investigate the extension of Compatible Discrete Operator schemes (CDO) to the unsteady, incompressible Stokes equations. CDO is a unifying framework on low-order mimetic schemes which preserve at the discrete level the structural properties of the PDEs, such as conservation laws, while still ensuring competitive computational performances. A CDO module is implemented in *Code_Saturne* and it is used for groundwater flows and storage-related problems in an industrial context. We present the results of a manufactured test case for the face-based version of CDO applied to the unsteady Stokes equations by using the Artificial Compressibility method. The latest developments towards the Navier-Stokes equations will be discussed as well.

Conjugate heat transfer: a big challenge for RANS modeling

G. Mangeon (EDF, UPPA), S. Benhamadouche (EDF), J.-F. Wald (EDF), R. Manceau (UPPA)

Predicting heat transfer in fluid flows and solid parts is crucial for EDF industrial applications, particularly in nuclear engineering. A reliable estimate of turbulent heat fluxes, the mean temperature and its variance is a key issue in order to improve the safety and efficiency of the nuclear power plants. Here, authors propose low-Reynolds number models to predict the turbulent heat fluxes and the temperature variance and its dissipation rate in the fluid and solid parts. These models use the elliptic blending approach to treat the near-wall region on the fluid side. This poster also speaks about a big issue of the dissipation rate of the temperature variance modeling: its discontinuity at the fluid/solid interface. This thermal variable is essential to estimate the temperature variance in the fluid and solid parts. Finally, a reliable estimate of the temperature variance is crucial to study for instance the thermal fatigue.

Integral formulation for fluid flow in congested media modelling
C. Colas, M. Ferrand, J.-M. Hérard, E. Le Coupanec (EDF)

We focus on a numerical method to compute fluid flows in a medium congested by obstacles. The integral formulation consists in integrating the equations of fluid mechanics over the fluid part of mixed control volumes including both fluid and solid zones. This technique uses the *Code_Saturne* finite volume pressure-correction compressible algorithm. Fast transient validation test case, describing a shock wave interaction with a set of obstacles, is performed. A good agreement is observed between the integral formulation numerical results and the fine CFD solution where the wall of obstacles are explicitly meshed.

Review of Atmospheric Case Studies with *Code_Saturne*
L. Makke, X. Wei, M. Nibart, V. Michaud (ARIA)

We want to present a review of some interesting CFD studies we carried out last year. - Training a customer to the use of radiative transfer module for an indoor application. - A methodology to study wet plume dispersion model - Intercomparison between *Code_Saturne* and Lagrangian Particles and Puff Dispersion Model to study rocket launch dispersion - Wind exposure study on the CDG Airport - Depolluting system solutions in dense urban area.

Prediction of hydrodynamic loads on a cable near a rough seabed
K.I. Kuznetsov (ENPC/LHSV), J.C. Harris (ENPC/LHSV), N. Germain (France Energies Marines)

In the present work we consider the forces on a fixed slender cylinder, laying on the seabed or with some vertical height, in order to understand the stability of submarine cables, in particular for cases with large seabed roughness. Semi-empirical Wake models have been developed which include some physical understanding of the problem, shown to be accurate for a given range of parameters specific to the oil and gas industry, and until now have been used for bottom-attached cylinders. For the general case, unlike the case of a cylinder resting on the seabed, one must handle complex vertical forces due to vortex shedding. Once this is taken into account, we are able to reproduce the horizontal and vertical forces on a cylinder with a Wake model, even for cases when the cylinder is not resting directly on the seabed. The model will be validated by a comparison of the efforts measured during basin tests.

Optimum design of spillways using a metaheuristic algorithm
F. Oukaili, Y. Bercovitz, C. Goeury, F. Zaoui, E. Le Coupanec (EDF)

Spillway shape optimization based on *Code_Saturne* Spillways are hydraulic structures provided for storage and detention dams to release surplus floodwater, which cannot be contained in the allotted storage space. They ensure that the water does not overflow and damage or destroy dams. Consequently, spillway is one of the most important component of a dam. Some failures of dams have been reported due to inadequate capacity or improper design of spillway. The water shedding capability of spillway depend on its geometric shape. Currently, design of most spillways is based on empirical studies. The aim of this work is to propose an optimized shape increasing the water shedding capability of the spillway by minimizing the upstream water level for a fixed discharge. Thus, the spillway performance is numerically evaluated with *Code_Saturne* simulations and improved with the help of a meta-heuristic. A comparison of the final optimized shape with the initial standard geometry is carried out.

Absorption of regular water waves by a sponge layer in the Volume Of Fluid module of *Code_Saturne*

P. Landesman (EDF/LHSV), C. Peyrard (EDF/LHSV), J.C. Harris (ENPC/LHSV), M. Benoît (AMU)

Absorption of surface water waves by sponge layers implemented in *Code_Saturne*'s VOF method In the context of my thesis's research work, aiming at simulating wave-structure interactions by coupling a potential solver to *Code_Saturne*'s VOF module with a domain-decomposition approach, I had to study the reflection of regular waves on the absorbing sponge layer. The study was conducted in a 2D numerical wave-tank, and the relaxation relied on linear source terms added to the momentum equation only or alternatively to both the Navier-Stokes equations. For given characteristics of the incident wave train, reflection was assessed through the combination of two probes measuring the instantaneous free surface elevation at two different locations of the tank. It was then possible to quantify the influence of various numerical parameters on the reflection coefficient: the sponge layer length, the intensity of the damping, the selection of equations on which the source terms were imposed, and the choice of the target solution. It was then found that the relaxation implemented in the VOF module of *Code_Saturne* was able to absorb waves over a reasonably short distance with low reflection rates. This is a promising conclusion, as relaxation strategies will be used for the viscous/inviscid coupling.

NEPTUNE_CFD simulations of heat transfer around cylindrical core fragment in sodium coolant

M. Tekavcic, M. Ursic (JSI)

NEPTUNE_CFD simulations of heat transfer around cylindrical core fragment in sodium coolant during a hypothetical core melt accident in innovative sodium cooled fast reactors, the rapid and intense heat transfer interaction between molten core material and sodium coolant could lead to a vapour explosion. The opaqueness of sodium causes difficulties for the experimental investigations of the pressurization process with the heat transfer and the vapour production around the melt. Thus, currently the investigation of the pressurization process in sodium must be supported using analytical research and precise numerical simulations. Presented work focuses on the two-dimensional simulations of heat transfer around a cylindrical core melt fragment in sodium coolant, which we perform with the NEPTUNE_CFD code. The initial conditions were set based on the expected conditions during the vapour explosion in sodium: pressures up to 10 MPa, velocities up to 20 m/s and fragment size of 0.1 mm. Reasonable agreement between the single-phase simulation results and the relevant heat transfer correlations was obtained. Our final objective is to perform the film boiling simulations in sodium to assess the applicability of the Epstein-Hauser heat transfer correlation.

Hybrid RANS-LES modeling of unsteady turbulent loads in hydraulic pumps

V. Duffal (EDF, UPPA), R. Manceau (UPPA), B. de Laage de Meux (EDF)

The development of a consistent continuous hybrid RANS/LES approach based on temporal filtering is presented. This new formalism leads to the development of the Hybrid Temporal LES (HTLES) method. An upgraded version, focusing on improving the model for wall-bounded flows, is integrated in *Code_Saturne*. Notably, a shielding function and an internal consistency constraint are added to impose the RANS behavior in the near-wall regions. The calibration and validation of the model are finally carried out on various test cases: decaying isotropic turbulence, channel and periodic-hill flows.

Control of the Movement of the Arc in a cascaded-anode plasma torch

R. Zhukovskii, C. Chazelas, A. Vardelle, V. Rat (Univ. Limoges)

This study is intended to develop a reliable and adaptive time-dependent model of D.C. electric arc plasma torch. The model works with various geometries and electric arc shapes; it is able to take into account arc motion in swirling gas flows and external magnetic field; it also describes thermal disequilibrium of Argon plasma. Thermal disequilibrium of plasma is considered by including two thermal scalars: electron enthalpy and heavy species enthalpy (Argon atoms and ions). The key feature of this study compared to the other ones in the field is that the two electrodes are coupled with the Argon plasma and it has advanced boundary condition for magnetic vector potential. The electrodes are needed in the computational domain for better prediction of magnetic field.

Transitional flow around a heated cylinder

S. Rolfo, C. Moulinec, D. R. Emerson (STFC)

Flow transition around a heated cylinder Overhead Power Lines (OHL) are a key component to transmit and distribute electrical energy. The aero-thermal design of OHL is generally carried out considering the cable as a perfect heated cylinder immersed in a uniform cross flow. The design also assumes that only two regimes occurs, depending on the Reynolds number (based on the line diameter and the incoming velocity): (a) natural convection for low velocities, e.g. below a wind speed of 0.6 m/s which roughly corresponds to a Reynolds number of 850 (b) force convection at higher Reynolds numbers. In both of these cases analytical correlations exist to compute the Nusselt. However, for moderate Reynolds numbers, the cylinder is generally operating in mixed convection and aero-thermal characteristics of the flow might largely vary from the assumption of simple force or natural convection. As a consequence, the focus of this work is to investigate the so called transition mode E, where large 3D structures appears in the wake of the cylinder. Particular focus will be given at the effect of the Prandtl number on the development of the 3D wake and the generation mechanism responsible for the transition. The effect of the angle between the undisturbed (or incoming) velocity and the gravity vector will be also presented.

Improving local wind simulations with Code_Saturne using data assimilation

C. Defforge, B. Carissimo, M. Bocquet, R. Bresson (EDF/CEREA)

Precise wind fields simulated by CFD models, such as the atmospheric module of *Code_Saturne*, are used for many environmental and safety micro-meteorological applications, such as dispersion modelling or wind potential assessment. Atmospheric simulations at local scale are largely determined by boundary conditions, which are provided, for instance, by outputs of meso-scale models (e.g., WRF). In order to improve the accuracy of the boundary conditions, especially in the lowest levels, data assimilation methods might be used to take available observations into account. Data assimilation methods have been generally developed for larger scale meteorology and deal with initial conditions. Among the existing methods, the iterative ensemble Kalman smoother (IEnKS) has been chosen as it is independent of the atmospheric model and it is able to handle non-linear operators. The IEnKS has been adapted to local scale atmospheric simulations by taking boundary conditions into account. This adapted version has previously been tested on a simple shallow-water model in 1D. Here, we analyse the performances of the IEnKS in 3D with the CFD model *Code_Saturne* using both twin experiments and real observations over a realistic, very complex topography. The IEnKS is also tested in urban conditions with observations provided by the Mock Urban Setting Test field campaign.

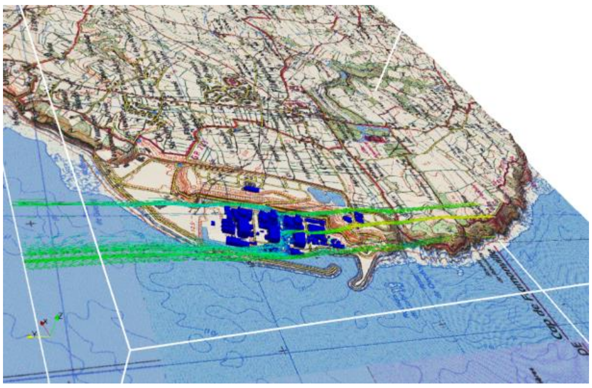
Abstracts

Oral Presentations

Code_Saturne: quality is in the air

Martin Ferrand (EDF/CEREA), Cédric Flageul (ENPC/CEREA)

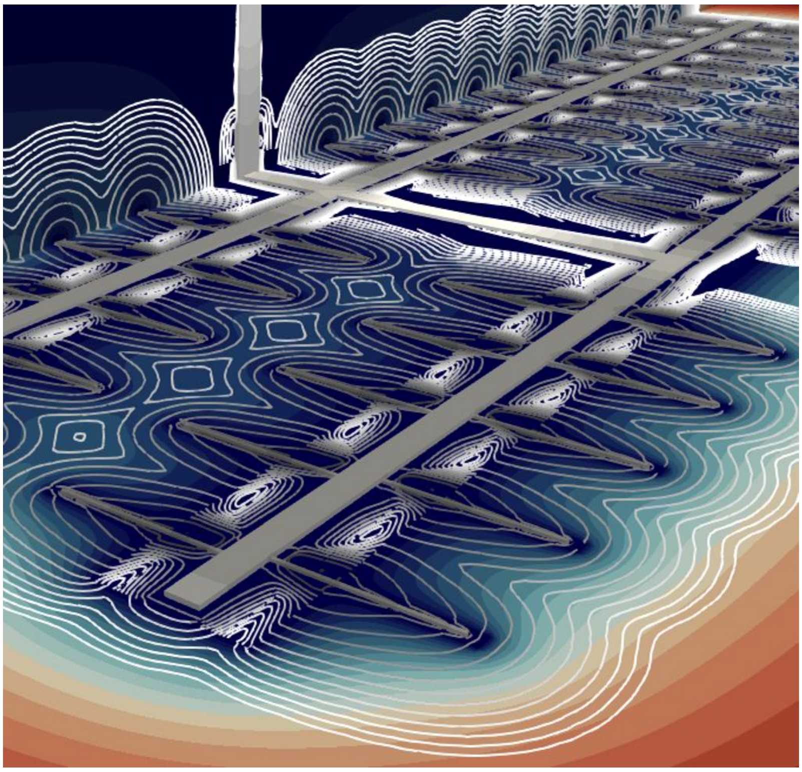
The presentation first introduces the specificities of the atmospheric module of *Code_Saturne* developed by the CEREA lab: energy is solved in terms of potential temperature for dry atmosphere and liquid potential temperature for humid atmosphere (in addition to humidity transport). In case of stable or unstable meteorology, buoyant turbulent production is taken into account. Rough wall functions taking buoyant effect into account are used. Chemical reactions and aerosol can be taken into account with shared models from Polyphemus platform (also developed by CEREA lab). Radiative transfer is treated with a 1D or a 3D approach (Discrete Ordinate Method). Dedicated pre-processing steps are automatized: a script produces meteo files used for initialization and boundary conditions. A major improvement has been recently done in the Salome-based auto mesher from a shapefile, allowing representation of several thousand buildings. Validation of the atmospheric module is synchronously performed in the same way as the rest of the code and some VnV cases are based on SIRTA experimental site on Palaiseau shelf. A data assimilation method has been recently tested with *Code_Saturne* to improve atmospheric simulations at local scale. This data assimilation method is used here to precisely estimate the boundary conditions though it is flexible and could be used for other *Code_Saturne* applications. *Code_Saturne* is involved in the downscaling approach (WRAPP methodology) implemented to estimate the annual energy production of offshore wind farms. Meteorological profiles from mesoscale simulations are used after a classification step for boundary/initial conditions. *Code_Saturne* is used at local scale to calculate the annual energy loss due to the wake effects in the wind farm. Tackling open boundaries is a key point for atmospheric module. Since version 5.2, a new treatment is available to deal with boundaries switching from inlet to outlet. New open boundaries and improvements in Salome based mesher will be illustrated on a study on the Villiers-sur-Marne city around the A4 motorway.



Code_Saturne for groundwater flow applications: Case of long-term safety studies

Jérôme Bonelle (EDF), Raphaël Lamouroux (EDF)

Groundwater flow is one of the applications available in *Code_Saturne*. In this presentation, this module is used for long-term safety studies related to the CIGEO project. CIGEO is a deep storage center for high-activity long-life nuclear wastes. In this long-term safety study, one aims at evaluating the distribution of radionuclides inside the computational domain as well as the quantity exiting the domain. Several difficulties arise when studying such phenomena: dealing with heterogeneous soil properties, generating a computational domain of length scale about 1km with small geometrical details (about 0.1m), simulating about 1,000,000 years. Thanks to numerical schemes called Compatible Discrete Operator (CDO), recently integrated, and an advanced usage of *Code_Saturne* 's mesh manipulations, simulations on the full geometry with meshes between 500 million and 1 billion cells were carried out. Results demonstrate the capability of *Code_Saturne* to handle efficiently such problem.



Code_Saturne and NEPTUNE_CFD activities in China

Tingting Xu (EDF R&D – China)

Code_Saturne, as a generic CFD software, has been known and recognized by more and more Chinese users. NEPTUNE_CFD, with its powerful two-phase flow model, caused strong interests of Chinese partners. During the past one year, R&D China continues the technical research work on application domain with Code_Saturne and NEPTUNE_CFD.

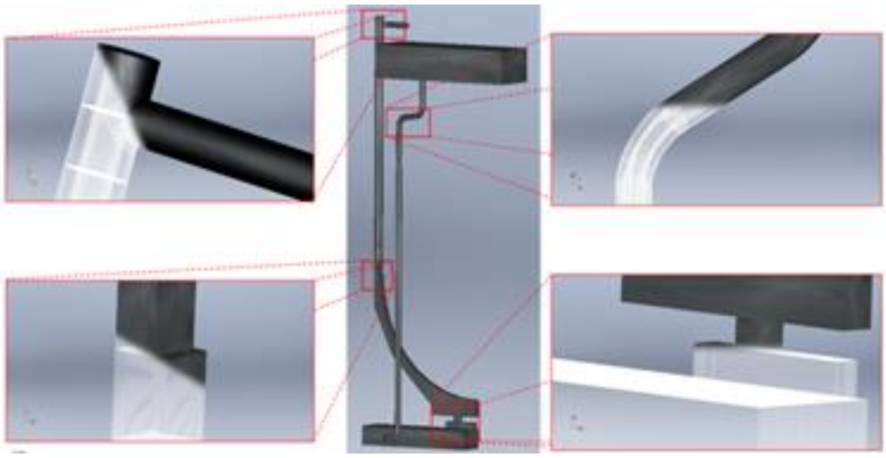
- *Code_Saturne*

EDF R&D works together with MFEE on Fuel Assembly project to do the research on hydraulic force loading in non-deformed/deformed structure. With local partner – CAS-IMR, an ingot segregation phenomenon were studied to understand the mechanisms behind ingot solidification and segregation by experiment & simulation.

- NEPTUNE_CFD

For severe accident, EDF has cooperated with CGN since 2016. CGN is launching the experiment on IVR. A benchmark with NEPTUNE_CFD and Fluent are launched to study the thermal-hydraulic behavior of metallic corium pool and external cooling loop. With Xi'An Jiaotong University, there is a project to do the NEPTUNE_CFD V&V research based on ECCS and ADS-4 experimental data.

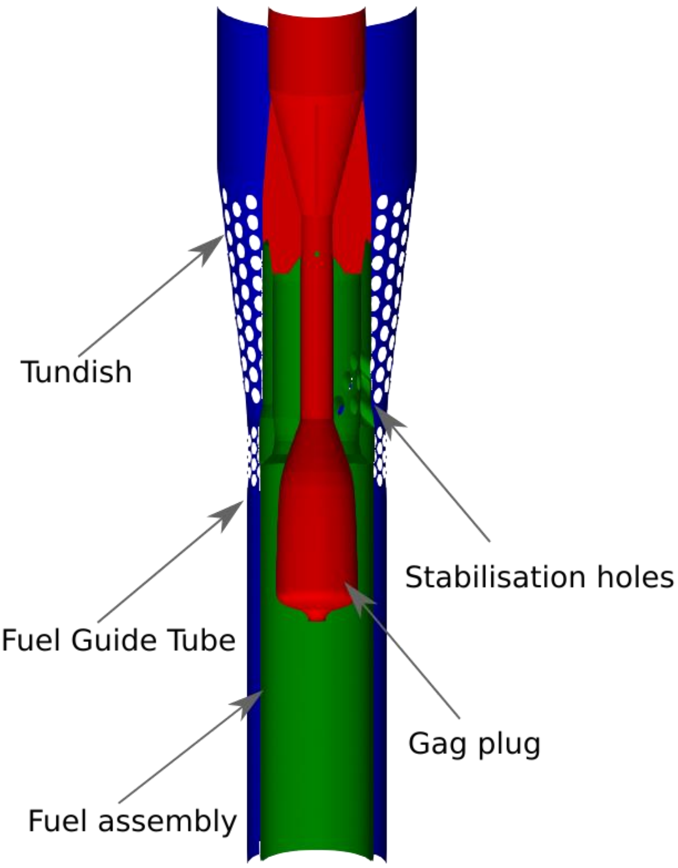
Since 2011, EDF R&D China works on popularizing *Code_Saturne* in the research and industry domain through the cooperation with Chinese partners and the organization of open-session training. Last year, a four-day *Code_Saturne* training was organized in Xi'An with a mini user club. The participants gave good feedback and demanded on advanced session in the future.



Recent applications of *Code_Saturne* at EDF R&D UK Centre

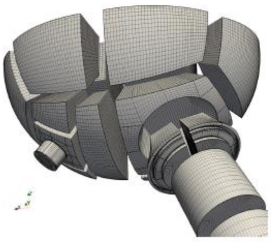
Juan Uribe (EDF R&D – UK Center)

EDF Energy is the sole operator of nuclear reactors in the UK, with 14 Advanced Gas-cooled Reactors (AGRs) currently in its fleet. The reactors use CO 2 as primary coolant, working at high temperatures and pressures. These reactors are fast approaching their end of life, with an estimated final shutdown by 2030. In order to maximise output, a Plant Lifetime EXtension (PLEX) programme has been put in place to assess the need for a safe operation through the next years. The objective of this talk is to present the latest uses of *Code_Saturne* in the different regions of the reactors where some problems are being tackled by the use of Computational Fluid Dynamics. These include flow inside the fuel assembly (figure 1) and its interaction with the box at the top of the reactor, flow inside the boilers and gas/steam release scenarios. The code is widely used by the Modelling and Simulation Team which is part of the Nuclear team at the EDF Energy R&D UK Centre. The team is based at the Modelling and Simulation Centre (MaSC) at The University of Manchester.

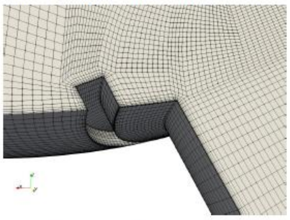


Measurement of primary coolant flow rate from a pressure difference
Romain Camy (EDF)

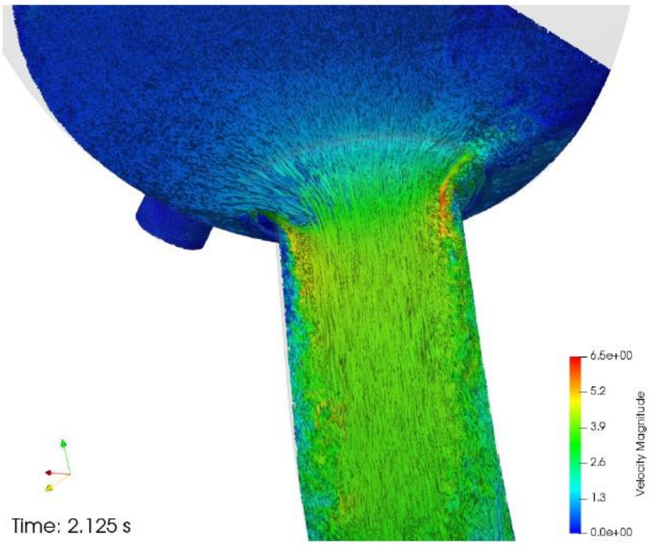
Measurement of primary coolant flow rate with ΔP_{elbow} aims at using in Reactor Coolant System the pressure difference in the elbow at the outlet of Steam Generator water box. This measure would be complementary and independent from current measure with a balance of enthalpy. An accurate prediction of the flow in this configuration is complicated because of complex flow patterns: transition from laminar to turbulent, many vortices oriented in different directions, multiple boundary layer separations, etc. Also the Reynolds number is high: 10^6 experimentally and 10^8 in an actual PWR. This presentation shows most recent results with SALOME CFD on this application. Simulations with wall-resolved RANS models and Large Eddies Simulations (LES) were performed with a large range of grid refinements from 330 K cells to 725 M. Numerical results are very consistent with each other and an independence of the results towards the grid refinement seems to be achieved for the LES between 169 M and 725 M cells.



(a) First level blocking of the grid with 330 K cells



(b) Zoom on boundary layer mesh (2.6 M cells)

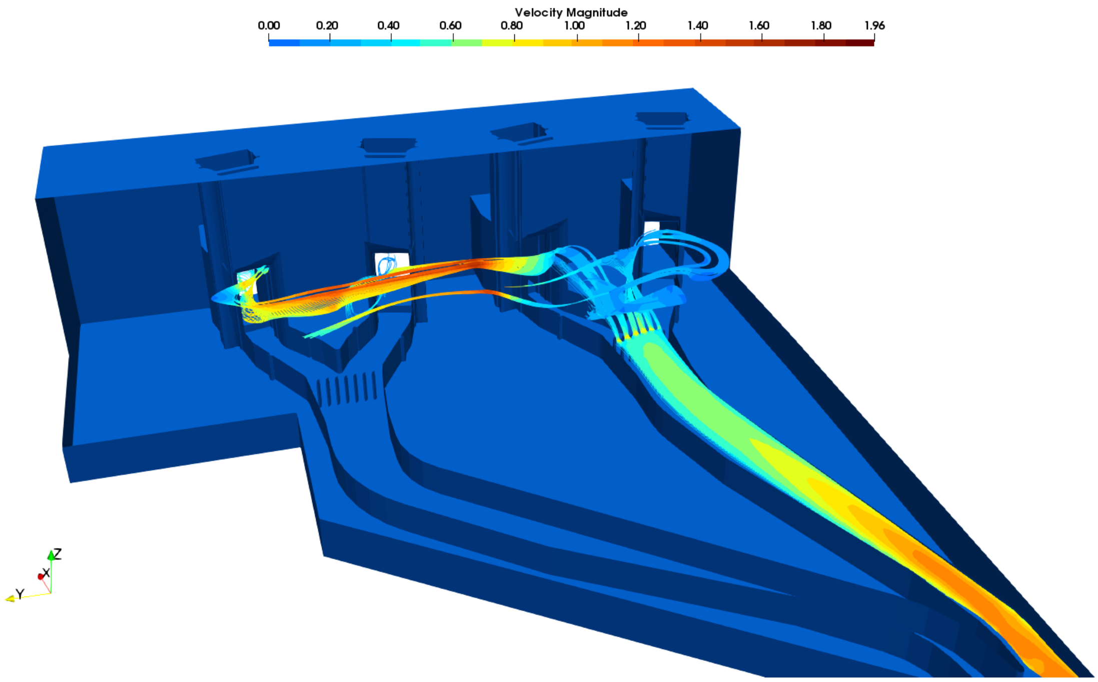


(c) LES, $Q = 0$ (169 M cells)

Design of submerged thermal diffusers by numerical modelling

Olivier Bertrand (ARTELIA)

Design of submerged thermal diffusers by numerical modelling In winter conditions, and in order to avoid the risk of freeze-up, it may be useful to discharge warm water upstream of the intake circuits. The system, implemented on several sites, consists in positioning diffusers just in front of the intake galleries. The positioning, dimensioning and thermal efficiency of these diffusers are studied by the use of the *Code_Saturne*. The sensitivity of the results is examined by comparing them with other software systems.



Avoiding segregation with Code_Saturne
Charles Demay (EDF)

This work deals with the modeling and simulation of solidification process occuring in ingot castings. During that process, heterogeneities in the solidified alloy, called macrosegregations, may be obtained, affecting the mechanical characteristics of the resulting piece. In order to evaluate and understand the formation of macrosegregations, a mixture model has been implemented in *Code_Saturne*. This model is an incompressible model with Boussinesq approximation and composed by four mixture conservation equations, i.e. momentum, mass, heat and solute conservation. It is complemented with a microsegregation model given by the lever rule. Regarding the mushy zone, a porosity term is added in the momentum conservation equation (Carman-Kozeny relation) and the solid phase is assumed motionless. Numerical experiments are performed on academic and industrial cases, comparing with other models and experimental data

Bayesian calibration of a two-group IATE model for air-water flows
N. Leoni (CEA), M.-G. Rodio (CEA), Emmanuel Kuidjo (CEA), P. Congedo (INRIA), O. Le Maître (CMAP)

The analysis of accidental scenarios in nuclear safety studies involves expensive simulations of two-phase flows focused on reproducing transitional two-phase regimes, from bubbly to annular one. Numerical simulations are affected by several sources of errors, including discretization errors, uncertainties in the physical parameters and modeling errors. In this context, the uncertainty quantification methods applied to the simulation allow, on the one hand, to define most prescriptive models and, the other one, to verify the sensitivity of our models/codes to some tuning parameters. In the present study, parametric uncertainty of an two-group IATE model is estimated by running a Bayesian calibration. We run simulations of vertical co-current cap-turbulent air-water flows in a rectangular channel using NEPTUNE_CFD and compare with experimental data. A sensitivity analysis, using a surrogate model constructed from polynomial chaos bases, is provided. This results in better predictions of the experiments and an associated measure of uncertainty.

Towards a numerical reactor for LOCA with NEPTUNE_CFD
N. Mérigoux, C. Caruyer (EDF)

This presentation focuses on the analysis of the thermohydraulic behaviour in the cold legs, the downcomer and the steam generator of a Pressurized Water Reactor primary circuit during an Intermediate-Break Loss Of Coolant Accident scenario located on one of the cold legs. The use of CFD to model the critical parts of this transient scenario can help to better understand and qualify previous results carried out with system-scale codes and presented in safety analysis. Firstly, a physical analysis of the accidental scenario is proposed to comprehend the modelling complexity on an industrial geometry. Secondly, a brief overview of NEPTUNE_CFD validation cases which are consistent with the physical phenomena highlighted in the first part will be given to reinforce the confidence that can be brought to the numerical results. Finally, a description of the engineering approach that has been used to model this LOCA on a French PWR is provided, with a particular focus brought on the physical analysis of the numerical results by comparison with the previous physical analysis and system-scale codes results.

Moving solids every way in multiphase flows with NEPTUNE_CFD

W. Benguigui, J. Laviéville (EDF)

In the field of energy production, there are possibilities for some components to move freely with slight or large displacements. For example, steam generator tubes vibrate because of the steam/water cross flows. The aim of the present work is to be able to simulate these free-motion phenomena with a unique numerical method in single or two-phase flows. To do so, a CFD code based on a two-fluid approach is used. A "discrete forcing" method is implemented in order to allow solid body motion in a single- or two-phase flow. The validation is performed with simple and industrial cases using experimental data and theoretical results. Using an existing implicit algorithm, a fluid-structure coupling based on the developed interface tracking method is implemented. Validated for single and two-phase flows, it is now possible to have a solid motion induced by fluid forces. Finally, feasibility test cases and industrial applications are computed for nuclear, hydraulic and solar fields. This validation shows the advantages of a discrete forcing method coupled to a fluid-structure interaction module in a code dedicated to nuclear applications. In the future, it will allow to study more in details phenomena appearing in nuclear power plants to increase our understandings.

Development of *Code_Saturne* for Rarefied Gas Flow

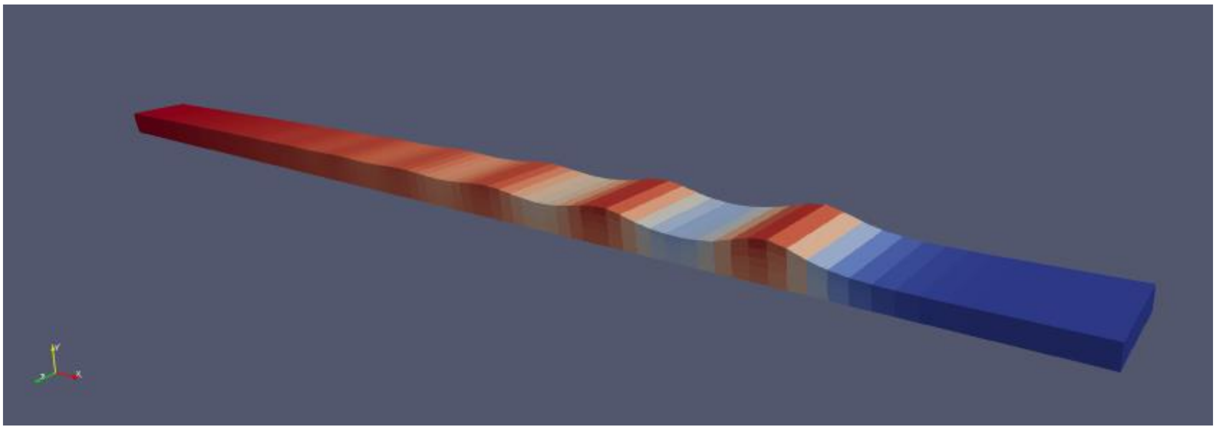
D. Emerson (STFC)

Development of *Code_Saturne* for Rarefied Gas Flow Accurately capturing rarefied gas flow represents a fundamental modelling challenge and problems exist in many industrial applications and scientific research facilities, including mass spectrometry, low-pressure environments, vacuum pumps, micro-electro-mechanical systems, high-altitude vehicles, and porous media. We have recently implemented the Method of Moments into *Code_Saturne* to extend its capability in handling complex rarefied gas flows and we will present our initial results.

Applications of the Arbitrary Lagrangian-Eulerian approach for modeling water waves

J. Harris (ENPC/LHSV)

The use of the ALE (Arbitrary Lagrangian-Eulerian) module in *Code_Saturne* permits, with proper selection of parameters, the modeling of water wave propagation and wave-structure interaction problems. Results have been validated against standard benchmarks and compare favorably with specialized wave models and experimental results. These validation tests include basin tests of forces and overturning moments on monopiles and gravity based foundations, the viscous decay of standing waves, and even for non-breaking wave runup. The resulting model can be used for novel applications, such as the development of a new type of wavemaker intended for the study of tsunamis, and even propagation of tsunamis on the globe.



Cleaning up with *Code_Saturne* and SALOME

N. Tonello (Renuda)

CleanTech plays an increasingly important part in everyday life and with regard to industrial processes, with applications ranging from limiting emissions to reducing waste through optimising processes or introducing new technologies. In turn, the quest for efficiency is pushing digital modelling, and CFD in particular, into industries which had so far relied on more traditional techniques. Renuda will present some of its recent applications of *Code_Saturne* and SALOME to water treatment and dust capturing projects and the applied, research and development work carried out over the past year as part of its collaborative research partnership with EDF R&D. Using *Code_Saturne*, Renuda worked with Veolia on the design and optimisation of different sections of a planned drinking water plant in Ivory Coast. The Lagrangian multiphase framework was utilised to evaluate the performance of wood dust separators in collaborative projects on plywood manufacturing with the heavy goods manufacturer, Siempelkamp. The approaches, models and salient results for these two series of projects will be described during the presentation. As part of its development work in collaboration with EDF R&D, the multiphase VoF model initially introduced by Renuda within the same partnership was augmented with the capability to account for interfacial forces. Its application in combination with the Lagrangian model was also demonstrated. A summary of this work and illustrative results will be presented.

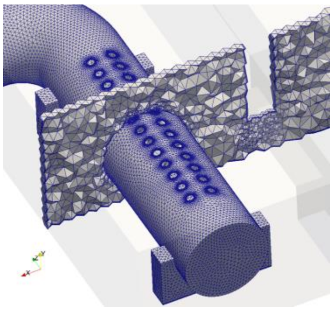


Fig. 1: Drinking water plant.

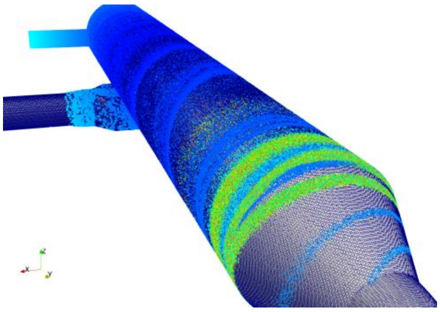


Fig. 2: Wood dust separation.

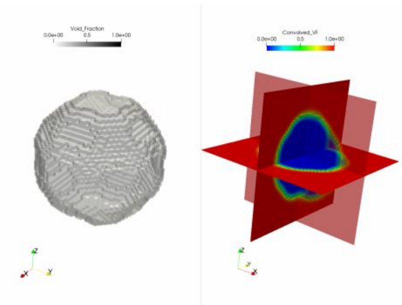


Fig. 3: Superficial force