The image is a composite of two photographs. The upper portion is a 3D simulation of a wind turbine. It features a white tower, a blue nacelle, and three blue blades. A complex grid surrounds the tower, and a color-coded flow field is shown near the blades, transitioning from blue at the top to red at the bottom, indicating air movement. The lower portion is a photograph of a real-world wind farm. It shows several white wind turbines standing on a hillside covered in dark green shrubs. A dirt path leads towards the turbines. The sky is clear and blue.

2025 edition,
November, 6

code_saturne & neptune_cfd Users' Meeting

code_saturne and neptune_cfd user day

Morning agenda



09h00–09h30

- 09h30 Introduction.
- 09h40 New features overview in code_saturne v9.0.
- 10h10 Presentation of Simvia, an EDF subsidiary.

Welcome / Breakfast

- P. Charles (EDF R&D)*
- Dev. Team (EDF R&D)*
- F. Leray (Simvia)*

Presentations – morning session (part 1)

- 10h15 An assessment of algebraic and differential Reynolds-Stress Models for a highly-bent serpentine aircraft intake.
- 10h35 Modelling and simulation of a three-phase stirred tank.
Modeling of sodium spray combustion with neptune_cfd.

- S. Hanrahan (Melbourne University)*
- R. Ansart (IMFT)*
- N. Kirov, O. Simonin (IMFT)*

11h05–11h30

Break

Presentations – morning session (part 2)

- 11h30 Study of vortex intrusion phenomena in dead legs.
- 11h50 Simulation of solar receivers for direct steam generation using neptune_cfd coupling.
- 12h10 Advancing code_saturne for hydraulics: validation and application to industrial problems.

- JF. Wald, J. Uribe (EDF R&D)*
- I. Aguilera (CNRS PROMES)*
- P. Asproulis, Y. Eude (RENUDA)*

12h30–14h00

Lunch / Poster session

code_saturne and neptune_cfd user day

Afternoon agenda



12h30–14h00

Lunch / Poster session

Presentations – afternoon session

- | | | |
|-------|---|-----------------------|
| 14h00 | A review of code_saturne developments in STFC UKRI. | S. Rolfo (STFC UKRI) |
| 14h20 | Overview of neptune_cfd simulations for reduced-scale filling experiments. | A. Doradoux (SIREHNA) |
| 14h40 | Simulation of breaking wave loads on a wind turbine foundation in the coastal zone with code_saturne. | M. Benoît (EDF R&D) |

15h00–15h25

Break

Flash session

- | | | |
|-------|--|--|
| 15h25 | Implementation of a log-normal modeling in neptune_cfd for polydispersed flows. Presentation of high-fidelity (WR-LES) simulation with code_saturne of turbulent flow under mixed convection regime within a heated rod bundle. Go Viking – Development and use of EDF CFD tools for vibration prediction in axial and crossflow conditions. On-the-fly construction of a ROM during code_saturne simulations: an urban boundary layer example. System monitoring of flow-accelerated corrosion through upscaling of CFD models coupling thermohydraulics and chemistry. Grand Challenges SELENA: Determination of inter-assembly flow redistributions with explicit modeling of mixing grids at large scale. | N. Cailler (EDF R&D) V. Duffal (EDF R&D) W. Benguigui (EDF R&D) K. Kuznetov (GRASP) B. Cellé (EDF R&D) R. Ceyrolle (EDF DT) |
| 16h10 | An overview of CEREA activities with code_saturne. Atmospheric flow modelling of floating offshore wind turbines: coupling between code_saturne and the aero-hydro-servo-elastic model DIEGO. | M. Ferrand, A. Mathieu (CEREA, EDF R&D) |
| 16h45 | Prospects in code_saturne & neptune_cfd. | Dev. Team (EDF R&D) |

17h00

Closure

Presentation of Simvia, an EDF subsidiary

by F. LERAY – SIMVIA

Simvia is an EDF subsidiary dedicated to open-source scientific simulation software for engineering. We build upon more than 35 years of EDF R&D expertise in multi-physics modeling and high-performance computing. Our mission is to empower industrial and academic partners with transparent, flexible, and sovereign simulation tools. Simvia

provides expert support, training, and development services around EDF's open-source codes such as `code_saturne`, Code aster, and Salomé-Meca. Through our open-source commitment, we help organizations regain control of their simulation capabilities and accelerate innovation.



An assessment of algebraic and differential Reynolds-stress models for a highly-bent serpentine aircraft intake

by S. K. HANRAHAN (1), J. A. SCHEIBEL (2), A. GROIS (2), M. MARCEL STÖSSEL (2), M. KOZUL (1), S. JAKIRLI (3), D. KOVZULOVIC (2), R. D. SANDBERG (1) – DEPARTMENT OF MECHANICAL ENGINEERING, THE UNIVERSITY OF MELBOURNE, PARKVILLE, VICTORIA, AUSTRALIA (1), INSTITUTE OF JET PROPULSION, UNIVERSITY OF THE BUNDESWEHR MUNICH, NEUBIBERG, BAVARIA, GERMANY (2), INSTITUTE FOR FLUID MECHANICS AND AERODYNAMICS, THE TECHNICAL UNIVERSITY OF DARMSTADT, DARMSTADT, HESSE, GERMANY (3)

Serpentine ducts are commonly used as intakes for modern low-bypass aeroengines, drawing atmospheric air through to the aero-engine compressor. Because of their highly-bent geometry, serpentine ducts develop significant unsteady flow features that can, in turn, cause compressor instabilities. Quantifying the performance of a coupled intake-compressor system requires analysis of a large parameter space, which is typically undertaken with URANS calculations based on linear eddy-viscosity models. In the present work, we study a highly-bent serpentine intake with three classes of turbulence model, and assess the intake as a turbulence mod-

elling benchmark. A comparison is made between the $k - \omega$ SST eddy-viscosity turbulence model, the explicit algebraic Reynolds-stress model of Wallin and Johansson (2000) [2], as well as the differential Reynolds-stress model of Manceau (2015) [1]. Comparisons are made between experimental wall pressure measurements and URANS calculations along the centreline, and radial cross-sections of the intake. Detailed analysis is undertaken in critical near-wall regions of the intake. Finally, based on the outcomes of the flow physics analysis, future modelling opportunities are discussed for algebraic and differential Reynolds-stress models.

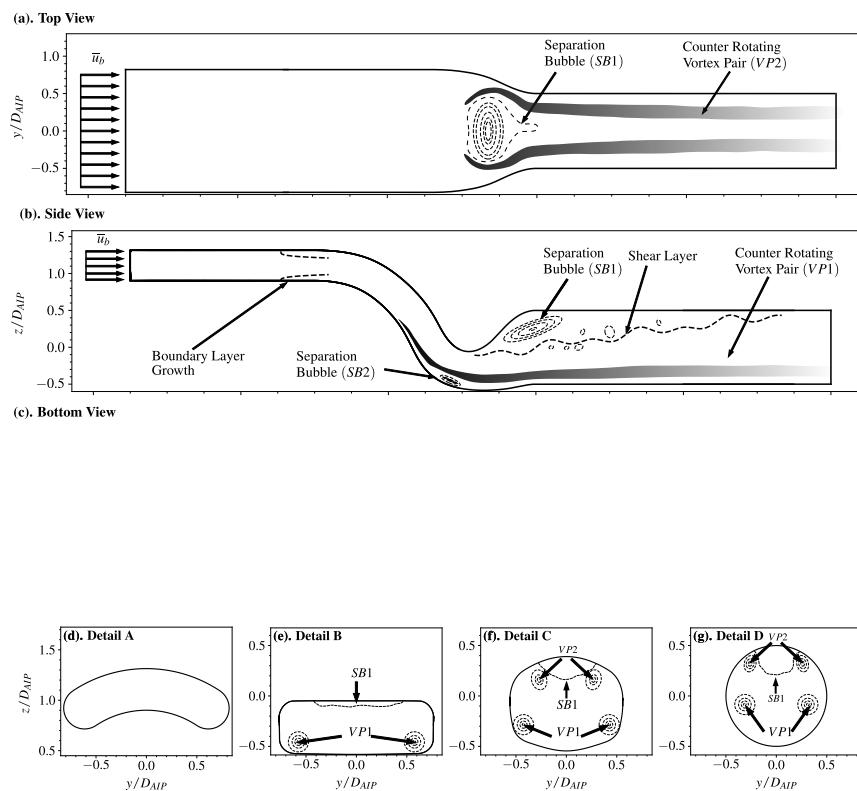


Figure 1: A simplified visualisation of the flow physics in the MEIRD benchmark. **(a)**. Top View ($x - y$ plane). **(b)**. Side View ($x - z$ plane). **(c)**. Bottom view ($x - y$ plane). Cross-sections through the MEIRD at **(d)**. $x/D_{AIP} = 2.92$, **(e)**. $x/D_{AIP} = 0.83$, **(f)**. $x/D_{AIP} = 0.29$, and **(g)**. $x/D_{AIP} = -0.03$

[1] Manceau, R. (2015). Recent progress in the development of the Elliptic Blending Reynolds-stress model. International Journal of Heat and Fluid Flow, 51, 195-220.

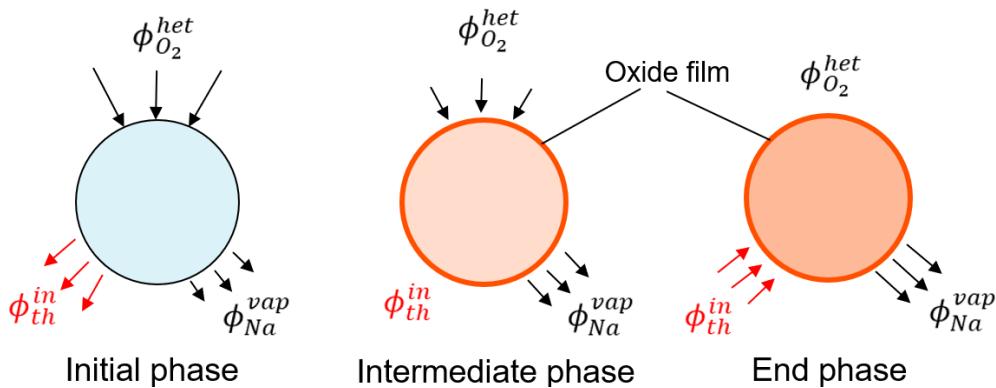
[2] Wallin, S., and Johansson, A. V. (2000). An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows. Journal of fluid mechanics, 403, 89-132.

Modeling of sodium spray combustion with neptune_cfd. Modelling and simulation of a three-phase stirred tank.

by NICOLAY KIROV – OLIVIER SIMONIN – ENRICA MASI – LYNDA PORCHERON – PIERRE PLION IMFT

The safe operation of IV generation sodium-cooled fast reactors requires accurate prediction of sodium spray combustion, which can occur during leaks or accidental releases. When liquid sodium is exposed to the atmosphere, it undergoes rapid oxidation, releasing significant heat and producing flammable reaction products. To capture these processes, spray combustion of dispersed sodium particles in air is numerically investigated using an N-Eulerian approach implemented within the `neptune_cfd` platform. The model simultaneously captures three coupled phenomena governing sodium-air reactions: droplet evaporation,

heterogeneous surface combustion, and turbulent gaseous combustion driven by the reactive sodium vapors. Evaporation and surface combustion are treated using a combined diffusion–chemical kinetics formulation, while gaseous combustion is modeled via a presumed beta-PDF approach, assuming either infinitely fast chemistry or chemical equilibrium. This integrated framework enables a fully coupled treatment of interphase mass, momentum, and energy exchange, overcoming limitations of existing lumped-parameter and Eulerian–Lagrangian methods.



Modelling and simulation of a three-phase stirred tank

by RENAUD ANSART – IMFT

Numerical modeling of processes is a major challenge for the development of new complex processes based on innovative concepts. Here, we represent the local behavior of a stirred liquid-gas-solid reactor with a complex geometry, based on an industrial configuration, by determining the operating conditions as a function of the properties of the particles according to the state of progress of a gas-solid chemical reaction. The design objectives of this process are to suspend the particles but also to ensure good contact between the gas and the particles, which are the two reactants. From a numerical point

of view, the objectives are to manage the coupling between the three phases involved: liquid, gas and solid. Particular attention needs to be paid to the various liquid-gas interfaces, since it is necessary both to predict the diameter of the bubbles and to consider the presence of this gaseous phase in the form of a gas sky at the top of the tank. Finally, agitation in this three-phase medium will be taken into account using a moving mesh. To simulate this process, we are using a N-Euler approach implemented in `neptune_cfd`.

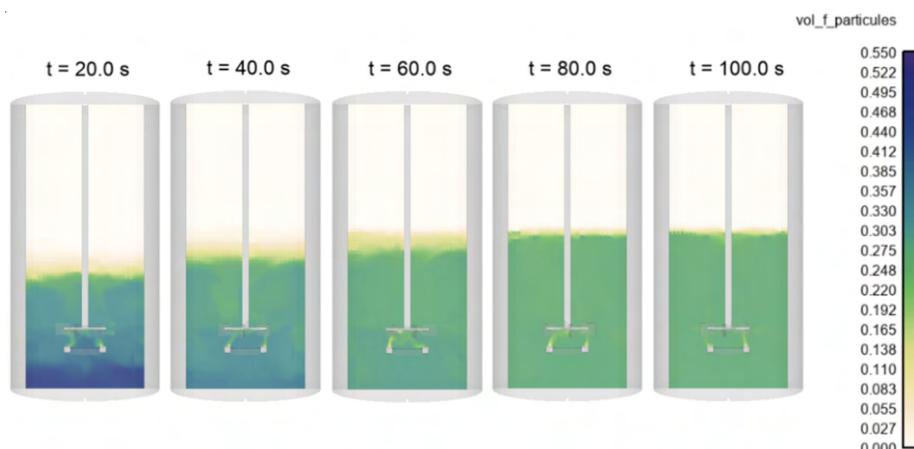


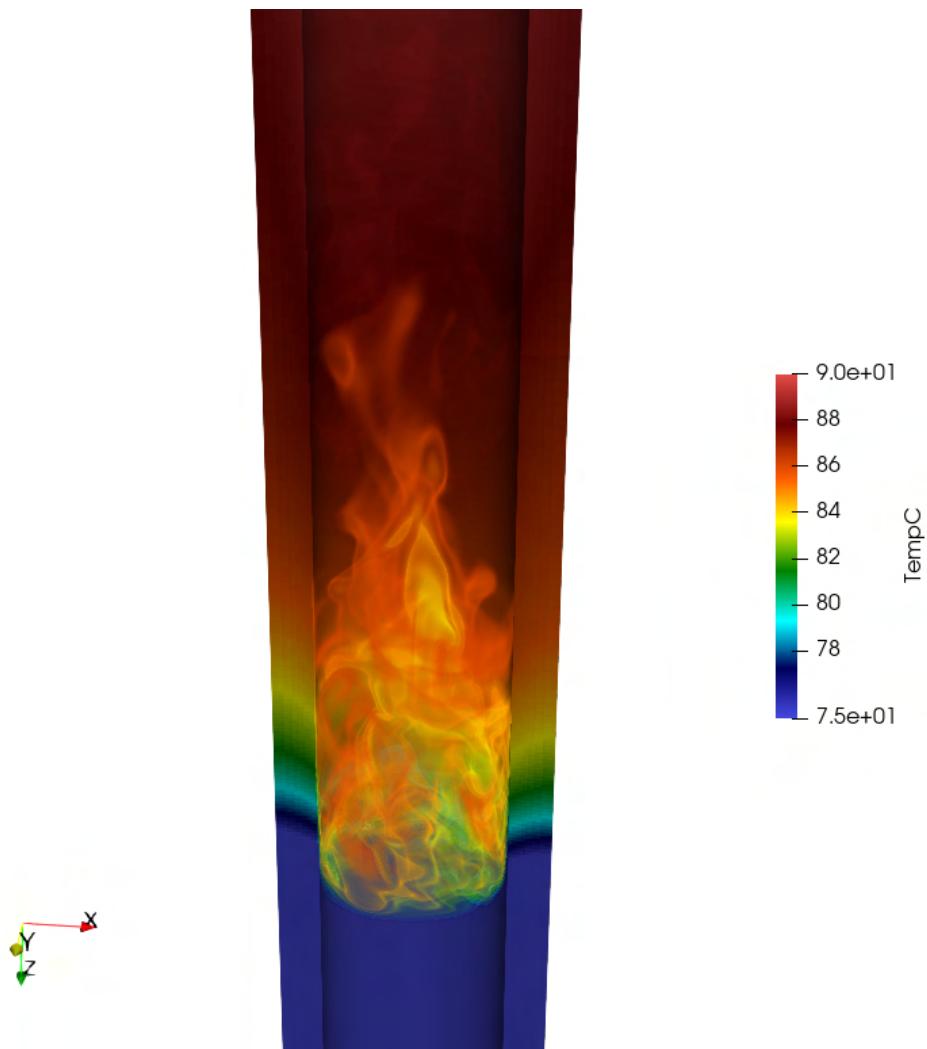
Figure 1: Evolution of the solid volume fraction over time.

Study of vortex intrusion phenomena in dead legs.

by J. F. WALD, J. URIBE – EDF R&D

The “dead branch” configuration could be encountered in industries that have piping systems such as the nuclear industry and, particularly, in the primary circuits of Pressurized Water Reactors (PWR). It might play a key role in issues related to thermal fatigue or stress corrosion, as the phenomenon might transport the temperature. The numerical prediction of dead leg flows using CFD remains a significant challenge for the engineering community. The difficulty stems from a combination of factors, including the vast disparity in length

and time scales between the fast flow in the main branch and the slow, evolving flow in the dead leg. The physical time required for the main vortex to establish itself can be on the order of several thousand seconds, necessitating extremely long simulation runtimes. Furthermore, accurately capturing the shear layer dynamics requires a fine mesh resolution at the junction. The purpose of the present work is to evaluate a URANS approach based on the k-omega SST model with conjugate heat transfer on existing validation cases for dead leg flows.

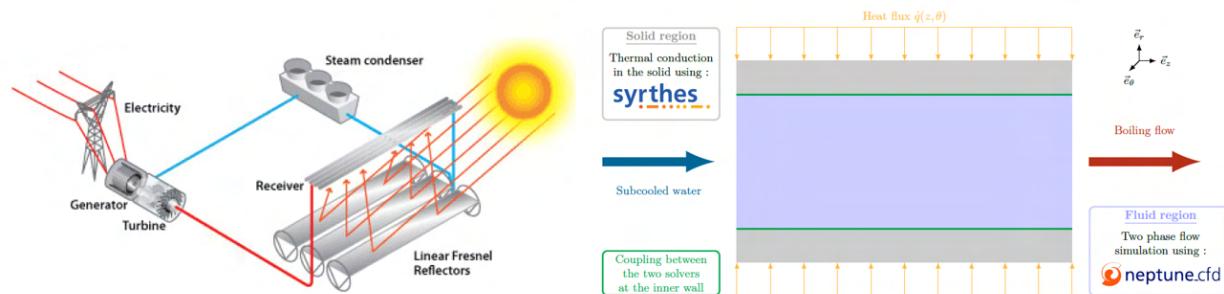


Numerical simulation of oxycombustion in fluidized bed at pilot and industrial scales with neptune_cfd

by I. AGUILERA – CNRS PROMES

Direct steam generation technology in solar receivers is positioned as a promising solution for decarbonising industry. In this technology, concentrated solar receivers are used to generate steam directly in the solar field, eliminating the need for heat transfer fluids, mainly molten salts. However, the coexistence of water and steam phases in the receiver introduces complexities and operational prob-

lems. At the PROMES laboratory, a small-scale experimental facility has been built to study the phenomenon of water boiling in horizontal and slightly inclined pipes. In parallel, coupled simulations have been carried out with **neptune_cfd** and SYRTHES to model both the fluid and solid parts of the receiver. The aim of this study is to compare the results of the simulations with the experimental results.



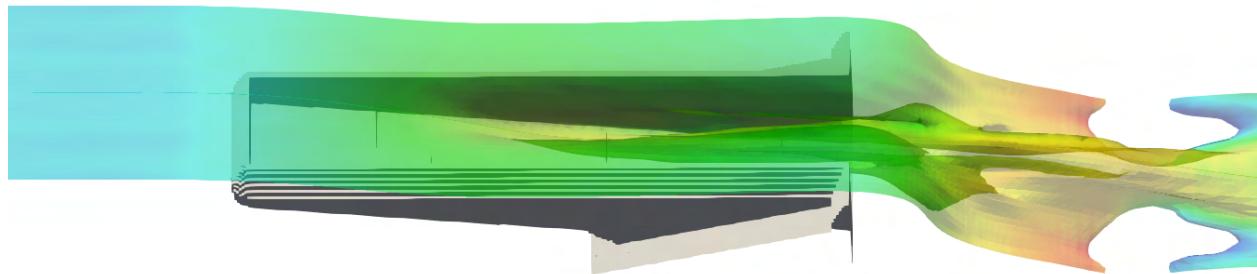
Advancing code_saturne for hydraulics: validation and application to industrial problems.

by P. ASPROULIS, Y. EUDE – RENUDA

Water is fundamental to life on earth, impacting a large range of essential topics and activities, including food production, sanitation, flooding and energy generation. Whilst access to safe drinking water has been declared a human right by the UN since 2010, population growth, conflicts, climate change, and environmental considerations are bringing added urgency to using water sensibly and responsibly. In turn, industry relies increasingly on digital models and studies to optimise its concepts for water pro-

duction, distribution and utilisation or, even, recycling.

Renuda will present some of its recent work in relation to the development and application of the VoF, separated two-phase flow functionality of code_saturne. Highlights include validating code_saturne on representative cases in collaboration with EDF, enhancing its usability for hydraulics application, and applying code_saturne to the design of an industrial water treatment plant.



Modelling water flow over a weir.

A review of code_saturne developments at STFC-UKRI

by C. MOULINEC, GU XIAOJUN, BO LIU, C. TSINGINOS, H. JONES, G. CARTLAND-GLOVER, WEI WANG AND S. ROLFO – THERMO-FLUID GROUP, SCIENTIFIC COMPUTING DEPARTMENT, STFC-UKRI

Over the past five years, the Thermo-Fluid group within STFC-UKRI has led and contributed to several software development projects using `code_saturne`. These projects extend the framework beyond its original focus on thermal-hydraulics to address new physics and applications. This presentation will highlight three independent examples of developments: SubChannelCFD, a micropolar solver, and the adaptation of `code_saturne` for image-based modelling.

The first development, SubChannelCFD, introduces a coarse-mesh methodology that combines traditional CFD with empirical correlations typical of sub-channel codes. This approach delivers robust and accurate results comparable to Reynolds-Averaged Navier–Stokes (RANS) simulations at a fraction of the computational cost. We have applied this methodology to several reactor types, including PWR and HTGR, under both operational and transient conditions (see Fig. 1).

The second development focuses on implementing the micropolar theory, a continuum approach that simulates fluids with microstructure without resorting to costly molecular or atomistic models. Such

flows occur in many natural and industrial processes, including landslides, avalanches, lateral spreading, blood flow with clots, polymers, granular flows, liquid crystals, suspensions of rigid and/or deformable particles, and even turbulent flows. The presentation will introduce the features of the new solver (see Fig. 2) and present its scalability performance.

The third development adapts `code_saturne` for image-based modelling to maximize the use of Micro-X-ray Computed Tomography (XRCT) data and enable efficient data-processing pipelines. This work integrates parallel HDF5 formats—typical of XRCT output—with `code_saturne`'s mesh manipulation capabilities to generate complex geometries on the fly (such as the electrode bilayer shown in Fig. 3). We resolve these geometries for diffusion processes using the CDO discretization.

Finally, we will discuss the importance of building a development community around `code_saturne` to consolidate contributions, streamline distribution and adoption, and create pathways for integrating community developments into the main release of the framework.

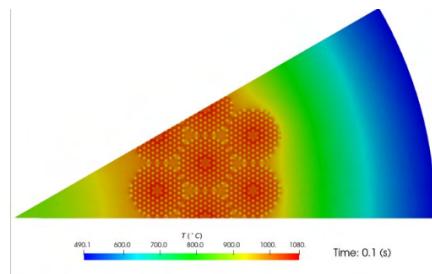


Figure 1: Cross section of a HTGR reactor core during a LOFA scenario.

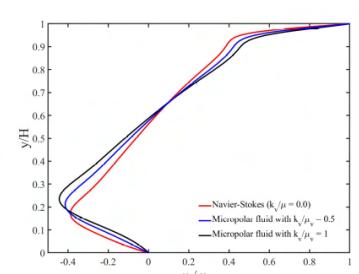


Figure 2: Cavity flow for different micropolar fluids.

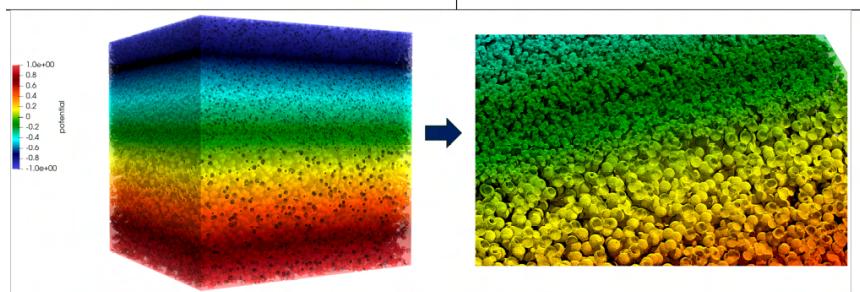


Figure 3: Potential across a section of bilayer electrode. Full case (L) and close up of cross section (R)

Simulation of the water filling of a cylindrical cavity using neptune_cfd

by T. DRAPPIER (1), A. DORADOUX (1), J. RUA (2), B. AGUILAR (3), V. ROIG (3), O. PRAUD (3) – NAVAL GROUP 44340 (1), NAVAL GROUP 83190 (2), IMFT (3)

The water-filling of a self-pressurized cavity, due to the ejection of a moving body, exhibits a complex two-phase flow involving different scales of gas structures. The time evolution of pressure at the bottom of the tube - and hence the filling time - depends strongly on the amount of bubbles that are created and on the pressure oscillations caused by the compressibility of the gas phase.

In an experimental setup (see below), a circular cylinder, initially in air, is placed beneath a water tank. The moving body is partially inserted inside the cavity, separating it from the tank. When the body is pulled out, a large bubble is driven by the body, caus-

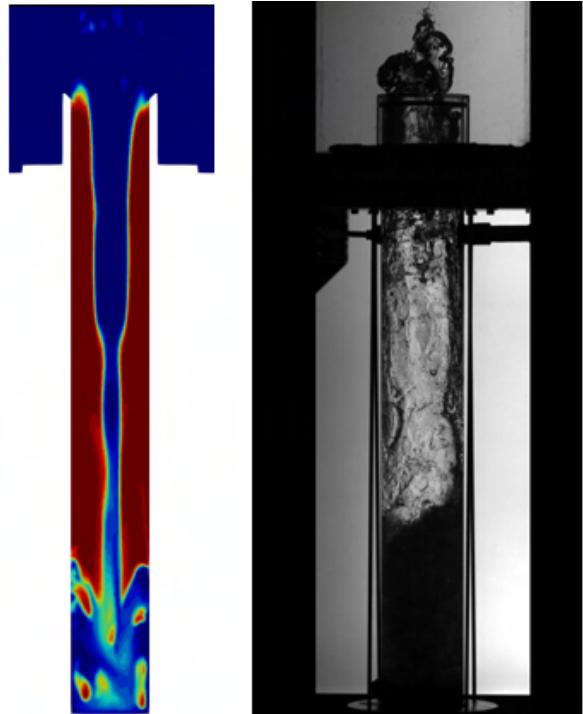
ing the expansion of the gas, that leads to the pinch-off of the bubble. Then, the cavity is gradually filled, with gas structures transitioning from large bubbles to smaller, finer bubbles as filling continues.

This experiment is simulated using *neptune_cfd* with a Euler-Euler approach and the Generalized Large Interface Model to handle multiple flow regimes (for both separated and dispersed topologies). The body motion is simulated using an immersed boundary method.

The bottom and top pressures are compared to experimental datas. The simulation gives encouraging results.



Experimental setup, IMFT.



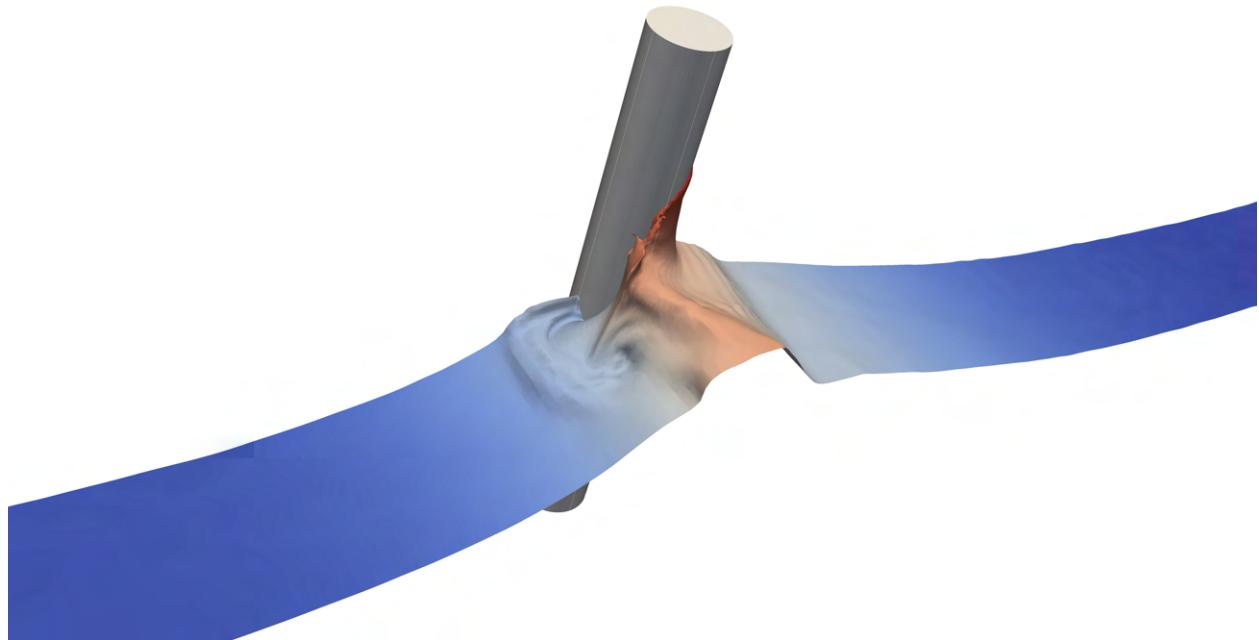
Volume fraction of gaz in a slice.

Simulation of breaking wave loads on a wind turbine foundation in the coastal zone with code_saturne

by W. ZHANG (1), J. HARRIS (1), M. BENOIT (1,2) – EDF R&D, LHSV (1), EDF R&D, LNHE (2)

Monopile foundations are widely used in the nearshore and coastal zones for offshore wind turbines. Wave breaking due to shoaling over sloping seabeds frequently subjects monopile foundations to strong and transient slamming forces, which are critical in the design process of these foundations. The benchmark experiments of Irschik et al. (2004) have been widely adopted for validating CFD models, with Li and Fuhrman (2022) demonstrating great agreement using different turbulence closure models. In this study, the open-source solver `code_saturne` is applied to simulate breaking waves and reproduce this benchmark experiment. The volume of fluid method is used to capture the complex free-surface

dynamics. Several turbulence models are tested to compare their ability to describe wave breaking and to capture the complex flow structures in the vicinity of the structure. The numerical predictions of free-surface elevations and breaking wave-induced forces show good agreement with the experimental data. These results provide a robust foundation for subsequent impact force calculations and allow to use the numerical model as a digital twin of the experiments to be performed in a wave flume at EDF R&D Laboratoire National d'Hydraulique et Environnement (LNHE) and Saint-Venant Hydraulics Laboratory (LHSV) in 2026.



Improvement of closure models for condensation processes in a two-phase flow simulation approach

by N. CAILLER (1, 2), S. MIMOUNI (1), W. BENGUIGUI (1), C. COLIN (2), T. BONOMETTI (2), P. FEDE (2) – EDF R&D (1), IMFT

The separated-effect TOPFLOW experiment is a bubble condensation test facility in which bubbly flows are obtained by injecting steam bubbles in sub-cooled water [1] (+ fig.1). This experiment is thus particularly suited for better estimating the Interfacial Area Concentration (IAC) used to close dispersed phase equations derived from the method of moments in `neptune_cfd`. This method of moments, and therefore the IAC, rely on Bubble Size Distributions (BSD), currently modeled in `neptune_cfd` with a Dirac (monodispersion) or quadratic (polydispersion) approach [2, 3]. However, data collected at different heights of the TOPFLOW facility show that BSD exhibit a lognormal behavior. In order to

improve the IAC estimation, a log-normal law has been implemented in `neptune_cfd`, needing a supplementary transport equation due to the increased number of degrees of freedom of this law [4]. Analytical expressions have been derived for the interfacial heat transfer source terms, interfacial forces and interfacial area transport equation source terms. Results show that, for most of the cases, the log-normal approach tends to better predict the IAC or at least as well as the quadratic one (fig.2). Nonetheless, current simulations clearly underestimate small bubbles population at the core, indicating insufficient condensation due to relative velocity discrepancies (fig.3).

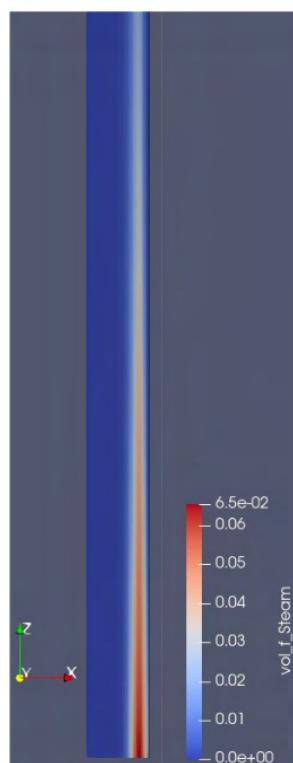


Fig. 1: Gas volume fraction at the end of a TOPFLOW simulation with `neptune_cfd`

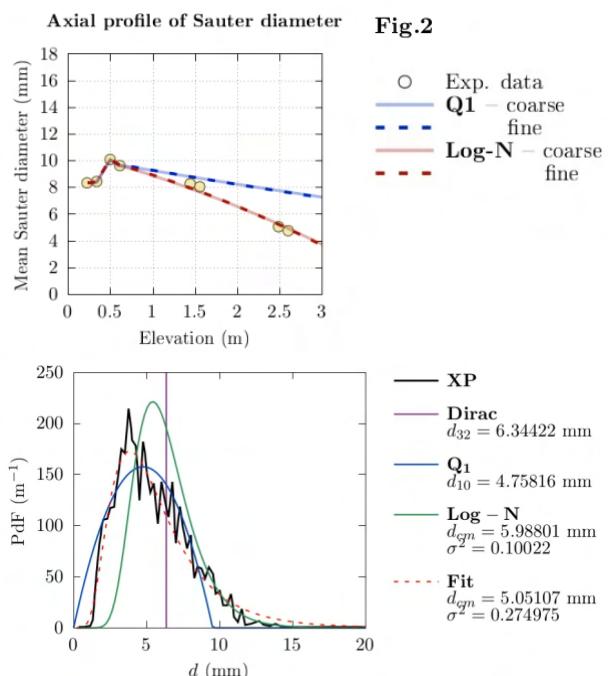


Fig. 3: Comparison between experimental and simulated BSD at a given cross-section of the facility

- [1] Lucas, D., & Prasser, H. M. (2007). Steam bubble condensation in sub-cooled water in case of co-current vertical pipe flow. Nuclear Engineering and Design, 237(5), 497-508.
- [2] Ruyer, P., Seiler, N., Beyer, M., & Weiss, F. P. (2007, July). A bubble size distribution model for the numerical simulation of bubbly flows. In 6th International Conference on Multiphase Flow, ICMF. Leipzig, Germany, July (pp. 9-13).
- [3] Herry, T., Raverdy, B., Mimouni, S., & Vincent, S. (2024). Modeling and simulation of bubble condensation using polydisperse approach with bubble collapse model. International Journal of Multiphase Flow, 176, 104846.
- [4] Kamp, A. M., Chesters, A. K., Colin, C., & Fabre, J. (2001). Bubble coalescence in turbulent flows: A mechanistic model for turbulence-induced coalescence applied to microgravity bubbly pipe flow. International Journal of Multiphase Flow, 27(8), 1363-1396.

High-fidelity Simulation of a Turbulent Mixed Convection Flow in a Heated Rod Bundle

by V. DUFFAL, A. LAIEB, E. LE COUPANEC, S. PUJET, S. BENHAMADOUCHE, R. VICENTE CRUZ, C. FLAGEUL, E. LAMBALLAIS – EDF R&D, CMCC FOUNDATION-EURO-MEDITERRANEAN CENTER ON CLIMATE CHANGE ITALY, CURIOSITY TEAM, PPRIME INSTITUTE CNRS

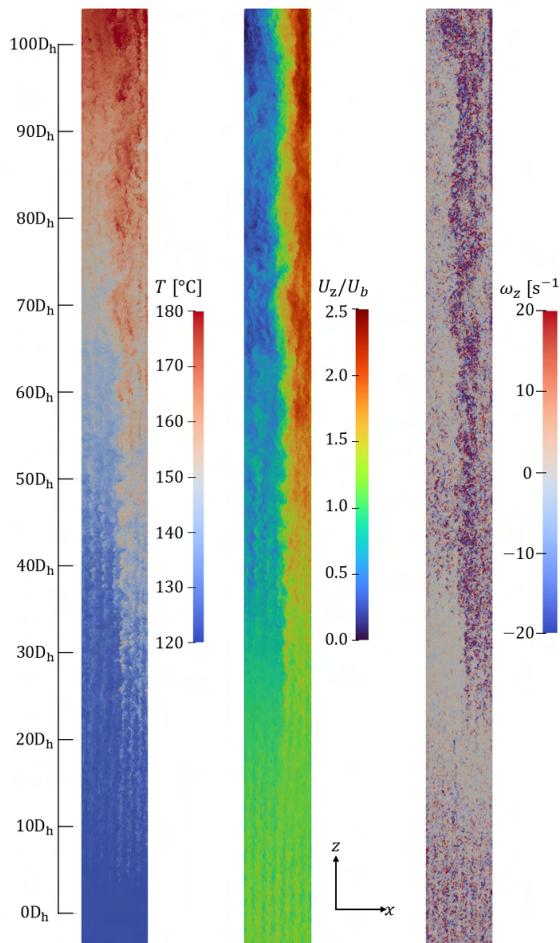
A WR-LES calculation is performed using code_saturne to investigate an asymmetrically heated upward flow within a tube bundle configuration. This low-Reynolds buoyancy-affected flow is representative of the mixed convection regime encountered in the core rod bundle, following the shutdown of the primary pumps during Main Steam Line Break transients.

code_saturne predictions are evaluated against a numerical experiment: a high-accuracy database provided by a reference quasi-DNS calculation performed at Pprime institute [Vicente Cruz et al., 2024]. Not only does this turbulent mixed convection flow exhibit strong cross-flows leading to the growth of

a thermal plume, but buoyancy-induced partial laminarization, turbulence redevelopment and mixing layers are noticed.

Despite these non-trivial physical phenomena, the very satisfactory match between the two calculations demonstrates the capability of the in-house industrial CFD solver code_saturne to carry out high-fidelity LES computations of buoyancy-affected flows within a heated rod bundle. The present simulation, relying on the Boussinesq approximation, paves the way for future calculations using more realistic variable fluid properties.

This presentation follows on from an article presented at the NURETH-21 conference (2025).



Go Viking – Development and use of EDF CFD tools for vibration prediction in axial and crossflow conditions.

by W. BENGUIUI – EDF R&D.

GO-VIKING (Gathering expertise On Vibration ImpaKt In Nuclear power Generation) is an Euratom project focused on NPP flow-induced vibration challenges. Through experimental and numerical investigations, these interactions are further studied and improved modeling methodologies are developed.

Axial and cross flow-induced vibration are studied in both single and two-phase flows. EDF contribution is focused on numerical simulations, the objective is to present our results with `code_saturne` and `neptune_cfd` on the different benchmarks and the associated developments.

Reduced order model for efficient prediction of dynamics of urban boundary layer in Paris

by K.KUZNETSOV, O.DUBOVIK, E.ALEKSEENKO – GRASP EARTH, LABORATORY OF ATMOSPHERIC OPTICS, UNIVERSITY OF LILLE, UNIVERSITÉ DU LITTORAL CÔTE D'OPALE.

Understanding and predicting the dynamics of the Urban Boundary Layer (UBL) is essential for improving urban weather forecasts, air quality modeling, and assessments of energy efficiency and thermal comfort. However, the high computational cost of full-scale Computational Fluid Dynamics (CFD) simulations limits their applicability for real-time or large-domain studies. This work introduces a Reduced Order Model (ROM) designed to efficiently reproduce UBL dynamics in complex urban environments such as central Paris.

The proposed ROM is based on Streaming Dynamic Mode Decomposition (sDMD), which extracts dominant spatio-temporal modes representing key physical processes including wind circulation, temperature stratification, and turbulence mixing. In the

offline stage, a comprehensive database of sDMD modes is built from CFD simulations under various meteorological conditions. During the online stage, these modes are adaptively interpolated to reconstruct the evolving UBL flow fields without performing new CFD runs.

Applied to a representative urban case, the ROM demonstrates strong agreement with high-fidelity CFD results, achieving root mean square errors below 8% for major flow and thermodynamic variables while providing a computational speedup exceeding three orders of magnitude. These results highlight the ROM's potential for real-time urban climate prediction, fast scenario evaluation, and integration into large-scale air-quality or digital-twin frameworks for cities like Paris.

CFD modeling of Flow-Accelerated Corrosion (FAC) coupled with electrochemistry in the secondary circuits of pressurized-water reactors

by B. CELLE, T. LASSEUR, S. MIMOUNI, G. SIMONINI, S. SHKIRSKIY, F. KANOUI – EDF R&D, UNIVERSITÉ PARIS CITÉ.

Keywords: Flow-accelerated corrosion, CFD coupled model, PWR secondary circuit

Flow-Accelerated Corrosion (FAC) is a phenomenon that causes wall thinning in low-alloy steel pipes. It results from complex interactions between hydrodynamics, water chemistry, temperature, and metal composition. FAC has long been a safety concern in the secondary circuits of Pressurized Water Reactors (PWRs), where several pipe ruptures have already occurred. Numerical modeling, using Computational Fluid Dynamics (CFD), provides a powerful tool to investigate this phenomenon under conditions representative of PWR secondary systems.

To achieve reliable predictions of FAC, this study focuses on developing a fully mechanistic model that incorporates species mass transport, electrochemical reactions, and water chemistry. In addition, it accounts for the protective effect of magnetite as a function of pH and temperature and include the influence of metal surface roughness on local hydrodynamics. The model was implemented using `neptune_cfd`, the multiphase CFD software co-developed by EDF, CEA, ASNR, and Framatome, which solves the mass, momentum, and energy equations for both the liquid and vapor phases.

The model was first applied and validated for single-phase water flow in a straight tube geometry, using the RANS approach for turbulence modeling. Experimental data were taken from pipe wall loss

measurements on the CIROCO test loop (EDF R&D, 1988), designed to study FAC under PWRs secondary circuit conditions in single-phase flow. The study investigated the effects of temperature, mean flow velocity, water pH at 25°C (controlled with ammonia), and hydraulic diameter, with the system pressure maintained at 70 bars.

Fig. 1 (left) indicates that the FAC rate increases with flow velocity, regardless of temperature, due to enhanced convection and turbulence, which the model accurately captures. We observe that the peak corrosion rate occurs at approximately 150°C, which is consistent with the experiment. The decrease in FAC rate with increasing temperature results from the protective effect of magnetite. The comparison between the model predictions and the experimental results revealed a generally good agreement. Several simulations were also conducted on an elbow geometry. Fig. 1 (right) shows that the FAC rate is maximal at the leading edge of the intrados and at the downstream of the extrados.

Using `neptune_cfd`'s tools, the model can further be applied to investigate two-phase effects on FAC, such as those caused by phase changes in secondary circuits, which will be addressed in future simulations.

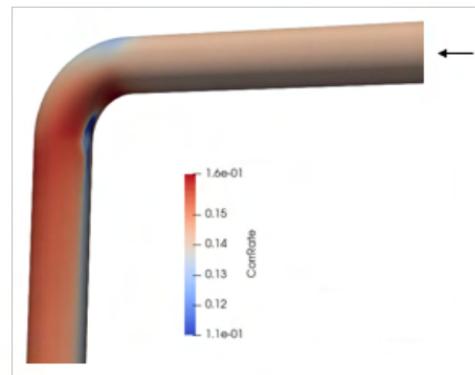
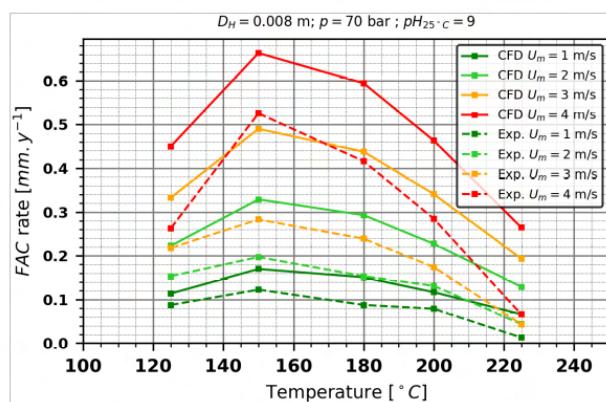


Figure 1. Left: FAC rate as function of temperature for $pH_{25^\circ C} = 9$ and various mean flow velocities compared with CIROCO measurements. Right: Spatial visualization of FAC rate on an elbow.

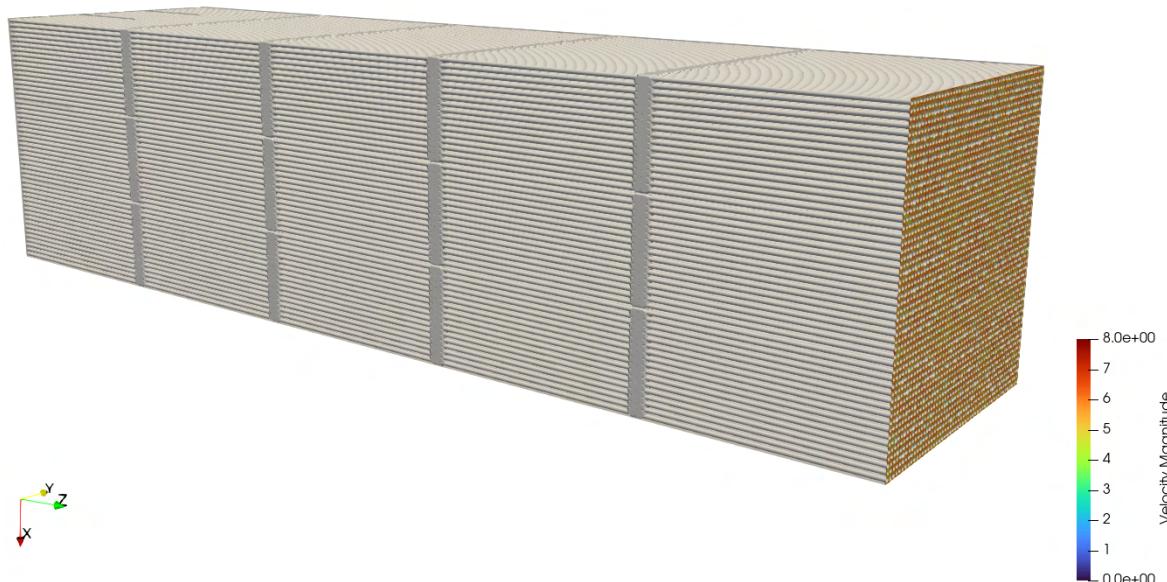
Grand Challenges SELENA – Determination of Inter-Assembly Flow Redistributions with Explicit Modeling of Mixing Grids at Large Scale

by R. CEYROLLE – EDF DT.

Within a reactor core, the inlet flow field can exhibit significant heterogeneities. For studies performed with component-scale codes, the accurate representation of inter-assembly flow redistribution dynamics is a key issue for EDF. In the absence of experimental data that could serve as a reference for this physical phenomenon, CFD simulations under reactor conditions ($Re = 500,000$) on a representative geometry—featuring both centripetal and centrifugal flow patterns—provide a first-level response for discriminating, in the short term, between predictions from different component codes.

With the computational resources typically available, the most relevant CFD approaches for scale-up objectives generally require strong modeling as-

sumptions regarding the mixing grids. These assumptions have a first-order impact on the flow redistribution dynamics. The access to the computational power of the SELENA cluster within the framework of the Grand Challenges made it possible to lift these assumptions by explicitly integrating the geometry of the mixing grids. The computational domain represents a 3×3 lattice of fuel assemblies over a height corresponding to five grid spans – i.e., four grid crossings – in a bare bundle configuration (without the representation of the end fittings). This high-fidelity simulation, involving 8.6 billion cells, minimizes modeling assumptions and provides a state-of-the-art reference result aimed at supporting scale-up towards component-scale modeling tools.



An overview of CEREA activities with code_saturne

by M. FERRAND – EDF R&D AND CEREA.

The first part of the presentation will provide an overview of CFD modeling activities using code_saturne conducted at CEREA (the joint EDF R&D / ENPC laboratory specializing in atmospheric environmental studies). These activities include the development of a dedicated module for cooling towers, now compatible with the humid atmosphere module. Ongoing research work also enables the modeling of the rain zone through a hybrid stochastic Lagrangian (for rain) / Eulerian RANS (for moist air) approach, accounting for polydispersion and phase interactions.

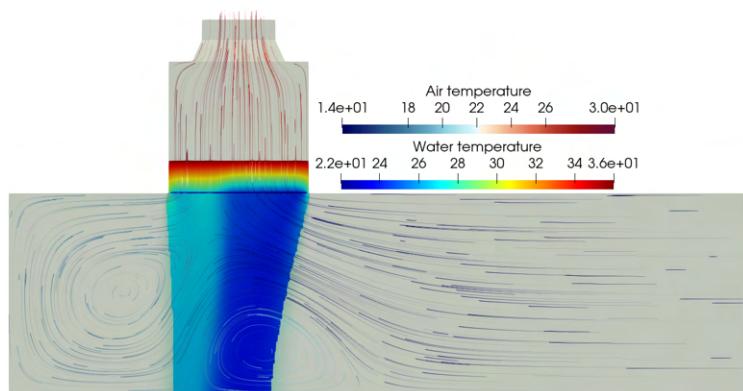
Other recent applications include the use of

code_saturne to build an atmospheric digital twin of the archery range during the 2024 Olympic Games, used for performance and environmental assessments; the modeling of a pressurized jet compared with an experimental campaign conducted by Ineris; and the simulation of pollutant releases on industrial sites under low-wind conditions.

Finally, the presentation will also address the development of advanced turbulence models and numerical schemes, particularly for second-order RANS and LES approaches, as well as the use of point clouds for atmospheric domain meshing.



Real-time forecast website. The forecasts allow the athletes to anticipate coming windy events and appreciate specific recirculation zones on the pitch due to stands, screen, trees and windy corridors.



Validation using the MISTRAL test bench at the Bugey nuclear power plant, owned by EDF.

Atmospheric flow modelling of floating offshore wind turbines: coupling between code_saturne and the aero-hydro-servo-elastic model DIEGO

by A. MATHIEU – EDF R&D AND CEREA.

One of the main challenges in wind turbine modelling is to accurately represent the wake, a region characterized by low wind speed and high turbulent kinetic energy downstream from the rotor. Although difficult to model, wake interactions significantly impact the wind farm performance by reducing energy production and increasing aerodynamic loads for downstream turbines. Consequently, understanding and mitigating wake effects has become a major challenge for the wind industry in recent years.

To this complexity is added the modelling of Floating Offshore Wind Turbines (FOWT), which can move in the six degrees of freedom under the ac-

tion of waves, wind and floater response. The effect of rotor movement on wake is an active and recent research topic in the scientific community.

In this study, we present results from CFD simulations using code_saturne to assess the capabilities of CFD to accurately predict the wake downstream of a FOWT using an oscillating actuator disk model in sway and surge motions. Additionally, we demonstrate the coupling between code_saturne and the aero-hydro-servo-elastic model DIEGO, enabling accurate reproduction of the turbines and floaters dynamics for different atmospheric and wave conditions.



List of posters

| | |
|---|--|
| Cost-Effective Thermal Hydraulics Modelling for Advanced Nuclear Reactor Technologies | by B. LIU, W. WANG, C. MOULINEC, S. ROFLO, E. IYAMABO, C. KATSAMIS & M. CHEVALIER STFC |
| Cost-Effective Modelling for Accident Scenarios of Advanced Nuclear Reactor Systems | by B. LIU, W. WANG, C. MOULINEC, S. ROFLO, E. IYAMABO, C. KATSAMIS & M. CHEVALIER STFC |
| A priori modelling of the temperature variance dissipation transport equation | by C. SANZ SOUAIT, J-F. WALD, S. BENHAMADOUCHE & R. MANCEAU EDF R&D, CNRS |
| An analytical Wall Function for improved Heat Transfer Modelling in <code>code_saturne</code> | by C. KATSAMIS, D. WILSON, E. LYAMABO, T.J. CRAFT & H. LACOIDES UNIVERSITY OF MANCHESTER, EDF R&D UK |
| System monitoring of Flow-Accelerated Corrosion through upscaling of CFD models coupling thermohydraulics and chemistry | by B. CELLE, T. LASSEUR, S. MIMOUNI, G. SIMONINI, S. SHKIRSKIY & F. KANOUI EDF R&D, UNIVERSITÉ PARIS CITÉ |
| <code>code_saturne</code> chez SIL3X | by CONTACT@SIL3X.FR SIL3X |
| Progress of the GPU port of <code>code_saturne</code> | by DEV. TEAM EDF R&D |
| Numerical study of flows in a lid-driven cavity square and circular sections | by J. GAUTHIER, R. MANCEAU, S. BENHAMADOUCHE ET AL. EDF R&D - MFEE |
| Towards an optimal agrivoltaic design for plant protection | by J. VERNIER, B. AMIOT ET AL. EDF R&D, CEREIA, INRAE, EDF POWER SOLUTIONS |
| Improvement of a film condensation model for simulating the thermodynamics of two-phase thermosyphon loops with the <code>neptune_cfd</code> code - application to passive safety systems | by E. ROLLIER, N. MÉRIGOUX, T. PRUSEK, M. MONTOUT & C. COLIN EDF R&D - IMFT |
| Modelling of the Dynamic Degradation of Breakwaters Using CFD-DEM | by M. BARCET, W. BENGUIGUI, T. BONOMETTI & P. FEDE EDF R&D, IMFT |
| CFD modeling of two-phase cross-flows in tube bundles | by F. BELTRAN, T. BONOMETTI, C. COLIN, W. BENGUIGUI & N. MERIGOUX EDF R&D, IMFT |
| Stochastic Lagrangian Modeling of Wet Cooling Towers Rain Zones | by L. VOLLE, M. FERRAND, P. MASSIN, C. DUQUENNOY & A. BARBE EDF R&D, CEREIA, EDF DTG |

Solitary wave simulations at CIH

by M. LÉOPOLD
CIH

Two-phase flow simulations at CIH

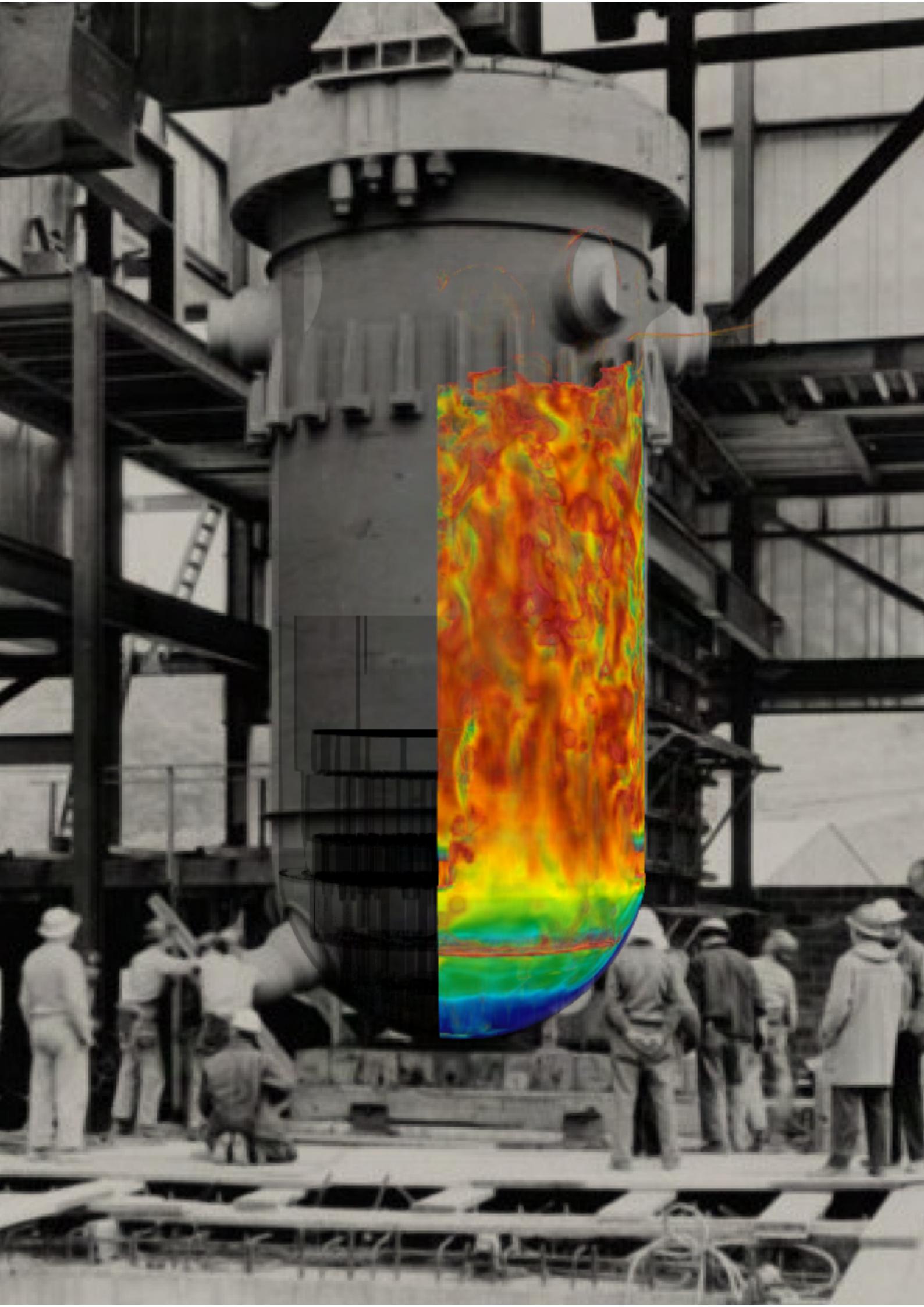
by R. BISSIÈRE & V.
LOIZEAU
CIH

Massively Parallel Workflows for Image-based Modelling

by H. JONES, C. MOULINEC,
J. LE HOUX & S. ROLFO
SCIENTIFIC COMPUTING,
STFC

Developing a multiscale continuum framework for exascale computing of fluids with
microstructure

by C. TSIGGINOS & C.
MOULINEC
SCIENTIFIC COMPUTING,
STFC



| | | |
|---|--|---|
| 09h00–09h30 | | Welcome / Breakfast |
| 09h30 | Introduction. | <i>P. Charles (EDF R&D)</i> |
| 09h40 | New features overview in code_saturne v9.0. | <i>Dev. Team (EDF R&D)</i> |
| 10h10 | Presentation of Simvia, an EDF subsidiary. | <i>F. Leray (Simvia)</i> |
| Presentations – morning session (part 1) | | |
| 10h15 | An assessment of algebraic and differential Reynolds-Stress Models for a highly-bent serpentine aircraft intake. | <i>S. Hanrahan (Melbourne University)</i> |
| 10h35 | Modelling and simulation of a three-phase stirred tank. Modeling of sodium spray combustion with neptune_cfd. | <i>R. Ansart (IMFT) N. Kirov, O. Simonin (IMFT)</i> |
| 11h05–11h30 | | Break |
| Presentations – morning session (part 2) | | |
| 11h30 | Study of vortex intrusion phenomena in dead legs. | <i>JF. Wald, J. Uribe (EDF R&D)</i> |
| 11h50 | Simulation of solar receivers for direct steam generation using neptune_cfd coupling. | <i>I. Aguilera (CNRS PROMES)</i> |
| 12h10 | Advancing code_saturne for hydraulics: validation and application to industrial problems. | <i>P. Asproulis, Y. Eude (RENUDA)</i> |
| 12h30–14h00 | | Lunch / Poster session |
| Presentations – afternoon session | | |
| 14h00 | A review of code_saturne developments in STFC UKRI. | <i>S. Rolfo (STFC UKRI)</i> |
| 14h20 | Overview of neptune_cfd simulations for reduced-scale filtering experiments. | <i>A. Doradoux (SIREHNA)</i> |
| 14h40 | Simulation of breaking wave loads on a wind turbine foundation in the coastal zone with code_saturne. | <i>M. Benoît (EDF R&D)</i> |
| 15h00–15h25 | | Break |
| Flash session | | |
| 15h25 | Implementation of a log-normal modeling in neptune_cfd for polydispersed flows. | <i>N. Cailler (EDF R&D)</i> |
| | Presentation of high-fidelity (WR-LES) simulation with code_saturne of turbulent flow under mixed convection regime within a heated rod bundle. | <i>V. Duffal (EDF R&D)</i> |
| | Go Viking – Development and use of EDF CFD tools for vibration prediction in axial and crossflow conditions. | <i>W. Benguigui (EDF R&D)</i> |
| | On-the-fly construction of a ROM during code_saturne simulations: an urban boundary layer example. | <i>K. Kuznetov (GRASP)</i> |
| | System monitoring of flow-accelerated corrosion through upscaling of CFD models coupling thermohydraulics and chemistry. | <i>B. Cellé (EDF R&D)</i> |
| | Grand Challenges SELENA: Determination of inter-assembly flow redistributions with explicit modeling of mixing grids at large scale. | <i>R. Ceyrolle (EDF DT)</i> |
| 16h10 | An overview of CEREA activities with code_saturne. Atmospheric flow modelling of floating offshore wind turbines: coupling between code_saturne and the aero-hydro-servo-elastic model DIEGO. | <i>M. Ferrand, A. Mathieu (CEREA, EDF R&D)</i> |
| 16h45 | Prospects in code_saturne & neptune_cfd. | <i>Dev. Team (EDF R&D)</i> |
| 17h00 | | Closure |