Code_Saturne user meeting

Thursday –April 2nd, 2015 – EDF Lab Chatou

8:30	**Welcome – Breakfast**	
9:00	Foreword	M. Ferrand *Head of Code_Saturne project*
9:05	Invited Lecture	H. Cordier *Head of EDF SEPTEN THL Group*
9:20	Latest news and prospects in *Code_Saturne* from 3.0 to 4.0	Y. Fournier *EDF R&D - MFEE*
9:40	Underground flow in *Code_Saturne*, applied to storage of nuclear waste	L. Le Tarnec and V. Stobiac *EDF R&D LNHE*
10:00	Porous modeling of AGRs pod boilers with *Code_Saturne*	J. Deamicis *EDF R&D UK centre*
10:20	**Break**	
10:50	RANS computations of a quasi-axial flow in an in-line tube bundle	B. Claudet *EDF R&D and EDF SEPTEN*
11:10	Numerical Analysis of Blood Flow in a Ventricular Assist Device Using High Performance Computing	M. Behbahani *Aachen University*
11:40	LES of a simplified HVAC system used for aero-acoustic predictions	S. Rolfo *Daresbury Lab.- UK*
12:00	**Lunch**	
13:40	Applications of *Code_Saturne* to Critical Multi-Physics Industrial Problems	N. Tonello and B. Angel *RENUDA*
14:00	Experimental and computational analysis of olive residues biomass-fired grates and modelling fly ash particles deposition on olive waste fired boiler	M.A. Díaz and P.J. Leal *GESTAMP ENERGY SOLUTIONS R&D Department*
14:20	Implementation of *Code_Saturne* 3.3 for atmospheric dispersion studies	F. Cohn *NUMTECH*
14:40	**Break - Poster session**	
15:30	Electric arc / plasma coupling: application to GTAW welding	D. Borel *EDF R&D MRI*
15:50	CFD simulations temperature behaviour of bitumen mud with *Code_Saturne* V4.0	G. Espinasse *BERTIN TECHNOLOGIE*
16:10	Automotive fan system simulation with *Code_Saturne*	Manuel Henner *Valeo Thermal Systems*
16:30	**Closure**	**F. Baron** ***Head of MFEE department***
	End of the day	

LIST OF POSTERS

Modelling a Low-Speed Centrifugal Compressor with *Code_Saturne*	A. Heffron *Queen Mary, University of London*
Péclet robust CDO schemes for advection-diffusion on polyhedral meshes	P. Cantin *EDF R&D – MFEE*
Modelling fly ash particles deposition on olive waste fired boiler Experimental and computational analysis of olive residues biomass-fired grates	M.A. Díaz, P.J. Leal, C. Yin and A. J. Gámez *Gestamp Energy Solutions R&D Department – Institute of Energy Technology, Aalborg University - Universidad de Cádiz – Escuela Superior de Ingeniería*
Adaptive Wall Treatment for the Elliptic Blending Reynolds Stress Model	J.-F. Wald *EDF R&D – MFEE*
Simulation of cavitating flows in hydraulic machineries using an homogeneous mixture model	B. De Laage De Meux *EDF R&D – MFEE*
3-D Atmospheric Infrared Radiative Transfer in *Code_Saturne* to Simulate Fog Formation	L. Makke *EDF R&D – MFEE*
Modelling of natural draft wet cooling tower behaviour in realistic atmospheric conditions using *Code_Saturne*	A. Chahine, B. Carissimo *EDF R&D – MFEE*
Influence of Potential Sleeve Leakage on Gas Mixing in AGR Fuel Assemblies	Juan Uribe, Charles Moulinec, Jim Gotts, Bing Xu, David R. Emerson, D.R. Laurence *EDF R&D UK Centre; STFC Daresbury Laboratory; EDF Energy, Nuclear Generation; EDF R&D, MFEE; The University of Manchester, School of MACE*

CFD and safety demonstration for Nuclear Power Plants – evolution or revolution?

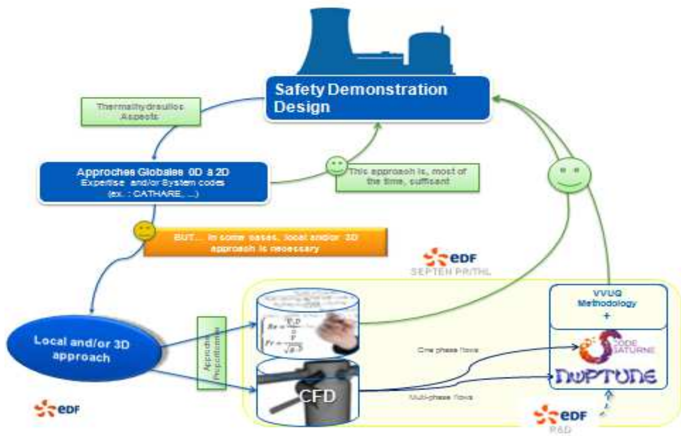
H. Cordier, A. Barthet, S. Bellet, S. Delépine, F. Duplat, B. Gaudron, W. Hay, F. Lelong, J.P. Simoneau, C. Terrier, A. Zanchetti - *EDF SEPTEN*

After several years of development, CFD now plays an integral role in the safety demonstration process of nuclear power plants.

This role has two main prerequisites: firstly, the user must have full confidence in the quality of the CFD calculations produced; secondly, the safety authorities must be equally convinced by the CFD calculations.

For the above to be possible, it is fundamental to use a pragmatic and proportionate method in order to maintain the balance between quality of results and time needed to obtain them. At EDF, this methodology can be summarized in 4 letters: V V U Q (Verification, Validation and Uncertainty Quantification).

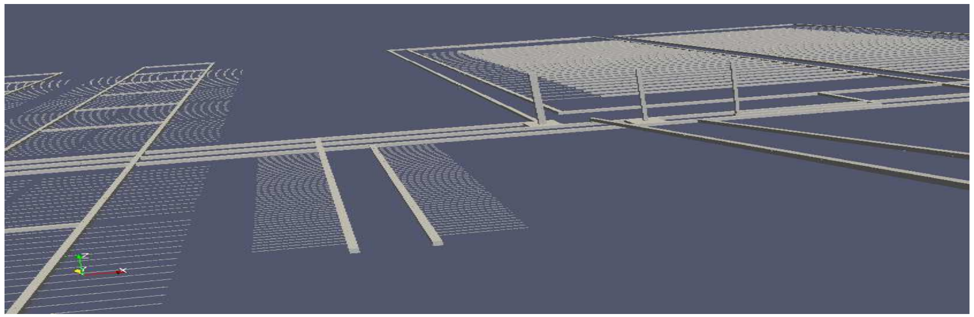
The purpose of this presentation is to summarize how this methodology is applied at EDF for nuclear safety studies and how it can help the wider CFD community to improve its capability to convince with CFD calculations.



Underground flow in *Code_Saturne*, applied to storage of nuclear waste

L. Le Tarnec and V. Stobiac – *EDF R&D LNHE*

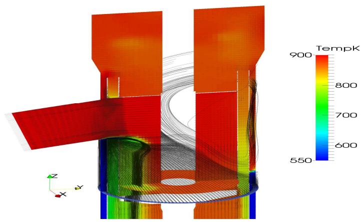
In the framework of safety assessment of radioactive waste repositories, it is necessary to model underground flow and transport on very complex geometries. A specific module was developed in *Code_Saturne*, able to treat standard hydrogeology models, in order to take benefits of current and future HPC optimizations of the software. The module is undergoing tests, aiming at demonstrate that specificities encountered in underground environment, especially severe discontinuities of physical properties, are correctly handled by *Code_Saturne*.



Porous modelling of AGRs pod boilers with *Code_Saturne*

J. Deamicis – *EDF R&D UK centre*

A porous model has been developed to represent the thermal-hydraulics of CO_2 flowing in the boilers of AGRs (Advanced Gas Reactors).To investigate the effect of the gas flow on the performance of the boiler, and to provide the correct thermal boundary condition to the gas side, the model is coupled with NUMEL, a 1D code developed by EDF Energy Generation that solves the mass, momentum and energy conservation equations of the water side. For each tube a NUMEL calculation is performed. The model is then coupled with SYRTHES to evaluate the temperature distribution in the main structural components of the boiler.



RANS computations of a quasi-axial flow in an in-line tube bundle

B. Claudet – *EDF R&D and EDF SEPTEN*

The flow behavior through tube bundles is of major interest in several nuclear engineering applications; fuel assemblies, steam generators… It is well known that performing fine CFD simulations for a whole core or a complete steam generator is not affordable today as it will lead to deal with too large computational meshes. The authors believe that this will remain the case for several decades. This pushes us to use much coarser approaches such us the one based on porous media (among others) in which one needs to impose pressure loss coefficients in the sub-channels to obtain the right mass flow rates. Unfortunately, the flow is not straight and can be oblique for several reasons. There are no correlations in the literature which give the pressure loss coefficient for low angles (quasi-axial flows) to the author's knowledge. The presentation which will be given is the very first step of a program which aims at giving correlations as a function of the angles (up to 10°), the Reynolds number and the pitch to diameter ration.

In the present study, one fixes the pitch to diameter ratio to a typical value found in a fuel assembly and the axial pressure gradient. The a posteriori obtained Reynolds numbers, based on the bulk velocity and the hydraulic diameter, are in the range [50.000, 70.000]. The sensitivity to the RANS turbulence model (standard k-ε, k-ω SST, SSG, EB-RSM), to the number of tubes (1x1, 2x2, 3x3, 4x4, 5x5 with periodic boundary conditions in the three directions) and to the mesh refinement are first studied for the axial flow and an inclined flow obtained from the first one by adding a pressure gradient in one of the two transverse directions. Then, the EB-RSM model is used with the appropriate mesh and tube configuration to obtain the pressure drop coefficient for several angles. LES computations might be available for *Code_Saturne* 2015 user meeting.

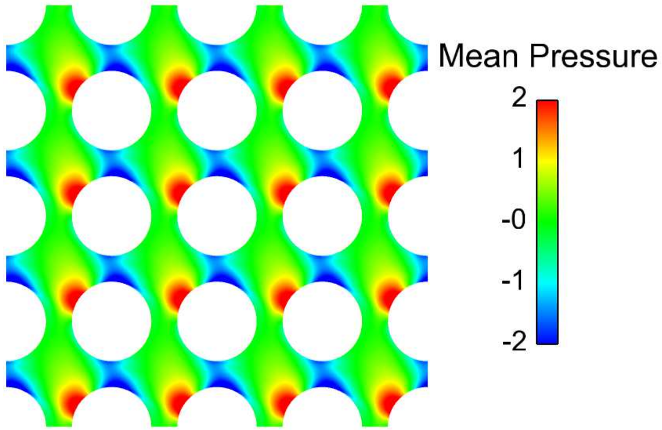


Figure 1: Typical mean pressure for an inclined flow in a 4x4 configuration (same pressure gradient in the axial and one of the two transverse directions)

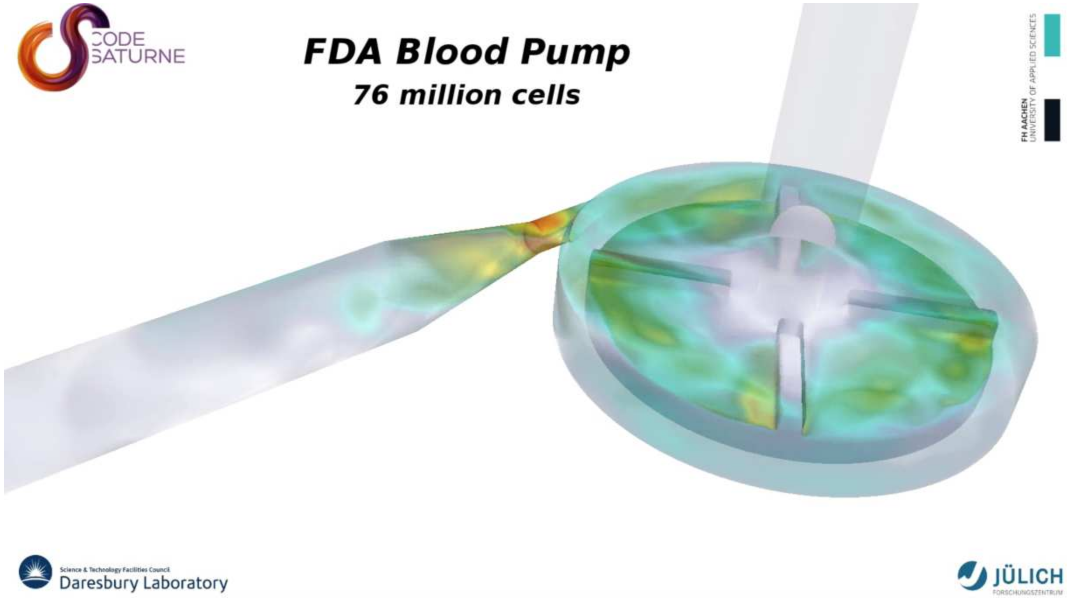
Numerical Analysis of Blood Flow in a Ventricular Assist Device Using High Performance Computing

M. Behbahani – *Aachen University*

Ventricular assist devices (VADs) provide long- and short-term support to chronically ill patients suffering from heart disease, which is the most common fatal disease in developed countries. These devices are expected to match the remarkable functionality of the natural heart, which makes their design a very challenging task. Blood pumps, the principal component of the VADs, must operate over a wide range of flow rates and pressure heads and minimize the damage to blood cells in the process. Mathematical models and computational fluid dynamics (CFD) have recently emerged as powerful design tools in this context.

For an international benchmark issued by the FDA (Food and Drug Administration) a study was undertaken using a 76 million element mesh which was produced in Salome. Using the flow solver *Code_Saturne* simulations could be performed for six test cases six test with different volume flow rates, pump speeds, and Reynolds numbers ranging from 210,000 to 293,000. *Code_Saturne* showed good scalability on the BlueGene/P and BlueGene/Q supercomputers and computations were typically run on 8,192 to 32,768 processors.

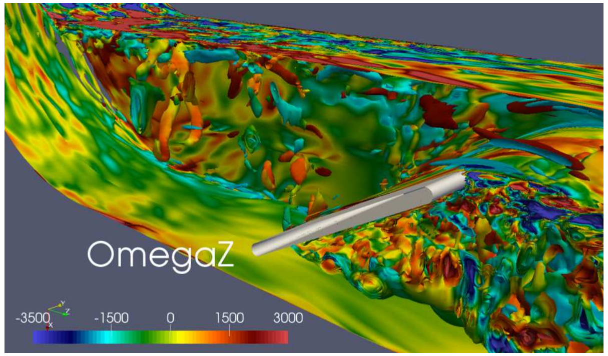
This study is a step towards the establishment of a reliable method to accurately predict blood flow and blood damage in fast rotating biomedical devices.



LES of a simplified HVAC system used for aero-acoustic predictions

S. Rolfo – *Daresbury Lab.- UK*

This work will present the flow inside a simplified Heat and Ventilation Air Conditions (HVAC) system. One of the main challenges with HVAC system is the noise generated by the turbulent flow, particularly in the case of detachments generated by obstacles inserted along the stream-wise direction of the flow. The current geometry is composed by a square bent duct with a flap positioned after the bend. The pipe is after exiting in a plenum forming a jet with a square cross section.



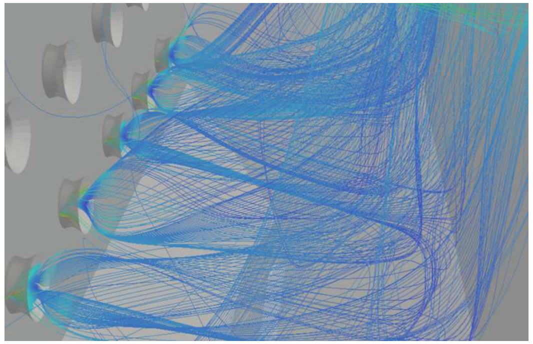
Despite the relatively simple geometrical definition, the flow presents some very complicated pattern as visible in the figure. A large recirculation area is clearly visible on the topside of the geometry just downstream the bend. These very large turbulent and very energetic structures are impinging on the top surface of the flap, creating very elongated structures on the top part of the duct. A large wake is also visible behind the flap and counter-rotating vortices can be identified both at the top and the bottom of the recirculation area.

In the presentation a complete description of the flow features will be described and identification of the noise sources will also be presented.

Applications of *Code_Saturne* to Critical Multi-Physics Industrial Problems

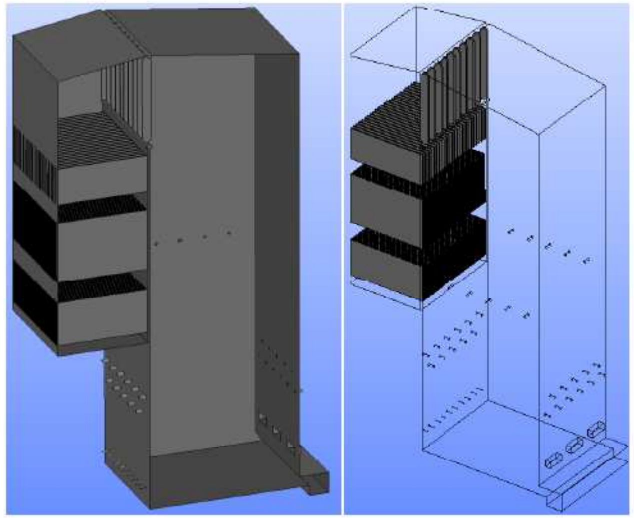
N. Tonello and B. Angel – *RENUDA*

CFD specialists Renuda will present an overview of some their activities using *Code_Saturne* and will then focus on three illustrative industrial applications making use of different *Code_Saturne* functionalities: liquid flow for the optimization of a waste water purification system in order to minimize system pressure drop and maximize flow homogeneity; multi-species flow for the conjugate heat transfer in an exhaust gas heat recovery unit and assessing the efficiency of different designs for burnt gas entrainment and system pressure drop; multiphase, multi-species flow and solid particulate combustion to assess the efficiency of a combined coal and biomass power plant under different operating conditions.



Experimental and computational analysis of olive residues biomass-fired grates and modeling fly ash particles deposition on olive waste fired boiler

M.A. Díaz and P.J. Leal – *GESTAMP ENERGY SOLUTIONS R&D Department*

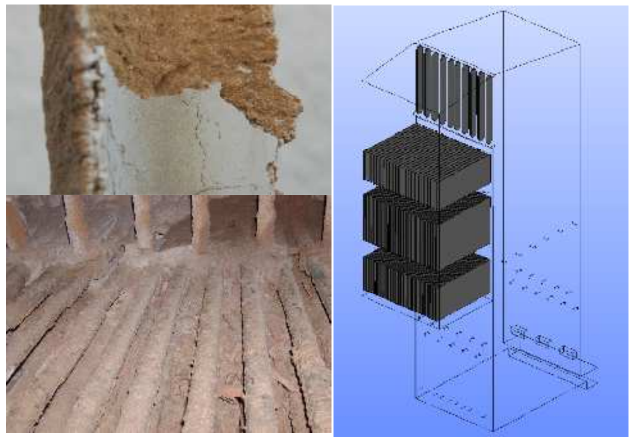


This presentation concerns the performed works, foreseen planned simulations and full-scale validation process of an olive-residue fired grate model for Biomass power generation plants in development as a result of a project of Gestamp Energy Solutions R&D Department and which is also focused to be the PhD dissertation of the authors at the University of Cádiz and supported by the University of Aalborg.

This model will allow biomass boiler design optimization in engineering stages as well as biomass boiler performance predictions considering the complexities of the olive wastes fuels (chemical compositions and particularities according to their ashes behavior) in order to maximize facilities dispatchabilities.

The model will be validated in existing Gestamp Renewables' biomass power plants burning olive residues through on-site combustion species measures with a temperature-controlled measurement probe specifically designed for either this purposes and deposition processes measurements.

The presentation also exposes the proposal of combining existing fly ash particle deposition computational models on heat transfer surfaces of biomass firing boilers. Ash particles of olive waste have high chlorine and alkali (K+Na) contents which give them a relative low melting temperature promoting a high level of fouling in a short lifetime.



The computational model integrates different deposition mechanisms and shedding. The sticky behaviour of ash is also modeled based on its chemical composition and temperature. A small boundary deposition model over a few tubes is performed with a high mesh density and once validated in Gestamp Renewables' power plants, it will be exported to a full-scale model. Validation will be performed by a special probe inserted in the boiler controlling the metal temperature.

Implementation of *Code_Saturne* 3.3 for atmospheric dispersion studies

F. Cohn – *NUMTECH*

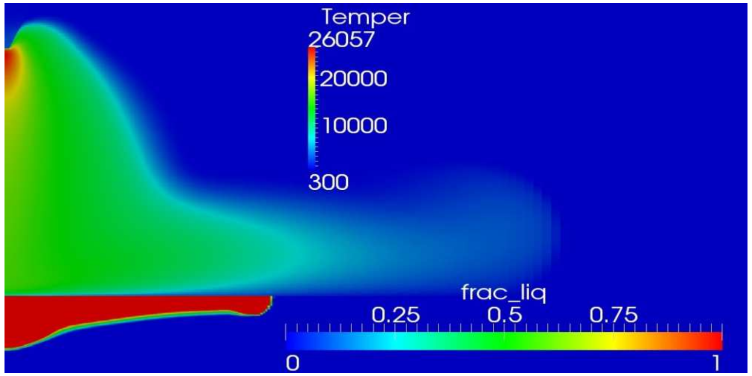
NUMTECH, a specialist consultancy in atmospheric dispersions of emissions linked with human activities recently implement *Code_Saturne* in their tools to achieve urban and industrial studies. This 3D model meets with specific needs, taking into account with much accuracy buildings and obstacles (sonic walls, windbreaks, hedges, pipes and complex geometries).
In this presentation, they will first introduce the methodology used to achieve industrial studies in compliance with regulatory requirements. And then compared the results will other models.

The presentation will finished with an overview of their activities in CFD.

Electric arc / plasma coupling: application to GTAW welding

D. Borel – *EDF R&D MRI*

In this presentation, we expose a physical and numerical modelling of the gas-tungsten arc welding (GTAW). We propose a coupling between two existing models developed at EDF: the electric arc and weld pool models. The aim of this work is to demonstrate that is possible to take into account plasma physics to improve welding simulations. We focus on the interfacial conditions between these two instances of *Code_Saturne* (3.2.4 version).



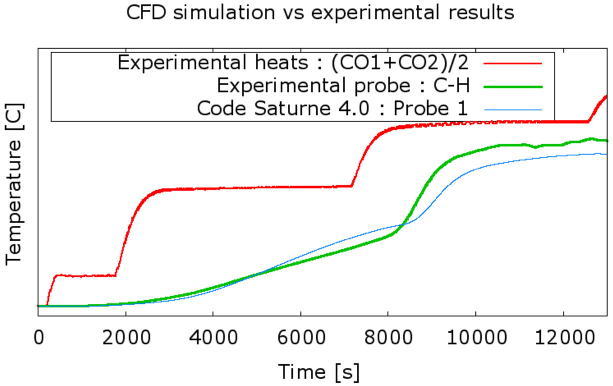
CFD simulations temperature behaviour of bitumen mud with *Code_Saturne* V4.0

G. Espinasse – *BERTIN TECHNOLOGIE*

Bitumen is a derivative of crude oil. Radioactive wastes in the form of dewatered sludge are incorporated in the bitumen. Comparing simulations CFD with recordings during time of several thermocouples immersed in 2kg of bitumen pot which is heated.

With a numerical model, we want to reproduce the experimental temperature behavior of bituminous sludge.

At room temperature, the transfers are very limited and heat exchange takes place by conduction through the material. The density change is significant from a threshold temperature rise. The presence of a density gradient is the establishment condition of a convection loop.



Automotive fan system simulation with *Code_Saturne*

Manuel Henner – *Valeo Thermal Systems*

Valeo has been doing numerical simulation for its components for many years, and it is now considered as a strong asset in the design process: faster development cycle, deep analyze of the physics, reduced costs, etc.

The ability of *Code_Saturne* to perform aerodynamic performance predictions is therefore investigated for the case of the fan system. The objective is to use this open source code with unlimited number of simultaneous simulations in a large Design of Experiment for the sake of optimization. Several thousand of cooling module simulations should be performed in the collaborative project named "Pepito" funded by the French National Research Agency, and investigations are therefore done to assess the code performances in terms of accuracy and CPU time.

Preliminary results will be presented on simple cases and compared to previous results with a different code. Predictions on an actual case are also compared to experimental results and show a very agreement.