Code_Saturne and Salome_CFD user meeting

Program

WebSite *BugTracker* *Forum*

8:00	**Welcome – Breakfast**	
9:00	Foreword	M. FERRAND *Code_Saturne* project leader
9:05	Introduction	D. BANNER Head of simulation program
9:20	Latest news and prospects in *Code_Saturne* and Salome_CFD	*Code_Saturne* DEV. TEAM EDF R&D - MFEE
9:50	Contribution of CFD in the study of EPR Spent Fuel Pool	R. CAMY EDF SEPTEN
10:10	Pool fire modelling for nuclear safety assessment – an industrial case	A. AMOKRANE EDF R&D - MFEE
10:30	**Break**	
11:00	Modelling of Heat Transfer Impairment in fuel pins of Advanced Gas-cooled Reactors	D. CALVO EDF R&D UK Centre
11:20	*Code_Saturne* in China: 2016-2017 activity overview	T. XU EDF China R&D Center
11:40	Validation of EDF's CFD tool *Code_Saturne* on a 40° bend configuration at high Reynolds numbers	N. LANCIAL EDF R&D - PRISME
12:00	**Lunch**	
13:30	Multi-Physics development and applications of *Code_Saturne*	N. TONELLO Renuda
13:50	Large Eddy Simulations of a Compressor Operating at Low Reynolds Number	O. WILSBY Cambridge Uni.
14:10	*Code_Saturne*, a useful tool to compute atmospheric flows: some recent case studies	L. MAKKE & M. NIBART ARIA Technologies
14:30	**Break - Poster and live demonstration session**	
15:30	Heat exchanger optimization using *Code_Saturne*	F. MASTRIPPOLITO CEA Tech LITEN, LMFA ECL Lyon, Valeo & EDF
15:50	Quantification of the Discontinuity of the Temperature Variance Dissipation at a Fluid-solid Interface: Wall-resolved LES of Turbulent Channel Flow with Conjugate Heat Transfer	C. FLAGEUL Josef Stefan Institute
16:10	CFD study of flow redistributions in PWR fuel assemblies	R. DENÈFLE EDF R&D - MFEE
16:30	Closure	J.P. CHABARD Scientific Director of EDF R&D
	Closing Reception	

Contribution of CFD in the study of EPR Spent Fuel Pool

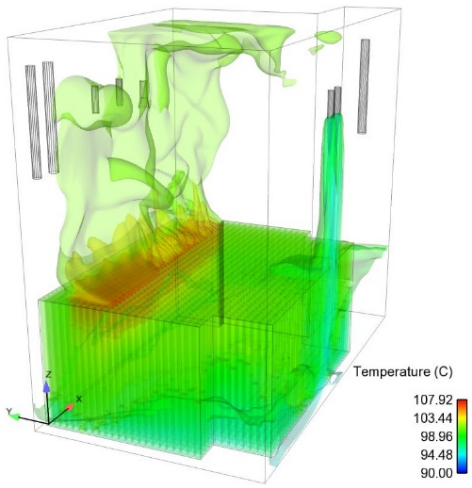
by R. Camy – EDF SEPTEN

The role of the Spent Fuel Pool (SFP) is to ensure the integrity of the fuel unloaded from the reactor core while performing biological shielding. In the event of a loss of the Fuel Pool Cooling and Purification System (FPCPS), the temperature in the pool would gradually increase until change of state occurs. Assuming that a make-up compensates the loss of water transformed into steam, the pool would reach a statistically stationary state where heat from the fuels is balanced by sensible heat of the make-up and latent heat of vaporization.

This presentation focuses on the SFP of the EPR (a third generation pressurized water reactor) in a configuration with Mixed Oxide (MOX) fuel. The modelling of the thermalhydraulics in the SFP relies on *Code_Saturne*, an open source 3-D Computational Fluid Dynamics (CFD) code. This modelling allows predicting the natural circulation flow that sets up in the SFP and hence the homogenization of the temperatures between the source term (the fuels) and the sink term (mainly the area where vaporization occurs).

The use of a single-phase CFD code leads to model the vaporization with a clipping at saturation temperature defined as a function of static pressure only. The very small scales of the geometric details of the racks and the fuels stored in them lead to use a porous approach with imposed head loss coefficients.

In this presentation highlights are put on the methodology and the modelling assumptions. A special care is given to the origins of some of the sensitivities done to demonstrate that the configuration under consideration is conservative. This work fits into the usual approach "Phenomena Identification and Ranking Table (PIRT), Verification, Validation and Uncertainty Quantification (VVUQ)" in use at EDF for CFD safety studies.

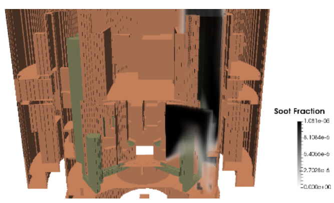


Pool fire modelling for nuclear safety assessment – an industrial case.

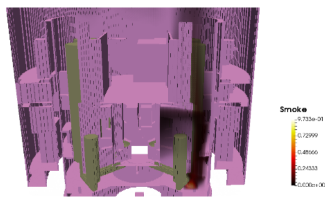
by A. Amokrane, B. Sapa & F. Nmira – EDF R&D MFEE

The need to use CFD for fire safety applications has been expressed over the ten past years as an answer to the inherent limitations of two-zone models, used usually in fire safety assessment studies. This need is specially accentuated by the growing necessity to undertake simulations of fires in large volumes and complex geometries.

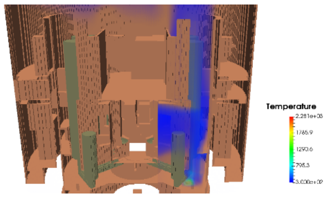
Physics of fire developments, in *Code_Saturne*, have been initiated since 2008. After having implemented, in the code, the necessary developments to enable the modelling of fires, the code is now able to accurately simulate well ventilated compartment fires, whose characteristic scales are of the order of few hundreds of cubic meters.



Code_Saturne also benefits from an ongoing dedicated process of verification and validation.



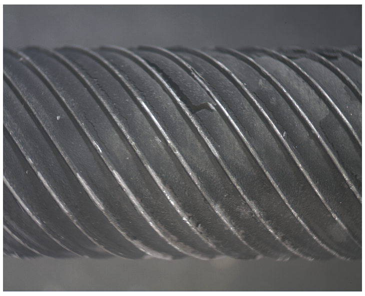
The main aim of the present study in to evaluate the ability of *Code_Saturne* to undertake an industrial scale fire study. We have retained in this case a pool fire scenario originating from a leakage in the Reactor Coolant Pump (RCP) of the reactor building of EFR FA3 power plant. This scenario is similar to the one previously used by SEPTEN in the past, for a study done with the code FDS (Fire Dynamic Simulator) which is the code of reference of the fire safety community. The results of the two codes were compared to assess their coherence.



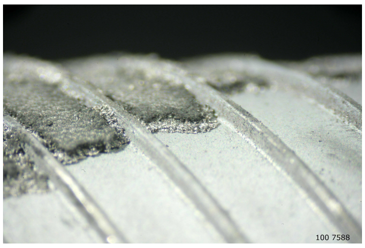
Modelling of Heat Transfer Impairment in fuel pins of Advanced Gas-cooled Reactors

by D. Calvo – EDF R&D UK Centre

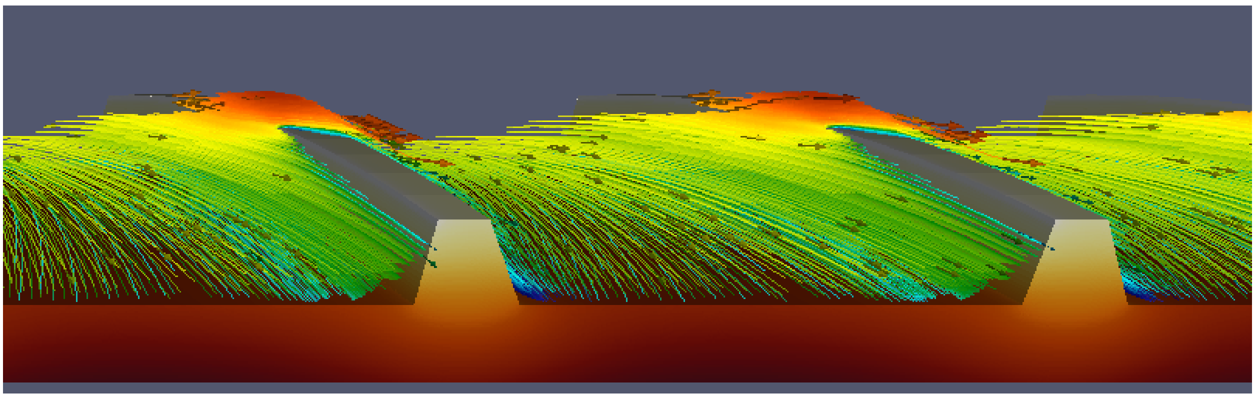
The Advanced Gas Cooled reactors in the UK use fuel pins arranged in assemblies of 36 pins each. These pins have ribs machined in a helical pattern to optimise the heat transfer and reduce the extra drag induced.



The pins are exposed to high temperatures in a carbon dioxide atmosphere. This leads to chemical reactions resulting in deposits which have an effect on the heat transfer performance. Two mechanisms are responsible. Firstly, the deposit presents a higher thermal resistance and secondly it modifies the surface geometry leading to a change in the heat transfer coefficient of the wall. The Heat Transfer Impairment (HTI) resulting from the deposition shortens the fuel life due to increase in temperature at the steel can surface.



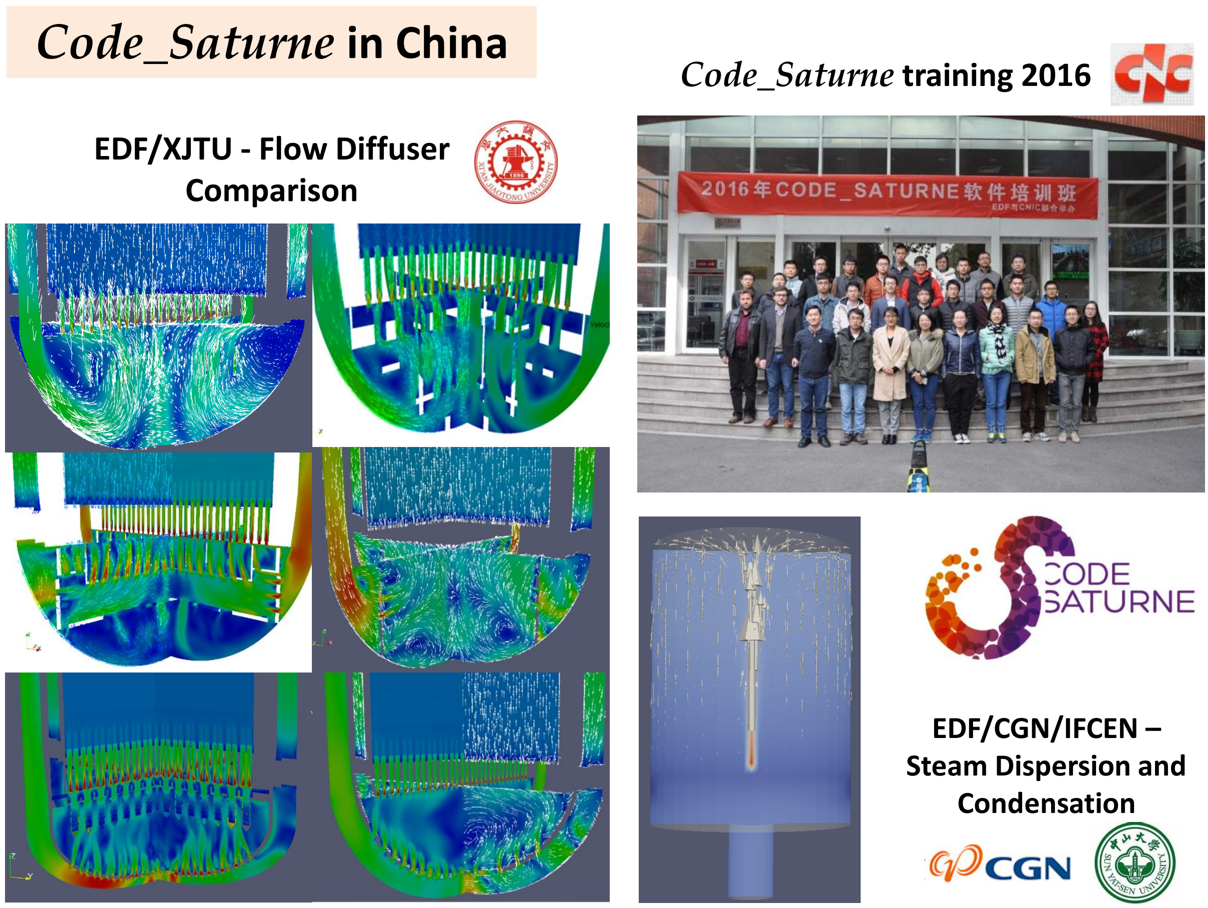
EDF Energy R&D UK Centre in collaboration with EDF Energy Generation and the University of Manchester has developed a model using *Code_Saturne* for studying this effect. The objective of the study is to obtain information on how the HTI varies depending of the deposition characteristics which are highly variable. The model follows a 2D periodic approach based on the helical multi-start geometry of the AGR fuel pins. It makes use of one of the state-of-the-art capabilities of *Code_Saturne* such as Internal Coupling of fluid-solid-solid with different thermal conductivities. The fluid model has been validated with experiments conducted at the University of Manchester specifically designed for this purpose. A sensitivity study has been conducted depending on deposit thickness and conductivity for different mass flow.



Code_Saturne in China: 2016-2017 activity overview

by T. Xu – EDF China R&D Center

Since June 2011, EDF R&D China Center works on popularize *Code_Saturne* in China through research cooperation and technical exchange with Chinese partners, trainings, presentations at international conferences, and so on. During the past six years, *Code_Saturne* attracts attention from both industrial and academic actors, and it is recognized as an industrially-proved, open-source, HPC-based software. Worked jointly with experts located in other EDF R&D centers in France, Germany, UK and Poland, EDF China R&D Center aims at strengthen cooperation between EDF and the Chinese scientific and industrial R&D actors in the energy sector, relying on numerical simulation capability to provide R&D support to EDF and its partners in China. The objective of this presentation is to illustrate the *Code_Saturne* activities in China during the past one year in link with local partners, including CGN, CNNC, XiAn Jiaotong Univ. and IFCEN. *Code_Saturne* has been used more and more at nuclear domain in China, as an important technical tool to deal with single-phase flow simulation from new reactor design, sever accident to environmental impact assessment. Its high performance computing (HPC) feature has also been well noticed via its implementation on Chinese HPC system. In the framework of EDF software valorisation project, the first *Code_Saturne* public training was organized in collaboration with the CAS-CNIC (Chinese Academy of Science, Computer Network and Information Center) in Beijing from 1^{st} to 4^{th} November, 2016. The training gathered together 24 participants from both academic institutes (CAS, XJTU, SJTU ...) and industrial fields (CGN, CNNC, SNPTC). In general, this was an opportunity to demonstrate the advanced simulation capabilities of the EDF tools and the associated expertise. With the good feedback, the second public training will be held on from 29th August to 1st September 2017 at XiAn, organized by EDF R&D China and XiAn Jiaotong University.



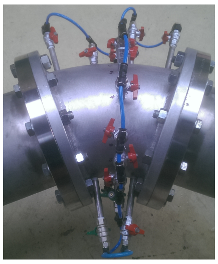
Validation of EDF's CFD tool *Code_Saturne* on a 40° bend configuration at high Reynolds numbers

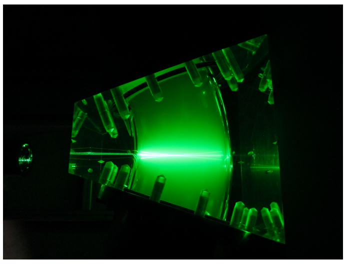
by N. Lancial, H. Gamel, E. Le Coupanec, A. Briant, J.-M. Favennec, S. Bellet & G. Lebreton– EDF R&D - PRISME, MFEE - EDF SEPTEN

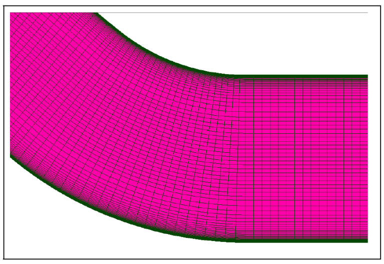
The flowrate in the Reactor Coolant System (RCS) of EDF's Pressurized Water Reactors (PWRs) in France is currently measured by performing an enthalpy balance between the primary and the secondary systems. Alternatively, the flowrate could be deduced from a differential pressure in a 40° bend between the intrados and extrados of the elbow, taking into account that the RCS flowrate is basically proportional to the square root of this differential pressure. In this context, a $\frac{1}{4}$ scale experimental flow test loop (EVEREST) has been set up at EDF Lab Chatou in order to validate EDF's CFD tool, *Code_Saturne*. EVEREST mock-up consists in a circular cross-section with a 40° bend between long straight sections upstream and downstream of the elbow. Pressure taps are located in a so-called "reference section" at 22.5° from the elbow inlet. Velocity profiles are measured with Laser Doppler Velocimetry (LDV).

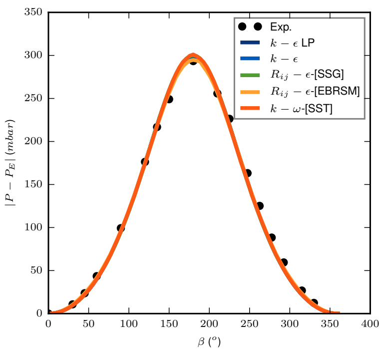
Investigations focus on a range of flowrates corresponding to Reynolds numbers from 8.71×10^5 to 1.48^6. Numerical simulations are performed with the EDF in-house and open-source CFD tool *Code_Saturne*. Simulations are carried out using several RANS turbulence models: two high Reynolds number turbulence models, namely the $k - \varepsilon$ with a linear production (LP) and the $R_{ij} - SSG$ with wall functions, one low Reynolds number model with a second moment closure, the EB-RSM and finally the $k - \omega$ SST. Quantitative and qualitative comparisons are performed.

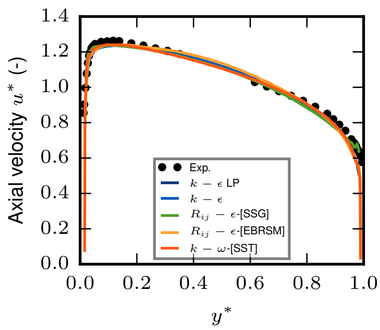
This validation exercise shows the ability of *Code_Saturne* to predict pressure and velocity fields with an adequate accuracy. The flow is well represented by all the models, notably secondary flows generated in the elbow, namely Dean vortices, are qualitatively observed.

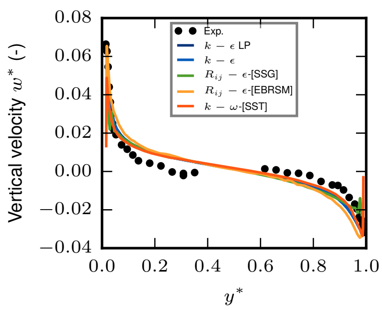










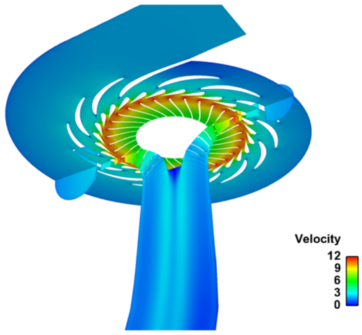


Multi-Physics Development and Applications of *Code_Saturne*

by N. Tonello – RENUDA

As part of its work with industrial clients and research with EDF R&D, Renuda has been developing and applying *Code_Saturne* and SALOME to model multi-physics problems with applications relevant within and outside the nuclear energy industry, including hydraulic energy, consumer product manufacturing, waste water treatment and the automotive sector.

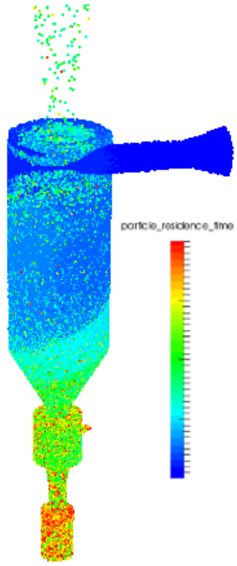
The newly released turbomachinery module has been used for a variety of applications. The research work carried out with EDF R&D on the Francis turbine validations as part of the Francis99 series of workshop will be presented, as well as further validations and HPC scale up studies of *Code_Saturne* performed by Renuda on rotating mixers.



Francis turbine: contours of velocity magnitude.

For two-phase flow, a model based on the drift flux Eulerian representation, making it possible to compute the flow, humidity and temperature in cooling towers was implemented in *Code_Saturne* collaboratively with EDF R&D. The development and application of this model on a representative tower will be presented.

The Lagrangian model was also used extensively in the context of waste water treatment. The presentation will discuss the modelling of hydrocyclones which was carried out with *Code_Saturne*.



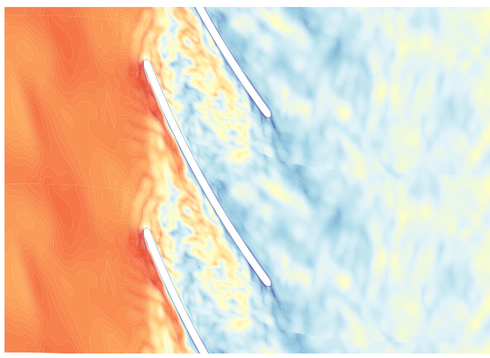
Hydrocyclone: particle residence time.

Large Eddy Simulations of a Compressor Operating at Low Reynolds Number

by O. Wilsby – Cambridge Uni.

Large Eddy Simulations of an axial compressor operating at low Reynolds number have been conducted. A mesh dependency study with and without sub-grid scale models has been completed and the investigation of different time steps has also been studied.

Comparisons are made with a coarser full annular case, with mesh rotation to validate the modelling of a single passage frozen mesh approach. The results are also post-processed to give an understanding of the acoustic sources present in the compressor. These results are then compared with steady RANS calculations performed with open source solver openFoam in order to validate simple semi-empirical models for estimating noise due to trailing edge and rotor stator interaction. The simulations are to be validated experimentally using hot wire anemometry.



Velocity contours at blade tip.

List of posters & demonstrations

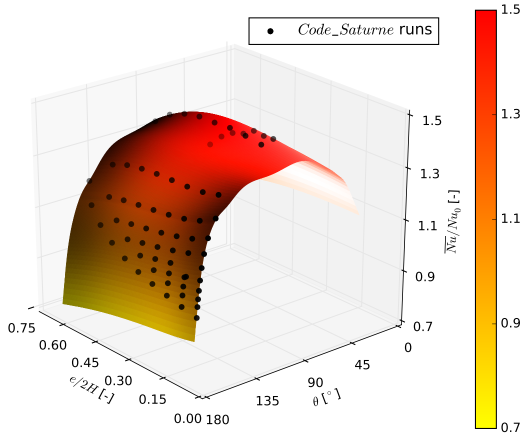
Vote for the best one!

Adaptation of the Lagrangian module of the CFD code *Code_Saturne* for near-field atmospheric dispersion of pollutants	by M. Bahlali, E. Dupont, B. Carissimo EDF R&D - MFEE
An Implicit Integral Formulation for the Modeling of Inviscid Fluid Flows in Domains Containing Obstacles	by C. Colas, M. Ferrand, J.-M. Hérard & E. Le Coupanec EDF R&D - MFEE & I2M
Ensuring the positiveness of the Reynolds Stress tensor for Differential Reynolds Stress Models	by C. Koren EDF R&D - MFEE
A Dual Mesh Hybrid RANS/LES Model for Wall-Bounded Flows	by P. Nguyen Uni. of Manchester
Meteorological fields clustering for a long term simulation of atmospheric flows	by A. Chahine, E. Dupont, B. Carissimo & L. Musson-Genon EDF R&D - MFEE, CEREA
Arbitrary Lagrangian-Eulerian ocean wave modeling at laboratory and field scale	by J. Harris Ecole des Ponts Paris-Tech
Modelling High-Speed Compressors with *Code_Saturne*	by Andrew Heffron & El-dad Avital Queen Mary University
Internal coupling of multi-scalars and multi-domains in *Code_Saturne*	by L. Garelli & M.. Storti UNL-CONICET
NEPTUNE_CFD: a multiphase solver powered by *Code_Saturne* HPC capabilities	by M. Guingo, C. Koren, J. Lavieville, N. Merigoux & S. Mimouni EDF R&D - MFEE
Multiphase flow in *Code_Saturne* using level set method	by Q. Rolland & C. Jause-Labert ABMI groupe
Simulation-based robust optimization approach to welding	by K. Dorogan, B. Iooss, M. Keller & L. Le Gratiet EDF R&D - PRISME
New approach (EK model) to simulate interactions between radionuclides and soil matrix with *Code_Saturne*	by G. Aynie, V. Loizeau & R. Lamouroux EDF R&D - LNHE
Elliptic Blending Models for turbulent heat fluxes	by G. Mangeon, R. Manceau, S. Ben-hamadouche, J.-F. Wald EDF R&D
Local modelling of sodium spray fires: study of aerosols	by J.-B. Rabilloud Uni. Paris Saclay
Interactions between Mixing, Flame Propagation, and Ignition in Non-Premixed Turbulent Flames	by X. Wang, V. Robin, C. Losier & A. Mura ENSMA & Université de Poitiers
Dynamic gradient reconstruction on *Code_Saturne* using task paradigm	by B. Lorendeau EDF R&D - MFEE
Live demontration of Salome_CFD: *Code_Saturne* - SYRTHES coupling	by M. Paolillo EDF R&D - HERMES

Heat exchanger exchanger optimization using *Code_Saturne*

by F. Mastrippolito – CEA Tech LITEN, LMFA ECL Lyon, Valeo & EDF

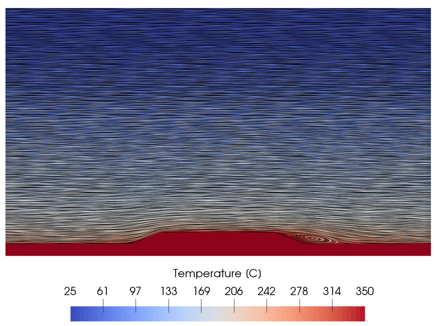
Heat exchangers are widely used in many industrial applications such as power engineering, chemical or petroleum industries. They are also used in cars, aviation, spatial applications, cryogenic engineering and other technology fields. In order to improve energy efficiency, working on heat transfer enhancement and heat exchanger design is very important.



Nusselt number Kriging-based metamodel of 77 Code_Saturne simulations.

This work will present shape optimization of heat exchanger with the goal to increase the thermal efficiency of the system. Heat exchanger physics is a multi-scale issue. As a consequence it is difficult to perform a CFD simulation of the complete system. A solution to this problem is to consider a local periodic pattern representative of the flow and then compute local and global performance indicators using integral method. We focus here on a 2D ribblet. Obvious objectives to achieve in this case are to maximize the convective heat transfer coefficient (non-dimensioned by the Nusselt number) and to minimize the head losses (expressed through a friction factor). They clearly are conflicting objectives. In this multi-objective optimization problem, the goal is to find the best trade-off between these objectives, called the Pareto set.

The optimization methodology uses genetic algorithm, *Code_Saturne* and Kriging-based meta-model. The genetic algorithm uses biology analogy to find the optimum: a population of individuals (designs) advances over several generations in order to find the Pareto set. For this it needs the solution of Nusselt number and friction factor for each design. In order to obtain a well-discretized Pareto set we find that 500 individuals over 100 generations are necessary. It makes 50 000 CFD points to compute, which is unrealizable in industrial applications. To avoid this without precision loss, a Kriging-based metamodel is used to extrapolate the results from a reduced number of CFD runs. This metamodel is then called by the genetic algorithm.



Temperature map and streamlines.

The ribblet case is computed using *Code_Saturne* under periodic conditions for velocity, pressure and temperature and internal coupling fluid/solid for the temperature. Then the Nusselt number and the friction factor are post-processed. Ribblet's shape varies from a square to a trapeze considering a constant height. Objectives are “metamodelized” and the genetic algorithm find the Pareto set.

Metamodel surfaces are presented as well as the Pareto set and some “optimal” designs.

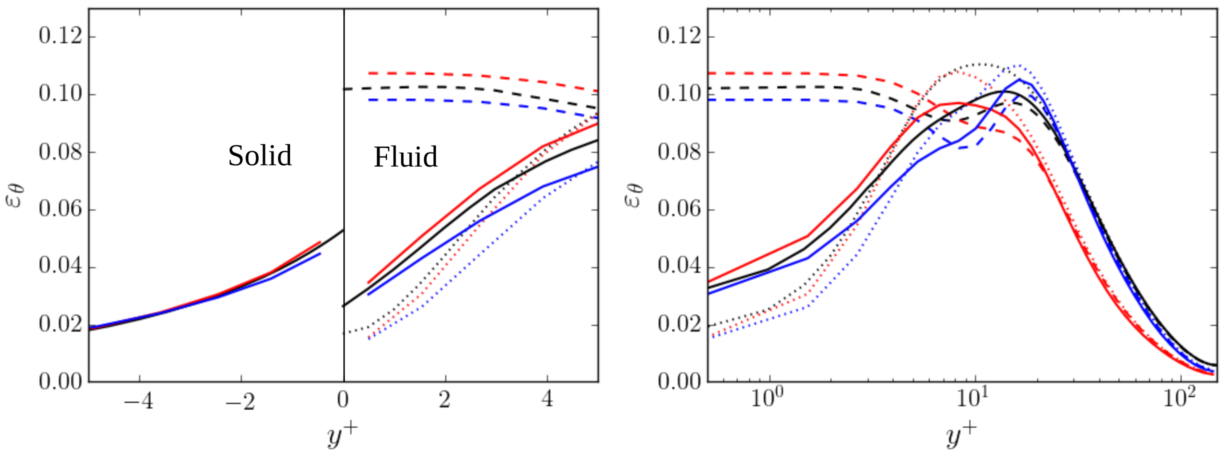
Quantification of the discontinuity of the temperature variance dissipation at a fluid-solid interface: Wall-resolved LES of turbulent channel flow with conjugate heat transfer

by C. Flageul & I. Tiselj – Josef Stefan Institute

Conjugate heat transfer represents the actual thermal coupling between a fluid and a solid part. It is of prime importance in nuclear industrial applications where fluctuating thermal stresses are a concern, *e.g.* in case of a severe emergency cooling (Pressurized Thermal Shock) or long-term ageing of materials such as thermal striping occurring in T-junctions. For such complex applications, numerical investigations often rely on Reynolds Averaged Navier Stokes (RANS) or wall-modelled Large Eddy Simulation (LES) approaches.

RANS models for conjugate heat transfer are relatively recent (Craft *et al.*, Journal of turbulence, Vol. 11, 2010). The authors have recently established that the dissipation rate (ε_θ) associated with the halved temperature variance ($\theta^2/2$) is discontinuous at the fluid-solid interface in case of conjugate heat transfer (Flageul et al., IJHFF, 08/2015, IJHMT, 05/2017). There is currently, to the authors' knowledge, no coupled RANS model for conjugate heat transfer taking this discontinuity into account. From an industrial perspective, LES remains the best option for thermal fatigue prediction but needs refinement at the wall, which makes it very expensive if not unaffordable, at high Reynolds numbers.

We assess the ability of wall-resolved LES to estimate this discontinuity of ε_θ on channel flows using latest developments in *Code_Saturne* (internal coupling). This is a step forward towards a rich validation database for future RANS models adapted to conjugate heat transfer.



Dissipation rate ε_θ. Black: DNS. Red: No SGS model. Blue: WALE. Dashed lines: imposed T. Dotted lines: imposed Q. Solid lines: Fluid-solid coupling. $Re_\tau = 400$, $Pr = 0.71$, $G = 0.5$, $K^2 = 2$ here.

Code_Saturne, a useful tool to compute atmospheric flows: some recent case studies

by C. Bonan, L. Makke and M. Nibart – ARIA Technologies

ARIA Technologies is a French SME specialized in atmospheric environment modelling. In this field ARIA Technologies edits several computer programs to compute indoor and atmospheric flows and also uses *Code_Saturne* CFD model, and especially its atmospheric module, since 2009 to perform studies with various objectives. Before this, MERCURE-ESTET was used, often facing some limitations in mesh size and/or CPU time because of the absence of parallelization. ARIA Technologies is also supporting some others companies or Institutes like EGIS and CNES to use *Code_Saturne*, as a standalone tool or call in the framework of specific software like the ARIACity.

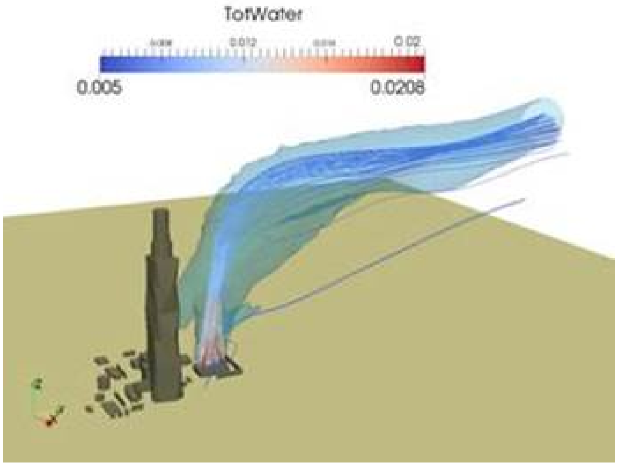
Highlights on different studies will be presented:

- "classical" risk assessment study where the objective is to estimate the toxic zone around an industrial area for a given incident scenario (AIR LIQUIDE);
- Air quality impact study in the vicinity of new road traffic infrastructure in Marseilles, with a

focus on tunnel portal (EGIS);

- Applied micro meteorology study to quantify the effect of the extension of a quarry (COLAS);
- A more original study for a solar plant where the objective is to compute the loss in solar energy due to water condensation plume (ALSTOM).

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



CFD study of flow redistributions in PWR fuel assemblies

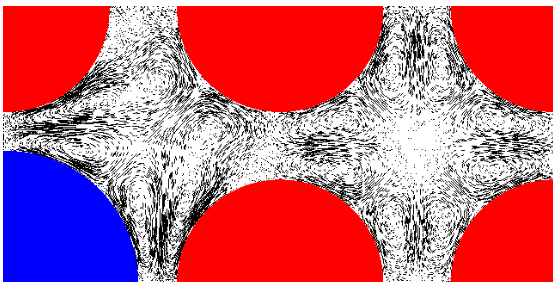
by R. Denèfle, M. C. Gauffre, S. Benhamadouche – EDF R&D MFEE

Computational Fluid Dynamics (CFD) is increasingly applied in nuclear reactor thermalhydraulic investigations. In addition to valuable experimental data, the use of well validated CFD codes is of great interest in order to improve the physical models and correlations implemented in larger scale tools such as “Component” or “System” scale codes utilized to evaluate safety parameters in industrial configurations. Actually, CFD simulations are valuable while dealing with particular physics or configurations for which experimental values are difficult to obtain.

Mixed cores involve potential oblique motions in the fuel assemblies. Such transverse flows may be the result of flow rate redistribution due to a change in the bundle configurations (thimble tubes) or to the use of different mixing grids (positions and/or geometries). This can naturally lead to different Critical Heat Flux (CHF) performances of the core that need to be assessed. In this context, CFD is an adequate tool to enlighten the physical phenomena for a better understanding of the internal redistributions occurring inside the fuel bundle and grids. CFD can also provide relevant information to prepare dedicated experimental set-up for such applications.

Relying on previous work on the validation of CFD in rod bundle and grid configurations such as EPRI NESTOR CFD round robin exercise or OECD-KAERI CFD benchmark, the main aim of the present work is to exploit CFD calculations performed with *Code_Saturne*, EDF in-house and open source CFD software, to improve the understanding of local redistributions occurring in fuel assemblies. For the purpose of our study, we will focus on two regions affected by flow rate redistributions: first, the bare bundle with typical subchannels (surrounded by four fuel rods) and thimble subchannels (surrounded by three fuel rods and a thimble rod), secondly, the Mixing Vane Grid (MVG) containing dimples and springs.

The study has been carried out with a stringent CFD methodology, evaluating the calculation sensitivity to the mesh refinement, the Reynolds number and the turbulence model in RANS, respectively. A 5×5 bundle geometry with a thimble rod in the middle at $Re = 20000$ allows the comparison of RANS (Eddy Viscosity and Reynolds Stress Models) with wall-resolved LES calculations, which provide uncertainty bounds applied later to higher Reynolds numbers RANS computations. The return of experience of the bare bundle study has been used to design the MVG study.



List of participants

NAME	First name	University / Company	Country	E-mail address
ALVAREZ	Dominique	EDF R&D	France	dominique.alvarez@edf.fr
AMOKRANE	Abdenour	EDF R&D	France	abdenour.amokrane@edf.fr
ANGOT	Guillaume	EDF R&D	France	guillaume-g.angot@edf.fr
ARCHAMBEAU	Frédéric	EDF-SOFINEL	France	frederic.archambeau@edf.fr
ARGAUD	Jean-Philippe	EDF R&D	Fance	jean-philippe.argaud@edf.fr
ASPROULIS	Panos	Renuda	Germany	panos.asproulis@renuda.com
AUDEBERT	Bruno	EDF R&D	France	bruno.audebert@edf.fr
AUDOUIN	Yoann	EDF	France	yoann.audouin@edf.fr
AULAGNIER	Céline	EDF DIPDE	France	celine.aulagnier@edf.fr
AUTRUSSON	Thibaut	EDF R&D	France	thibaut.autrusson@edf.fr
AYNIE	Gabrielle	EDF R&D	France	gabrielle.aynie@edf.fr
BAHLALI	Meissam	EDF R&D	France	meissam.bahlali@edf.fr
BANNER	Didier	EDF R&D	France	didier.banner@edf.fr
BATAIL	Rémi	AREVA	France	remi.batail@areva.com
BAUDRY	Cyril	EDF R&D	France	cyril.baudry@edf.fr
BENETTI	Luis	EDF	France	ramosbhluis@gmail.com
BENHAMADOUCHE	Sofiane	EDF R&D	France	sofiane.benhamadouche@edf.fr
BERLAND	Julien	EDF R&D	France	julien-j.berland@edf.fr
BILLARD	Flavien	Dassault Aviation	France	flavien.billard@dassault-aviation.com
BONELLE	Jérôme	EDF R&D	France	jerome.bonelle@edf.fr
BOIS	Guillaume	CEA	France	guillaume.bois@cea.fr
BORE	Clément	EDF R&D	France	clement.bore@edf.fr
BORTOLOTTI	Damien	EDF	France	damien.bortolotti@edf.fr
BOUCKER	Marc	EDF R&D	France	marc.boucker@edf.fr
BOULANT	Guillaume	EDF R&D	France	guillaume.boulant@edf.fr
BOULOUMOU	Ouafa	NÃľodyme	France	o.bouloumou@neodyme.fr
BRUNET	Florian	Open Cascade	France	florian.brunet@opencascade.com
BURKHART	Stephane	DGA	France	stephane.burkhart@intradef.gouv.fr
CALVO MAYO	Daniel	EDF Energy - UK Centre	United Kingdom	daniel.calvo@manchester.ac.uk
CAMY	Romain	EDF SEPTEN	France	romain.camy@edf.fr
CARISSIMO	Bertrand	CEREA ENPC-EDF R&D	France	carissim@cerea.enpc.fr
CARTLAND-GLOVER	Gregory	STFC	United Kingdom	greg.glover@stfc.ac.uk
CARUSO	Ange	EDF R&D	France	ange.caruso@edf.fr
CARUYER	Céline	EDF R&D	France	celine.caruyer@edf.fr
CERVANTES	Michel	Luleå university of Technology	Sweden	michel.cervantes@ltu.se
CHABARD	Jean-Paul	EDF R&D	France	jean-paul.chabard@edf.fr
CHAHINE	Ali	EDF R&D	France	ali-externe.chahine@edf.fr
CHARWATH	Marcus	EDF R&D	France	marcus.charwath@edf.fr
CHEN	Fei	GoldWind	China	627362120@qq.com
COLAS	Clément	EDF R&D	France	clement.colas@edf.fr
CORDIER	Hervé	EDF SEPTEN	France	herve.cordier@edf.fr
CROUZET	Fabien	EDF R&D	France	fabien.crouzet@edf.fr
DBJAY	Sylvain	AKKA Technologies	France	sylvain.dbjay@akka.eu
DE AMICIS	Jacopo	EDF Energy - UK Centre	United Kingdom	jacopo.deamicis@edfenergy.com
DE LAAGE DE MEUX	Benoît	EDF R&D	France	benoit.de-laage-de-meux@edf.fr
DE SANTIS	Andrea	Nuclear Research and Consultancy Group (NRG)	Netherlands	a.desantis@nrg.eu
DE SOZA	Thomas	EDF R&D	France	thomas.de-soza@edf.fr
DEFFORGE	Cécile	EDF R&D	France	cecile.defforge@edf.fr
DELMAS	Josselin	EDF R&D	France	josselin.delmas@edf.fr
DENEFLE	Romain	EDF R&D	France	romain.denefle@edf.fr
DERI	Enrico	EDF R&D	France	enrico.deri@edf.fr
DEVIC	Catherine	EDF	France	catherine.devic@edf.fr
DOROGAN	Kateryna	EDF R&D	France	kateryna.dorogan@edf.fr
DUPONT	Eric	EDF R&D	France	eric.dupont@edf.fr
EL WESHAHY	Gihan	EDF	France	gihan.el-weshahy@edf.fr
ESPINASSE	Gerard	BERTIN	France	espinasse@bertin.fr
EUDE	Yohann	Renuda	France	yohann.eude@renuda.com
FAYDIDE	Bernard	CEA	France	bernard.faydide@cea.fr
FERRAND	Martin	EDF R&D	France	martin.ferrand@edf.fr
FERRARI	Jérôme	EDF R&D	France	jerome.ferrari@edf.fr
FLAGEUL	Cédric	Institut Jozef Stefan	Slovenia	cedric.flageul@ijs.si
FLOUR	Isabelle	EDF R&D	France	isabelle.flour@edf.fr
FONTAINE	Jacques	EDF R&D	France	jacques-j.fontaine@edf.fr
FOURNIER	Yvan	EDF R&D	France	yvan.fournier@edf.fr
FREYDIER	Philippe	EDF SEPTEN	France	philippe.freydier@edf.fr
GARELLI	Luciano	CIMEC - UNL CONICET	Argentina	lucianogarelli@gmail.com
GAUFFRE	Marie-Charlotte	EDF R&D	France	marie-charlotte.gauffre@edf.fr
GEAY	Anthony	EDF R&D	France	anthony.geay@edf.fr

GORE	Nathalie	Open Cascade	France	nathalie.gore@opencascade.com
GRECO	Angelo	OPTIMISE	France	greco.angelo4@gmail.com
GUINGO	Mathieu	EDF R&D	France	mathieu.guingo@edf.fr
HARRIS	Jeffrey	Ecole des Ponts ParisTech	France	jeffrey.harris@enpc.fr
HEFFRON	Andrew	Queen Mary, University of London	United Kingdom	a.p.heffron@qmul.ac.uk
HELENE	Mathieu	EDF R&D	France	mathieu.helene@edf.fr
HÉRARD	Jean-Marc	EDF R&D	France	jean-marc.herard@edf.fr
HÜLSEMANN	Frank	EDF R&D	France	frank.hulsemann@edf.fr
HURISSE	Olivier	EDF R&D	France	olivier.hurisse@edf.fr
JAMELOT	Erell	CEA	France	erell.jamelot@cea.fr
JAUSE-LABERT	Clément	ABMI	France	c.jauselabert@abmi-engineering.com
KNOWLES	Robin	CFD Engine	United Kingdom	robin@cfdengine.com
KUMBARO	Anela	CEA	France	anela.kumbaro@cea.fr
KUZNETSOV	Konstantin	Ecole des Ponts, Laboratoire d'Hydraulique Saint-Venant	France	konstantin.kouznetsov@gmail.com
LACAZEDIEU	Elisabeth	EDF R&D	France	elisabeth.lacazedieu@edf.fr
LAMOUROUX	Raphaël	EDF R&D	France	raphael.lamouroux@edf.fr
LANCIAL	Nicolas	EDF R&D	France	nicolas.lancial@edf.fr
LAVAL	Damien	DCNS	France	damien.laval@dcnsgroup.com
LE COUPANEC	Erwan	EDF R&D	France	erwan.lecoupanec@edf.fr
LE FRIEC	Christophe	Thermic Simulation & Recherche	France	bureaux@tsr-be.com
LEFEBVRE	Vincent	EDF R&D	France	vincent.lefebvre@edf.fr
LELONG	Franck	EDF SEPTEN	France	franck-f.lelong@edf.fr
LOIZEAU	Vincent	EDF R&D	France	vincent.loizeau@edf.fr
LORENDEAU	Benjamin	EDF R&D	France	benjamin.lorendeau@edf.fr
LORENTZ	Eric	EDF R&D	France	eric.lorentz@edf.fr
MA	Li	ISAE-Supaéro/EDF chatou	France	malidamien93@gmail.com
MAKKE	Laurent	ARIA Technologies	France	lmakke@aria.fr
MANGEON	Gaëtan	EDF R&D	France	gaetan.mangeon@edf.fr
MARTÍN SÁNCHEZ	Alexandra	EDF R&D	France	alexandra.martin-sanchez@edf.fr
MASTRIPPOLITO	Franck	CEA	France	franck.mastrippolito@cea.fr
MERIGOUX	Nicolas	EDF R&D	France	nicolas.merigoux@edf.fr
MINIER	Jean-pierre	EDF R&D	France	jean-pierre.minier@edf.fr
MONFORT	David	EDF R&D	France	david.monfort@edf.fr
MONTOUT	Michaël	EDF R&D	France	michaelmontout@yahoo.fr
MOULINEC	Charles	STFC Daresbury Laboratory	United Kingdom	charles.moulinec@stfc.ac.uk
NAURY	Sylvie	CEA	France	sylvie.naury@cea.fr
NGUYEN	Philipp	The University of Manchester	United Kingdom	philipp.nguyen@postgrad.manchester.ac.uk
OUKACINE	Fadila	EDF R&D	France	fadila.oukacine@edf.fr
PAOLILLO	Martine	EDF R&D	France	martine.paolillo@edf.fr
PAPACONSTANTINOU	Thomas	EDF R&D	France	thomas.papaconstantinou@edf.fr
PUSCAS	Maria adela	CEA	France	maria-adela.puscas@cea.fr
QUEMERAIS	Eric	ONERA	France	eric.quemerais@onera.fr
QUIBEL	Lucie	EDF	France	lucie.quibel@edf.fr
RABILLOUD	Jean-Baptiste	Université Paris Saclay	France	jb_rabilloud@icloud.com
RAYNAUD	Christelle	EDF	France	christelle.raynaud@edf.fr
ROBIN	Vincent	ENSMA Université de Poitiers	France	vincent.robin@isae-ensma.fr
ROLLAND	Quentin	ABMI	France	q.rolland@abmi-engineering.com
ROLLAND	Romain	DGA	France	robin.simon@intradef.gouv.fr
SALVATORE	Patricia	CEA	France	patricia.salvatore@cea.fr
SANTAMARIA	Jorge	EDF R&D	France	jorge.santamaria@edf.fr
SAPA	Bertrand	EDF R&D	France	bertrand.sapa@edf.fr
SENECHAL	Dorothée	EDF R&D	France	dorothee.senechal@edf.fr
SHAMS	Afaque	Nuclear Research and Consultancy Group (NRG)	Netherlands	shams@nrg.eu
SICCAMA	Arné	Nuclear Research and Consultancy Group (NRG)	Netherlands	siccama@nrg.eu
SIMON	Robin	DGA	France	robin.simon@intradef.gouv.fr
STANCIU	Mugurel	EDF R&D	France	mugurel.stanciu@edf.fr
STOBIAC	Vincent	EDF R&D	France	vincent.stobiac@edf.fr
STORTI	Mario	CIMEC (CONICET, UN Litoral, Argentina)	Argentina	mario.storti@gmail.com
TONELLO	Nicolas	Renuda	United Kingdom	nicolas.tonello@renuda.com
TOUAZI	Ouardia	EDF R&D	France	ouardia.touazi@edf.fr
TRUCHOT	Benjamin	INERIS	France	benjamin.truchot@ineris.fr
TRUONG	Ngoc-minh	AKKA TECHNOLOGIES	France	ngocminh.truong@akka.eu
URIBE	Juan	EDF Energy	United Kingdom	juan.uribe@edfenergy.com

VIVALDI	Daniele	IRSN	France	daniele.vivaldi@irsn.fr
WALD	Jean-François	EDF R&D	France	jean-francois-j.wald@edf.fr
WEI	Xiao	ARIA Technologies	France	xwei@aria.fr
WENDUM	Denis	EDF R&D	France	denis.wendum@edf.fr
WILSBY	Oscar	University of Cambridge	United Kingdom	ow222@cam.ac.uk
XU	Tingting	EDF China	China	tingting.xu@edf.fr
YAU	Xavier	EDF R&D	France	xavier.yau@edf.fr

Here today