

SECONDARY FLOWS IN PWR HOT LEGS, CFD MIGHT BRING SOME LIGHT ON THE FLOW BEHAVIOR

S. BELLET¹, S. BENHAMADOUCHE² and A.FERRET¹

¹ EDF Nuclear Engineering - Basic Design department, France

² EDF R&D – Fluids Dynamics, Energy and Environment Department, France
serge.bellet@edf.fr, sofiane.benhamadouche@edf.fr

Abstract

Secondary flows play an important role in several locations of the primary circuit of a PWR. A full program for a better understanding of these kinds of flows with CFD computations has been launched at EDF. Since the beginning of 2009, EDF started new activities around CFD simulations of swirling and secondary flows. A progressive approach (increasingly close to the industrial needs) is adopted with different items relative to bibliography study, academic to semi-industrial test cases, simplified upper plenum and hot leg, mock-up and reactor simulations. The present paper focuses on a recent work, based on a pure CFD approach, dealing with the understanding of the complex structure of secondary flows which are observed in the hot leg. Different simulations have been performed to find the origin of the secondary flows in the upper plenum and also to understand their evolution along the hot leg.

A first simulation (empty plenum) enlighten on the origin of the two “main” secondary structures widely observed in other numerical simulations and experiments. Two counter rotating vortices are created due to a geometrical effect (asymmetric location of the hot leg). The axis that separates these two structures is rather vertical at the inlet of the hot leg and turns clockwise while going through the leg. The rotation is not that important. A second simulation (with the guide tubes) enlighten on the role of the RCCA guide tubes located as an obstruction of flow in front of the hot leg nozzle inlet. It is shown that secondary motions are created along the tubes due to an upward/downward flow and these structures can have a non negligible trace at the hot leg inlet. However, in particular far from the leg inlet, the global structure consists in two counter rotating vortices similar to the one observed on an empty plenum but with a different orientation (the axis that separates the two structures is shifted by around 20° compared to the configuration without guide tubes) and a similar rotation. Concerning the temperature, the global distribution of the temperature is similar in both cases, in particular at the beginning of the hot leg. While going further in the hot leg, the influence of the secondary motions is clearly observe, what makes the temperature distribution less homogenous and less smooth.

One of the most important conclusions is also that Reynolds Stress Models supported by an adequate mesh refinement can capture complex secondary flows, which is not the case of standard approaches such as the ones which use a standard k-ε model for example.

KEYWORDS

Thermal-hydraulic, swirl, secondary flow, upper plenum, hot leg, PWR

Introduction

Figure 1 shows the different locations where swirling or secondary flows might be crucial in the understanding of the local or global flow structure in a PWR primary circuit. More precisely, these kinds of flows are located in the U-bend upstream the steam generator, in the vessel lower-plenum, along the fuel assemblies and in the upper-plenum/hot leg as described in [4]. In this paper, one focuses on the upper plenum/hot leg geometry and more precisely on the flow structure along the leg and its effect on the temperature distribution. This is directly linked to the issue of temperature heterogeneity in the hot leg often call " T_{HOT} streaming" phenomena.

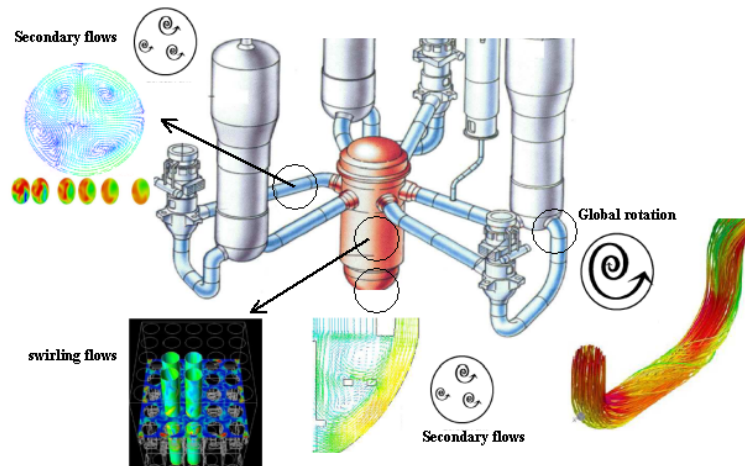


Figure 1 Several locations of secondary flows in the primary circuit of a PWR

The flow in the upper plenum encounters several obstacles of different kinds. Several experiments have been dedicated to study the flow and temperature fields in this region and more particularly along the hot legs. In France, Banquise mock-up (1300 MW reactor configuration, see Figure 2) have been used for this purpose. PIV measurements and thermal tracing allowed to show a complex velocity field driven by secondary flows in the hot legs. Understanding these structures is of major interest in order to predict the distribution of the temperature in the hot leg. In 1999 (see [2]), EDF carried out numerical simulations (with N3S-EF in-house CFD code at that time) on this configuration in order to validate the CFD approach. These studies largely contributed to understand the behaviour of the flow into the upper plenum and the hot legs of a 1300 MW reactor. Measurements/calculations comparisons were conducted for the velocity and the scalar fields along the hot legs (see Figure 3). Caruso et al. [2] highlighted that the global structure of the flow is well predicted with standard approaches for turbulence modelling (1st order closures). In particular, they concluded that a CFD code such as N3S-EF (with a standard $k-\epsilon$ turbulence model) is able to predict the secondary vortices obtained by the experiment even if the position of their cores was not always well located. Since this period, other simulations have been conducted with others codes (*Code_Saturne* [1] and *Star-CD*). Some limitations are still observed in particular when one moves to other reactor configurations.

All these observations led EDF to think that the secondary structures of the flow in the hot legs play an important role, in particular if one considers the transport of the temperature. That is the reason for the special interest given by EDF to these phenomena and that is why it launched a program of research/studies on this topic. This program is detailed hereafter.



Figure 2 Banquise experiment (1/5th scale of a PWR 4 loops)

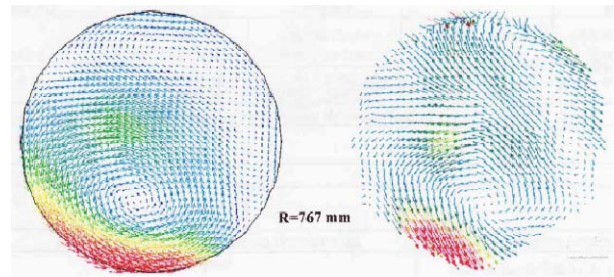


Figure 3 Comparison between numerical and experimental results (Left: N3S-EF code - Right: BANQUISE exp)

1. EDF CFD program for a better understanding of secondary flows

EDF believes that CFD and the experimental approaches have to be utilized simultaneously. The experiments are used to obtain several physical information on a given configuration and to validate the codes. However, they are expensive, in particular if one wants to represent reactor conditions (high Reynolds numbers, high pressures and temperatures, ...) and they might be confined and not representative of the “real” reactor configurations (confinement effects because of side walls in the case of fuel rods for example). On the other hand, CFD, after being verified and validated thanks to experimental and very fine numerical data, might be used either to have more information particularly in zones which can not be reached by experimental devices or to study parametric configurations by varying few parameters of the flow (inlet conditions, heat fluxes, ...) at a lower cost than experiments. In other cases, CFD computations might be used to configure an experiment and to know *a priori* which regions are of interest and thus where to measure the desired quantities. The authors believe that we are still far from doing CFD computations in “blinded” conditions without any validation process. However, the previous arguments show clearly that CFD simulations are and will remain a major tool for engineering decisions.

One of the major issues in CFD is the uncertainty introduced by the numeric's (discretization errors, numerical scheme, ...) and by the models (turbulence models for example). In order to take into account turbulence phenomena for example, the constant growth of computational power (High Performing Computations, HPC) allows to carry out large computations with several approaches. While using Large Eddy Simulation (LES) technique, higher Reynolds numbers (10 000 to 100 000) then few years ago are affordable today for limited computational domains. However, using this technique on a complex geometry such as the upper plenum, even at a moderate Reynolds number is still not affordable. While using RANS techniques, higher order models (such as Reynolds Stress Models (RSM)) than the first order standard k-epsilon model (still widely used in the industrial CFD community) are today affordable at high to very high Reynolds numbers (1 000 000 to 10 000 000) and on large computational domains (these estimations include the use of wall functions). The authors believe, as has been recently argued by Chabard and Laurence [3], that both techniques

(LES and RANS) are mandatory and will remain complementary in the near future. Turbulence modelling, the numerical schemes and the grid refinements are then crucial issues in predicting swirling and rotating flows.

Since the beginning of 2009 (see [4]), EDF started new activities around the simulation of swirling and secondary flows. A progressive approach (progressively evolving towards our industrial needs) is proposed, the main objective being a better understanding swirling and secondary flows in general and more particularly in the upper part of the vessel (upper plenum and hot legs):

1. A bibliography review on secondary flows in order to get the state of art on this topic,
2. Simulations of several academic to semi-industrial test-cases (see [5], [6], [7], [8]) such as the square duct and few “vortex” tubes. Sensitivity studies [4] are carried out concerning turbulence modelling, the numerical schemes and the grid refinement and the results are compared to the available experimental or fine CFD data,
3. **Simulation of a simplified geometry of the upper plenum and the primary hot legs using the feedback of the two first steps.**
4. Simulations of EPR mock-up (upper plenum and the hot legs) with the feedback of the three first steps,
5. A reactor simulation with realistic conditions (the issue of the dependency of the configuration to the Reynolds number is kept in mind and investigations has started in the 3rd step using a simplified geometry).

The present paper focuses on step 3, some recent results are presented and discussed in the next section.

2. Numerical simulations on a simplified Upper plenum / Hot leg

2.1 The simplified geometry

The main goal is not to simulate the real flows but to improve our understanding of the role played by the RCCA guides tubes and particularly the guides tubes located close to the hot leg inlet. Only a quarter of the upper plenum is considered in order to limit the computational needs. Symmetries are assumed on the vertical sides (the real dimensions of a PWR 1300 are utilized). Figure 4 shows the simplified geometry used in the present study to mimic a reactor configuration. Simplified inlet conditions are used by interpolating and simplifying a real core configuration : the bulk velocity in the hot leg, the density, the viscosity and the velocity/temperature distributions at the inlet (upper core plate) are representative of a real reactor configuration.

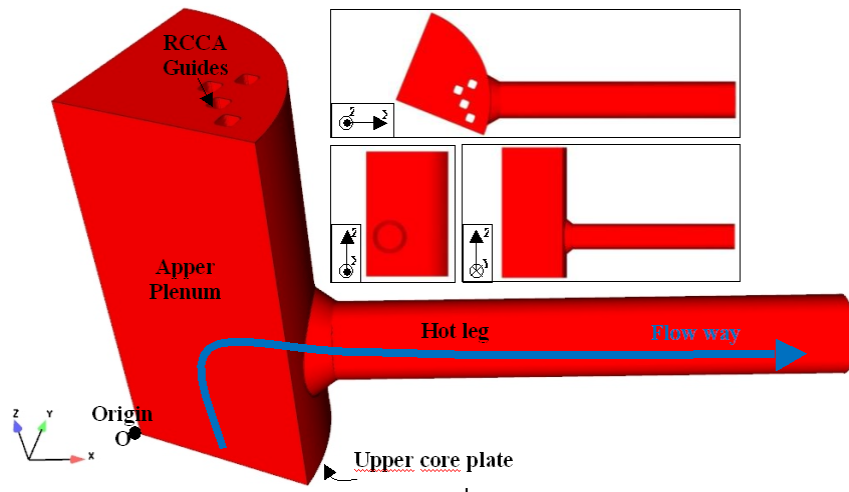


Figure 4 Upper plenum/hot leg simplified geometry

2.2 Mesh generation

The authors give a particular care to the meshing effort because they consider, as it is widely shared by the CFD community in industry, that the spatial discretization is a key point to obtain satisfactory results in particular with a collocated finite volume discretization and depending on the turbulence model (RANS and LES will not require the same numerical options and thus the same mesh quality).

Different meshes were created with ANSYS ICEM_CFD 12 software. With this tool, it has been possible to obtain fully hexahedral conforming meshes around the RCCA guides tubes with the blocking approach. Several meshes with a progressive refinement have been generated for sensitivity studies. Only two meshes are shown in the present study in Figure 5.

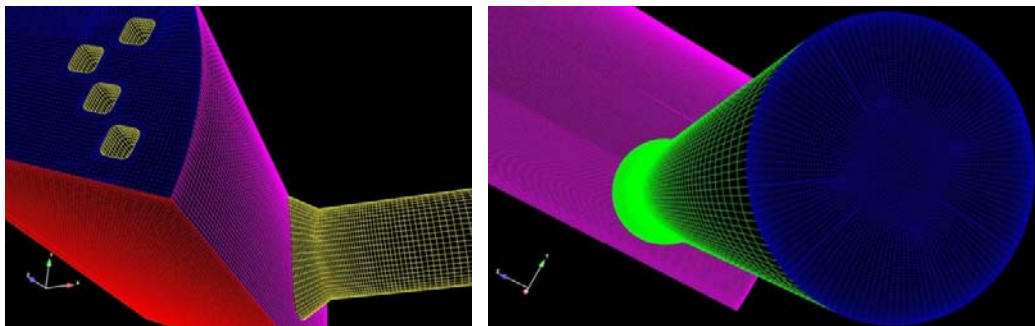


Figure 5 Coarse (0.7 M elements) and refined (5.5 M elements) Meshes

2.3 The numerical simulations

2.3.1 The main parameters

All the simulations run under nominal steady state reactor conditions (Pressure: 153.7 bars, Temperature: 326 °C). Temperature was treated as a passive scalar. The Reynolds number is around 10^8 in the hot leg under reactor conditions. Inlet conditions are extrapolated from reactor flow rate distribution considering each fuel assembly position.

2.3.2 First step : simulation of a quarter empty upper plenum with its hot leg (with the refined mesh)

Here, the aim is to study the behavior of the flow without RCCA guides, focusing the analysis on velocity field especially on the transition region between the upper plenum and the hot leg to understand the origin of the secondary structures and how they propagate along the circular duct.

Several cuts at different locations are performed (see Figure 6) and the tangential velocity is plotted (the origin giving x can be found in Figure 4). On these different cuts along the hot leg, one can see the progressive formation of the two counter rotating secondary structures (called respectively S1 and S2). At the beginning an asymmetric convergence of the velocity vectors is observed (see the stagnation point respectively in Figure 6 and Figure 7 (called stop point in this figure)). The impact of this asymmetric flow on the hot leg wall induces then the two counter rotating secondary flows. A sketch of the tangential velocity evolution along the leg is presented in Figure 7. The axis which separated the two counter rotating vortices turns clockwise while going through the hot leg (this is also clearly observed on Figure 6).

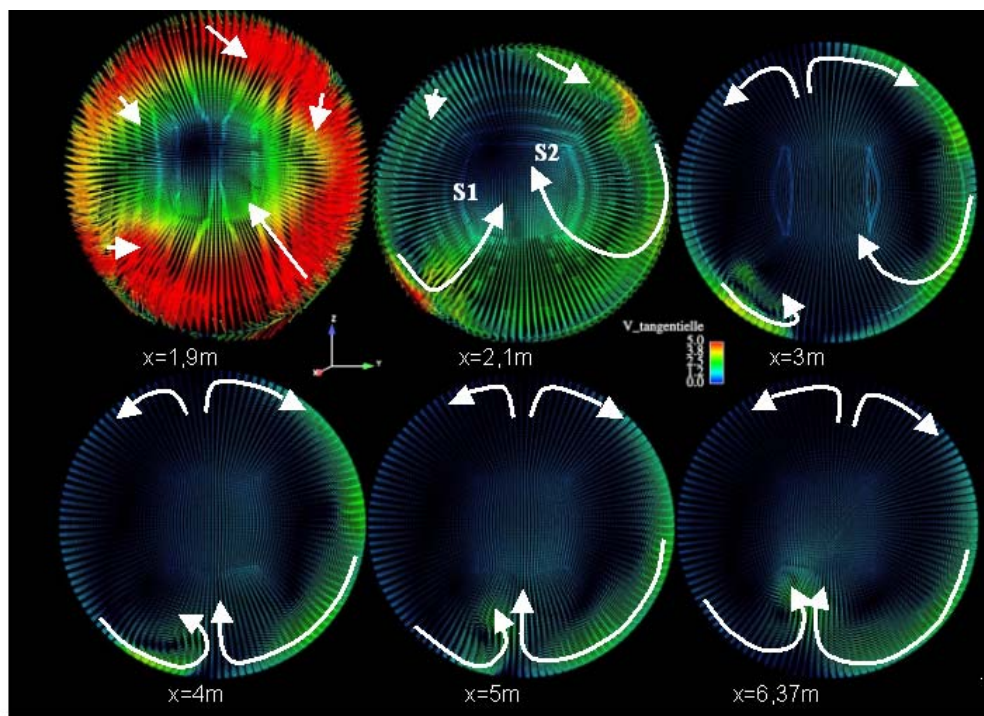


Figure 6 The tangential velocity at several cuts along the hot leg (view from the outlet of the leg, empty upper plenum)

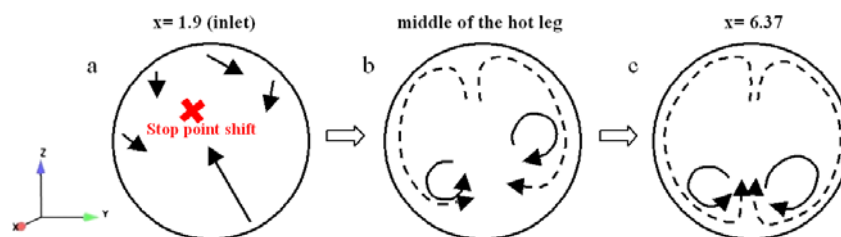


Figure 7 Sketch of the counter rotating structures along the hot leg for an empty upper plenum

2.3.3 Second step : Simulation of a quarter upper plenum with four RCCA guide tubes (with the refined mesh)

In the upper plenum, the flow motion is globally similar to the one observed previously except around the RCCA guide tube. The observation of the velocity field (see Figure 8 for the central guide) reveals that upstream (resp. downstream) the RCCA guide tubes an upward (resp. an opposite downward/upward) flow takes place. Therefore, behind the tube (downstream side of the guide tube), at the junction of the two opposite flows, new secondary structures are generated (the mechanism is probably more complex, but one notices that the secondary structures emanate from the stagnation region where the two opposite flows meet). Figure 9 shows these two counter-currents (1 and 4) facing each other and accordingly generating a horizontal flow at the point (2). This secondary flow is carried or enhanced by the global surrounding upward flow (3), to form two secondary structures: B1 and B2 (see the numbering of the guide tubes in Figure 8). These structures have a velocity close to 3 m/s on their periphery and a 10 cm diameter (one recalls that the bulk velocity in the hot leg is around 18 m/s and that the diameter of the leg is about 70 cm).

The same mechanism appears on the others guide tubes. Due to the different distance and locations (from the tubes to the inlet nozzle), the structures have different intensities and may also interact. Some of them disappear very quickly. The tube A (see Figure 8) gives two structures, then the tube D creates also two. All these structures are very similar to B1 and B2 represented in Figure 9. Finally, all these structures combine each other's then spread along the hot leg.

Figure 10 shows the different structures just upstream the hot leg nozzle. One can observe that the structures emanating from the guide tube A have disappeared as this guide is far and its structures might be broken by the one emanating from tubes B and D.

Further downstream, at the inlet of hot leg Figure 11, one can still see most of these structures. Some are missing and some new ones are generated (with a prime on the figure). S1 and S2 (see Figure 6 for the case without guide tubes) still exist and seem to be the two secondary structures coming from the upper plenum.

In order to compare more precisely the two configurations with and without the RCCA guide tubes, the tangential streamlines are plotted in Figure 12. Although they are more complex than in the first test-case, one can observe the main two counter rotating structures when the four guide tubes are represented. At the hot leg inlet nozzle, there is a small difference concerning the location and the shape of the stagnation point of the flow. Along the leg one observes the different positions and shapes of the two counter rotating structures. While the axis between the two structures is almost vertical with an empty plenum at the hot leg inlet, it makes an angle of around 45° with the vertical axis. The rotation of this axis is moderate with and without the guide tubes (a rotation of few degrees is observed and clockwise in both cases). It seems then that the secondary structures do not drastically modify the global topology of the flow in the hot leg which consists in two counter-rotating vortices but has an impact on the location of these two structures. One can also observe that the intensity of the tangential velocity is higher when the guide tubes are represented. The structure with the highest tangential velocity (the one on the right in both simulations) seems to push the other one, what creates the global rotation.

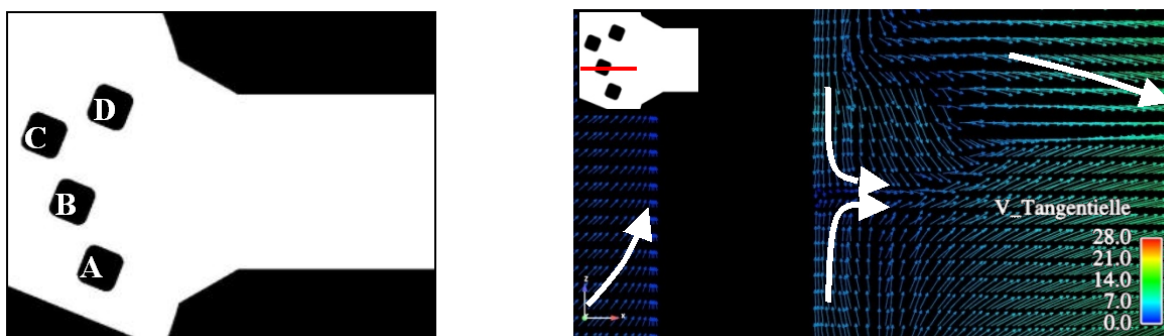


Figure 8 Instantaneous velocity field for the red cut of the guide tube B

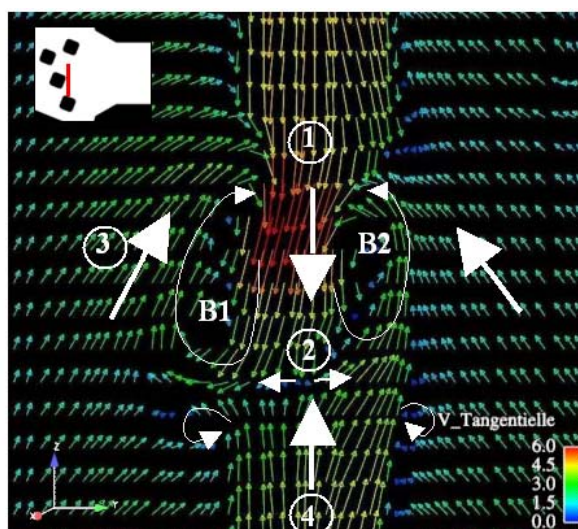


Figure 9 Velocity field downstream the RCCA guide tube B (view from the outlet of the hot leg)

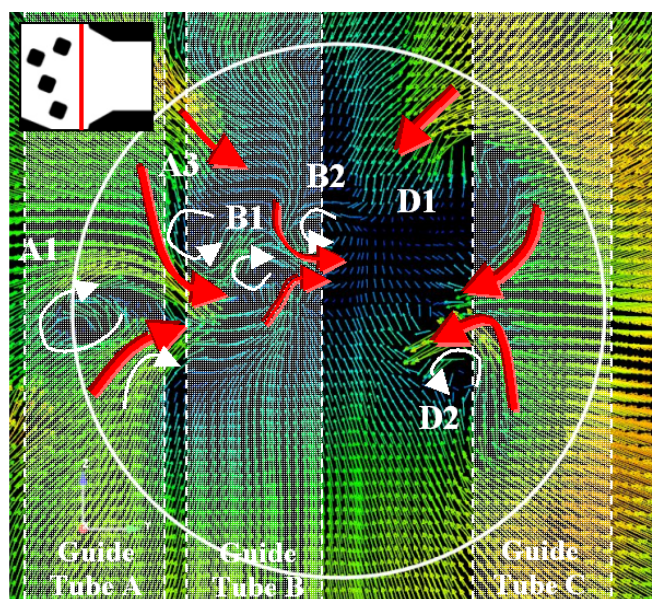


Figure 10 Velocity field in the upper plenum close (upstream) to the hot leg nozzle

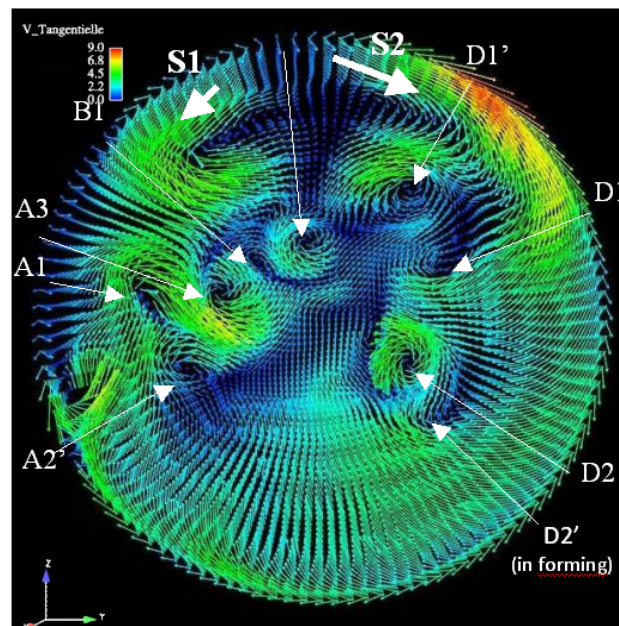


Figure 11 Tangential velocity field at the hot leg inlet nozzle (view from the outlet of the hot leg)

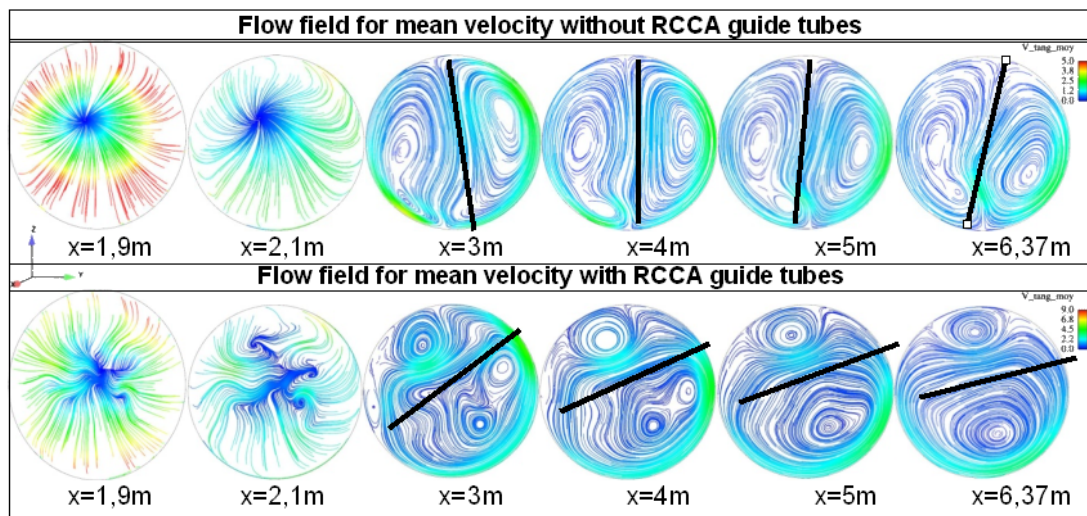


Figure 12 Stream lines of mean velocity along the hot leg (view from the outlet of the hot leg)

2.3.4 Temperature field

A temperature distribution that mimics the nominal reactor conditions have been imposed at the upper plenum inlet. Figure 13 shows the temperature evolution in the plenum and at the entrance of the hot leg. A usual observation for this kind of flow (which shows also that the velocity at the inlet is somehow realistic) can be made: the temperature is higher on the upper region of the hot leg as the center of the upper core plate is hotter than its periphery.

Figure 14 shows how the temperature evolves along the hot leg for the two main configurations; with and without guide tubes. On the one hand, at the hot leg nozzle inlet, the global temperature

behaviour is almost the same in the two configurations with or without RCCA guide tubes. The temperature field is however more disturbed when the guide tubes are represented. On the other hand, the thermal spots (regions) location/evolution are different. The cold region which is first located in the lower part of the section is restricted to the left while going through the leg. The hot region is very smooth without the guide tubes and is largely influenced by the secondary structures in the second case. The hot fluid stays longer close to the center of the duct when the guide tubes are represented. One could think that the secondary motions due to the presence of the guide tubes will enhance the mixing. The first analysis provided herein does not show this phenomenon (although the mixing seems different in the two configurations).

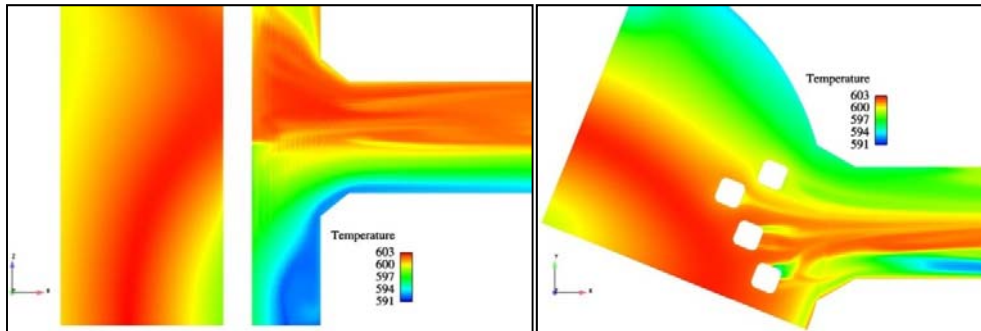


Figure 13 Temperature field (the minimum and maximum temperature are in Kelvin, they correspond respectively to 318°C and 330°C)

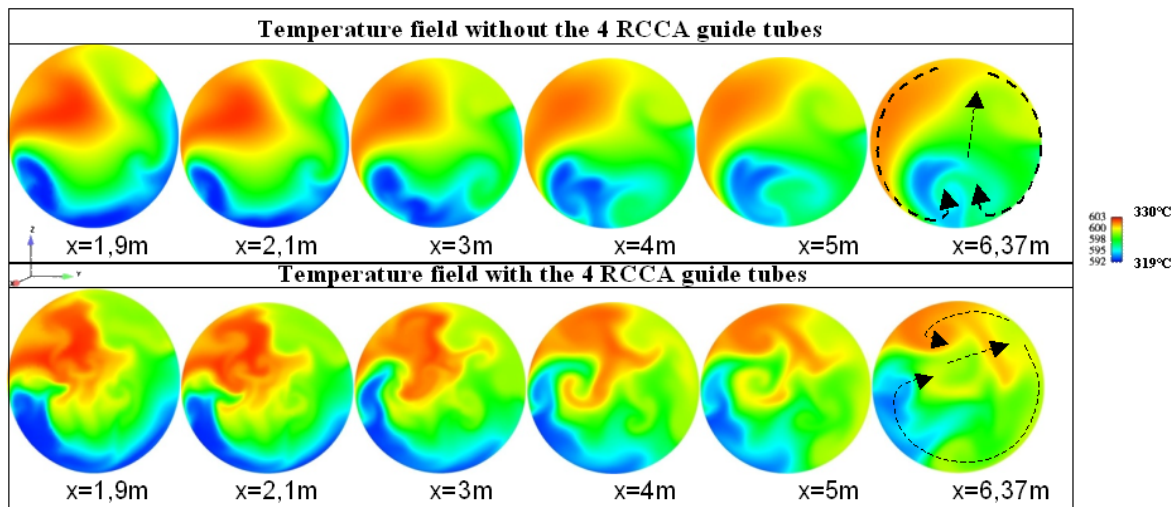


Figure 14 Temperature field evolution along the hot leg (view from the outlet of the leg)

2.3.5 Sensitivity studies: main results

Some sensitivity studies have been carried out concerning the mesh refinement, the turbulence model and the Reynolds number.

The first one confirms that it is necessary to use a sufficiently refined mesh to capture the secondary structures. Typically a structures seems to need around 50 cells (as a minimum value) to be captured. A coarser mesh leads to a poor prediction of the secondary motions.

Concerning the turbulence model, the results confirm the point of view developed in § 1 and confirmed in [8]: the richer result comes with the use of a second order closure. Indeed, Figure 15 underlines that

the standard k- ϵ model does not detect the secondary structures, even with the fine mesh. This is due to the fact that it can not reproduce the anisotropy of the flow, which is strong in the present case. Concerning the Reynolds number effect, the results show an impact on the position of the structures at the outlet of the hot leg, but some additional simulations and investigations will be engaged to statute on this effect and to validate the main conclusions of the present physical analysis.

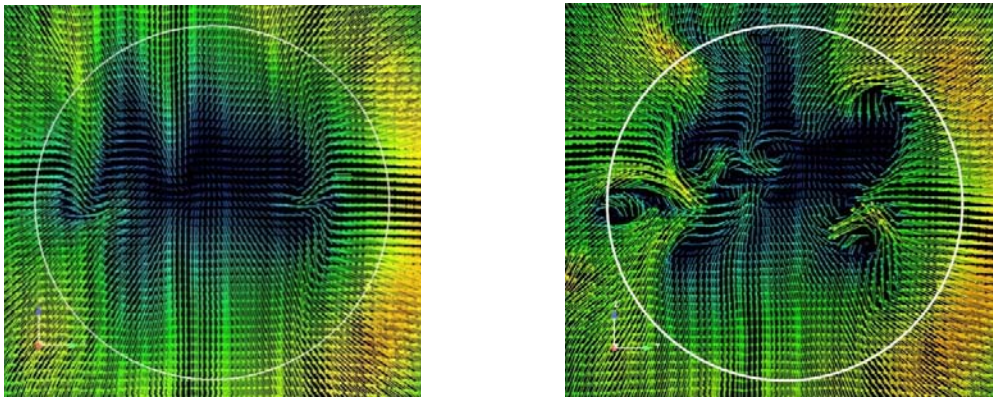


Figure 15 Sensitivity to the turbulence model (left: k- ϵ model, right: RSM model)

3. Conclusions

A progressive approach (increasingly closer to the industrial needs) was adopted by EDF with different items relative to bibliography study, academic to semi-industrial test cases validation, simplified upper plenum and hot leg, mock-up and finally reactor simulations.

The present study focused on a simplified quarter upper plenum and its hot leg in reactor conditions. Two simulations have been carried out: the first one with an empty plenum and the second one by representing four guide tubes.

The first simulation (empty plenum) enlighten on the origin of the two “main” secondary structures widely observed in other numerical simulations and experiments. Two counter rotating vortices are created due to a geometrical effect (asymmetric location of the hot leg). The axis that separates these two structures is rather vertical at the inlet of the hot leg and turns clockwise while going through the leg. The rotation is not that important. The second simulation (with the guide tubes) enlighten on the role of the RCCA guide tubes located as an obstruction of flow in front of the hot leg nozzle inlet. It is shown that secondary motions are created along the tubes due to an upward/downward flow and these structures can have a non negligible trace at the hot leg inlet. However, in particular far from the leg inlet, the global structure consists in two counter rotating vortices similar to the one observed on an empty plenum but with a different orientation (the axis that separates the two structures is shifted by 45° compared to the configuration without guide tubes) and a similar rotation. Concerning the temperature, the global distribution of the temperature is similar in both cases, in particular at the beginning of the hot leg. While going further in the hot leg, the influence of the secondary motions is clearly observe, what makes the temperature distribution less homogenous and less smooth.

One of the most important conclusions is also that Reynolds Stress Models supported by an adequate mesh refinement can capture complex secondary flows, which is not the case of standard

approaches such as the ones which use a standard k- ϵ model for example. More investigations are however needed to study the influence of the Reynolds number.

These studies contribute to a better understanding of the physical flow behaviour and notably demonstrate that the secondary structures play an important role in the transportation of the thermal spots coming from the core outlet. These different results lead us to provide some recommendations for the next step of the program which should concern CFD validation against the experimental data from mock-up representative of reactor geometry. Our approach will also allow to progress in the PIRT approach of the temperature heterogeneity phenomena in the hot leg which is often call "T_{HOT} streaming" phenomena.

4. References

- [1] Archambeau, F. Méchitoua, N. and Sakiz, M, (2004) "Code_saturne: a finite volume code for the computation of turbulent incompressible flows – industrial applications", Int. J. on Finite Volume, Electronical edition: <http://averoes.math.univ-paris13.fr/html> ISSN 1634(0655) (2004).
- [2] A. Caruso, A. Martin, A. Leal de Sousa, S. Bellet, E. Martino and G. Mignot, "Numerical study of the flow into the upper plenum and the hot legs of a 1300 PWR : assessment of experimental model", *Nureth 9, San Francisco, October 3-8, 1999*.
- [3] J-P. Chabard and D. Laurence "Heat and fluid flow simulations for deciding tomorrow's energy", *Proceedings of Turbulent, Heat and Mass Transfer*, 14-18 September, Rome, Italy, 2009.
- [4] S. Bellet and S. Benhamadouche "Impact of rotating and secondary flows on PWR primary loops, CFD might bring some light on the flow behavior", *proceeding of Icone 18, Xi'an, China, May 2010*.
- [5] J.J. Derksen (2005), "Simulations of confined turbulent vortex flow". *Computers & Fluids* 34, 301–318.
- [6] F. E. Erdal and S. Shirazi, "LDV Measurements of a swirling flow in a cylinder with one tangential inclined inlet", *Proceedings of ASME FEDSM 2001: 2001 ASME Fluids Engineering Division Summer Meeting*, New Orleans, Louisiana, May 29-June 1, 2001
- [7] A. Gupta and R. Kumar, (2007) "Three-dimensional turbulent swirling flow in a cylinder: Experiments and Computations", *International journal of Heat and Fluid Flow*, 28 (2007) 249–261, 2007
- [8] S. Jakirlić, K. Hanjalić and C. Tropea, (2002) "Modeling Rotating and Swirling Turbulent Flows: A Perpetual Challenge", *AIAA J.*, Vol. 40, No. 10