EFFECTS OF THE UPSTREAM ELBOWS FOR THERMAL FATIGUE STUDIES OF PWR T-JUNCTION USING LARGE EDDY SIMULATION

Thomas PASUTTO  
EDF R&D Fluid Mechanics and Heat Transfer  
6, quai Watier, 78401, Chatou, France  
thomas.pasutto@edf.fr

Christophe PENIGUEL  
EDF R&D Fluid Mechanics and Heat Transfer  
6, quai Watier, 78401, Chatou, France  
christophe.peniguel@edf.fr

Jean Michel STEPHAN  
EDF R&D Mechanics Material and Components  
Les Renardieres, 77818, Moret sur Loing, France  
jean-michel.stephan@edf.fr

Keywords: T-junction, RHR, Large Eddy Simulation, Upstream Elbows

ABSTRACT

Thermal fatigue of the coolant circuits of PWR plants is among the major issues for nuclear safety. The problem is very acute in mixing zones, like Tee junctions. In these zones, the conjunction of high levels of turbulence with large water temperature differences can lead to large thermal fluctuations at the wall. Studies on the subject have been already done at EDF using an unsteady conjugate heat transfer approach (Peniguel et al., 2003), (Sakiz et al., 2004). The fluid flow was solved with a Large Eddy Simulation technique in order to deal with the unsteady aspects of the problem which may cause large mechanical stresses. However at that time, due to CPU cost required by these simulations, the computational domains were limited to a small part of the T-junction, and simplified inlet conditions were used for the velocity at the inlet.

Advances in computer capabilities allow now to extend the geometry upstream of the junction. The purpose of this paper is to investigate some of the effects induced by the upstream elbows. Three simulations of the former Residual Heat Remover (RHR) junction found in N4 type PWR nuclear plants, are done with the CFD tool Code_Saturne developed at EDF (conjugate heat transfer is not considered in the present study). The results of the computation including the upstream elbows are compared with the results of two simulations using simplified velocity profile and synthetic turbulent inflow conditions.

In the present study, upstream elbows have little influence on the flow dynamic. As expected, the mean and RMS temperature field are twisted when elbows are taken into account, but, as it was found in previous studies (Peniguel et al., 2003), these new computations confirm that no specific frequency seems to appear downstream of the T-junction.

1. INTRODUCTION

Civaux Reactor was shut down in may 1998 following a leak of primary coolant from a pipe in the Residual Heat Removal (RHR) system. The cracks discovered in the pipework were clearly the result of thermal fatigue caused by turbulent mixing of hot and cold flows. Since this incident, RHR pipeworks have been redesigned and replaced at all the EDF's N4 units, and large research programs were launched to address this issue.

To explain why a failure occurred, experimental approaches have been extensively used with both, real scale mock-up and on site measurements. But theses approach may become quite costly or difficult to achieve, when dealing with high temperature, pressure or potentially dangerous environments (as it is the case in nuclear industry). Moreover, on site experimental approaches often lack the flexibility needed when a sensitivity study is desired.

For these reasons, numerical investigations have been done at the same time to gain some understanding of the phenomena taking place in mixing zone. These simulation were done using an unsteady and 3D approach : the fluid flow was solved with a Large Eddy Simulation technique, and thermal coupling between the fluid and the solid, as well as the thermal conduction inside the wall was taken into account.
Due to the cost of a CFD approach using Large Eddy Simulation, just a small portion surrounding the location where cracks have occurred was simulated, and simplified velocity inlet conditions were used for these simulations. Advances in computer capabilities allow now to extend the size of the computational domain. The aim of this study is to investigate some of the effects induced by the elbows located at the upstream of the residual removal system. The simulations results obtained with an extended domain (including the upstream elbows) are compared with the simulations results obtained with a truncated computational domain and simplified inlet velocity profile. Two strategies are used to estimate the shape of the simplified inlet velocity profile. The first one deals with the use of well-know turbulent pipe flow correlations, while the second one involves RANS precursor computations of the flow in the upstream elbows.

2. NUMERICAL APPROACH

The characteristics of the flows in mixing zones usually involve a Reynolds numbers. Hence, Direct Numerical Simulation is totally out of reach of the most powerful computers. On the other hand, Reynolds Averaged Navier Stokes models (like ke or more advanced models) can deal with such flows quite easily. Yet, they sometimes lack accuracy in specific complex areas of the flow, and most of them can only yield mean or low order averaged quantities, and totally overlook the instantaneous aspects, which are essential for thermal fatigue studies. In the last few years, the growth of computer capacity has made mixing zones accessible to Large Eddy Simulation techniques. In LES, only the smaller scales of turbulence are modeled, whereas the large energy carrying structures are computed. Thus, LES provides 3D time dependent solutions, on which any kind of signal processing can be done, at much higher Reynolds numbers than DNS.

2.1 The CFD Tool Code_Saturne

The EDF finite volume CFD code, Code_Saturne, is used to solve Navier-Stokes equations (1) on unstructured meshes. The flow is assumed incompressible and Newtonian and the density is only a function of temperature. Using LES, the filtered Navier-Stokes equations can be written as follows (the filtering operator is omitted for clarity sake), together with the equation for the temperature.

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \rho U_i \frac{\partial U_i}{\partial x_j} &= 0 \\
\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} &= -\frac{\partial p}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + (\rho - \rho_0) g_i \\
\rho \frac{\partial T}{\partial t} + \rho U_i \frac{\partial T}{\partial x_j} &= -\frac{\partial}{\partial x_j} \left[ \lambda \frac{\partial T}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left( \frac{\mu_f}{\sigma} \frac{\partial U_i}{\partial x_j} \right)
\end{align*}
\]

In these equations, \( U_i \) are the filtered components of the velocity, \( p^* \) stands for the pressure (minus the reference hydrostatic pressure), \( \rho \) is the density, \( \mu_f \) and \( \mu_s \) are the molecular and subgrid-scale viscosities, \( T \) is the temperature, \( \lambda \), \( C_p \), and \( \sigma \) are respectively the fluid conductivity, specific heat and turbulent Prandtl number. It should be noted that in RANS approaches, the equations are the same, but the turbulent viscosity models the turbulent effects on the whole energy spectrum (whereas in LES the sub-grid viscosity only represents the small scale structures associated to the high frequencies fluctuations). In our computations a Smagorinsky model has been used in LES calculations, and standard k-\( \omega \) model has been used for RANS calculations. More details on the turbulent models available in Code_Saturne can be found in (Arachambeau et al., 2004) and (Benhamadouche et al., 2002).

Before taking on the present study, Code_Saturne has been tested on a large number of academic and industrial cases (homogeneous turbulence, channel flows, tube bundles, T-junction, coaxial jet, ...).

2.2 Numerical Technique Used to Solved the Fluid Equations

In the collocated finite volume approach used in Code_Saturne, all variables are located at the gravity centers of the cells (which may take any shape). The momentum equations are solved by considering an explicit mass flux (the three components of the velocity are thus uncoupled). Velocity and pressure coupling is insured by a SIMPLEC prediction/correction method with outer-iterations (Ferziger, Peric, 1999). The Poisson equation is solved with a conjugate gradient method. The collocated discretisation requires a Rhie and Chow interpolation in the correction step to avoid oscillatory solutions (Rhie, Chow, 1983). This interpolation has been used in the present application, although it doesn't seem essential for unstructured meshes. For LES calculations, second order schemes are used in space (fully centered scheme for the velocity components, centered scheme with slope test for the temperature) and time (Crank-Nicolson with linearised convection). A second order Adams-Bashforth method is used for the part of the diffusion involving the transposed velocity gradient, to keep the velocity components uncoupled. For RANS calculations, second order schemes are used in space (centered with slope test for all the variables), and a first order implicit scheme is used in time.

2.3 Wall Treatment

Due to the phenomena affecting the large energy-carrying eddies near the wall, well resolved LES of wall flow become very expensive at high Reynolds number. Then methods to bypass the wall layer are required to perform simulation of industrial interest, at a reasonable cost. Many issues have been developed in the RANS context to model boundary layers flows. Ours computations rely on the same regular wall-function. As the grid is very coarse near the wall in comparison of the typical size of the turbulent structures of the boundary layer, cells adjacent to the wall contain a large number of eddies. The idea is then to consider the sample of near-wall eddies large enough, for statistical arguments to be usable (only theirs average effect of the vortices remain appreciable). The validity of this assumption has been checked with LES of channel and periodic hill flows. These simulations confirm that standard wall functions have difficulties to deal with recirculating...
zones, and may, in specific areas, lead to a large error in velocity prediction. Although they generally work quite well, and provides proper results in the major part of the flow where boundary layers are attached. More details on the wall function available in Code_Saturne can be found in (Montfort et al., 2006). One can notice that wall functions are no required for the temperature : conjugate heat transfer is not considered in the present study, and null heat flux conditions are directly applied at the wall.

2.4 Inlet Treatment

In RANS calculation, the inlet velocity conditions may be given with specification of a mean velocity profile and a level of turbulent kinetic energy (or a profile when available). In LES, the resolved variables are related to the turbulent fluctuations, and instantaneous values are required at the inlet. As the temperature is uniform at the inlet, inlet conditions for the temperature are obvious. Conditions for the velocity are more complex, because velocity fluctuations have a spatial and temporal correlation. During the last years, several attempts to generate turbulent inlet condition for LES have been made. The most widely used approach consists in adding a random perturbations to the mean velocity. The method, however, is known to be less suited for cases where inflow condition play a major role in the development of boundary layer.

Recently, a different method has been integrated in Code_Saturne to generate fluctuations that, unlike the random technique, provides spatial and temporal correlations. The approach is based on a vortices generation technique. A short description of the method is given here, and more details can be found in (Jarrin et al., 2003). At the inlet, velocity \( \vec{U} \) is divided into a mean part \( \langle \vec{U} \rangle \), and a fluctuating part \( \vec{u}' \) which is assumed null in the streamwise direction. Starting from the analytical expression of the velocity field \( \vec{U}(\vec{x}) \) induce by a single vortex located at the point \( \vec{x}_k \) :

\[
\vec{u}'(\vec{x}) = \sum_{k=1}^{N} \alpha \tau \left( 1 - \exp \left( \frac{-|\vec{x} - \vec{x}_k|}{2a^2} \right) \right) \exp \left( \frac{|\vec{x} - \vec{x}_k|}{2a^2} \right) \left[ \frac{\vec{u}'(\vec{x})}{\| \vec{u}'(\vec{x}) \|} \right] \left[ \vec{x} - \vec{x}_k \right]
\]

\( \vec{u}' \) is considered as the velocity field generated by many vortices :

\[
\vec{u}(\vec{x}) = \sum_{k=1}^{N} \vec{u}_k(\vec{x})
\]

where \( \vec{x} \) is a point of the inlet plan, \( \alpha \) is the vortex diameter \( \Gamma \) its intensity, and \( N \) the number of vortices contained in the inlet plane. One can notice that, unlike of the random fluctuations, \( \vec{u}' \) is divergence free. At the first step time, vortices position are given randomly, and moved after following the relation bellow :

\[
dx_{\vec{x}}{dt} = \sum_{k=1}^{N} \vec{u}_k(\vec{x})
\]

To avoid development of large unphysical structures in the streamwise direction, vortices are destroyed and regenerated after a time \( \tau \). \( \Gamma \) is defined in order that velocity fluctuations reach statistically the value of the turbulent kinetic energy specified at the inlet. Sizes and lifespan of the vortices are estimated following the relation :

\[
\alpha = \frac{k^{1/2}}{\varepsilon} \quad \text{and} \quad \tau = \frac{k^{1/2}}{\varepsilon \langle U' \rangle}
\]

\( k, \varepsilon \) and \( \langle U' \rangle \) are respectively the turbulent kinetic energy, the dissipation and the mean streamwise velocity. These values are the only data that the method required.

3. CONFIGURATION AND SIMULATION PARAMETERS

3.1 Physical Parameters

Fluid is water at a pressure of 36 bars. In this study, the horizontal hot branch and the vertical cold branch are set respectively at a temperature of 20°C and 180°C. The total flow rate is set to be 550 m³/hour and the velocity ratio between the cold flow branch and the total flow rate is 20%.

The physical properties of the fluid and the solid (density, viscosity, diffusivity, ...) are calculated using polynomial approximations that were defined by regressions over a sample of values taken from an in-house database, in the considered range of temperatures.

3.2 Meshes

The global geometry of the RHR circuit is presented in figure 1. In order to study effects induced by the upstream elbows, the fluid part is divided into two sub-domains which are colored in two different gray levels on the figure 2.

- The first one surrounds the T-junction and the location where cracks occurred. Mesh of this domain (Mesh 1) is identical to the mesh used in the previous study of the junction. It contains roughly 400 000 cells and was created with the mesh generator Gibi and Ideas.

- The second sub-domain represents the sequence of pipeworks and elbows which are located at the upstream of the cold and hot branch (Mesh 2 and 3). These meshes are created from an extrusion of the T-junction 2D discretisation in a section with the mesh generator Simail. They contain roughly 365 000 cells for the hot line, and 85 000 cells for the cold line.

The pipework diameter is 254 mm, and the dimensions of the circuit are mentioned in figure 3 and 4.
Figure 2: View of the different computational domains

Figure 3: Dimensions of the RHR circuit (left view)

Figure 4: Dimensions of the RHR circuit (front view)

Figure 5 presents the global mesh corresponding to the whole domain. As it can be seen, all cells are hexahedric, and the discretisation is rather fine both normally and azimuthally. Figure 6 shows close views which underline how fine and uniforms the cells in all the directions are. However, in the present study, small bumps corresponding to the weld seam have been omitted. With this refinement, distance from the wall of the first calculation point in the fluid can be as large as 1400 wall units.

3.3 Inlet Strategy

Three computations have been done.
- The first one uses the global mesh including the T-junction and the upstream elbows. The fluid domains are automatically merged by Code_Saturne which allows to use separated meshes. Inlet conditions are specified on the plan HB0 and CB0 which are defined on the figure 2. Although it probably has not many influence, vortex method is used at each inlet. Indeed, in this simulation, pipe length is long enough for the flow to become established, and details of the inlet conditions, have probably no influence on the flow dynamics beyond a distance. The mean velocity, kinetic energy and dissipation profile used at the inlet, are evaluated in this computation from experimental correlation of fully developed turbulent pipe flow. Details on the correlation can be found in (Boyer, Laurence, 2002).
- The second calculation uses only the first fluid domain (Mesh 1). Inlet conditions are specified just upstream of the plane HB1 and CB1. Mean velocity, turbulent kinetic energy are specified with the same correlations than the one used in the previous case.
• The third calculation is also done with the truncated fluid domain, but mean velocity and others data used by the vortex method are evaluated with RANS precursor computations of the upstream flows. As the RANS simulation time step can be ten times greater than the LES time step, mean flow data are quick and easy to obtain.

Tables 1 gives a summary of the mesh and inlet conditions used in each computation. The figure 7 shows a snapshot of the typical instantaneous velocity fields specified at the inlet in the computation 2 and 3 (small vortices are visible). One can already notice that means velocity fields are not symmetrical in the third case, and show evidences of mean tangential velocity.

<table>
<thead>
<tr>
<th>Case</th>
<th>LES Mesh</th>
<th>Inlet data used by the vortex method</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1 + 2 + 3</td>
<td>Fully turbulent pipe-flow correlation</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>Fully turbulent pipe-flow correlation</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>Precursor RANS computation using the meshes 2 and 3</td>
</tr>
</tbody>
</table>

Table 1 : Summary of the inlet data and mesh used for LES computation

4. COMPUTATIONAL RESULTS

Initial conditions present no difficulties, and are wiped out very quickly. In the first computation, 2 seconds are needed for a particle to be convected from the inlet to the outlet. 30 seconds of physical time have been simulated for this case. This simulation correspond to a large computational effort (around 500 hours of CPU on a on 16 parallel Opteron processors). Time averages are calculated for this case over the last 20 seconds. With the truncated fluid domain, less than 1 second is needed to wipe out the transient state. 8 seconds of physical time have been simulated for the computations using the truncated mesh, and time averages are computes over the last 6 seconds.

4.1 Results With the Upstream Elbows

Figure 8 presents instantaneous and mean temperature fields in the symmetry plan of the T-junction for case 1. One may notice the strong irregularity of the stratification. Figure 9 and 10 present respectively the mean and the RMS temperature (obtained by time averaging of the instantaneous temperature field). These figures indicate that a strong mixing takes place downstream of the T-junction. Figures 11 and 12 present the evolution of the mean and RMS temperature in the four sections ZM1 to ZM4. One may notice that mean fields are not symmetrical, and RMS temperature decreases for the sections located further of the junction.
Figure 11: Case 1 - Mean temperature in the sections MZ

Figure 12: Case 1 - Temperature RMS in the section MZ

Next figures focus on the unsteady flow dynamics aspects. Figure 13 presents time evolutions and Spectral Power Density (SPD) of the normal velocity at the center of the sections HB1 and CB1. Figure 14 and 15 present similar plots for the temperature in the sections MZ2 and MZ4. Three probes are considered in the section MZ2. One is at the center, the two others are located near the wall on the extrados and intrados side of the junction. Only the two last probes are considered in the section MZ4. As expected, due to the mixing, the fluid temperature fluctuations are globally much smaller in the section MZ2 than in the section MZ4. One can underline that no specific frequency appears in the spectra. These results seem to be in contradiction with the computation recently done by several authors (Chapuilot et al., 2005). One cannot find on the spectra, evidence of large velocity pulsations upstream of the junction, and velocity fluctuations as well as temperature fluctuations, involve a wide range of frequencies (like turbulent phenomena).

Figure 13: Case 1 - Velocity time evolutions and SPD in the sections HB1 and CB1

Figure 14: Case 1 - Temperature time evolutions and SPD in the section MZ2

Figure 15: Case 1 - Temperature time evolutions and SPD in the section MZ4

4.2 Results With the Truncated Domain

Figure 16 and 17 present mean temperature and RMS temperature fields in the middle plan for the second and the third computation. Figure 18 and 19 present the evolution of theses quantities for the sections taken in the mixing zone. These figures point out some of the effects induced by the upstream elbows. As expected, the fields are nearly symmetrical in this computation (small asymmetries are still noticeable, probably due to the relatively short duration of the time averaged in this computation). The hot flow moves away from the intrados beyond the section ZM1, and maximum RMS temperature is around 5% lower in the section MZ2 and MZ3 than in the first computation. Differences are also noticeable on the figure 20 which shows comparison of the mean velocity field in the section HB1 and CB1 between the different calculations. One can notice on this picture the pretty strong secondary motion induce by the upstream geometry, especially in the hot branch where the flow is swirled by the elbows sequences.
Concerning the third calculation, one can notice that mean velocity effects seem pretty well reproduced when using RANS precursor inlet data (figure 20). Unfortunately, agreement is no so good further, as shown on the figure 21 which presents comparisons of the mean and RMS temperature near the wall. Predictions of the mean temperature are in better agreement with the results of the first computation (especially near the intrados), but the mixing remains globally too weak, and the stratification is stiffer at the outlet than in the first computation. This phenomenon is probably due to the weakness of the turbulent kinetic energy level in the precursor RANS calculation, compared to that found in the first calculation.

Figure 20: Mean tangential velocity colored by the mean normal velocity in the section HB1 and CB1

5. CONCLUSIONS

This paper has investigated the effect induced in the scope of the thermal fatigue studies, by the upstream geometry of the former RHR T-junction. Large Eddy Simulations of the flow have been done with the EDF’s CFD tool Code_Saturne using a “full” and a “truncated” mesh, in order to see how mean and fluctuating quantities are affected when upstream elbows are neglected.

This work confirms the major conclusions of the previous study done RHR T-junction (Peniguel et al., 2003), although the computations are similar but not completely identical (the total flow rate is 550 m$^3$/h instead of 1000 m$^3$/h and conjugate heat transfer is not considered in the present study). As expected, mean and RMS temperature fields are twisted when upstream geometry is taken into account, and thermal fluctuations are slightly more important on one side of the T-junction. However, one can underline that no specific frequency appears, and one cannot find evidence of large velocity pulsations triggered by the elbows at the upstream of the junction (fluctuations involve wide range of frequencies). This result is similar to that of our previous computation, but seem in disagreement with the conclusions suggested by several authors (Chapuliot et al., 2005).
Computation of the upstream flow with RANS precursor calculations allows to save a significant part of the CPU time required by the LES including the upstream elbows. Prediction of the mean temperature is reasonably correct, but predictions of RMS temperature are too weak.

Complementary works are planned at EDF to investigate other configurations (like T-junction with unequal diameters), and special effort are likely to be devoted to improve CPU aspects and the near wall treatment.

### 6. NOMENCLATURE

- $U, u_i$: Velocity, velocity component (m/s)
- $T$: Temperature (°C)
- $p^*$: Pressure (without hydrostatic part) (bars)
- $\rho$: density (kg/m$^3$)
- $g$: Gravity, gravity component (m/s$^2$)
- $\mu_e$: Molecular viscosity (kg/ms)
- $\mu_t$: Subgrid-scale / turbulent viscosity (kg/ms)
- $\lambda$: Fluid conductivity (kg m$^3$ °K)
- $C_p$: Specific heat (m$^2$/s$^2$ °K)
- $\sigma$: Turbulent Prandtl number
- $k$: the turbulent kinetic energy (m$^2$/s$^2$)
- $\varepsilon$: dissipation (m$^2$/s$^3$)
- $\Gamma$: intensity of the velocity fluctuations (m/s)
- $\alpha$: Size of the vortices (m)
- $N$: Number of vortices
- $\tau$: Life time of the vortices (s)

### 7. REFERENCES

Peniguel C., Sakiz M., Benhamadouche S., 2003, “Presentation of a numerical 3D approach to tackle thermal striping in a PWR nuclear T-Junction”, Proceeding ASME-PVP, Cleveland.


