SIMULATION OF ATTENUATION OF THERMAL FLUCTUATIONS NEAR A PLATE IMPINGED BY JETS

Jan-patrice SIMONEAU          Odile GELINEAU
Design and thermal hydraulics department  Design and thermal hydraulics department
Novatome – Framatome                  Novatome – Framatome
10, rue Juliette Récamier            10, rue Juliette Récamier
69456 LYON cedex06                     69456 LYON cedex06
France                                France
Ph. : 33 4 72 74 73 75                 Ph. : 33 4 72 74 72 26
Fax : 33 4 72 74 73 25                  Fax : 33 4 72 74 73 25
e-mail : jpsimoneau@framatome.fr    e-mail : ogelineau@framatome.fr

KEY WORDS : Fluid dynamics – Turbulence – Thermal striping.

ABSTRACT

In nuclear reactors, and especially in liquid sodium cooled ones, the combination of temperature differences inside cooling fluid, turbulent flows and high heat transfer coefficients is a potential source of the thermal striping process. Such a phenomenon has been studied for several years by using Large Eddy Simulation models (ref. 1). The present paper focuses on the attenuation of the thermal fluctuations in the boundary layer. The behaviour of the fluid within this zone drives the temperature amplitude reduction. Its knowledge is therefore of prime importance for subsequent mechanical analyses. The aim of this paper is to present a Large Eddy Simulation model implemented in the Star-cd code, to treat a test case for which experimental data are available. The test case chosen is a Japanese water experiment (ref 2) : two parallel jets at different temperatures impinge a flat steel plate. The Star-cd code and the Large Eddy Simulation model are first described. The experiments is then briefly presented. Finally, results and analysis are drawn.

INTRODUCTION : 

In nuclear reactors, and especially in liquid sodium cooled ones, the thermal striping process may occur because of the combination of temperature differences inside cooling fluid, turbulent flows and high heat transfer coefficients. Such a phenomenon has been studied for several years by using Large Eddy Simulation models (ref. 1). The present paper focuses on the attenuation of the thermal fluctuations in the boundary layer. The behaviour of the fluid within this zone drives the temperature amplitude reduction. Its knowledge is therefore of prime importance for subsequent mechanical analyses. The aim of this paper is to present a Large Eddy Simulation model implemented in the Star-cd code, to treat a test case for which experimental data are available. The test case chosen is a Japanese water experiment (ref 2) : two parallel jets at different temperatures impinge a flat steel plate. The Star-cd code and the Large Eddy Simulation model are first described. The experiments is then briefly presented. Finally, results and analysis are drawn.

1- THE STAR-CD CODE :
The general-purpose code Star-cd for fluid mechanics and thermics (ref. 4) is used. The main characteristics of this software are listed below :
- 3 dimensional code.
- Finite volume formulation.
- Unstructured grid capabilities including advanced features : local refinement, non-conformant grids with arbitrary interfaces. Meshes are mainly hexahedrons and prisms.
Matrices inverted by bi-conjugate gradient method with preconditioning.
PISO algorithm for pressure linked equations (the pressure solver is semi-implicit: pressure fields are corrected at every iteration in order to achieve both momentum and continuity balances).
Numerical schemes up to third order.

The Navier-Stokes and energy equations are solved in primary variables (velocity, pressure and temperature). For this specific study, computations are run in transient regime. The turbulence is taken into account as described below.

2- THE SUBGRID SCALE MODEL
In a turbulent flow, one can observe all-sized eddies, so that this flow can be represented by a continuous spectrum of eddies ranging from great structures (order of magnitude of the domain) to the smallest eddies (of the Kolmogorov scale order). An explicit calculation of all the eddies is not reachable in a 3D configuration and the Large Eddy Simulation so proposes to compute the large eddies until the discretisation mesh size, while the other ones (subgrid scale eddies) are appropriately modelled.

The subgrid scale model here used is the selective structure function model (ref. 5, 6) with adaptations. In the structure function model, a “box” filter is used to split each flow variable into a large scale part (solved via Navier-Stokes and energy equations) and a subgrid part. The subgrid contribution is proportional to the spatial correlation of the velocity between two points (called structure function) and introduced in the Navier-Stokes and energy equations via an additional turbulent viscosity \( \nu_t \). The selective feature is that this viscosity is applied only if the flow is “sufficiently” 3D, i.e. if the angle between vorticity vector at the considered point and the averaged vorticity vector of the neighbouring points is large enough. Note that the magnitude of this viscosity is much lower than the turbulent viscosity calculated by a k-\( \epsilon \) model.

The adaptation of this model consists of:

- The turbulent viscosity is set to zero in the viscous sublayer, the thickness of this sublayer is evaluated by \( y^+ < 3 \).

In the grid (presented later), local refinement implies locally very high size variations of meshes. A threshold is set to limit high non physical values of the turbulent viscosity is those peculiar zones. The threshold is \( \nu_t / \nu < 15 \), where \( \nu \) is the molecular viscosity.

3- THE WATER TESTS:

Those tests has been conducted in Toshiba Japanese laboratories (see ref. 2) using the apparatus shown below. The configuration represents two parallel jets (one hot and one cold) impinging a flat plate. Both hot and cold fluids are delivered from nozzles to the steel test piece. The nozzles and the test piece are immersed in a water tank (see fig. 1). The temperatures are measured with thermocouples (see figure 1) both:

- on the steel plate surface,
- inside the fluid at 2 mm from the wall

The 0.25 mm diameter thermocouples allow short time constants: 20 ms for fluid measurement and 30 ms for wall.
The nozzles are almost rectangular (5 x 18 mm) and separated by 19.1 mm.
The test retained for the present study is characterised by:

- \( T_{\text{hot}} = 46^\circ \text{C}, T_{\text{cold}} = 15^\circ \text{C} \)
- Distance nozzle outlet – plate = 38 mm
- Jet outlet velocity = 3.36 m/s

4- THE NUMERICAL MODEL:

The geometrical domain extends up to the tank walls, but also represents precisely the nozzles. Furthermore, the boundary layer along the plate is discretised very precisely. The number of meshes remains reasonable (110 000 cells) thanks to the local refinement feature allowed by the Star-cd code.
The figure 2 below shows the total domain, made of large meshes at the periphery but including fine ones near the jets as seen on figures 3 and 4. The boundary layer thickness is less than 1 mm along the plate. The grid is refined near the plate and the first mesh is located \( 4 \times 10^{-3} \) mm, i.e. \( y^+ \approx 0.8 \).

Figure 3 shows also the boundary conditions of the model:

- Hot and cold inlets: velocity and temperature prescribed, no turbulence.
- Outlet: located behind the plate, zero normal gradients for each variable.
- Walls: Laminar friction.

The physical properties of the fluid are water characteristics and taken as constants at mean temperature, except for the laminar viscosity. For that property, decreasing with temperature, a linear function of temperature is used: \( \mu = 147 \times 10^{-3} - 1.94 \times 10^{-4} \) T. (T in °C and \( \mu \) in Pa.s).
The coupled conduction in the test piece is computed simultaneously. Effects of steel thermal inertia on neighbouring fluid is therefore taken into account. This phenomenon may be not negligible in the attenuation process.
5- RESULTS:

A preliminary steady computation is first performed in order to get a mean initial field. For this steady case only, a standard k-ε model is used.

The transient run is then launched. The time step must be chosen (by the user) to keep the CFL criterion less than about 10 to 50. The value of $10^{-3}$ s satisfies this criterion (the normal velocities near the plate, where meshes are very fine, are very low).

The CPU and physical times are:

<table>
<thead>
<tr>
<th>CPU time (450MHz Sun workstation)</th>
<th>Physical time</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 days 12 s</td>
<td></td>
</tr>
</tbody>
</table>

The first times of the transient run correspond to the establishment of the fluctuations.

5-1 Mean temperature profile:

Figure 5 presents the longitudinal profile of mean temperature along the plate surface and shows a good agreement in the region of impact with the experimental results. (Numerical results are plotted only in the region of fine grid).

5-2 Temperature fluctuations:

5-2-1 Amplitude:

Figure 6 presents the time history of the temperature field in the test piece and in the water between the nozzles and this plate. The plots are separated by 0.1 s (temperature in K). One sees:

- the instability along the hot jet (same instability occurs also along the cold one),
- slight moving of jets and subsequent alternation of cold and hot eddies near the plate between the two jets,
- slow variation of wall temperature

The time history of temperature at one location is reported on figure 7 for both wall and fluid (zoom on 2 seconds). The temperature is normalised with the maximum temperature difference, and is expressed in %.

The longitudinal profile of temperature fluctuations on wall surface (peak to peak value normalised and expressed in %) is plotted on figure 8. For the central point (mid point between hot and cold jets), the wall and fluid peak to peak fluctuations are reported below:

<table>
<thead>
<tr>
<th>Peak to peak amplitude (%)</th>
<th>Wall</th>
<th>Fluid outside boundary layer (thickness &lt; 1 mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LES computation</td>
<td>42.5</td>
<td>90</td>
</tr>
<tr>
<td>Experiment</td>
<td>40.5</td>
<td>85</td>
</tr>
</tbody>
</table>

The fluctuations computed by the LES model are found a bit larger than experimental ones both for wall and fluid temperature (comparison performed only in the region of fine meshes). Nevertheless the attenuation is around 55% for both LES computation and experiments.

5-2-2 Frequency:

The frequencies of the fluctuations are assessed thanks to the "zero-cross method" in the ref. 2. Hence, the same method is used here. The table 3 shows an acceptable prediction of the main frequency for the wall surface temperature.

<table>
<thead>
<tr>
<th>Main frequency (Hz)</th>
<th>LES results</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>7.5</td>
<td>5 – 7</td>
</tr>
</tbody>
</table>

The spectrum of the instant temperature at mid point is plotted on figure 9. The peak of the main frequency is pointed out (7.5 Hz), it corresponds to the pulsation of the jets, seen on the history plots of figure 6.

5-3 Other calculations:

In order to evaluate the influence of some parameters such as the LES model or the mesh size, other computations have been run. They are not detailed here but the main conclusions are the following:

- A pseudo Direct Navier-Stokes simulation (DNS) has been performed with the same fine mesh. The difference with the LES approach is that no subgrid model is used. Large eddies are solved explicitly on a fine mesh but the influence of small eddies is not taken into account via a turbulent viscosity. The results show:
  - Under evaluation of the amplitude of temperature fluctuations, (33 %)
  - Over evaluation of the main frequency (16.5 Hz)
  - Rather good mean temperature profile, but not as good as LES one.

- A second LES computation has also been run on a coarser mesh (the first point in the boundary layer is at $y^+ = 2$ instead of 0.8 and the total number of cells is 60000 instead of 110000). The results are:
  - Good evaluation of the amplitude of fluctuations (41 %),
  - Good mean temperature profile (not as good as fine mesh one)

Over evaluation of main frequency (12 Hz) but better result than DNS
CONCLUSION:

The present paper shows the capability of computational fluid dynamics codes to assess the turbulent thermal fluctuations on the reactor component walls due to the thermal striping phenomenon. A fine discretisation of the boundary layer and especially of the viscous sublayer, and the coupled conduction within steel plate permits a good evaluation of the thermal attenuation near the wall showing the benefit brought by the LES model. Other configurations have to be investigated (mixing tees, fluctuating stratification interfaces, ... ) and influence of subgrid numerical modelling may also be carried on. Hence, such a work needs to be continued by taking advantage of future thermal striping experiments and modelling possibilities of computers.

REFERENCES


NOMENCLATURE:

\( k \) turbulent kinetic energy (m\(^2\)/s\(^2\))
\( \varepsilon \) rate of dissipation of \( k \) (m\(^2\)/s\(^3\))
\( y^+ \) non dimensional distance from wall
\( \mu \) dynamic molecular viscosity (Pa.s)
\( \nu \) kinematic molecular viscosity (m\(^2\)/s)
\( \nu_t \) turbulent viscosity (m\(^2\)/s)
\( T \) temperature (°C or K)
\( T_{\text{hot}, \text{cold}} \) temperature of hot, cold jet
\( \Delta t \) time step (s)
LES Large Eddy Simulation
DNS Direct Navier-Stokes simulation

Figure 1 – Experimental set-up
Figure 2 - grid

Figure 3 – grid - detail

test piece
nozzles outlets

Figure 4 – grid – zoom on nozzles and test piece

test piece
Cold jet nozzle
Hot jet nozzle
Normalised temperature %

Figure 5 – mean temperature longitudinal profile
Figure 6 – sequential plots of temperature field (Δt = 0.1 s)
Figure 7 – $T = f(t)$ – wall and fluid computed temperatures

Figure 8 – Peak to peak fluctuation amplitude along plate