NUMERICAL ANALYSIS OF THERMAL STRATIFICATION PHENOMENON IN BENT PIPES

Marco Pellegrini, Hiroshi Endo and Hisashi Ninokata

1 Tokyo Research Laboratory for Nuclear Reactors, Tokyo Institute of Technology
2-12-1 Ookayama, Meguro-ku, Tokyo, 152-8550, Japan
*pellegrini.m.aa@m.titech.ac.jp

2 Japan Nuclear Energy Safety Organization (JNES)
3-17-1, Toronamon, Minato-ku, Tokyo, 105-0001, Japan

ABSTRACT

During protected loss of flow (PLOF) accidents in fast breeder reactors the temperature at the core outlet experiences variation depending on the pump coastdown. In case of long flow coastdown the temperature at the core outlet will decrease and sodium stratification will occur in the upper plenum due to the effect of gravity. The geometrical characteristics of nuclear reactor such as Monju (presence of shrouds with circumferential holes), can moreover create currents with different temperature which move towards the hot-leg piping system. As a result uneven temperature distribution is likely to appear at the inlet of the pipe which can set the conditions for occurrence of thermal stratification in horizontal piping.

In the present work CFD tools are employed as a validation of an experimental benchmark for the understanding of essential phenomena occurring in a thermally stratified flows in pipes with bends. Moreover the discussion of such events is connected to their influence in association with natural convection establishment inside the reactor.

Finally the numerical results point out the impact of the chosen turbulence modeling on the prediction of the experimental data and discussion is provided as an attempt to motivate the encountered issues as a starting point for a future development.

1. INTRODUCTION

Thermal stratification occurs when two fluid with different densities come into contact and, under the effect of buoyancy forces, the lighter fluid will tend to move above the heavier one. At first studied as a geophysical phenomenon in relation to river estuaries, cold air currents and muddy streams, the stratification phenomenon has assumed large importance in the engineering field in relation to chemistry and nuclear fields.

If thermal stratification is created inside the horizontal piping system of a nuclear reactor because of uneven temperature distribution at the inlet, two density currents moving with different velocity will be created, in which the hotter and faster front will occupy the upper region of the pipe. This phenomenon will affect the thermal stresses in the piping system due to fluid-wall interaction and differential circumferential stresses will be created impacting on the piping integrity.

Another aspect related to thermal stratification is that, since further stresses between the two density currents and higher local velocities of the hot front are created, additional pressure losses can be introduced inside the system.

Furthermore it is essential to note that the piping system of nuclear reactors presents numerous bends in order to accommodate the thermal stresses during transient operations.
The above motivation led Viollet (1987a, 1987b) and Viollet et al. (1987) to set up a water experiment for stratification analysis of the Super-Phenix reactor. The domain investigated by the author is shown in Fig. 1 in which a U pipe experiences a temperature increase at the inlet.

Due to the mock-up similarities to our present purpose, the above mentioned experiment was chosen as validation of the numerical tools selected for the study. It is important to underline that Viollet (1987a) attempts a two-dimensional numerical study of the problem which will fail in the evaluation of the swirl and secondary flows introduced by the geometry, which is expected to play an important role in turbulence creation.

The above considerations therefore moved the interest in the validation of the available CFD tools in order to verify the capability of the state of art in turbulence modeling for the investigation of the role of geometry and buoyancy effect in such transients.

In the present work therefore, after the description of the physics governing the experiment in relation with the associated non-dimensional numbers, the calculation methodology applied in a commercial CFD code is delineated. Finally the results in relation with the phenomena of interest are presented and discussion is proposed about the motivation which lead to the current achievements.

### 2. EXPERIMENTAL DESCRIPTION

#### 2.1. Physics

Thermal stratification in pipes can occur in general for three main reasons:

- Non uniform temperature distribution at the pipe inlet;
- Temperature variation at the inlet (up or down-ramp);
- Heat loss through the pipe.

Because the piping system of nuclear reactors are isolated in order to avoid external thermal losses during normal operations, the third point is generally neglected in the analysis. In the experiment performed by Viollet (1987a) in order to create a discontinuity the temperature was increased (decreased) in order to create a hot-shock (cold-shock) wave in the vertical pipe which, as a...
consequence, leads to uneven temperature distribution at the inlet of the nearly horizontal section. In the case chosen as validation, the hot-shock experiment is analyzed in which the hot front reaches the horizontal pipe and it “accommodates” above the lower layer of cold fluid increasing its velocity. At the instant in which the fluid reaches the second bend the buoyancy and centrifugal forces (acting on the fluids both as body forces but in different directions), create a region of instability with wide coherent vortex which enhance the mixing of the two density currents. The effect of the two bends during the stratification will be different on the flow. In presence of gravity indeed the bending direction will have different effects on the fluid. In this case indeed, while the first bend will create a stable stratified flow, the upward bend will be instead responsible for the vortex creation.

2.2. Non-dimensional Numbers

Some non dimensional numbers control the physics of the process and they can be summed up in Table 1.

<table>
<thead>
<tr>
<th>F</th>
<th>Re</th>
<th>Pe</th>
<th>( t_\alpha )</th>
<th>R/d</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.22</td>
<td>5000</td>
<td>30,000</td>
<td>7.5</td>
<td>1.52</td>
</tr>
</tbody>
</table>

The first number, defined as Froude number \((F)\), represents the strength of the applied transient, it is defined as the ratio between the inertial and buoyancy force. In the related experiment a value of \(F\) greater than unity will lead the hot flow to occupy the bottom part of the first bend (due to high value of the inertia) and as a result instability will be created in the horizontal pipe. Since stratification might occur during PLOF accident the number of Froude is assumed to be small so that instability and creation of mechanical entrainment (wave) will not likely appear. \(Re\) and \(Pe\) number are related through the \(Pr\) number of the fluid. They show that the flow is in the transition region of turbulence and that the fluid employed is water whose \(Pr\) is almost 6, which is about 600 times greater than the sodium one. The non dimensional time \(t_\alpha\) represents the non dimensional length of the transient which accounts for the effects of the stratification. The temperature increase will cease when the hot wave will be propagated until approximately the middle of the nearly horizontal pipe and before reaching the second bend. Finally the ratio of the bend radius and the pipe diameter shows that the geometry will apply a strong curvature of the flow leading to recirculation and flow separation.

3. CALCULATION METHOD

The present work is evaluated through the commercial CFD code STAR-CCM+ 5.02.009. Hereafter the Navier-Stokes equations solution, turbulence modelization and numerical practices are explained and motivated in relation to the present case.

3.1. URANS Governing Equations

Because of the unsteady nature of the problem (i.e. unsteady boundary conditions) the analysis was held through Unsteady RANS approach where the angle brackets represent the ensemble averaging:

\[
\frac{\partial \langle u_i \rangle}{\partial x_i} = 0
\]

\[
\frac{D \langle u_i \rangle}{Dt} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \langle u_i \rangle}{\partial x_j} \right) - \langle u_i u_j \rangle \delta_{ij} + \rho \beta \frac{\partial (T - T_e)}{\partial x_i} \delta_{ij}
\]

\[
\frac{D \langle T \rangle}{Dt} = \frac{\partial}{\partial x_i} \left( \frac{\kappa \partial \langle T \rangle}{\partial x_i} \right) - c_p \langle u_i u_j \rangle \delta_{ij}
\]

In equation (1) the terms \(\langle u_i u_j \rangle\) need closure, the pressure \(P\) is expressed as \(p + \rho \beta (x - x_0)\) where \(x_0\) represents the reference altitude which is set at the pressure outlet. The body force introduced by gravity was treated as an equation of state. This equation is obtained by writing the Taylor series for \(\rho\) as a function of \(T\), considering the pressure constant and truncating at the first order. The fluid is treated with variable density and the density function is introduced in the code as a user defined function as follows:

\[
\rho(T) = \rho - \rho \beta (T - T_e)^2
\]

\(\rho = 997.561\ \text{[kg/m}^3\text{]}\), \(T = 300\ \text{[K]}\), \(\beta = 3.91427 \times 10^{-4}\ \text{[1/K]}\)
3.2. Closure Models

The proper choice of the modelization depends on the kind of transient and on the geometrical characteristics of the domain. It is well known (Baglietto, 2004) that a fluid moving in a bent pipe experiences swirl and secondary flows due to the centrifugal force created by the pipe. It was demonstrated that the basic hypothesis by Boussinesq, which assumes the Reynolds stresses as linearly dependent on the mean rate of strain, fails in case of strong streamline curvature. Non linear turbulence models (quadratic and in particular cubic formulations) are able to take into account this effect and, together with a low $y^+$ wall treatment, are shown to be successful in the prediction of the velocity field such as adverse pressure gradient and flow separation (Lien et al (1996) and Baglietto (2004)). For this reason the $k-\varepsilon$ standard cubic model, as implemented in STAR-CCM+ is employed for the evaluation. The formulation of the Reynolds stresses employed is expressed by the following equation:

\[
\begin{align*}
\langle u_i u_j \rangle & = \frac{2}{3} \delta_{ij} - v \Delta S_{ij} + C_{\nu} \frac{k}{\varepsilon} [S_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} S_{ij}] + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] \\
& + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] \\
& + C_{\nu} \frac{k}{\varepsilon} [S_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} S_{ij}] + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] \\
& + C_{\nu} \frac{k}{\varepsilon} [S_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} S_{ij}] + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] \\
& + C_{\nu} \frac{k}{\varepsilon} [S_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} S_{ij}] + C_{\nu} \frac{k}{\varepsilon} [\Omega_{ij} \delta_{j} - \frac{1}{3} \delta_{ij} \delta_{j} \Omega_{ij}] \quad (3)
\end{align*}
\]

Where $S_{ij}$ is the mean rate of strain tensor, $\Omega_{ij}$ is the mean vorticity tensor, $k$ is the turbulent kinetic energy, $\varepsilon$ is the rate of turbulent dissipation and $\mu_t$ is the turbulent viscosity. The coefficient employed are based on those proposed by Lein et al. (1996). The two scalars $k$ and $\varepsilon$ are solved through two additional equations. While we avoid writing these equations for the low $y^+$ wall treatment, which can be found in the code UserGuide, it is worth noting that a source term due to the buoyancy effect appears in both equations. This term can be written in short as:

\[
G = \beta \frac{\mu_t}{\sigma_t} \frac{\partial T}{\partial x_z} \quad (4)
\]

Where $\sigma_t$ is the turbulent Prandtl number and the term acts only in the direction parallel to gravity. In the $\varepsilon$ equation this term is multiplied by a constant defined usually as $C_{\varepsilon}$ which is specified following Viollet (1987a) as follows:

\[
C_{\varepsilon} = \begin{cases} 
1 & \text{for } G \geq 0 \\
0 & \text{for } G < 0 
\end{cases} \quad (5)
\]

Meaning that the influence of gravity on the turbulence appears only on stably stratified flows.

For the closure of the energy equation instead it was decided to employ the simplification of a constant turbulent Prandtl number equal to 0.9.

Moreover, in order to compare the effects of the improved turbulence modelization, the same case was run with the classic model $k-\varepsilon$ Realizable where the $C_\mu$ coefficient in the description of the turbulent viscosity is expressed as shown by Lumley (1978). A similar definition of the $C_\mu$ is done for the cubic model which in addition contains the contribution of the vorticity. Finally for the $k-\varepsilon$ Realizable a two-layer wall treatment was applied with the description given by Wolfshtein (1968); with this practice, as underlined by the code manual, the same mesh could be applied with different wall treatments.

In the further study $k-\varepsilon$ realizable case will be referred to as Case 1 while $k-\varepsilon$ standard cubic non-linear will be defined Case 2.

3.3. Spatial and Numerical Discretization

Grid was created with an hybrid mesh as shown in Fig. 2 in order to respect the constrains requested by Case 2, which automatically satisfy those for Case 1. For the bends and the nearly horizontal pipe a polyhedral grid was employed for its multidirectional characteristics. The inlet and the outlet pipe were extruded starting from the polyhedral mesh with 50 and 250 layers respectively. On the wall, in order to solve the boundary layer and to respect the constrains of a low $y^+$ wall treatment (in the first center cell $y^+< 1$) a prism layer with 30 layers and 15 mm thickness was created for the whole pipe (Fig. 2 b) and d)).
3.4. Initial and boundary conditions

In order to obtain a low value of the residuals during the whole transient it resulted vital the correct initialization of the velocity, pressure and temperature fields before running the effective transient. Steady state analysis cannot provide a satisfying level of convergence for the energy equation since condition of heat transfer is absent (adiabatic walls) and therefore a preliminary unsteady simulation with imposed temperature at the inlet (i.e. 300 K which is the initial temperature value assumed for the present case) was performed without the effect of gravity.

For the evaluation of the effective transient instead the inlet velocity remains constant and the temperature is increased linearly in order to satisfy the chosen Froude number (Table 1).

Moreover, even though the transient can be evaluated completely through non-dimensional numbers, for sake of clarity and better understanding of the further results, it is preferable to show the dimensional values governing the transient, which are listed in Table 2.

<table>
<thead>
<tr>
<th>d [m]</th>
<th>R [m]</th>
<th>V [m/s]</th>
<th>Tin [K]</th>
<th>ΔT [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>0.38</td>
<td>0.0178</td>
<td>300</td>
<td>6.83</td>
</tr>
</tbody>
</table>
4. RESULTS

Fig. 3 Velocity and temperature contours at non-dimensional time $2\tau$, on the symmetry plane and on three sections on the second bend at $45^\circ$, $75^\circ$, $90^\circ$ inclination. The left side of the three sections represents the internal part of the bend. Figures a) and b) refer to $k$-$\varepsilon$ realizable model while c) and d) to $k$-$\varepsilon$ standard cubic non-linear.

In this particular accident where the unsteadiness is created because of unsteady boundary condition (i.e. hot-shock) it is not possible to provide meaningful average data but results are provided in precise instants in space and time which are useful for the understanding of the flow. The author provides the experimental data at non-dimensional time $2\tau$ so that the hot wave will reach approximately 15D. This means that the front will exceed the second bend location and geometrical influence can be evaluated.

4.1.1. Velocity and Temperature Fields

Fig. 3 reveals that in the nearly horizontal part of the pipe the two fluid behave similarly and that the hotter fluid moves with a higher velocity over the colder one in a stably stratified flow. This result is similar to the two different turbulence modeling as shown in Fig. 3.

On the other side when the low density current enters the bend and penetrates the vertical and cold region, the two models produce very different results. Fig. 3 a) and b) reveal that in Case 1 the fluid after the bend keeps behaving as stably stratified flow adhering to the internal wall and that low levels of recirculation are created. This appears in contradiction with the common knowledge that in case a hot current is introduced below a more dense fluid, the current will ascend thanks to the buoyancy force, but creating oscillations in the main flow.

On the other hand results provided by Case 2 (Fig. 3 c) and d)) demonstrate that the density current shifts towards the center of the pipe influencing the creation of coherent structures which then propagate.
in the direction opposite to the gravity thanks to the buoyancy force. This has the result to create a more uniform temperature field which is what observed by Viollet (1987a).

4.1.2. Stratified Region: Fluid Entrainment

The analysis of the stratification region in the nearly horizontal pipe is essential in order to have an estimation of the length the transient and how long it persist after the duration of the imposed up-ramp.

In case of water, the relatively high value of Prandtl number (i.e. $Pr = 6$) leads to have momentum diffusivity which is more influent respect the thermal diffusivity. Therefore in the stratification region, where the diffusion of momentum is low, the heat transfer is reduced showing the temperature field as shown in Fig. 3 b) and c) (i.e. two distinct fluid regions are created).

Crapper and Linden (1973) showed that the width of the interface increases due to fluid entrainment in order to balance the heat flux occurring from the top to the bottom of the flow. If the heat flux could not be balanced by diffusion then mechanical entrainment is created and waves appear.

The investigation of this issue, which in case of water could appear of a secondary importance, will instead have a great influence in case of low Prandtl number fluid (e.g. sodium or liquid metals in general) because the interface tends to widen quicker than in the water case. Thus a proper evaluation of the length of the stratification is highly recommended for our purposes. In relation to this measuring lines are created on the plane of symmetry (Fig. 4) in order to show the evolution of the temperature profile along the pipe length for the visualization of the differences in the fluid entrainment predicted by the two models. In Fig. 5 the temperature profile in three sections along the nearly horizontal region are plotted. It can be seen that Case 1 provides the width of the interface to be almost double compared to Case 2. This difference can derive from the different definition of the turbulent viscosity and the ability of the $k-\varepsilon$ cubic to dump the turbulence in presence of laminarization of the flow, which is the case for the stratified region. Even though there are no experimental results for the width of the stratified region the prediction of Case 2 appears more reliable in particular comparing with the results of Viollet (1987a) where the temperature in the measuring points (Fig. 4) remains constant for the whole transient.

4.1.3. Temperature Comparison

Experimental data have been provided for 12 points along the pipe located at 1 cm far from the wall (Fig. 4). The comparison with experimental results is shown in Fig. 6. The results show persistence of stratification in the nearly horizontal region which was already evaluated qualitatively in Fig. 3 and which is well predicted by both models.

On the other hand in the points downstream the second bend, (namely points: 14, 15, 16 and 17 in Fig. 4 and S/D almost 15 and 20 in Fig. 5) computational data show different results from the experimental evaluations. In particular Case 1 approximates better the general behavior of the temperature distribution but it lacks of accuracy in the prediction of the internal side of the bend (point 13) because of inaccuracy in the prediction of turbulence in streamline curvature. Case 2 instead shows a net improvement in the prediction of the external part of the flow while decreasing in accuracy of the concave part of the bend (point 12). Indeed even though the present case shows a more physical behavior compared to the Case 1, it predicts an inversion of temperature which is not present in the experimental results.

---

3 The measuring points numbers are kept in accordance with the Viollet (1987a) experiment.
For the discussion about this issue it essential to note that in this region indeed turbulence is assumed to decrease while laminarization and flow separation occur. Even though the non linear models were developed and set-up in order to improve estimations of these particular flows, the lack of accuracy can be attributed to highly strong flow curvature, to very low Reynolds number and to the modeling of buoyancy effect through constant parameters. In this context these results claim for insights about the turbulence characteristics in this particular conditions.

Fig. 5 Comparison of the interface width predicted by the different models at non dimensional time $\frac{t}{t_r}$.

Fig. 6 Experimental comparison of temperature difference. Dashed lines refer to outer surface of the bend while solid to the inner at non dimensional time $\frac{t}{t_r}$.

4.1.4. Pressure

A short remark must be done about the pressure distribution inside the pipe, since pressure evolution was one of the mean objective of the study. The qualitative evaluation of the pressure shows that this value increases notably (almost 30 times) from the case of isothermal flow. This consequence can be of impact in the pressure modeling in system codes which account for the whole reactor during the simulation of the accident. Actually not large difference can be noticed from the pressure drops evaluated by the two modelization (Case 1 and Case 2), in this sense nevertheless we can assert that the
understanding of turbulence is not mandatory only for the correct value of pressure introduced but furthermore for the understanding of the evolution and persistence of the stratification which, through additional pressure losses, can likely influence the instauration of natural convection inside the nuclear reactor.

5. CONCLUSIONS

A validation benchmark of a thermally stratified experiment in water was carried out through CFD evaluation. The results show the ability of the present state of art turbulence modeling (non linear models) to predict the occurrence of stratification in the horizontal section of the pipe. While this has been claimed as central point by the author of the experiment, much weaker results are instead performed for the condition at the exit and downstream of the bend. Indeed as pointed out during the above discussion the correct evaluation of the flow in those locations are of deeper importance for the estimation of the total length of the transient, which is assumed to cover an important role in case of sodium cooling.

Finally the results point out some issues which can be answered through a different modelization of the effect of gravity on turbulence and isothermal analyses of similar conditions can validate the use of non linear modeling in condition of highly strong adverse pressure gradient and high flow curvature in flow characterized by low Reynolds number.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>Froude number</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>Pe</td>
<td>Peclet number</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>R</td>
<td>bend curvature [m]</td>
</tr>
<tr>
<td>d</td>
<td>pipe diameter [m]</td>
</tr>
<tr>
<td>g</td>
<td>gravity [m/s²]</td>
</tr>
<tr>
<td>ǻt₀</td>
<td>non dimensional time</td>
</tr>
<tr>
<td>tᵢ</td>
<td>applied temperature increase [s]</td>
</tr>
<tr>
<td>ǻT</td>
<td>temperature increase [K]</td>
</tr>
<tr>
<td>V</td>
<td>average flow velocity [m/s]</td>
</tr>
</tbody>
</table>

Greek Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ</td>
<td>density [kg/m³]</td>
</tr>
<tr>
<td>κ</td>
<td>thermal conductivity [W/mK]</td>
</tr>
<tr>
<td>β</td>
<td>volumetric thermal expansivity [1/K]</td>
</tr>
<tr>
<td>σₜ</td>
<td>turbulent Prandtl number</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>i</td>
<td>i-th coordinate</td>
</tr>
<tr>
<td>j</td>
<td>j-th coordinate</td>
</tr>
</tbody>
</table>

REFERENCES


Viollet P.L., (1987b) “The modeling of turbulent recirculating flows for the purpose of reactor thermal-


STAR-CCM+ UserGuide 5.02