RANS simulations of turbulent and thermal mixing in a T-junction

M. Aounallah*, M. Belkadi**, L. Adjlout***, O. Imine****

*Laboratoire d’Aéro-Hydrodynamique Navale, USTOOran MB, Algeria, E-mail:aounallah_2000@yahoo.fr
**Laboratoire d’Aéro-Hydrodynamique Navale, USTOOran MB, Algeria, E-mail:belkadigma@yahoo.fr
***Laboratoire d’Aéro-Hydrodynamique Navale, USTOOran MB, Algeria, E-mail:adjloul@yahoo.fr
****Laboratoire d’Aéronautique et Systèmes Propulsifs, USTOOran MB, Algeria, E-mail: imine_omar@yahoo.fr

crossref http://dx.doi.org/10.5755/j01.mech.19.3.4663

1. Introduction

The large temperature fluctuations of the fluid can cause a serious disturbance in the operating conditions of some machines which require uniform temperature field at its inlets. For this reason, engineers in many industrial applications try to determine the shortest length of the pipelines to avoid this problem. Moreover, in the place where cold and hot fluids are mixed, a high cycle thermal fatigue in surrounding structure occurs. This phenomenon is significant for structural integrity and safety of the plant. Throughout the world, many reactors were shut down due to the leakage in light water circuit such as the Japanese PWR Tomari-2 in 2003 and the French PWR Civaux in 1998.

Recently some experiments have produced reliable data for validation of computational fluid dynamics (CFD) calculations. Westin et al. [1-2] describes new experimental data of thermal mixing in a T-junction to be used for comparisons. The authors have found that the LES and DES results were in qualitative good agreement with the new experimental data published also when fairly coarse computational meshes were used. Walker et al. [3] have carried out a T-junction mixing experiment with wire-mesh sensors and they obtain important information on the scale of turbulent mixing patterns by cross-correlating the fluctuations signal recorded at different locations within the measuring plane of the sensor. Kimura et al. [4] have studied the influence of upstream elbow in the main pipe of a mixing tee facility. Measured temperature showed that fluctuation intensity near the wall was larger in the case with elbow than in the straight case under the wall jet condition. Naik-Nimbalkar et al. [5] have carried out experiments and numerical investigations of thermal mixing in a T-junction with water. The numerical predictions of the velocity and the temperature fields are found in good agreement with the experimental data.

In the last decades, the flow in T-junctions becomes a challenging test case for CFD. The majority of the numerical contributions underline a number of difficulties principally related to turbulence modeling and the coupling between the turbulence and the heat flux. Walker et al. [6] have performed steady-state calculations with ANSYS-CFX-10 using the k-ε, k-ω SST and RSM models. It was found that both turbulent mixing and turbulent momentum transport downstream of the side-branch connection are underestimated by all the three models and the calculated transport scalar and velocity profiles are less uniform than the measured ones. Better results were obtained by increasing of the model coefficient Cε in the k-ε model leading to an improvement of velocity profiles. Frank et al. [7] have simulated the turbulent isothermal and thermal mixing phenomena using ANSYS CFX 11.0 with unsteady Reynolds-averaging SST and RSM and with scale-resolving SAS-SST turbulence models. It has demonstrated that unsteady SST or RSM turbulence models are able to satisfactorily predict the turbulent mixing of isothermal water streams in a T-junction in the horizontal plane and transient thermal striping was observable from the SAS-SST solution. Chaplilot et al. [8] have inspected the incident of the residual heat removal system of the Civaux unit 1 reactor. Sinkunas et al. [9] have used a method for the calculations of heat transfer and friction in laminar film with respect to variability of liquid physical properties. The dependencies of stabilized heat transfer and friction on temperature gradient for laminar film flow were estimated analytically. Using the CAST3D code, the thermo-hydro-mechanical simulation has demonstrated that the critical point of the accident was the appearance of a crack on the outside of the bend and its rapid propagation through the wall.

Passuto et al. [10] have simulated the turbulent flow using LES technique with Code_Saturne developed by EDF in order to follow the influence of the mean and fluctuating quantities when upstream elbows are neglected in a T-junction. Many others works relating to the thermal mixing using LES were documented in the literature; see e.g. Kuczaj et al. [11], Lu et al. [12].

To gain some understanding of the phenomena taking place in the mixing zone in T-junctions, numerical investigations have been carried out to determine the thermal mixing length. The simulations were done using steady 3D approach and the turbulent fluid motion was solved with RANS: k-ε standard, k-ω standard, k-ω SST and RSM models. The tests were conducted to predict the flow field and the temperature distribution inside a horizontally oriented T-junction with a straight main pipe and a side branch coming in under an angle of 90°.

2. Problem position

The problem treated is basically a three dimensional turbulent thermal flow inside T-pipes with an angle of 90°. Fig. 1 shows the geometrical features of the horizontally oriented T-junction under consideration and the coordinates chosen. The simulation domain consists of the main pipe with a length of 80 inch and a diameter of 6 inch and the side branch which is 30 inch long and 2 inch diameter. The junction is positioned at 1/4 of length of the main pipe. Cold water flows from the left of the main pipe at
15°C and the hot water incomes from the small branch at 50°C. The temperature difference is set to 35°C. The approximate cold and hot flow rates are 30 and 20 m³/h giving inlet bulk velocity values of 0.45–2.74 m/s and the corresponding Reynolds numbers are (0.7–1.37) × 10⁵ respectively. During the simulations, it is assumed that there is no heat exchange with the exterior and all the thermophysical properties of water (viscosity, diffusivity and the specific heat at constant pressure) are set constant except density is function of temperature.

The only the overall forms of

\[ u \mu \varepsilon \]

(1)

\[ \frac{\partial}{\partial x_j}(\rho u_j) = 0 \]

(2)

\[ \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial}{\partial x_j}(\rho u_i u_j) \left[ (\mu + \mu_i) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + G_k - \rho \varepsilon \]

(5)

\[ \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{2}{3} \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right), \quad i = 1, 2, 3 \]

(6)

\[ \mu_i = \rho C_k \frac{k^2}{\varepsilon} \]

(7)

3. Grid generation

Fig. 2 shows the junction zone of the computational domain meshed with hexahedral control volumes. The geometry and the mesh are generated using Gambit preprocessor taking into account the boundary layer refinement with 6 layers near both pipes walls. The height of the first cell is calculated through the estimation of the y+ value which guarantees the use of the high Reynolds number turbulence models with an acceptable accuracy. The mesh quality is excellent since 71% of total cells have an equisize skew coefficient less than 0.1 and 19% between 0.1 and 0.2. The remaining cellules have this coefficient between 0.2 and 0.4. Several tests of grid sensitivity were carried out to get independent solution and finally a grid resolution of 575 280 hexahedral cells is employed.

4. Mathematical formulation

The problem treated is a steady three-dimensional flow in a main pipe with an incoming branch of 90°. The Reynolds number based on the velocity at the centreline and the diameter of the cold inlet is set to 0.7 10⁷. The non-isothermal viscous incompressible flow inside the pipes is described by the steady-state Navier-Stokes equations and the conservation of energy balance. The governing equations are defined as follows

\[ \frac{\partial}{\partial t}(\rho u_j) = 0 \]

(1)

\[ \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial}{\partial x_j}(\rho u_i u_j) \left[ (\mu + \mu_i) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + G_k - \rho \varepsilon \]

(5)

\[ \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{2}{3} \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right), \quad i = 1, 2, 3 \]

(6)

\[ \mu_i = \rho C_k \frac{k^2}{\varepsilon} \]

(7)

The turbulent viscosity is modelled by four turbulence models: the standard k-ε model of Launder [13], the k-ω Standard of Wilcox [14] and the k-ω SST of Menter [15]. The RSM model of Launder [16] closes the RANS equations by solving seven Reynolds stresses transport equations, together with an equation for the dissipation rate. For a simple presentation, only the overall forms of equations are given.

4.1. k-ε Standard model

\[ \frac{\partial}{\partial t}(\rho k) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \]

(5)

\[ \frac{\partial}{\partial t}(\rho \varepsilon) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\mu} \frac{\varepsilon}{k} \left( G_k - C_{2\mu} \rho \frac{\varepsilon^2}{k} \right) \]

(6)

\[ \mu_t = \rho C_k \frac{k^2}{\varepsilon} \]

(7)

4.2. k-ω Standard model

\[ \frac{\partial}{\partial t}(\rho k) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k \]

(8)

\[ \frac{\partial}{\partial t}(\rho \omega) = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \omega}{\partial x_j} \right] + G_m - Y_\omega \]

(9)

\[ \mu_t = \alpha^* \frac{\rho k}{\omega} \]

(10)
and $\omega$ due to turbulence. The coefficient $\alpha^*$ damps the turbulent viscosity causing a low-Reynolds-number correction. The constants are: $\sigma_k = 2.0$ and $\sigma_\omega = 2.0$.

4.3. $k-\omega$ SST model

$$\frac{\partial}{\partial x_j} \left( \rho \frac{k}{\omega} u_j \right) = - \frac{\partial}{\partial x_k} \left[ \rho \frac{k}{\omega} u_i u_j + p \left( \delta_{ij} + \delta_{ij} u_j \right) \right] + D_{L\omega} + P_\omega + \Phi_\omega - \epsilon_\omega$$

$$+ \frac{1}{2} \rho \epsilon$$

$$\frac{\partial}{\partial x_j} \left( \rho \omega u_i \right) = - \frac{\partial}{\partial x_k} \left[ \rho \frac{\omega}{\sigma_\omega} u_i u_j + p \left( \delta_{ij} + \delta_{ij} u_j \right) \right] + C_{\omega} \frac{k}{k} \rho \frac{\epsilon^2}{k}$$

$$\mu_\omega = \rho C_\mu \frac{k^2}{\epsilon}$$

$D_{L\omega}, P_\omega$ do not require any modeling, they represent the molecular diffusion and the stress production respectively. However, $\Phi_\omega$ and $\epsilon_\omega$ represent the pressure strain and the dissipation and need to be modeled to close the Reynolds stress equations. The constants are defined as: $C_k = 1.0$, $C_{\omega} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 0.82$, $\sigma_\omega = 1.0$.

4.4. RSM model

$$\frac{\partial}{\partial x_j} \left( \rho u_i u_j \right) = - \frac{\partial}{\partial x_k} \left[ \rho u_i u_k u_j + p \left( \delta_{ik} + \delta_{ik} u_k \right) \right] +$$

$$+ D_{L\omega} + P_\omega + \Phi_\omega - \epsilon_\omega$$

$$\frac{\partial}{\partial x_j} \left( \rho \frac{k}{\omega} u_i \right) = - \frac{\partial}{\partial x_k} \left[ \rho \frac{k}{\omega} u_i u_j + p \left( \delta_{ij} + \delta_{ij} u_j \right) \right] +$$

$$+ \frac{1}{2} \rho \epsilon$$

$$\mu_\omega = \rho C_\mu \frac{k^2}{\epsilon}$$

In order to obtain a fully developed flow at the hot and cold inlets, a separate computation is firstly conducted on a small cylinder with periodic conditions for both tubes and for the four models of turbulence. The results are obtained after 25 seconds of a transient computation mode; this time seems to be sufficient to achieve a fully developed flow since there is no considerable change in the profile shapes of the velocity components, the turbulent kinetic energy and its dissipation rate. Fig. 3 compares the dimensionless velocity profiles applied at the cold and hot boundary inlets. For both inflows, the velocity profiles intended by the turbulence models experienced collapse approximately in one curve. The velocity, turbulent kinetic energy its dissipation rate planes are saved to be read as boundary conditions at the inlets for the T-junction simulations.

**Fig. 3 Velocity profiles applied as the boundary conditions**

A - cold inlet, B - hot inlet

**6. Results**

6.1. Convergence

Table compares some convergence characteristics of the simulations conducted for the different turbulence models tested. Numerically, it is clearly seen that each model requires its own iteration number needed to reach convergence. It is also remarked that the CPU time is proportional to the iteration number except for the RSM model which requires more CPU time with an iteration number less than that needed by the $k-\omega$ Standard. It is due certainly to the large number of equations to be solved with the RSM compared to those of the two transport equations.
models. The convergence is also well recognized by verifying the net imbalance. A very slight imbalance of mass flow rate is observed; without a doubt it is due to numerical diffusion. The temperature at the outlet is well predicted by all the models tested with a slight difference not exceeding 0.8°C.

### Table

<table>
<thead>
<tr>
<th>Models</th>
<th>Iterations number</th>
<th>CPU time</th>
<th>Net imbalance of mass flow rate kg/s</th>
<th>Temperature outlet °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k-\varepsilon$</td>
<td>500</td>
<td>1h 20’</td>
<td>-1.54 $10^{-7}$</td>
<td>21.13</td>
</tr>
<tr>
<td>$k-\omega$</td>
<td>1900</td>
<td>3h 10’</td>
<td>1.16 $10^{-8}$</td>
<td>20.90</td>
</tr>
<tr>
<td>$k-\omega$ SST</td>
<td>860</td>
<td>2h 50’</td>
<td>-6.20 $10^{-8}$</td>
<td>20.35</td>
</tr>
<tr>
<td>RSM</td>
<td>1500</td>
<td>3h 40’</td>
<td>-8.88 $10^{-5}$</td>
<td>21.10</td>
</tr>
</tbody>
</table>

#### 6.2. Temperature distribution

Fig. 4 shows the comparison of the dimensionless temperature distribution in the median plane of $z$-direction for different turbulence models. The numerical predictions show good qualitative agreement between the turbulence models used. The large mass flow rate of the cold water and the small one of the hot stream have made that the hot water does not inward the upper wall of the main pipe and consequently, there is no thermal effect on the structure close to this region. For all the models tested, the upper wall temperature of the main pipe remains constant and equals to the cold temperature. It is also visibly that most heat transfer occurs just in the lower region close the junction. The green zone near the bottom wall of the mixing part represents a division of hot water counted by cold one since the branch pipe is centered in the perpendicular plan of the main pipe. The mixing region predicted by both standard $k-\varepsilon$ and RSM models seems to be small and centered in the core of the tube than that simulated by both $k-\omega$ models which allocate a more melange area going to the outlet.

Fig. 5 Cross-sectional distribution of the dimensionless temperature distribution

In Fig. 6, the dimensionless temperature profiles in the $y$-direction at $x/L = 0.375$ are presented for the various turbulence models. Qualitatively, the same trend is reproduced. The warm water injection upstream this section has increased the fluid temperature from the wall to the center of the pipe. In the remaining region, the dimensionless temperature diminishes quickly and falls to zero due to the important flow mass rate of the cold water compared to the hot one. The pick of the dimensionless temperature is somewhat indistinguishable when both $k-\omega$ models are used with a slight displacement in the $y$-direction, while it is underestimated with the $k-\varepsilon$ model and overestimated with the RSM model.

Fig. 5 Dimensionless temperature distribution in the stream-wise direction $z/D = 0$

Fig. 6 Dimensionless temperature profiles at $x/L = 0.375$
6.3. Velocity distribution

The dimensionless velocity component in the x-direction profile plotted in the y-direction at $x/L = 0.375$ is shown in Fig. 7. In overall, the numerical predictions obtained by the various turbulence models attest a good qualitative agreement for the two transport equations models while the flow seems to be much accelerated with the RSM model. For all the turbulence models, it can be aware that the flow goes faster in the upper rayon than one in the power part. This behavior can be also confirmed by the inequality of the diameter and the flow mass rate of the two pipes.

Fig. 7 Dimensionless velocity component profile at $x/L = 0.375$

6.4. Turbulent kinetic energy

Fig. 8 shows the dimensionless kinetic energy in the median plane of z-direction for the different turbulence models. It can be noticed that the high level of turbulence occurs always where the thermal mixing takes place. The area where the maximum turbulent kinetic energy is located with the standard $k-\varepsilon$ and the RSM models is so large compared with that visualized by both $k-\omega$ models. The dimensionless turbulent kinetic energy profiles in the y-direction at $x/L = 0.375$ are presented for the different turbulence models in Fig. 9. Good qualitative agreement between models is observed with an overestimation on behalf of the RSM model as mentioned previously. The turbulent kinetic energy reaches its maximum in the center, where the thermal mixing is high and decreases in both directions towards the walls.

Fig. 8 Dimensionless turbulent kinetic energy distribution in the stream-wise direction $z/D = 0$

Fig. 9 Dimensionless turbulent kinetic energy profiles at $x/L = 0.375$

7. Conclusion

In the present study, the effect of the RANS turbulence models on the turbulent and thermal fluid mixing is studied. The flow examined is a non-isothermal steady 3D flow in a turbulent regime ($Re = 0.7 \times 10^3$). The aim of this paper is to determine the mixing length where homogeneous temperature distribution is established. Exhaustive comparisons have been presented for different flow and temperature parameters function of the different turbulence models.

In general, the results obtained agree qualitatively. Unfortunately, the standard $k-\varepsilon$ and the RSM models predict the flow field and the temperature distribution with some discrepancies, whereas both $k-\omega$ models are reasonably close between them. It has been numerically demonstrated that the mixing length, at which constant temperature distribution occurs, is at its end ($x/L = 1$) if a tolerance of 1°C is considered. Further simulations on a longer pipe are strongly encouraged to assist in elucidating the length mixing determination.

References


The turbulent and thermal mixing in a horizontally oriented T-junction is investigated numerically. The objective of the present study is to determine the mixing length in a T-junction where homogeneous temperature distribution is established in the cross section. A steady state three-dimensional turbulent flow is considered with a Reynolds number of $0.7 \times 10^5$ at the inlet of the main cold pipe. Grid is generated in a stretched manner so that strong gradients near the wall regions are accounted for as required. Four turbulence models are tested to provide closure for the Reynolds stress tensor: the $k$-$\varepsilon$ standard, the $k$-$\omega$ standard, the $k$-$\omega$ SST and the RSM models. For all simulated cases, good qualitative agreement is obtained. Quantitative comparisons show that the standard $k$-$\varepsilon$ and the RSM models give too low or high predictions, whereas both $k$-$\omega$ models are reasonably close between them. It is found that a length of 80 inch is enough to get homogeneous temperature if a tolerance of 1°C is considered. Further simulations on a longer pipe are strongly encouraged to assist in elucidating the length mixing determination.

Keywords: T-junction, thermal mixing, turbulence models.