Numerical Simulation of Turbulent Thermal Mixing in a Rectangular T-Junction

Bouchera Sara Bessaid¹, Ahmed Zineddine Dellil¹, Fadéla Nemdili², Abbès Azzi²

¹Département ELM, Institut de Maintenance et de Sécurité Industrielle, Université Oran 2, Oran, Algeria.
²Laboratoire Aero Hydrodynamique Navale, (LAHN) USTO-MB, Oran, Algeria

Université des Sciences et de la Technologie d’Oran, Mohamed Boudiaf
Faculté de Génie Mécanique, BP1505 El-Mnaouar, 31000, Oran, Algeria

Corresponding author email: bestsara2010@hotmail.fr
Received 4 Dec 2017, Revised 25 Mar 2018, Accepted 28 Mar 2018

Abstract

This work reports three-dimensional simulation results of thermal mixing in rectangular T-junction configuration at high Reynolds number. The validation data are provided by an experimental study done at the Department of Mechanical Engineering of Mie University, Japan. The T-Junction was selected as a benchmark for thermal mixing in the ERCOFTAC Workshop held in EDF Chatou, France, 2011. Reynolds Averaged Navier-Stokes (RANS), Unsteady Reynolds Averaged Navier-Stokes (URANS) and Scale-Adaptive Simulation (SAS) were performed with CFD code using finite volume method. Velocity and thermal field, as well as the turbulent stresses, are reported and compared to experimental data in several longitudinal stations. It was found from the comparison that URANS methodology can’t reproduce the striping phenomenon and secondly that the SAS model fits better than the SST model with experimental data. Additional contours of averaged longitudinal velocity end thermal field, as well as the flow structures developing in the channels, are presented and discussed.

Key Words: Turbulence models, SAS-SST, URANS, T-Junction, striping

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>H, B</td>
<td>spacing between the wall planes</td>
</tr>
<tr>
<td>l</td>
<td>length of T-junction</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>U, V</td>
<td>velocity</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl Number</td>
</tr>
<tr>
<td>x, y, z</td>
<td>Cartesian coordinates</td>
</tr>
<tr>
<td>Δt</td>
<td>time step</td>
</tr>
<tr>
<td>μ</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
</tbody>
</table>
1. Introduction

Thermal mixing in T-junction configurations is found in various industrial equipment, including chemical reactors, combustion chambers, piping systems in power plants, and HVAC (Heating, Ventilating, Air-conditioning) units used for automobile air-conditioning systems [1]. Here two streams of fluids with different velocities, temperatures, and/or concentrations are mixed by turbulence. The phenomenon could potentially lead to thermal fatigue failures in energy cooling systems when cyclic stresses are imposed on the piping system due to rapid temperature changes in regions where cold and hot flows are intensively mixed. Additionally, the hot and cold fluids impinge at nearly right angles, a situation lending itself to advanced CFD analysis. The fluctuating thermal field which leads to thermal fatigue can be of dramatic consequence in relation to the nuclear reactor cooling systems [2]. In many studies, thermal stripping has been identified in light water reactors in particular as incidents of high-cycle fatigue at coolant mixing T-junctions [3].

Flow separation and reattachment, secondary flow, the anisotropy of turbulent stresses and heat transfer (including thermal stripping) are some of the complex flow features associated with the T-junction. In order to develop effective methods to promote mixing and control thermal stripping, one has to resort to advanced experimental and modeling strategies to fully understand the detailed flow and heat transfer characteristics. It is thus expected that the choice of turbulence models is a key element for successful prediction. For such applications, past experience shows that statistical time-average models need to be replaced by more sophisticated scale-resolving strategies. This paper uses the well-documented experiment done by Hirota et al., [4] as benchmark for CFD validation. In their project, Hirota et al., [4] have been conducting an experimental study on turbulent mixing of hot and cold airflows in a T-junction with rectangular cross-section. In their experimental data, the authors provide detailed measurements of both dynamical and thermal turbulent fields. This test-case has been selected by the 15th ERCOFTAC Workshop on Refined Turbulence Modelling (Chatou, 2011) [5] as a benchmark for turbulent convection mixing. Additionally, to steady-state and transient RANS simulation with the Shear Stress Transport (SST) turbulence model of Menter[6], this paper presents results from the very promising new strategy of Scale Adaptive Simulation model proposed by Menter and Ergov[7].

2. T-Junction test case descriptions

A series of detailed measurements of turbulent thermal mixing was carried out at the Department of Mechanical Engineering, Mie University, Kurimamachiya-Cho, Japan by Prof. Masafumi Hirota [4]. Fig.1 shows a sketch of the test channel, coordinate system, dimensions, and boundary conditions.
The flow in the main horizontal channel, which is the larger duct, is at 12°C and the flow in the vertical branch corresponds to the hot flow at 60°C. The two flows are maintained at the same blowing velocity of 2.7 m/sec. The rectangular cross-section of the main channel has a dimension of 0.12 x 0.06 m while for the branch it is 0.12 x 0.03 m. The working fluid is air at Pr=0.71. Experimental measurement data are provided for selected planes in both streamwise and spanwise directions as illustrated in Fig. 2.

### Fig. 1. Schematic diagram of the test channel

### Fig. 2. Location of measurements and comparison data [4]

#### 3. Computational domain, boundary conditions and computational meshes

Three hexahedral meshes were generated for the geometry of the T-junction test case, starting with a moderate grid resolution (1.3 Million) and then a refined mesh showing reasonably good near-wall refinement with about 3 Million mesh nodes. As all computational domains are rectangular, the mesh
quality is maintained at the best level with strictly Cartesian meshes and adequate refinement near all wall boundaries. Nominal velocity of 2.7 m/sec in accordance with experimental data in both main and branch pipes have been specified. The given inlet length allows for a fairly well developed turbulent velocity profile at the mixing in the T-junction. In addition, a medium turbulence intensity level of 5% is specified at the main inlet while only 2% is set for the branch. Zero averaged static pressure outlet boundary condition (BC) has been used for the outlet cross-section and non-slip BC’s with automatic wall treatment are used for all walls of the domain. Firstly, a sensitivity study on the two meshes was carried out using steady-state RANS simulation with the Shear Stress Transport (SST) turbulence model [6]. Profiles of mixing temperatures are compared and showed small differences in regions of large gradient; so the fine mesh is adopted and considered as mesh independent solution. This mesh is built by 90 x 90 nodes in the main channel and 90 x 90 nodes in the branch channel. The main channel has 60 nodes from the main inlet until the branch and 150 nodes from the branch until the outlet boundary, while the branch is made with 60 nodes from branch inlet until the main channel. So, the total mesh size is nearly 3 million nodes.

The length of the computational domain is set to 4 times the width of the branch (B) from inlet to the first edge of the T-junction and 10 times from the last edge of the T-junction to the outlet boundary, while the branch is 4 times longer.

Conforming to the experimental test, the coordinate system is set with x downstream, y cross flow, and span wise, with the (x,y,z)=(0,0,0) origin downstream of the centre of the vertical branch. Results will be provided along specified lines as showed in Fig. 2.

4- The turbulence models

The first order, two equations model called Shear Stress Transport turbulence model of Menter[6] is used as a reference for three kinds of computations. The first one uses the steady-state SST turbulence model (RANS), the second one uses the unsteady formulation also with SST turbulence model (URANS) and then the new Scale-Adaptive Simulation (SAS). The SST turbulence model is known to provide a good compromise by combining the k-omega model of Wilcox in the near the wall region and the high Reynolds k-ε model in the outer region. The use of the two models is realized via a blending function, which switches dynamically and smoothly from one to zero depending on the geometrical position of the integration point. The near-wall field is resolved by the use of the automatic wall functions. A detailed explanation of the model formulation and test case validations can be found in specific literature of Menter’s group. The first computation was conducted until convergence using the steady-state RANS SST model. For the second computation, the solution obtained previously with SST model has been used as an initialization for a transient URANS SST simulation. The time step was set at Δt=0.001 second with a second-order backward time discretization. The 10-4 maximal residual was reached with less than three
loops per time step. As expected from other researchers investigating similar T-junction but with a circular cross-section, the behavior of the URANS SST solution is quickly approaching a steady-state solution in terms of velocity and temperature fields after some initial transient behavior.

This can be explained by the nature of the URANS strategy itself, which can’t provide any spectral content, even if the grid and time step resolution would be sufficient for that purpose. This behavior is a natural outcome of the RANS averaging procedure, which eliminates all turbulence content from the velocity field. So, URANS can only work in situations of a ‘separation of scales’. But for this practical application and as stated before, experimental observations clearly state strong and high-frequency temperature fluctuations at pipe walls downstream of the T-junction, which can lead to high-cycle thermal fatigue, crack formation and pipeline break, e.g. in pipelines in power plants. So, in that situation there is a real need to use other turbulence modeling strategies like Large Eddy Simulation (L.E.S) or Detached Eddy Simulation (D.E.S). L.E.S is based on the concept of filtering the flow field by means of a spatial filter. The specific super-grid part of the flow with its turbulent fluctuating content is directly predicted whereas the sub-grid scale (SGS) part is modeled, assuming that these scales are more homogeneous and universal in behavior. This approach can give very interesting results for this test case but has the inconvenience to be very expensive especially in terms of resolution near solid walls [7 and 8]. To overcome the restriction in terms of computational grid sizes and consequently the running time, D.E.S combines L.E.S and URANS strategies and gives a very promising tool to predict industrial flows [7 and 8].

Nevertheless, there is still an ambiguity in mixing two different physics in the same computation (averaged and instantaneous values). The so-called Scale-Adaptive Simulation (SAS) model was recently proposed by Menter and Egorov[9] as a new method for the simulation of unsteady turbulent flows. A complete description of the SAS model can be found in the related publications and only a brief description is provided here.
Fig. 3. Velocities, stresses and temperatures at the longitudinal stations

X / B = 0  X / B = 1  X / B = 2  X / B = 4  X / B = 5

- Experiment; —— RANS-SST; —— SST SAS

X / B = 0  X / B = 1  X / B = 2  X / B = 4  X / B = 5

Fig. 3. Velocities, stresses and temperatures at the longitudinal stations
While all two equations turbulence models use the same transport equation of kinetic energy as first equation, they show large difference in formulating the second equation. The construction of this second equation is not as straight and clear as the first one. According to Menter and Egorov [9], the Rotta’s k-kL turbulence model is well suited for a term-by-term modeling and shows some interesting features compared to other approaches. Nevertheless, the weakest part made in this model is in neglecting the second velocity derivative and maintaining the third one. The model is then suited only for homogenous turbulence and need additional terms to be applied in the near wall zones. Menter and Egorov [9] suggest then to replace problematic third velocity derivatives with the second one. The formulation proposed by Menter can operate in standard RANS mode, but has the capability of resolving the turbulent spectrum in unsteady flow regions.

This is done by use of the von Karman length-scale, which is a three-dimensional generalization of the classic boundary layer definition. The mathematical formulation of the SST-SAS model differ from those of the SST URANS model by an additional SAS source term in the transport equation for the
turbulence eddy frequency. The main feature of the method is its capability to adapt the length-scale automatically to the resolved scales of the flow field rather than the thickness of the turbulent (shear) layer. So, the SAS solution automatically applies the RANS mode in the attached boundary layers, but allows a resolution of the turbulent structures in the detached regime. This behavior is in much better agreement with the true physics of the flow, as was also shown for other test-cases by Menter and Egorov [9]. Contrary to LES or DES technics, the SST-SAS model operates in frame of URANS formulation with the so-called “LES”-like capability without an explicit dependency on the grid spacing. So, a third computation was done with SST-SAS model using the quasi steady-state result from the preceding URANS-SST as initial conditions. The transient simulation by using the SAS-SST scale resolving turbulence model approach has been carried out for 3 second real time with a time step of $\Delta t=0.0001s$.

5- Solution methodology

The present simulations were conducted using the finite-volume code Fluent. In the solver package, the solution of the governing equations is obtained by using finite volume method with multi-blocs hexahedral structured grids. The momentum and continuity equations are coupled through the SIMPLE pressure correction scheme. The spatial discretization consisted of a bounded central-differencing scheme by Leonard for the nonlinear terms and the second-order central scheme for the viscous terms.

6- Computational grid

The original version of the SST-SAS model (Menter and Egorov [9]) has undergone certain evolution and the latest model version has been presented in Egorov and Menter [9]. One model change is the use of the quadratic length scale ratio $(L/L_{vk})^2$ in the Equation 1, 2 below, rather than the linear form of the original model version. The use of the quadratic length scale ratio is more consistent with the derivation of the model and no major differences to the original model version are expected. Another new model aspect is the explicitly calibrated high wave number damping to satisfy the requirement for an SAS model that a proper damping of the resolved turbulence at the high wave number end of the spectrum (resolution limit of the grid) must be provided. In the following the latest model version of the SST-SAS model (Egorov and Menter [9]) will be discussed, which is also the default version in ANSYS CFX. The governing equations of the SST-SAS model differ from those of the SST RANS model [6] by the additional SAS source term $Q_{SAS}$ in the transport equation (3) for the turbulence eddy frequency $\omega$:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = P_k + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) - \rho \omega c_k \omega \text{Eqs.(1)}$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) =\alpha^2 P_k - \rho \beta \omega^2 + Q_{SAS} + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + (1 - F_1) \frac{2\rho}{\sigma_{\omega 2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \text{Eqs.(2)}$$

Where $\sigma_{\omega 2}$ is the $\sigma_\omega$ value for the $k - \varepsilon$ regime of the SST model. The additional source term $Q_{SAS}$ reads for the latest model version Egorov and Menter [9]:

1
\[ Q_{SAS} = \max \left[ \rho \xi_2 K S^2 \left( \frac{L}{L_{eq}} \right)^2 - C \cdot \frac{2 \rho k}{\sigma_{\Phi}} \max \left( \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j}, \frac{1}{k^2} \frac{\partial k}{\partial x_j} \right) \right] \text{Eq. (3)} \]

The model parameters in the SAS source term Equation are:

\[ \xi_2 = 3.51, \sigma_{\Phi} = 2/3, C = 2 \]

The discretization of the advection is the same as that for the SST-DES model, beside the fact that no RANS-shielding is performed for the SAS-SST model. The grids carried out with software ANSYS-ICEM are of hexahedral type and tetrahedral. The grid is carried out so that it is refined on the level of the rib and of the undulation (see Figure 4) for a validation of the model of turbulence to knowing the SST. Three levels of mesh refinement were used and tested, which consisted of approximately 1,300,000, 4,000,000, and 3,000,000 hexahedral elements. So, the grid with 3,000,000 hexahedral cells is adopted in all present computations.

\[ \text{Fig 5.grid} \]

7- Results and discussion

7.1- Validation

SST-RANS and SST-SAS time-averaged results including the measurement data for comparison are shown in Figures 3 and 4. Globally, velocities, stresses and thermal profiles at the five locations of the channel are qualitatively well reproduced by the two models. Comparing the two models, the SAS model is by far in better agreement with experimental data especially for the three first measurement stations, while some discrepancy is observed in the two last ones. The reasons for this difference can be explained partly by low spatial and temporal resolution in regard to turbulence model requirements. Another source of discrepancy can be due to differences in inlet boundary conditions between experiment and numerical runs. In fact, fully developed velocity and high level of turbulence intensity are assumed at inlet whereas it is not clear if this is the case in experimental work. While the longitudinal velocity is well reproduced almost in all stations, the vertical velocity shows some considerable differences between the two models and the experimental data. An example in a station
(x/B=2), the qualitative experimental V/U0 profile is slightly well captured by the SAS model, while the RANS model fails to reproduce the two positive values upper and lower the midline position (y/H=4). Negative values reproduced by the RANS model and the week positive value by the SAS model show that numerical results predict a smaller separation bubble compared to the experimental one. Turbulence intensity differences are also highlighted by remarkable differences between the experimental data and simulation results for longitudinal and vertical fluctuation profiles. Nevertheless, both models reproduce high turbulence activities in the lower part of the domain (less than y/H=4), while the upper part is globally free from turbulence.

7.2- Flow and thermal field description:

All results described and discussed here are obtained from SAS computation. Instantaneous flow separation and typical developing vortex structures downstream of the T-junction at t = 1 second real-time are shown in Figure 6.

![SST](image1)

![SAS](image2)

**Fig.6.** Iso-surfaces of the Q-criteria colored by the longitudinal velocity, Q = 500 [s^-2]

The visualization is based on iso-surfaces of the so-called Q-criteria, the function of vorticity and the strain rate of the flow field. Figure 7 clearly shows that SAS model can give more details for the irregular turbulent vortex structures forming immediately downstream of the T-junction and being convected with the flow along the main pipe.
From experimental data provided by Hirota et al., [4], the hot flow entering the channel is separated at the downstream edge of the T-junction forming a large separation bubble along the bottom wall of the main channel. The reattachment point is located nearly at four times the width of the branch after the last edge of the T-junction. The location of the reattachment point is confirmed by Figure 7 showing...
the spanwise variation of the longitudinal flow velocity distribution where the gray regions correspond to reverse flow. The comparison to experimental data from Hirota et al., [4] is fairly well including the boundary layer reattachment point and the 3D behavior of the flow development. When approaching the wall side (Z/A=0.875) the height of the bubble decreases and the flow reattachment point moves in the negative direction. Streamwise evolution of cross-sectional distribution of $U/U_0$ (left) and secondary flow velocities (right) are shown in Figure 8.

![Figure 8. Streamwise evolution of cross-sectional distribution of $U/U_0$ (left) and secondary flow velocities (right).](image)

At the first station (x/B=0), the upward direction is dominated over the cross section on the main channel, while at the second and third (x/B=1 and 2 respectively) stations the upward flow becomes weaker. We remind here from previous figures that cross section x/B=1 and 2 correspond to the bubble separation and x/B=4 correspond to the flow reattaching point. So, at x/B=2, a longitudinal vortex develops near the lower corner, while it disappears at x/B=4. This behavior is related to the
streamline curvatures and the flow reattachment point. Streamwise evolution of the cross-sectional mean temperature normalized by the cold and hot flow temperatures are shown in Figure 9.

![Streamwise evolution of the cross-sectional mean temperature distribution](image)

**Fig. 9.** Streamwise evolution of the cross-sectional mean temperature distribution

The hot (bottom) and cold (top) regions are separated by the thermal mixing layer were the temperature vertical gradient ($dT/dy$) is at its maximal values. Surprisingly and contrary to the 3D evolution stated before, the thermal field shows the very nearly uniform distribution in the spanwise direction. This behavior is also reported by the experimental study of Hirota et al. [4] and is explained by the fact that the longitudinal vortex is confined into the hot flow region below the thermal mixing layer and doesn’t exert any influence on the temperature distribution.
8- Conclusion

The T-Junction experiment carried out at the Department of Mechanical Engineering of Mie University, Japan is investigated here numerically with the use of SST-RANS, SST-URANS, and SST-SAS models. The last one being in the frame of scale resolving simulation gives LES-like results and still stays in URANS strategy. Comparing to SST-RANS, the SAS model has been shown to yield accurate results for this complex flow in T-junction. The present study showed also that such quality results are very difficult to obtain using a URANS approach.

References

5. Wiki-projets sp2mi.univ-poitiers.fr/bin/View/workshop chatou 2011/webhome
7. O. Braillard, Y. Jarny, and G. Balmigere, Thermal load determination in the mixing Tee impacted by a turbulent flow generated by two fluids at large gap of temperature, ICONE13-50361, 13th International Conference on Nuclear Engineering, Beijing, China,(2005) 16-20.