COBEM-2017-1156
NON-ISOTHERMAL FLUID FLOW IN A T-JUNCTION

Alysson Dantas Ferreira[1]
Severino Rodrigues de Farias Neto[2]
Universidade Federal de Campina Grande, Department of Chemical Engineering, Campina Grande, Brazil
alysson.dantas@eq.ufcg.edu.br[1], s.fariasn@gmail.com [2]

Abstract. Due to the high industrial competitiveness, companies are constantly looking for technologies that make the price of their products more accessible to the market, investing in research in the areas of optimization or development of new low-cost and low-maintenance equipment. The mixing process is an operation that is present in the most diverse industrial sectors. In this sense, the purpose of this work is to study, through computational fluid dynamics techniques, a simple geometry for a mixer device and to analyze non-isothermal fluid flow in a T mixer, comparing the results with more than one software. The Salome computational program was used to generate the mesh of a T junction, Ansys CFX and OpenFOAM to obtain the results of the numerical simulations. With the results of the simulation it was possible to evaluate the quality of the mathematical models used, comparing them with experimental data obtained by Naik-Nimbalkar et al. (2010) and performing a statistical treatment. It was also possible to evaluate the results of the different software used, thus, informs which one presented better response to the data of the experiment. The data of the simulations were shown to be coherent and were statistically validated, further enhancing confidence in the results. The proposed mathematical model represented with fidelity the phenomenon of mixing of a junction T.

Keywords: CFD, T-Junction, Statistics treatment, Hydrodynamic performance.

1. INTRODUCTION

The mixing process is constantly used in various types of chemical industries, there are different types of equipment to carry out the operation, where most are expensive, spacious and complex. In the last decades several studies have been carried out in this area for the development of new equipment, such as a "T" junction that leads to the formation of cross flow jets.

The bifurcation or junction "T" is a variation industrially commonly used as a mixer, separator or reactor because it is a simple construction tool and low cost. In "Fig. 1", some of the possibilities of fluid inlet and outlet are presented. It is a type of jet in confined cross flow and has been considered a good method to promote mixing as well as heat and mass transfer with or without chemical reaction, separation of two or more phases.
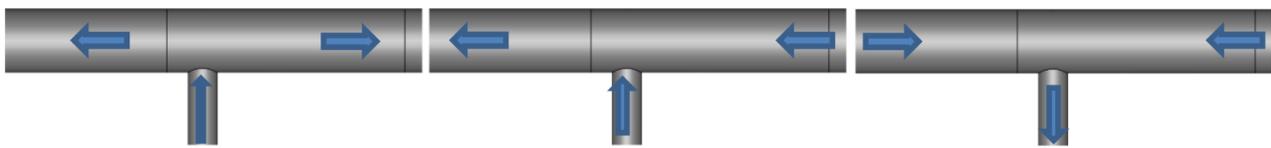

Figure 1. Possibilities of fluid inlets and outlets.

The bifurcation is used for separation, generally when a mixture formed by two immiscible phases flows through the main pipe of the "T" junction. The lighter phase, for example, gas bubbles in a gas-liquid mixture, tends to be diverted to the side branch at a greater ratio than the heavier phase. As a consequence, there is a greater concentration of the gas in the mixture at the inlet of the T junction and differs along the flow to the outlet. A method to predict this behavior of each phase in the two output branches of a "T" junction (main branch and lateral branch) is of great importance since the design of pumps or other equipment connected to these lines is strongly influenced by the concentration of each phase (Oliveira *et al.*, 1992).

In the chemical process industry, turbulent jets are widely used for the mixing of miscible liquids. The common configuration of the turbulent jet mixer for efficient mixing of two fast-reacting fluids is the cross-flow jet confined in a T-mixer or side-entry mixer. In this situation a fluid stream is injected radially by the lateral branch into a tube with another fluid stream, generally having a flow regime of a developed fluid (laminar or turbulent) where the reaction takes place.

In the study of T-junction, we highlight the mixture of hot and cold fluid streams, which in many situations can lead to large temperature fluctuations in a short period of time. According to Ogawa *et al.* (2010) if these temperature fluctuations reach large amplitudes of temperature variation can lead to thermal fatigue of the pipe structure, which can lead to cracks that will cause fluid leakage. The compression of this phenomenon is of main importance in assessing the structural integrity and safety of nuclear power plants and thermoelectric power generation plants.

According to Lin and Ferng (2016) many simulations using computational fluid dynamics (CFD) with different turbulence models have been conducted to investigate the mixing characteristics of T-junctions. Hu and Kazimi (2006) studied temperature fluctuations caused by thermal mixing in T-junctions using the large eddy simulation (LES) turbulence model. The results showed that the calculated maximum temperatures were somewhat higher than the measurements. Wang and Mujumdar (2007) developed a three-dimensional (3-D) model with the standard $\kappa - \varepsilon$ turbulence model to predict the flow and mixing characteristics of multiple and multi-set opposing jets. Good agreement between the simulated and the experimental results was obtained. Using the LES turbulence model, Lee *et al.* (2009) simulated the coolant temperature fluctuations at a mixing T-junction of equal pipe diameters. The calculated normalized mean temperatures and fluctuating temperatures were in good agreement with the measurements.

The purpose of this work is to explain the importance of confined cross-flow in the processes of separation, mixing and reaction using T junction. The aim of this research was to perform an analysis and physically understand the phenomena of temperature fluctuations involved in the mixing of two fluid streams with different temperatures, with the aid of two computer programs: one is open source, OpenFOAM, and the other is commercial, Ansys CFX.

2. COMPUTATIONAL PROCEDURE

The physical domain studied is based on the work of Naik-Nimbalkar *et al.* (2010) and illustrated in "Fig. 2". The study domain is highlighted by the dashed area, a T-shaped connection, characterized by a main duct of 1.2 m in length and a diameter of 0.05 m connected to a secondary duct whose length is 0.6 m and the diameter is 0.025 m. The position of the secondary duct is at the midpoint between the inlet and outlet of the main pipe. The representative meshes of the study domain were generated in the SALOME 8.2 software. The computational domain was constructed with tetrahedral elements, containing hexahedral elements in the region of the viscous layer.
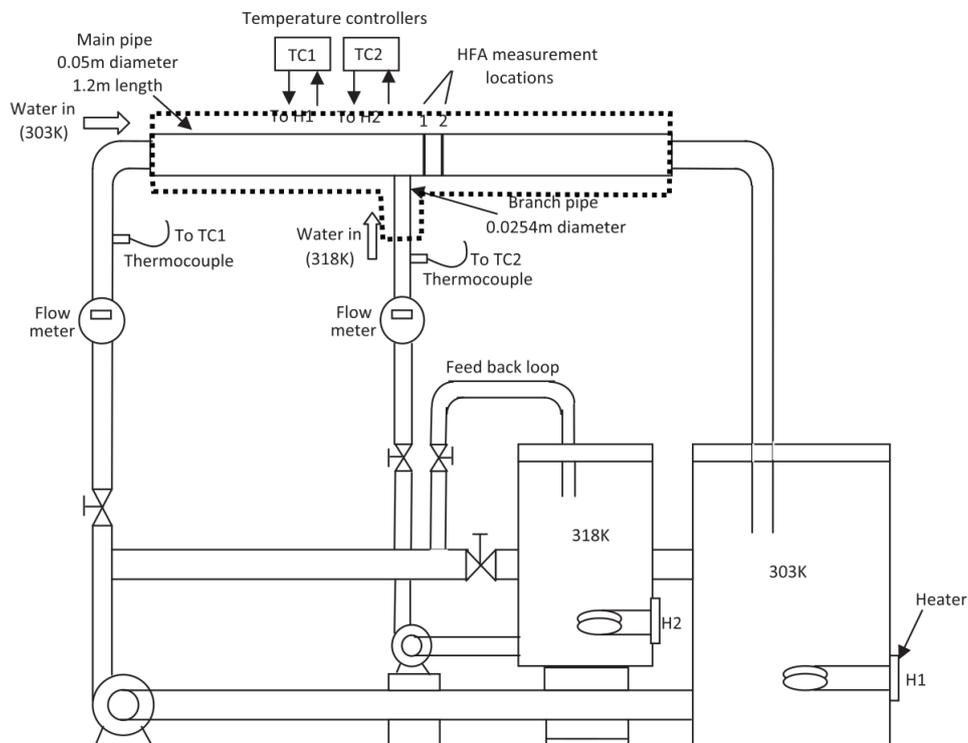

Figure 2. Experimental apparatus used by Naik-Nimbalkar *et al.* (2010).

To perform the mesh convergence study, the study domain was divided into three regions as shown in "Fig. 3". Three meshes were constructed with different amounts of elements, varying only the maximum size of each element in zone 2, as shown in "Tab. 1".

In the flow of the mixer type T, it was considered that the regime is in steady state, is non-isothermal, turbulent, does not occur the presence of a chemical reaction and was considering the gravitational effect. The walls were considered

static, without roughness, without slip and with incompressible fluid. The $\kappa - \varepsilon$ turbulence model was used. From these considerations, the simulation was performed in the OpenFOAM and Ansys CFX software, in which the following transport equations are implemented:
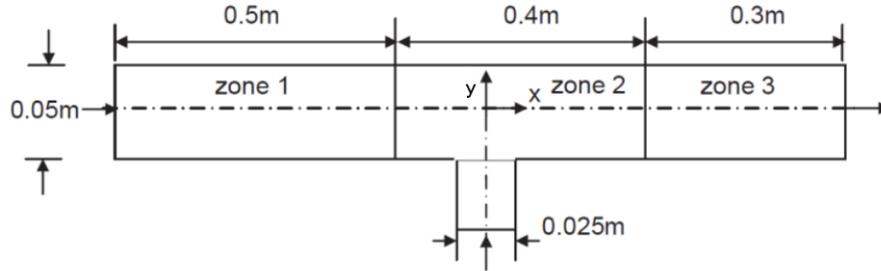

Figure 3. Refinement Zones.

Table 1. Maximum size of elements in each zone.

Meshes	Zone 1	Zone 2	Zone 3	N of Elements
Mesh 1	3 mm	2 mm	3 mm	1980021
Mesh 2	3 mm	1.5 mm	3 mm	3811402
Mesh 3	3 mm	1 mm	3 mm	4106444

Continuity Equation

$$\nabla (U) = 0 \tag{1}$$

Moment Equation

$$\nabla (UU) + \nabla p - \nabla \left[\nu \left(\nabla U + (\nabla U)^T - \frac{2}{3} tr (\nabla U)^T I \right) \right] = [1 - \beta(T - T_{ref})] \tag{2}$$

Energy Equation

$$\nabla (UT) - \nabla \left[\left(\frac{\nu}{Pr} + \frac{\nu_t}{Pr_t} \right) \nabla T \right] = 0 \tag{3}$$

$\kappa - \varepsilon$ **Model**

$$\frac{\partial \kappa}{\partial t} + \nabla \cdot (\kappa U) = \nabla \left[\left(\nu + \frac{\nu_t}{\sigma_\kappa} \right) \nabla \kappa \right] + 2\nu_t S_{ij} \cdot S_{ij} - \varepsilon \tag{4}$$

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon U) = \nabla \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{\kappa} 2\nu_t S_{ij} \cdot S_{ij} - C_{2\varepsilon} \frac{\varepsilon^2}{\kappa} \tag{5}$$

The following boundary conditions were adopted:

• Inlet section - horizontal tube: prescribed velocity of 1 m/s and temperature of 303K;
• Inlet section - secondary branch: prescribed velocity of 0.5 m / s and temperature of 318K;
• Outlet section - average pressure of 101325 Pa;

The fluid used in the present work was water and its properties are listed in "Tab. 2".

3. RESULTS AND DISCUSSION

The analysis of the dependence of the meshes was performed with the help of Ansys CFX 15 and OpenFOAM software. The temperature and axial velocity values were collected at positions L1 and L2, which are shown in "Fig. 4". The axial velocity and temperature were normalized and the result is found in "Fig. 5".

In "Fig. 5" are represented the values of the temperatures and velocities dimensionless as a function of the dimensionless position.

Table 2. Properties and parameters of the fluid.

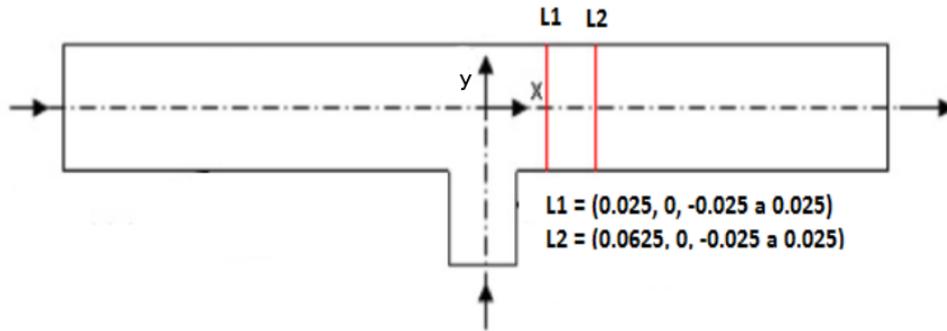| Properties	Water
Thermal expansion coefficient	$4.185X10^4 K^{-1}$
Reference Temperature	$318K$
Kinematic viscosity	$1X10^{-6}\frac{m^2}{s}$
Laminar Prandtl Number	0.7
Turbulent Prandtl Number	0.9

L1 = (0.025, 0, -0.025 a 0.025)
L2 = (0.0625, 0, -0.025 a 0.025)

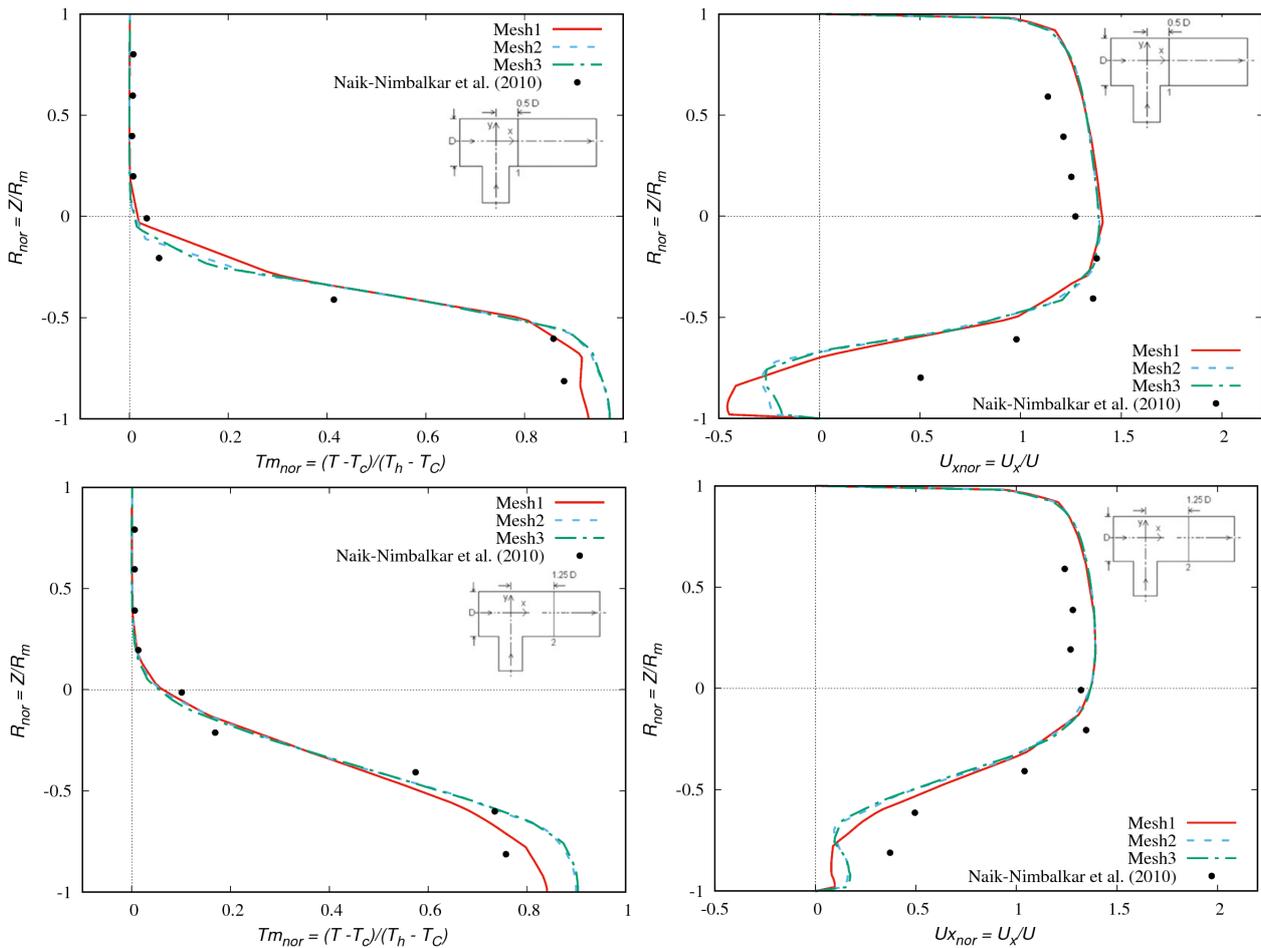Figure 4. Representation of the positions of the lines in the pipe where the values of velocity and temperature were taken.

Figure 5. Mesh Dependency

It is observed that the results indicate that there is practically no great influence of the mesh on the behavior of the temperature and velocity components, especially between the meshes 2 and 3. Thus, the mesh 2 was chosen to generate the results shown in the "Fig. 7", "Fig. 8" and "Fig. 9".
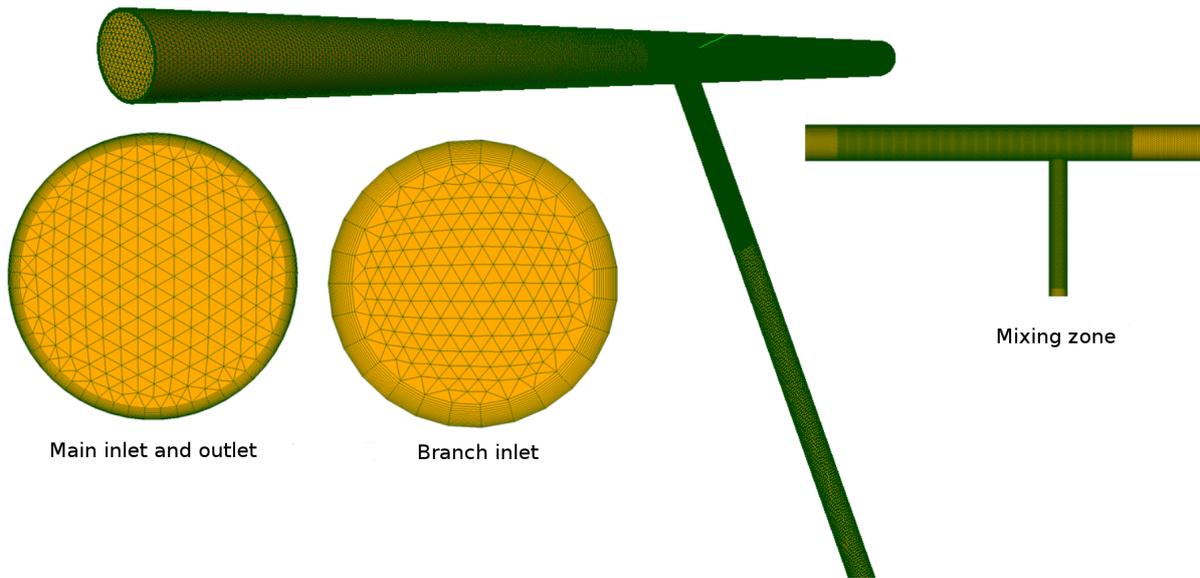
Figure 6. Mesh 2 for the mixer T.

The results of the simulation using OpenFOAM and Ansys CFX, as well as the experimental data of Naik-Nimbalkar *et al.* (2010) are shown in "Fig. 7", in which are shown the temperature and axial velocity profiles at positions L1 and L2 shown in "Fig. 4".
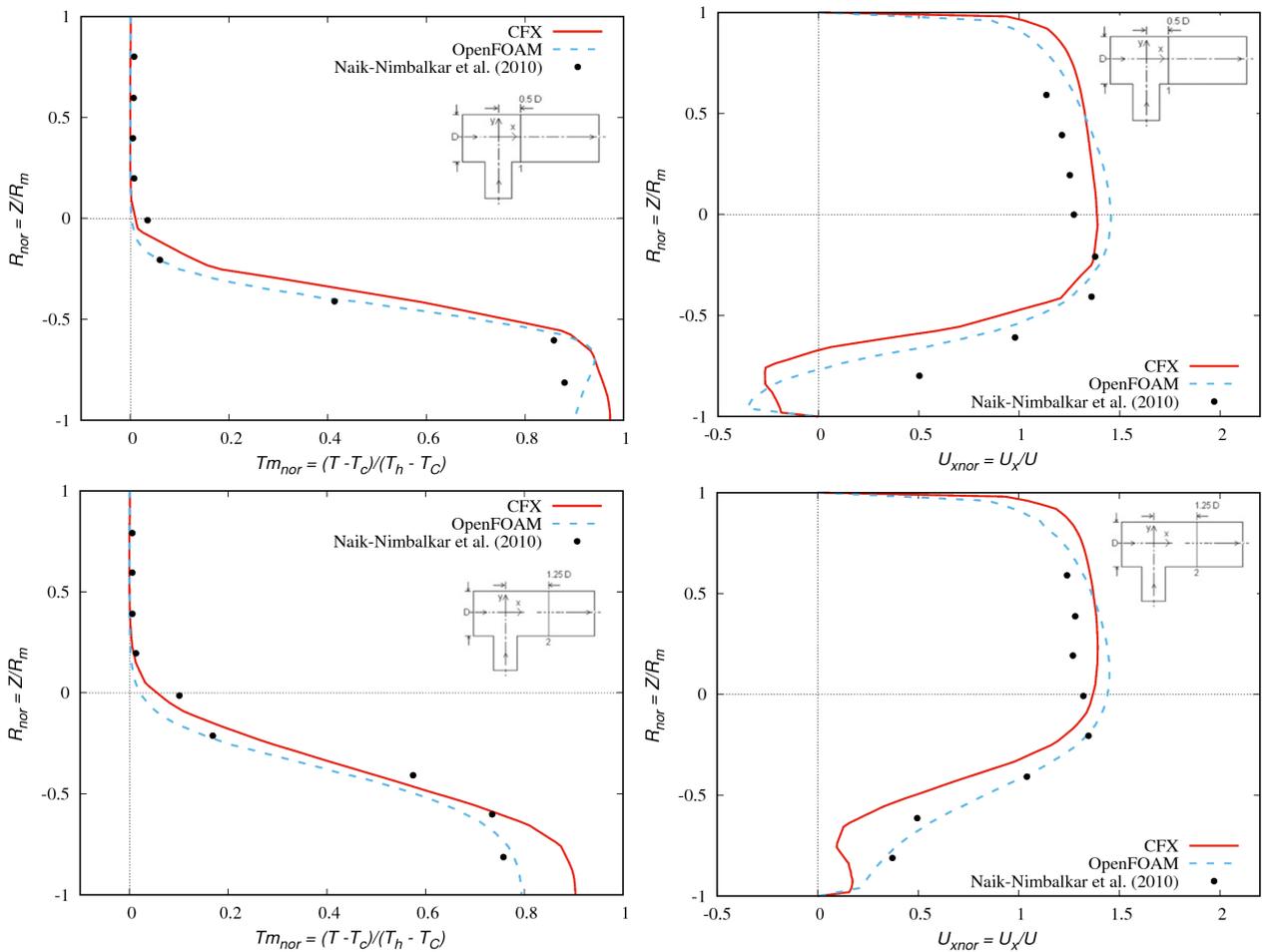

Figure 7. Comparison between OpenFOAM and CFX.

Analyzing "Fig. 7", it is observed that the numerical results obtained by both software seem to be in good agreement

with the experimental data of Naik-Nimbalkar *et al.* (2010), however, it is known that many of the mathematical models are simplifications of real phenomena and when solved numerically present truncation errors. It should also be taken into account that the obtaining of the experimental data are subject to errors, such as the calibration and the gauging of the equipments used. The sum of these small errors can lead or, more specifically, lead to the observed discrepancies. In this sense, a region of confidence was determined statistically with 5 % of significance.
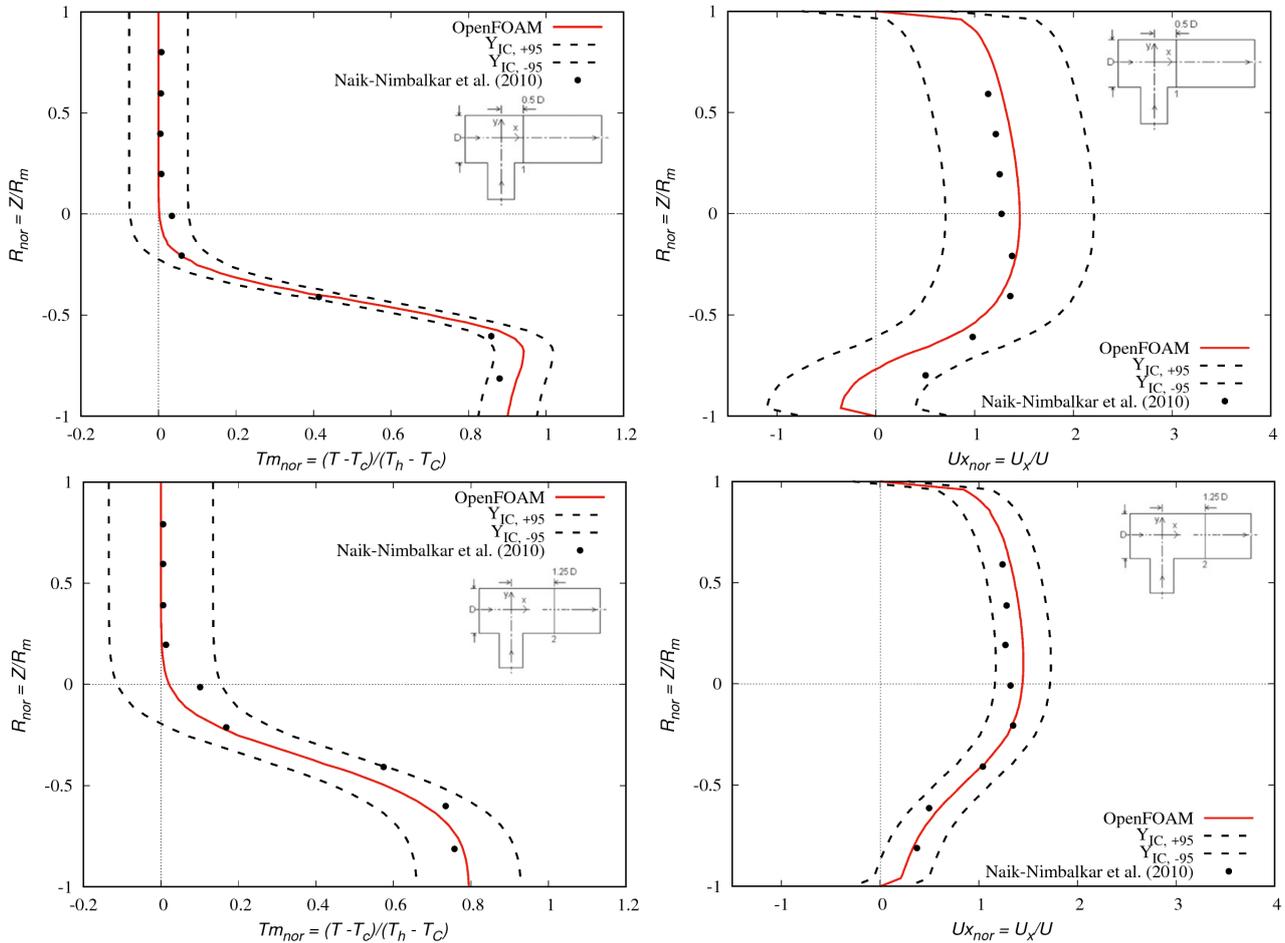

Figure 8. Confidence region of the OpenFOAM's data.

Analyzing "Fig. 8" and "Fig. 9", it is observed that the numerical results obtained with both the CFX and OpenFOAM and the experimental data remained within the region of confidence, thus guaranteeing a validation of the numerical results in relation to the experimental data.

Comparing the results of both software, it is observed that the lower and upper limits of the regions of confidence regarding the results of OpenFOAM are closer to the values of the simulation, that is, the standard deviation of the numerical results in relation to the experimental data are smaller than the standard deviations obtained by CFX.

In "Fig. 10" and "Fig. 11" the temperature and axial velocity distribution are shown respectively along the T mixer, these fields were generated by the OpenFOAM.

It is noted in "Fig. 10" is a temperature variation below the axis of the main pipe, which indicates the mixing region of the secondary branch fluid with the horizontal pipe fluid. As the jet advances, due to mixing, the average temperature values become lower in the lower region of the main tube. In the region of interaction of the hot fluid current with the cold fluid current, a thermal fluctuation occurs, which according to Ayhan and Sokmen (2012), this fluctuation induces cyclical thermal stresses, which results in a thermal fatigue and may cause an unexpected failure in the tube material.

Analyzing "Fig. 11" there is a sudden change in velocity near the axis of the main pipe, which indicates the point of interaction between the fluid of the secondary branch with the fluid of the main pipe. It is still noted that in the lower region of the mixing zone, close to the side entrance, a negative axial velocity, which characterizes the formation of a recirculation zone.
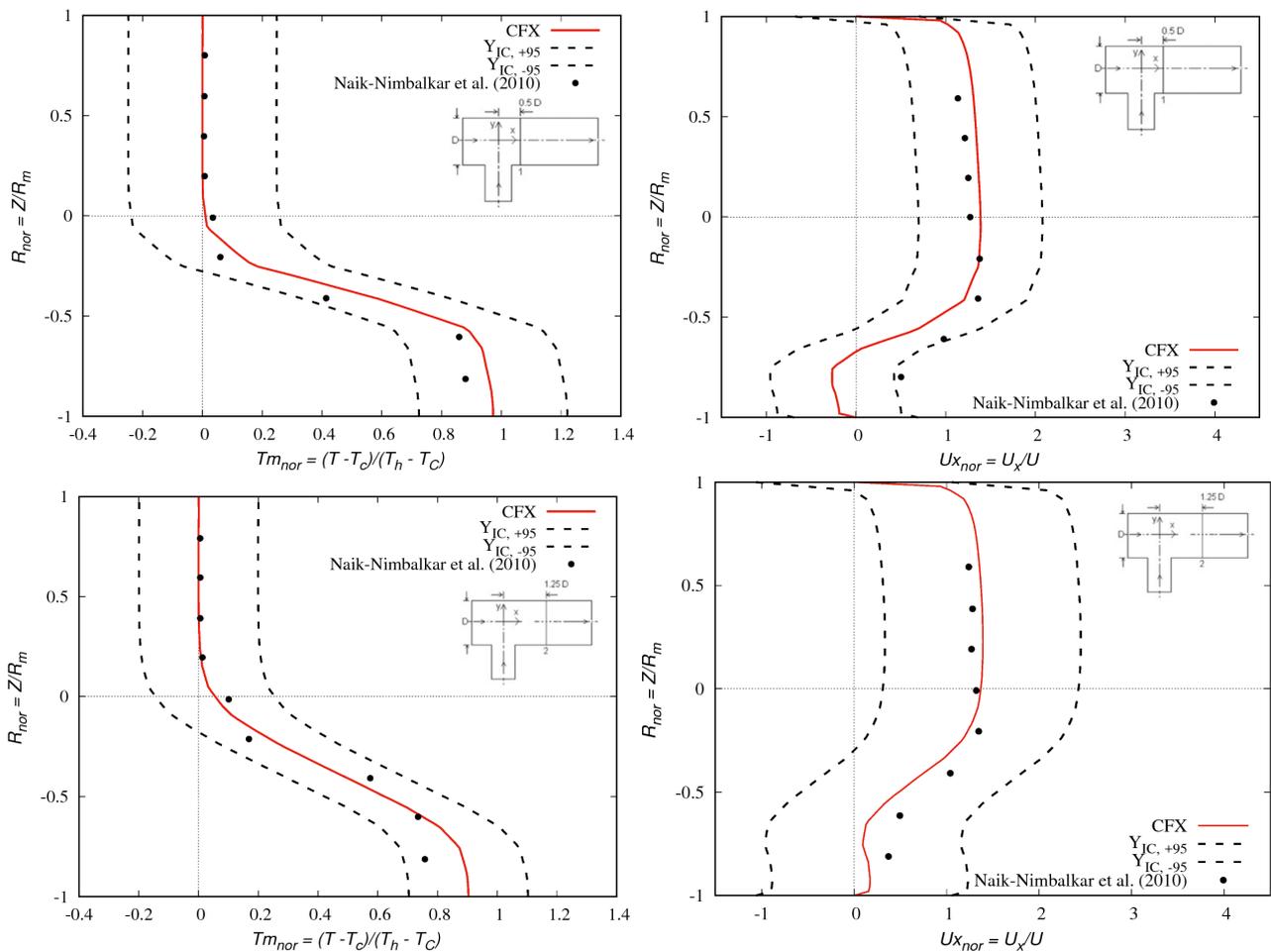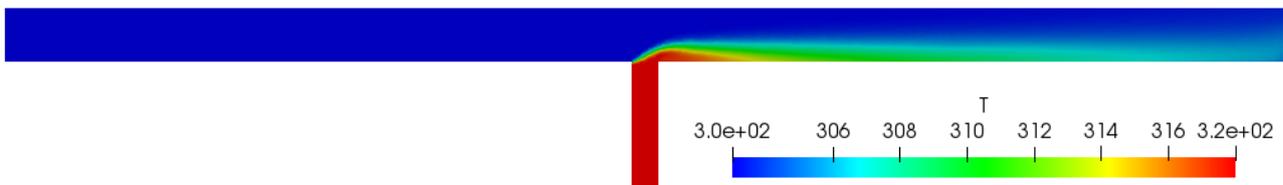
Figure 9. Confidence region of the CFX's data

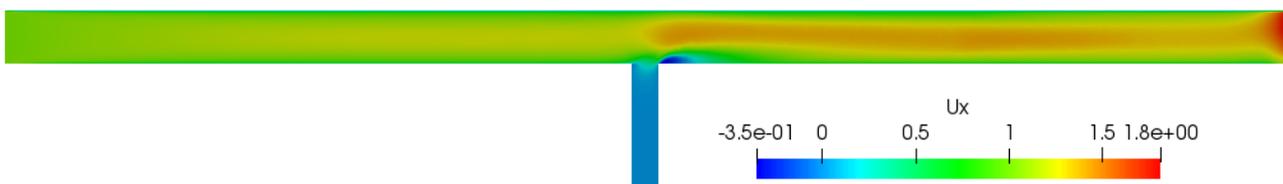Figure 10. Temperature distribution along mixer T.

Figure 11. Distribution of the axial velocity along the mixer T.

4. CONCLUSION

About the results obtained, it is concluded that the mathematical models of mass, momentum and energy conservation suggested by this work and the standard $\kappa-\varepsilon$ turbulence models were able to predict the temperature and fluid fluctuations of the fluid currents in the T junction, validated experimentally and statistically.

The numerical results of the OpenFOAM and CFX software presented a good agreement with the experimental results, with OpenFOAM presenting a lower standard deviation, consequently representing more faithfully the physical phenomena under the conditions currently studied.

5. REFERENCES

Ayhan, H. and Sokmen, C.N., 2012. "Cfd modeling of thermal mixing in a t-junction geometry using les model". *Nuclear Engineering and Design*, Vol. 253, pp. 183–191.

Hu, L. and Kazimi, M., 2006. "Les benchmark study of high cycle temperature fluctuations caused by thermal striping in a mixing tee". *International Journal of Heat and Fluid Flow*, Vol. 27, pp. 54–64.

Lee, J., Hu, L., Saha, P. and Kazimi, M., 2009. "Numerical analysis of thermal striping induced high cycle thermal fatigue in a mixing teer". *Nuclear Engineering and Design*, Vol. 239, pp. 833–842.

Lin, C. and Ferng, Y., 2016. "Investigating thermal mixing and reverse flow characteristics in a t-junction using cfd methodology". *Applied Thermal Engineering*, Vol. 102, pp. 733–741.

Naik-Nimbalkar, V.S., Patwardhan, A.W., Banerjee, I., Padmakumar, G. and Vaidyanathan, G., 2010. "Thermal mixing in t-junctions". *Chemical Engineering Science*, Vol. 65, pp. 5901–5911.

Ogawa, H., Igarashi, M., Kimura, N. and Kamide, H., 2010. "Experimental study on fluid mixing phenomena in t-pipe junction with upstream elbow". *Nuclear Engineering and Design*, Vol. 240, pp. 3055–3066.

Oliveira, A.G., Silva, G.P., Geraldelli, W. and Ropelato, K., 1992. "O crescimento do uso de fluidodinâmica computacional (cfd) nas tecnologias de destilação, fcc e coqueamento retardado na engenharia básica de abastecimento da petrobras." *Relatorio Tecnico*.

Wang, S. and Mujumdar, A., 2007. "Dsmc simulation of gas mixing in t-shape micromixer". *Applied Thermal Engineering*, Vol. 27, pp. 2370–2377.

6. RESPONSIBILITY NOTICE