Simulation of mixing flows with different temperatures in a T-junction using OpenFOAM code

Najmeh Jafari Ouregani, V I Melikhov, O I Melikhov
National Research University “Moscow Power Engineering Institute
Krasnokazarmennaya 14, Moscow 111250, Russia
vladimir.melikhov@erec.ru

Abstract. This paper aims to evaluate the frequency of velocity and temperature fluctuations in the mixing region using OpenFOAM code. Turbulent mixing of fluids at different temperatures can lead to temperature fluctuations at the pipe material. These fluctuations, or thermal striping, inducing cyclical thermal stresses and resulting thermal fatigue, may cause unexpected failure of pipe material. Therefore, an accurate characterization of temperature fluctuations is important in order to estimate the lifetime of pipe material. Thermal fatigue of the coolant circuits of nuclear power plants is one of the major issues in nuclear safety. To investigate thermal fatigue damage, the OECD/NEA-Vattenfall T-Junction Benchmark was initiated to test the ability of state-of-the-art Computational Fluid Dynamics (CFD) codes to predict the important parameters affecting high-cycle thermal fatigue in mixing tees. In this study, to simulate the standard problem described above, the OpenFOAM code is used, which is an open integrated platform for numerical simulation of problems in continuum mechanics. At the first with Salome-meca code, a computational grid was created, consisting of about 450,000 nodes, and k-eps model and RANS models were used to simulate turbulence. OpenFOAM code results were compared with the available experimental results. The results were found to be in well-agreement with the experimental results in terms of amplitude and frequency of temperature and velocity fluctuations.

1. Introduction

Investigation of mixing of two liquid streams with different temperatures in the tee junction is a complex thermophysical problem and is of great practical and theoretical interest. For the tee junction connection of pipelines with mixing of flows, the processes of turbulent mixing and vortex formation are characteristic, accompanied by pulsations of pressure, temperature and velocity. As a result, cyclic thermal stresses arise on the wall of the tee junction, which can cause cracks in the walls of the pipelines. Temperature inhomogeneity of the surface of pipelines is one of the main causes of damage to power equipment. Calculation of temperature fields is an important task from the point of view of ensuring the safety of power equipment. Interest in the subject first arose in the context of Liquid Metal Fast Breeder Reactors (LMFBRs) in the early 1980s, but then received renewed impetus following the incident at the Civaux-1 plant in France in 1998 in which both circumferential and longitudinal cracks appeared near a T-junction in the Residual Heat Removal (RHR) system [1].

T-junction mixing experiments have been conducted at a number of facilities in France, Germany, Sweden, Japan and Switzerland, but previously unreleased test data became available from tests carried out in November 2008 at the Älvkarleby laboratory of Vattenfall Research and Development in Sweden. These became the basis of the blind CFD benchmarking exercise described in this document [2].
several experimental and computational studies investigating the fluid dynamic and thermal effects have
been reported. Experimental T-junction studies can be found in literature (Hu and Kazimi, 2006; Faidy, 2003; Metzner and Wilke, 2005; Zboray et al., 2007; Westin et al., 2008).

2. Computational model

2.1 Material Properties
The experimental setup is manufactured in Plexiglas and consists of a main horizontal pipe with inner
diameter (D2) 140 mm for the cold water flow (Q2), and a vertically oriented pipe with inner diameter
(D1) 100 mm for the hot water flow (Q1). The working fluid in this test is deionized tap water between
19°C and 36°C. Values are displayed in Table 1 for temperatures of relevance.

| Temperature (°C) | Density (kg/m³) | Thermal conductivity (W/m K) | Dynamic viscosity (N s/m²) | Heat capacity (J/kg K) | Molecular Prandtl number (-) |
|-----------------|-----------------|-----------------------------|---------------------------|-----------------------|-----------------------------|
| 19              | 998.49          | 0.5996                      | 1.028E-3                  | 4176                  | 6.988                       |
| 36              | 993.7           | 0.6287                      | 0.7053E-3                 | 4152                  | 4.342                       |

This vertical branch pipe was connected through a tee junction to the middle of the main pipe. The
junction itself is constructed from a solid Plexiglas block into which the main and branch pipes fit. After
the tee, thermocouples were installed in several sections of the main pipe on its side walls, which made
it possible to record the process of mixing hot and cold water [3]. The test rig is illustrated in Figure 1.
2.2 Boundary Condition

The volumetric flow rates used in the test are given in Table 2, together with the locations where the velocity distributions over the pipe cross-sections were measured. The temperatures for the cold and hot inlets are also given at these cross-sections. As a consequence of the thorough flow mixing in the upstream stagnation chambers, the temperatures of the inflowing streams were assumed to be uniform at the measuring locations, and only one thermocouple was placed there. The flow path of hot and cold water and their mixing is shown in Figure 1.

| Inlet/designation | Temperature (°C) | Pipe diameter (mm) | Measuring location (mm) | Volumetric flow rate (litres/s) |
|-------------------|------------------|--------------------|-------------------------|---------------------------------|
| Main/InCo         | 19               | 140 (D2)           | -420 (-3D2)             | 9.0                             |
| Branch/InHo       | 36               | 100 (D1)           | -310 (-3.1D1)           | 6.0                             |

3. Methods and Materials

3.1 OpenFOAM simulation code

OpenFOAM is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics. It has a large user base across most areas of engineering and science, from both commercial and academic organizations. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics [4].

3.2 Salome–meca simulation software

Salome is a free software that provides a generic platform for pre- and post-processing for numerical simulation. It is based on an open architecture made of reusable components. It is open source, released under the GNU Lesser General Public License, and both its source code and binaries may be downloaded from its official website. Salome is a favorite preprocessor without license costs in academic world with Code Aster, Code Saturne and OpenFOAM for hpc with FEM and CFD [5].

To solve this model in OpenFOAM code, the first step is to create the geometry and mesh of the T-Junction. Because of its complexity, shape geometry is best drawn in other Engineering Simulation Software such as Ansys or Salome-meca. In this case, had been used the Salome-meca program to draw the geometry and create the mesh. Below are the results of meshing in Salome-meca. The number of meshes created is about 450,000 nodes. in Figure 2 is shown. After meshing in Salome-meca, the generated mesh is transferred by using the “ideasUnvToFoam pipe2.unv” command in the OpenFOAM code terminals, to continue the calculations in the OpenFOAM. The next step is to choose the right solver. In this study used the “buoyantBoussinesqPimpleFoam” model [6]. This solver used for buoyant, turbulent flow of incompressible fluids.
4. Results and discussion
All experimental data presented in this paper are the property of Vattenfall Research and Development Laboratory and have been used with permission.

4.1 Temperature distribution
Inlet temperatures and temperature fluctuations were recorded using thermocouples in the experiment. Since it was not always possible to keep a constant temperature level for the inflow conditions over time, non-dimensional (normalized) temperature taken to be as a means to compare computational and experimental results. A non-dimensional temperature $T^*$ shall be reported at each thermocouple location. $T^*$ is the actual temperature minus the cold flow inlet temperature, divided by the difference between hot and cold inlet temperatures: that is,
\[ T^* = \frac{T - T_{cold}}{T_{cold} - T_{hot}} \]  

Figure 4. Non-dimensional mean temperatures near the wall obtained from the OpenFOAM code and experimental results (0°: top, 90°: right, 180°: bottom, 270°: left side of the pipe). Blue line – OpenFOAM, red line – experiment.

4.2 Velocity distribution

Figure 6-13 presents horizontal and vertical time averaged velocity profiles at different distances. At the top of the pipe, streaming flow through main pipe is blocked due to high momentum ratio of branch pipe; therefore, velocity magnitude decreases at the top of the pipe, as expected. Momentum ratio of the flows in the main and branch pipes has an influence on the intensity of fluctuation, which consequently affects the intensity of thermal load. As seen in Figures. 12-13, after mixing, fluid comes to hydraulic equilibrium in the neighborhood of x = 4.6D. Hydraulic equilibrium is the state of a system in which velocity profiles reach turbulence velocity profile. This behavior shows that, in thermal mixing cases, flow reaches hydraulic equilibrium first and the thermal equilibrium [3].
Figure 6. Time-Averaged U at x = 1.6D, z = 0

Figure 7. Time-Averaged U at x = 1.6D, y = 0

Figure 8. Time-Averaged U at x = 2.6D, z = 0

Figure 9. Time-Averaged U at x = 2.6D, y = 0

Figure 10. Time-Averaged U at x = 3.6D, z = 0

Figure 11. Time-Averaged U at x = 3.6D, y = 0

Figure 12. Time-Averaged U at x = 4.6D, z = 0

Figure 13. Time-Averaged U at x = 4.6D, y = 0
5. Conclusions

In the present investigation the turbulent mixing of water streams in a T-junction in the horizontal plane have been investigated. A CFD study of turbulent thermal mixing of two water streams having different temperatures in a T-junction was performed and compared with the experimental results. The results of RANS computations resulted in sufficiently accurate predictions of temperature and velocity fluctuations which are important in order to characterize thermal fatigue. Investigations have shown, that Reynolds averaging based RANS turbulence models are able to satisfactorily predict the turbulent mixing of isothermal fluid in T-junctions, while the thermal mixing of water streams of different temperature in T junctions is a challenging test case for CFD methods. Although there are several useful findings about the prediction of thermal load, thermal mixing phenomenon is still a challenging issue for the nuclear technology due to the computation time cost and CPU requirements.

References

[1] Obabko A V, Fischer P F, Karabasov T J S, et al 2011 CFD validation in OECD/NEA T-junction Benchmark Rep Argonne Nat Lab ANL/NE11/25 Lemont
[2] Smith B L, Mahaffy J H, Angele K and Westin J 2011 Report of the OECD/NEA Vattenfall T-junction benchmark exercise Tech Rep OECD/NEA. NEA/CSNI/R 51
[3] Ayhan H and Sokmen C N 2012 CFD modeling of thermal mixing in a T-junction geometry using LES model Nucl. Eng. Des. 253 183–191
[4] OpenFOAM, http://www.openfoam.com
[5] Salome- meca, https://www.salome-platform.org
[6] Sam Khaniani N 2017 Simulation of Flow and Heat Transfer with OpenFOAM 3.x Iran Tehran Andishehsara p 380
[7] Hu L M and Kazimi M S 2006 LES benchmark study of high cycle temperature fluctuations caused by thermal striping in a mixing tee Int J Heat Fluid Flow 27 54–64
[8] Faidy C 2003 Thermal fatigue in mixing tees a step by step simplified Procedure In 11th International Conference on Nuclear Engineering (Tokyo:Japan) 20–23
[9] Metzner K J and Wilke U 2005 European THERFAT project-thermal fatigue evaluation of piping system “Tee”-connections Nucl. Eng. Des. 235 473–484
[10] Zboray R, Manera A, Niceno B and Prasser H M 2007 Investigations on mixing phenomena in single-phase flows in a T-junction geometry Proc. of the 12th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (Pittsburgh PA:USA).
[11] Westin J, Mannetje C, Alavyoon F, Vebre P, Andersson L, Andersson U, Eriksson J, Henriksson M and Andersson C 2008 High-cycle thermal fatigue in mixing tees Large Eddy Simulations compared to a new validation experiment In 11th International Conference on Nuclear Engineering (Orlando FL: USA)