Non-isothermal vortex flow in the T-junction channel

T A Baranova¹, Yu V Zhukova¹, A D Chorny¹, A N Skrypnik³², R A Aksyanov² and I A Popov²

¹A.V. Luikov Heat and Mass Transfer Institute of NAS of Belarus, Minsk, Belarus
²Kazan National Technical University named after A.N. Tupolev – KAI (KNTU-KAI), Kazan, Russian Federation
³Technische Universität Dresden, Institut für Verfahrenstechnik und Umwelttechnik, Dresden, Germany

Abstract. In this work we present the numerical simulation of coolant mixing modes in the T-junction. We shows that the RANS approach is beneficial for a qualitative flow analysis to obtain relatively agreed averaged velocity and temperature. Moreover, traditionally, the RANS approach calculates only the averaged temperature distribution. It should also be emphasized that unlike the LES approach, the steady RANS approach cannot express a local flow structure in intense mixing zones. Nevertheless, apparently the used RANS approach should be used for assessing the quality of computational grids, boundary conditions in order to use the LES approach for further numerical simulation.

1. Introduction
One of the requirements of reliable assessment of coolant thermal and hydraulic parameters in different elements of thermal power equipment (TPE) (boilers, different-construction steam generators, heat exchangers, pipelines, fuel elements, and other heat-loaded elements of nuclear reactor plants, etc.) is that heat loads (first of all, thermal and hydraulic) occurring in such plants can give rise to thermal fatigue and mechanical vibrations under normal TPE conditions [1]. Historically, the hydrodynamic flow and hydraulic shock stability problem (the pipe and pipe junction strength problem) is quite old, for example, see Professor N.E. Zhukovsky’s classical work ‘About hydraulic shock in water pipes’ [2]. Thermal and hydraulic assessments are used as the initial data for solution of cyclic strength and brittle failure tasks with the purpose to analyze the stress-strain state and strength of TPE elements [1]. Convective heat transfer processes in such elements are as a rule accompanied by temperature fluctuations. These fluctuations manifest themselves at boiling crises, unstable steam generation on heated surfaces, coolant flowrate fluctuations, unsteady convective transfer, etc. Temperature fluctuations cause wall temperature stresses to change significantly. In combination with hydraulic load and corrosion of the coolant operating medium, they can destruct TPE elements [1]. This is illustrative of such phenomena as the emergency situations: the failure of the structural elements of a nuclear power plant (NPP Sivo (France) 1998 ; on power unit No. 1, the radiative water leak from the first coolant circuit due to forming pipe cracks; NPP Genkai (Japan), 1998 ; the leak in the capacitor of power unit No.1, 2011; the leak in the cooling system of power unit No. 3; NPP Tange (Belgium), 2012; the outer shell erosion of power unit No. 2; NPP Farley (USA), 2013; the unplanned carbon dioxide emission on power unit No. 2; NPP Tsuruga (Japan), 1981; the radiation emission on power unit No. 2, the discharge
of 16 tons of radiative water; NPP Loviisa (Finland), 1990; accidents on power unit No. 1, the main pipeline failure and erosion-corrosion failures of pipelines at the junction of flow meters; NPP Novovoronezh (USSR), 1990; on power unit No. 5, the weld failure of the pipe valve due to the corrosion-mechanical welding defects (lack of root penetration) developing when acted upon by operation factors).

The task of predicting thermal fatigue and mechanical vibration impact is quite labor-intensive since it needs to determine both hydrodynamic and temperature parameters in near-wall flows of working media, to calculate thermal conductivity and mechanical stresses in solid materials of the walls of TPE elements, as well as in the most unfavorable cases, for example, to analyze the influence on the strength of forming fractures, etc. [1]. Usually, thermal fatigue and mechanical strength are calculated in two main stages. At the first stage, the task of hydrodynamics and heat transfer in flows of working media is solved with allowance for the geometric features of TPE elements. At the second stage, the stress-strain state of TPE elements is determined, when the obtained temperature fields, as well as hydrodynamic ones are used as a load source. In this situation, calculations or numerical simulation of geometrically simple TPE elements, which qualitatively reproduce a separate physical effect, are important for the understanding of this effect, the validity of the used mathematical model, and the reliability of thermal and hydraulic calculation. For example, as applied to NPP reactor plants, thermal fatigue is studied for two test geometric configurations: a) T-junction, b) two or more co-current jets issuing into the wall-bounded space [3]. The T-junction is a widely used component, with the help of which pipelines are combined in different TPE systems.

The objective of the present work was to study turbulent mixing of non-isothermal flows in the T-junction using numerical simulation to determine the distribution of hydrodynamic and temperature parameters depending on the mixing modes in such a junction. Thus, the first-stage task of thermal fatigue prediction in TPE elements was solved. The typical exhaust tee was taken as a variant of the mixing component embodiment of the T-junction. Hot and cold coolants were mixed in the exhaust tee (Figure 1). Numerical simulation was performed, assuming that the inner walls of the T-junction were thermally insulated, not considering the heat removal into the walls. It was assumed that in the inlet cross-section of the T-junction, the coolant flows had different flowrates and temperatures.

![Figure 1. T-junction.](image)

The particular values of the geometric and dynamic parameters corresponded to the WATLON data [4]. Water with a temperature $T_m$ is supplied with a velocity $V_m$ through the main pipe with the inner diameter $D_m = 150$ mm. At the distance $L = 804$ mm from the main pipe entrance at an inclination angle of 90°, the socket is placed – the pipe with the inner diameter $D_b = 50$ mm, through which water is
supplied at a temperature $T_b$ with a velocity $V_b$. According to [4, 5], for all considered cases the flow regime corresponded to the turbulent one.

2. Mixing modes in the T-junction

These mixing modes depend on a number of factors. The velocity ratio $V_m/V_b$ or, as proposed in [6], the momentum ratio $M_m/M_b$ is the defining parameter. A significant influence will also be made by the coolant temperature ratio $T_m/T_b$ at the entrance of the main pipe and of the socket.

Assess the influence of the socket orientation relative to the gravity force action (socket flow direction along or against the gravity force vector). The influence will be significant at natural convection. Since mixing is considered in the water T-junction, the limiting case related to the jet buoyancy within the natural convection regime must be assessed.

At natural convection, the Richardson number $Ri = Gr/Re^2 > 1$, hence $Gr > Re^2$ or, presenting them in the dimensional form, we have $g\beta\Delta TD_b \gg V_b^2$, where $Re$ is the Reynolds number and $Gr$ is the Grashof number. Thus,

$$V_b << \sqrt{g\beta\Delta TD_b} \quad (1)$$

Work [7] contains the value of the water temperature expansion coefficient $\beta$ depending on the temperature and pressure [7]. Substituting the needed numerical values into (1) yields $V_b \ll 0.086\sqrt{\Delta T}$. The temperature drop $\Delta T = 15^\circ$ is known from the WATLON data [4]. Hence, the location of the socket relative to the gravity force direction affects the nature of water issuing from the socket at the issuing velocity $V_b \ll 0.3333$ m/s under the considered experiment conditions. Thus, as the experimental data [4, 6] show, the socket location is not critical when considering the mixing in the zone where the socket is attached to the main pipe of the T-junction, since $V_b \geq 0.5$ m/s for all mixing modes. The contribution of the mass gravity force is relatively small.

According to [6], the following mixing modes determined in [4] and cited in Table 1 will be considered. The near-wall jet regime is characterized by the socket jet pressed against the wall of the main pipe where high momentum is available. In the deflecting jet regime, the momentum flows are compared. Socket flow presses main pipe flow against the upper wall of the channel. Main pipe flow deflects socket flow from the vertical direction, trying to press it against the lower wall of the channel. In the impact jet regime, the socket flow has a very high velocity in comparison to the main pipe flow one (the socket flow is high). The socket jet blocks the entire cross-section of the main pipe and reaches its upper wall. Thus, bearing in mind the momentum $M_m$ in the main pipe and the momentum $M_b$ in the socket, it is possible to judge about the mixing picture of two flows in the T-junction. Such an approach was used for determining the mixing modes [4, 6] (see Table 1). The momentum ratio at the entrance of the main pipe and of the socket was indicated as $M_m/M_b$ where

$$M_m = D_mD_b\rho_mV_m^2, \quad M_b = \frac{\pi}{4}D_b^2\rho_bV_b^2 \quad (2)$$

More traditional for flow regimes is the presentation based on the ratio of the Reynolds numbers constructed through the flow parameters at the entrance of the main pipe and of the socket. The momentum ratio $M_m/M_b$ and the Reynolds number ratio $Re_m/Re_b$ are related as follows:

$$\frac{M_m}{M_b} = \frac{D_mD_b\rho_mV_m^2}{\frac{\pi}{4}D_b^2\rho_bV_b^2} = \frac{4}{\pi}\frac{D_m\rho_mV_m^2}{D_b\rho_bV_b^2} = \frac{Re_m^2}{Re_b^2} \left(\frac{4}{\pi}\frac{D_m\rho_mV_m^2}{D_b\rho_bV_b^2}\right) \quad (3)$$

Table 1 cites the mixing modes vs. the flow Reynolds number ratio in the main pipe and the momentum ratio in the socket.

**Table 1:** T-junction flow regimes according to [4, 6]
Regime | \( \text{Re}_m/\text{Re}_b \) | \( \text{M}_m/\text{M}_b \) | Flow visualization
--- | --- | --- | ---
Near-wall jet | \( \text{Re}_m/\text{Re}_b > 2.36 \) | \( \text{M}_m/\text{M}_b > 1.35 \) | ![Flow visualization](image1)
Deflecting jet | \( 1.2 < \text{Re}_m/\text{Re}_b < 2.36 \) | \( 0.35 < \text{M}_m/\text{M}_b < 1.35 \) | ![Flow visualization](image2)
Impact jet | \( \text{Re}_m/\text{Re}_b < 1.2 \) | \( \text{M}_m/\text{M}_b < 0.35 \) | ![Flow visualization](image3)

3. **Numerical simulation of hydrodynamics and heat transfer in the T-junction**

The Reynolds-averaged steady Navier–Stokes equations, the continuity equation, and the energy equation formulated for enthalpy (RANS approach) were solved using numerical simulation. Boussinesq’s eddy viscosity hypothesis and Menter’s shear stress transport \( \kappa–\omega \) model [8] were used for closure. The thermophysical properties of coolant (density, viscosity, thermal conductivity) were assigned in the form of the temperature-dependent piecewise linear functions.

At the entrance of the main pipe and the socket, the bulk velocity and temperature values were assigned; as turbulent characteristics the level of turbulent fluctuations \( T_u = 1\% \) and the hydraulic diameter were taken to determine the turbulence level at the entrance. The soft boundary conditions in terms of both hydrodynamic and thermal parameters were predetermined at the computational domain outlet. The T-junction walls were assumed to be thermally insulated. It can be noted that, when the additional task of admixture transport in the T-junction at the wall thermal insulation conditions is solved, the temperature and admixture transport processes will be similar. Conservative scalar theory [9] can be used for analyzing transport processes.

Numerical simulation was performed using the gasdynamic solver Ansys Fluent 19.1. Numerical simulation used the finite volume method [10]. The computational grid (Figure 2) was densely packed in the mixing zone of two flows – the one issuing from the main pipe and the other issuing from the socket. The minimum size of the cell in the near-wall boundary layer was 0.1 mm, the maximum cell size in the flow – 4 mm. The cell size was chosen to minimize the influence of artificial viscosity, arising due to the approximation error, on numerical simulation results, i.e. the cell linear size provided small artificial viscosity in comparison to effective (molecular + turbulent) viscosity [11].
Figure 2. Computational grid fragment in the mixing zone of the T-junction.

Thus, coolant steady flow was considered. At that, the iteration convergence of searching for a solution meant that the steady flow regime was present. The fact, showing the convergence absence when solving such a task, revealed that the unsteady mixing mode was substantially unsteady. The task convergence was controlled through the residual value. Calculations were stopped after achieving the pressure residual value of $10^{-4}$ and the temperature residual value of $10^{-8}$.

4. Numerical simulation results of the mixing modes in the T-junction

Within the near-wall jet regime (see Table 1) the jet with the temperature $T_b$ is issuing from the socket into the main flow with the temperature $T_m$ and is pressed against the channel wall as shown in Figure 3. Downstream the main pipe, the socket flow with the temperature $T_b$ near the wall is gradually mixed with the flow with the temperature $T_m$ in the main pipe. As seen from Figure 3, for this mixing mode, the RANS mixing results show that mixing occurs more slowly in comparison to the LES results [5]. In the RANS approach, the mixing zone is longer. This fact is confirmed by the dimensionless temperature distribution $T^* = (T - T_b)/(T_m - T_b)$ in the channel cross-sections at the distances of $0.5D_m$ and $1.0D_m$ downstream behind the socket center (Figure 4). The RANS results are compared to the LES results [6]. As seen from the temperature distributions in the channel cross-sections, the low-temperature region is maintained up to the distance $x = 1.0D_m$. It is found that the region sizes are smaller than those determined by the LES approach [6]. For a more detailed comparison, Figure 5 presents the averaged temperature profile along the vertical line at a distance of $0.5D_m$ behind the socket center. The abscissa axis is the dimensionless temperature $T^*$ and the ordinate axis – the dimensionless distance $y/D_m$.

The experimental and numerical simulation results show that the dimensionless temperature is practically constant in the main flow ($y/D_m > -0.1$) and rapidly decreases at the mixing layer boundary ($-0.3 < y/D_m < -0.1$). The temperature near the pipe wall slightly increases due to the fact that the circulating flow captures some part of the immiscible medium with a higher temperature from the mixing layer (Figure 3). Figure 5 illustrates that regardless of the numerical simulation method used in this region, the predicted results and the WATLON data [4] do not agree – the RANS approach underestimates the temperature value, while the LES approach overestimates it. As described in [12], the LES results obtained with the use of different computational grids practically did not differ between themselves and demonstrated the overestimated temperature value near the main pipe wall. The LES results [13, 14], in which the numerical simulation used the near-wall functions and the computational grids were formed with the thermal boundary layer resolution, also showed the overestimated temperature value near the main pipe wall.
Figure 3. Averaged temperature distribution in the middle section of the T-junction at $\text{Re}_m/\text{Re}_b = 5.76$ obtained by the following approaches: a – LES [5]; b – RANS.

Figure 4. Distribution of the dimensionless temperature $T^*$ at $\text{Rem}/\text{Re}_b = 5.76$: a – LES approach [6]; b – RANS approach

Thus, regardless of the numerical simulation approach (RANS or LES), the problem remains how to adequately describe the temperature field at intense mixing of coolant flows. However, from the viewpoint of solving the thermal fatigue, the LES overestimation of the temperature value can be considered as the positive factor since it allows predicting the start of critical phenomena with some time.
Figure 5. Distribution of the dimensionless temperature $T^*$ in the section at the distance $z = 0.5D_m$ behind the socket center at $Re_m/Re_b = 5.76$: 1 – WATLON data [4]; 2 – RANS approach; 3 – LES approach [5].

The averaged velocity distribution in Figure 6 demonstrates that near the surface of the main pipe downstream behind the socket, there are formed the stagnation zone of the flow issuing from socket and the recirculating flow zone that somewhat presses the main pipe flow, which results in its velocity increase. The latter is seen in the case of both the RANS and LES approaches.

Figure 6. Absolute averaged velocity distribution in the middle section of the T-junction at $Re_m/Re_b = 5.76$: a – LES approach [5]; b – RANS approach.

Figure 7 presents the profile of the dimensionless averaged longitudinal velocity $U_{mean}/U_m$ along the vertical line at the distance $z = 0.5D_m$ downstream behind the socket center. The abscissa axis is the dimensionless averaged velocity and the ordinate axis – the dimensionless distance $y/D_m$. The overestimated velocity value is seen in the main pipe center. It is obtained by the LES approach [5] in comparison to the WATLON data [4] and by the RANS approach. The LES results [13] showed the same behavior, assuming that the numerically simulated profile depends on the velocity profile assigned as the boundary conditions at the main pipe entrance. One of the possible reasons for the differences between experiment and numerical simulation was that in the experiment, the flow at the pipe entrance was not fully developed (the velocity profile is flat). Apparently, this explains the agreement between the RANS results and the experimental data for $y/D_m > -0.1$ since, when the RANS approach was used in our numerical simulation, the velocity was constant at the computational domain inlet.
The flow velocity in the WATLON experiment [4] is almost constant for the main flow ($y/D_m > -0.1$) and decreases in the direction to the wall, at which the socket is placed (Figure 7). In this region, the LES results well enough show both this tendency and the change in the numerical velocity unlike the RANS approach where its value is underestimated. Here, the LES approach advantage is apparently seen from the viewpoint of reflecting the complex local unsteady vortex structure of the flow unlike the RANS approach that yields only the averaged flow pattern.

![Figure 7. Distribution of the dimensionless velocity in the section at the distance behind the socket center for Re_m/Re_b=5.76: 1 – WATLON experiment [4]; 2 – RANS approach; 3 – LES approach [5].](image)

In the case of the deflecting jet, mixing occurs in a peculiar cocoon shaped as a divergent cone downstream behind the socket when a significant change in the dimensionless temperature is not seen near the pipe wall as shown in Figure 8. The dimensionless temperature mainly changes in the central flow part. Moreover, as shown in Figure 8, b, the RANS results qualitatively agree with the LES results.

In the case of the impact jet, the coolant issuing from the socket reaches the opposite wall of the main pipe (see Table 1). This results in the fact that already at a sufficiently small distance downstream the main pipe behind the socket, a mixed medium is formed as shown in Figure 9 at $z = 0.5D_m$. At that, downstream the main pipe, unlike the near-wall jet, the decreased temperature region exists in the upper part of the main pipe (Figure 9). Flows with different temperatures are mixed sufficiently intensely already in the cross-section $z = 1.0D_m$, the main pipe has a pronounced near-wall increased temperature region, a mixing layer, and a decreased temperature region. As shown in Figure 9, b, the RANS results demonstrate a mixing delay in comparison to the LES results [6].

![Figure 8. Distribution of the dimensionless temperature T* at Re_m/Re_b = 5.76: a – LES approach [6]; b – RANS approach](image)
5. Conclusion

The numerical simulation of coolant mixing modes in the T-junction shows that the RANS approach is beneficial for a qualitative flow analysis to obtain relatively agreed averaged velocity and temperature. Moreover, traditionally, the RANS approach calculates only the averaged temperature distribution. The changes in the temperature fluctuations [15] important for the thermal fatigue task are not considered in this mathematical model. It should also be emphasized that unlike the LES approach, the steady RANS approach cannot express a local flow structure in intense mixing zones. Nevertheless, apparently the used RANS approach should be used for assessing the quality of computational grids, boundary conditions in order to use the LES approach for further numerical simulation.

The work was financially sponsored by the Ministry of Education and Science of the Russian Federation as part of the fulfillment of obligations to fulfill obligations under the Agreement 075-03-2020-051/3 dated of 09.06.20 and Belarusian Foundation for Fundamental Research (project F21MS-011).

References
[1] Birger I A 1993 Calculation of the strength of machine parts: handbook (Moscow, Mashinostroenie) 640 p
[2] Zhukovsky N E 1949 About water hit in water pipes (Moscow-Leningrad: GITTL) 104 p
[3] Assessment of computational fluid dynamics (CFD) for nuclear reactor safety problems 2008 Nuclear Energy Agency, Committee on the safety of nuclear installations. (NEA/CSNI/R) 180 p
[4] Igarashi M et al. 2002 Experimental study on fluid mixing for evaluation of thermal striping in T-pipe junction 10th Int. Conf. on Nuclear Engineering: Proc. (Arlington, USA) 22255.
[5] Utanohara Y 2018 Conjugate numerical simulation of wall temperature fluctuation at a T-junction pipe Bulletin of the JSME Mechanical Engineering Journal 5(3) 1–23
[6] Kamide H et al. 2009 Study on mixing behavior in a tee piping and numerical analyses for evaluation of thermal striping Nucl. Eng. Des. 239 58–67
[7] Vihner Ya M 1976 Handbook for hydraulics, hydraulic machines and hydraulic drives (Minsk: Vysheshshaya Shkola) 416 p
[8] Menter F R 2003 Ten years of industrial experience with the SST turbulence model (Begell House, Inc)
[9] Sosinovich V A 2007 Turbulent motion of incompressible media. (Minsk: A.V. Luikov Heat and Mass Transfer Institute of NAS of Belarus) 189 p
[10] Patankar S V 1991 Computation of Conduction and Duct Flow Heat Transfer (CRC Press) 370 p
[11] Aksenov A A et al. 2017 Numerical investigations of mixing non-isothermal streams of sodium

Figure 9. Distribution of the dimensionless temperature $T^*$ at $Re_m/Re_b = 0.91$: a – LES approach [6]; b – RANS approach
coolant in T-branch Computer Research and Modeling. 9(1) 95–110
[12] Utanohara Y, et al. 2016 Numerical simulation of long-period fluid temperature fluctuation at a mixing tee for the thermal fatigue problem Nucl. Eng. Des. 305 639–652
[13] Nakamura A et al. 2010 Benchmark simulation of temperature fluctuation using CFD for the evaluation of the thermal load in a T-junction pipe Seventh Korea-Japan Symp. on Nucl. Therm. Hydr. and Saf. Proc. (Chuncheon, Korea) N7P-0011.
[14] Tanaka M 2010 Thermal mixing in T-junction piping system concerned with high-cycle thermal fatigue in structure J. Nucl. Sci. Technol. 47(9) 790–801.
[15] Qian S 2015 High-accuracy CFD prediction methods for fluid and structure temperature fluctuations at T-junction for thermal fatigue evaluation Nucl. Eng. Des. 288 98–109.