Annals of Nuclear Energy 124 (2019) 245–256

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene

Detached eddy simulation of turbulent and thermal mixing in a T-junction

Dong Gu Kang [a,b], Hanbee Na [a], Chi Young Lee [c,*]

[a] Korea Institute of Nuclear Safety, 62 Gwahak-ro, Yuseong-gu, Daejeon 34142, Republic of Korea
[b] Nuclear and Radiation Safety Department, University of Science and Technology, 217 Gajeong-ro Yuseong-gu, Daejeon 34113, Republic of Korea
[c] Department of Fire Protection Engineering, Pukyong National University, 45 Yongso-ro, Nam-gu, Busan 48513, Republic of Korea

A R T I C L E I N F O

Article history:
Received 11 May 2018
Received in revised form 5 September 2018
Accepted 1 October 2018

Keywords:
DES
Vattenfall T-junction test
CFD
Turbulent thermal mixing
Mixing tee

A B S T R A C T

Turbulent thermal mixing is one of the major degradation mechanisms of thermal fatigue, called high cycle thermal fatigue, and a mixing tee has been known as typical component susceptible to high cycle thermal fatigue. From a numerical analysis point of view, accurate prediction of turbulent flow and associated thermal fields in a T-junction is an essential task; that requires computational fluid dynamics (CFD) with advanced turbulence modeling. The detached eddy simulation (DES) model is hybrid turbulence model which combines classical Reynolds Averaged Navier-Stokes (RANS) formulations with elements of large eddy simulation (LES) method. The DES model has a benefit from computational cost point of view, but the studies of its applicability to industrial problems seem to have been conducted relatively insufficiently. Therefore, in this study, transient CFD analysis using the DES model was performed against Vattenfall T-junction test, and the applicability of DES model to turbulent thermal mixing was evaluated by comparing with its experimental data. For the comparison of velocities, the DES results were in good agreement with the experimental data. For the comparison of temperature, the calculated results were generally in good agreement, but at separation region, a large difference of mean temperature was observed. For the locations where the wall temperature variation is large in which the risk of thermal fatigue is expected to be higher, it was seen that the low frequency oscillations are dominant and the energy begins to decrease from ~4 Hz. In conclusions, it was confirmed that the DES turbulence model has a capability to simulate turbulent thermal mixing phenomenon in a mixing tee and the CFD analysis using that model can provide reliable results for the assessment of the structural integrity of such a piping system.

1. Introduction

Thermal fatigue is a significant degradation mechanism affecting structural failure of various components and systems, and it mainly occurs in the piping system where fluids with different temperature meet together; these thermal loads generate cyclic mechanical stresses on the pipe wall over a long period and it can lead to crack initiation and propagation. Most of thermal fatigue stresses are caused by following thermal loads.

(1) thermal shock induced by significant temperature gradient along the axial direction of the pipe, which occurs at the coolant injection with high flow rate such as safety injection into reactor coolant system or spray in the pressurizer
(2) thermal stratification due to the fluid density difference with low flow rate, which mainly results in global bending deflection and local pear-type deformation in horizontal portions of piping system such as surge line and spray line of the pressurizer
(3) turbulent thermal mixing characterized by random fluid motion and temperature variation, which occurs in certain piping system such as T-junction where two flows with different temperature mix together.

Since the thermal fatigue due to thermal shock loading and thermal stratification have been well understood through a lot of research, they have been considered in design process and can be monitored by plant instrumentation systems. However, turbulent thermal mixing is difficult to be taken into account in conventional simplified fatigue analysis due to its irregularity and complexity.

* Corresponding author.
E-mail address: cylee@pknu.ac.kr (C.Y. Lee).

https://doi.org/10.1016/j.anucene.2018.10.006
0306-4549/

Turbulent thermal mixing is often referred to as thermal striping, and fatigue due to that is called high cycle thermal fatigue. In nuclear engineering society, thermal striping was initially issued for liquid metal cooled fast breeder reactors in the 1980s, and interest has shifted into light water reactors after several incidents of piping failure in nuclear power plants such as Civaux Unit 1 in France and Tsuruga Unit 2 in Japan, and so on (Claude, 2003; Hu and Kazimi, 2006; Muramatsu and Ninikata, 1996; Sugano et al, 2000). A mixing tee has been known as typical component susceptible to high cycle thermal fatigue and that of residual heat removal system is seen to be most vulnerable part in nuclear power plants (Hu and Kazimi, 2006). In order to evaluate high cycle thermal fatigue, the magnitudes and frequencies of coolant temperature fluctuations near a piping wall should be identified; to do that many experimental mock-up tests and numerical simulations have been conducted. From a numerical analysis point of view, accurate prediction of turbulent flow and associated thermal fields in a T-junction is an essential task; that requires computational fluid dynamics (CFD) with advanced turbulence modeling.

Since the CFD could yield various results depending on the methodologies such as turbulence model, discretization scheme, mesh resolution, and so on, many benchmark studies against experiments have been conducted to confirm the applicability of CFD to thermal striping problem. Among those studies, the T-junction thermal mixing test carried out by Vattenfall Research and Development, has been paid attention because it was selected as OECD/NEA sponsored CFD benchmark exercise (OECD/NEA, 2009); so the corresponding CFD simulations have been performed (Ayhan and Sökmen, 2012; Frank et al., 2010; Gritskevich et al., 2014; Höhne, 2014; Jayaraju et al., 2010; Kim and Jeong, 2012; Kuczaj et al., 2010; Ndombo and Howard, 2011; Sakowitz et al., 2014). The CFD studies against other various experiments have also been conducted (Hu and Kazimi, 2006; Kuhn et al., 2010; Naik-Nimbalkar et al., 2010; Selvam et al., 2015). The results of these studies can be summarized as follows.

(1) Unsteady Reynolds Averaged Navier-Stokes (URANS) based approach has difficulties in simulating fluctuating velocities and temperatures clearly observed in the experiments.
(2) large eddy simulation (LES) model has capability to predict flow and temperature field in the T-junction and amplitude and frequency of fluctuations accurately.
(3) fluctuations near the wall exhibited dominant frequencies of several Hz or Strouhal number in order of 0.5.
(4) wall-resolved approach gives more accurate results than wall function based simulation for predicting fluctuations close to the wall.
(5) inlet conditions, especially inlet turbulence doesn't affect the bulk parameters much, but it has an effect on the near wall flow.

However, the accurate prediction of turbulent flow and thermal mixing using CFD is still challenging task, and the validation of CFD method is still required.

If only considering the prediction accuracy of turbulent flow, direct numerical simulation (DNS) would be the best approach, but it is not practical to high Reynolds number problem because it requires tremendous number of meshes to directly resolve the whole range of turbulence scales. Therefore, in most of previous studies, the LES method have been widely used. In LES, large eddies are explicitly calculated, while small scales are modeled by using a subgrid-scale model. Since only large eddies are resolved in the LES, much coarser mesh and larger time-step can be used compared with those of DNS. However, for many industrial application, it still requires a large number of meshes and its computational cost is much higher than that of RANS based simulation; especially, experience has shown that the use of LES in boundary layer flows at high Reynolds numbers is prohibitively expensive (Spalart et al., 1997). Meanwhile, detached eddy simulation (DES) model has been developed to overcome this limitation of LES turbulence model, and it has hybrid approach which combines classical RANS formulations with elements of LES method (Spalart et al., 1997; Strelets, 2001; Spalart et al., 2006; Menter, 2012). However, the studies of its applicability to industrial problems seem to have been conducted relatively insufficiently (e.g. only one out of 29 submissions in OECD/NEA T-junction project). Therefore, in this study, transient CFD analysis using the DES turbulence model was performed against Vattenfall T-junction test, and the applicability of DES model to turbulent thermal mixing was evaluated by comparing with its experimental data.

2. Vattenfall T-junction thermal mixing test

The facility of Vattenfall T-junction thermal mixing test is illustrated in Fig. 1. Cold water of 19 °C was supplied through a horizontal pipe with inner diameter 140 mm (D_m in Fig. 1), and hot water of 36 °C was provided from a vertically oriented pipe with inner diameter 100 mm (D_b in Fig. 1). The tee section was made from Plexiglas block as shown in Fig. 1. The upstream length of horizontal pipe was more than 80 pipe diameters (fully developed flow), and that of vertical one was approximately 20 pipe diameters (developing flow). The inlet volumetric flow rates of cold and hot water were 0.009 and 0.006 m^3/s, respectively. Velocity profiles upstream of the test section were measured using laser doppler velocimetry (LDV) at 3 and 3.1 diameters upstream of cold and hot inlet pipes, respectively. For downstream velocity, particle image velocimetry (PIV) measurements have been used at 1.6, 2.6, 3.6, and 4.6 diameters downstream of the test section. Temperature fluctuations near the pipe walls were measured using thermocouples located 1 mm from the wall at 2, 4, 6, 8, 10, 15, and 20 diameters and around circumference of the pipe such as 0°, 90°, 180°, and 270° (OECD/NEA, 2009).

3. Turbulence model and CFD methodology

3.1. DES turbulence model

The DES model has been originally proposed by Spalart et al. (1997). The DES model functions as a subgrid-scale model in regions where the maximum grid spacing is much smaller than the flow turbulence length-scale, and as a RANS model in regions where it is not (Strelets, 2001). In original Spalart-Allmaras (S-A) model (Spalart et al., 1997), the distance to the closest wall, d_w, is replaced by new DES length scale, l;

$$l = \min(d_w, C_{DES}\ \Delta) \tag{1}$$

$$\Delta = \max(\Delta_x, \Delta_y, \Delta_z) \tag{2}$$

where C_{DES} is a constant and Δ is the maximum length of local grid cell. Therefore, in the attached boundary layer, due to the significant grid anisotropy ($\Delta_x \approx \Delta_y \gg \Delta_z$), in accordance with (1), $l = d_w$, and the model reduces to the RANS model. Otherwise, once a field point is far enough from wall ($d_w > C_{DES}\Delta$), the model performs as a subgrid-scale version of the S-A model (Strelets, 2001).

Strelets (2001) has proposed new idea of the model that the DES length scale can be obtained by turbulent length predicted by RANS model instead of d_w. In that work, the $k-\omega$ shear stress transport (SST) model was adopted, because it is consistently considered as one of the best two-equation RANS models, particularly for separation prediction (Menter, 1993; Strelets, 2001). In the