Nuclear Engineering and Technology xxx (2017) 1–17

Contents lists available at ScienceDirect

Nuclear Engineering and Technology

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

Original Article

Numerical analysis of two experiments related to thermal fatigue

Q11 Ulrich Bieder*, Paolo Errante

Q1 *DEN-STMF, Commissariat à l'Énergie Atomique et aux Énergies Alternatives, Université Paris-Saclay, F-91191 Gif-sur-Yvette, France*

ARTICLE INFO

Article history:
Received 7 October 2016
Received in revised form
11 December 2016
Accepted 30 January 2017
Available online xxx

1. Introduction

Q2 During the past few decades, thermal fatigue issues have been observed in components of nuclear power plants in places where two fluids at different temperatures were mixed. If not detected in time, temperature fluctuations driven by turbulent flow can induce thermal stress cycles on wall material and can provoke crack initiation with consecutive structural weakening. T-junctions are a typical component where hot and cold fluids come together in the form of jets in crossflow and where temperature fluctuations can act on walls. Several failures have been observed in nuclear reactor coolant systems downstream of safety injection lines, which are connected via T-junctions to the primary circuit. Leakage incidents due to thermal fatigue in such T-junctions have occurred in France, such as in 1998 in the light water reactor at Civaux [1] and in 1991 in the sodium-cooled fast breeder PHENIX reactors [2]. Several experiments have been carried out to analyze temperature fluctuations in such mixing tees. Related to Civaux, temperature fluctuation and crack behavior were examined in a scaled model experiment under reactor conditions of pressure and temperature. Flow velocity field was measured separately in a small scale and isothermal experiment [3]. The PHENIX incident was the subject of intense numerical analysis including the organization of an international IAEA benchmark [2].

The most significant thermal–hydraulic parameters that play important roles in the formation and subsequent progression of structural damage are related to the thermal gradient and its time derivative. Thus, temperature differences between flows (ΔT), fluctuating frequencies (Ω) and number of cycles (N) are determining variables to be used in thermal fatigue analysis. The various experiments [3] demonstrated that, typically, thermal loads characterized by ΔT of about 160 K and Ω between 3 Hz and 10 Hz are favorable conditions for crack initiation. Besides the aim of determining the critical intervals of these parameters, experimental data have also been produced to validate computational fluid dynamics (CFD) codes and to initiate further model development. In fact, CFD tools show the interesting potential to predict thermal striping problems correctly, which thereby can be avoided. CFD application can lead to a deeper comprehension of complex operating conditions and geometries, which are difficult and/or expensive to reproduce in experimental investigations [4].

To obtain consistent time-averaged fields, turbulent flows can be described with good accuracy and small computational costs by using two-equation turbulence models based on Reynolds averaged Navier–Stokes (RANS) equations. To obtain reasonable values for thermal fatigue analysis in terms of local temporal fluctuations, large eddy simulations (LES) show the potential to find the desired parameters, regardless of its more expensive computational costs. Based on these two turbulence modelling approaches, one calculation methodology was developed at Commissariat à l'Énergie Atomique (CEA) to analyze thermal fatigue phenomena with CFD for *new applications*. This methodology consists of two successive steps:

1) A well-documented reference experiment that is physically close to the new application is analyzed. Two types of calculations are performed:
   a) Preliminary RANS calculations to understand the flow field globally as well as to evaluate details on the meshing requirements and the boundary condition treatments. These calculations are called *scaling calculations.*
   b) Concluding LES to obtain access to turbulence statistics. These calculations are called *analyzing calculations.*
2) The new geometry is then analyzed mainly by LES, supported by some selected RANS calculations, and taking into account the previously obtained experience. These calculations are called *production calculations.*

This method is presented here in detail. Advantages and shortcomings of the method and of the turbulence modelling

* Corresponding author.
E-mail address: Ulrich.BIEDER@cea.fr (U. Bieder).

http://dx.doi.org/10.1016/j.net.2017.01.018

approaches LES and RANS are discussed for the example of flows in two experimental facilities: the Vattenfall T-junction experiment [5] serves as a well-documented reference experiment and the CEA TRANSAT rectangular jet experiment [6] serves as the new application.

Q3

2. The numerical approach

2.1. The TrioCFD code

The TrioCFD (http://www-trio-u.cea.fr/) code [7] is used to perform calculations for both experimental test cases. Turbulence is treated either by RANS equations with the linear eddy viscosity k-ε model or by LES.

2.2. Unsteady Navier–Stokes equations

The fluid is assumed to be incompressible and Newtonian. Buoyancy effects are not taken into account. The instantaneous velocity u of this fluid can be expressed by the equation of mass conservation [Eq. (1)] and the equation of momentum conservation [Eq. (2)]. Einstein's notation is used.

Q4

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\upsilon_{eff}\cdot\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

The effective viscosity υ_{eff} is the kinematic viscosity of the fluid and is defined by the turbulence model.

2.3. RANS equations

In Reynolds-averaged turbulence approaches, the nonlinearity of the Navier–Stokes equations gives rise to Reynolds stress terms that are modeled by turbulence models. To model the Reynolds stress, almost all turbulence models for industrial applications are based on Boussinesq's concept of eddy-viscosity, which assumes that the Reynolds stresses are aligned with the main strain rates:

$$-\overline{u_i' u_j'} = \upsilon_T\cdot\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}k\delta_{ij} \tag{3}$$

This approach leads to the Reynolds averaged mass conservation equation and the Navier–Stokes equations. For the RANS approach, Eqs. (1, 2) are written for the Reynolds averaged velocity U_i and $\upsilon_{eff} = \upsilon + \upsilon_t$. In the study presented here, the turbulent viscosity is calculated from the well-known k-ε model by using the following formulation:

$$\upsilon_t = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

$$\frac{\partial k}{\partial t} + \frac{\partial (U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_t}{\sigma_k}\right)\cdot\frac{\partial k}{\partial x_j}\right] - \varepsilon + P \tag{5}$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial (U_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_t}{\sigma_\varepsilon}\right)\cdot\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\cdot P\cdot\frac{\varepsilon}{k} - C_{\varepsilon 2}\cdot\frac{\varepsilon^2}{k} \tag{6}$$

$$P = -\overline{u_i' u_j'}\cdot\frac{\partial U_i}{\partial x_j}, \text{ with } \overline{u_i' u_j'} \text{ calculated by Eq. (3)} \tag{7}$$

The following empirical coefficients are used: $C_\mu = 0.09$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$.

2.4. Filtered Navier–Stokes equations

In LES, a filtering operation is applied to the instantaneous turbulent quantities of Eqs. (1, 2). The subgrid-scale stress tensor τ_{ij} that appears is calculated using an analogy to the Boussinesq eddy viscosity concept:

$$\tau_{ij} = -\left(\overline{u_i\cdot u_j} - \overline{u_i}\cdot\overline{u_j}\right) = \upsilon_{SGS}\cdot\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \frac{1}{3}\tau_{ii}\delta_{ij} \tag{8}$$

Then, for LES, Eq. (2) is written for the filtered velocity $\overline{u}_i$ and $\upsilon_{eff} = \upsilon + \upsilon_{SGS}$. With the aim of better reproducing the transition from laminar to turbulent flow and obtaining a correct wall-asymptotic-behavior of the turbulent viscosity, the wall adaptive local eddy-viscosity model [8] is applied. This model offers advantages of the dynamic Smagorinsky model without requiring explicit filtering operations. The turbulent viscosity of the wall adaptive local eddy-viscosity model is calculated according to the following equations ($C_W = 0.5$):

$$\upsilon_{SGS} = \left(C_W \Delta\right)^2\cdot\frac{\left(s_{ij}^d\cdot s_{ij}^d\right)^{3/2}}{\left(S_{ij}\cdot S_{ij}\right)^{5/2} + \left(s_{ij}^d\cdot s_{ij}^d\right)^{5/4} + 10^{-6}} \quad \text{with} \tag{9}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \text{ and } s_{ij}^d = \frac{1}{2}\cdot\left(\left(\frac{\partial \overline{u}_i}{\partial x_j}\right)^2 + \left(\frac{\partial \overline{u}_j}{\partial x_i}\right)^2\right) - \frac{1}{3}\cdot\left(\frac{\partial \overline{u}_i}{\partial x_i}\right)^2\cdot\delta_{ij} \tag{10}$$

2.5. Numerical solution of the conservation equations

2.5.1. Discretization method

TrioCFD [7] uses a finite volume based finite element approach on tetrahedral cells to integrate in conservative form all conservation equations over the control volumes belonging to the calculation domain. As in the classical Crouzeix–Raviart element, both vector and scalar quantities are located in the centers of the faces. The pressure, however, is located in the vertices and at the center of gravity of a tetrahedral element, as shown in [9] for the two-dimensional case. This discretization leads to very good pressure/velocity coupling and has a very dense divergence free basis. Along this staggered mesh arrangement, the unknowns, i.e. the vector and scalar values, are expressed using nonconforming linear shape-functions (P1-nonconforming). The shape function for the pressure is constant for the center of the element (P0) and linear for the vertices (P1).

2.5.2. Convection, diffusion and time scheme

For RANS calculations, the first-order Euler backward implicit scheme is used for the time integration. This scheme ensures good stability of the steady state solution. A second-order MUSCL type convection scheme is applied. The diffusion term is discretized by a second-order centered scheme. For LES, the second-order explicit Adams–Bashforth scheme is used for time integration. The used time step respects the Courant–Friedrichs–Levy stability criteria < 0.8. A slightly stabilized second-order centered convection scheme is applied. The diffusion term is discretized by a second-order centered scheme.

Please cite this article in press as: U. Bieder, P. Errante, Numerical analysis of two experiments related to thermal fatigue, Nuclear Engineering and Technology (2017), http://dx.doi.org/10.1016/j.net.2017.01.018