Nuclear Engineering and Design 299 (2016) 201–213

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Reynolds stress turbulence model applied to two-phase pressurized thermal shocks in nuclear power plant

Nicolas Mérigoux*, Jérôme Laviéville, Stéphane Mimouni, Mathieu Guingo, Cyril Baudry

Electricité de France, R&D Division, 6 Quai Watier, 78401 Chatou, France

HIGHLIGHTS

- NEPTUNE_CFD is used to model two-phase PTS.
- k-ε model did produce some satisfactory results but also highlights some weaknesses.
- A more advanced turbulence model has been developed, validated and applied for PTS.
- Coupled with LIM, the first results confirmed the increased accuracy of the approach.

ARTICLE INFO

Article history:
Received 17 June 2015
Accepted 3 July 2015
Available online 6 August 2015

ABSTRACT

Nuclear power plants are subjected to a variety of ageing mechanisms and, at the same time, exposed to potential pressurized thermal shock (PTS) – characterized by a rapid cooling of the internal Reactor Pressure Vessel (RPV) surface. In this context, NEPTUNE_CFD is used to model two-phase PTS and give an assessment on the structural integrity of the RPV. The first available choice was to use standard first order turbulence model (k-ε) to model high-Reynolds number flows encountered in Pressurized Water Reactor (PWR) primary circuits. In a first attempt, the use of k-ε model did produce some satisfactory results in terms of condensation rate and temperature field distribution on integral experiments, but also highlights some weaknesses in the way to model highly anisotropic turbulence. One way to improve the turbulence prediction – and consequently the temperature field distribution – is to opt for more advanced Reynolds Stress turbulence Model. After various verification and validation steps on separated effects cases – co-current air/steam-water stratified flows in rectangular channels, water jet impingements on water pool free surfaces – this Reynolds Stress turbulence Model (R_{ij}-ε SSG) has been applied for the first time to thermal free surface flows under industrial conditions on COSI and TOPFLOW-PTS experiments. Coupled with the Large Interface Model, the first results confirmed the adequacy and increased accuracy of the approach in an industrial context.

1. Introduction

Nuclear power plants (NPP) operating equipments are subjected to a variety of ageing mechanisms. Ageing effects of the Reactor Pressure Vessel (RPV) have the potential to be a life-limiting condition for a NPP as the RPV is impossible or economically unviable to replace pressurized thermal shock (PTS) transient is characterized by a rapid cooling (i.e. thermal shock) of the downcomer and internal RPV surface, followed sometimes by re-pressurization of the RPV. Thus, a PTS event raises a potentially significant challenge to the structural integrity of the RPV in a Pressurized Water Reactor (PWR). In this context, NEPTUNE_CFD is developed – within the framework of the NEPTUNE project (Guelfi et al., 2007) – and used to model two-phase PTS in an industrial configuration and provide temperature and pressure fields – required to assess the integrity of the RPV. In the same time, experimental programmes such as COSI and TOPFLOW-PTS have been set up to provide additional relevant data to further develop two-phase CFD modelling in such a configuration, and validate it.

The first available choice was to use standard first order turbulence model (k-ε) to model high-Reynolds number flows encountered in PWR primary circuits. In a first attempt, the use of k-ε model did produce some satisfactory results in terms of condensation rate and temperature field distribution on separated effects and integral experiments. These computations also highlight that the way to model turbulence – particularly in the

* Corresponding author. Tel.: +33 130879101.
E-mail address: nicolas.merigoux@edf.fr (N. Mérigoux).

http://dx.doi.org/10.1016/j.nucengdes.2015.07.015
0029-5493/

impinging jet areas, where an overestimation of the turbulence production is expected – can have a significant impact on the results. One way to improve the turbulence prediction – and consequently the temperature field distribution – is to opt for more advanced Reynolds Stress turbulence Model (RSM). This allows to model highly anisotropic turbulence that is encountered in PWR and integral experiments such as COSI and TOPFLOW-PTS.

To this end and consistently with the work already done on the k-ε model (Coste et al., 2010), this paper presents NEPTUNE_CFD code validation against six different experiments which have been selected to allow separated effects and integral validations. To add some confidence in the validation results, a verification step has first been performed on some very simple 1D stratified channel flows. It mainly allows us to verify the velocity and turbulence fields' behaviour near the boundaries and at the interface. The next four cases are useful for separated effects validation dedicated to two-phase PTS. The Fabre et al. (1987) experiment is a co-current smooth and wavy Air/Water STratified (AWST) flow in a rectangular channel with detailed measurements of turbulence and velocities. The Lim et al. (1984) experiment is also a co-current smooth and wavy Steam/Water STratified (SWST) flow in a rectangular channel with measurements of the steam flow rates at six axial positions along the channel. The Bonetto and Lahey (1993) and the Iguchi et al. (1998) experiments deal with a water jet impingement on a water pool free surface in air environment. In the first one, radial void fraction profiles are given at different heights, whereas in the second one, axial mean velocity and RMS turbulence components are measured. Finally, this Reynolds Stress turbulence Model (R_{ij}-ε SSG), coupled with the Large Interface Model (LIM) has been applied for the first time to thermal free surface flows on COSI and TOPFLOW-PTS experiments – representative of a French 900 MWe PWR, respectively scaled 1/100 and 1/2.5 – under Loss Of Coolant Accident (LOCA) conditions for a more integral validation. The measurements include temperature profiles at different positions and global condensation rates.

As expected, the first results obtained to model PTS scenarios seem to go in the right way and a more in-depth validation has been launched to get a general tendency on various test cases and industrial scenarios. Moreover, this approach for modelling turbulence – using RSM – shows that it is almost not more CPU time-consuming than using first order turbulence model. It has also shown that it does not deteriorate the numerical stability of computations (which is crucial to achieve robustness in industrial configurations).

2. Model description

The purpose of this paper being to present the validation and use of RSM with NEPTUNE_CFD in a PTS context, the following section will only describe the main principles of the used models – without any details on the governing equations. Next sections will be dedicated to the verification and validation steps.

2.1. Two-phase model and solver

NEPTUNE_CFD is a three dimensional two-fluid code developed more especially for nuclear reactor applications. This code is based on the classical two-fluid one-pressure approach (Ishii, 1975), (Delhaye et al., 1981), and is able to simulate multi-component multiphase flows by solving a set of three balance equations for each field (Méchitoua et al., 2003), (Guelfi et al., 2007), (Mimouni et al., 2009a,b). These fields can represent many kinds of multiphase flows: distinct physical components (e.g. gas, liquid and solid particles); thermodynamic phases of the same component (e.g. liquid water and its vapour); distinct physical components, some of which split into different groups (e.g. water and several groups of different bubble diameters); different forms of the same physical components (e.g.: a continuous liquid field, a dispersed liquid field, a continuous vapour field, a dispersed vapour field).

The discretization follows a 3D full-unstructured finite-volume approach, with a collocated arrangement of all variables. Numerical consistency and precision for diffusive and advective fluxes for non-orthogonal and irregular cells are taken into account through a gradient reconstruction technique. Convective schemes for all variables, except pressure, are centred/upwind scheme. Velocities components can be computed with a full centred scheme. The solver is based on a pressure correction fractional step approach. Gradients are calculated at second order for regular cells and at first order for highly irregular cells.

A set of local balance equations for mass, momentum and energy is written for each phase. These balance equations are obtained by ensemble averaging of the local instantaneous balance equations written for each phase. When the averaging operation is performed, the major part of the information about the interfacial configuration and the microphysics governing the different types of exchanges is lost. As a consequence, a number of closure relations must be supplied for the total number of equations (the balance equations and the closure relations) to be equal to the number of unknown fields. We can distinguish three different types of closure relations. Those which express the inter-phase exchanges (interfacial transfer terms), those which express the intra-phase exchanges (molecular and turbulent transfer terms) and those which express the interactions between each phase and the walls (wall transfer terms) (Mimouni et al., 2009b).

2.2. Large interface model

Two-phase PTS CFD scenarios involve interfaces between liquid and vapour which are generally much larger than the computational cells size: the “large interfaces”. Specific models to deal with them were developed and implemented in NEPTUNE_CFD: it is the Large Interface Model (LIM) (Coste, 2013). It includes large interface recognition, interfacial transfer of momentum (friction), heat and mass transfer with Direct Contact Condensation (DCC). The LIM takes into account large interfaces which can be smooth, wavy or rough.

Regarding the interface recognition, the method implemented in NEPTUNE_CFD is based on the gradient of liquid fraction. The first step consists in computing a refined liquid fraction gradient, based on harmonic or anti-harmonic interpolated values of liquid fraction on the faces between the cells (Laviéville and Coste, 2008). This refined gradient allows us to detect the cells belonging to the Large Interface (LI). The models – specific LI's closure laws – developed and implemented in NEPTUNE_CFD (Coste et al., 2007, 2008; Coste and Laviéville, 2009) are written within a three-cell stencil (LI3C) around the large interface position (including the two liquid and vapour neighbouring cells located in LI's normal direction). This stencil is used to compute, on both the liquid and gas sides, the distance from the first computational cell to the large interface. Both distances are used in the models written in a wall law-like format. In this manner, only physically relevant values are used by choosing the interface side where the phase is not residual and the effect of the LI's position with regard to the mesh is limited.

2.3. Turbulence model

When solving the momentum equation, a turbulent stress tensor has to be modelled (or neglected when the flow is considered laminar). Several turbulence models are available in NEPTUNE_CFD to solve all kind of high-Reynolds multiphase flows. Regarding