IJCSS International Journal of Complex Systems in Science

Int. J. Complex Systems in Science
vol. **2(1)** (2012), pp. 43–48

Turbulence modelling of a thermal stratification CFD model

Mónica Martínez[1,†], Rafael Miró[1], Teresa Barrachina[1],
Sergio Chiva[2] and Gumersindo Verdú[1]

[1] *Institute for Industrial, Radiophysical and Environmental Safety (Isirym), Universitat Politècnica de València*

[2] *Mechanical Engineering and Construction Department, Universitat Jaume I*

Abstract. The OECD/NEA ROSA project test 1-1 was conducted in 2006 with the objective to obtain the multidimensional temperature distributions in cold legs and downcomer during the Emergency Core Cooling System (ECCS) water injection in a Pressurized Water Reactor (PWR) for verification of computer codes and models. In this paper, 3D Computational Fluid Dynamics (CFD) study of OECD ROSA (Rig-of- Safety Assessment) project test 1-1, using the commercial CFD code Ansys-CFX v13 is presented. The analysis was focused on the turbulence models which are the most relevant physical models responsible for models errors. Steady-state calculations were performed with different turbulence models: Standard $k-\varepsilon$, RNG $k-\varepsilon$, Shear Stress Transport and Reynolds Stress Model. Numerical results for all the turbulence models selected could be considered satisfactory for the prediction of thermal stratified flow. However, it is necessary to establish a procedure to evaluate the error and uncertainty due to aspects as mesh refinement, time step and turbulence models.

Keywords: turbulence modelling, CFD codes, thermalhydraulics, nuclear engineering
MSC 2000: 76TXX, 76FXX, 65KXX, 76RXX, 80A20

† **Corresponding author:** momarlia@iqn.upv.es
Received: November 26th, 2012
Published: December 17th, 2012

1. Introduction

Commercial CFD codes are employed in engineering and scientific applications as aerospace, automotive, chemical and nuclear industries to research, simulate and optimize their processes. The focus is on the fluid variables (density, pressure, velocity, temperature or energy) but grid generation, turbulence modelling and solver parameters play an important role on the accuracy and stability of the solution.

One of the CFD application areas for nuclear engineering purposes is the study of the Pressurized Thermal Shock (PTS) in a Pressurized Water Reactor (PWR). A PTS can occur during Loss of Coolant Accident (LOCA), when cold water injection from the Emergency Core Cooling System (ECCS) is needed. The thermal stratification of the cold leg flow could provoke the growth of a cold plume in the PWR downcomer. The pressure vessel in contact with the cold plume suffers a fast cooling at high pressure, affecting the structural integrity of the PWR vessel.

OECD ROSA (Rig-of Safety Assesment) project was carried out in the Japanese ROSA/LSTF (Large-Scale Test Facility) to obtain an experimental database to validate code predictive capability and accuracy for the simulation of temperature stratification during ECCS water injection [1]. In this paper, the influence of turbulence models on the results of the OECD ROSA test 1-1 project using the commercial CFD code Ansys-CFX v13 are presented. The study proposes the evaluation of several Reynolds Average Navier Stokes (RANS) turbulence models: Standard $k{-}\varepsilon$, RNG $k{-}\varepsilon$, Shear Stress Transport Model (SST) and two Reynolds Stress Models (RSM).

2. Turbulence modelling

An important aspect to solve CFD equations is the phenomenon of the turbulence in a fluid flow. Turbulence appears in most of the natural fluid flows and consists of random fluctuations of the flow properties. Also, it is inherently three dimensional and time dependent. There are several approaches for the turbulence based on the detail to capture the phenomenon [2]; however the computational resources increase with the complexity of the method. Commercial CFD codes use RANS simulations based on the solution of the average Navier-Stokes equations in a reasonable computational time. The momentum flow configuration can be represented by the Reynolds average Navier Stokes equations for turbulent flow, in Cartesian coordinates as follow:

$$\rho\frac{\partial U_i}{\partial t} + \rho U_j\frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2\mu S_{ji} - \overline{\rho u'_j u'_i}\right) \tag{1}$$

where ρ is the density, U is the average part of the velocity, i and j are the indexes for the Cartesian directions, P is the average part for the pressure,the $2\mu S_{ji}$ represents the viscosity term and the last term is known as the Reynolds-Stress tensor.

In order to solve all mean-flow properties of the turbulent flow under consideration, it is required a prescription for computing the Reynolds-Stress tensor. There are different models to compute it. RANS Eddy-viscosity models and RANS Reynolds-Stress models (RSM) are the most commonly used.

The RANS Eddy-viscosity models (Standard $k-\varepsilon$, RNG $k-\varepsilon$ and SST) are two-equation models based on the Boussinesq eddy-viscosity approximation whilst RANS Reynods-Stress Models are based on the transport equations for the individual components of the Reynolds stress tensor. These models are the standard models of practically all CFD codes. However, a complete study of the influence of these models is required in fluid problems with strong buoyancy effects, where the density of the fluid differs from the main stream density [3].

3. CFD model

The analysis was performed by the commercial CFD code Ansys CFX v13 [4]. The Best Practice Guidelines [5] for the use of CFD in Nuclear Reactor Safety applications were followed during the development of the CFD model.

The CFD simulation analyzed was the cold leg A with coolant mixing considering single-phase conditions. Mass flow rates were given as inlet boundary conditions, while the value of pressure was imposed as an outlet boundary condition at the lower part of the downcomer. Fluid recirculation was included as inlet boundary condition on the top of the downcomer. Adiabatic wall boundary conditions were selected for the walls while symmetry conditions were imposed in the right and left sides of the downcomer model. Figure 1 shows the ANSYS-CFX model details. An unstructured tetrahedral mesh with prismatic near wall elements was generated automatically. The mesh is composed of 1108933 elements.

Water properties were extracted from the tables of Ansys library IAPWS-IF97 in the range of 273 K and 823 K at a pressure of 15.5 MPa. The buoyancy forces were considered and the buoyancy reference density was taken as 763.32 kg/m^3, which is an approximate average value of the expected domain density. The initial conditions of the flow in the main pipe assigned were a temperature of 553.5 K and a relative pressure of 15.5 MPa, whilst 296.5 K was assigned in the ECCS line. Test conditions and experimental data were extracted from the Final Data Report of OECD/NEA ROSA Test 1-1 (ECCS Water injection under natural circulation condition).

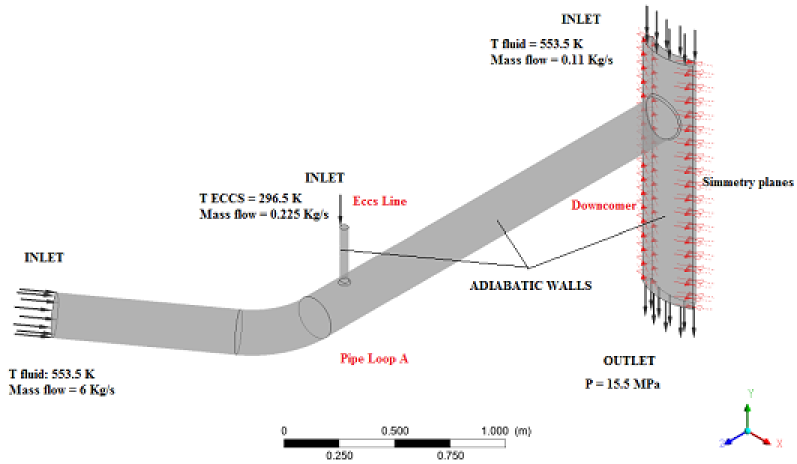

Figure 1: Ansys-CFX model details

4. Discussion and Results

A large number of steady-state calculations were executed. The simulations were performed using parallel local processing HP-MPICH. CFX Solver was run on a PC with two processors Intel Core i-5 2.3 GHz and a RAM memory of 4Gb, under Windows 7 Home Premium. Simulations were performed using the upwind numerical scheme (first order). A root mean square (RMS) residual target value of 10^{-6} was defined as the convergence criteria for the simulation in double precision. Automatic time step was selected. Typical computation time for default grid case was about 3 hours. Three two-equation models (Standard $k-\varepsilon$ model, RNG $k-\varepsilon$ and SST) and two Reynolds Stress Model (SSG and BSL) were selected to study its influence on the temperature stratification range. Scalable wall functions were selected for turbulence models. Moreover the full buoyancy model and the total energy heat transfer model were solved.

Thermal stratification can be observed in all numerical results. The cold water stays at the bottom while the hot water occupies the upper part of the pipe. The cold bottom layer does not mix with the warmer upper water layer. Moreover, comparison between measured and calculated results along a line near the ECCS injection pipe (middle TE-2 plane) indicates that all turbulence models give rather similar results. Those reults are shown in figures 2 and 3.

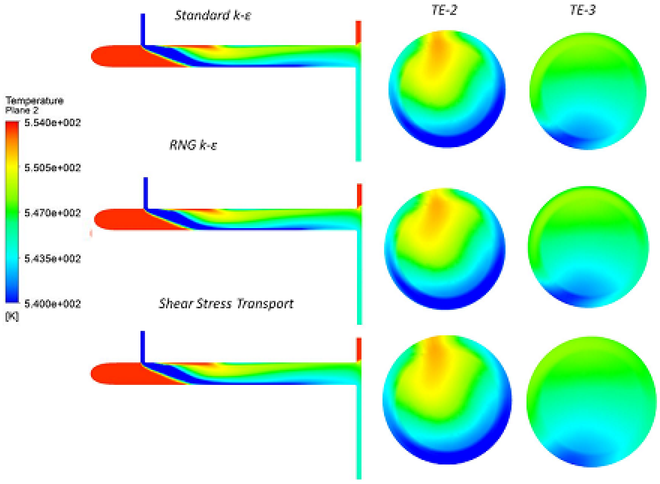

Figure 2: Temperature distributions. RANS Eddy-viscosity models

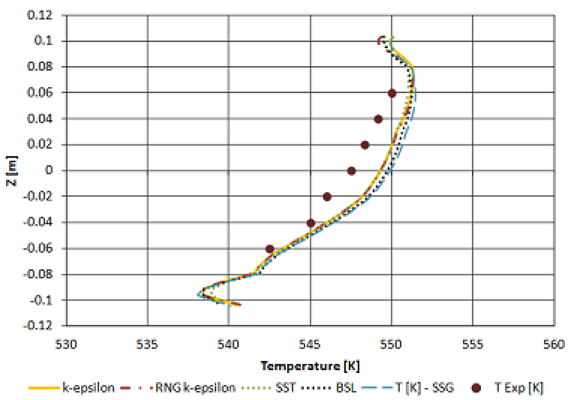

Figure 3: Comparison between different turbulence models. Experimental data and numerical results

5. Conclusions

This paper constitutes a first approach of the assesment of the ANSYS-CFX code to simulate thermal stratification phenomena. The calculations performed with upwind scheme, full buoyancy model and total energy with the five turbulence models capture the stratification process in the cold leg. All the numerical simulations produced similar results and were in good agreement with the experimental data compared. However, it is necessary to define a procedure to evaluate error and uncertainty due to aspects such as mesh refinement, time step and turbulence models.

References

[1] JAEA, *Final data report of OECD/NEA ROSA project test1-1* (2008).

[2] D. Wilcox, *Turbulence Modeling for CFD, Ed. DCW Industries* (1998).

[3] T. Farkas and I. Toth, *Fluent analysis of a ROSA cold leg stratification test, Nuclear Engineerign and Design* **240**, 2169-2175 (2010).

[4] *Ansys CFX Reference Guide. Release v13. ANSYS, Inc.* (2010).

[5] J. Mahaffy et al., *Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications. NEA Report* NEA/CSNI/R(2007)5.