EVALUATION OF TURBULENCE MODELS OF A CFD MODEL FOR NUCLEAR ENGINEERING PURPOSES

M. Martínez[1], T. Barrachina[1], R. Miró[1], S. Chiva[2] and G. Verdú[1]

[1]Institute for Industrial, Radiophysical and Environmental Safety (Isirym), Universitat Politècnica de València, Valencia, Spain
[2]Mechanical Engineering and Construction Department, Universitat Jaume I, Castellón, Spain

momarlia@iqn.upv.es, tbarrachina@iqn.upv.es, rmiro@iqn.upv.es, schiva@emc.uji.es, gverdu@iqn.upv.es

Abstract

Computational Fluid Dynamic (CFD) is the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of computer-based simulation. These codes are employed in engineering and scientific applications where the focus is on the fluid variables (density, pressure, velocity, temperature or energy).

Turbulence consists of random fluctuations of the various flow properties and it is inherently three dimensional and time dependent. An enormous amount of information would be required to describe a turbulent flow. There are several approaches for the turbulence based on the detail to capture the phenomenon; however the computational resources increase with the complexity of the method. Because of the random fluctuations of the flow properties, the solution needs a statistical approach. Commercial CFD codes use Reynolds Averaged Navier Stokes (RANS) models which they solve the average Navier-Stokes equations in a reasonable computational time:

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

where U is the average part of the velocity, i and j are the indexes for the cartesian directions, P is the average part of the pressure, $2\mu S_{ji}$ represents the viscosity term, and $-\rho\overline{u'_j\,u'_i}$ quantity 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 (*k-ε*, *k-ω*, *SST*) and RANS Reynolds-Stress models are the most commonly used. The knowledge of the physics and mathematics of these turbulence models is required to achieve the correct solution of a particular fluid problem, particularly for nuclear engineering purposes.

In this paper, the results of the comparison between different turbulence models in the OECD ROSA test 1-1 project, using the commercial CFD code ANSYS-CFX, are presented. The OECD/NEA ROSA Project Test 1-1was 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. The purpose of this study is the assessment of the code for simulating cases in which the buoyancy forces are relevant. Several commonly used Reynolds Averaged Navier–Stokes models have been selected for the analysis: the Standard *k–ε*, the *RNG k–ε*, the Shear Stress Transport (*SST*) model and Reynolds Stress models (RSM). This paper explains the different turbulence models and their influence on buoyancy and mixing effects in the fluid flow studied.