Comparison of two turbulence models in simulating an axisymmetric jet evolving into a tank

F Zidouni Kendil¹, D-V Danciu², D Lucas², A Bousbia Salah³ and A Mataoui⁴

¹ Nuclear research Center of Birine, Ain-Oussara, Algeria
² Institute of Safety Research, Helmholtz-Zentrum Dresden-Rossendorf, Dresden, Germany
³ Theoretical and Applied Fluid Mechanics Laboratory, Faculty of Physics - USTHB, Algiers, Algeria
⁴ Department of mechanical and Nuclear Engineering University of Pisa-2, Pisa, Italy

E-mail: zidounifaiza@yahoo.fr
E-mail: d.danciu@hzdr.de

Abstract. Experiments and computational fluid dynamics (CFD) simulations have been carried out to investigate a turbulent water jet plunging into a tank filled with the same liquid. To avoid air bubble entrainment which may be caused by surface instabilities, the free falling length of the jet is set to zero. For both impinging region and recirculation zone, measurements are made using Particle Image Velocimetry (PIV). Instantaneous- and time-averaged velocity fields are obtained. Numerical data is obtained on the basis of both $k-\varepsilon$ and SSG (Speziale, Sarkar and Gatski) of Reynolds Stresses Turbulent Model (RSM) in three dimensional frame and compared to experimental results via the axial velocity and turbulent kinetic energy. For axial distances lower than 5cm from the jet impact point, the axial velocity matches well the measurements, using both models. A progressive difference is found near the jet for higher axial distances from the jet impact point. Nevertheless, the turbulence kinetic energy agrees very well with the measurements when applying the SSG-RSM model for the lower part of the tank, whereas it is underestimated in the upper region. Inversely, the $k-\varepsilon$ model shows better results in the upper part of the water tank and underestimates results for the lower part of the water tank. From the overall results, it can be concluded that, for single phase flow, the $k-\varepsilon$ model describes well the average axial velocity, whereas the turbulence kinetic energy is better represented by the SSG-RSM model.

1. Introduction

After numerous developments achieved in the 70’s and 80’s, the science of Computational Fluid Dynamics (CFD) and more specifically the part that deals with the Reynolds-Averaged Navier-Stokes (RANS) equations has touched a development plateau mainly caused by the difficulty of properly modeling turbulence. A general turbulence model that holds good predictions for a wide range of flow fields has yet to be developed. Many researchers explain that such a general model can not be developed due to the inherent shortcomings of the RANS equations. This explains the large amount of work performed in Large-Eddy Simulations (LES) and Direct Numerical Simulations (DNS).
However, from a practical point of view, the use of LES or DNS as design tools is far from reality due to the high computational costs associated with these techniques. Therefore, it is envisioned that RANS equations associated with turbulence modeling will be the main CFD tool used by the industry and part of the research community in the near future. As a result of this trend, there is a need to improve the accuracy and reliability of solutions for turbulent flow fields obtained from RANS equations. In light of this discussion, it seems natural to suppose the existence of an extensive database with the comparative performance of several turbulence models for different flow fields. The present work is intended to add to the literature that deals with the comparative study of turbulence models.

The investigated flow field is a turbulent water jet evolving vertically into a tank filled with water. It is known that a water jet impinging into a quiescent pool of water is often the cause for air entrainment. Bubbles are entrained mostly due to jet surface instabilities. To avoid air entrainment in this study, the free falling length of the jet is set to zero. The flow is single phase flow. Such a configuration is often encountered in many industrial applications. One prominent example where plunging jets can be applied is the emergency core cooling (ECC) which is activated during the loss of coolant accident (LOCA). In this case, a break in one of the reactor inlet pipes is assumed. Water is injected into the pipe and bubbles are entrained and contribute to the mixing between the fluids in the pipe (1). The reactor pressure vessel is subjected to stresses. For the integrity of aged reactor pressure vessels, the mixing characteristics must be assessed. Depending on the size of the leak, its location, and the operating conditions of the considered nuclear power plant, either single-phase or two-phase flow conditions may be encountered. The flow in this case is extremely complicated, so that a similar scenario is simplified to a laboratory-scale water tank to study jet impingement without any heat transfer. The configuration should be adjusted progressively to the real configuration, such as modeling the jet with air entrainment and study the air bubble behavior under the jet. The work presented in this paper reveals the first steps and covers flow characteristics without air entrainment.

2. Instrumentation and methodology

Figure 1 shows a schematic of the experimental setup. Water is pumped out of the tank and reinjected back in through a 16 mm diameter and 500 mm length smooth Plexiglas pipe used as a nozzle. The jet impinges into a 300 mm × 300 mm × 500 mm rectangular Plexiglas tank filled with deionized water. Sequences of the flow over the center vertical plane of the tank are recorded using a CCD camera with a resolution of 1376 × 1040 pixel and a maximum frequency of 5 Hz. To follow the flow correctly, the water is seeded with PMMA Rhodamine B fluorescent particles with a density of 1.016 g/cm$^3$ and diameter of ca. 20 mum. The particles need to be small enough to effectively follow the flow, and large enough to reflect a sufficient amount of light to be detected by the camera. A twin Nd: YAG high-energy (400 mJ) pulsed laser is used as illumination source of the seeded flow. The laser is equipped with sheet optics and has a pulse width of 10 mus. To ensure that only the scattered laser light can reach the CCD of the camera and that the unwanted background light is suppressed, a high-pass optical filter is used with the camera. The camera is operated in the double frame/double exposure mode. Double frame images can be evaluated using cross-correlation. Image analysis is performed using DaVis 7.2 provided by LaVision and the PIVMAT Toolbox for Matlab.

For comparison, the geometry used for the simulation is sketched in Figure 2.

The experimental test matrix consists of four different jet lengths chosen for testing: 0 cm (the nozzle is in contact with the pool surface - no air entrainment occurs, i.e. single phase case), 5 cm, 7.5 cm and 10 cm. Also, for each jet length, six different nozzle exit velocities are investigated, ranging from 1 m/s to 2 m/s. Each test case expands over a period of 40 s measurement time. This paper presents the experimental results for a single phase flow test.
Figure 1. Experimental arrangement for PIV measurements

Case: \( v_0 = 1 \, m/s \) and \( L_j = 0 \, cm \).

Instantaneous as well as time-averaged velocities are determined from the recorded data. The results are used for the analysis of the flow structure and turbulence fluctuations. Considering that images are acquired in a double frame/double exposure mode, cross-correlation is chosen for data analysis. Each image of the recorded set is divided in so-called interrogation windows. The algorithm computes the cross-correlation of all interrogation windows between the first exposed frame and subsequent exposure. This yields one velocity vector for each interrogation window.
3. Simulation approach

The numerical simulations were performed using the commercial CFD package ANSYS-CFX 12.0 to solve the Navier-Stokes equations via a finite volume method. The turbulence \( k - \epsilon \) and SSG-RSM models are used for prediction in order to perform a comparative study for the experimental measurement of axial velocity and turbulence kinetic energy. The flow field for statistically unsteady flow is governed by the following equations in Cartesian tensor notations:

\[
\frac{\partial U_i}{\partial x_i} = 0 \quad (1)
\]

\[
\frac{\partial U_i}{\partial t} + U_i \frac{\partial U_i}{\partial x_i} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial U_i}{\partial x_i} - \overline{u_i u_j} \right) - g \quad (2)
\]

The three terms on the right-hand side of equation 2 represent the pressure gradient, the viscous stress and body force (gravity), respectively.

3.1. Turbulence modeling

3.1.1. Standard \( k - \epsilon \) model

The model adopted for this purpose is based on the standard \( k - \epsilon \) formulation of Launder and Spalding (1974) (5) to simulate the turbulent phenomena in the continuous liquid flow. The governing equations for turbulent kinetic energy \( k \) and turbulent dissipation \( \epsilon \) are:

\[\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_i} \right] + (P_k - \rho \epsilon) \quad (3)\]

\[\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho U_i \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial \epsilon}{\partial x_i} \right] + \frac{\epsilon}{k} (C_{\epsilon 1} P_k - C_{\epsilon 2} \rho \epsilon) \quad (4)\]

The model constants are those of standard \( k - \epsilon \). The turbulent kinetic energy production term, \( P_k \), is deduced through equation 5:

\[P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} \quad (5)\]

3.1.2. SSG turbulence model

These models are based on transport equations for all components of the Reynolds stress tensor and the dissipation rate. They do not use the eddy viscosity hypothesis, but solve directly the transport of Reynolds stresses in the fluid. Algebraic Reynolds stress models solve algebraic equations for the Reynolds stresses, whereas differential Reynolds stress models solve differential transport equations individually for each Reynolds stress component (2). The exact production term and the inherent modeling of stress anisotropies theoretically make Reynolds stress models more suitable for complex flows. However, practice shows that they are often not superior to two-equation models. The Reynolds averaged momentum equations for the mean velocity components are given by:

\[\frac{\partial p U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_i} - \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] = -\frac{\partial}{\partial x_i} \left( P + \frac{2}{3} \mu \frac{\partial U_k}{\partial x_k} \right) - \frac{\partial}{\partial x_i} (\rho \overline{u_i u_j}) + S_M \quad (6)\]

where \( P \) is the static pressure, \( S_M \) is the sum of body forces and the fluctuating Reynolds stress contribution is \( \rho \overline{u_i u_j} \).

In the differential stress model a separate transport equation must be solved for each of the six Reynolds stress components of \( \rho \overline{u_i u_j} \). The standard Reynolds stress model is based on the \( \epsilon \) equation. The CFX solver solves the following equation for transport of the Reynolds stresses:
\[
\frac{\partial (\rho u_i u_j)}{\partial t} + \frac{\partial }{\partial x_k} (\rho U_k u_i u_j) - \frac{\partial }{\partial x_k} \left[ \left( \delta_{kl} + C_{S} \frac{k}{\epsilon} u_k u_l \right) \frac{\partial (u_i u_j)}{\partial x_l} \right] = P_{ij} - \frac{2}{3} \delta_{ij} \rho \epsilon + \Phi_{ij}
\]

where the production term \( P_{ij} \) is given by:

\[
P_{ij} = \rho u_i u_j \frac{\partial U_j}{\partial x_k} - \rho a_k u_k \frac{\partial U_i}{\partial x_k}
\]

and \( \Phi_{ij} \) represents the pressure strain correlation and can be split into a slow term, also known as the \textit{return-to-isotropy} term, and a fast term as shown below:

\[
\Phi_{ij} = \Phi_{ij,1} + \Phi_{ij,2}
\]

The SSG model was developed by Speziale, Sarkar and Gatski (1991) (3) and uses a quadratic relation for the pressure strain correlation. The pressure-strain correlation depends on the anisotropy tensor \( a_{ij} \), the mean strain rate tensor \( S_{ij} \) and vorticity tensor \( \Omega_{ij} \):

\[
\Phi_{ij,1} = -\rho \epsilon \left[ C_{S1} a_{ij} + C_{S2} \left( a_{ij} a_{kj} - \frac{1}{3} a_{mn} a_{mn} \delta_{ij} \right) \right]
\]

\[
\Phi_{ij,2} = -C_{r1} Pa_{ij} + C_{r2} \rho k S_{ij} + C_{r3} \rho k S_{ij} \sqrt{a_{mn} a_{mn}} + C_{r4} \rho k \left( a_{ik} S_{jk} + a_{jk} S_{ik} - \frac{2}{3} a_{kl} S_{kl} \delta_{ij} \right) + C_{r5} \rho k \left( a_{ik} \Omega_{jk} + a_{jk} \Omega_{ik} \right)
\]

and the tensors are given by:

\[
a_{ij} = \frac{\mu u_i u_j}{k} - \frac{2}{3} \delta_{ij}
\]

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\]

\[
\Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right)
\]

The production due to the buoyancy is neglected in this study and the standard wall function approach is chosen (4).

4. CFD model and boundary conditions

A cylindrical tank with a diameter of 30 cm filled with water is considered. The nozzle diameter for the jet injection is 16 mm. The geometry consists of a quarter of the domain in order to reduce computational time. Azimuthal symmetry around the jet axis is considered. The structured mesh has 60 cells for the total height of the domain. For the radius of the water inlet, 16 uniform cells are used and 30 cells for the opening.

The symmetric boundary condition was applied at the lateral boundaries, including the centerline. No slip wall boundary condition was set at the lateral- and bottom-wall of the tank. A logarithmic velocity profile was provided at the inlet boundary and a constant pressure at the exit. The optimal computational grid was found to be \( 46 \times 46 \times 75 \) nodes. The scalable wall functions were applied to specify the turbulence in the near wall region. Boundary conditions needed to be applied for the circular nozzle. The average inlet velocity was \( v_0 = 1 \) m/s, with a chosen maximum value of \( v_0 = 1.2 \) m/s, in accordance with the experimental measured values. The flow was considered to be fully developed at the nozzle exit. For the free surface, free slip wall boundary was used by considering only a single phase flow.
5. Results and discussion

5.1. Axial velocity field

The predicted distribution of the axial velocity in five axial representative positions under the jet impact point is compared to the measured ones in Figure 3. The decay of the velocity on the jet center line can be observed. In the immediate region under the free surface, velocity gradients are high. After the first 5\,cm, the velocity decreases slowly and reaches very small values close to the bottom of the tank. A difference of maximum ±2\,cm/s between the $k-\epsilon$ model and experiments is obtained. Except for the region near to the jet exit ($y < 5\,cm$), the SSG-RSM model shows an overestimation of the axial velocity on the jet center line and also a rapid decrease of the velocity from the jet center line to the wall.

![Figure 3. Radial profile of axial velocities with increasing distance from the jet impact point](image-url)
5.2. Turbulence kinetic energy

The distribution of the turbulence kinetic energy is showed in Figure 4. Similar to the velocity results, the $k-\epsilon$ model matches well the experiments for the region immediate to the jet impact point ($y < 5$ cm), whereas it underestimates it in deeper regions. Also noticeable is a minimum of the turbulence close to the jet exit, which corresponds to the jet core. The SSG-RSM model shows better agreement in the region far from the jet exit. This can be explained by the growing anisotropy of the turbulence in the deeper region, where small vortices appear due to the reversed flow induced by the bottom wall of the tank.

![Figure 4](image-url)
5.3. Formation of vortices and flow pattern

The vorticity contours predicted by the SSG turbulence model are illustrated in Figure 5. This figure indicates the presence of two main vortical structures. The first structure is formed around the jet axis and has a toroidal form corresponding the maximum velocity and minimum kinetic energy, whereas the second one is produced near the bottom-wall where the dynamical values are lower. There is also a stratification of vortex structures near the bottom region. The amplitude of vortical structures gradually diminishes and becomes practically zero at the bottom wall of the tank. The penetration of the outer layer of the jet and the back flow developed by the jet while impinging on the bottom wall produce a separation of these structures, leading thus to an increase in the size of the second vortex.

**Figure 5.** Iso-vorticity surface given by the SSG model ($\Omega_i = 5.78 \text{ s}^{-1}$) and colored after the turbulent eddy dissipation

The vortex showed in Figure 5 can be reproduced by observing the velocity vector field developed near to the bottom of the tank as shown in Figure 6. The presence of the vortex is due to the flow being limited by the bottom wall of the tank. A reversed flow occurs in this
manner and the recirculation vortex forms. The center of the vortex varies with the used model. The $k-\epsilon$ model gives a more accurate prediction of the position of the vortex than the SSG-RSM. It is known that the SSG-RSM is recommended for rotating flows or flows where anisotropic turbulence is generated. Thus, it can be concluded that the anisotropy of the Reynolds stress is not of high importance for this configuration.

6. Concluding remarks

The study performs numerical simulation based on the standard $k-\epsilon$ turbulence model and the SSG-RSM model for predicting the flow behavior of a vertical water jet plunging into a tank filled with the same liquid. Experimental results corresponding to a single-phase flow test case are selected for comparison with the simulations. It was found that the $k-\epsilon$ model offers good predictions of the axial velocity in most regions of the jet flow, whereas the turbulence kinetic energy is underestimated, especially in the deeper region of the tank. The discrepancy is due to neglecting the turbulence anisotropy effect caused by the back flow developed by jet when impinging on the bottom wall. The turbulence kinetic energy obtained with the SSG-RSM shows good agreement to the measurements. A toroidal form is evidenced by calculation of second order turbulence model SSG-RSM.

References

[1] Egorov, Y. & Boucker, M. & Martin, A. & Pigny, S. & Scheuerer, M. & Willemsen S. 2001 Validation of CFD codes with PTS-relevant test cases. 5th Euratom Framework Programme 1998-2002, European Commission, Key action: Nuclear Fission.

[2] Launder, B. E. & Reece, G. J. & Rodi, W. 1975 Progress in the developments of a Reynolds-stress turbulence closure. Journal of Fluid Mechanics, 68, pp. 537–566.

[3] Speziale, C. G. & Sarkar, S. & Gatski, T. B. 1991 Modeling the pressure-strain correlation of turbulence: an invariant dynamical system approach. Journal of Fluid Mechanics, 277, pp. 245–272.

[4] Grotjans, H. & Menter, F. R. 1998 Wall functions for general application of CFD codes. In ECCOMAS 98 Proceedings of the Fourth European Computational Fluid Dynamics Conference, pp. 1112–17, Wiley & Sons.

[5] Launder, B. E. & Spalding, D. B. 1974 The numerical computation of turbulent flows. Computational Methods Appl. Mech. Eng., 3, pp. 269–289.