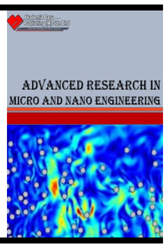
Penerbit Akademia Baru

Journal of Advanced Research in
Micro and Nano Engineering

Journal homepage: https://akademiabaru.com/submit/index.php/armne
ISSN: 2756-8210



Numerical Modelling of Hydraulic Jump Using Mesh-based CFD method and Its Comparison with Lagrangian Moving-Grid Approach

Daniel John Ebrahim Bryant[1], K. C. Ng[1,*]

[1] Department of Mechanical, Materials and Manufacturing Engineering, Faculty of Science and Engineering, University of Nottingham Malaysia, 43500 Semenyih, Selangor, Malaysia

ABSTRACT

Hydraulic jump is a phenomenon in fluid mechanics that has a high research interest due to its energy dissipating behaviour in hydraulics. This study aims to simulate a hydraulic jump using RNG k-ϵ and SST k-ω turbulence models. This study adopts the Eulerian fixed mesh approach in ANSYS FLUENT, where the free surface was modelled using Volume of Fluid (VOF) method to simulate multiphase flow of water and ambient air. Transient analysis is performed using an implicit discretization scheme. Results of open surface water levels and longitudinal velocity profiles are computed and compared with experimental result and those obtained using the Lagrangian approach. Both models show good agreement with experimental data in terms of the free surface water level, with the SST k-ω showing the most similar trend whilst RNG k-ϵ providing a better roller length. The SST k-ω model showed the poorest performance in predicting the mean longitudinal velocity profile as compared to the RNG $k - \epsilon$ model.

Keywords:
Turbulence modelling; Computational Fluid Dynamics; Hydraulic Jump

1. Introduction

Leonardo Da Vinci was the first to notice the phenomenon of hydraulic jumps, whereas Bidone [1] was the first to analyse hydraulic jumps experimentally and analytically [2]. Nowadays, there is an interest in research towards hydraulic jump for kinetic energy dissipation in hydraulic structures such as spillways, chutes and gates [3]. Hydraulic jump is a phenomenon where supercritical flow is transformed into subcritical flow [4]. When inertia forces are dominant, the Froude number, $Fr > 1$ and classifies supercritical flow, whilst when $Fr < 1$, the flow is classified subcritical [5]. Hydraulic jumps typically occur in open channels resulting in a sudden rise of open surface level [5, 6] and large amount of energy dissipation due to turbulent mixing [7]. Due to the intense turbulent nature of hydraulic jumps, even modern research still lacks full understanding of the phenomenon [2].

As compared to physical experimentation, it is more economical of using computational fluid dynamics to investigate energy dissipation and turbulent flow characteristics in hydraulic jump. In early stages of research, Eulerian approach of a fixed mesh was adopted. Chippada *et al.* [7] numerically modelled a hydraulic jump with k-ϵ turbulence model, finding that the recirculation zone plays a dominant role in turbulent energy generation and dissipation. Souders and Hirt [8] considered multiphase flow to model air entrainment in a hydraulic jump, which contributes to hydraulic jump

* *Corresponding author.*
E-mail address: khaiching.ng@nottingham.edu.my

design to reduce cavitation damage. In recent studies, Witt *et al.* [9] uses Volume of Fluid (VOF) model to model the free surface and air entrainment in a hydraulic jump.

Newer studies investigated the Lagrangian approach, first introduced by Gingold and Monaghan [10], and Lucy [11]. Javan and Eghbalzadeh [12], which this paper performs comparisons with, used Lagrangian moving grid approach coupled with the standard k-ϵ turbulent model. Their results have shown good agreement with the experimental data. Lopez and Garrote [13] used smoothed particle hydrodynamics (SPH) to simulate a hydraulic jump, finding that using SPH with the sophisticated k-ϵ turbulence model increased accuracy at the expense of more computational time. Chern and Syamsuri [14] used SPH method to simulate a hydraulic jump on a corrugated bed, showing that a corrugated bed can dissipate more energy than a flat bed.

This study adopts the Eulerian mesh-based method in ANSYS FLUENT with RNG k-ϵ and SST k-ω turbulence models. The Volume of Fluid (VOF) approach is used to track the air-water interface. The suitability of both RNG k-ϵ and SST k-ω turbulence models are then determined by comparing the numerical results against the published experimental and numerical data.

2. Methodology

2.1 Geometry

Figure 1 presents the schematic of the domain geometry and boundary conditions. The domain

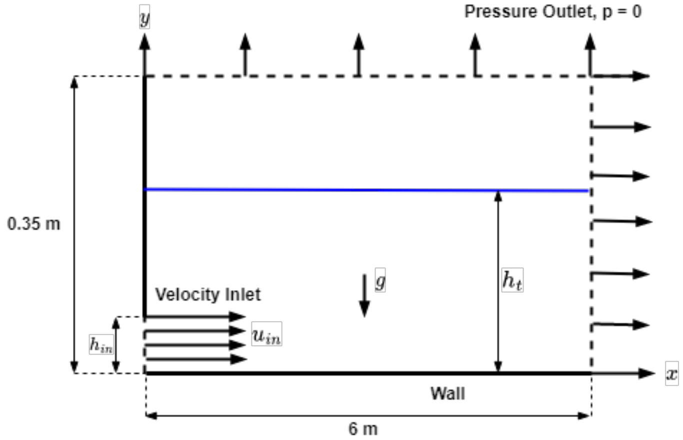


Fig. 1. Schematic of domain with dimensions and boundary conditions

has an overall height of 0.35 m and even though the region of interest in this study is at $0 < x < 2m$, the domain length is set to 6m to prevent reflection of water from the outlet as reported by Javan and Eghbalzadeh [12]. The height of the water is initially set to h_t = 0.206 m and the height of the velocity inlet h_{in} = 0.015 m. Gravitational acceleration, g = 9.81 m/s^2 which its inclusion is important for open channel flow.

2.2 Numerical setting

At the inlet, velocity, turbulent intensity and turbulent dissipation rate were set to 3.14 m/s, 0.0014 m^2/s^2 and 2.2x10^{-5} m^2/s^3, respectively. The domain has a pre-set water level of 0.206 m before the simulation starts, which would reduce computational time of filling up the channel. The pressure outlet facing the y-direction could be considered as the 'environment' to simulate ambient air conditions. The wall above the inlet acts as a sluice gate between the investigated region and another body of water on the left side of the grid which is not considered in the current analysis.

The mesh is split into two sections: $0 < x < 2$ m and $2 < x < 6$ m. This is to create a finer grid at the area of interest ($0 < x < 2$ m) and use a coarser grid at the rest of the region ($2 < x < 6$ m). Both grids are mapped, and the finer grid has a consistent element size of 0.0075 m.

Two different turbulence models are used and compared. This study uses RNG k-ϵ and SST k-ω turbulence models. For time discretization, this study adopts a first-order implicit discretization scheme. As mentioned by Bayon-Barrachina and Lopez-Jimenez [15], despite its longer computational time, implicit time discretization scheme provides more stability. A time step size of 0.05s was used and the simulation was executed up to 300 seconds.

3. Results

This section compares the predicted velocity and water surface level profiles (using ANSYS FLUENT) against those published numerical results of Javan and Eghbalzadeh [12] using the Lagrangian moving grid technique coupled with the standard k-ϵ turbulence model and the experimental results of Long *et al.* [16].

3.1 Water Fraction Profiles

The water volume fraction profiles from $0 < x < 2$ m and $0 < y < 0.35$ m for RNG k-ϵ and SST k-ω are presented in Figure 2 and Figure 3 respectively. It can be seen that both models give very similar results, where RNG k-ϵ predicts more air entrainment in the turbulent region. SST k-ω predicts a relatively smooth ascent of water level whilst the RNG k-ϵ prediction is slightly more ragged.

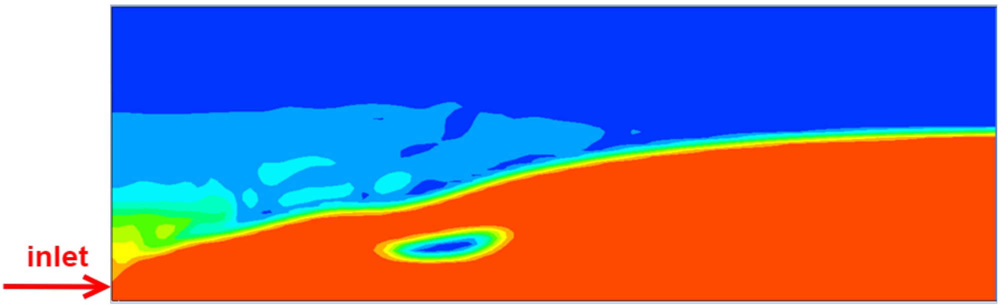


Fig. 2. Water volume fraction profile using RNG k-ϵ

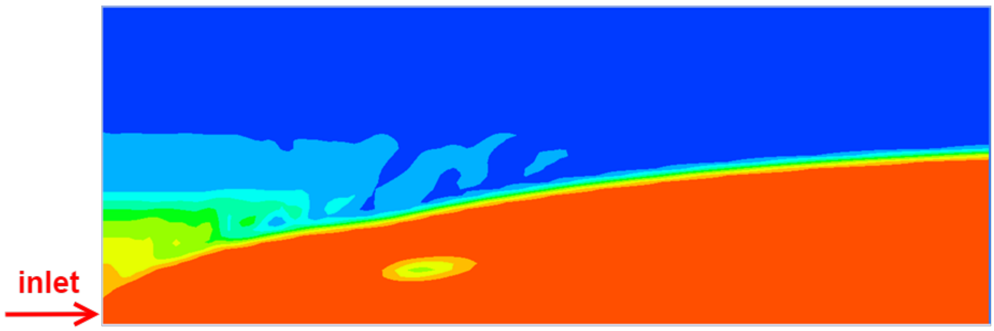


Fig. 3. Water volume fraction profile using SST k-ω

3.2 Water Open Surface Profile

To capture the location of the water surface profile, an iso-surface is created throughout the domain with a volume fraction of water restricted to 0.51. Here, any computational cell with a water volume fraction of 0.51 is displayed, creating a contour of the water surface. Figure 4 compares the free surface levels predicted using different methods. It can be seen that the SST k-ω turbulence model best fits the experimental data. RNG k-ϵ result shows a steeper ascent within the jump and the Lagrangian moving-grid technique with the standard k-ϵ model [12] overestimates the overall water level but shows a very similar gradient of ascent against the experimental result. Nevertheless, the standard k-ϵ Lagrangian method best captures the free surface close to the gate wall at $0 < x/h_{inlet} < 10$.

RNG k-ϵ best captures the roller length, which is the length at which the free surface rises, as the RNG k-epsilon plot reaches the maximum free surface height at the same point the experimental curve reaches. This is in line with Bayon-Barrachina and Lopez-Jimenez [15], who found the least error of roller length using RNG k-ϵ.

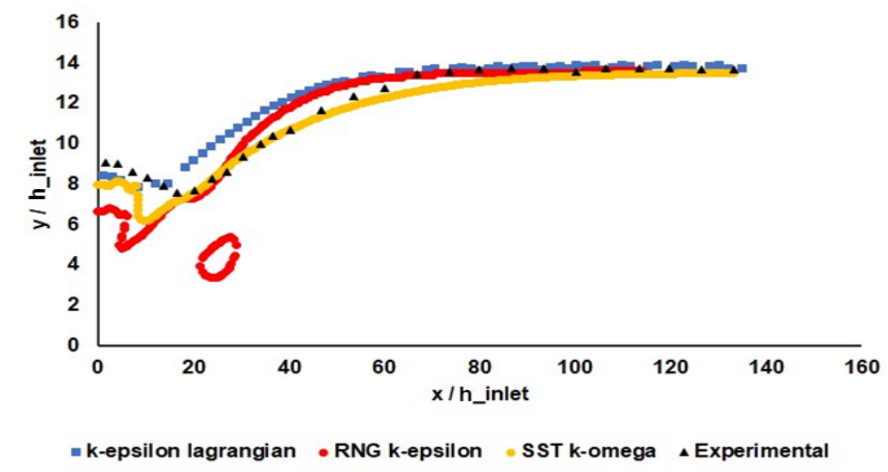


Fig. 4. Non-dimensionalized free surface water levels for different turbulence models and experimental data

3.3 Longitudinal Velocity Profile

Figure 5 presents the non-dimensionalized maximum mean velocity profile along the domain length. It can be seen that all 3 turbulence models show good agreement with the experimental data of Long *et al.* [16]. The hump predicted using the standard k-ϵ model is due to a boundary condition used at the inlet by Javan and Eghbalzadeh [12] where there is a downward vertical velocity, thereby contributing to a higher mean velocity. The RNG k-ϵ results show good agreement with the experimental data, and the results collapse with those of the Lagrangian approach with standard k-ϵ model from Javan and Eghbalzadeh [12] further downstream. At downstream region, all 3 turbulence models overpredict the mean velocity. The SST k-ω model shows the poorest performance in general as witnessed from Figure 5. This observation is consistent with that of Bayon-

Barrachina and Lopez-Jimenez [15] who stated that RNG k-ϵ model has an overall better performance than SST k-ω model when simulating hydraulic jump.

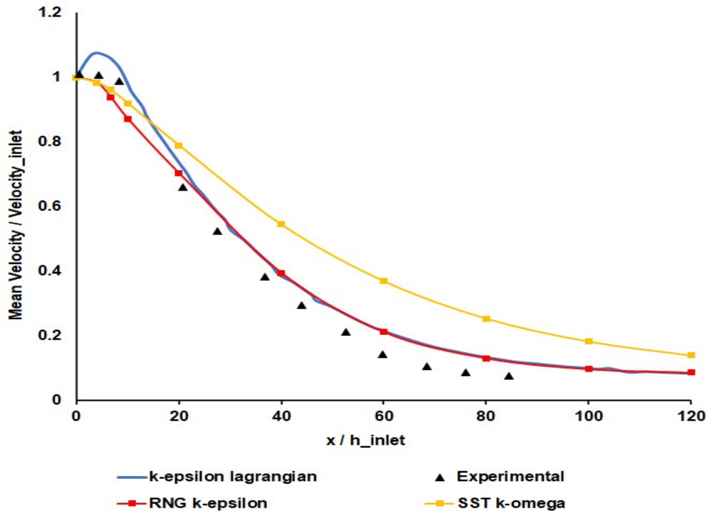


Fig. 5. Non-dimensionalized maximum mean velocity profile along the domain length for different turbulence models and experimental data

4. Conclusion

All 3 turbulence models accurately simulate the water surface level, with the standard k-ϵ producing the best results near the gate wall, SST k-ω showing good accuracy in the water level rise and RNG k-ϵ giving the most accurate result in roller length estimation.

The SST k-ω model exhibits the poorest performance in estimating the mean longitudinal velocity profile, whilst the RNG k–ϵ result compares quite well with the experimental data and it is almost identical to the numerical result generated using the Lagrangian moving-grid approach that uses the standard $k - \epsilon$ model.

Further studies could involve more turbulence models in the search of a more accurate turbulence model for simulating hydraulic jump.

Acknowledgement

This research was funded by Ministry of Higher Education (MoHE) Malaysia under the Fundamental Research Grant Scheme (FRGS): FRGS/1/2021/TK0/UNIM/02/6.

References

[1] G Bidone. 1823. *Experiences Hydrauliques Sur Le Remous, et Sur La Propagation Des Ondes.* Paris.

[2] De Padova, Diana, and Michele Mossa. 2021. “Hydraulic Jump: A Brief History and Research Challenges.” *Water* 13 (13): 1733. https://doi.org/10.3390/w13131733.

[3] Abbaspour, A., and D. Farsadizadeh. "Numerical study of hydraulic jumps on corrugated beds using turbulence models." *Turkish Journal of Engineering and Environmental Sciences* 33, no. 1 (2009): 61-72. https://doi:10.3906/muh-0901-7

[4] Mirzaei, Hamid, and Hossein Tootoonchi. "Experimental and numerical modeling of the simultaneous effect of sluice gate and bump on hydraulic jump." *Modeling Earth Systems and Environment* 6, no. 4 (2020): 1991-2002.

https://doi.org/10.1007/s40808-020-00835-5
[5] Retsinis, Eugene, and Panayiotis Papanicolaou. "Numerical and experimental study of classical hydraulic jump." *Water* 12, no. 6 (2020): 1766. https://doi.org/10.3390/w12061766
[6] Mortazavi, Milad, Vincent Le Chenadec, Parviz Moin, and Ali Mani. "Direct numerical simulation of a turbulent hydraulic jump: turbulence statistics and air entrainment." *Journal of Fluid Mechanics* 797 (2016): 60-94. https://doi:10.1017/jfm.2016.230
[7] Chippada, S., B. Ramaswamy, and M. F. Wheeler. "Numerical simulation of hydraulic jump." *International Journal for Numerical Methods in Engineering* 37, no. 8 (1994): 1381-1397. https://doi.org/10.1002/nme.1620370807
[8] Souders, David T., and C. W. Hirt. "Modeling entrainment of air at turbulent free surfaces." In *Critical Transitions in Water and Environmental resources Management*, pp. 1-10. 2004. https://doi.org/10.1061/40737(2004)187
[9] Witt, Adam, John Gulliver, and Lian Shen. "Simulating air entrainment and vortex dynamics in a hydraulic jump." *International Journal of Multiphase Flow* 72 (2015): 165-180. https://doi.org/10.1016/j.ijmultiphaseflow.2015.02.012
[10] Gingold, Robert A., and Joseph J. Monaghan. "Smoothed particle hydrodynamics: theory and application to non-spherical stars." *Monthly notices of the royal astronomical society* 181, no. 3 (1977): 375-389. https://doi.org/10.1093/mnras/181.3.375
[11] Lucy, L. B. "A numerical approach to the testing of fusion processes [J]." *Astrophys J* 82 (1977): 1013-1024. https://doi.org/10.1086/112164
[12] Javan, M., and A. Eghbalzadeh. "2D numerical simulation of submerged hydraulic jumps." *Applied Mathematical Modelling* 37, no. 10-11 (2013): 6661-6669. https://doi.org/10.1016/j.apm.2012.12.016
[13] López, David, Roberto Marivela, and Luis Garrote. "Smoothed particle hydrodynamics model applied to hydraulic structures: a hydraulic jump test case." *Journal of Hydraulic Research* 48, no. sup1 (2010): 142-158. https://doi.org/10.1080/00221686.2010.9641255
[14] Chern, Ming-Jyh, and Sam Syamsuri. "Effect of corrugated bed on hydraulic jump characteristic using SPH method." *Journal of Hydraulic Engineering* 139, no. 2 (2013): 221-232. http://dx.doi.org/10.1061/(ASCE)HY.1943-7900.0000618
[15] Bayon-Barrachina, Arnau, and Petra Amparo Lopez-Jimenez. "Numerical analysis of hydraulic jumps using OpenFOAM." *Journal of Hydroinformatics* 17, no. 4 (2015): 662-678. https://doi.org/10.2166/hydro.2015.041
[16] Long, D., P. M. Steffler, and N1 Rajaratnam. "LDA study of flow structure in submerged hydraulic jump." *Journal of Hydraulic Research* 28, no. 4 (1990): 437-460. https://doi.org/10.1080/00221689009499059