

COMPARATIVE OF RANS AND LES SIMULATIONS IN FLOW PATTERN MODELING: LITERATURE REVIEW

PERBANDINGAN SIMULASI RANS DAN LES DALAM PEMODELAN POLA ALIRAN: KAJIAN LITERATUR

ELSY THIANSY*, TEDY A. CAHYADI, and SHOFA R. HAQ
Master of Mining Engineering Program, UPN "Veteran" Yogyakarta,
Jalan Padjajaran, Sleman, Yogyakarta, Indonesia.
*Corresponding author's e-mail: elsythiansy@gmail.com

ABSTRACT

The fidelity of turbulent flow modelling is critical aspect in engineering design. This study conducts comparative assessment of Reynolds-Averaged Navier–Stokes (RANS) and Large Eddy Simulation (LES) models in Computational Fluid Dynamics (CFD) through a literature review. The findings show that RANS is efficient for large-scale simulations with low computational cost but less accurate in capturing turbulence details. LES, on the other hand, offers higher accuracy in representing turbulent structures but requires much greater computational resources. This review concludes that the determination of an optimal modeling framework necessitates a balanced consideration of computational resource constraints, simulation domain scale, and the required level of predictive accuracy.

Keywords: CFD, flow characteristics, turbulence, RANS, LES.

ABSTRAK

Pemodelan aliran turbulen penting dalam desain rekayasa. Penelitian ini bertujuan untuk membandingkan model Reynolds-Averaged Navier–Stokes (RANS) dan Large Eddy Simulation (LES) dalam Dinamika Fluida Komputasi (CFD) melalui tinjauan pustaka. Temuan menunjukkan bahwa RANS efisien untuk simulasi skala besar dengan biaya komputasi rendah, tetapi kurang akurat dalam menangkap detail turbulensi. Di sisi lain, LES menawarkan akurasi yang lebih tinggi dalam merepresentasikan struktur turbulen, tetapi membutuhkan sumber daya komputasi yang jauh lebih besar. Tinjauan ini menyimpulkan bahwa penentuan kerangka pemodelan yang optimal memerlukan pertimbangan yang seimbang antara keterbatasan sumber daya komputasi, skala domain simulasi, serta tingkat akurasi prediktif yang dibutuhkan.

Kata kunci: CFD, karakteristik aliran, turbulensi, RANS, LES.

INTRODUCTION

Modeling fluid flow patterns is a crucial aspect in various engineering fields, such as civil engineering, mechanical, environmental engineering, and industrial processes, utilizing advances in numerical simulation technology, particularly CFD (Ahadi, Bergstrom and Mazurek, 2020). Several previous studies have used two approaches in CFD simulation to handle turbulent flows: RANS and LES as presented in Table 1. RANS and LES are the two predominant

approaches used in CFD for turbulence modelling (Yan *et al.*, 2018; Rodríguez Berrio *et al.*, 2025). There are several other simulations in CFD simulation, such as Direct Numerical Simulation (DNS), Unsteady RANS (URANS) and Hybrid Models such as Detached Eddy Simulation (DES). However, these simulations are computationally prohibitive and some other simulation methods are actually derivatives or variations of RANS or LES (Chaouat, 2017; Aultman *et al.*, 2022; Fridayana *et al.*, 2022).

CFD analysis is useful for evaluating the flow and deposition of sediment in stormwater ponds (Ahadi, Bergstrom and Mazurek, 2020). CFD simulations accurately predict main characteristics such as velocity profiles and energy dissipation with a high degree of accuracy based on experimental data (Bayon *et al.*, 2016; Masrur, 2016; Ahadi, Bergstrom and Mazurek, 2020; Desti, Putra and Kushadiwijayanto, 2021; Nurhaliza, Putra

and Kushadiwijayanto, 2023). These results strengthen the role of CFD as a credible tool in hydraulic analysis, while providing opportunities for further development in aspects such as aeration, microscopic turbulence and the influence of channel geometry using the CFD method in order to understand the behavior of flow velocity and pressure.

Table 1. Previous research

Researcher	Year	Title	Research Result
Iberl <i>et al.</i> (2025)	2025	A-posteriori assessment of mixed models for Large Eddy Simulation of polydisperse multiphase flows	This study demonstrates that LES is capable of effectively predicting two-phase turbulent flow, but the choice of SGS model significantly impacts accuracy. Mixed SGS models are shown to be superior to standalone models in capturing multiphase turbulence interactions (such as bubble breakup, shear layer, and ejection-sweep phenomena).
Nurhaliza, Putra and Kushadiwijayanto (2023)	2023	<i>Studi numerik pola aliran di sekitar pintu air menggunakan pendekatan komputasi dinamika fluida</i>	The simulation results show a linear relationship between the average flow velocity and the Froude number, with a high coefficient of determination (R2) (≥ 0.95), indicating a strong correlation. The flow category identified from the simulation results is supercritical, which has the potential to cause antidunes as the basic form of flow. The use of the LES equation in this research provides more accurate results in capturing turbulent flow patterns around the sluice gate
Kadivar, Tormey and McGranaghan (2023)	2023	A comparison of RANS models used for CFD prediction of turbulent flow and heat transfer in rough and smooth channels	This study shows that the RANS model is able to predict turbulent flow in smooth and rough channels with good accuracy, especially if the roughness geometry is modeled explicitly (roughness-resolving), with Realisable k- ϵ + EWT showing the best overall performance.
Alvindriyove, Rozi and Fajar (2021)	2021	<i>Studi numerik pengaruh ketinggian step terhadap karakteristik aliran melalui backward-facing step</i>	This study shows that the RANS model with SST k- ω in this study is effective in predicting macroscopic turbulent flow characteristics (separation, recirculation, reattachment, pressure distribution, Cf, TKE) in BFS flow.
Desti, Putra and Kushadiwijayanto (2021)	2021	<i>Simulasi lompatan hidrolis pada aliran kanal terbuka menggunakan pendekatan komputasi dinamika fluida</i>	This study shows that LES is able to model the increase in velocity near the channel bottom, the decrease in the water column, and the increase again in the water surface that occurs due to turbulent effects.
Ahadi, Bergstrom and Mazurek (2020)	2020	Computational fluid-dynamics modeling of the flow and sediment transport in stormwater retention ponds: A review	This study shows that the RANS (Reynolds-Averaged Navier–Stokes) approach is widely and centrally used in various CFD studies discussed to simulate turbulent flow in retention ponds.
Ahadi, Bergstrom and Mazurek (2019)	2019	Application of the two-fluid model to prediction of sediment transport in turbulent open channel flow	This study uses the RANS (Reynolds-Averaged Navier–Stokes) model to simulate the average turbulent flow of water in an open channel. RANS successfully captures the main flow structure and near-wall effects, which are important for sedimentation models.
Wang, Wu and Zhu (2019)	2019	Numerical simulation of flow separation over a	The LES model in this study is capable of predicting BFS turbulent flow phenomena quite well, particularly in capturing large-scale vortices

Researcher	Year	Title	Research Result
		backward-facing step with high Reynolds number	and shear layer dynamics. However, precision in the near-wall boundary layer is limited by grid resolution, leading to overprediction of the top-wall separation and Reynolds number stress.
Cheng, Hsu and Chauchat (2018)	2018	An Eulerian two-phase model for steady sheet flow using large-eddy simulation methodology	This research shows that LES is able to capture the turbulent eddy structure that influences sediment movement, including the ejection (particles lifted upwards) and sweep (particles pressed downwards) phenomena.
Sukhodolov <i>et al.</i> (2017)	2017	Turbulent flow structure at a discordant river confluence: Asymmetric jet dynamics with implications for channel morphology	The RANS model effectively predicts flow and turbulence patterns in river confluences, with results consistent with field data, although it is less accurate in capturing small-scale turbulence dynamics.
Bayon <i>et al.</i> (2016)	2016	Performance assessment of OpenFOAM and FLOW-3D in the numerical modeling of a low Reynolds number hydraulic jump. Environmental modelling and software	In this study, RANS models turbulence with an additional empirical model, namely the RNG k- ϵ model. However, the additional empirical model RNG k- ϵ is able to predict the mean flow and global parameters quite accurately, but has limitations in capturing the details of the local turbulence structure in the swirling roller area, especially stress anisotropy, vortex interaction, and complex pressure distribution
Masoudian <i>et al.</i> (2016)	2016	A RANS model for heat transfer reduction in viscoelastic turbulent flow	This study shows that in smooth channels, the RANS model is able to predict velocity and temperature profiles with an error of <5%. However, in rough channels, RANS with a roughness-resolving approach is quite accurate in predicting the effects of roughness on drag (Cf) and heat transfer (Nu) if using an appropriate model such as Realizable k- ϵ with Enhanced Wall Treatment (EWT).
Li <i>et al.</i> (2016)	2016	Effects of parallel processing on Large Eddy Simulations in ansys fluent	This study simulates turbulent flow in a rectangular duct using LES. LES preserves the time fluctuations (transients) in the flow, which is very important for the study of complex flow dynamics such as in the case of small-dimensional narrow channels in this study.

However, until now there is still very limited research directly discussing and comparing LES and RANS simulation models in modeling flow patterns, especially in controlling and optimizing fluid turbulence conditions. In fact, both LES and RANS have different characteristics in capturing turbulence phenomena (Lopes *et al.*, 2024; Quaresma, Romão and Pinheiro, 2025). LES is more detailed in capturing large turbulence structures but is computationally expensive, while RANS is more efficient because the computation time is short but the turbulent structure is modeled only statistically (Blocken, 2018; Ibarra-Udaeta *et al.*, 2020; Fazlizan, Muzammil and Al-Khawlani, 2025). In the RANS turbulence solution, it is not simulated explicitly, where all turbulent effects are modeled using a time-averaged approach (Chipongo, Khiadani and Sookhak Lari, 2020;

Ballesteros Martínez and Gaukel, 2023; Mishra *et al.*, 2024; Liu *et al.*, 2025). While LES is simulated directly (explicit resolution), where only small scales (subgrids) are modeled (Marefat, Alam and Pope, 2024). This is what makes LES more sensitive to geometric shapes than RANS which only provides an average picture (Wan Ismail *et al.*, 2024; Samuels *et al.*, 2025; Tang *et al.*, 2025). These differences have the potential to produce different flow predictions, especially in modeling flow structures behind obstacles (eg steps) that play an important role in the deposition process (Vita *et al.*, 2020; He *et al.*, 2021; Knopp *et al.*, 2021). Therefore, this comparative study of two models is essential for designing effective sediment control systems, offering both academic insights and practical applications for sedimentation pond design.

The collected literature is then explained to understand the methodology and results on the topic studied. This research is expected to be able to determine accurate model equations for use in CFD simulations.

METHOD

The data used in this research comes from a number of research findings that have been carried out and published in online journals, both national and international. Strategic in collecting journals of various literature through the use of accredited journal sites, for example Scholar, Research Gate, and ScienceDirect. Where this research uses 6 international journals and 6 national journals.

Each selected journal is identified in the form of a summary table containing author, year, research plan, sample, instruments and findings to facilitate grouping and discovery of research concepts shown in Figure 1.

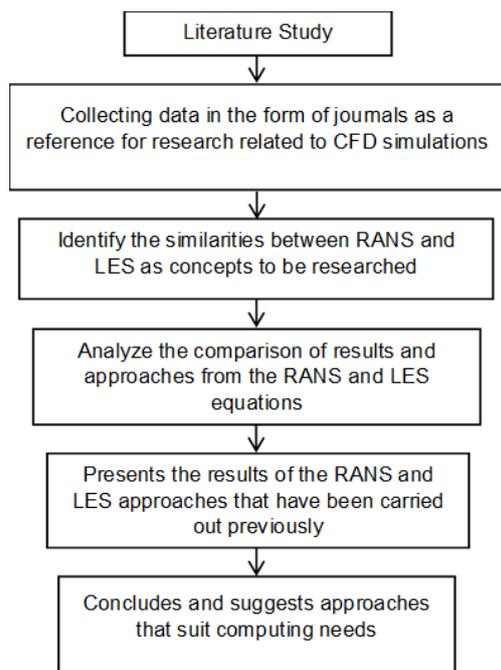


Figure 1. Research Flow Chart

The analytical method in this research uses comparative analysis. Where to compare the results and approaches of several existing equations to find similarities, differences and patterns of relationships between existing studies.

RESULTS AND DISCUSSION

RANS

RANS models are capable of accurately simulating average velocity patterns and lateral momentum distributions in accordance with semi-empirical free jet theory (Desti, Putra and Kushadiwijayanto, 2021). Simulations using the RANS model show consistent results with field measurement data, although there are some differences in the turbulence scale and secondary circulation patterns. This RANS model is effective in predicting flow patterns and turbulence interactions in river confluences, although there are several limitations in capturing complex small-scale dynamics (Sukhodolov *et al.*, 2017). RANS can be used effectively to model open channel flow conditions that have solid bottom walls requiring good resolution of the turbulent field near the walls (Ahadi, Bergstrom and Mazurek, 2019). Research conducted by Masrur (2016) show that the flow characteristics behind the step are influenced by channel geometry and initial disturbances in the upstream flow. in detail, as well as evaluating the effect of geometry modifications on the stability and efficiency of flow in the Backward-Facing Step (BFS) channel.

Research conducted by Alvindriyove, Rozi and Fajar (2021) carried out a simulation at the grid creation stage, the number of cells used in this simulation was 474439, the minimum orthogonal quality value was 0.71 and the maximum skewness value was 0.49. The mesh discretization scheme at high mesh density around the wall is as shown in Figure 2.

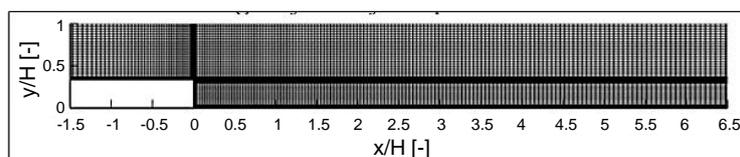


Figure 2. Mesh on forward-facing step (Alvindriyove, Rozi and Fajar, 2021)

From the grid simulation results, simulations were carried out with step height variance $h/H = 0.33, 0.47$ and 0.60 . The plots are arranged from low to high step heights. Where the results of the simulation produce the flow structure and velocity distribution which can be seen in Figure 3. The color gradations visible in the plot represent small and large velocities distributed throughout the BFS section. On the other hand, the chain of arrows that extends from the inlet to the outlet symbolizes the flow structure. The flow structure shows the flow separation point at the end of the step, the recirculation zone at the front of the step, and the reattachment area in the downstream. The increase in recirculation occurs at the same time as the step increases, with the largest recirculation occurring during the step $h/H = 0.60$ and is also followed by a reattachment length. Below the step, lower flow recirculation appears, and the opposite rotation is the main recirculation or primary vortex, known as the secondary vortex. The lowest velocity recirculation zone is marked with a blue gradient.

The pressure distribution in the simulation through the implementation of $Re = 1.3 \times 10^4$ and various step heights $h/H = 0.33, 0.47,$ and 0.60 is shown in Figure 4. The pressure magnitude is represented by the color gradation at the bottom of the plot, starting from blue to red. Upstream appears to have a small pressure area which is symbolized by the blue contour. On the other hand, downstream has a pressure area that increases in the x-axis direction.

It is characterized by a color gradient from blue on the steps to red in the outlet area. The area of low pressure downstream, where the pressure is smaller than upstream, is the flow recirculation area. In other words, the recirculation area, apart from the low velocity, also has low pressure. If viewed based on pressure comparisons at each step $h/H = 0.33, 0.47,$ and 0.60 , changes in pressure distribution occur with each increase in step height. In step $h/H=0.33$ the pressure is highest downstream and decreases continuously as the height of the step increases. Areas that

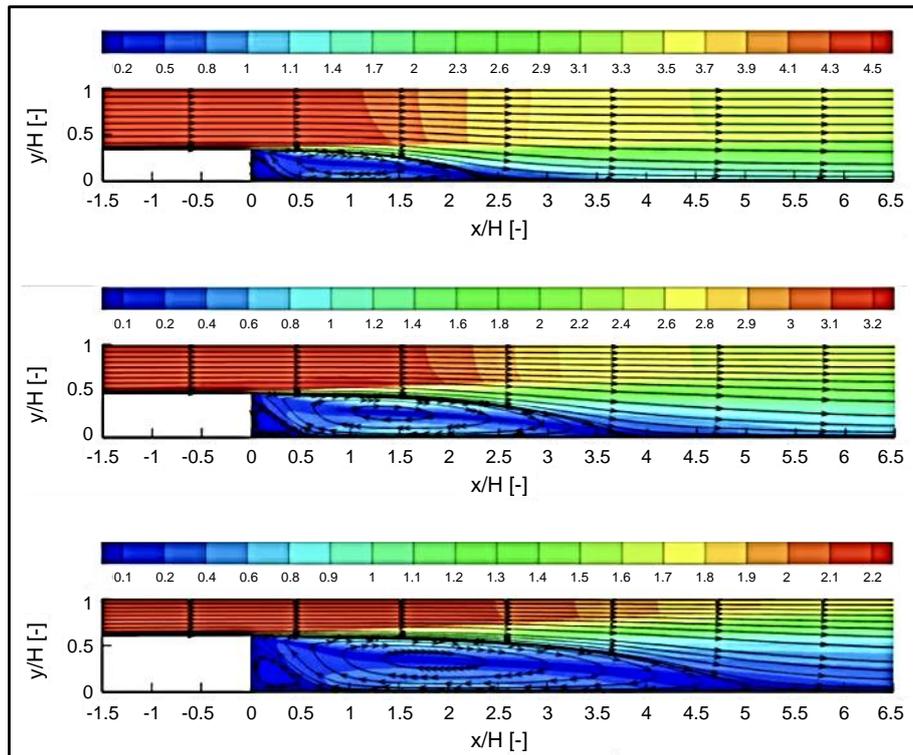


Figure 3. Flow structure and velocity distribution at $Re = 1.5 \times 10^4$ (Alvindriyove, Rozi and Fajar, 2021)

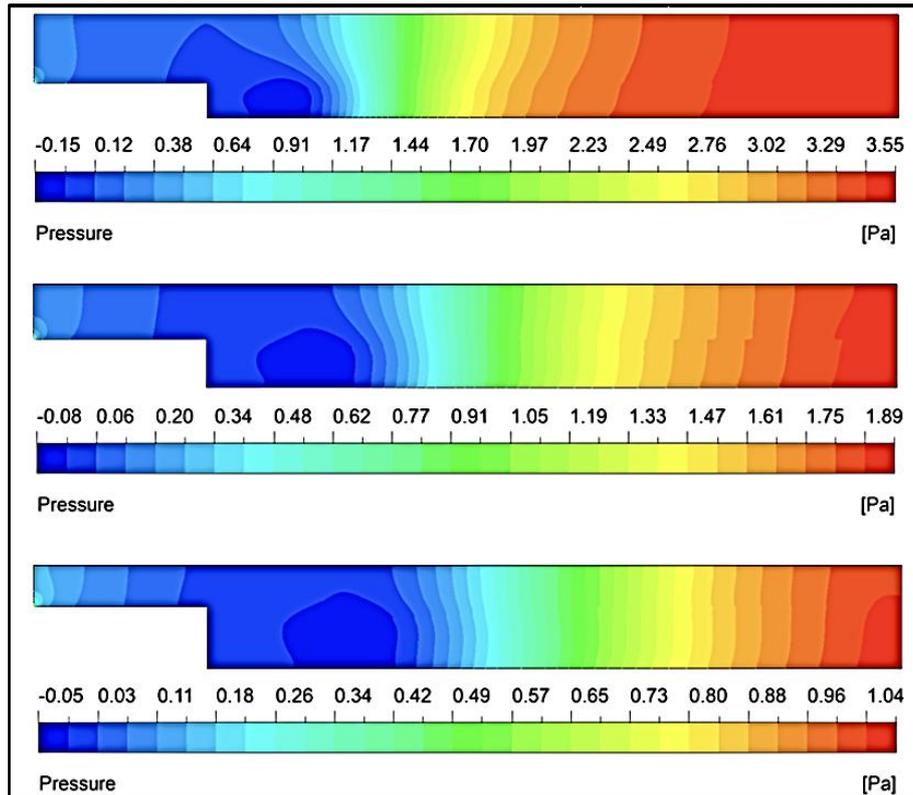


Figure 4. Pressure distribution at $Re = 1.5 \times 10^4$ (Alvindriyove, Rozi and Fajar, 2021)

have pressure fluctuations where the pressure changes rapidly in the area around the reattachment point.

The distribution of turbulent kinetic energy at $Re = 1.5 \times 10^4$ at various step heights $h/H = 0.33, 0.47$ and 0.60 is shown in Figure 5. The gradient is represented from blue to red as shown below in the plot. Based on the findings, the magnitude of the turbulent kinetic energy decreases as the size of the step increases. It appears that the highest turbulent kinetic energy is in the flow simulation, using the step $h/H = 0.33$ and the smallest is at the highest step, $h/H = 0.60$.

LES

The LES model can capture the phenomena of shear layer instability and drag crisis well. Simulations show that the boundary layer transition is influenced by vortices in the shear layer, which play a role in changes in drag and flow separation.

The LES model is able to capture flow patterns and vortex formation in very detail,

especially in recirculation areas behind buildings or flow barriers This equation is able to capture water phenomena that can change at any time (unsteady) (Nurhaliza, Putra and Kushadiwijayanto, 2023). The use of the LES equation can provide more accurate results in capturing turbulent flow patterns. Based on the simulation results using the LES approach, it is effective in modeling turbulent flow in pipe elbows because it is able to capture physical phenomena such as non-uniform velocity distributions, vortices and shear stress fluctuations (Desti, Putra and Kushadiwijayanto, 2021). In a study conducted by Li *et al.* (2016), the use of the LES equation successfully produced a stable and realistic velocity profile in channel flow, depending on the choice of subgrid model and mesh size.

In contrast to the RANS equation, in research conducted by Desti, Putra and Kushadiwijayanto (2021), the parameter testing process with the LES equation was carried out in a simulation based on the variance of grid sizes in types I, II, III, IV, V from the smallest grid size to the large grid size which can be seen in Figure 6.

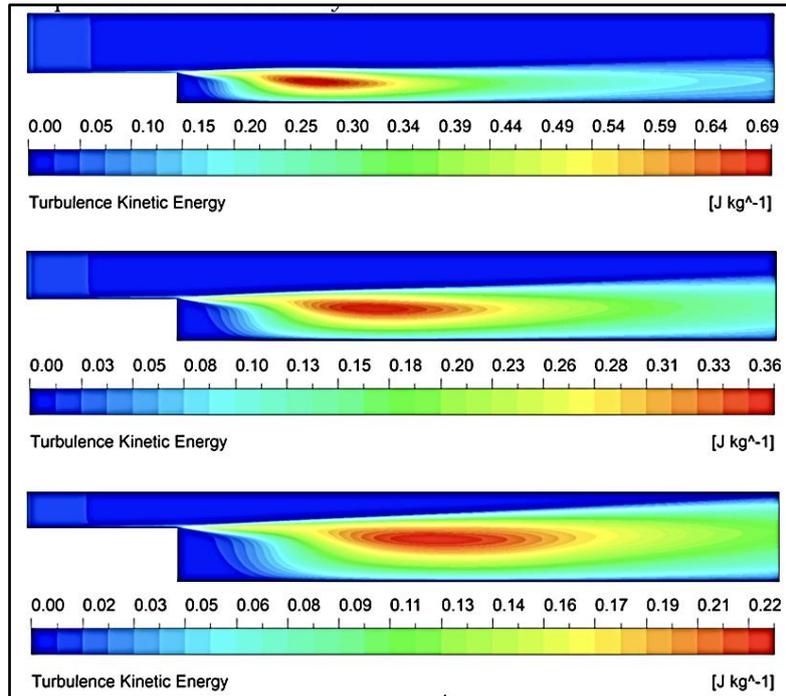


Figure 5. Distribution of turbulent kinetic energy at $Re = 1.5 \times 10^4$ (Alvindriyove, Rozi and Fajar, 2021)

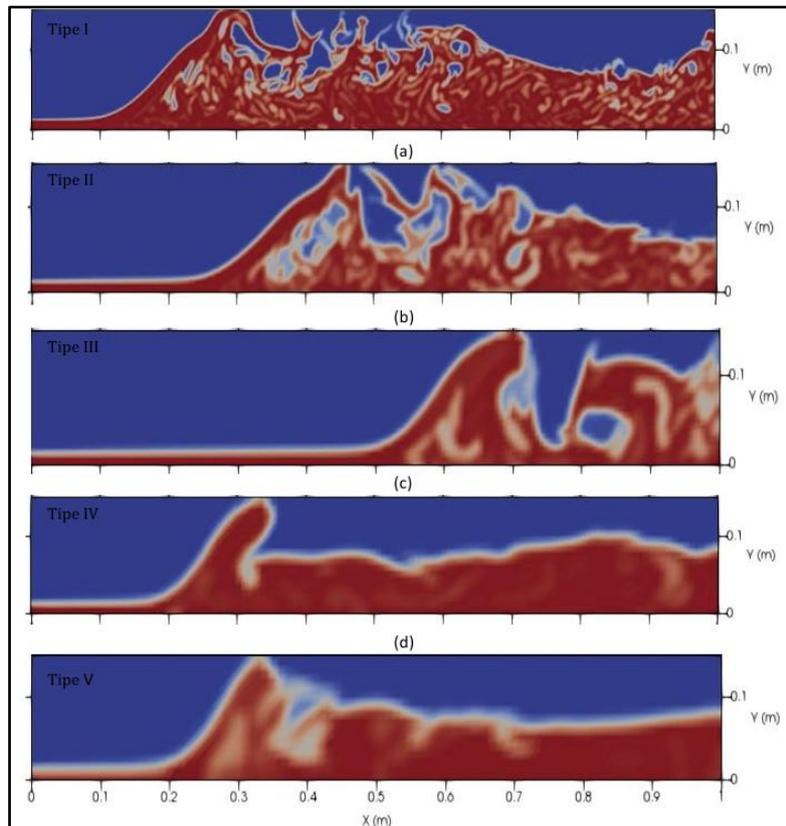


Figure 6. Free surface shape at grid sizes Types (a) I, (b) II, (c) III, (d) IV, and (e) V (Desti, Putra and Kushadiwijayanto, 2021)

The results of the analysis from the simulation carried out show that Type I is closer to the reference data. This is because the grid in type I is lower in size than the other grids, making simulation findings that are closer to the reference data.

The main difference between the two lies in the turbulence model used. Where the LES .turbulent fluctuations, but is still able to model the main characteristics of BFS flow, such as separation zones, reattachment length, and pressure distribution. Thus, this table shows that despite differences in the level of complexity and resolution of the simulations In contrast, the RANS model used by Alvindriyove is more computationally efficient because it only calculates the time average of both models remain relevant for studying BFS flows. The choice between RANS or LES depends largely on accuracy requirements and available computing resources especially in terms of velocity fluctuations and turbulent kinetic energy. However, this model requires much greater computational time due to the finer mesh resolution and timestep scale turbulence structures directly, thus providing more detailed simulation results model has the advantage of capturing large. Based on the

results of the discussion, a comparison between RANS and LES can be seen in Table 2.

CONCLUSION AND SUGGESTION

RANS and LES simulations show a similar trend. The greater the influence of inertial forces, whether through increasing Re or increasing geometric dimensions, results in a longer recirculation zone behind the steps. Both simulations suggest that the geometric design can be adjusted to create a sufficiently stable and slow flow transition zone, allowing sediment particles time and space to settle effectively.

A comparison of RANS and LES simulations shows that both are consistent in predicting the general direction of flow patterns. RANS tends to produce more damped velocity profiles, while LES is able to more accurately depict the dynamics of eddy shedding and interactions. This suggests that the advantage of LES lies in its ability to capture transient flows, while RANS places more emphasis on speed and computational efficiency.

Table 2. Comparison of RANS and LES

Equality	Principle	Turbulence Resolution	Computing Needs	Excess	Lack
RANS	Modeling the effects of turbulence via an averaging approach in the Navier-Stokes equations	It only predicts average turbulence, without capturing small-scale fluctuations	Low (computationally efficient)	Suitable for engineering simulations with long-term turbulent flows. Efficient for engineering designs such as aerodynamics and hydrodynamics	Does not capture detailed turbulence structures. Not suitable for transient and small-scale phenomena
LES	Simulates large-scale turbulence explicitly, while small-scale turbulence is modeled	Captures large eddy directly, but models small scale with a subgrid approach	Medium to high (more expensive than RANS, but cheaper than DNS)	1. More accurate in capturing turbulent dynamics than RANS 2. Suitable for complex flows such as atmospheric vortices, open channel turbulence, etc.	1. Still requires subgrid modeling 2. Expensive for industrial scale simulations

ACKNOWLEDGEMENT

The authors express gratitude to the head and lecturer of the Mining Engineering Master Program, UPN "Veteran" National Development University Yogyakarta, for the guidance and substantial contributions to the research.

REFERENCES

- Ahadi, M., Bergstrom, D.J. and Mazurek, K.A. (2019) 'Application of the two-fluid model to prediction of sediment transport in turbulent open channel flow', *Physics and Chemistry of the Earth, Parts A/B/C*, 113, pp. 73–82. Available at: <https://doi.org/10.1016/j.pce.2019.06.001>.
- Ahadi, M., Bergstrom, D.J. and Mazurek, K.A. (2020) 'Computational fluid-dynamics modeling of the flow and sediment transport in stormwater retention ponds: A review', *Journal of Environmental Engineering*, 146(9). Available at: [https://doi.org/10.1061/\(ASCE\)EE.1943-7870.0001784](https://doi.org/10.1061/(ASCE)EE.1943-7870.0001784).
- Alvindriyove, Rozi, K. and Fajar, B. (2021) 'Studi numerik pengaruh ketinggian step terhadap karakteristik aliran melalui backward-facing step', *Jurnal Teknik Mesin*, 9(4), pp. 553–558.
- Aultman, M., Wang, Z., Auza-Gutierrez, R. and Duan, L. (2022) 'Evaluation of CFD methodologies for prediction of flows around simplified and complex automotive models', *Computers & Fluids*, 236, p. 105297. Available at: <https://doi.org/10.1016/j.compfluid.2021.105297>.
- Ballesteros Martínez, M.Á. and Gaukel, V. (2023) 'Time-averaged analysis and numerical modelling of the behavior of the multiphase flow and liquid lamella thickness inside an internal-mixing ACLR nozzle', *Flow, Turbulence and Combustion*, 110(3), pp. 601–628. Available at: <https://doi.org/10.1007/s10494-023-00406-5>.
- Bayon, A., Valero, D., García-Bartual, R., Vallés-Morán, F. José and López-Jiménez, P.A. (2016) 'Performance assessment of OpenFOAM and FLOW-3D in the numerical modeling of a low Reynolds number hydraulic jump', *Environmental Modelling & Software*, 80, pp. 322–335. Available at: <https://doi.org/10.1016/j.envsoft.2016.02.018>.
- Blocken, B. (2018) 'LES over RANS in building simulation for outdoor and indoor applications: A foregone conclusion?', *Building Simulation*, 11(5), pp. 821–870. Available at: <https://doi.org/10.1007/s12273-018-0459-3>.
- Chaouat, B. (2017) 'The state of the art of hybrid RANS/LES modeling for the simulation of turbulent flows', *Flow, Turbulence and Combustion*, 99(2), pp. 279–327. Available at: <https://doi.org/10.1007/s10494-017-9828-8>.
- Cheng, Z., Hsu, T.-J. and Chauchat, J. (2018) 'An Eulerian two-phase model for steady sheet flow using large-eddy simulation methodology', *Advances in Water Resources*, 111, pp. 205–223. Available at: <https://doi.org/10.1016/j.advwatres.2017.11.016>.
- Chipongo, K., Khiadani, M. and Sookhak Lari, K. (2020) 'Comparison and verification of turbulence Reynolds-averaged Navier–Stokes closures to model spatially varied flows', *Scientific Reports*, 10(1), p. 19059. Available at: <https://doi.org/10.1038/s41598-020-76128-9>.
- Desti, D., Putra, Y.S. and Kushadiwijayanto, A.A. (2021) 'Simulasi lompatan hidrolis pada aliran kanal terbuka menggunakan pendekatan komputasi dinamika fluida', *PRISMA FISIKA*, 9(2), pp. 172–178. Available at: <https://doi.org/10.26418/pf.v9i2.49501>.
- Fazlizan, A., Muzammil, W.K. and Al-Khawlani, N.A. (2025) 'A review of computational fluid dynamics techniques and methodologies in vertical axis wind turbine development', *Computer Modeling in Engineering & Sciences*, 144(2), pp. 1371–1437. Available at: <https://doi.org/10.32604/cmcs.2025.067854>.
- Fridayana, E.N., Hadiwidodo, Y.S., Satrio, D. and Irawan, E.N. (2022) 'Studi model turbulensi pada Vertical Axis Water Turbine (VAWT) menggunakan metode Computational Fluid Dynamics (CFD)', *Syntax Literate: Jurnal Ilmiah Indonesia*, 7(6), pp. 8844–8863.

- He, K., Minelli, G., Wang, J., Gao, G. and Krajnović, S. (2021) 'Assessment of LES, IDDES and RANS approaches for prediction of wakes behind notchback road vehicles', *Journal of Wind Engineering and Industrial Aerodynamics*, 217, p. 104737. Available at: <https://doi.org/10.1016/j.jweia.2021.104737>.
- Ibarra-Udaeta, I., Portal-Porras, K., Ballesteros-Coll, A., Fernandez-Gamiz, U. and Sancho, J. (2020) 'Accuracy of the cell-set model on a single vane-type vortex generator in negligible streamwise pressure gradient flow with RANS and LES', *Journal of Marine Science and Engineering*, 8(12), p. 982. Available at: <https://doi.org/10.3390/jmse8120982>.
- Iberl, A., Trautner, E., Klein, M. and Hasslberger, J. (2025) 'A-posteriori assessment of mixed models for Large Eddy Simulation of polydisperse multiphase flows', *International Journal of Multiphase Flow*, 189, p. 105231. Available at: <https://doi.org/10.1016/j.ijmultiphaseflow.2025.105231>.
- Kadivar, M., Tormey, D. and McGranaghan, G. (2023) 'A comparison of RANS models used for CFD prediction of turbulent flow and heat transfer in rough and smooth channels', *International Journal of Thermofluids*, 20, p. 100399. Available at: <https://doi.org/10.1016/j.ijft.2023.100399>.
- Knopp, T., Korsmeier, P., Strelets, M. and Guseva, E. (2021) 'Study of RANS Turbulence Models for Turbulent Wake Flow in Adverse Pressure Gradient', in *New Results in Numerical and Experimental Fluid Mechanics XIII*. Springer Cham, pp. 58–67. Available at: https://doi.org/10.1007/978-3-030-79561-0_6.
- Li, P., Eckels, S.J., Zhang, N. and Mann, G.W. (2016) 'Effects of parallel processing on Large Eddy Simulations in ANSYS fluent', in *Volume 1B, Symposia: Fluid Mechanics (Fundamental Issues and Perspectives; Industrial and Environmental Applications); Multiphase Flow and Systems (Multiscale Methods; Noninvasive Measurements; Numerical Methods; Heat Transfer; Performance); Transport Phe*. American Society of Mechanical Engineers. Available at: <https://doi.org/10.1115/FEDSM2016-7884>.
- Liu, S., Xu, L., Chen, B., Deng, Z., Li, P., Zhao, R. and Chen, Q. (2025) 'Recent progress in optimization of RANS turbulence models for accurate urban airflow and contaminant dispersion simulations', *Sustainable Cities and Society*, 125, p. 106336. Available at: <https://doi.org/10.1016/j.scs.2025.106336>.
- Lopes, D., Puga, H., Teixeira, J., Lima, R., Grilo, J., Dueñas-Pamplona, J. and Ferrera, C. (2024) 'Comparison of RANS and LES turbulent flow models in a real stenosis', *International Journal of Heat and Fluid Flow*, 107, p. 109340. Available at: <https://doi.org/10.1016/j.ijheatfluidflow.2024.109340>.
- Marefat, H.A., Alam, J.M. and Pope, K. (2024) 'Toward scale-adaptive subgrid-scale model in LES for turbulent flow past a sphere', *Fluids*, 9(6), p. 144. Available at: <https://doi.org/10.3390/fluids9060144>.
- Masoudian, M., Pinho, F.T., Kim, K. and Sureshkumar, R. (2016) 'A RANS model for heat transfer reduction in viscoelastic turbulent flow', *International Journal of Heat and Mass Transfer*, 100, pp. 332–346. Available at: <https://doi.org/10.1016/j.ijheatmasstransfer.2016.04.053>.
- Masrur, M.. (2016) *Simulasi numerik 3D pengaruh penambahan triangular bump pada aliran backward-facing step*. Institut Teknologi Sepuluh Nopember.
- Mishra, B.K., Rai, S., Gavhane, K. and Deshpande, M. (2024) 'Turbulence modeling in navier-stokes equations: Challenges and approaches', *International Research Journal of Modernization in Engineering Technology and Science*, 6(4), pp. 6933–6940. Available at: <https://doi.org/10.56726/IRJMETS53610>.
- Nurhaliza, N., Putra, Y.S. and Kushadiwijayanto, A.A. (2023) 'Studi numerik pola aliran di sekitar pintu air menggunakan pendekatan komputasi dinamika fluida', *PRISMA FISIKA*, 10(3), pp. 241–250. Available at: <https://doi.org/10.26418/pf.v10i3.57851>.
- Quaresma, A.L., Romão, F. and Pinheiro, A.N. (2025) 'A comparative assessment of Reynolds-averaged navier-stokes and large-eddy simulation models: Choosing the best for pool-type fishway flow simulations', *Water*, 17(5), p. 686. Available at: <https://doi.org/10.3390/w17050686>.
- Rodríguez Berrio, J.F., Castaño Usuga, F.A., Correa, M.A., Rodríguez Cortes, F. and

- Saldarriaga, J.C. (2025) 'Comparative CFD analysis using RANS and LES models for NOx dispersion in urban streets with active public interventions in Medellín, Colombia', *Sustainability*, 17(15), p. 6872. Available at: <https://doi.org/10.3390/su17156872>.
- Samuels, C., Watling, T.C., Aleksandrova, S., Medina, H., Holtzman, R., Rusli, I. and Benjamin, S. (2025) 'Numerical investigation of the effect of an oblique flow entry on the pressure losses in square channels', *Fluid Dynamics* [Preprint].
- Sukhodolov, A.N., Krick, J., Sukhodolova, T.A., Cheng, Z., Rhoads, B.L. and Constantinescu, G.S. (2017) 'Turbulent flow structure at a discordant river confluence: Asymmetric jet dynamics with implications for channel morphology', *Journal of Geophysical Research: Earth Surface*, 122(6), pp. 1278–1293. Available at: <https://doi.org/10.1002/2016JF004126>.
- Tang, T.-T., Li, F.-Q., Wang, G.-Y., Yan, J. and Lu, Z.-K. (2025) 'Comparative study of RANS models for simulating turbulent flow and heat transfer in corrugated pipes', *Water*, 17(17), p. 2649. Available at: <https://doi.org/10.3390/w17172649>.
- Vita, G., Salvadori, S., Misul, D.A. and Hemida, H. (2020) 'Effects of inflow condition on RANS and LES predictions of the flow around a high-rise building', *Fluids*, 5(4), p. 233. Available at: <https://doi.org/10.3390/fluids5040233>.
- Wan Ismail, W.H., Mohamad, M.F., Ikegaya, N., Chung, J., Hirose, C., Abd Razak, A. and Mohd Azmi, A. (2024) 'Comprehensive comparisons of RANS, LES, and experiments over cross-ventilated building under sheltered conditions', *Building and Environment*, 254, p. 111402. Available at: <https://doi.org/10.1016/j.buildenv.2024.111402>.
- Wang, F., Wu, S. and Zhu, S. (2019) 'Numerical simulation of flow separation over a backward-facing step with high Reynolds number', *Water Science and Engineering*, 12(2), pp. 145–154. Available at: <https://doi.org/10.1016/j.wse.2019.05.003>.
- Yan, Z., Zhong, Y., Lin, W.E., Savory, E. and You, Y. (2018) 'Evaluation of RANS and LES turbulence models for simulating a steady 2-D plane wall jet', *Engineering Computations*, 35(1), pp. 211–234. Available at: <https://doi.org/10.1108/EC-11-2016-0397>.

