http://journal.uir.ac.id/index.php/JGEET

E-ISSN : 2541-5794
P-ISSN : 2503-216X

Journal of Geoscience,
Engineering, Environment, and Technology
Vol 10 No 3 2025

RESEARCH ARTICLE

Comparison of Laminar, RNG, and LES Model for Wave Propagation Simulation with FLOW-3D

Rizaldi Caesar Yuniardi [1,*], Dinar Catur Istiyanto[1], Ika Wulandari[1], Yofan Tahamano Deo Harita[1], Raka Firmansyah[1], Reno Arief Rachman[1]

[1] *Research Center for Hydrodynamics Technology, National Research and Innovation Agency, Sukolilo, 60112 Surabaya, Indonesia.*

* Corresponding author : riza005@brin.go.id
Tel.:+62-823-209-54732; fax: +62-823-209-54732
Received: Jul 28, 2023; Accepted: Jul 24, 2025.
DOI: 10.25299/jgeet.2025.10.3.13933

Abstract

Understanding wave transformation and its interaction with coastal structures is critical for shoreline protection and design. While physical modeling has traditionally supported such studies, its high cost has led to increased reliance on numerical modeling. This study uses FLOW-3D to simulate wave propagation in a scaled 2D coastal wave channel and compares the performance of three turbulence models: Laminar, Renormalized Group (RNG) k-ε, and Large Eddy Simulation (LES). Simulations are based on a 3-meter wave height, 12-second period, and 7-meter water depth, with wave elevation recorded at six probes along the domain. Results show that the LES model achieved the most accurate prediction, with a significant wave height of 3.01 meters at the structure location—an error of only 0.33%—outperforming RNG and laminar models. These findings highlight the superior turbulence resolution of LES in capturing energy dissipation and wave evolution. The study provides practical guidance for coastal engineers in selecting turbulence models based on accuracy and computational trade-offs. Future research should include model validation with experimental or field data and extend to irregular wave conditions to enhance real-world applicability.

Keywords: Numerical Modeling, CFD, FLOW-3D, Laminar, RNG, LES.

1. Introduction

Over the past few decades, research in the field of Coastal Engineering in coastal areas has been growing and garnering significant interest. The coastal area is a highly dynamic environment characterized by continuously changing coastlines and interconnected ecosystems, with key areas of focus including wave dynamics, shoreline protection, beach erosion, port and breakwater design, reclamation and dredging technology, estuary dynamics, and offshore structure design (Reno Arief Rachman et al., 2023). Hydraulic physical modeling and laboratory experiments are commonly utilized for planning and investigating the physical phenomena that occur in prototypes. To simplify and reduce costs, a physical modeling approach with a model scale smaller than the prototype is often used, especially in the field of hydraulics and ship design, to proportionally represent the forces in the model (Hugues, 1996). But, along with the high cost of physical modeling, researchers have increasingly turned to using numerical modeling. Numerical modeling has significantly increased with the rapid development of computer capabilities, providing continuous algorithms to obtain accurate solutions. Nonetheless, validating and calibrating numerical results with experimental and prototype data remain essential for ensuring accuracy and reliability.

Computational Fluid Dynamics (CFD) method is a branch of numerical modeling that can be used to solve and predict fluid flow, heat and mass transfer, chemical reactions, and transport phenomena. The CFD method has been widely used in hydraulic engineering research topics, especially open channels starting in the 1980s. Various studies have been conducted, including research on circulation in lakes (Georgoulas et al., 2010; Simons, 1974), flow in rivers (Baranya and Jozsa, 2006), sediment deposition and transport (Huggins et al., 2004), laminar flow (Sahu et al., 2009), all of which are integral to environmental and hazard assessments in dynamic coastal zones (Ondara et al., 2020).

The CFD modeling approach, particularly using FLOW-3D, has been applied to model spillway buildings, solving the Reynolds-averaged Navier Stokes (RANS) equation (Ho et al., 2003; Kim et al., 2010; Savage and Johnson, 2001). For regulating flow in numerical simulations, the Navier-Stokes equations and turbulence models are solved using numerical methods (Parsaie et al., 2015b, 2015a; Parsaie and Haghiabi, 2017; Versteeg and Malalasekera, 2007). Generally, CFD predicts turbulent flow through three approaches: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds-averaged Navier Stokes (RANS). RANS is considered accurate, requiring relatively low computational effort. Various options in RANS include k-ε, k-ω, and k-ω SST turbulent models, each with its advantages and disadvantages.

However, despite the extensive use of CFD in wave modeling, limited studies have compared the performance of different turbulence models under field-scale wave propagation scenarios. Most previous research either focused on idealized or small-scale laboratory setups or evaluated model performance only for steady flow conditions (Dogan et al., 2017; Kumari et al., 2025; Lee and Wahab, 2019). This leaves a gap in understanding how well different turbulence models capture dynamic wave behavior in coastal-scale environments.

This study addresses this gap by comparing three turbulence models—Laminar, RNG, and LES—under realistic wave conditions (3 m wave height, 12 s wave period, 7 m depth). The focus on these three models allows us to contrast the lower-complexity laminar approach, the commonly used RANS-based RNG model, and the more computationally demanding but potentially more accurate LES approach. The exclusion of other RANS models (e.g., k-ε, k-ω) was deliberate, as RNG offers a balanced trade-off between stability and realism for free surface flows (Roy-Biswas and Sen, 2022), and previous studies have shown RNG performing comparably or better than other RANS models in certain wave and vortex conditions (Dogan et al., 2017; Lee and Wahab, 2019).

Therefore, the objective of this study is to assess the relative accuracy of these turbulence models in simulating wave height transformation and propagation using FLOW-3D, with particular attention to performance at the structure location where precision is critical for design applications.

2. Theoretical Background

Fluid flow can be categorized as either laminar or turbulent, based on the regularity of particle motion. In laminar flow, fluid particles move in parallel layers with relatively uniform velocities. In contrast, turbulent flow exhibits chaotic motion, fluctuating velocity fields, and eddy formations. The distinction between these regimes is governed by the Reynolds number (Re), defined by equation (1) below.

$$Re = \frac{u.L}{\vartheta} \qquad (1)$$

$$\vartheta = \frac{\mu}{\rho} \qquad (2)$$

Where ϑ is the kinematic viscosity of the fluid (m2/s), L is the characteristic length (in open channels, which is considered the hydraulic radius) (m), u is the average velocity (m/s), ρ is the density of the fluid (kg/m3), and μ is the dynamic viscosity (Pa.s).

In general, Re < 2000 corresponds to laminar flow, and Re > 4000 indicates turbulence. For this study's wave conditions (wave height = 3 m, wave period = 12 s, depth = 7 m), the velocity is estimated as 1.57 m/s. Using a hydraulic radius L = 7 m and $\nu \approx 1.0 \times 10^{-6}$ m^2/s, we compute:

$$Re \approx (1.57 \times 7) / (1.0 \times 10^{-6}) = 1.1 \times 10^{7}$$

This confirms the fully turbulent nature of the flow. Turbulence modeling is therefore necessary.

In hydraulic engineering, physical modeling remains a dependable method to analyze flow over structures, yet it is costly and scale-dependent (Ettema Robert et al., 2000; Hager and Pfister, 2010; Suprapto, 2013). As computing power has advanced, numerical modeling using Computational Fluid Dynamics (CFD) has become a powerful alternative. CFD software like FLOW-3D solves the Navier-Stokes equations (FLOW-3D, 2019), which describe fluid motion based on conservation of mass, momentum, and energy, that can be expressed in equation (3) to equation (6).

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad (3)$$

The components of Momentum in the x-direction, y-direction, and z-direction are expressed in equations (4), (5), and (6), respectively.

$$\frac{\partial}{\partial t}(\rho u) + \frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho vu) + \frac{\partial}{\partial z}(\rho wu) = -\frac{\partial \rho}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad (4)$$

$$\frac{\partial}{\partial t}(\rho v) + \frac{\partial}{\partial x}(\rho uv) + \frac{\partial}{\partial y}(\rho vv) + \frac{\partial}{\partial z}(\rho wv) = -\frac{\partial \rho}{\partial y} + (v + v_t)\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \qquad (5)$$

$$\frac{\partial}{\partial t}(\rho w) + \frac{\partial}{\partial x}(\rho uw) + \frac{\partial}{\partial y}(\rho vw) + \frac{\partial}{\partial z}(\rho ww) = -\frac{\partial \rho}{\partial z} + (v + v_t)\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \qquad (6)$$

Where u, v, and w are the velocity components in the x, y, and z directions, respectively, vt is the turbulent kinematic viscosity, t is time, and P is pressure.

Turbulent flow, characterized by its chaotic and multiscale nature, presents significant challenges in simulation, making the selection of the most appropriate turbulence model a complex task that depends on factors such as flow characteristics, desired accuracy, and available computational resources (Gildeh, 2013). Since direct modeling of turbulence is often infeasible in practical simulations, turbulence models are used to approximate the effects of unresolved, smaller eddies. FLOW-3D offers six turbulence modeling options, with the Laminar, RNG, and LES models chosen for this study to represent varying levels of modeling complexity. The laminar model, assuming no turbulence, is typically applied in low-Reynolds number or highly viscous flows and, although not suitable for high-energy wave propagation, serves as a baseline for comparison.

The Renormalized Group (RNG) k-ε model is an extension of the standard RANS (Reynolds-Averaged Navier-Stokes) turbulence model. It solves two transport equations for turbulent kinetic energy (k) and its dissipation rate (ε). Derived from renormalization group theory, this model includes additional terms to improve its accuracy in predicting rapidly strained, curved, and swirling flows (Yakhot and Orszag, 1986); (Setiawan and Darmawan dan Harto Tanujaya, 2022). It has also shown reliable performance in dam-break simulations and surge wave propagation, capturing general flow characteristics effectively (Roy-Biswas and Sen, 2022) and providing reasonable accuracy in simulating hydraulic jumps and impinging flows (Guo et al., 2020).

In contrast, the Large Eddy Simulation (LES) model offers a higher-fidelity alternative that resolves large-scale turbulent structures while using subgrid-scale models for the smallest eddies. LES model directly computes turbulent flow structures resolved by the computational grid while approximating smaller features. This makes LES results more comprehensive and informative, especially in cases with complex flow phenomena (FLOW-3D, 2019). LES excels in capturing time-dependent flow phenomena, such as wave breaking and vortex shedding, which are critical in coastal applications. It has been found to outperform RANS-based models in simulating flow around hydraulic structures, accurately predicting vortex interactions and unsteady behaviors (Dogan et al., 2017). LES also demonstrated superior performance in simulating dam-break waves over mobile beds (Heydari and KhoshKonesh, 2016), free-surface profiles in stepped spillways (Yalcin et al., 2023), and turbulent wake flows around submerged structures (Kumari et al., 2025). In a CFD-physical modeling comparison for a hydro power spillway, LES effectively captured dominant frequencies and fluctuating pressure patterns not resolved by RNG models (Duró et al., 2012).

In summary, the selection of these three turbulence models—Laminar, RNG, and LES—allows for a comprehensive evaluation of their relative accuracy and computational efficiency in modeling field-scale wave propagation in coastal environments. This tiered comparison is essential for understanding the trade-offs between simplicity and realism in numerical wave modeling.

3. Methodology

3.1 Stage of Study

This study uses a quantitative numerical modeling approach in FLOW-3D to compare fluid flow behavior under laminar and turbulent wave conditions. The simulation process consists of three core components: Pre-Processor, Solver Manager, and Postprocessor.

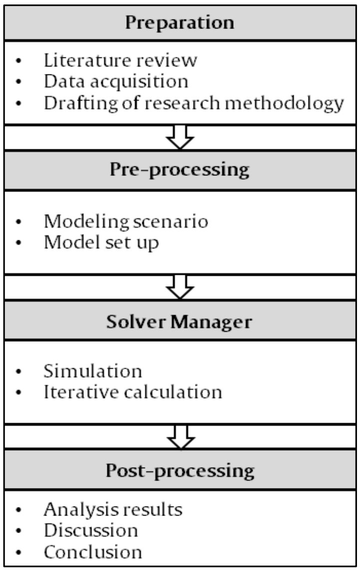

Fig. 1. Flowchart of stage activities.

In the Pre-Processor stage, the geometry of the domain is created, the mesh is generated, and the boundary and initial conditions are defined. This includes specifying the fluid's physical properties—such as density (ρ = 1000 kg/m^3), dynamic viscosity (μ = 1.0 × 10^{-3} Pa·s), and temperature (assumed constant at 25°C). The meshing process uses a Cartesian grid with a uniform cell structure, a base mesh size of 0.25 m was used, resulting in approximately 2.49 million computational cells for the full domain. A mesh sensitivity analysis was conducted by comparing simulation results at multiple resolutions.

Simulations were run using FLOW-3D v12.0 with the Solver Manager performs time-stepping iterations based on the selected turbulence model, solving the Navier-Stokes equations with volume of fluid (VoF) methods for surface tracking. The Postprocessor visualizes the output in terms of wave height, velocity field, and pressure distribution. Results were analyzed through probe data, surface plots, and time-series graphs. A flowchart outlining the full simulation workflow is shown in Fig. 1.

3.2 Model

The simulation is based on a scaled physical model of the 2D wave channel at the Port Infrastructure Engineering and Coastal Dynamics Laboratory in Yogyakarta. A 1:30 geometric scale is applied to field dimensions, with the domain set to 1080 m (length) × 60 m (width) × 48 m (height) in prototype scale. In the FLOW-3D model, the effective domain height is adjusted to the water depth (7 m), and the width is reduced to minimize computational demand without compromising physical accuracy.

The wave input is a regular Stokes second-order wave with a wave height of 3 m and period of 12 s, propagated through the X-min boundary using FLOW-3D's built-in wave generation functions. The wave length (L) is calculated to be 122.9 m, and six wave probes (P0–P5) are placed at locations corresponding to 0L, 1L, 2L, 4L, 6L, and 7L from the wave source. The final probe (P5) is located at 7L, where a structure is planned for later analysis.

The domain boundaries were defined as follows: X-min was set as a wave-generating inlet using Stokes second-order formulation; X-max used an outflow condition with a damping zone to absorb waves. Y-min and Y-max were treated as symmetry planes to simplify lateral flow. Z-min was assigned as a no-slip wall representing the channel bed, and Z-max was defined as a pressure boundary to capture free-surface motion using the Volume of Fluid (VOF) method. These boundary conditions are summarized in **Error! Reference source not found.**, and the probe configuration is illustrated in Fig. 2.

Table 1. Boundary Condition in Model.

Boundaries	Boundary Condition
Xmin	Wave*; H=3 m, T=12 s, d=7 m
Xmax	Outflow**
Ymin	Symmetry
Ymax	Symmetry
Zmin	Wall
Zmax	Specified pressure

* Stokes and Cnoidal wave
** with absorbing layer 1L

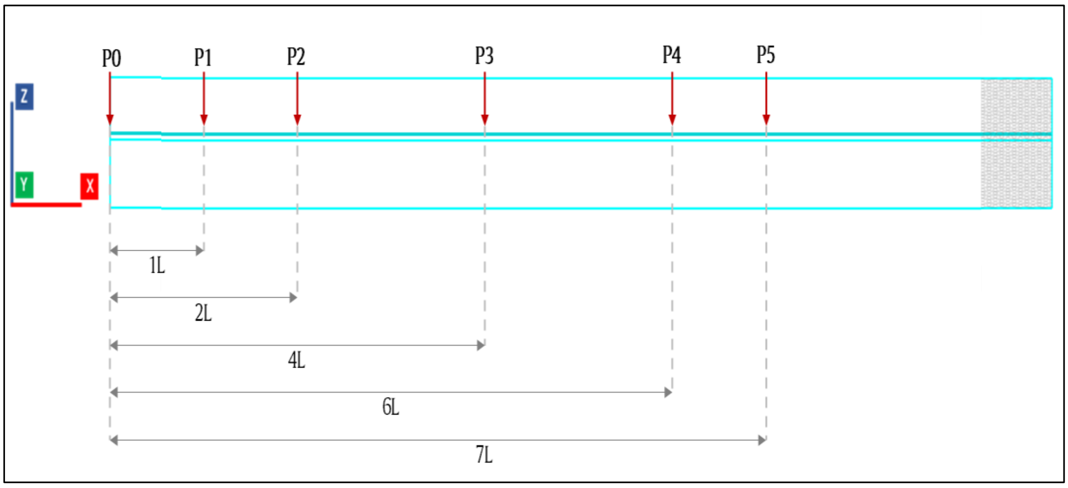

Fig. 2. Model Domain and Probes Assignment in FLOW-3D.

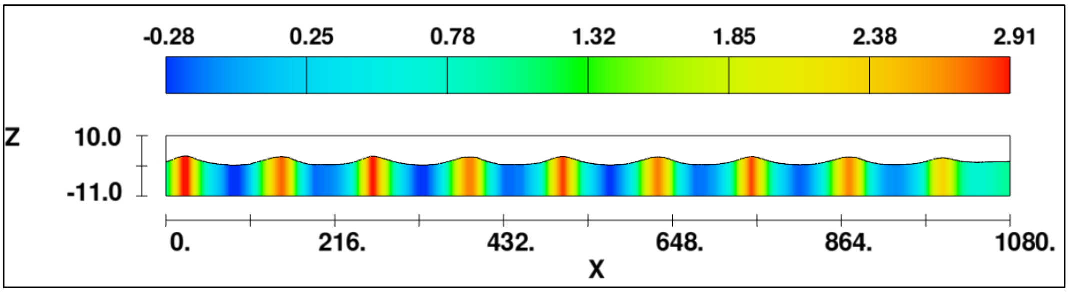

Fig. 3. Water Level Elevation Contours Result From FLOW-3D.

4. Results and Discussion

4.1 Water Level Elevation

The FLOW-3D simulation results provide time-series data for water level elevation at six wave probes (P0 to P5), corresponding to positions from the wave generation zone to the structure location. These simulations used a target input wave height of 3 meters and a wave period of 12 seconds. Sample water surface contour outputs are shown in Fig. 6, and time-series water elevation graphs for each model (laminar, RNG, and LES) are presented in **Error! Reference source not found.** to **Error! Reference source not found.**.

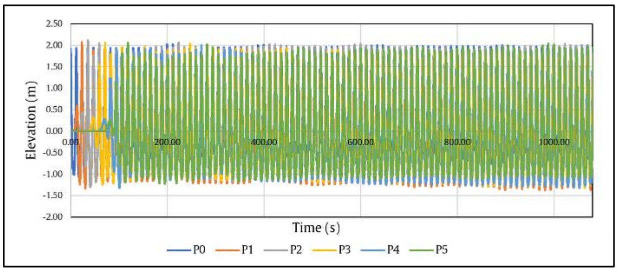

Fig. 4. Time Series Water Level Elevation at P0 – P5 for Laminar Model.

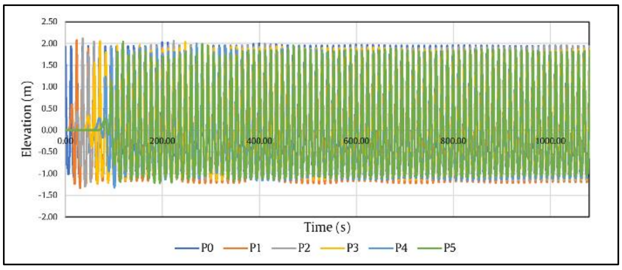

Fig. 5. Time Series Water Level Elevation at P0 – P5 for RNG Model.

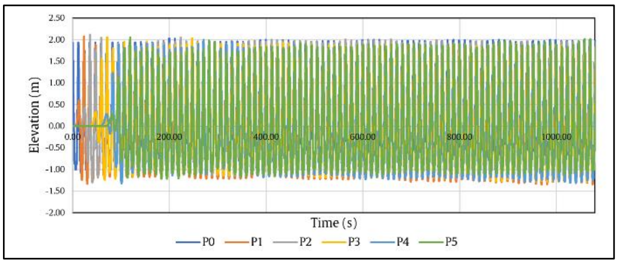

Fig. 6. Time Series Water Level Elevation at P0 – P5 for LES Model.

Each turbulence model captures the wave progression and decay as it propagates through the domain. As expected, a gradual reduction in wave height is observed moving from P0 to P5 due to energy dissipation. These wave elevation results form the basis for further comparison using derived significant wave heights.

4.2 Significant Wave Height

Significant wave heights were extracted using a zero up-crossing analysis from the time-series data at each probe. These results are summarized in Table 2 and visualized in Fig. 7. The analysis focuses particularly on probe P5, located at 7L from the inlet, where the structure is planned.

Table 2. Summary of Significant Wave Height and Period.

Probe	Laminar		RNG		LES	
	Hs (m)	Ts (s)	Hs (m)	Ts (s)	Hs (m)	Ts (s)
P0	3.13	11.75	3.12	11.75	3.16	11.74
P1	3.07	11.74	3.08	11.75	3.14	11.74
P2	3.03	11.73	3.08	11.74	3.12	11.74
P3	2.94	11.73	3.03	11.70	3.08	11.71
P4	2.92	11.68	2.98	11.67	3.02	11.67
P5	2.90	11.70	2.95	11.68	3.01	11.70

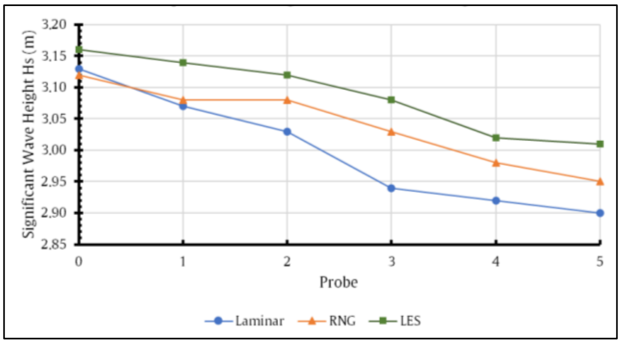

Fig. 7. Comparison of Significant Wave Height at P0 – P5 for Laminar, RNG, and LES Model.

To evaluate model accuracy, Root Mean Square Error (RMSE) and Mean Absolute Error (MAE) were computed by comparing the simulated significant wave heights at P5 with the design value of 3.00 m, as shown in Table 3 below.

Table 3. Error at P5, RMSE, and MAE of significant wave heights predicted by each turbulence model.

Model	Hs at P5 (m)	Error at P5 (%)	RMSE (m)	MAE (m)
Laminar	2.90	3.33%	0.084	0.078
RNG	2.95	1.67%	0.072	0.063
LES	3.01	0.33%	0.105	0.088

The RMSE and MAE values calculated across probes P0 to P5 show that the RNG model yields the lowest overall error, with an RMSE of 0.072 m and an MAE of 0.063 m, compared to 0.105 m and 0.088 m for LES, and 0.084 m and 0.078 m for the laminar model. This suggests that RNG offers a more uniform prediction across the entire domain. However, a closer inspection at the structure location (P5) reveals that the LES model produces the most accurate result, predicting a wave height of 3.01 m—only 0.33% above the input value of 3.00 m. This contrast can be explained by the underlying characteristics of the turbulence models. While RNG, as a RANS-based model, smooths out turbulent fluctuations and thus provides stable average performance over distance, it may underperform in zones where localized, unsteady flow structures dominate—such as near coastal structures. LES, by resolving large-scale eddies and capturing energy dissipation more precisely, excels in such dynamic regions, which is why it performs best at the structure location where precision is critical. In contrast, the laminar model consistently underpredicts wave height due to its neglect of turbulence altogether. These findings underscore the importance of selecting a turbulence model based not only on overall statistical performance, but also on the location and sensitivity of the design objectives within the modeled domain.

5. Discussions

The simulation results demonstrate that the Large Eddy Simulation (LES) model outperforms both the RNG and laminar models in predicting wave propagation, particularly at the structure location (P5). This superior performance is due to LES's ability to directly resolve large-scale turbulent eddies that dominate energy transport in breaking waves and near-structure flows. By computing unsteady, grid-resolved turbulence, LES accurately captures energy dissipation and the evolution of wave forms. Previous studies confirm that LES provides higher fidelity in modeling wave–structure interactions and free-surface turbulence, particularly in areas like the lee of submerged breakwaters where RANS fails to conserve turbulent kinetic energy (Chang et al., 2020).

RNG and other RANS-based models, while computationally more efficient, rely on time-averaged approximations that smooth out transient features critical to coastal wave modeling. These models often underpredict turbulence intensity and miss eddy-induced mixing and surface deformation. Similar trends are reported in wave-driven flow studies near breakwaters, where LES captured stronger vortices and more realistic sediment dispersion compared to RANS (Protsenko, 2024), and in unsteady sloshing simulations (Liu et al., 2016). Likewise, (Giannissi et al., 2021; Tominaga and Stathopoulos, 2011) emphasized that LES better reproduces unsteady turbulence structures in environmental flows.

Despite its accuracy, LES demands significantly greater computational resources, including finer grids and smaller time steps. This trade-off is well documented, with hybrid RANS/LES models developed to reduce cost while retaining key LES benefits in turbulent zones (Boehler et al., 2020). In contrast, the laminar model, which entirely neglects turbulence, leads to unacceptable underestimations in wave height—up to 3.3% at the structure location in this study—posing a risk in design applications.

The practical impact of selecting an inappropriate model can be substantial. Underestimating wave height may lead to under-designed structures, increasing the risk of overtopping, failure, or inaccurate assessments of scour and sediment transport. For high-stakes design scenarios such as seawalls, revetments, or port structures, LES provides a higher confidence level in predicting wave-induced forces and flow behaviors. RNG may offer a reasonable balance for broader planning or early-stage design where resource limitations exist.

In conclusion, LES proves to be the most accurate and physically representative model for wave transformation and coastal turbulence in this study. However, RNG remains a practical middle ground for less sensitive applications. The findings emphasize the importance of matching the turbulence model to the precision and scale of the engineering task.

6. Conclusion

This study used FLOW-3D to simulate wave propagation in a scaled 2D wave channel, based on input parameters of 3-meter wave height, 12-second wave period, and 7-meter water depth. The comparative analysis of three turbulence models—Laminar, RNG, and Large Eddy Simulation (LES)—revealed that the LES model produced the most accurate results, achieving a significant wave height of 3.01 meters at the structure location, with only a 0.33% deviation from the target input height.

These results suggest that LES should be prioritized in simulations requiring high precision, especially for modeling wave transformation near critical infrastructure, where accurate prediction of wave energy and flow behavior is essential for design reliability. In contrast, RNG may offer a suitable alternative in early design stages or when computational resources are limited, while the laminar model is not recommended for high-Reynolds number coastal flow scenarios due to its inability to capture turbulence and energy dissipation.

This study provides valuable guidance for coastal engineers and CFD practitioners in selecting appropriate turbulence models based on the trade-off between accuracy and computational cost. To ensure broader applicability, future work should include validation using physical model data or field measurements to verify numerical accuracy. Additional research should also explore random wave inputs using realistic spectra such as JONSWAP, Pierson-Moskowitz, or Bretschneider, and incorporate wave–structure interaction modeling to simulate more complex coastal engineering scenarios or long-period wave phenomena, which have been shown to affect coastal sea level and hydrodynamic patterns (Khoirunnisa et al., 2020).

Acknowledgements

We extend our gratitude to the management and staff of the Research Center for Hydrodynamics Technology - National Research and Innovation Agency, particularly to the members of the Port Infrastructure and Coastal Building Technology Research Group and the Port Infrastructure and Coastal Dynamics Technology Laboratory, as well as all those who provided assistance in conducting this research.

References

Baranya, S., Jozsa, J., 2006. Flow Analysis In River Danube By Field Measurement and 3D CFD Turbulence Modelling. Periodica Polytechnica (Budapest University of Technology and Economics 50, 57–68.

Boehler, M., Sudesh, A., Turner, M., 2020. Hybrid RANS-LES of Shaped Hole Film Cooling on an Adiabatic Flat Plate at Low Reynolds Number. Open Journal of Fluid Dynamics 10, 317–341. https://doi.org/10.4236/ojfd.2020.104019

Chang, Y.S., Do, J.D., Jeong, W.M., Hyun, S.K., Baek, W.-D., Choi, J.H., Jin, J.-Y., 2020. Comparison of Turbulent Flows and Suspended Sediment Particle Motions Simulated around a Submerged Breakwater Using

RANS and LES. Ocean Science Journal 55, 1–16. https://doi.org/10.1007/s12601-020-0009-7
Dogan, S., Yagmur, S., Goktepeli, I., Ozgoren, M., 2017. Assessment of Turbulence Models for Flow around a Surface-Mounted Cube. International Journal of Mechanical Engineering and Robotics Research 237–241. https://doi.org/10.18178/ijmerr.6.3.237-241
Duró, G., De Dios, M., López, A., Liscia, S.O., 2012. Physical Modeling and CFD Comparison: Case Study of a Hydro-Physical Modeling and CFD Comparison: Case Study of a Hydro-Combined Power Station in Spillway Mode Combined Power Station in Spillway Mode PHYSICAL MODELING AND CFD COMPARISON: CASE STUDY OF A HYDRO-COMBINED POWER STATION IN SPILLWAY MODE.
Ettema Robert, Arndt, R., Roberts, P., Wahl, T., 2000. Hydraulic modeling: Concepts and practice. American Society of Civil Engineers.
FLOW-3D, 2019. FLOW-3D® v.12 Users Manual.
Georgoulas, A.N., Angelidis, P.B., Panagiotidis, T.G., Kotsovinos, N.E., 2010. 3D Numerical modelling of turbidity currents. Environmental Fluid Mechanics 10, 603–635. https://doi.org/10.1007/s10652-010-9182-z
Giannissi, S.G., Tolias, I.C., Melideo, D., Baraldi, D., Shentsov, V., Makarov, D., Molkov, V., Venetsanos, A.G., 2021. On the CFD modelling of hydrogen dispersion at low-Reynolds number release in closed facility. Int J Hydrogen Energy 46, 29745–29761. https://doi.org/10.1016/j.ijhydene.2020.09.078
Gildeh, H.K., 2013. Numerical Modeling of Thermal/Saline Discharges in Coastal Waters.
Guo, G., Zhang, R., Yu, H., 2020. Evaluation of different turbulence models on simulation of gas-liquid transient flow in a liquid-ring vacuum pump. Vacuum 180, 109586. https://doi.org/10.1016/j.vacuum.2020.109586
Hager, W.H., Pfister, M., 2010. Hydraulic modelling – an introduction: principles, methods and applications. Journal of Hydraulic Research 48, 557–558. https://doi.org/10.1080/00221686.2010.492104
Heydari, M., KhoshKonesh, A., 2016. The Comparison of the Performance of Prandtl| Mixing Length, Turbulence Kinetic Energy, K-e, RNG and LES Turbulence Models in Simulation of the Positive Wave Motion Caused by Dam Break on the Erodible Bed. Indian J Sci Technol 9. https://doi.org/10.17485/ijst/2016/v9i7/87856
Ho, D., Boyes, K., Donohoo, S., Cooper, B., 2003. Numerical Flow Analysis for Spillways, in: 43rd ANCOLD Conference. pp. 24–29.
Huggins, D.L., Piedrahita, R.H., Rumsey, T., 2004. Analysis of sediment transport modeling using computational fluid dynamics (CFD) for aquaculture raceways. Aquac Eng 31, 277–293. https://doi.org/10.1016/j.aquaeng.2004.05.007
Hugues, S., 1996. Physical Models and Laboratory Techniques in Coastal Engineering.
Khoirunnisa, H., Arif Rachman, R., Sari Ningsih, N., Syamsudin, F., 2020. The Role of Decadal Kelvin Wave in the Western of Sumatra and Along the South Coast of Java using Frequency-Wavenumber 2D Spectral Analysis. Journal of Geoscience, Engineering, Environment, and Technology 5, 40–46. https://doi.org/10.25299/jgeet.2020.5.1.3889
Kim, S.-D., Lee, H.-J., An, S.-D., 2010. Improvement of hydraulic stability for spillway using CFD model, International Journal of the Physical Sciences.
Kumari, S., Tiwari, H.L., Chavan, R., 2025. Analyzing the effectiveness of turbulence models in predicting flow around tandem piers. Ocean Engineering 321, 120467. https://doi.org/10.1016/j.oceaneng.2025.120467
Lee, H.C., Wahab, A.K.A., 2019. Performance of different turbulence models in predicting flow kinematics around an open offshore intake. SN Appl Sci 1, 1266. https://doi.org/10.1007/s42452-019-1320-8
Liu, D., Tang, W., Wang, J., Xue, H., Wang, K., 2016. Comparison of laminar model, RANS, LES and VLES for simulation of liquid sloshing. Applied Ocean Research 59, 638–649. https://doi.org/10.1016/j.apor.2016.07.012
Ondara, K., Dhiauddin, R., Wisha, U.J., Rahmawan, G.A., 2020. Hydrodynamics Features and Coastal Vulnerability of Sayung Sub-District, Demak, Central Java, Indonesia. Journal of Geoscience, Engineering, Environment, and Technology 5, 32–39. https://doi.org/10.25299/jgeet.2020.5.1.3996
Parsaie, A., Haghiabi, A.H., 2017. Computational Modeling of Pollution Transmission in Rivers. Appl Water Sci 7, 1213–1222. https://doi.org/10.1007/s13201-015-0319-6
Parsaie, A., Haghiabi, A.H., Moradinejad, A., 2015a. CFD modeling of flow pattern in spillway's approach channel. Sustain Water Resour Manag 1, 245–251. https://doi.org/10.1007/s40899-015-0020-9
Parsaie, A., Yonesi, H.A., Najafian, S., 2015b. Predictive modeling of discharge in compound open channel by support vector machine technique. Model Earth Syst Environ 1. https://doi.org/10.1007/s40808-015-0002-9
Protsenko, S. V., 2024. Modelling Turbulent Flows near Coastal Structures Using Various Turbulence Models. Computational Mathematics and Information Technologies 8, 55–62. https://doi.org/10.23947/2587-8999-2024-8-1-55-62
Reno Arief Rachman, Haryo Dwito Armono, Dinar Catur istiyanto, Khusnul Setia Wardani, Hanah Khoirunnisa, Reni Wijayanti, 2023. Study of Hydrodynamic Patterns of Tanjung Pasir Banten Waters to Support the NCICD Sea Wall Development Plan. Journal of Geoscience, Engineering, Environment, and Technology 8, 1–9. https://doi.org/10.25299/jgeet.2023.8.1.10801
Roy-Biswas, T., Sen, D., 2022. Critical analysis of turbulence models for simulating positive surge waves in open channels in a RANS-VOF setup. Environmental Fluid Mechanics 22, 1253–1281. https://doi.org/10.1007/s10652-022-09889-8
Sahu, M., Khatua, K.K., Patra, K.C., Naik, T., 2009. Developed Laminar Flow In Pipe Using Computational Fluid Dynamics, in: 7th International R &D Conference on Development and Management of Water and Energy Resources. Bhubaneswar.

Savage, B.M., Johnson, M.C., 2001. Flow over Ogee Spillway: Physical and Numerical Model Case Study. Journal of Hydraulic Engineering 127, 640–649. https://doi.org/10.1061/(asce)0733-9429(2001)127:8(640)

Simons, T.J., 1974. Verification of Numerical Models of Lake Ontario. J Phys Oceanogr Volume 4, 507–523.

Suprapto, M., 2013. Increase spillway capacity using labyrinth weir, in: Procedia Engineering. Elsevier Ltd, pp. 440–446. https://doi.org/10.1016/j.proeng.2013.03.039

Tominaga, Y., Stathopoulos, T., 2011. CFD modeling of pollution dispersion in a street canyon: Comparison between LES and RANS. Journal of Wind Engineering and Industrial Aerodynamics 99, 340–348. https://doi.org/10.1016/j.jweia.2010.12.005

Versteeg, H.K., Malalasekera, W., 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Second Edition. ed. Pearson Education Limited, Edinburgh Gate.

Yakhot, V., Orszag, S.A., 1986. Renormalization Group Analysis of Turbulence I. Basic Theory. J Sci Comput Volume 1.

Yalcin, E.E., Ikinciogullari, E., Kaya, N., 2023. Comparison of Turbulence Methods for a Stepped Spillway Using Computational Fluid Dynamics. Iranian Journal of Science and Technology, Transactions of Civil Engineering 47, 3895–3911. https://doi.org/10.1007/s40996-023-01127-5