MedCrave
Step into the World of Research

International Journal of Hydrology

Review Article

Open Access

CrossMark

Applications of computational fluid dynamics (CFD) in the design and analysis of hydraulic structures

Abstract

Hydraulic structures are engineering buildings constructed for the purpose of storing, transmitting, controlling, and converting water into energy. Their importance stems from their critical function in ensuring water security, sustainable water use, energy supply, and disaster risk reduction for society. Computational Fluid Dynamics (CFD) has become an indispensable tool in the design and analysis of hydraulic structures, offering a numerical alternative or complement of physical modelling and empirical approaches. The design, operation, and safety of hydraulic structures necessitate the accurate analysis of complex hydrodynamic processes. Owing to their high cost and time requirements, traditional physical modeling methods are increasingly being superseded by CFD techniques. This study reviews recent scientific investigations employing the OpenFOAM, Flow-3D and ANSYS Fluent software packages, with a view to assessing their efficacy, comparative advantages and modelling approaches in the analysis of hydraulic structures. Within the scope of literature review, basin management employing the SWAT model, the hydraulic mechanisms of historical water conveyance channels, and innovative coastal protection units are also incorporated into the assessment. The findings indicate that CFD simulations exhibit a high degree of correlation with experimental data and serve as a critical decision-support tool in complex flow problems.

Keywords: (CFD), hydraulic structures, OpenFOAM, FLOW-3D, ANSYS fluent, numerical modeling

Volume 10 Issue 4 - 2026

Zeyneb KILIÇ
Department of Civil Engineering, Adıyaman University, Türkiye

Correspondence: Zeyneb KILIÇ, Department of Civil Engineering, Engineering Faculty, Adıyaman, Adıyaman University, Türkiye

Received: November 13, 2026 | **Published:** November 24, 2026

Introduction

Hydraulic structures fulfill socially critical functions such as water security, energy production, agricultural sustainability, and disaster risk reduction. The safe, efficient, and sustainable design of these structures necessitates not only engineering practices but also scientific research and advanced analytical methods. Hydraulic structure engineering investigates the behavior of systems—including dams, spillways, bridge piers, and coastal structures—under the dynamic influence of water. The numerical modeling of phenomena encountered in the design of these structures, such as cavitation, local scour, turbulent flow, air entrainment, and energy dissipation, is of vital importance for preserving structural integrity and ensuring operational safety. Numeric simulation became an important tool in hydraulic engineering and is a vital extension to a hydraulic scale model test.[1]

Although traditional methods rely on laboratory-scale physical models, scale effects and high operational costs impose limitations on these processes. In the contemporary context, with advancing computational technologies, modern numerical fluid dynamics solvers such as OpenFOAM, Flow-3D, and ANSYS Fluent have emerged as the most effective tools in the field of hydraulic structures, offering optimized solutions to engineering problems. Numerical methods in the field of CFD are a very powerful and important tool for hydraulics and hydraulic engineering. Various academic and commercial codes are available. Each of codes have its special application area and advantages. In the studies, the results of the numeric simulations generally match very well with the results of the physical scale model test. The next step will be further investigation and to prefer the most accurate simulations by analysing the hydraulic structures to achieve required improvements in the process.[2]

CFD provides a numerical approximation to solve the governing equation of fluid movement. CFD has become one of the most important simulation methods in classical and modern engineering apllications. Instead of relying only on physical prototypes and laboratory testing, engineers can use CFD simulation to predict how liquids, gases, heat and particles behave inside or around a product, machine or industrial process.[3]

The aim of this study is to examine modern CFD solvers used in the numerical modeling of complex hydraulic phenomena such as cavitation, local erosion, turbulent flow, air entrainment, and energy dissipation encountered in the design and analysis processes of water structures such as dams, spillways, bridge abutments, and shore structures. In this context, the study aims to comparatively evaluate OpenFOAM, FLOW-3D, and ANSYS Fluent, which are among the most widely used open-source and commercial software programs today, in terms of their fundamental features, numerical methods, turbulence models, free surface tracking approaches, and application areas. Furthermore, the study aims to highlight the advantages offered by these software programs over the limitations of traditional physical model experiments, such as scale effects and high costs, and to systematically present the current state and future potential of numerical modeling in water structure engineering in light of a literature review.

Material and methods

This study was conducted in accordance with the principles of a systematic comprehensive literature review, related to studied issue. After gathering the relevant literature, evaluations were made using sections related to the importance of the subject. In conclusion, this study is expected to contribute to the relevant literature and to similar studies to be conducted on this subject.

Submit Manuscript | http://medcraveonline.com

©2026 KILIÇ. This is an open access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and build upon your work non-commercially.

Copyright: ©2026 KILIÇ.

Software-based lıterature assessment

Flow-3D applications

The Flow-3D software, by virtue of the "Volume of Fluid" (VOF) method employed for the high-accuracy modeling of free-surface and time-dependent (transient) flows, has acquired a wide array of applications in hydraulic structure projects.

Spillway and Cavitation Analysis: Aşırok[4] investigated the cavitation potential of the Güllübağ Dam spillway model, validating the software's proficiency in modeling pressure and velocity distributions against experimental measurements.[4] Similarly, Demeke et al.[5] employed three-dimensional (3D) hydrodynamic modeling to optimize the Tendaho Dam spillway in Ethiopia, recommending a lowering of the stilling basin floor elevation that reduced flow velocity from 20 m/s to 6 m/s to prevent channel overflows.[5] Furthermore, the flow and turbulence characteristics within the voids of spillway gate plates constitute a critical area of analysis for predicting flow safety.[6]

Vortex and Sediment Transport: Evranos[7] modeled air-entraining vortices at water intake structures, analyzing the effectiveness of anti-vortex devices in preventing energy losses.[7] Kelleci[8] examined sediment separation efficiency and flow characteristics in vortex-type settling basins, reporting that the k-ω turbulence model; so it is exhibited high performance in capturing the settling dynamics of coarse-grained sediments.[8]

Bridge and Coastal Structures: Shikh Hasan[9] numerically simulated soil–flow interaction by investigating local scour around circular bridge piers arranged in a triangular configuration.[9] Aksoy,[10] focusing on flow–soil–structure interactions in coastal regions, emphasized Flow-3D's competence in the pre-processing stage while highlighting its limitations in data visualization.[10] Within the scope of probabilistic design of innovative deep-sea coastal structures, the domestically produced "Piblok+" coastal protection unit was verified through Flow-3D simulations as offering high stability and economic advantages under challenging marine conditions.[11]

ANSYS fluent applications

ANSYS Fluent is recognized for its industry-accepted broad spectrum of turbulence models, advanced mesh flexibility, and robust solver algorithms.

Comparative Analyses: In a study conducted by Rajaa and Kamel,[12] the flow over the ogee-type weir of the Mandali Dam was modeled using both ANSYS Fluent and Flow-3D.[12] It was concluded that both software packages achieved agreement exceeding 90% with experimental findings regarding free-surface profile and velocity distribution, thereby serving as mutual calibration and validation tools in engineering processes.

Hydraulic Jumps: Viti et al.[13] reported that Reynolds-Averaged Navier–Stokes (RANS) models could predict mean flow variables in hydraulic jump simulations with accuracy exceeding 90%, demonstrating that software such as Fluent offers high reliability for engineering designs.[13]

OpenFOAM applications

Owing to its open-source and free of charge, OpenFOAM is preferentially adopted, particularly within academic environments, for the development of novel algorithms, the customization of libraries, and licensing cost efficiency.

Turbulence and Algorithm Development: Shahriari et al.,[14] developed a custom solver to accelerate Large Eddy Simulations (LES) in water engineering, achieving a 40% increase in computational speed.[14] This study demonstrated that complex flow regimes at the intake structures of river-type hydropower plants can be successfully modeled using IDDES and SST turbulence models.

Mesh Optimization in Complex Structures: Grivalszki[15] demonstrated that, in three-dimensional flows around complex hydraulic structures, the k-ω SST model minimizes erroneous turbulence production in the vicinity of the free water surface, and revealed the critical role of the automatic cell generation tool, snappyHexMesh, in simulation accuracy.[15]

Basin-scale analyses and historical structures

In addition to micro-scale CFD analyses, the macro-level sustainability of water resources and hydraulic structures is also investigated through numerical models. In the study conducted by Çolak[16] the hydrological processes of the Upper Tigris Sub-Basin were examined using the SWAT (Soil and Water Assessment Tool) model, yielding high predictive performance in estimating precipitation–runoff relationships and in basin-based water management optimization.[16]

Conversely, hydraulic analyses conducted to comprehend the engineering philosophy of historical water structures enable the deciphering of past mechanisms. The case of the Historical Ferhat Water Channel, examined by Kurşun[17] revealed the structure's mechanism for maintaining resistance to sediment accumulation over time. The hydraulic analyses determined that the channel was designed with a distinctive turbulent flow regime that inherently prevents sediment deposition, reflecting the sophistication of advanced Roman engineering.[17] Recent research on hydrodynamic performance enhancement and flow characteristics in open channels also clearly demonstrates the effects of spur dike placement configurations on energy dissipation.[18]

Comparatıve assessment and modelıng approaches

In light of the literature data obtained, the selection of software for hydraulic structure projects is directly contingent upon the nature of the physical problem under consideration, the geometric complexity, and the computational budget. In the tables below, the capabilities of the numerical models and the principal findings derived from the case analyses are systematically compared Table 1. Regarding modeling approaches, RANS equations retain their predominance in industrial designs due to the balance between speed and accuracy, whereas advanced methods such as LES are observed to be preferred for more detailed turbulence analysis and for resolving complex phenomena such as air entrainment.[13] Water is one of the most essential needs for life; therefore, water conservation, water structures and all water-related activities are very important Table 2.[19,20]

Citation: KILIÇ Z. Applications of computational fluid dynamics (CFD) ın the design and analysis of hydraulic structures. *Int J Hydro.* 2026;10(4):136–139. DOI: 10.15406/ijh.2026.10.00438

Copyright: ©2026 KILIÇ.

Table 1 Numerical Models Employed in the Design of Hydraulic Structures and Their Capabilities

Software / Model	Primary Hydraulic Application Area	Solution Capability / Highlights Feature	Accuracy / Agreement Ratio
FLOW-3D	Spillways, Cavitation, Free-Surface Flows, Coastal Structures	Free-surface profiles, high-velocity flow optimization, cavitation risk prediction	High correlation exceeding 90% with experimental data
OpenFOAM	Complex Hydraulic Structures, 3D Flow Dynamics	Open-source CFD simulations, flexible turbulence modeling	High agreement with experimental measurements and verifiability
ANSYS Fluent	Ogee-Type Weirs, General Flow Mechanisms	Precise prediction of velocity distributions and pressure variations	Experimental agreement exceeding 90% in Ogee Weir designs
SWAT Model	Large-Scale Basin Management, Hydrological Processes	Prediction of precipitation–runoff relationships, basin-based water management	High predictive performance in large-scale analyses

Table 2 Numerical Approaches and Engineering Findings in the Resolution of Different Hydraulic Problems

Examined Structure	Numerical Approach Employed	Basic Engineering Problem	Obtained Numerical Finding and Modification
Güllüba□ Dam Spillway	FLOW-3D Simulation	Cavitation potential and structural safety risk	Cavitation risk analyzed numerically; structural solution proposals developed
Tendaho Dam Spillway	3D Hydrodynamic Modeling	Channel overflows and uncontrolled high flow velocities	Flow velocity reduced from 20 m/s to 6 m/s; modifications proposed to prevent channel overflows
Ogee-Type Weirs	ANSYS Fluent & FLOW-3D	Agreement between experimental models and numerical data	Numerical models found to achieve full agreement exceeding 90% with experimental findings
Water Intake Structures	CFD and k-ω Turbulence Model	Air-entraining vortices causing energy losses and sediment accumulation	Efficacy of anti-vortex devices demonstrated; sediment separation efficiency optimized through basin geometry
Piblok+ Coastal Protection Unit	FLOW-3D Modeling	Wave–structure interaction and stability in deep-sea structures	Domestically produced units confirmed to offer both economic and structural stability advantages under challenging marine conditions
Upper Dicle Sub-Basin	SWAT Model	Prediction of hydrological processes over large areas	High success in predicting precipitation–runoff relationships and water management optimization
Historical Ferhat Water Channel	Engineering and Hydraulic Analysis	Structure's mechanism of resistance to sediment accumulation over time	Channel determined to be designed with a turbulent flow regime preventing sediment deposition, reflecting advanced Roman engineering

Conclusion

This review study has examined the applications of CFD in the design and analysis of hydraulic structures through the solvers OpenFOAM, FLOW-3D, and ANSYS Fluent. The findings demonstrate that CFD has become a powerful alternative and complementary tool to physical modeling in the present day. FLOW-3D offers superior VOF performance in free-surface flows; ANSYS Fluent provides a comprehensive turbulence and multiphase model library; and OpenFOAM offers flexibility and customization advantages through its open-source structure.

It has been emphasized that the selection of a solver should be based on problem characteristics, computational resources, and the desired level of accuracy. Limitations of physical models, such as scale effects and high costs, can be effectively overcome through numerical methods. Nevertheless, challenges persist in areas including mesh resolution, turbulence model selection, and the modeling of multiphase phenomena such as air entrainment.

CFD software in the field of hydraulic structures constitutes not merely a visualization or validation tool, but also a powerful engineering platform wherein critical design parameters are optimized. Recent studies generally demonstrate that, numerical models have been successfully validated against experimental data in technical domains such as the prevention of cavitation damage, sediment transport control, and energy dissipation in open channels. Especially in cases involving complex 3D flow dynamics, traditional 1D models prove insufficient in their predictive capabilities, making 3D simulations an essential requirement for structural safety, operational sustainability, and project economics. In the future, the integration of artificial intelligence and high-performance computing technologies is expected to further advance CFD applications. In conclusion, all three solvers have become indispensable tools in hydraulic engineering, and the combined use of numerical and physical methods offers the most reliable approach.

Citation: KILIÇ Z. Applications of computational fluid dynamics (CFD) ın the design and analysis of hydraulic structures. *Int J Hydro.* 2026;10(4):136–139. DOI: 10.15406/ijh.2026.10.00438

Copyright: ©2026 KILIÇ.

Acknowledgements

None.

Conflicts of interest

Authors declare no conflict of interest exists.

Funding

None.

References

1. Kılıç Z. Bibliometric analysis of computational fluid dynamics (CFD) on hydraulic energy. *Black Sea J Eng Sci*. 2025;8(3):598–604.
2. Gabl R, Gems B, Plorer M, et al. Numerical simulations in hydraulic engineering. In: *[Book Title Missing]*; 2015:chap 8.
3. Javadi A. *Applications of Computational Fluid Dynamics in Engineering*. World Academies; 2026.
4. Aşırok E. *Güllübağ Barajı Dolusavak Modeli Üzerindeki Kavitasyon Potansiyelinin İncelenmesi* [master's thesis]. Fen Bilimleri Enstitüsü; 2019.
5. Demeke GK, Asfaw DH, Shiferaw YS. Three-dimensional hydrodynamic modeling to improve the design of Tendaho Dam spillway, Ethiopia. *Water*. 2019;11(1):82.
6. Özün MF. *Dolu Savak Plaka Boşluklarındaki Akım ve Türbülans Karakteristiklerinin Deneysel Olarak İncelenmesi* [master's thesis]. İstanbul Teknik Üniversitesi; 2019.
7. Evranos OB. *Su Alma Yapılarında Vorteks Oluşumunun Sayısal Olarak İncelenmesi ve Anti-Vorteks Düzeneklerin Etkinliği* [master's thesis]. İzmir: Dokuz Eylül Üniversitesi, Fen Bilimleri Enstitüsü; 2018.
8. Kelleci A. *Vorteks Tipi Çökeltim Havuzlarında Akım Karakteristiklerinin ve Sediment Verimliliğinin Sayısal Olarak İncelenmesi* [master's thesis]. Fen Bilimleri Enstitüsü; 2021.
9. Shikh Hasan OR. *Numerical Simulation of Scour Around a Group of Circular Bridge Piers Arranged in a Triangular Pattern* [master's thesis]. Gaziantep Üniversitesi; 2018.
10. Aksoy E. *Kıyı Bölgelerindeki Akım-Zemin-Yapı Etkileşimlerinin Sayısal Modellenmesi* [master's thesis]. 2017.
11. Uğurlu A. *Derin Deniz Kıyı Yapılarının Olasılıksal Tasarımı ve Yenilikçi Bir Koruma Ünitesi Geliştirilmesi: Piblok+ Örneği* [dissertation]. Gazi Üniversitesi; 2023.
12. Rajaa AI, Kamel AH. Comparison of numerical simulation of flow over ogee weir using ANSYS Fluent and FLOW-3D. *[Journal/Conference Name Missing]*. 2021.
13. Viti N, Valero D, Gualtieri C. Numerical simulation of hydraulic jumps. Part 1: experimental data for validation and review of existing numerical studies. *Environ Fluid Mech*. 2019;11(1):36.
14. Shahriari S, Schneider J, Zenz G. Application of OpenFOAM in water engineering: from algorithm development to hydraulic structure analysis. *Österr Wasser Abfallwirtsch*. 2020;72:291–307.
15. Grivalszki P, Fleit G, Baranya S, et al. Assessment of CFD model performance for flows around a hydraulic structure of complex geometry. *Period Polytech Civ Eng*. 2021;65(1).
16. Çolak YE. *Yukarı Dicle Alt Havzası Hidrolojik Süreçlerinin SWAT Modeli ile İncelenmesi* [master's thesis]. Fen Bilimleri Enstitüsü; 2025.
17. Kurşun G. *Tarihi Su Yapılarının Malzeme ve Hidrolik Açısından İncelenmesi: Ferhat Su Kanalı Örneği* [master's thesis]. Fen Bilimleri Enstitüsü; 2020.
18. Ulu AE. *Açık Kanallarda Mahmuz Yerleşim Düzenlerinin Akım Karakteristikleri ve Enerji Sönümü Üzerine Etkilerinin İncelenmesi* [dissertation]. Dicle Üniversitesi; 2023.
19. Kılıç Z. The importance of water and conscious use of water. *Int J Hydrol*. 2020;4(5):239–241.
20. Kılıç Z. Water pollution causes, negative effects and prevent methods. *İstanbul Sabahattin Zaim Üniv Fen Bilim Enst Derg*. 2021;3(2):32–35.