

Computational Investigation of the Flow Characteristics in a Ski Jump Bucket Energy Dissipator Using FLOW-3D

Mr. Yash Garud ¹

¹M.Tech. Civil & Environmental Technology,
Department of Technology,
Savitribai Phule Pune University,
Pune, Maharashtra, India

Dr. V.S.Chavhan ²

²Assistant Professor,
Department of Civil Engineering,
AISSMS College of Engineering,
Pune, Maharashtra, India

Abstract - The hydraulic performance of spillway energy dissipators is a critical consideration in the design and safety of high-head dams, where inadequate energy dissipation can result in severe downstream scour and structural instability. Ski-jump bucket energy dissipators are widely employed to redirect high-velocity flows away from the dam toe and reduce the erosive effects of discharged water. The present study investigates the flow characteristics and hydraulic performance of a ski-jump bucket energy dissipator using Computational Fluid Dynamics (CFD). A three-dimensional numerical model of the spillway was developed in AutoCAD and imported into FLOW-3D for simulation. Free-surface flow was modelled using the Volume of Fluid (VOF) method, while turbulence effects were represented using the Renormalization Group (RNG) $k-\epsilon$ turbulence model. Simulations were performed for multiple discharge conditions by applying appropriate boundary conditions, material properties, and numerical parameters. The hydraulic behaviour of the flow was evaluated in terms of velocity distribution, pressure distribution, fluid fraction, jet trajectory, and energy dissipation characteristics. Numerical predictions were validated against corresponding experimental observations, demonstrating good agreement with percentage errors within acceptable engineering limits. The results indicate that increasing discharge significantly influences flow velocity, jet trajectory length, and overall hydraulic performance of the ski-jump bucket. Furthermore, the CFD model successfully reproduced the free-surface flow behaviour and provided detailed visualization of complex hydraulic phenomena that are difficult to obtain through conventional physical modelling. The study demonstrates that FLOW-3D is a reliable and efficient tool for analysing spillway hydraulics and can serve as a cost-effective alternative to experimental investigations for the design, evaluation, and optimization of ski-jump bucket energy dissipators.

Keywords - Computational Fluid Dynamics (CFD); FLOW-3D; Ski-Jump Bucket; Spillway Hydraulics; Energy Dissipation; Free-Surface Flow.

I. INTRODUCTION

Spillways are among the most critical hydraulic structures in dam engineering, providing a safe means of releasing excess reservoir water during flood events while protecting the structural integrity of the dam. Water flowing over spillways possesses a high level of kinetic energy due to the conversion of potential energy, and if this energy is not adequately dissipated, severe downstream erosion, scour, and structural damage may occur. Consequently, the design of efficient

energy dissipation systems is an essential aspect of hydraulic engineering, directly influencing the safety, durability, and operational performance of dams.

Ski-jump bucket energy dissipators are extensively employed in high-head dams because they project the high-velocity water jet away from the dam toe into a downstream plunge pool, thereby reducing the erosive effects of the discharged flow. The hydraulic performance of a ski-jump bucket depends on several parameters, including discharge, bucket geometry, approach flow conditions, tailwater depth, jet trajectory, and flow velocity. These parameters govern the effectiveness of energy dissipation and influence the hydraulic behaviour of the downstream flow. Therefore, understanding the flow characteristics of ski-jump bucket spillways is essential for the design and optimization of safe and efficient hydraulic structures.

Traditionally, hydraulic performance has been evaluated through physical model studies conducted in laboratory environments. Although physical modelling provides reliable experimental data, it is often associated with high construction costs, extended testing durations, scaling limitations, and restricted measurement capabilities. In addition, obtaining detailed information regarding velocity fields, pressure distributions, turbulence characteristics, and free-surface behaviour through experimental investigations alone is often difficult. These limitations have encouraged the increasing adoption of numerical modelling techniques for analysing complex hydraulic flow phenomena.

Computational Fluid Dynamics (CFD) has emerged as an effective tool for investigating free-surface flows in hydraulic engineering. By numerically solving the governing equations of fluid motion, CFD enables detailed visualization and quantitative analysis of flow behaviour within complex hydraulic structures. Modern CFD software allows engineers to evaluate velocity distributions, pressure variations, turbulence characteristics, jet trajectories, and energy dissipation mechanisms with high spatial resolution while significantly reducing the cost and time associated with experimental studies.

Among the available CFD platforms, FLOW-3D has gained widespread acceptance for modelling free-surface hydraulic flows because of its robust numerical algorithms and

specialized capabilities for simulating complex water–air interactions. The software employs the Volume of Fluid (VOF) method to accurately capture free-surface interfaces and offers several turbulence models suitable for hydraulic applications. In particular, the Renormalization Group (RNG) $k-\epsilon$ turbulence model has demonstrated improved performance in predicting rapidly varying turbulent flows commonly encountered in spillways and energy dissipators. These capabilities make FLOW-3D a suitable numerical tool for analysing the hydraulic behaviour of ski-jump bucket energy dissipators.

Several researchers have investigated spillway hydraulics using both experimental and numerical approaches. Previous studies have demonstrated that CFD can successfully reproduce free-surface flow characteristics and provide satisfactory agreement with laboratory observations. Numerical investigations have reported accurate prediction of velocity fields, pressure distributions, and flow patterns for ogee spillways and related hydraulic structures. However, comparatively fewer studies have focused on the detailed hydraulic performance of ski-jump bucket energy dissipators under varying discharge conditions using validated three-dimensional CFD models. Furthermore, comprehensive analyses relating discharge variation to jet trajectory, hydraulic characteristics, and energy dissipation remain limited.

The present study addresses this gap by performing a three-dimensional CFD investigation of the flow characteristics of a ski-jump bucket energy dissipator using FLOW-3D. A numerical model of the spillway geometry was developed and simulated under multiple discharge conditions. The free-surface flow was modelled using the Volume of Fluid (VOF) technique, while turbulence effects were represented using the RNG $k-\epsilon$ turbulence model. The hydraulic behaviour of the flow was evaluated through analysis of velocity distribution, pressure distribution, fluid fraction, jet trajectory, and energy dissipation characteristics. The numerical predictions were subsequently compared with corresponding experimental observations to assess the accuracy and reliability of the CFD model.

The findings of this study demonstrate the applicability of CFD as an efficient and reliable approach for analysing spillway hydraulics and provide valuable insights into the hydraulic performance of ski-jump bucket energy dissipators. The results contribute to the understanding of free-surface flow behaviour and support the use of numerical modelling as a practical alternative to conventional physical model investigations for the design and optimization of hydraulic structures.

II. LITERATURE REVIEW

Computational Fluid Dynamics (CFD) has become an indispensable tool for investigating hydraulic flow behaviour in spillways, energy dissipators, and other water-retaining structures. The continuous advancement in numerical methods, turbulence modelling, and computational resources has significantly enhanced the capability of CFD to simulate complex free-surface flows with high accuracy. Compared with conventional physical model studies, CFD offers detailed visualization of velocity fields, pressure distributions, turbulence characteristics, and flow trajectories while reducing the time and cost associated with experimental investigations.

Several researchers have successfully employed CFD techniques to analyse flow over spillways and validate numerical predictions against laboratory observations. Previous studies have demonstrated that numerical models can accurately reproduce discharge characteristics, velocity distributions, pressure variations, and free-surface profiles under different hydraulic conditions. The application of Reynolds-Averaged Navier–Stokes (RANS) equations coupled with the Volume of Fluid (VOF) method has been widely accepted for modelling air–water interfaces in spillway flows, providing satisfactory agreement with experimental measurements.

Among the various CFD software packages available for hydraulic simulations, FLOW-3D has gained widespread recognition because of its specialized capabilities for modelling free-surface flows. The software incorporates the TruVOF technique for interface tracking and provides several turbulence models suitable for hydraulic engineering applications. Its ability to simulate transient flow behaviour, jet formation, hydraulic jumps, and complex air–water interactions has resulted in its extensive use in spillway design, river hydraulics, dam engineering, and energy dissipation studies.

Previous numerical investigations have evaluated the hydraulic performance of ogee spillways, gated spillways, side-channel spillways, and energy dissipators using FLOW-3D and other CFD platforms. These studies have reported good agreement between numerical predictions and laboratory experiments, demonstrating that CFD can reliably estimate hydraulic parameters such as flow velocity, pressure distribution, water surface profiles, discharge capacity, and energy dissipation. The incorporation of free-surface tracking techniques and advanced turbulence models has further improved the predictive capability of numerical simulations for rapidly varying hydraulic flows.

The selection of an appropriate turbulence model plays a significant role in determining the accuracy of CFD simulations. Among the commonly employed turbulence models, the Renormalization Group (RNG) $k-\epsilon$ model has been widely adopted for spillway analyses because it provides improved prediction of rapidly strained and highly turbulent flows compared with the standard $k-\epsilon$ model. Several comparative studies have concluded that the RNG $k-\epsilon$ model offers better agreement with experimental observations for spillway hydraulics, particularly in regions characterised by strong turbulence, jet impingement, and free-surface deformation.

Despite the substantial progress achieved in numerical modelling of spillway hydraulics, comparatively fewer investigations have focused specifically on the hydraulic behaviour of ski-jump bucket energy dissipators under varying discharge conditions using validated three-dimensional CFD models. Most published studies have primarily concentrated on ogee spillways or generalized spillway configurations, while detailed analyses of jet trajectory characteristics, pressure distribution, velocity variation, and energy dissipation performance in ski-jump bucket systems remain limited. Furthermore, several existing investigations evaluate only individual operating conditions without presenting a comprehensive assessment of hydraulic performance over a range of discharge conditions.

The present study addresses this research gap by developing a three-dimensional CFD model of a ski-jump bucket energy dissipator using FLOW-3D. Free-surface flow is simulated using the Volume of Fluid (VOF) method, while turbulence is represented by the RNG $k-\epsilon$ model. Numerical simulations are performed for multiple discharge conditions, and the hydraulic behaviour is evaluated through velocity distribution, pressure distribution, fluid fraction, jet trajectory, and energy dissipation characteristics. The numerical predictions are validated against corresponding experimental observations to assess the reliability of the computational model. The findings contribute to the growing application of CFD in hydraulic engineering and demonstrate the capability of FLOW-3D as an effective tool for the analysis and optimization of spillway energy dissipators.

III. NUMERICAL METHODOLOGY

A. Computational Model

A three-dimensional Computational Fluid Dynamics (CFD) model was developed to investigate the hydraulic behaviour of flow over a ski-jump bucket energy dissipator. The numerical simulations were performed using FLOW-3D, a CFD software specifically designed for modelling free-surface hydraulic flows. The software solves the governing equations of fluid motion using the finite volume method and accurately captures complex air-water interfaces through the Volume of Fluid (VOF) technique.

The numerical model was developed to simulate free-surface flow over the spillway for multiple discharge conditions and to evaluate important hydraulic parameters, including velocity distribution, pressure distribution, jet trajectory, and energy dissipation.

B. Geometry Development & Mesh

The geometric model of the spillway and ski-jump bucket energy dissipator was prepared using AutoCAD based on the dimensions of the experimental model. Initially, a two-dimensional profile of the spillway was created and subsequently extruded to generate a three-dimensional model representing the spillway geometry. The completed model was exported in STL (Stereolithography) format and imported into FLOW-3D for numerical analysis.

The imported geometry is positioned within the computational domain to ensure sufficient upstream and downstream flow regions for accurate simulation of the free-surface flow.

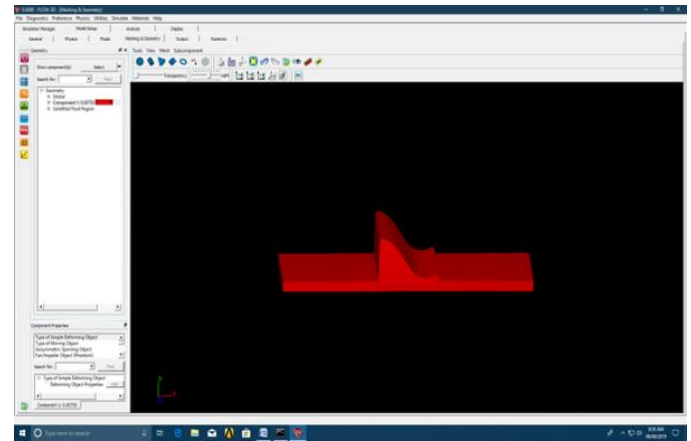


Figure 1. Three-dimensional geometry of the ski-jump bucket spillway.

The computational domain was discretized using a structured Cartesian mesh generated within FLOW-3D. A mesh block enclosing the complete spillway geometry was created using the built-in meshing tools provided by the software.

A mesh cell size of approximately 0.01 m was adopted to provide a suitable balance between computational accuracy and simulation time while adequately resolving the hydraulic features of the flow. The computational domain was extended above the spillway to accommodate the free-surface jet generated by the ski-jump bucket.

The generated mesh was verified prior to simulation to ensure complete coverage of the computational domain and appropriate representation of the spillway geometry.

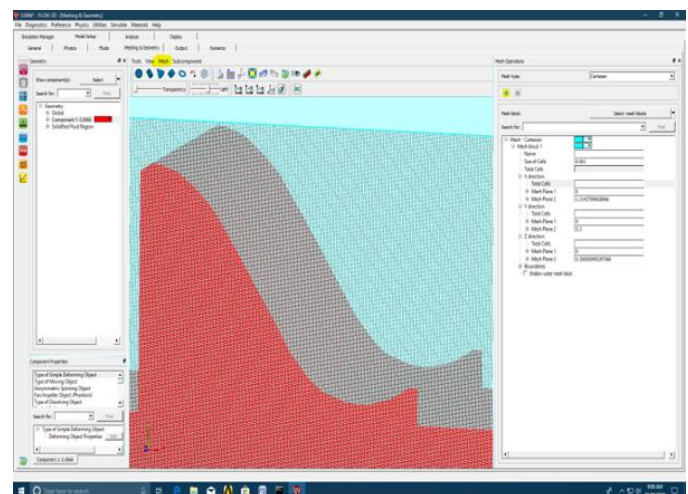


Figure 2. Computational mesh used for numerical simulation.

C. Numerical Model

The interface between water and air was simulated using the Volume of Fluid (VOF) method. This technique tracks the volume fraction of water within each computational cell and enables accurate prediction of rapidly varying free-surface flows.

The VOF approach is particularly suitable for spillway hydraulics because it effectively captures jet formation, splash regions, hydraulic jumps, and air–water interaction without requiring explicit tracking of the fluid interface.

Turbulence effects were represented using the Renormalization Group (RNG) $k-\epsilon$ turbulence model. This model was selected because of its improved capability to predict rapidly strained and highly turbulent flows commonly encountered in spillway hydraulics.

Compared with the standard $k-\epsilon$ model, the RNG formulation incorporates additional terms that improve the prediction of turbulence intensity, streamline curvature, and recirculating flow regions. Consequently, it has been widely adopted in numerical investigations of spillways, energy dissipators, and hydraulic structures.

D. Boundary Conditions

The inlet boundary was specified using a **velocity inlet** corresponding to the discharge under investigation. Separate simulations were carried out for each discharge condition. A **pressure outlet** boundary condition was assigned at the downstream boundary to allow unrestricted outflow from the computational domain.

Water at a temperature of **293 K** was selected as the working fluid, and gravitational acceleration of **9.81 m/s²** was applied in the negative vertical direction. Appropriate initial water elevations were prescribed to establish realistic free-surface conditions before the commencement of each simulation.

Table 1. Boundary conditions in the CFD simulations.

Boundary	Specification
Inlet	Specified Velocity
Outlet	Specified Pressure
Fluid	Water
Temperature	293K
Gravity	9.81m/s

E. Simulation Parameters

The numerical simulations were performed under transient flow conditions using the default numerical schemes available in FLOW-3D. **The total simulation time was 20 s**, which was sufficient to establish stable hydraulic conditions throughout the computational domain.

The simulation outputs included velocity distribution, pressure distribution, fluid fraction, total head, particle trajectories, and free-surface profiles. These variables were subsequently used to evaluate the hydraulic performance of the ski-jump bucket energy dissipator.

The principal numerical parameters employed in the present investigation are summarized in Table 2.

Table 2. Numerical simulation parameters.

Parameter	Value
Software	FLOW-3D
Flow Type	Transient
Solver	Finite Volume Method

Parameter	Value
Turbulence Model	RNG $k-\epsilon$
Free Surface Method	Volume of Fluid (VOF)
Working Fluid	Water
Simulation Time	20s
Temperature	293 K

F. Post-processing

The numerical results were analysed using the post-processing modules available in FLOW-3D. Velocity contours, pressure contours, fluid fraction distributions, and free-surface profiles were generated to evaluate the hydraulic characteristics of the flow.

The jet trajectory and hydraulic behaviour downstream of the spillway were determined using sectional visualization tools, while quantitative parameters including trajectory length and energy dissipation were extracted for comparison with the corresponding experimental observations. This comparison was used to assess the accuracy and reliability of the numerical model.

IV. MODEL VALIDATION

Validation of the Numerical Model

Validation of the numerical model is an essential step in assessing the reliability of Computational Fluid Dynamics (CFD) simulations. In the present study, the numerical predictions obtained from FLOW-3D were validated by comparing the computed energy dissipation values with the corresponding experimental observations available for identical hydraulic conditions.

The validation was carried out for four representative discharge conditions covering the range of flow rates considered in the investigation. The same spillway geometry, hydraulic conditions, boundary conditions, and operating parameters were adopted in both the numerical and experimental analyses to ensure a consistent basis for comparison.

The comparison between the CFD predictions and experimental observations is presented in Table 3.

Table 3. Comparison between CFD and Experimental Energy Dissipation

Discharge (m ³ /s)	CFD Energy Dissipation (%)	Experimental Energy Dissipation (%)	Percentage Error (%)
0.00962	54.47	48.45	15.69
0.00735	41.13	46.61	11.75
0.00595	55.72	53.67	3.69
0.00431	55.22	52.01	5.81

The comparison demonstrates good agreement between the numerical predictions and the corresponding experimental results. The percentage error varied from **3.69% to 15.69%**, indicating that the CFD model was capable of reproducing the hydraulic behaviour of the ski-jump bucket energy dissipator with acceptable engineering accuracy.

The smallest deviation was observed for the discharge of **0.00595 m³/s**, where the numerical prediction differed from the experimental value by only **3.69%**. The maximum deviation of

15.69% occurred at the highest discharge condition of 0.00962 m³/s. This difference may be attributed to numerical discretization, turbulence modelling assumptions, mesh resolution, and uncertainties associated with experimental measurements.

Overall, the validation results confirm that the **FLOW-3D** numerical model provides reliable predictions of the hydraulic performance of the ski-jump bucket energy dissipator. The observed level of agreement between the CFD simulations and experimental observations demonstrates the suitability of the adopted numerical methodology for investigating free-surface flow characteristics and energy dissipation in spillway structures.

To further illustrate the agreement between the numerical and experimental results, a comparison plot of CFD and experimental energy dissipation values is presented in Figure 3. The close correlation between the two datasets indicates that the developed numerical model accurately captures the principal hydraulic characteristics of the flow and can be confidently employed for the analysis of ski-jump bucket energy dissipators.

V. RESULTS AND DISCUSSION

The CFD simulations were performed to evaluate the hydraulic behaviour of flow over the ski-jump bucket energy dissipator under different discharge conditions. The numerical results obtained from FLOW-3D were analysed in terms of velocity distribution, pressure distribution, free-surface flow characteristics, jet trajectory, and energy dissipation. The simulations successfully captured the complex hydraulic behaviour associated with free-surface flow over the spillway and provided detailed visualization of the flow field. The numerical predictions were also compared with experimental observations to assess the accuracy and reliability of the computational model.

A. Velocity Distribution

The velocity contours obtained from the numerical simulations indicate a continuous increase in flow velocity along the spillway profile as water accelerates under the influence of gravity. The highest velocities were observed near the lip of the ski-jump bucket, where the potential energy of the reservoir water was converted into kinetic energy before the flow was projected downstream.

As the discharge increased, the velocity magnitude within the spillway also increased, resulting in greater jet momentum and longer downstream trajectories. The CFD simulations successfully captured the acceleration of the flow and clearly illustrated the formation of the free jet after leaving the bucket. The velocity contours also demonstrated smooth flow development without significant numerical instability, indicating the effectiveness of the adopted numerical model.

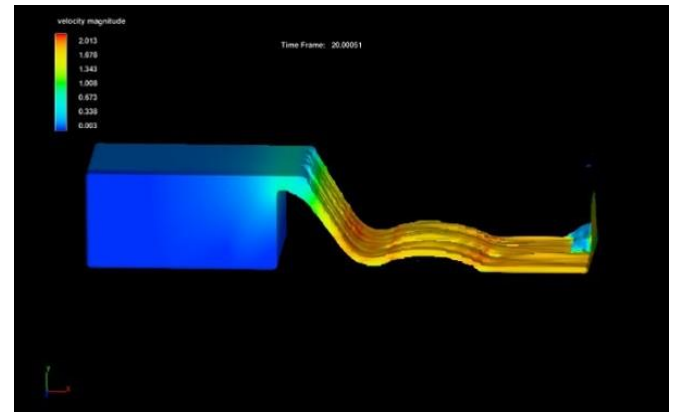


Figure 3. Velocity magnitude contour for the ski-jump bucket spillway.

The velocity contour demonstrates the gradual acceleration of flow along the spillway profile due to the conversion of potential energy into kinetic energy. The highest velocity values occur near the lip of the ski-jump bucket, where the flow is projected downstream as a free jet. The contour also indicates a smooth velocity distribution without abrupt fluctuations, confirming the numerical stability of the simulation. As the discharge increases, the flow possesses greater momentum, resulting in higher exit velocities and longer jet trajectories. These observations are consistent with the expected hydraulic behaviour of ski-jump bucket energy dissipators.

B. Pressure Distribution

The pressure contours revealed significant variations along the spillway profile. Higher pressure values were observed near the spillway surface due to the interaction between the flowing water and the spillway geometry, whereas lower pressure regions developed within the free jet after the flow left the bucket.

The pressure distribution varied with discharge, although the overall pressure pattern remained similar for all simulated cases. The numerical model effectively predicted the transition from high-pressure regions along the spillway surface to atmospheric pressure conditions within the downstream jet. These results demonstrate the capability of FLOW-3D to accurately simulate pressure variations in rapidly varying free-surface flows.

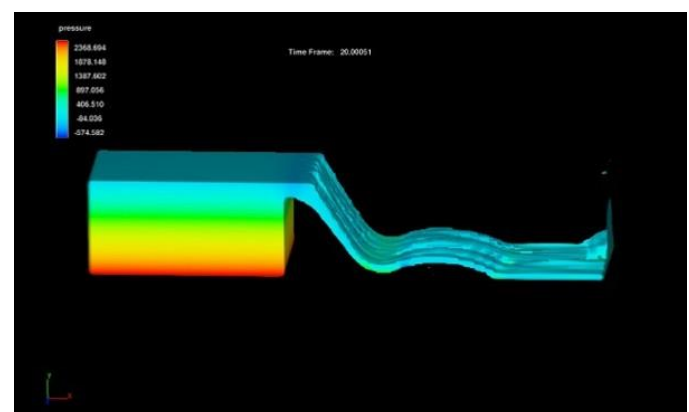


Figure 4. Pressure Contour of the ski-jump bucket spillway.

Free-Surface Flow Behaviour

The free-surface flow was analysed using fluid fraction contours generated through the Volume of Fluid (VOF) method. The numerical simulations accurately captured the air–water interface throughout the computational domain and clearly represented the formation of the free jet downstream of the ski-jump bucket.

The flow remained stable under all discharge conditions, and no abnormal flow separation or excessive numerical oscillations were observed during the simulations. The VOF method successfully reproduced the free-surface characteristics, demonstrating its suitability for modelling rapidly varying hydraulic flows encountered in spillway structures.

C. Jet Trajectory Characteristics

The ski-jump bucket effectively projected the flow away from the spillway toe, forming a well-defined free jet before entering the downstream plunge region. The trajectory length was found to be strongly influenced by discharge.

An increase in discharge resulted in greater jet momentum and consequently increased trajectory length, while lower discharge conditions produced comparatively shorter trajectories. The simulated jet profiles were consistent with the expected hydraulic behaviour of ski-jump bucket energy dissipators and corresponded well with the experimental observations.

The ability of the CFD model to accurately reproduce the jet trajectory demonstrates the effectiveness of FLOW-3D in simulating complex free-surface flow phenomena associated with spillway hydraulics.

D. Effect of Discharge on Hydraulic Behaviour

The discharge had a significant influence on the hydraulic performance of the ski-jump bucket energy dissipator. As the discharge increased, flow velocity, jet momentum, and trajectory length generally increased because of the greater kinetic energy available within the flow. Corresponding variations were also observed in pressure distribution and free-surface geometry.

Despite these variations, the overall flow behaviour remained hydraulically stable throughout the investigated discharge range. The numerical simulations consistently predicted smooth flow development over the spillway profile and effective deflection of the water jet by the ski-jump bucket. These observations confirm that the hydraulic performance of the energy dissipator is strongly dependent on discharge while maintaining stable flow characteristics under the simulated operating conditions.

E. Comparison with Experimental Results

The numerical predictions were compared with experimental observations to evaluate the reliability of the CFD model. The comparison demonstrated good agreement between the simulated and measured energy dissipation values, with percentage errors remaining within acceptable engineering limits.

The minimum percentage error obtained during validation was 3.69%, whereas the maximum observed error was 15.69%. The relatively small deviations between the numerical and

experimental results indicate that the adopted numerical methodology accurately reproduces the principal hydraulic characteristics of the ski-jump bucket energy dissipator.

Minor differences between the numerical predictions and experimental measurements may be attributed to mesh discretization, turbulence model assumptions, numerical approximations, and unavoidable uncertainties associated with laboratory measurements.

Figure 5 Illustrates the comparison between the CFD-predicted and experimentally measured energy dissipation values for the investigated discharge conditions.

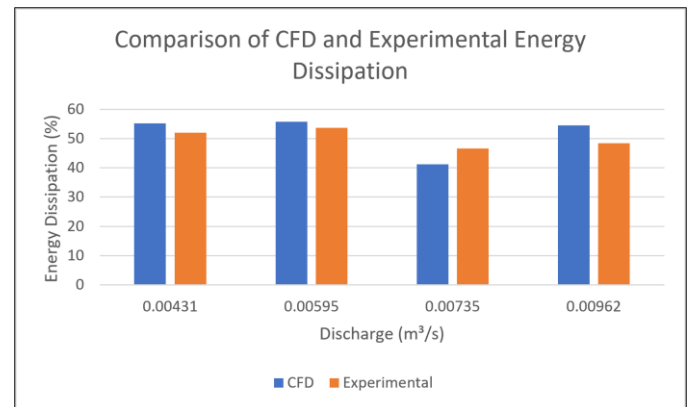


Figure 5. Comparison of CFD and Experimental Energy Dissipation

VI. ENGINEERING IMPLICATIONS

The findings of the present study demonstrate the practical applicability of Computational Fluid Dynamics (CFD) as a reliable tool for analysing the hydraulic performance of ski-jump bucket energy dissipators. The numerical simulations successfully captured the complex free-surface flow characteristics, including velocity distribution, pressure variation, jet trajectory, and energy dissipation, providing comprehensive insights into the hydraulic behaviour of the spillway system.

The validated CFD model offers several advantages over conventional physical model studies. While laboratory investigations remain essential for final verification, CFD enables engineers to perform preliminary design evaluations, assess multiple operating conditions, and optimize spillway configurations with significantly reduced time, cost, and resource requirements. The capability to visualize internal flow fields and hydraulic parameters that are difficult to measure experimentally further enhances the usefulness of numerical modelling in hydraulic engineering applications.

The results also indicate that discharge has a significant influence on the hydraulic performance of the ski-jump bucket energy dissipator. Variations in discharge directly affect flow velocity, pressure distribution, and jet trajectory, emphasizing the importance of evaluating spillway performance under different hydraulic conditions during the design process. The numerical model developed in this study can therefore be used as an effective decision-support tool for assessing the operational performance of spillway energy dissipators before physical implementation.

Furthermore, the satisfactory agreement between the numerical predictions and the experimental observations confirms the capability of FLOW-3D to simulate complex spillway hydraulics with acceptable engineering accuracy. This establishes confidence in the use of CFD for analysing existing spillway structures, investigating design modifications, and supporting rehabilitation projects where detailed hydraulic assessment is required.

Overall, the methodology presented in this study demonstrates that CFD-based numerical modelling can complement conventional experimental investigations and contribute to the design of safer, more efficient, and economically optimized spillway energy dissipation systems. The approach adopted in this work can also be extended to other hydraulic structures, including ogee spillways, stepped spillways, chute spillways, stilling basins, and related energy dissipation systems.

VII. CONCLUSION

A three-dimensional Computational Fluid Dynamics (CFD) model was successfully developed using FLOW-3D to investigate the hydraulic behaviour of a ski-jump bucket energy dissipator under varying discharge conditions. Based on the numerical simulations and validation against experimental observations, the following conclusions can be drawn:

1. The developed CFD model successfully simulated the free-surface flow over the ski-jump bucket energy dissipator and effectively predicted the hydraulic characteristics of the spillway under different operating conditions.
2. The Volume of Fluid (VOF) method accurately captured the air–water interface and free jet formation, while the RNG k - ϵ turbulence model provided stable and realistic predictions of turbulent flow behaviour throughout the computational domain.
3. The numerical simulations demonstrated that discharge significantly influences the hydraulic performance of the ski-jump bucket. An increase in discharge resulted in higher flow velocities, greater jet momentum, and longer jet trajectories, thereby affecting the overall energy dissipation characteristics of the spillway.
4. The pressure distribution obtained from the CFD simulations indicated smooth hydraulic behaviour over the spillway profile, with higher pressure regions occurring along the spillway surface and lower pressure zones developing within the free jet after leaving the bucket.
5. Validation of the numerical model against experimental observations demonstrated good agreement between the CFD predictions and measured results. The percentage error ranged from **3.69% to 15.69%**, confirming that the developed numerical model provides acceptable engineering accuracy for analysing spillway hydraulics.
6. The numerical model successfully reproduced the major hydraulic phenomena associated with ski-jump bucket energy dissipators, including velocity distribution, pressure variation, free-surface flow characteristics, and jet trajectory, demonstrating the capability of FLOW-3D to simulate complex spillway hydraulics.

7. The present study confirms that CFD can serve as an effective complement to conventional physical model investigations by providing detailed hydraulic information while reducing the time, cost, and effort associated with laboratory experimentation. The developed modelling approach can therefore be applied during the preliminary design, performance evaluation, and optimization of spillway energy dissipation systems.

Overall, the findings demonstrate that FLOW-3D is a reliable and efficient numerical tool for analysing the hydraulic performance of ski-jump bucket energy dissipators. The methodology presented in this study contributes to the application of CFD in hydraulic engineering and provides a practical framework for future investigations involving spillway design, energy dissipation, and free-surface flow analysis.

ACKNOWLEDGMENT

The author gratefully acknowledges the guidance and support of Dr. V. S. Chavhan, Assistant Professor, AISSMS College of Engineering, Pune, whose valuable insights, technical expertise, and continuous encouragement significantly contributed to the successful completion of this research.

The author also acknowledges the Department of Technology, Savitribai Phule Pune University, Pune, for providing the necessary academic support and research facilities. The constructive assistance and cooperation received from faculty members and colleagues during the course of this work are also sincerely appreciated.

REFERENCES

- [1] Bombardelli, F. A., Meireles, I., & Matos, J. (2011). Laboratory measurements and numerical modeling of aerated spillway flows: A review. *Journal of Hydraulic Research*.
- [2] Flow Science Inc. (2023). FLOW-3D User Manual. Flow Science Inc., Santa Fe, New Mexico, USA.
- [3] Gautam, H., Ahmad, Z., & Sharma, P. (2023). CFD-based modeling for computing discharge coefficient of an ogee spillway using FLOW-3D. *Proceedings of the International Association for Hydro-Environment Engineering and Research (IAHR)*.
- [4] Hirt, C. W., & Nichols, B. D. (1981). Volume of Fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39(1), 201–225.
- [5] Hughes, W. C., Savage, B. M., & others. (2012). Application of Computational Fluid Dynamics to spillway hydraulic analysis. *Journal of Hydraulic Engineering*.
- [6] Karami Moghadam, M., et al. (2019). Physical modeling of ski-jump spillway to evaluate dynamic pressure in plunge pools. *Water*, 11(8), 1687.
- [7] Oertel, M., & Bung, D. B. (2012). Initial stage of air entrainment in self-aerated spillway flows. *Journal of Hydraulic Research*.
- [8] Savage, B. M., & Johnson, M. C. (2001). Flow over ogee spillways: Physical and numerical investigations. *Journal of Hydraulic Engineering*.
- [9] Weller, H. G., Tabor, G., Jasak, H., & Fureby, C. (1998). A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in Physics*, 12(6), 620–631.
- [10] Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B., & Speziale, C. G. (1992). Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A*, 4(7), 1510–1520.
- [11] Hurtig, K., Vasquez, J., & Croockewit, J. (2013). Verification of CFD simulations of flip bucket spillway performance using physical model results. *Proceedings of the Canadian Dam Association Annual Conference*.

- [12] Meireles, I., Bombardelli, F. A., Matos, J., & others. (2014). Numerical simulation of hydraulic jumps using Reynolds-averaged Navier–Stokes models. *Journal of Hydraulic Research*.
- [13] Bayon, A., Lopez-Jimenez, P. A., Vallés-Morán, F. J., & others. (2016). Performance assessment of OpenFOAM and FLOW-3D in the numerical modelling of a low Reynolds number hydraulic jump. *Environmental Modelling & Software*, 80, 322–335.
- [14] Hinge, G., et al. (2010). Experimental investigation of stepped weir energy dissipators for spillways. *Proceedings of Hydraulic Engineering Conference*.
- [15] Nikrou, P., & Pirboudaghi, S. (2024). Numerical analysis of cavitation dynamics on free ogee spillways using the Volume of Fluid (VOF) method. *arXiv*.
- [16] IS 4997:1968. Criteria for Design of Hydraulic Jump Type Stilling Basins with Horizontal and Sloping Aprons. Bureau of Indian Standards, New Delhi.
- [17] USBR. (1987). *Design of Small Dams* (3rd ed.). United States Bureau of Reclamation.
- [18] Chow, V. T. (1959). *Open-Channel Hydraulics*. McGraw-Hill, New York.
- [19] French, R. H. (1985). *Open-Channel Hydraulics*. McGraw-Hill.
- [20] Chanson, H. (2004). *The Hydraulics of Open Channel Flow: An Introduction*. Elsevier