CFX Analysis of Kaplan Turbine

Mangesh Chaudhari[1], Omkar Shaha[2], Shantanu Dange [3], Shivraj Shinde[4], Siddhant Shinde[5], Rishikesh Yadav[6]
[1,2,3,4,5,6]*Mechanical Department, Vishwakarma Institute Of Technology*

***Abstract*—Hydropower has always been seen as a dependable and clean source of energy, and at the heart of it, turbines like the Kaplan turbine play a major role in how efficiently energy is generated. In this study, an attempt has been made to understand the real flow behaviour inside a Kaplan turbine using Computational Fluid Dynamics (CFD) analysis carried out in ANSYS CFX. Instead of relying only on theoretical assumptions, a 3D model of the turbine was developed and tested under different working conditions to see how water actually interacts with the runner blades, guide vanes, and draft tube. The focus was mainly on observing how pressure and velocity change within the turbine and where losses or disturbances in flow tend to occur. Certain regions were also studied more carefully to identify possible chances of cavitation, which can affect performance over time. By slightly varying parameters like flow rate and blade angle, it became easier to notice how sensitive the turbine performance is to operating conditions. The results from the simulation helped in getting a clearer picture of where improvements can be made in design to reduce losses and improve efficiency. Overall, this work shows how practical and useful CFD tools can be when it comes to studying and improving real engineering systems like hydraulic turbines.**

***Index Terms*—Kaplan turbine, computational fluid dynamics, ANSYS CFX, hydropower, flow analysis, turbine efficiency, cavitation, pressure and velocity distribution**

I. INTRODUCTION

Hydropower has remained one of the most practical and reliable ways of generating clean energy, especially where water resources are easily available. In such applications, the Kaplan turbine is commonly preferred because it works efficiently under low head and high flow rate conditions. Unlike simple impulse turbines, it is a reaction type, where both pressure and velocity of water contribute to energy generation. What makes the Kaplan turbine stand out is its ability to adapt to changing conditions through adjustable runner blades and guide vanes, helping it maintain good efficiency over a wide range of operation. The basic construction includes a spiral casing that distributes water evenly, guide vanes that direct the flow, runner blades where the actual energy conversion takes place, and a draft tube that helps in recovering energy before water exits. Gaining a clear understanding of how water behaves inside these components is important for improving overall turbine performance.

1.1 Project Objectives

The objectives of this project are aimed at understanding the working and performance of a Kaplan turbine in a practical way. The study focuses on analyzing flow behaviour and key parameters like pressure, velocity, and head, to identify losses and improve overall efficiency through detailed CFD analysis.

1.2 Flow Behaviour Through Turbine

The first objective of this study is to understand how water actually flows through the turbine during operation. Instead of just looking at theoretical values, the aim is to closely observe the movement of water as it passes through different components like the guide vanes, runner blades, and draft tube. By analyzing the flow behaviour, it becomes easier to identify areas where energy losses occur or where the flow becomes uneven or turbulent. This understanding helps in visualizing how effectively the turbine is converting hydraulic energy into mechanical energy and also provides a base for improving overall performance.

1.3. Analysis of Head, Pressure, and Velocity Distribution in the Turbine

The second objective of this work is to study how head, pressure, and velocity change as water moves through the turbine. These parameters are directly linked to how efficiently the turbine converts hydraulic energy into useful mechanical output. Rather than treating them as fixed values, this study looks at how pressure gradually drops from the inlet to the outlet, how velocity varies at different sections, and how effectively the available head is being utilized. Small changes in these factors can significantly affect performance, so observing them in detail becomes important. By analyzing these variations, it becomes possible to identify regions where the flow is uneven, energy losses are higher, or conditions are not ideal. This approach helps in building a more realistic understanding of the turbine's internal behaviour and provides useful insights for improving its design and overall efficiency in practical applications.

II. LITERATURE SURVEY

The literature review looks at previous studies related to Kaplan turbines and CFD analysis to understand how researchers have approached similar problems. By going through existing work, it becomes easier to identify key findings, common methods, and gaps that still need attention, which helps in shaping the direction of the present study.

1. The literature survey shows that many researchers have worked on improving the efficiency of Kaplan turbines, especially for low head hydropower applications. Most studies focus on the design and optimization of runner blades using CAD modelling and CFD analysis techniques. It has been observed that even slight changes in blade geometry and angle can influence the turbine's performance and power output. The reviewed paper mainly highlights the optimization of runner blade profiles by testing different blade angles and comparing their performance using CFD analysis. It also emphasizes the validation of simulation results with theoretical data to ensure accuracy. The findings show that proper blade modification can lead to a noticeable improvement in power output and overall efficiency. Overall, previous research clearly indicates that CFD tools are highly useful in understanding flow behaviour and enhancing turbine design.

2. The literature survey of the selected paper mainly focuses on the design and performance improvement of a Kaplan turbine runner using both theoretical calculations and CFD analysis. The study highlights that hydropower is one of the most reliable renewable energy sources, and turbine efficiency largely depends on proper blade design and flow management. Earlier works discussed in the paper compare experimental and CFD results, showing that advanced simulation methods can closely predict real turbine performance. The paper further emphasizes that traditional theoretical designs often rely on assumptions, which may lead to differences when compared with practical results. A key highlight of the study is the optimization of runner blades using CFD, where parameters like blade angle and number of blades are adjusted to improve efficiency. The survey clearly indicates that combining theoretical design with CFD analysis is essential for achieving accurate and efficient turbine performance in real-world applications.

3. The literature survey of another study looks at how the actual flow of water inside a Kaplan turbine affects its performance. Instead of focusing only on theory, the researchers tried to understand what really happens to the flow as it passes through the runner and draft tube. They found that small factors like blade angle, guide vane position, and flow disturbances can have a noticeable impact on efficiency. One important point highlighted in the paper is the formation of vortices and uneven flow patterns, which can lead to energy losses if not properly controlled. The study mainly emphasizes observing real flow behavior under different operating conditions to get a clearer picture of turbine performance. Overall, it shows that paying attention to internal flow characteristics and making slight design adjustments can help improve efficiency, and that CFD tools play a very useful role in analyzing and optimizing turbine performance in a practical way.

III. METHODOLOGY EXPERIMENTAL

The methodology adopted in this work focuses on analyzing the performance of a Kaplan turbine using a combination of 3D modeling and Computational Fluid Dynamics (CFD). The process began with the creation of a detailed 3D model of the turbine runner using Solid Edge. The geometry included essential parts such as the hub, blades, and shaft. While designing, basic proportions and blade orientation were considered to ensure realistic flow interaction.

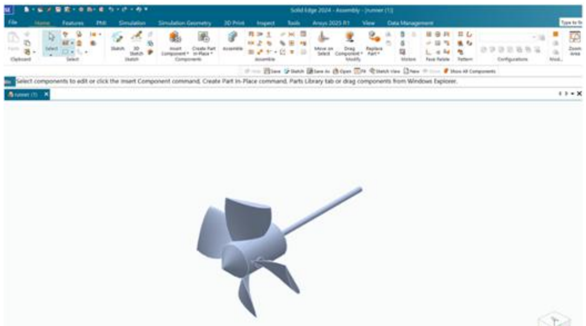

Figure 1 - 3D CAD Model Design 1

After completing the geometry, the model was imported into ANSYS CFX for further analysis. A cylindrical fluid domain was created around the runner to represent the water flow region. This domain helps simulate real operating conditions inside the turbine. The geometry was then discretized using a tetrahedral mesh, ensuring finer elements near blade surfaces to capture flow variations accurately.

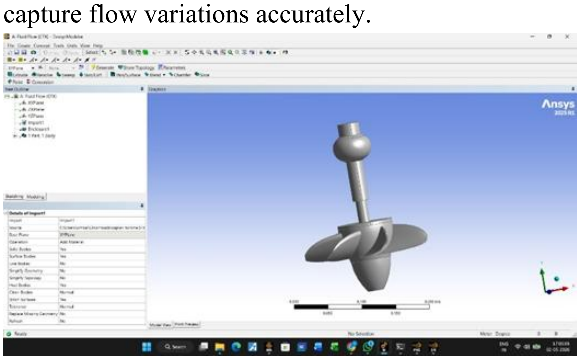

Figure 2 - 3D CAD Model Design 2

The figure 2 shows the 3D geometric model of the rotating component developed using ANSYS Design Modeler. The model consists of a central shaft connected to a multi-blade impeller, designed to facilitate fluid movement and mixing within the domain. The geometry is created with proper dimensional considerations to ensure smooth flow interaction and realistic simulation conditions. The helical blade profile is adopted to enhance flow circulation and improve fluid handling efficiency.

Special attention has been given to maintaining geometric symmetry and smooth surface transitions in order to minimize numerical errors during meshing and simulation. The finalized model is then used as the computational domain for further CFD analysis, including flow behaviour, pressure distribution, and turbulence characteristics

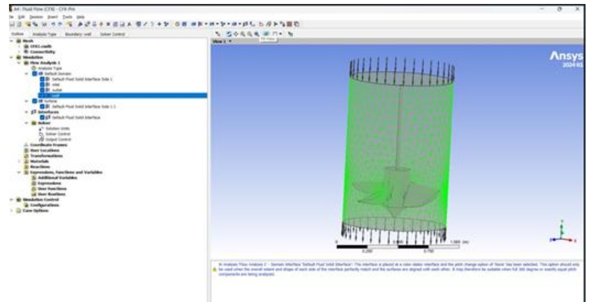

Figure 3 - Meshed Model 1

Once meshing was completed, boundary conditions were applied. The inlet was defined with a uniform velocity corresponding to the flow rate, while the outlet was assigned a pressure boundary condition. The runner was defined as a rotating domain with a specified angular velocity. A steady-state analysis was performed using an appropriate turbulence model to capture realistic flow behavior.

For the second turbine design (refer Fig 4), a separate meshing approach was carried out considering its different geometry and structural features. An unstructured tetrahedral mesh was generated using ANSYS Meshing, as it is well-suited for handling complex blade shapes and curved surfaces. Compared to the first design, this geometry required careful refinement around the blade edges, hub region, and shaft connection to accurately capture flow variations. A suitable element size was selected to maintain a balance between accuracy and computational effort, while local mesh refinement was applied in regions where higher gradients of pressure and velocity were expected. The mesh quality was evaluated using standard criteria such as skewness and element quality to ensure numerical stability during simulation.

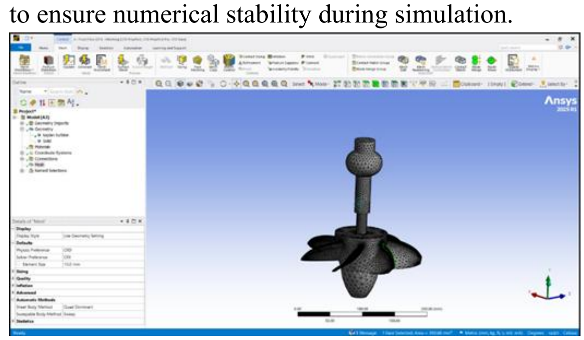

Figure 4 - Meshed Model 2

Key equations used in the analysis include:
1. Continuity Equation
2. Momentum Equation (Navier–Stokes)
3. Power Equation:

$$P = \rho g Q H$$

4. Efficiency Equation:

$$\eta = \frac{P_{output}}{P_{input}} \times 100$$

These equations help in evaluating flow characteristics and turbine performance.

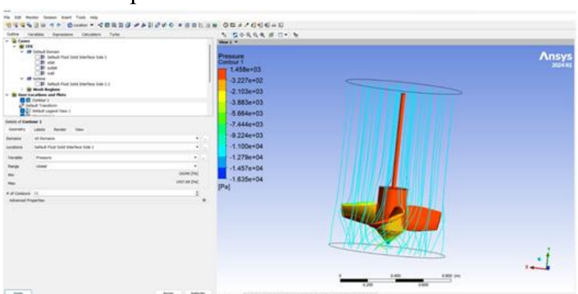

Figure 5 - Pressure Distribution Contour

The figure 5 shows illustrates the pressure contour distribution along with flow streamlines inside the domain. Higher pressure regions are observed near the blade/impeller surfaces (red zone), while lower pressure regions (blue/green) appear along the downstream flow path. The streamlines indicate the flow direction and reveal the interaction between the rotating component and the surrounding fluid, highlighting pressure variation and flow behavior within the system.

The figure 6 shows the convergence history of turbulence residuals during the simulation. It can be observed that both turbulence kinetic energy (k) and turbulence frequency (ω) residuals drop sharply in the initial iterations, indicating rapid stabilization of the solution. As the iterations progress, the rate of reduction becomes gradual, and the residuals approach a steady, low value. This trend confirms that the solution is converging and the numerical results have reached a stable condition.

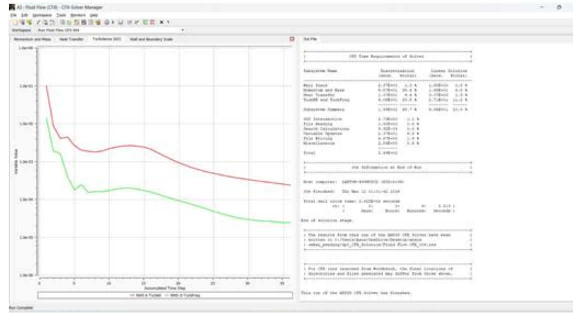

Figure 6 - Turbulence residual convergence plot (design 1)

The convergence of the simulation was evaluated using RMS residual plots for turbulence quantities (k and ω). Initially, higher residuals are observed due to flow field initialization; however, with increasing iterations, a steady decline is seen, indicating solution stabilization.
The ω residual shows a rapid decrease, reaching the order of 10^{-4}, while the k residual gradually reduces and stabilizes around 10^{-2}, both within acceptable convergence limits for turbulent flow simulations. The solver output also confirms stable numerical behavior with all equations satisfying convergence criteria.
Overall, the consistent reduction in residuals indicates that the solution has achieved satisfactory convergence and reliability.

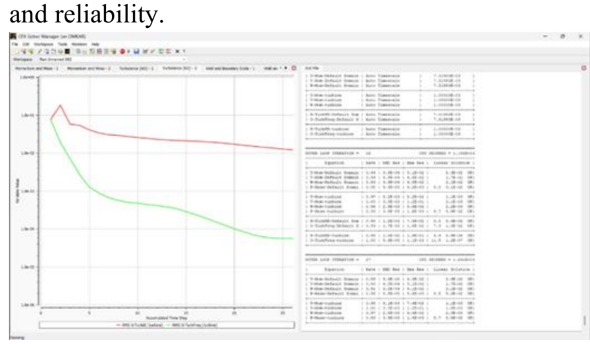

Figure 7 - Turbulence residual convergence plot (design 2)

The figure 8 presents the convergence history of momentum and mass residuals during the simulation. All residuals show a sharp decrease in the initial iterations, indicating rapid stabilization of the flow solution. As the iterations progress, the residuals gradually reduce and approach lower values, demonstrating steady convergence.

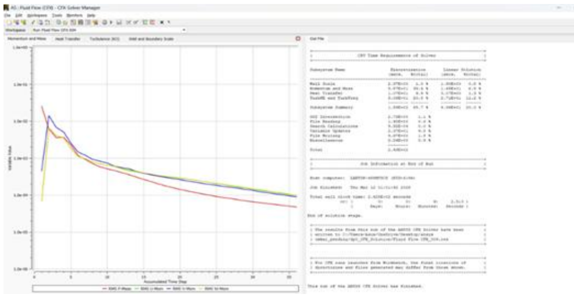

Figure 8- Mass and momentum residual convergence plot (design 1)

The smooth and consistent decline without major oscillations confirms the numerical stability and reliability of the obtained solution. Furthermore, the close trend followed by all residual curves indicates good coupling between governing equations, ensuring that continuity and momentum equations are being satisfied simultaneously. This behavior reflects the accuracy of the numerical setup and appropriateness of boundary conditions used in the simulation.

The figure 9 shows the convergence history of mass and momentum residuals for the turbine domain during the simulation. Initially, the residuals exhibit a slight fluctuation due to the adjustment of the numerical solution; however, a rapid decrease is observed within the first few iterations, indicating quick stabilization of the flow field. As the solution progresses, all residuals follow a consistent downward trend, gradually reaching lower magnitudes. The mass residual (continuity) drops significantly compared to the momentum components, confirming that mass conservation is effectively achieved. The momentum residuals in all three directions (U, V, and W) also show stable convergence with minimal oscillations. This smooth and steady reduction in residual values indicates good numerical stability, proper solver settings, and reliable convergence of the solution.

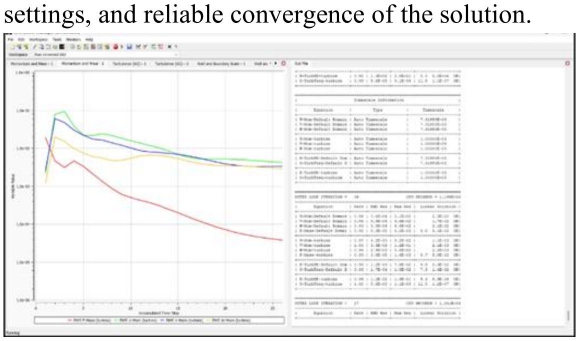

Figure 9- Mass and momentum residual convergence plot (design 2)

After running the simulation, results such as pressure variation, velocity distribution, and flow streamlines were obtained. These results were carefully analyzed to understand how water interacts with the runner blades and to identify regions of energy loss or inefficient flow. The methodology provides a clear approach to studying turbine behavior and improving its overall efficiency using CFD tools.

CFD Setup Details:
The simulation was performed using a steady-state solver with water as the working fluid, considering a density of approximately 1000 kg/m³. A tetrahedral mesh was used for discretizing the computational domain, with finer elements near the blade surfaces for better accuracy. The turbulence effects were modeled using the k-ε model. The turbine runner was defined as a rotating domain with a specified rotational speed (RPM), and a rotor-stator interface was applied to handle the interaction between stationary and rotating parts. Convergence of the solution was ensured by maintaining residual values below 10^{-5}.

CFD Setup Details
1. Solver Type: Steady State
2. Fluid: Water ($\rho = 1000$ kg/m³ approx.)
3. Turbulence Model: k-ε or SST
4. Mesh Type: Tetrahedral
5. Rotating Domain: Defined using rotational speed (RPM)
6. Interface: Rotor-Stator Interface.
7. Convergence Criteria: Residuals $< 10^{-5}$.

These settings were chosen to closely represent real operating conditions while maintaining numerical stability. Proper selection of turbulence model and mesh refinement ensures accurate prediction of flow behavior around the runner blades. The setup allows reliable evaluation of pressure, velocity distribution, and overall turbine performance.

IV. RESULTS AND DISCUSSIONS

The CFD analysis was carried out to evaluate the performance of the Kaplan turbine and to understand how water behaves inside it under given operating conditions. The results obtained from the simulation mainly focus on pressure distribution, velocity variation, and overall flow behavior, which together give a clear picture of turbine efficiency. The pressure distribution across the runner blades shows a noticeable variation from inlet to outlet. A higher-pressure region is observed at the blade inlet, which gradually reduces towards the outlet side. This pressure difference across the blade surfaces is what drives the rotation of the runner and enables energy conversion. The contour plot clearly represents this variation, where red regions indicate high pressure and blue regions represent low pressure.

Comparison of Both Designs

Parameter	Design 1	Design 2	Remarks
Mass & Momentum Convergence	Fluctuations present	Smooth and stable	Design 2 better
Turbulence (k–ω) Residuals	Faster drop, lower values	Slower, slightly higher	Design 1 better
Residual Smoothness	Minor oscillations	Uniform decay	Design 2 better
Numerical Stability	Good	More stable	Design 2 better
Overall Performance	Better Turbulence Resolution	Better stability	Design 2 preferred

Design 1 exhibits faster convergence and achieves lower residual values for turbulence parameters, indicating better prediction of flow physics. In contrast, Design 2 demonstrates smoother and more stable convergence for mass and momentum equations, ensuring improved numerical stability and conservation. Overall, Design 2 is preferable for stability, while Design 1 offers advantages in turbulence resolution.

The pressure distribution across the runner blades shows a noticeable variation from inlet to outlet. A higher-pressure region is observed at the blade inlet, which gradually reduces towards the outlet side. Some localized low-pressure zones can also be seen near the trailing edges of the blades, which may indicate a slight risk of cavitation if conditions are not properly controlled. The velocity distribution inside the turbine highlights how water accelerates as it moves through the guide vanes and enters the runner. Higher velocities are observed in the blade passage regions where direct interaction with the blades occurs. The flow appears mostly smooth, although minor disturbances can be noticed due to changes in geometry and rotation. These variations are important, as sudden changes in velocity can lead to energy losses.

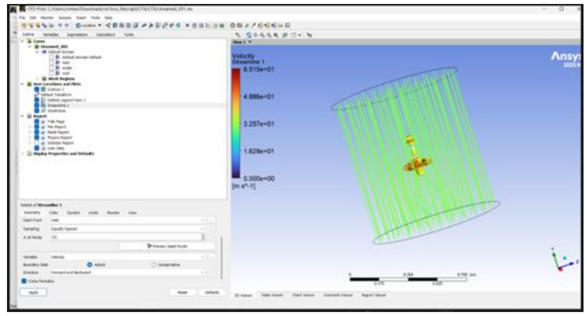

Figure 10 - Velocity Contour / Streamline Plot

Flow behaviour plays a very important role in determining turbine performance. The streamlines indicate that water generally follows a smooth path through the turbine. However, slight swirling and small vortices can be observed near the outlet region. Ideally, the flow should exit in a straight direction, but the presence of swirl suggests that some energy is still left unused, indicating scope for improvement.

From a performance point of view, efficient energy conversion depends on maintaining smooth and uniform flow. Any turbulence or uneven distribution leads to losses. The turbine performance can be related using equations like $P = \rho g Q H$ and $\eta = \frac{P_{out}}{P_{in}}$, which connect flow conditions to power and efficiency.
Overall, the results show that blade design has a strong influence on pressure difference and flow pattern. Better flow uniformity improves efficiency, while irregularities reduce performance. Minor differences between CFD and theoretical results are expected due to assumptions in calculations. Further improvements can be achieved by optimizing blade angles and controlling flow conditions more effectively.

V. CONCLUSION

The present study successfully demonstrated the use of CFD analysis to understand the performance and internal flow behaviour of a Kaplan turbine. By developing a 3D model and simulating it in ANSYS CFX, important parameters such as pressure distribution, velocity variation, and flow patterns were analyzed in detail. The results clearly showed that a proper pressure difference across the runner blades is essential for effective energy conversion, while smooth and well-directed flow plays a key role in improving overall efficiency. A detailed assessment of residual convergence for both designs highlights a clear trade-off between numerical stability and turbulence resolution. Design 2 demonstrates superior performance in terms of mass and momentum convergence, exhibiting a smooth and monotonic decay of residuals with minimal oscillations. This indicates stronger numerical stability and more consistent satisfaction of fundamental conservation laws, which is critical for ensuring solution robustness. In contrast, Design 1 shows faster convergence and achieves comparatively lower residual levels for turbulence parameters (k–ω), suggesting a stronger capability in resolving turbulent flow characteristics. However, the presence of residual fluctuations in mass and momentum equations points to relatively lower numerical stability.

Considering that stable convergence of continuity and momentum equations forms the foundation of a physically reliable CFD solution, Design 2 is selected as the preferred configuration. Its consistent and stable convergence behavior outweighs the marginal advantage of Design 1 in turbulence residual reduction, ensuring greater confidence in the overall simulation accuracy and repeatability of results.

REFERENCES

[1] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed. Pearson Education, 2007.

[2] J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications. New York, NY, USA: McGraw-Hill, 1995.

[3] S. V. Patankar, Numerical Heat Transfer and Fluid Flow. Washington, DC, USA: Hemisphere Publishing, 1980.

[4] D. C. Wilcox, “Reassessment of the scale-determining equation for advanced turbulence models,” AIAA Journal, 1988.