Proceedings of 14th IOE Graduate Conference
Peer Reviewed
Year: 2023 Month: December Volume: 14
ISSN: 2350-8914 (Online), 2350-8906 (Print)

Study of the Effects of Blockage in Centrifugal Pump using CFD Simulation

Bibek Adhikari [a], Nawraj Bhattrai [b], Kamal Darlami [c]

[a, b, c] Department of Mechanical and Aerospace Engineering, Pulchowk Campus, IOE, Tribhuvan University, Nepal
[a] 078msmde005.bibek@pcampus.edu.np, [b] bnawraj@gmail.com, [c] darlami.kd@pcampus.edu.np

Abstract
Centrifugal pumps play a crucial role in various industrial sectors, facilitating liquid transport in applications ranging from construction to petrochemicals. Ensuring their efficient and reliable operation is paramount. However, these pumps are susceptible to faults and defects throughout their lifecycle, necessitating the development of diagnostic tools for early detection and prevention. Understanding the complex dynamics within the pump is essential for optimal performance. This study explores the utilization of OpenFOAM, an open-source software, in Computational Fluid Dynamics (CFD) to analyze the flow field within centrifugal pumps. Both steady-state simulations, employing the simpleFoam solver with the Multiple Reference Frame (MRF) approach, and unsteady transient simulations, utilizing the pimpleFoam solver with the Sliding Mesh approach, are conducted. A CFD model of the Oberdorfer 60P pump is developed to simulate fluid flow patterns, focusing on critical parameters such as flow rate, head, pressure distribution, and force characteristics. Visualization of the flow field is achieved using the ParaView utility of OpenFOAM, depicting velocity vectors and static pressure distribution. Blockages within the pump lead to decreased static pressure near the inlet, resulting in increased pressure changes across the fluid domain. The flow velocity near the outlet section of the pipe gets reduced which results in the decrement of the discharge at the outlet.

Keywords
CFD, OpenFOAM, Centrifugal Pump, Turbomachinery, Blockage, Frozen rotor, MRF

1. Introduction

1.1 Background

Centrifugal pumps play a crucial role in various engineering applications and find widespread use across industries, agriculture, and domestic settings. Their popularity stems from their simplicity in design, ease of installation and maintenance, as well as their remarkable efficiency in fluid transfer. These pumps are versatile, capable of handling a wide range of pressures and flow rates, making them ideal for lifting liquids such as water, chemicals, oils, and other fluids from one location to another. However, they are less suitable for dealing with highly viscous substances or fluids containing significant solid content. The design of a centrifugal pump comprises several essential components, including the impeller, casing, inlet and outlet ports, and shaft. The pump's performance depends on various factors, including its design, operating conditions, and the specific properties of the fluid being pumped. Key performance parameters that needs to be studied for centrifugal pumps encompass pressure head, flow rate, and efficiency.

Since well before the industrial revolution, centrifugal pumps have been used in a variety of engineering applications. Denis Papin, a French inventor, invented the first real centrifugal pump in 1687 A.D., which is where this technology first emerged. The simple design of this early pump, which was primarily utilized for local drainage, included a straight vane impeller and a casing with a spiral volute. However, it was towards the middle of the 19th century that the centrifugal pump as we know it today developed and took shape[1].

Modern centrifugal pumps have seen significant enhancements in their impeller design, casing shape, and sealing mechanisms, resulting in improved pump performance and efficiency. There has been a lot of research done on investigating different design and performance parameters. However, since there are so many design variables at play, predicting the design and performance of these pumps continues to be a challenging issue. Additionally, traditional methods involving building and testing physical models comes at a high cost and require a lot of time. Testing and prototyping centrifugal pumps can be a challenging, time-intensive, and costly endeavor. The process of designing and assessing the performance of these pumps is intricate. The inner workings of the pump, involving fluid flow, are often difficult to comprehend and visualize. Furthermore, the presence of faults within a centrifugal pump only adds complexity, making it exceedingly challenging to predict the pump's performance accurately. To achieve cost-effectiveness and time efficiency in pump design, it becomes crucial to forecast their performance before actual manufacturing. This necessitates a thorough understanding of fluid flow behaviour within the various sections of the pump.

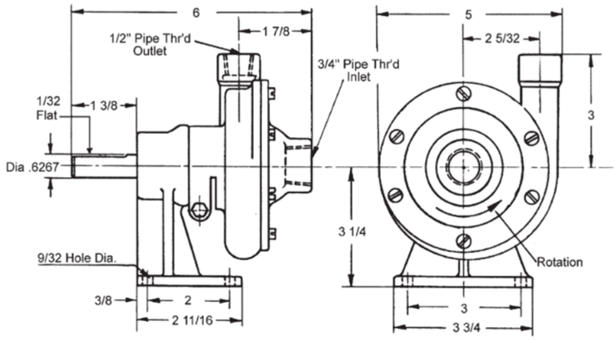
One approach to gaining insight into fluid flow behavior and predicting pump performance is through experimental model testing. However, this approach is known for being laborious, time consuming and expensive [2]. As a result, Computational Fluid Dynamics (CFD) has now emerged as the go-to approach for analyzing centrifugal pump design and performance. CFD analysis has become the predominant approach in centrifugal pump design. CFD enables the accurate prediction and comprehension of the complex fluid flow within the pump. By doing so, it speeds up the design process by skipping the time consuming stages of physical model and prototype testing. Jaiswal [3] in his paper shows that when discharge rises, head falls, power input rises, and pump efficiency rises. The Shear Stress Transport (SST) turbulence model yields superior results compared to other models. Lei at al. [4] used both the direct and inverse methods to investigate how the blade wrap angle affected the centrifugal pump's performance. Most research is primarily concerned with the design process and parameter optimization. The paper shows that the hydraulic loss becomes larger as the blade outlet angle is made larger.

Commercial CFD softwares like ANSYS, STAR-CCM+ are user-friendly but doesn't allow users to modify the code and thus limits the user to use the available solvers and code. However, OpenFOAM is an open-source software that offers wide range of solvers allowing users to modify the code. Nilson et al. [5] and Xie [6] studied the rotor-stator interaction using openFOAM in the ERCOFTAC Centrifugal Pump. The 2D and 3D models of the pump were generated to investigate the interaction between the flow in the impeller and in the diffuser. Both steady-state and transient simulations were employed for the study of the pump and the result was compared with the actual experimental measurements. David [7] explores the applicability of the openFOAM for simulating rotor-stator turbomachinery. Fully 3D transient simulations of the pump are conducted in openFoam and is compared with the ANSYS CFX. The thesis also investigates the phenomenon of cavitation in pumps. The results obtained using the openFOAM are found to be comparable to those obtained from ANSYS CFX. Huang et al. [8] performed numerical simulation and performance prediction of a centrifugal pump's flow field using openFOAM. The results by openFOAM and ANSYS-Fluent were compared and it was found that OpenFOAM had high accuracy in the prediction of the pump performance.

Bordoloi et al. [9, 10] in his paper proposed a method to identify the occurrence and severity of the blockages and the cavitation inside the pump. The inlet flow area of suction pipe is constricted to simulate the blockage in the pump. The pressure signals were recorded at different blockage levels running at different speeds. This pressure signals was used as a input to a deep learning model to classify the blockage level and cavitation severity. Additionally, the study describes how vibration signals can be used to diagnose blockage levels and impede cavitation at various pump speeds using machine learning methods called SVMs. Kumar et al. [11] conducted research on identifying blockage faults in the inlet of the centrifugal pump using data from the multiple sensors such as accelerometer, pressure transducer . The research uses deep learning algorithm for multiclass classification of the blockage level in the inlet pipe and the severity level. Rapur et al. [12, 13] carried out a research study aimed at classifying various types of faults in a centrifugal pump operating at known and unknown test speeds. Study specifically focused on identifying independently occurring and co-occurring faults in the pump. The paper focuses on suction blockage and impeller defects. Vibration data were collected for both healthy and faulty pumps. They used tri-axial vibration data from the pump and employed the Support Vector Machine algorithm on the processed data for different faults with Radial Basis Function kernel to classify the results. The study found promising results and accuracy for finding distinct faults or multiple coexisting faults.

1.2 Testcase Description

The 60P Oberdorfer centrifugal pump has been chosen for the performance study as shown in the *Figure 1*. The pump consists of an impeller with an outer diameter of 91mm and 5 backward curved blades. The features and design parameters for the centrifugal pump are shown in the *Table 1*.



[Source: `https://www.oberdorferpumps.com/en-us/centrifugal-pumps/end-suction/60p`*]*

Figure 1: Geometry of the Pump 60P Oberdorfer

Table 1: Feature and Design Parameter of the Oberdorfer 60P Pump

Rotational Speed	1725 to 3450 RPM
Ports	Inlet: 23mm, Outlet: 18mm
Flow	Upto 1.5 Liter Per Second (LPS)
Head	Maximum 16m
Temperature (Celsius)	20
Number of Blades	5

2. Design Study

2.1 Modeling of the Fluid Domain

The 3D model of the pump was prepared for the simulation as in the *Figure 2*. The CAD model consists of inlet cover, volute casing and an impeller rotor inside the volute casing as shown in the *Figure 3*. The diameter of the inlet pipe is 23mm and that of the outlet pipe is 18mm. A computational fluid domain of the centrifugal pump was created as shown in the *Figure 4* for the simulation. The fluid domain consists of three subdomains:

- Inlet Domain
- Impeller Domain
- Volute Domain

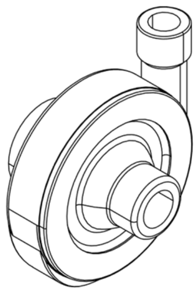


Figure 2: CAD Model of Oberforfer 60P Pump

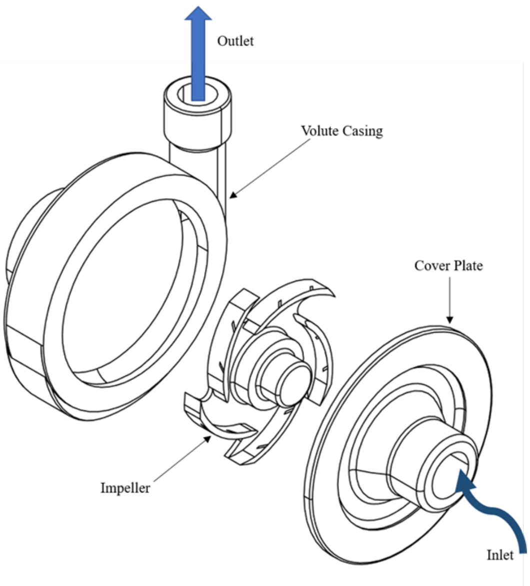


Figure 3: Exploded View of the Oberdorfer 60P Pump

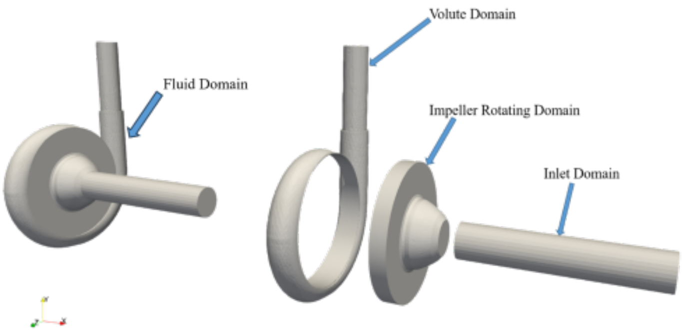


Figure 4: Computational Fluid Domain with Exploded View

2.2 Mesh Generation

After the preparation of the fluid domain, a mesh of appropriate size is created for the computational domain. The size of the mesh elements is selected such that mesh convergence is acquired. The total number of elements is 336087. The mesh for individual fluid domain is shown as in the *Figure 5*. The *Figure 6* shows mesh at the inlet domain.

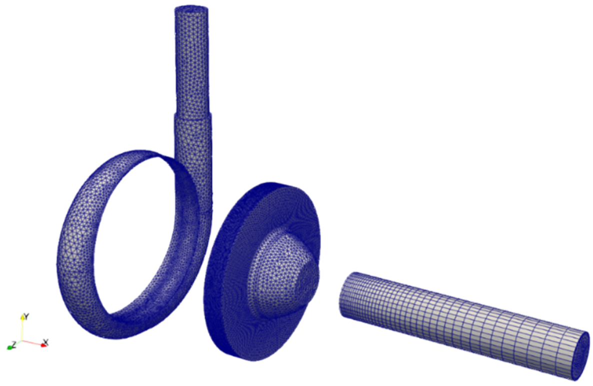


Figure 5: Mesh of Individual Fluid Domain

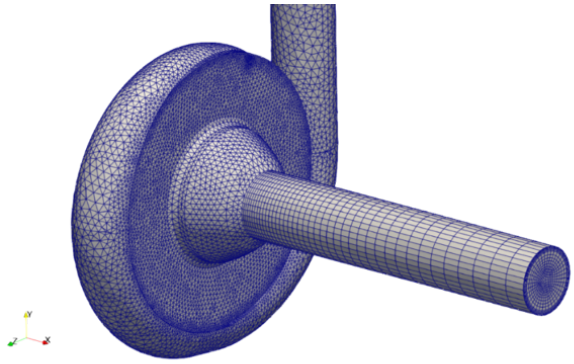


Figure 6: Zoomed in View Showing Mesh at the Inlet Domain

The whole fluid domain is discretized into three separate fluid domains. The inlet and the volute domain are static zones whereas the impeller domain is set as the rotating zone while using the multiple reference frame approach. while performing the simulation.The *Table 2* shows the number of individual mesh element used in the simulation.

Table 2: Domain wise Mesh Element Table

Domain	Element Number
Inlet	25160
Impeller	221880
Volute	89047

2.3 Mesh Independence

Mesh independence study refers to the state where the solutions of a simulation remain relatively unchanged as the mesh resolution is increased. It means that the outcome of the simulation is not explicitly affected by the size of the mesh. For the mesh independence study, the element size is increased in successive steps and the accuracy of its corresponding solution is examined. Generally, refined mesh consists of larger element size which in turn increases the computational time. Whereas, coarse mesh results in a smaller number of element size decreasing the computational time. Therefore, it is critical to perform mesh independence study in order to ensure a balance between solution accuracy and computational time efficiency.

In this study, steady state simulation is ran using the simpleFOAM solver in openFOAM. The pump is run at 1800 rpm and a mass flow rate of 0.504 Ltr/s is supplied as a mass flow rate inlet boundary condition. The boundary condition at the outlet face is set as a pressure-outlet with a value of 1 atm. The forces and the moments on the rotor are observed in order to check for convergence of the solution. Then the number of the mesh size is gradually increased to check for the mesh independence. The result from the mesh independence test is shown in the *Figure 7*.

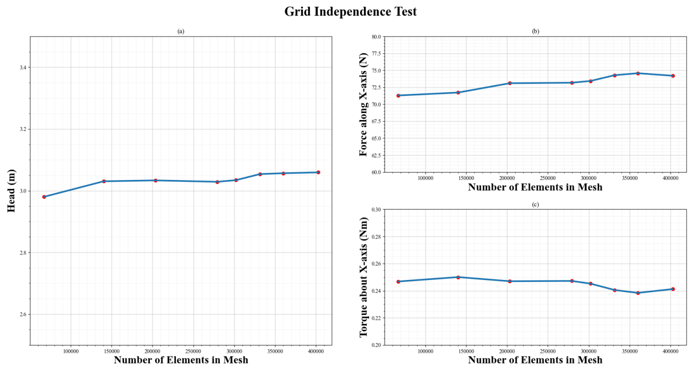


Figure 7: Mesh Independence Test (a) Head versus Number of Elements, (b) Force versus Number of Elements, (c) Torque versus Number of Elements

The above plot consists of three different individual plots. The plot (a) shows the head generated with successive number of mesh elements and the plot (b) and (c) shows the force and torque generated due to the flow. For a mesh with around 300000 – 350000 elements, the test yields acceptable results such that the pressure head generated doesn't vary.

2.4 Case Setup and Simulation

After the creation of the mesh of the fluid domain in the ANSYS Meshing, the mesh is imported into the openFOAM. Several boundary conditions are imposed in order to run the simulation. In general, velocity inlet and pressure outlet are the most widely used boundary conditions for the simulation of a centrifugal pump. Other boundary conditions include pressure inlet-pressure outlet, total pressure inlet-mass flow rate outlet, etc. As for this simulation, the boundary conditions used are as follows:

- Mass flow rate inlet at the inlet patch
- Static Pressure of 1 atm at the outlet patch

Steady-state CFD simulation of the centrifugal pump was done using simpleFoam using the Multiple Reference (MRF) approach. Here, the solver solves the governing equations for both the rotating and the stationary regions. The governing equations for the rotating regions contain an additional source term for Coriolis force. In the case of steady-state simulation, the head and the forces on the impeller blades are monitored for the convergence of the solution. Similarly, transient CFD simulation of the centrifugal pump was done using pimpleFoam solver using the dynamic mesh approach. Here, the fluctuations of the head, the forces and the moments acting on the impeller blades are monitored with time for convergence. Also, two pressure probes at two different locations in the pump are set and the fluctuation of the static pressure at the two-probe location with time is also monitored and recorded.

3. Results and Discussion

Both the Steady state and Unsteady state CFD simulation of the centrifugal pump is run with various inlet conditions i.e., by varying mass flow rate at the inlet and respective flow behavior is monitored. The variations of the mass flow rate at the inlet corresponds to the severity of blockage at the inlet. To study the fluctuations of variables in the flow field due to blockage, pressure and velocity contours are plotted for various mass flow rate at the inlet. Also, two pressure probes are placed in the fluid domain to study the pressure fluctuation with time for transient simulation.

3.1 Simulation Results Validation

A set of simulations is conducted to confirm the validity of the approach employed in our analysis. Initially, the obtained results are cross-referenced with the performance chart provided by the manufacturer of the Centrifugal pump, specifically the Oberdorfer 60P model. The *Figure 8* shows the graph of head versus the rpm at the inlet discharge of 0.5 LPS.

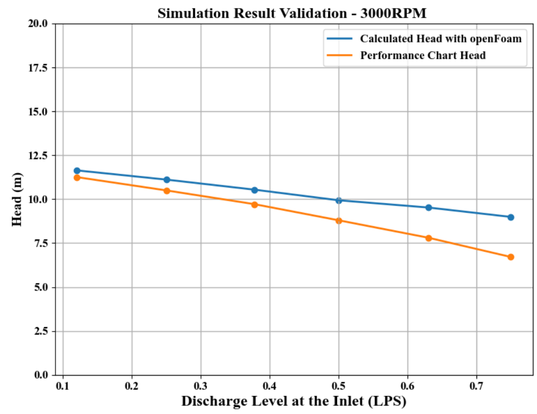


Figure 8: Result validation at 0.5 LPS

The Figure shows a close agreement between the results from the simulation and from the performance chart. The head predicted by the openFOAM are higher than the head given by the pump performance chart with a mean head difference of 0.715 meters.

A sequence of steady-state simulations is conducted, maintaining a constant rpm of 3000 while varying the inlet discharge from 0.12 LPS to 0.75 LPS. The predicted head values obtained through OpenFOAM are then compared with those from the performance chart. Agreement between the predicted and chart head values is observed at lower inflow rates; however, discrepancies emerge at higher discharge rates exceeding 0.5 LPS, as depicted in *Figure 9*. This divergence is attributed to increased turbulence within the pump at high rpm and discharge rates. Consequently, the utilization of the kEpsilon turbulence model in the simulation may influence the discrepancy in head results obtained from OpenFOAM.

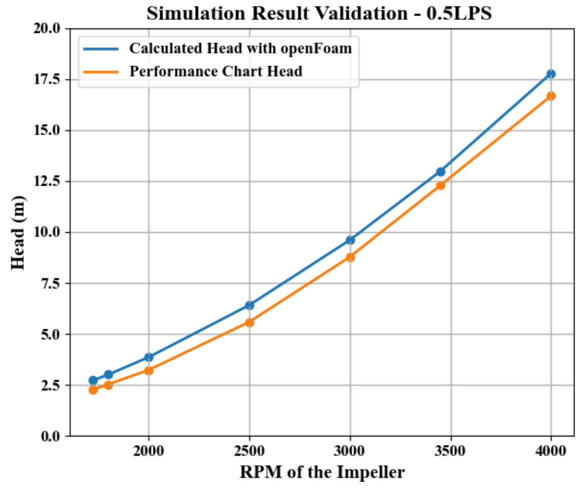


Figure 9: Result validation at 3000 RPM

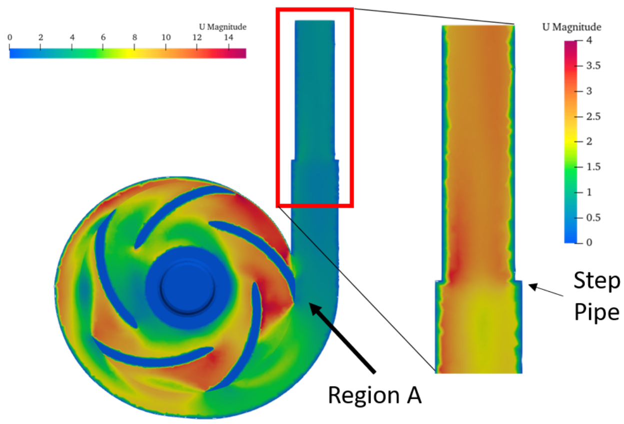


Figure 12: Flow Development at Outlet Pipe

3.2 Flow Development

A steady state simulation is performed to understand the flow development inside the centrifugal pump. The *Figure 10, 11 and 12* shows the pattern of flow development inside the centrifugal pump running at a RPM of 3000 with inlet flowrate of 0.63 LPS. Boundary layers can be seen at both the inner wall of the inlet and outlet section of the centrifugal pump in the *Figure 11* and *12*.

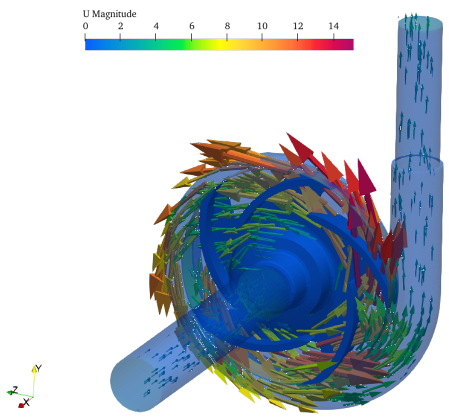


Figure 10: Flow Development Inside the Pump

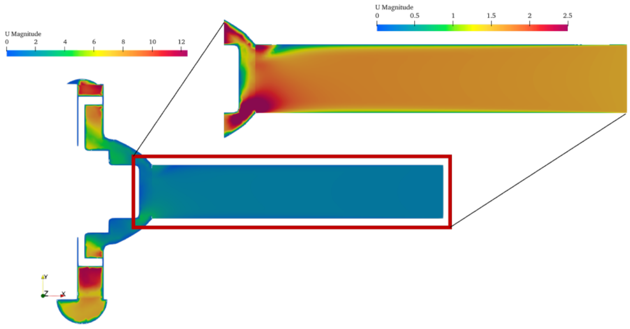


Figure 11: Developed Flow at Inlet Section

The velocity flow field increases from the eye of the impeller till the outlet of the impeller. The velocity then starts to drop down with the opening of the volute passage. The velocity decreases till the flow reaches the Region A. After this, the flow develops and the velocity becomes constant as shown in zoomed section of the outlet pipe. The flow velocity once again increases after it encounters a step in the outlet pipe.

3.3 Pressure Distribution

The *Figure 13* shows the pressure contour plotted for three different inlet flowrate. The decrease in the inlet flow rate leads to the decrement in the local static pressure at the inlet section of the pump. The change in static pressure across the pump domain increases with severity of the blockage level. The minimum static pressure decreases and the maximum static pressure increases.

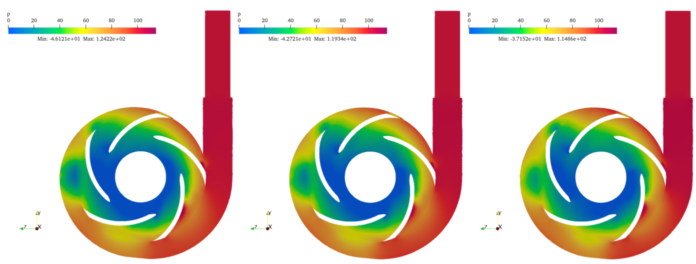


Figure 13: Static Pressure Distribution in the pump (steady-state) left: 0.504 LPS, mid: 0.63 LPS, right: 0.757 LPS

3.4 Velocity Vectors

The *Figure 14* shows the velocity vectors plotted for three different inlet flowrate from 0.504 LPS to 0.757 LPS. The velocity inside the pump gets decreased when the inlet flowrate is decreased. With decrease in inlet flow rate, the maximum flow velocity inside the pump gets reduced from 15.766 to 15.26. Thus, with presence of blockage or foreign materials i.e., blockage leads to decrease in flowrate, the flow velocity inside the centrifugal pump gets reduced.

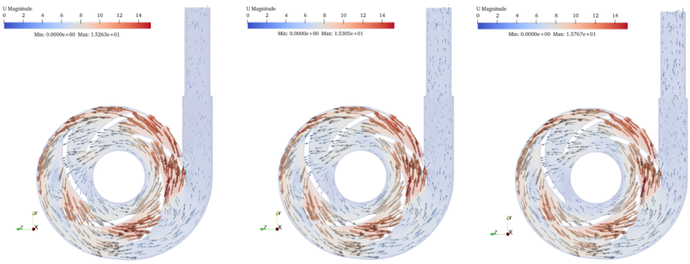


Figure 14: Velocity Vectors in the pump (steady-state) left: 0.504 LPS, mid: 0.63 LPS, right: 0.757 LPS

3.5 Pressure Probes

For transient simulation, two pressure probes are kept at two different locations to capture the pressure signals. The coordinate of the two-pressure probe are provided as follows:

- Probe 1 (-0.0035, -0.041, -0.045)
- Probe 2 (-0.0035 , 0.046, 0)

The *Figure 15* shows the front section of the centrifugal pump where the location of the two pressure probes are annoted. The probe 1 is located near the outlet section of the pump whereas the probe 2 is located vertically just above the eye of the impeller.

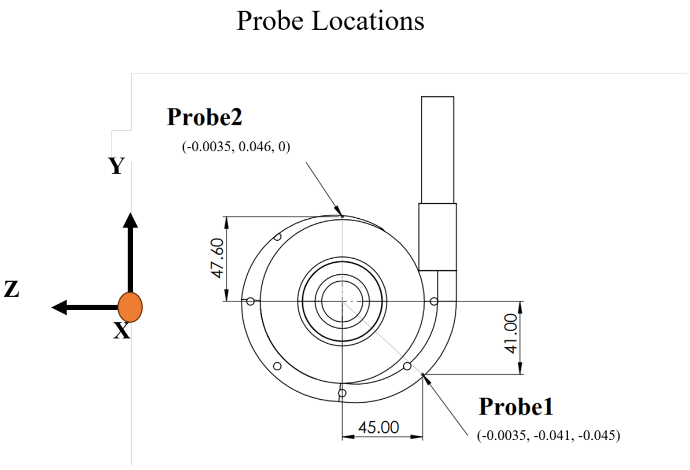


Figure 15: Probe Location

The *Figure 16* shows the plot of static pressure at the two-pressure probe with time. From the plot, it is clearly seen that the pressure is fluctuating about a mean value and the peaks are repeated at regular interval. Since the pump consists of impeller with five blades, the pressure at each probe location spikes exactly five times and repeats again. The time taken for the pressure peak to repeat is roughly about 0.02 seconds. The time averaged pressure value at the probes is:

- Probe 1: 98.2Kpa
- Probe 2: 45.048Kpa

Since, the Probe 1 is near to the discharge section of the pump, the pressure distribution at that regions will be higher than that of regions near the Probe 2.

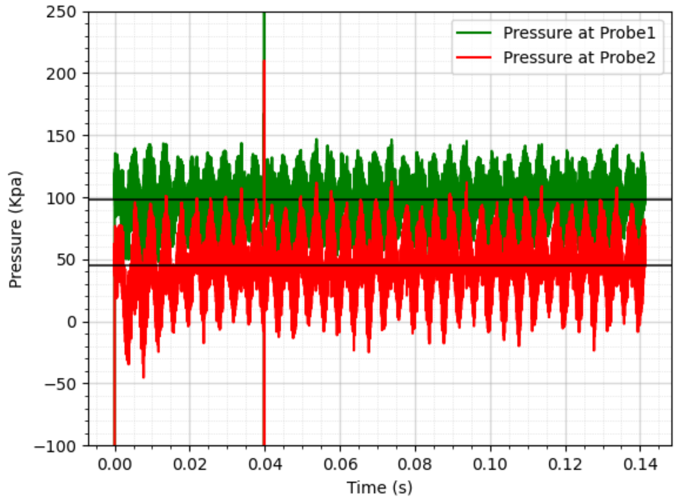


Figure 16: Probe Pressure with Time

3.6 Static Pressure and Flow Velocity with Blockage

The presence of upstream blockage reduces mass flow at the inlet, decreasing flow velocity and static pressure. Steady-state simulations at 1725 rpm assess these parameters, indicating potential cavitation risks. The blockage level at the inlet of the pipe is increased by decreasing the inlet flow rate at a base rpm of 1725. The *Figure 17* shows the plot of the variation of the static pressure with increasing amount of blockage level at the two-probe position and also at the inlet of the pump.

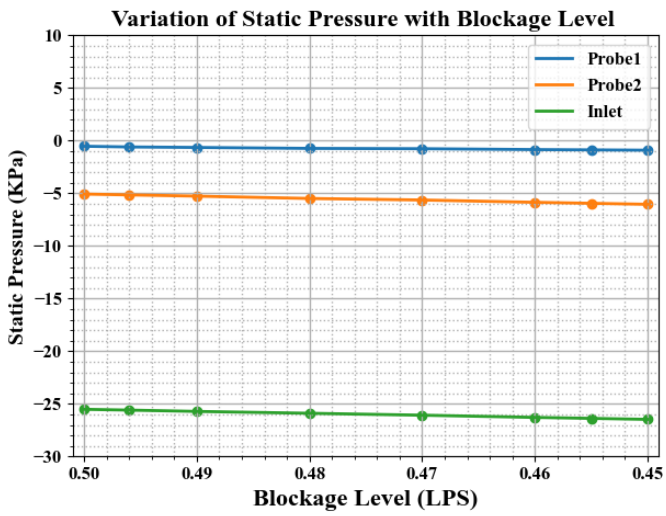


Figure 17: Static Pressure with Blockage

It is evident that the value of the static pressure at all these three positions i.e., at the two-probe position, probe1 and probe2 and at the inlet patch of the pump decreases with the increase in the blockage level.

The *Figure 18* shows the variation of the magnitude of the Y-Velocity at the outlet of the pump. It is seen that the magnitude of the y – velocity at the outlet of the pump decreases with the introduction of the blockage. For a base inlet flow rate of 0.5 LPS, the magnitude of the y – velocity is around 2 m/s whereas with the decrease in the inlet flow rate, this magnitude of y – velocity keeps on decreasing.

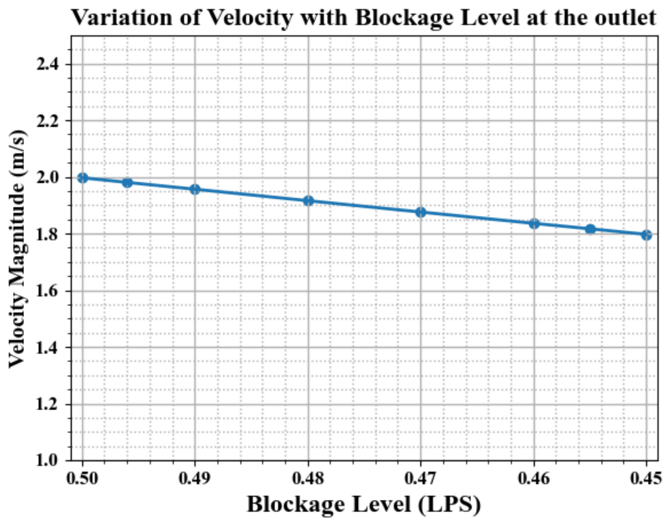


Figure 18: Flow Velocity with Blockage

3.7 Transient Head and Steady-state Head

The *Figure 19* shows the steady-state and transient results for the pressure head H over time t. The simulation is ran at rpm of 3000 for a discharge of 0.504 LPS. The black line y = 9.93 is a steady-state head obtained.The time averaging of the transient head obtained from the simulation gives a mean head of about 9.73 meters shown in red color which is about 2.11 % lower than steady state solution.

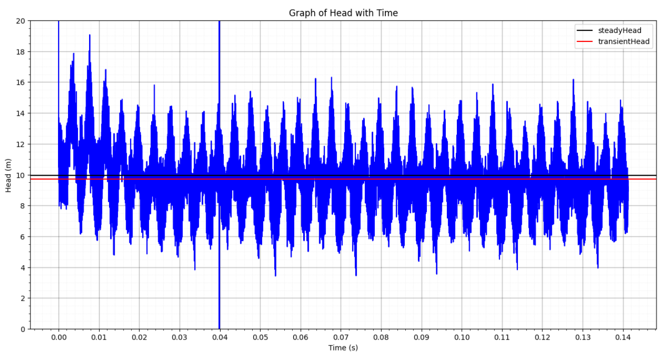


Figure 19: Head Versus Time for 0.504 LPS

4. Conclusion

The open source software OpenFOAM 2012 was chosen to simulate the fluid flow of a centrifugal pump. The model of the centrifugal pump has been created and meshed with both structured and unstructured grid. Steady state simulation of the pump has been performed for various inlet flowrate. The pump performance has been presented in terms of pressure contour, velocity vectors and hydraulic pressure head. With presence of blockage at the intake of the centrifugal pump, the static pressure at the inlet region gets reduced as well as the flow discharge at the outlet gets reduced. The value of the static pressure near the inlet region gets decreased with the successive introduction of blockage at the intake of the pump which in turn increases the change in static pressure across the pump whole domain. In the transient simulation the pressure head fluctuates about a mean head value. The time averaged value of the transient pressure head is approximately equal to the steady-state pressure head.

Acknowledgments

The authors are grateful to the Department of Mechanical and Aerospace Engineering, and Incubation, Innovation and Entrepreneurship Center (IIEC) of Institute of Engineering, Pulchowk Campus for providing the necessary equipment for performing the simulation of the centrifugal pump.

References

[1] Stavros I Yannopoulos, Gerasimos Lyberatos, Nicolaos Theodossiou, Wang Li, Mohammad Valipour, Aldo Tamburrino, and Andreas N Angelakis. Evolution of water lifting devices (pumps) over the centuries worldwide. *Water*, 2015.

[2] S R Shah, S V Jain, R N Patel, and V J Lakhera. Cfd for centrfugal pumps; a review of the state of the art. *Procedia Engineering*, 2013.

[3] Narayan P Jaiswal. Cfd analysis of centrifugal pump: A review. *Journal of Engineering Research and Applications*, 2014.

[4] TAN Lei, ZHU Baoshan, CAO Shuliang, BING Hao, and WANG Yuming. Influence of blade wrap angle on centrifugal pump performance by numerical and experimental study. *Chinese Journal of Mechanical Engineering*, 2013.

[5] Hakan Nilsson, Olivier Petit, and Matin Beaudoin. The ercoftac centrifugal pump openfoam case-study. *IAHR*, 2009.

[6] Shasha Xie. Studies of the ercoftac centrifugal pump with openfoam, 2010.

[7] David A Smith. Modelling cavitation in centrifugal pumps using openfoam, 2016.

[8] Si Huang, Yifeng Wei, Chenguang Guo, and Wenming Kang. Numerical simulation and performance prediction of centrifugal pump's full flow field based on openfoam. *Processes*, 2019.

[9] D.J. Bordoloi, Rajiv Tiwari, and Aakash Dewangan. Blockage and cavitation detection in centrifugal pumps from dynamic pressure signal using deep learning algorithm. *Measurement*, 2020.

[10] D.J. Bordoloi and Rajiv Tiwari. Identification of suction flow blockages and casing cavitations in centrifugal pumps by optimal support vector machine techniques. *J Braz. Soc. Mech. Sci. Eng.*, 2017.

[11] Dhiraj Kumar, Aakash Dewangan, Rajiv Tiwari, and D.J. Bordoloi. Identification of inlet pipe blockage level in centrifugal pump over a range of speeds by deep learning algorithm using multi-source data. *Measurements*, 2021.

[12] Rapur Rapur and Rajiv Tiwari. Experimental fault diagnosis for known and unseen operating conditions of centrifugal pumps using msvm and wpt based analyses. *Measurements*, 2019.

[13] Rapur Rapur and Rajiv Tiwari. Experimental time-domain vibration based fault diagnosis of centrifugal pumps using support vector machine. *ASCE-ASME Journal of Risk and Uncertainty in Engineering Systems*, 2017.