TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

THESIS NO: M-79-MSMDE-2021-2023

Sensitivity Solution of Off-Design Conditions in Centrifugal Pump

by

Sunil Sharma

A THESIS

SUBMITTED TO THE DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTERS OF SCIENCE IN

MECHANICAL SYSTEMS DESIGN AND ENGINEERING

DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

LALITPUR, NEPAL

NOVEMBER 2023

COPYRIGHT

TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING
PULCHOWK CAMPUS
DEPARTMENT OF MECHANICAL AND AEROSPACE EGINEERING

The undersigned hereby certify that they have read, and recommended to the Institute of Engineering for acceptance, thesis entitled "**Sensitivity solution of off-design conditions in Centrifugal Pump**" submitted by Sunil Sharma in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Systems Design and Engineering.

Supervisor, Dr. Tri Ratna Bajracharya
Professor
Department of Mechanical and Aerospace Engineering,

Supervisor, Dr. Sudip Bhattrai
Assistant Professor
Department of Mechanical and Aerospace Engineering,

External Examiner, Er. Sumit Maharjan

Committee Chairman, Dr. Sudip Bhattrai
Head of Department
Department of Mechanical and Aerospace Engineering

Tribhuvan University
Institute of Engineering
Department of Mechanical and Aerospace Engineering
Pulchowk C...

Date: 2023-11-26

ABSTRACT

Centrifugal pumps are most widely used pumps in industry and household usage for fluid flow. During usage of pump, pump speed and discharge may need to be varied. Also during continuous usage of pump, different physical defects may arise. These conditions would create off-design conditions in pump which are studied in this paper. The paper outlines a technique for efficiently obtaining the numerical solution of a pump operating in off-design conditions by leveraging the solution obtained under design conditions. This method is referred to as a "sensitivity solution", wherein small parameter adjustments are made, and simulations are conducted starting from an already converged solution. Test was done for change in impeller speed and it was found that for one percentage change, sensitivity solution was obtained in one-fourth time compared to its baseline simulation. For the flow rate variation, for two percentage change, sensitivity solution is obtained in one-fifth time. Results support use of sensitivity method for solving flows for variations up to ten percent. Variation in output parameter is seen to be higher in case of speed changes compared to the flow rate.

When sensitivity simulation method is tested for defect depth in impeller blade, solution is obtained in one-tenth time compared to baseline simulation for up to 60% increase in defect depth. In case of speed and flow rate deviation, sensitivity simulation time is increasing as deviation from base case is increasing while for defect depth, the sensitivity simulation time do not change significantly for depth changes tested.

ACKNOWLEDGEMENT

I would like to express my deep gratitude to my research supervisors Prof. Dr. Triratna Bajracharya and Assistant Professor and Head of Department, Dr. Sudip Bhattrai for their patient guidance, mentorship and useful critiques to all doubts and queries throughout the period of my research.

I would like to express sincere gratitude Prof. Dr. Laxman Poudel, Coordinator, M.Sc. in Mechanical Systems Design and Engineering. My special thanks to the entire family of the Department of Mechanical and Aerospace Engineering for the constant support and coordination.

I am indebted to Er Bibek Adhikari, Er Sandip Gewali, Er Anup Pandey, Er Amrit Tiwari and Er Sanjeeb Adhikari for their valuable suggestions and ideas to carry out the research work.

I would also like to acknowledge all teachers, friends and faculty members who directly and indirectly provided their help while undertaking the tasks. Lastly, my sincere gratitude goes to my family for their moral support, encouragement and patience throughout my work.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE: INTRODUCTION

1.1 Background

Centrifugal pumps are the most widely used pumps in industries especially in process industries. They are used for transportation of fluids or increasing pressure of fluids. They work on principle of Bernoulli with velocity being diffused to create pressure at the discharge.

A centrifugal pump consists of an impeller which is driven by a motor/driver. The fluid enters at the eye of the impeller and discharges radially. The outer casing of the pump is made in usually volute shape and discharge from the pump is tangential to the impeller. The impeller having blades give the fluid velocity as the impeller spins and blades trap fluid in between them. This speed causes the fluid to move outward as well due to the centrifugal force acting on it. As fluid moves outwards, the region of low pressure is caused near the impeller eye which causes the fluid to be sucked into the pump from the suction pipe. But the pressure at the eye needs to above certain limit to avoid cavitation. As fluid comes towards impeller tip the fluid velocity is increasing. When it leaves the impeller it reaches the wall where velocity is converted into pressure. The velocity decreases along the volute casing which has increasing cross section as fluid moves towards the outlet of the pump, resulting in pressure increase (Bachus & Cusstodio, 2003).

Centrifugal pumps are most widely used pumps in case of industries as they are capable of handling large volumes of fluid. They regulate the process flow which is vital section of many industries. Some of the use cases of centrifugal pumps are power plants, building services, biomedical applications, oil–refineries, industry and water engineering, chemical and process industries, etc. (Karassik & McGuire, 2012).

Like any machinery, continuous use of centrifugal pumps also causes problems in the machinery depending on the fluid transferred and the operation conditions. The major faults that occur in the pump are blockages, bearing failure, impeller failure, excessive vibration, etc. The faults occurring in the pumps can be classified as mechanical, hydraulic or others. This classification helps to pitch the best maintenance effort and thus help reduce the failure of the pumps.

In industries, the major task to perform for the pump to operate in sound condition is maintenance. Scheduled and preventive maintenance are the most common maintenance methods prevalent. Condition monitoring and the prediction of faults helps to keep the operating of the pump in good condition and the increase the longevity of pumps. As with the maintenance efforts increase the cost of maintenance increases. These maintenance activities also take a lot of time and energy. This increases down time in the industry. This problem may be solved by using sensors that collect data about the pump operation. The data collected can be processed and analyzed to find out if any fault exist on the pump.

This research is linked to Tiwari et al. (2020) to create simulation data that can be fed into machine learning algorithms for prediction of fault and severity. Geometry of the pump used in the simulation is going to be the pump being used in the Machine Fault Simulator (Rapur & Tiwari, 2017, Tiwari et al. 2020). These studies have created experimental data, a larger portion of which to feed to machine learning algorithms for training them and then testing the prediction based on the remaining data. The results have found good accuracy in selecting the faults appearing.

CFD is powerful tool that is used to solve the flow of fluid in complex situations. While proprietary CFD software often offers user-friendly interfaces, it has a drawback in that its underlying code is not accessible, which limits users' ability to customize models and algorithms to meet their specific needs. In contrast, OpenFOAM, an object-oriented CFD software with open-source origins, not only offers a range of standard solvers but also gives users the freedom to access and modify the source code.

Time for simulation is often limited by the computational resources available. If a larger data is to be extracted, then simulation is also costly and time consuming. Techniques need to be used so as to reduce the time.

Sensitivity solution technique is used for reducing the simulation time for a problem when one of the variables of the original solution is varied by small value. This is different from sensitivity analysis where full simulation is obtained but in reduced time compared to the original simulation. Sensitivity solution methods are used in aerodynamic applications. This research works on the numerical model of the pump to use sensitivity methods for creating larger dataset. For this open source software OpenFOAM v2212 is used.

1.2 Research Gap

Since centrifugal pumps are the heart of any process industry, their operation can be halted by any faults. Condition monitoring and preventive maintenance help to maintain this. But, they alone cannot predict the severity of faults existing inside of the pump. Besides that, maintenance and traditional prediction based maintenance activities seldom can be done while the pump is in running condition. There is availability of literature and manuals from manufacturers for such type of maintenance or checking of the pump for the faults. But, the use of other methods of data collection and their use with the help of machine learning is a new field of study. The study done till now are based on experiments which require a lot of cost and time for operation. If data needs to be extracted based on the small change in faults, experimentation results in tedious repeated tasks. The small change in the fault parameter thus can be simulated. Simulation is also useful to understand the off-design operation of the pump since some pumps need to operate under varying condition. Use of simulation also becomes tedious and costly if there are more variables or the extent of a variable is more.

Sensitivity techniques for finding full solution is still not widely used in case of pumps. This paper aims not to create a specific sensitivity solver for solving the fluid flow in the pump but to exploit OpenFOAM, which can be used for sensitivity simulation. This research tries to solve the problem of the data extraction for the different design flows and faults for centrifugal pump. Method of sensitivity is used so as to reduce the computation time associated with the extent of a variable affecting the output variable of the pump. Previous researches did the work to divide the fault severity into classes or levels but not directly quantifying the effect of fault. This research helps solve this problem of fault quantification. The data output from the research can be used to check the fault level using an appropriate algorithm. This in turn helps for optimum maintenance time or activity.

1.3 Objective of Research

1.3.1 Main Objective

To perform Sensitivity solution of off-design conditions in Centrifugal Pump.

1.3.2 Specific Objectives

- To prepare case setup of the pump and run simulation for design conditions.
- To perform baseline and sensitivity simulations for different pump speeds, flow rate and defect geometry.
- To find maximum sensitivity uncertainty parameter for the speed, flow rate and fault.
- To create low cost data generation for off-design conditions in centrifugal pump.

1.4 Assumptions and Limitations

The limitations for this research are listed as follows:

- The research is limited to numerical analysis.
- Limited experimental data available for the validation of the simulation data.
- The time/iterations for convergence of a solution depended upon the number of cores to which domain was decomposed. To reduce this variation, all simulations were performed on same CPU and same number of decomposition. Some variations were seen despite this effort.

1.5 Organization of Report

This report consists of five chapters that covering the research on sensitivity solution of centrifugal pump. Mentioned below are the chapters along with their main features.

Chapter 1: Introduction

This chapter includes information about the centrifugal pump working, its faults, use of CFD methods for pump, gap in research and objectives of the research.

Chapter 2: Literature review

This chapter explains the theories and previous studies related to this research. It also includes the description of the solver software that is used in the research.

Chapter 3: Methodology

This chapter shows the methodology that has been adopted in the research in order to achieve the objectives. The geometrical information and boundary conditions used for solving the numerical model are presented.

Chapter 4: Results and Discussion

This chapter illustrates the findings of the activities that are performed within the research frame including the flow field, pump performance and validation and comparison of baseline and sensitivity simulation times. The chapter also discusses on the explanation and validation of the outcomes.

Chapter 5: Conclusions and Recommendations

This chapter presents the conclusion of the research of sensitivity solution of off-design conditions in centrifugal pump using OpenFOAM. Recommendations for the further use of research and ways to further speed up sensitivity simulation are provided.

Appendices contain the pump drawings and performance curves.

CHAPTER TWO: LITERATURE REVIEW

2.1 Centrifugal Pump

Centrifugal pump is a type of rotodynamic pump which transfers fluid not by the direct displacement of the fluid, but by the dynamic forces acting on the fluid. The pressure head produced by the pump depends on many factors like speed of rotation of pump, casing and fluid properties.

Centrifugal pump works in basic principle but the flow analysis in centrifugal pump is complex due to the presence of unsteadiness, secondary flows, cavitation and turbulence. The use of computational method for analysis and its progress has increased recently. Researchers have used Computational Fluid dynamics (CFD) technique for prediction of pump performance for the design and off-design conditions, parametric effect in pump operation, cavitation prediction and effects, etc. For solving the flow analysis of centrifugal pump, unsteady Reynolds- averaged NS equations along with k-epsilon turbulence model is found to be appropriate (Shah et al. 2013).

2.2 Centrifugal Pump Faults and their Detection

There are different failure modes occurring in the centrifugal pumps and categorized them into hydraulic, mechanical and other failure modes explaining the causes for them as well. The major failure modes are bearing failure, erosion, cavitation, pressure pulsations, axial and radial thrust, blockages, corrosion, suction and discharge recirculation and seal failure (McKee et al., 2011). Wang et al. (2022) shows that the the erosion on blades due to sand mixed water occurs on trailing edge of pressure surface and near the blade inlet.

Continuous usage of centrifugal pumps without maintaining at high speeds usually results in failure of the mechanical components and sometimes cause critical damage to whole pump assembly. The failure modes can that appear are mechanically induced, operationally developed, system faults, or any combination of these. Failure occurring due to faulty parts like Misalignment of parts, bent rotor or bearing failure fall under mechanically induced faults. System faults encompass leakage and faulty installation. Operationally developed faults develop during the running condition of pump, for example, cavitation, blockages and flow related problems. This study collected vibrational data from healthy and faulty centrifugal pumps and diagnosed the presence of the fault type and the chosen the level of fault using Support Vector machine. The high prediction accuracy for the distinct and multi coexisting faults provided by the Support Vector Machine is found promising for centrifugal faults diagnosis, mainly in industrial use (Rapur & Tiwari, 2017).

Kumar et al. (2021) used multi source data to investigate the blockage level in suction pipe of centrifugal pump. The data that were used in the investigation was motor, vibrational data and pressure measurement. The data produced from different sources and on different severity levels were used to train a deep learning algorithm which after training would predict what type of the fault it was. The results of the research were found to be promising as the algorithm can predict the if there was blockage in inlet pipes and severity levels defined during experiments.

Chen et al. (2021) also studied fault detection in the centrifugal pumps using the vibration data. The faults for which the study was done included machine seal failure, cavitation and impeller damage. The results show 82% accuracy for the prediction of different types of faults using machine learning method. The experimental verification was done using a centrifugal pump operating at 3 levels of flow rate: design flow rate, and 0.8 and 1.2 times the design flow rate. The vibration data collected and processed to extract its features and compared with test results. Mahalanobis distance showed higher accuracy to predict the faults of centrifugal k-nearest neighbor pump.

Tiwari et al. (2020) introduced a method for identifying blockage and cavitation in centrifugal pumps, assessing their severity using pressure signals, and leveraging deep learning algorithms for this purpose. The authors extracted features from the time-domain pressure signals and conducted extensive experiments to identify the most effective combination of four features, maximizing classification accuracy. They also fine-tuned hyper parameters to further enhance the classification accuracy. Their visual observations revealed that cavitation tended to be more prevalent at higher pump speeds and increased blockage levels. Additionally, they noticed that the performance of the data classifier improved as the fault severity increased. These observations suggest that the algorithm successfully predicts the severity of cavitation and blockage faults. However, the effectiveness of this technique relies significantly on the sensor's ability to capture the signal accurately, which, in turn, depends on the sensor's placement or location.

Fu et al. (2014) present a study that delves into both experimental and numerical investigations concerning the flow instabilities and cavitation phenomenon occurring in a centrifugal pump when operating at low flow rates. A three-dimensional (3D) numerical model is employed to simulate the internal flow within the pump, encompassing extended portions of the inlet and outlet ducts. Cavitation was observed to manifest across a broad spectrum of low flow rates, resulting in a characteristic head-drop curve with a creeping pattern. The cavitation took shape in non-axisymmetric forms, primarily attaching themselves to the suction sides of the pump blades. As anticipated, the occurrence of these cavities was found to be contingent upon the pump's flow coefficient and cavitation number. The experimental aspect of the study focused on visualizing the internal flow patterns using high-speed digital recordings and analyzing the pressure pulsations near the impeller's eye through fast response pressure transducers. The experimental findings revealed that the unsteady behavior of the internal flow in the centrifugal pump, especially when operating at low flow rates, exhibited distinct low-frequency oscillations. Furthermore, under specific conditions, these low-frequency pressure fluctuations were closely linked to the flow instabilities triggered by cavitation phenomena at low flow rates. Ultimately, the hydraulic performance predictions for the centrifugal pump obtained through numerical simulations demonstrated a high degree of agreement with the corresponding experimental data.

2.3 Simulation and Sensitivity Solving

Computational Fluid Dynamics (CFD) research aims to push the boundaries of practical engineering applications into "non-traditional" domains. Balancing the requirements for computational flexibility and seamless code integration can be challenging. To address this, a shift in coding approach is proposed, emphasizing object orientation, library components, and equation mimicking as a way forward. OpenFOAM, a C++ object-oriented library for Computational Continuum Mechanics (CCM), is introduced as a solution to achieve efficient and flexible implementation of complex physical models. This is accomplished by emulating the form of partial differential equations within the software, providing code functionality in a library format. The open-source nature of OpenFOAM allows users to achieve the desired versatility in physical modeling without sacrificing support for complex geometry and execution efficiency (Jasak, 2009).

With the continuous advancement of computational technology, CFD has gained widespread applicability in both industrial and non-industrial sectors (Sakran, 2015). Proprietary CFD software, while offering user-friendly interfaces, often restricts access to the underlying code, limiting users' ability to customize models and algorithms to their specific needs. In contrast, OpenFOAM, an object-oriented CFD software with open-source roots, not only provides a suite of standard solvers but also empowers users to explore and modify the source code, allowing for extensive customization (Li et al., 2009). This adaptability and transparency have led to a surge in research exploring OpenFOAM's application in the study of fluid dynamics within pump machinery.

One of the solvers available in OpenFOAM is SimpleFoam, designed for steady-state simulations of incompressible, turbulent flow using the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. This solver employs a segregated solution approach, where equations for each system variable (velocity, pressure, and turbulence-related variables) are solved sequentially. The non-linearity arising in the momentum equation is addressed by computing it based on the velocity and pressure values from the previous iteration. This approach maintains the coupling between the momentum and pressure equations, preventing the occurrence of high-frequency oscillations in the solution, often referred to as the "checkerboard effect." The iterative solution process begins with the momentum equation, yielding a velocity field that may not initially satisfy the continuity equation (i.e., it may not be divergence-free). Subsequently, the momentum and continuity equations are used to construct a pressure equation with the goal of obtaining a pressure field that, when inserted back into the momentum equation, yields a divergence-free velocity field. After correcting the velocity field, the equations for turbulence are solved. This iterative procedure is repeated until convergence is achieved.

Huang et al. (2019) shows the results of steady-state and transient numerical simulation done using OpenFOAM 5.0 for solving full flow field of a pipeline centrifugal pump (specific speed= 65) in flow rate range 0.3 Q_d ~1.4 Q_d (Q_d is design flow rate). The turbulence models chosen for the flow governing equations were standard k-ε and k-ω

SST (Shear-Stress Transport). The solvers used were, SimpleFoam for steady state solving and pimpleDyMFoam for transient calculations. The resulting pressure, flow velocity and streamlines were displayed in ParaView and relationship between the vortex and the hydraulic loss in the pump were analyzed. Based on flow fields from simulation, the pump performance parameters like input power, efficiency and head were calculated. Pump performances predicted by OpenFOAM were compared with results from Ansys Fluent for the same calculation model, grids and boundary conditions. It was found that in current case, prediction of pump parameters OpenFOAM had higher accuracy.

In the realm of turbulent flows, the k-ε model emerged as a widely adopted choice for internal flows, offering an efficient means of simulating engineering-related fluid behaviors (Nallasamy, 1987). Notably, the investigation conducted by Liu et al. (2012) ascertained that the conventional k-ε turbulence model outperformed the RNG k-ε and k-ω SST models when predicting the performance of centrifugal pumps. In contrast, Shojaeefard et al. (2012) demonstrated that the k-ω SST model yielded notably precise evaluations of near-wall flow, exhibiting superior accuracy over the k-ε model in a centrifugal pump simulation.

In case of simulations that have changing parameters, there is method of sensitivity solving which helps to solve small changes by resuming iterations from the results of primary or baseline simulation. The use of the sensitivity solver helps reduce the computation costs but that depends on the number of variables (Bhattrai et al. 2018).

Standingford & Forth (2003) developed a sensitivity solver using automatic differentiation tool to create forward sensitivity version of Flite3D. The validation of the work is done by comparing the sensitivities of lift, drag and side force with respect to the angle of attack, calculated using sensitivity solver and central differencing.

Bhattrai et al. (2018) presents an efficient method for quantification of the uncertainty using the surrogate modeling based on the gradient-enhanced kriging. The paper demonstrates the effectiveness of the proposed methodology on assessing uncertainty through a case study of a hypersonic trailing flap edge. However, the same methodology can be used to assess the uncertainty quantification in case of the pump with faults. The use of gradient enhanced kriging in the uncertainty quantification allowed for a faster convergence of the results. The proposed method provides a promising approach for efficient uncertainty quantification of a complex system. The method can help reduce the computational cost and time required for uncertainty quantification.

In research by Salehi et al. (2018), authors explore the analysis of uncertainty quantification concerning a centrifugal pump characterized by a low specific-speed design. The investigation encompasses both operational and geometrical uncertainties as factors of consideration. The primary focus of this study is the evaluation of the pump's performance. It reveals that the probability density functions (PDFs) representing the head coefficients demonstrate a relatively flat pattern, whereas those for hydraulic efficiency exhibit more pronounced bell-shaped distributions. It's worth noting that, throughout the study, the observed head coefficient values for various flow rates consistently fall within the range of non-deterministic PDF values. The examination of the pump's robustness underscores that variations in the head coefficient, resulting from assumed uncertainties, are notably more substantial compared to fluctuations in hydraulic efficiency. Geometrical uncertainties are found to have a minimal impact on the pump's head, with rotational speed emerging as the primary influencing factor. Furthermore, the research highlights that geometrical uncertainties play a pivotal role in affecting efficiency variations, particularly at the optimal efficiency point. However, their significance diminishes with higher flow rates, where the importance of flow rate and rotational speed becomes more prominent.

CHAPTER THREE: METHODOLOGY

The research is advancing on the following methodology.

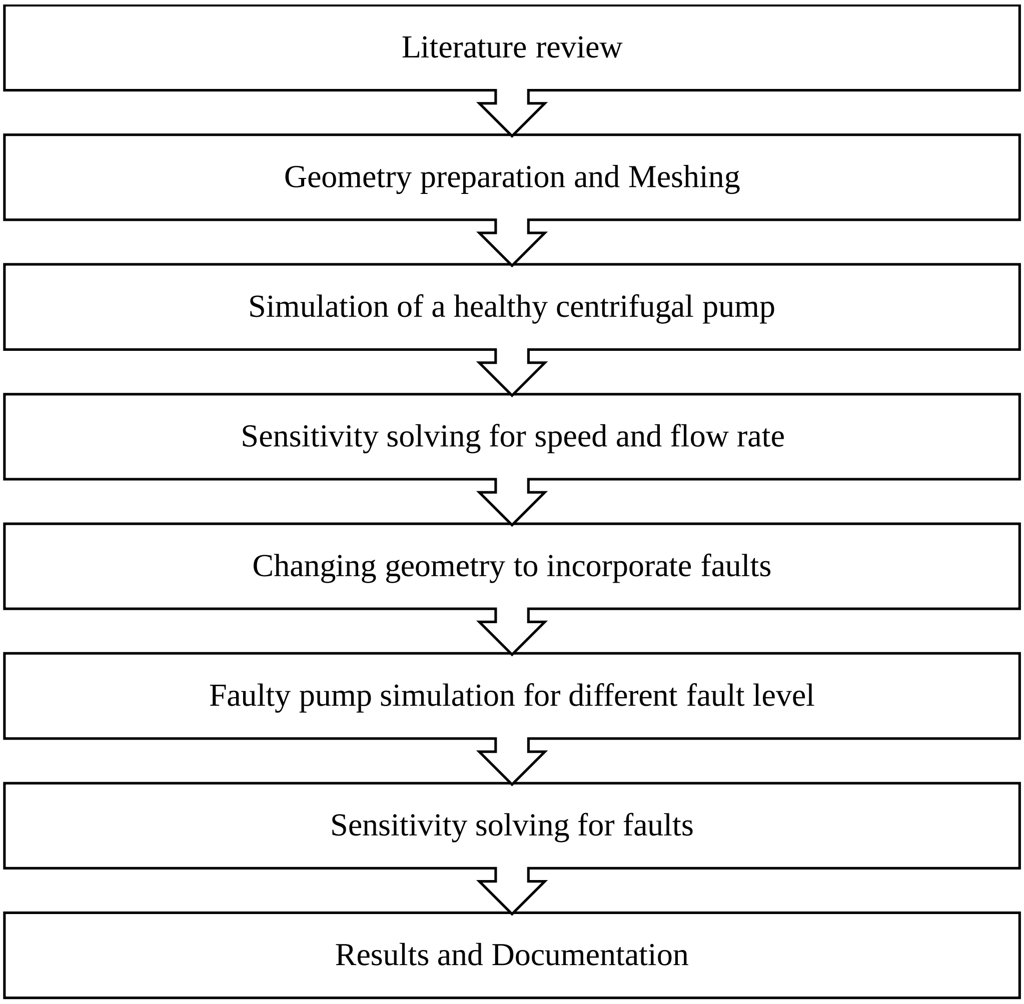


Figure 1: Flowchart for Research

3.1 Literature Review

Literature review for the research is carried out from different internet sources, books, articles and journals. Review of literature for the research is continuous work that has been carried out during the full research period to find out the methods of solving, output comparison, verification and validation. Since the research contains simulation related activities, internet based forums and pages are extensively used as well. Literature review has helped in extending knowledge for the field as well as find the suited methods for data extraction.

3.2 Geometry Preparation and Meshing

The centrifugal pump geometry going to be used in the research is the same as that of the used for researches of Tiwari et al. (2020) and Kumar et al. (2021). Their experimental setup is shown in Appendix. The pump was used on Machine Fault Simulator for the experiments related to faults that may occur in centrifugal pumps.

3.2.1 Geometry of Pump and Preparations

The pump that is used for this study is 60P series pump by Oberdorfer pumps. It is a small bronze centrifugal pump that is designed for pulley drives. The pump assembly consists of rotating impeller, volute casing and cover plate on the inlet end. The impeller is semi-open type and has five blades. The shaft at the other end would be coupled to pulley and belt driven by motor. Rotational speed of is 1725 to 3450 rpm.

Figure 2: Pump geometry (Source: Oberdorfer pumps)

From the obtained CAD file of the pump, fluid domain for the simulation is prepared using Boolean methods. CAD software Solidworks and Catia has been used for preparation of the geometry. During computational domain preparation, small parts inside of the fluid domain which posed difficulty during meshing and simulation are removed. Removal of sharp corners and small parts could help in easy meshing and prevented from making relatively small mesh elements and elements with higher aspect ratio. The inlet and outlet pipes of the pump are extended to four times the diameter of the corresponding pipes for providing the length for flow development.

3.2.2 Meshing of the Fluid Domain

Meshing is the process of dividing a complex continuous domain into small, simple geometric elements to assist numerical analysis. Mesh is prepared using algorithm and there are software to do it as well. Often manual input is required as parameters depending on type of the elements and size wanted. Finer mesh is usually associated with more accurate results but the finer it gets the requirement of the computational resource is higher.

The meshing of the fluid domain is primarily done in Ansys Mesh. The mesh is saved and converted into the format used in OpenFOAM. Mesh prepared is mainly unstructured with tetrahedral elements in the main flow region and volute section and structural grid is prepared in the inlet pipe. Refinement is done near the rotating surfaces so as to capture the flow details around moving walls similar to (Caruso & Meskell, 2020). The different mesh created are made in such a way that the mesh metrics like non-orthogonality and skewness are taken into account. Meshing is performed by dividing the geometry into three separate zones: inlet zone, impeller zone and volute zone; and merging the meshes into one for simulation. Inlet and outlet of the pump are extended to four times their diameters so as to provide path for flow development and prevent effect of boundary conditions in fluid domain simulation (Alemi et al. 2014).

Inlet section is cylinder and structured mesh with majority of hexahedral elements is prepared in this domain. The pump is tested with unstructured mesh in input as well and found to have no significant difference in output parameters and the convergence time or similarly numbered structured mesh. Number of element in inlet region in chosen mesh is 20,700. The volute and impeller regions have thin and irregular geometries in which structured grid is tedious to produce. So, unstructured grid with majority of tetrahedral elements is produced in volute and impeller region. Impeller region has 200,053 elements and volute region has 110,926 elements.

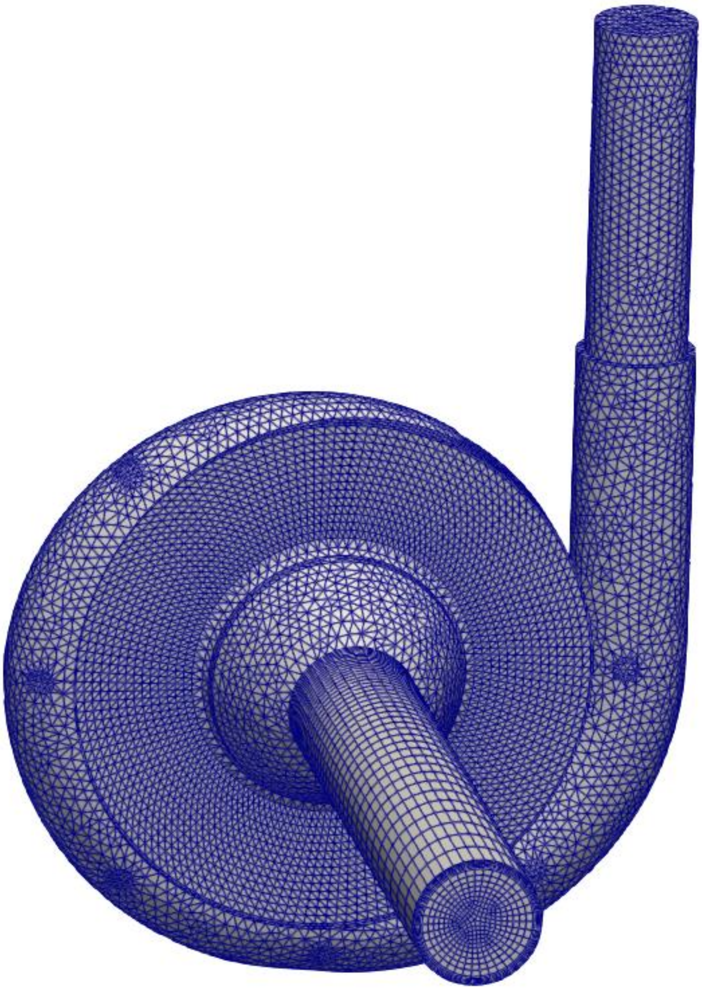


Figure 3: Fluid Domain Mesh

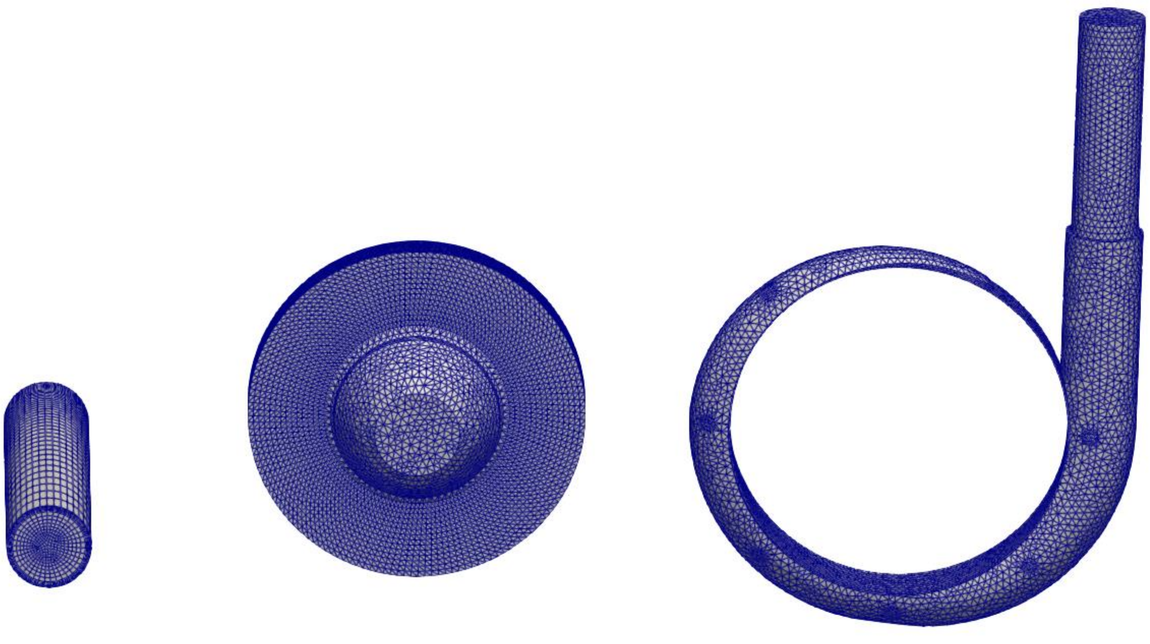


Figure 4: (from left) Inlet, Impeller and Volute Domain

Reynold's number of the flow in the inlet pipe is around 35,000 which in the regime of turbulent region. The dimensionless number, y+ is checked in the flow. In this study for chosen mesh, the y+ values are within the range of 30 to 330 with the rotating regions having on average 200 and inlet pipe has lower y+ values as compared to the volute and impeller regions. Log law is satisfied within the value of y+ in range 30~500 (Versteeg & Malalasekera, 2007). For this y+ values obtained, mesh is considered to be resolved near walls (Caruso & Meskell, 2020).

3.2.3 Grid Independence Test

Grid convergence study is done to find out the required minimum number of the elements for the simulation so that the results do not differ based on the mesh. Mesh independence test was done for different grids and head, torque on impeller and force along x-axis on impeller of the pump is checked as the output variable. Salehi et al. (2018) has tested grid sensitivity checking the head coefficient and efficiency in the pump model and assumed converged grid for 0.4% difference in head coefficient and 1.5% on the efficiency. The mesh is assumed to be grid independent when checked in three grids and less than 2% variation is considered in output quantities (Caruso & Meskell, 2020). Variation targeted in this study is around 0.5% for the output parameters which is less if not equal to the variation targeted by similar studies.

In this study, pump is run under 3000 rpm and 0.631 lps according to boundary conditions described in later topic. It is found that when number of elements increased from 3,00,000 to 3,30,000, output parameters changed around 0.5% and when changed from 3,30,000 to 3,80,000 elements, the difference in the output variables (force, torque and total head) is found less than 0.1% (Appendix). The finer mesh is chosen among 300,000 and 330,000 so as to get accurate solution. The mesh is with 330,000 elements is considered to be grid independent and further work on refining mesh has no significant effect on the parameters that are studied in the paper.

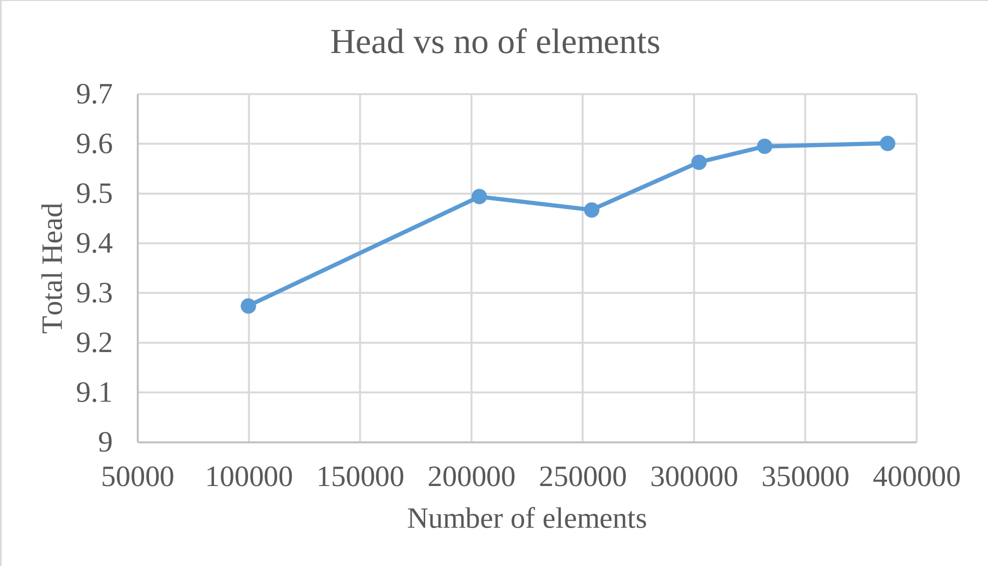


Figure 5: Total Head in Grid Independence test

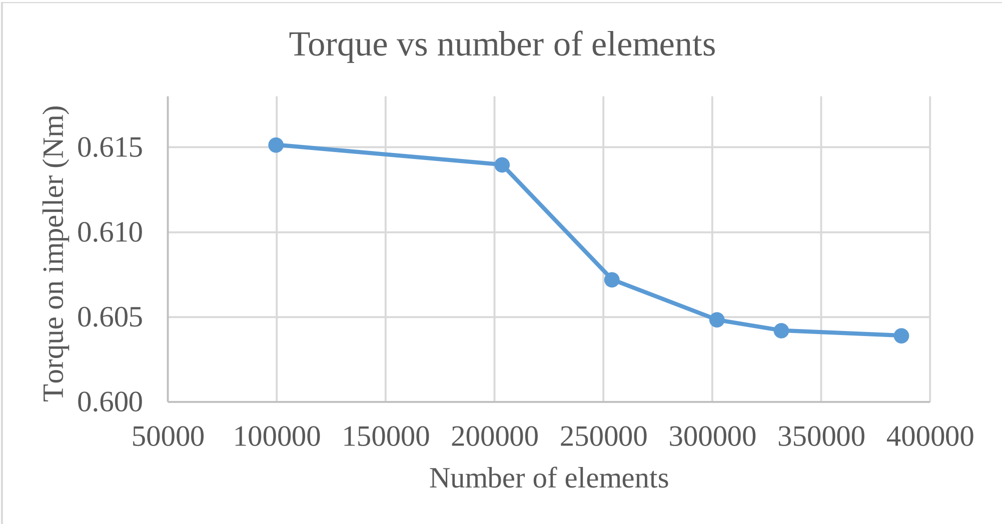


Figure 6: Torque on impeller in Grid Independence test

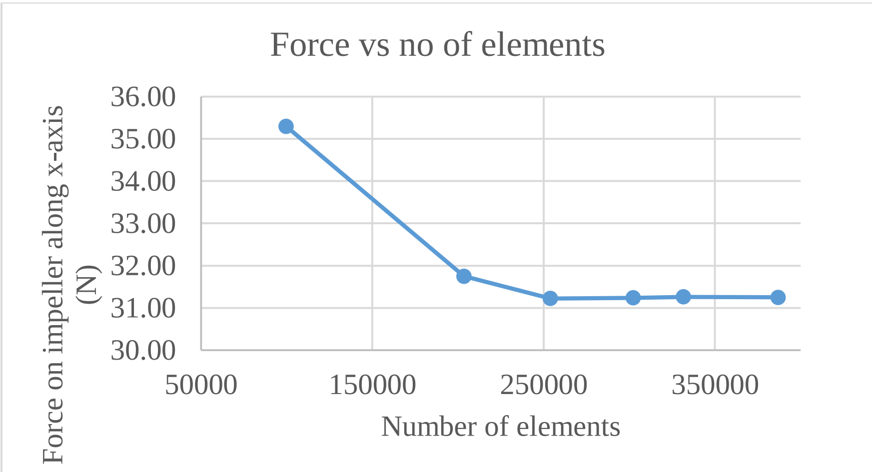


Figure 7:Force (along x) on Impeller in Grid Independence test

3.3 Simulation of a Healthy Centrifugal Pump

The prepared mesh is sent to the next step of the simulation done in OpenFOAM. Initial and boundary conditions is set as per requirement and nature of the work. Both steady and transient study are done in the study. It was expected that k-epsilon turbulence model matches the results more closely similar to Huang et al. (2019). So, k-epsilon Turbulence Model is chosen as per the convergence set. Since it is a water pump, the fluid properties are water are provided.

3.3.1 Boundary Conditions

Mass flow rate is defined at the inlet of the pipe which gave the inlet velocity. At the outlet the pressure is set to zero-gauge pressure. The walls are set as stationary no-slip wall. The impeller was set as rotating no-slip wall The impeller zone is set as rotating fluid zone with same speed as the impeller. Interfaces are defined for common faces of inlet and impeller zone and impeller and volute zone. Boundary values for the Eddy viscosity ratio was set to 10 and Turbulence intensity at 5% (Alemi et al. 2014) which gave values for Turbulence kinetic energy (k), Turbulence dissipation (ε) and specific turbulence dissipation (ω). Flux transfer between rotating impeller and stationary volute and inlet mesh is done using Arbitrary mesh interface for both transient and steady cases.

3.3.2 Pump Characteristics

The major output quantities to be monitored were the pressure at the probe locations, head developed by the pump (H), input power, and efficiency(η). These quantities will be evaluated in simulation and compared for validation. These quantities are defined as follows:

$$H = \frac{P_{out} - P_{in}}{\rho g} + h$$

$$Power = Tw$$

$$\eta = \frac{\rho g Q H}{Power}$$

In above equations, P_{out} and P_{in} represent the area averaged total pressure of the pump at the outlet and inlet respectively, h is the vertical distance from outlet to inlet, T is the torque acting on the surface of the impeller vanes, w is the angular velocity of the impeller given by $(2\pi N/60)$ and Q is the volumetric flow rate through the pump. Fluid being pumped is water at normal conditions.

3.3.3 Steady Simulation

Steady state simulation of the pump is done in OpenFOAM using SimpleFoam solver. Method used is frozen rotor method based on Multiple Reference Frame (MRF) approach implying zero relative mesh motion between stationary and rotating fluid zones. Impeller was set as movingwallVelocity. Impeller domain is set according to MRFProperties file in OpenFOAM. For convergence of steady case, residual control is set to 10^{-3} for pressure and10^{-4} for velocity, k, omega and epsilon.

3.3.4 Transient Simulation

Transient simulation is done in OpenFOAM using pimpleFoam solver which is based on PIMPLE (Pressure Implicit with Splitting of Operators) algorithm. Finite volume schemes are same as that set for the steady state simulation. Boundary conditions is same as that for the steady state simulation. Impeller's computational domain is defined through dynamicMeshDict file, utilizing the sliding mesh technique. Time step is established at $5*10^{-5}$ second, same as Huang et al. (2019) and which is also the time between the readings for the pressure probe in Tiwari et al. (2020). Pressure probe are set in the locations as set in same experiments. The locations of the pressure probes in shown in appendix. Time step was set to adjustable and maximum Courant Number is set to 1 for simulation so as to maintain numerical stability. Maximum of 50 iterative loops were allowed for each time step to obtain a solution.

3.3.5 Validation of results of healthy pump

The outcomes that is obtained from steady and transient simulation are then validated with available data. The transient data is compared with the results from previous work on the pump. The steady simulation results performed are compared with the performance curve available from the manufacturer's data sheet. Also the results are compared with the results from Ansys Fluent under same boundary conditions.

3.4 Sensitivity Solving for Speed and Flow Rate

Sensitivity solving involves changing uncertainty parameter so that time for solution is less and the parameter change from base case is as large as possible. This optimization is done by hit and trail method. The problem is to find the optimum value of the uncertainty parameter in the sensitivity solving so that the number of baseline simulations for the can be decreased and reasonable estimates for the output variable is obtained. This is done for the speed change of the pump. For this, steady simulation of the pump is run at a certain speed and the converged solution is obtained. From this converged solution, speed is changed near the initial speed and new simulation is run. The speed change is set in batches and maximum speed change which can be solved in low time compared to that can be obtained using the converged solution is found. Similar procedure is done by changing the flow rate through the pump. While performing sensitivity solution, it is checked whether the results in output parameters vary from that of the baseline simulations.

3.5 Changing Geometry to Incorporate Faults

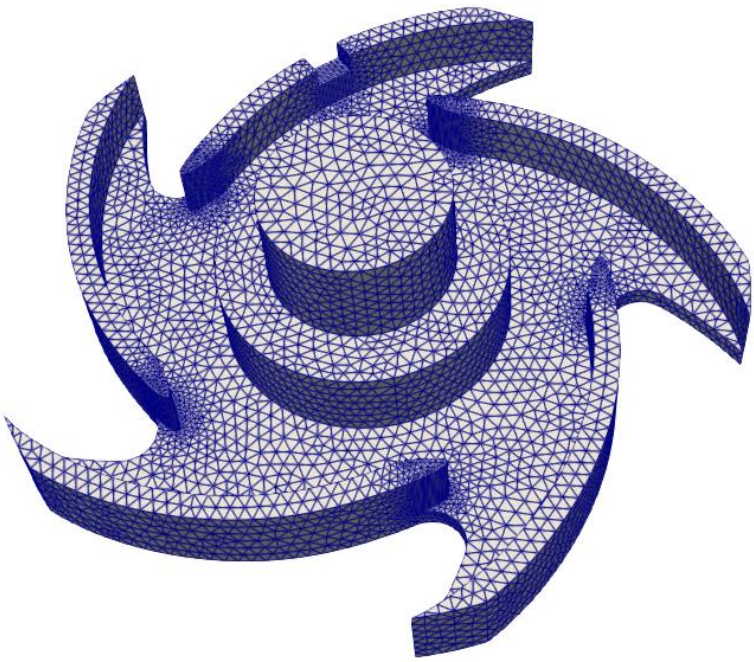


Figure 8: Faulty impeller blade showing defect in topmost blade

The next step in the research is to tweak the geometries of the pump to incorporate the impeller fault. The fault generated is of the following type. A 5 mm width portion is cut out on a blade and is given a certain depth. The depth is the input parameter for which the sensitivity simulation is performed. The fault generated may not match the faults that are usually created during pump life but this fault chosen here can give general idea about the flow disturbance due to a fault and the fault is also easy in preparing the mesh for sensitivity simulation. One geometry is prepared with fault on single blade only while other geometry is prepared with identical fault on all blades.

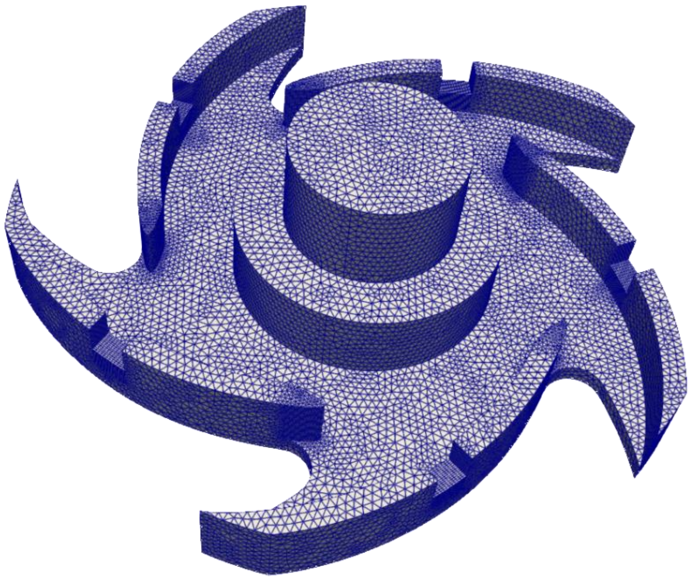


Figure 9: Impeller defect on all blades

3.6 Impeller Fault Simulation

After geometries are prepared for impeller fault with different severity, simulation is done for each of them to extract the pressure data at output. Due to the requirements related to sensitivity solution, the impeller region could not be directly meshed but prepared into three regions and then they are merged together in the simulation. Three regions are crack, enclosure and the impeller. The enclosure region is inside the impeller region and crack region resides inside of the enclosure region. The impeller is composed of unstructured mesh while the enclosure and crack regions have structured mesh prepared in such way that number of elements in the mesh and each face do not change while the crack depth (geometry) changes.

After meshing, boundary conditions are applied similar to earlier case. These simulations are the baseline simulations for the next step of the research, the sensitivity solution for the faults.

3.7 Sensitivity Solution of Impeller Fault

Uncertainty parameter of fault (depth) is decided for each of the fault and its value is initially supposed up. The small change in the severity of faults is simulated from the converged solution. The goal here is to reduce the number of iterations for the small change in faults so that the number is quite small compared to that of the number of iterations required for the convergence of solution in the baseline simulation. For this, a batch analysis is done, by selecting list of number of iterations that the solution for sensitivity solution is supposed to be converged. From the results of the sensitivity solution, the number of iterations required for the sensitivity solution is compared to that of the number of iterations required for the baseline simulation for the fault. From the comparison, the maximum parameter change is aimed to determined which can be simulated from sensitivity solving in less time compared to baseline simulation.

3.8 Results and Documentation

The results of the research are validated as available. Findings are addressed and properly documented. Conclusions are drawn as per the findings of the study. Recommendation are placed so as to be helpful for future works extending the research. The documentation is done as per the requirement of the Department of the Mechanical and Aerospace Engineering.

CHAPTER FOUR: RESULTS AND DISCUSSION

4.1 Pump Flow Field and Characteristics

Base speed for the pump case is 3000 RPM and run at a volumetric flow rate of 0.631 lps. Fluid pumped is water in viscous case in k-epsilon model. To see the effect of pump speed variation in the flow, pump is run at various speeds in steady state cases. The cases are run till convergence and the result at the converged time is discussed in the following figures. The flow fields of the different cases are viewed in ParaView. The results of 2700, 3000 and 3300 RPM are discussed below and velocity vectors and pressure distribution for the same are plotted in figures.

Pump characteristics discussed and plotted here are input power to the pump and the efficiency. These are plotted for speed ranges from 2600 RPM to 3400 RPM.

The pressure field shows lower static pressure around the inlet side. Constant pressure is seen in the outlet which is due to the pressure outlet boundary condition. The head is computed based on the total pressure at inlet and outlet and since the flow velocities at the inlet and outlet are fixed due to the mass flow inlet boundary condition, there is distinct static pressure difference seen in the three cases. Pressure is seen to be decreasing as the speed increases and due to this the head is increasing with the speed following the equation for total head. The negative pressure values indicate lower than atmospheric pressure. The pressure magnitudes are in kPa.

Velocity vectors in the pump are plotted similarly for the pump running at different speed but a fixed discharge. Figures are shown below and the magnitudes are in m/s. In the figure, it can be seen that the vectors at the inlet and outlet are identical in three speeds corresponding to the same volumetric flow rate inside them. The difference in the vectors is seen in the impeller region where higher magnitudes of velocity are seen in higher RPM than lower. This is due to the rotational speed of impeller imparted to the fluid. The higher speed in impeller is causing the pressure to drop more in case of higher speeds.

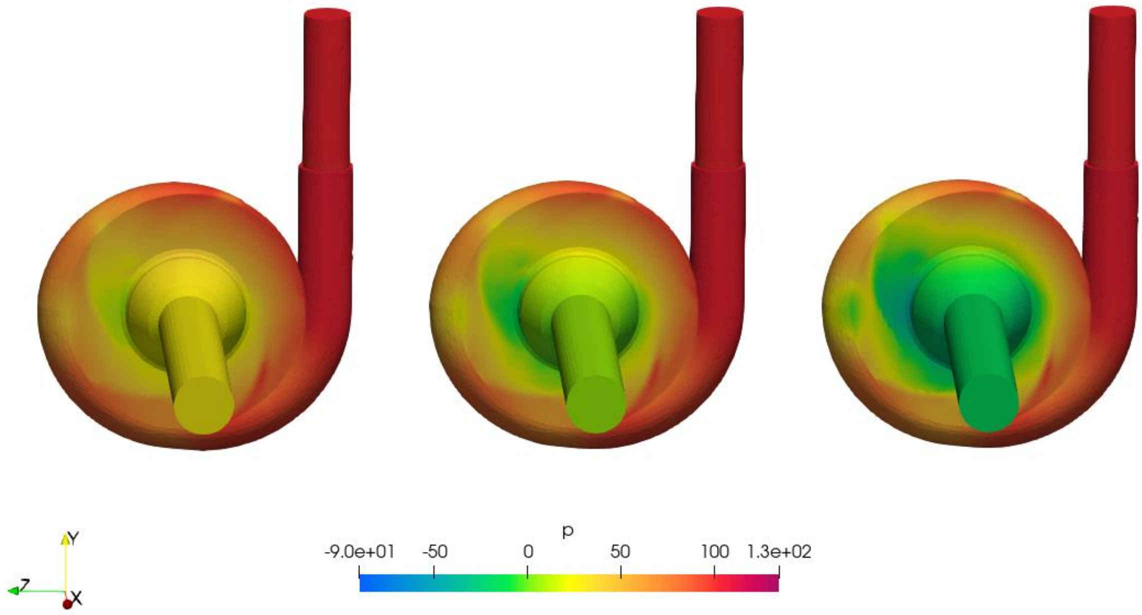


Figure 10: Pressure field variation with speed (2700 RPM, 3000 RPM, 3300 RPM)

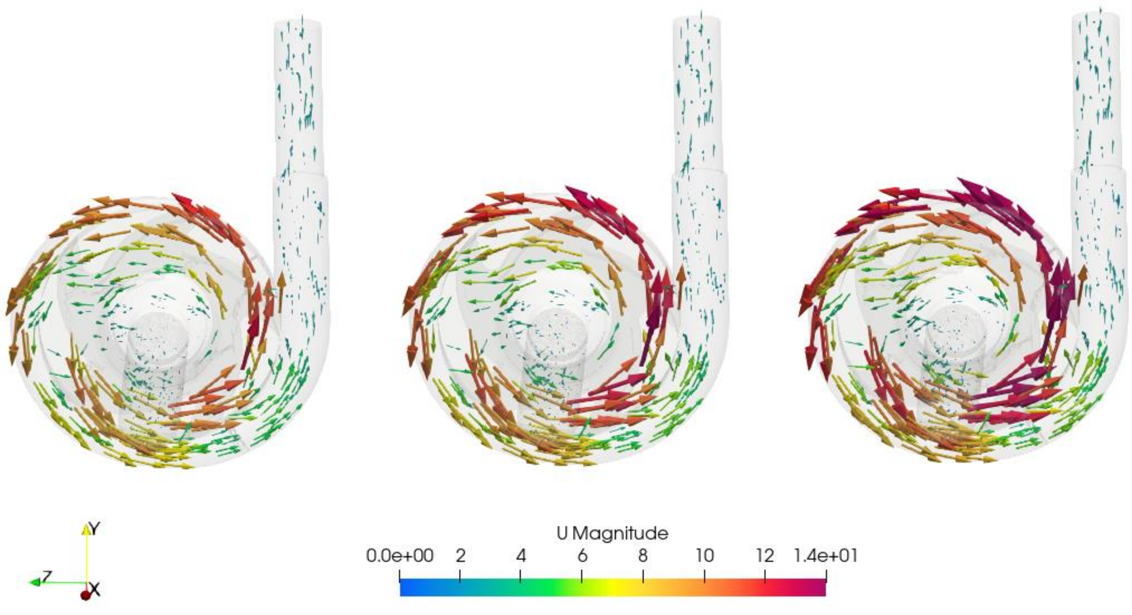


Figure 11: Velocity vectors in the pump (2700 RPM, 3000 RPM, 3300 RPM)

The Input power and efficiency in the ranges is calculated according to corresponding equations and plotted in following figures. Input power is seen to rise with increase in speed while efficiency is seen dropping.

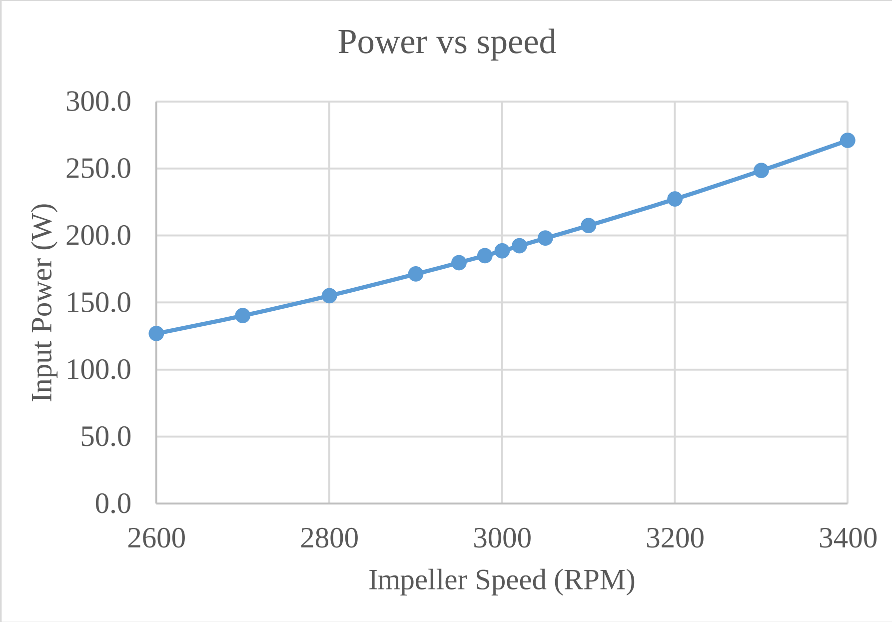


Figure 12: Input Power variation with speed

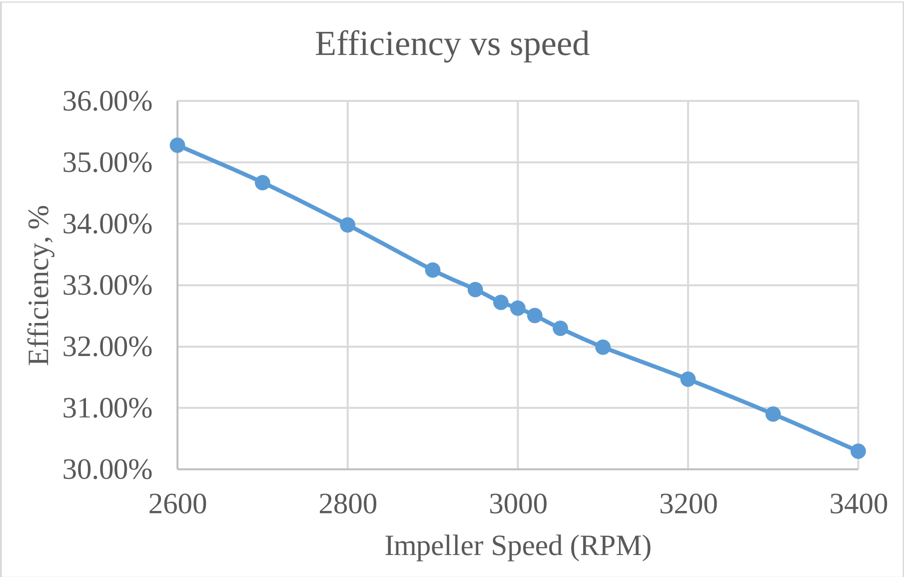


Figure 13: Pump efficiency with speed

4.2 Validation of OpenFOAM Results

The pump head obtained using OpenFOAM is compared with that of the Ansys Fluent in the speed ranges of 2400 RPM to 3600 RPM which is 600 RPM above and below or base speed of 3000 RPM. Head obtained using OpenFOAM is slightly higher compared to result from Fluent at the same speed. Comparison of the head generated by the pump in CFD with the performance curve by manufacturer shows that the values of head are higher for the both OpenFOAM and Ansys Fluent.

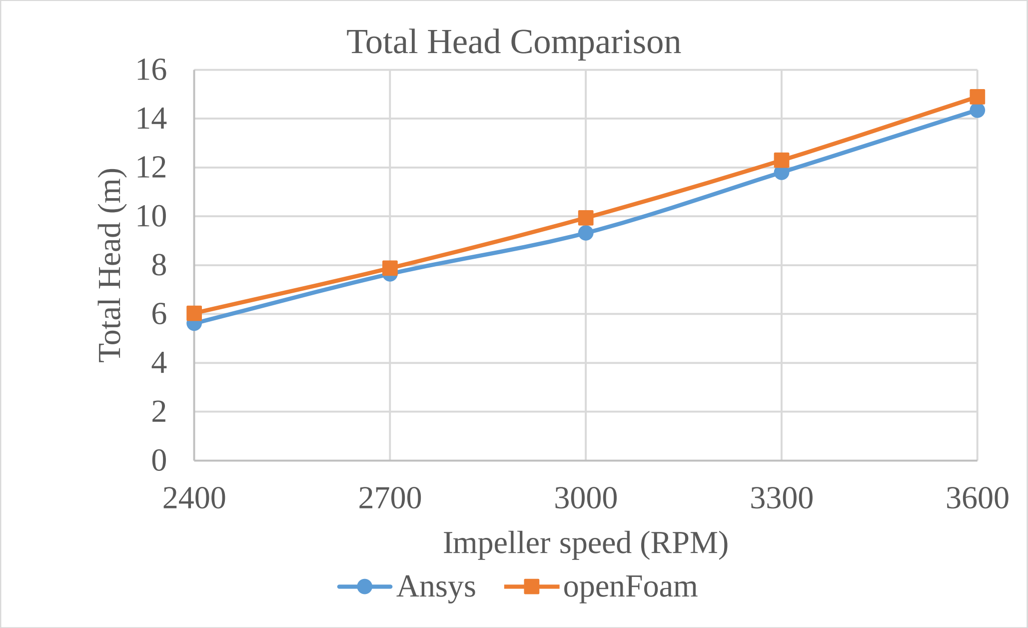


Figure 14: Total Head comparison between OpenFOAM and Ansys

The result of the simulation (head and input power) is compared with the performance charts available from manufacturer. Head obtained from simulation is seen to have offset from the head from manufacturer's chart. This may be due the frictional losses in case of the manufacturer's curve caused by rough surface in the walls of the pump. Surface roughness is not accounted in case of the pump under simulation. As the speed of the pump is increased; the head is seen to increase in simulation as expected. The head obtained from OpenFOAM is offset from performance curve by average of 1.3m.

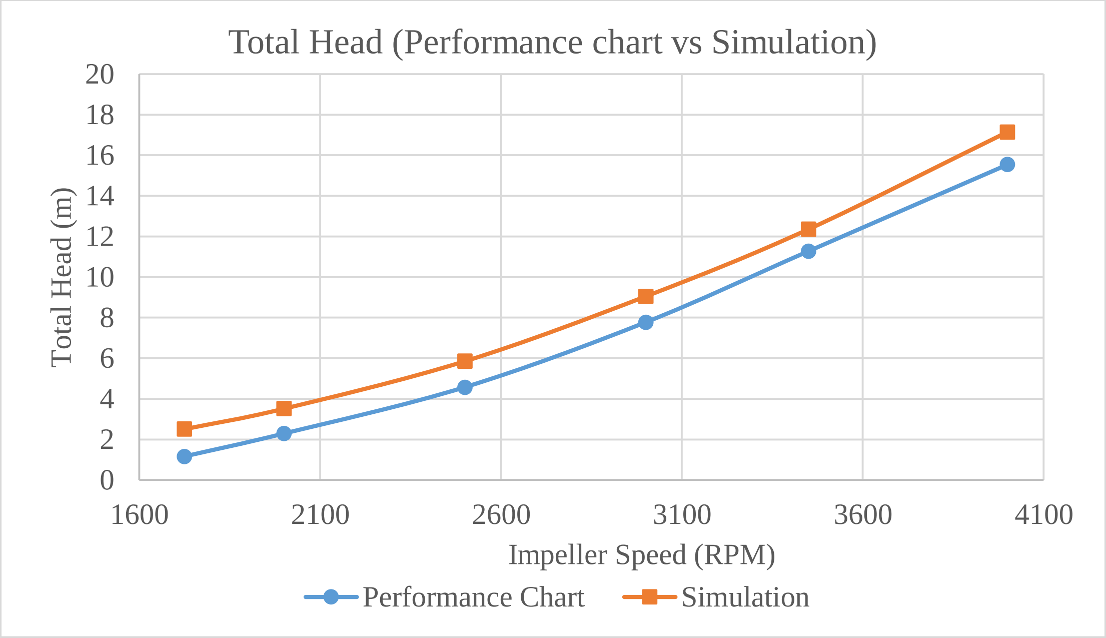


Figure 15:Head comparison between Performance Chart and Simulation

Input power is seen to match better between manufacturer's chart and simulation results. As expected, the value of power increases with speed. Values and errors are in Appendix as a table.

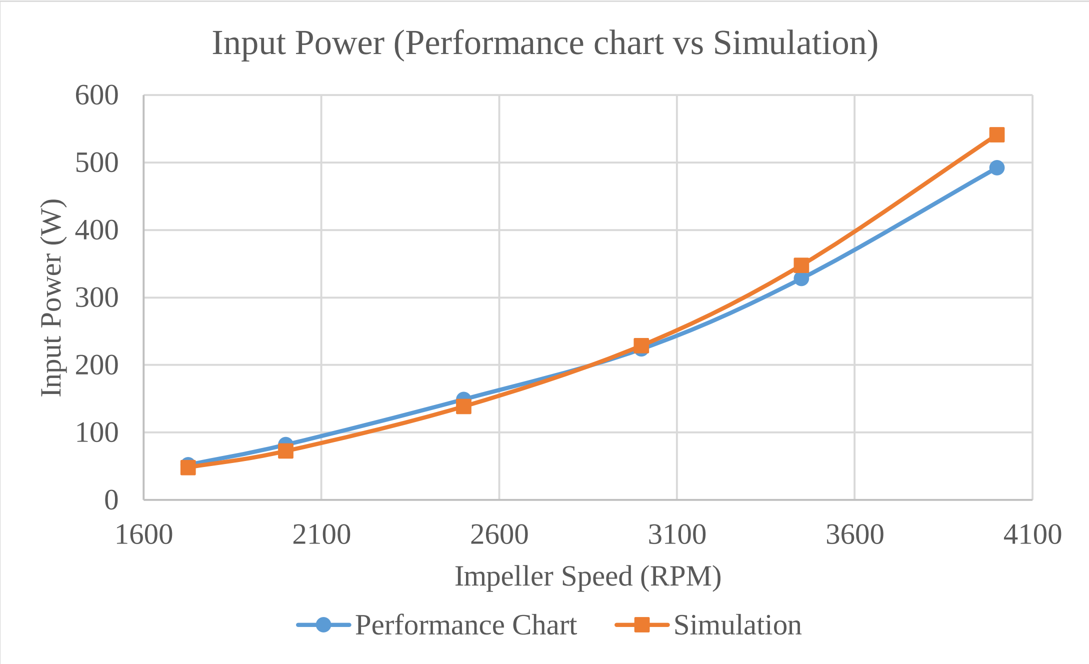


Figure 16: Input Power comparison between Performance Chart and Simulation

4.3 Transient Study

Two pressure probe locations are placed in simulation which recorded the pressure values for each time step. The plot of the pressure values for the two probes for the time is shown in below graph. In the graph, each series is fluctuating about a mean level and the wave is roughly repeating in 0.02 second which is the time taken by impeller to complete one revolution in the run speed of 3000 RPM. The plot thus contains data for five impeller revolutions. In one revolution of impeller (0.02 second), there are five peaks in the curve which correspond to each of the five blades of the impeller.

When plotting the pressure values from the two probes, it can be clearly seen that probe 2 pressure values reside at higher levels compared to the probe 1 pressure values. Also the peaks of pressure have higher amplitude in case of probe 1. The reason for this is probably due to the location. Probe 1 is located at the start of the volute, so it is near to the blades causing to create higher peak pressure. In case of probe 2 which is at greater distance compared to probe 1, the effect of blades is minimized due to the flow between blade and the probe.

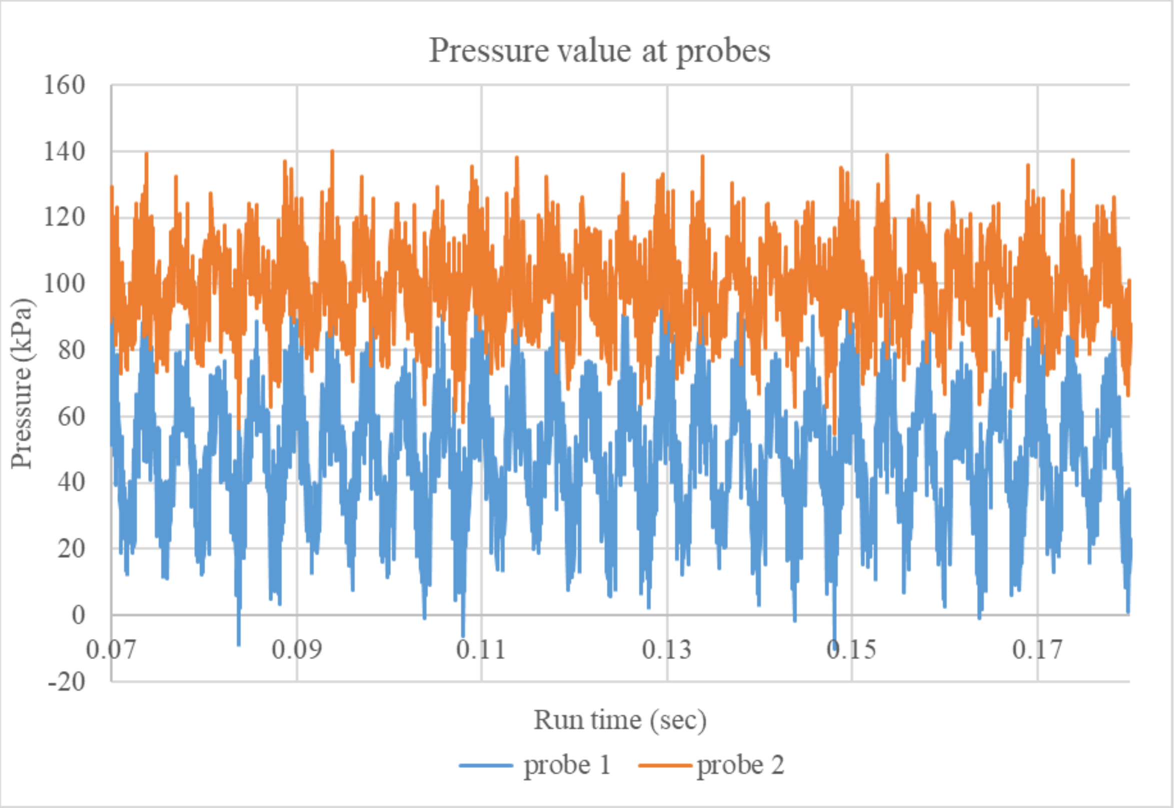


Figure 17: Pressure probe reading for the two pressure probes

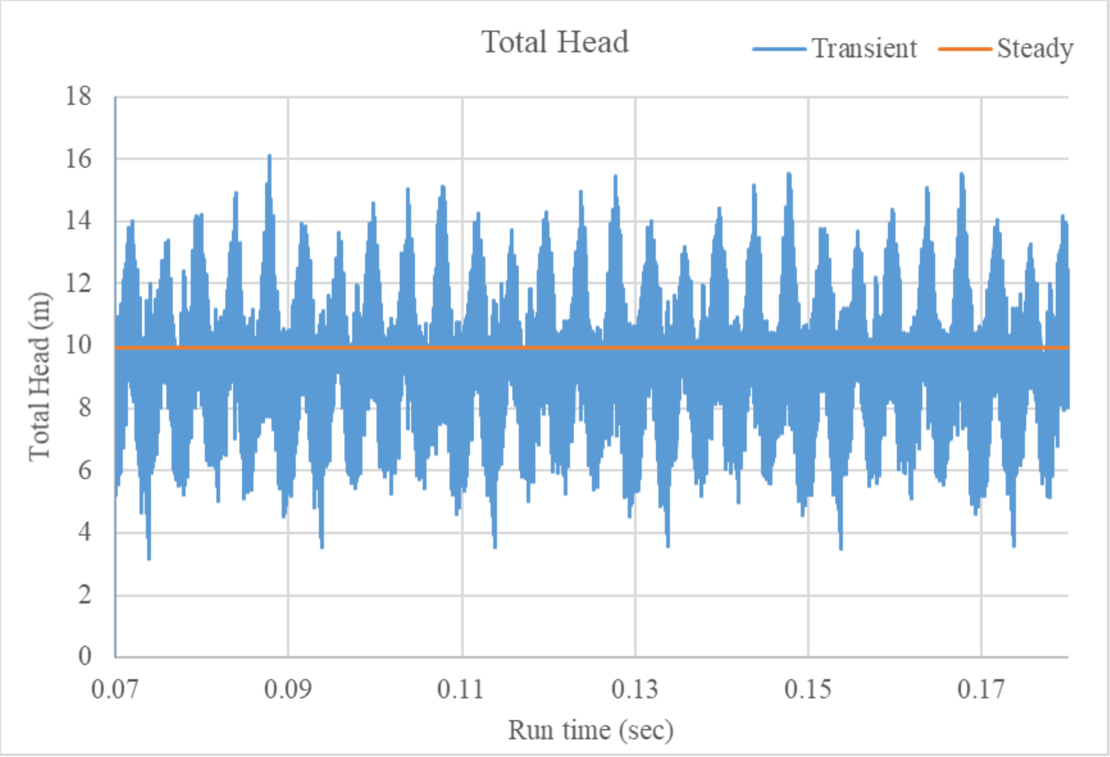


Figure 18: Head vs time plot

Total head versus time graph is plotted for the study and it is seen that the head is increasing as the flow is being developed and tries to reach a stable value. The head vs time graph same as seen in Huang et al. (2019). Transient simulation is also performed for 1800 RPM and similar plot is obtained with the case but the periodic nature was with different frequency (shown in Appendix). The pressure vs time plot obtained is compared with that of Tiwari et al. (2020) where the pressure was measured in same probe locations, but the results could not be validated.

4.4 Sensitivity Solution of Speed Deviation in Pump

Pump needs to be run at different speeds for controlling the flow rate or the head requirement of the pump. The speed of the pump is subject to change according to head or flow rate requirement. Sensitivity solution of pump speed is done by solving the steady case to convergence at different speeds at and around 3000 RPM base speed. The number of iterations required for convergence for each of the cases are noted and plotted as iteration from t=0 (baseline simulation). Next, same steady cases are run from the converged solution of 3000 RPM speed (sensitivity simulation) and the number of iterations required for the convergence are plotted in same graph as follows.

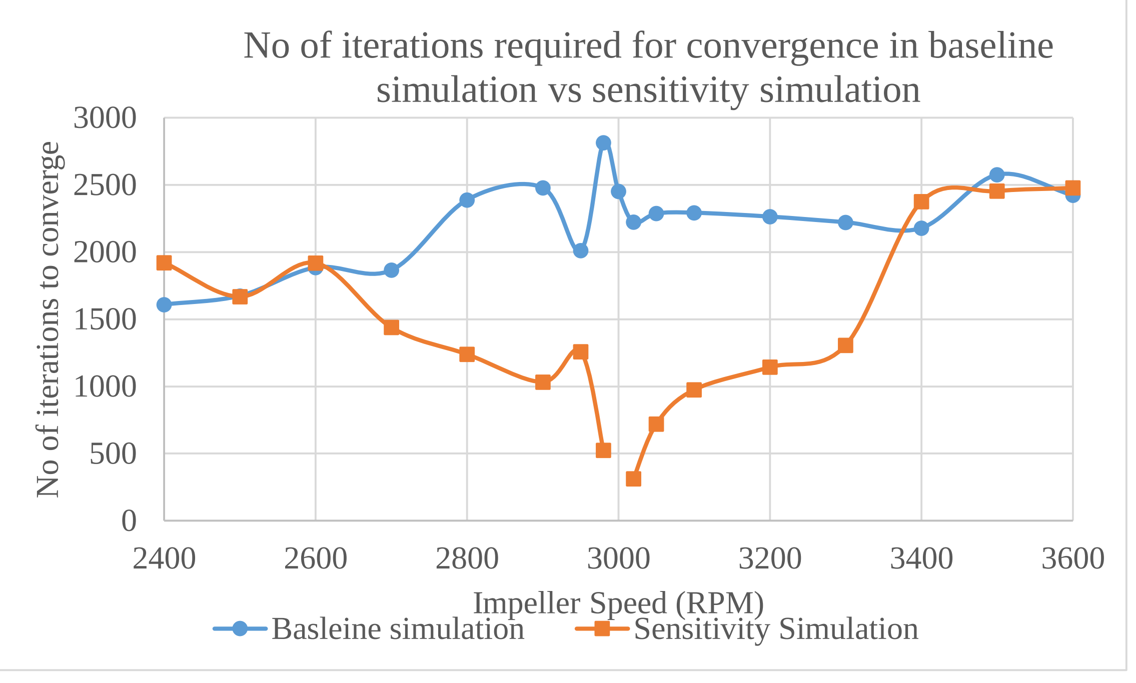


Figure 19: Convergence comparison of sensitivity solution of speed change

From the graph, it can be seen that up not all cases converge at same number of iterations for the cases run from t=0. There are high fluctuations compared to the mean number of iterations. But when run from the converged solution of 3000 RPM (sensitivity solution), a general trend is seen that as the difference of speed becomes high the number of iterations required for convergence is increasing. At speed change of 600 RPM below and above 3000 RPM, higher number of iterations are required for convergence in sensitivity simulation compared to baseline simulation. This suggests that sensitivity solution is not good option in cases of such. There is also trend of requiring less number of iterations for the lower speed cases in the range tested.

Solution can be found in less time using sensitivity solution for 10% speed change from the base speed of 3000 RPM. Similar results may be extracted for the other base speeds. For example, speed of 2400 and 3600 RPM may be chosen which can be used to solve the 10% speed changes around it. For speed changes up to 200 rpm above and below, sensitivity solution is providing iterations in about half iterations. For 1% speed change, sensitivity solution is providing results in one-sixth time. If simulation data is desired for whole speed range of the pump, baseline simulations can be done for 600 RPM difference and for range in between sensitivity simulation can be done. This lowers the computational cost for the whole data range.

Table below shows the comparison of head from sensitivity solution and baseline simulations when run from t=0. The maximum deviation in output variable is less than 0.4% which suggests that results from sensitivity solution are as dependable as the baseline simulations.

Table 1: Head comparison in sensitivity solution

RPM of pump	Head (baseline simulation), meters	Head (Sensitivity simulation), meters	% deviation
2600	7.233	7.22	0.18%
2700	7.857	7.862	0.06%
2800	8.517	8.537	0.23%
2900	9.201	9.212	0.12%
2950	9.567	9.575	0.08%
2980	9.784	9.8	0.16%
3000	9.938		
3020	10.099	10.066	0.33%
3050	10.334	10.338	0.04%
3100	10.723	10.731	0.07%
3200	11.552	11.557	0.04%
3300	12.406	12.406	0.00%
3400	13.264	13.265	0.01%

4.5 Sensitivity Solution of Flow Rate Deviation

Similar approach is applied for the flow rate variation of the pump. Design flow rate of 0.631 l/s is selected and pump is run at the speed of 3000 RPM. Pump flow rate is varied both below and above this flow rate keeping other parameters constant. The number of iterations for all these baseline simulations are noted. Now from the converged solution obtained initially, sensitivity solution for the different flow rates (whose baseline simulations are already recorded) are run and iterations to specified convergence is noted and plotted in same graph as baseline simulations.

Baseline simulations do not converge at same number of iterations. From the graph, it can be seen that the sensitivity simulation works to give results for broad range compared to the speed sensitivity solutions. The trend is as the percent change in flow rate are increase from the design speed, the simulation sensitivity solution is taking more iterations. For 2% change in the flow rate, the sensitivity simulation converges in one-fifth time compared to time required for baseline simulation and for 15% change, sensitivity simulation can help reduce time by half.

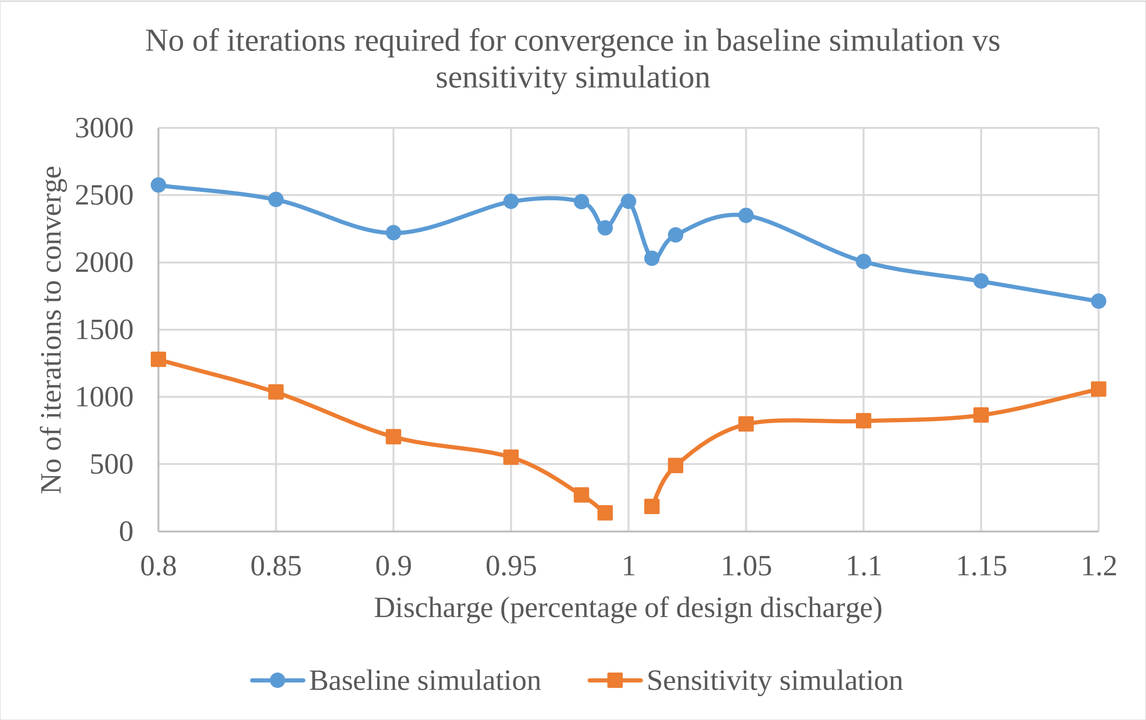


Figure 20: Convergence comparison of sensitivity solution of flow rate change

Comparing the results of the simulation in terms of output parameters, it is seen to have results comparable to baseline simulation. Error of maximum 0.4% can be seen. Data related to this is in Appendix. Comparing to speed variation, it can be seen that the variation in the speed causes significant variation in output parameters compared to the flow rates variation. If data is to be acquired for full range of flow rate, baseline simulations can be done for changes of 0.25 lps and in between the baseline simulations, sensitivity solution can be performed, so as to get low cost data.

4.6 Sensitivity Solution of Depth of Impeller Defect

Sensitivity and baseline simulation are performed for the defect size for the single defect case. The impeller defect introduced in one of the blade is seen to reduce the head developed by 1.7%. While the change in the output parameters were not realized to much extent, the cases are seen to converge, pointing that the effect of increase in the depth is miniscule compared to the amount which would cause performance change.

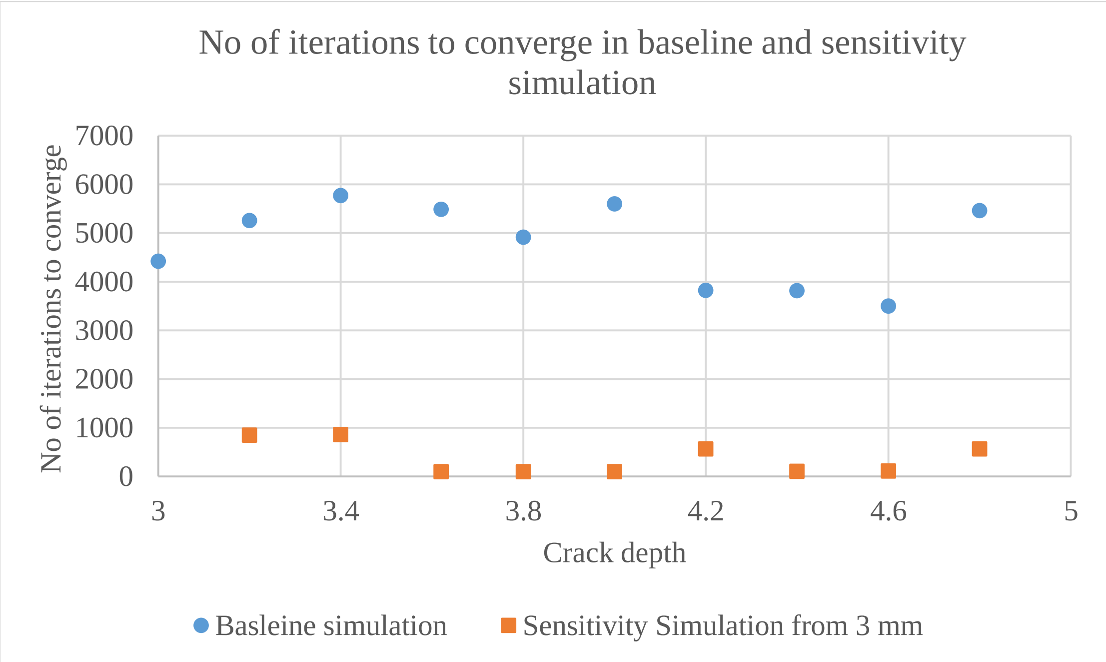


Figure 21: Convergence comparison of sensitivity solution of depth of defect (defect on single blade)

In this case, sensitivity solution is performed from converged solution of 3 mm depth supposing that the depth of defect increases in future. It is seen that iterations to convergence is random for both baseline and sensitivity simulation. From the graph above, it can be seen that the sensitivity simulations converge at one-tenth time (average) compared to the average baseline simulation time. From this graph, it can be noted that, the number of iterations to converge in sensitivity solution do not show a trend. This is different from the cases of speed and flow rate sensitivity where the time for convergence increased as the uncertainty increased.

For evaluating the effectiveness of sensitivity solution, measured variables were pump head and the pressure at two probe locations. With increase in the depth of the crack from the given depth, the head change was not found to be significantly changed. This led to conclusion that the crack is able to introduce head change up to a certain level only, increase in crack depth do not affect the total head created. But, trend is seen in the pressure probe near the start of the volute (See Appendix). This may be attributed to the nearness of the probe to the blade as this probe is near the start of the volute. Measurement and Analysis of pressure data at probes for detection of faults is the basis of Tiwari et al. (2020). When similar defect is introduced in all of the five blades, head developed by the pump falls by around 12%

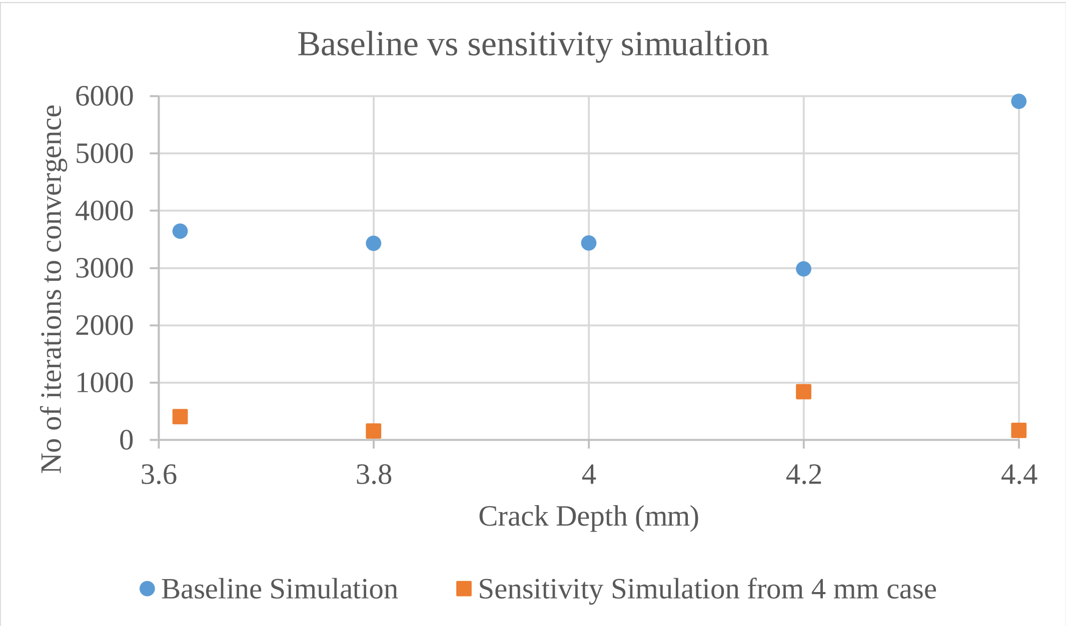


Figure 22: Convergence comparison of sensitivity solution of depth of defect (defect on all blades)

For defect on all blades, sensitivity simulation is performed from converged solution of 4 mm defect. In this case, sensitivity is performed of 10% around the 4 mm depth. Similar to defect on single blade, the solution time for baseline and sensitivity solution seem random. Sensitivity simulation has on average produced solution in one-eighth time compared to baseline simulation.

Although the exact shape of the erosion in the impeller blades cannot be predicted, this study may help to gain an insight on the performance change due to the specific defect. This study also sets the limitations for the amount of change that can be checked. Sensitivity solution helps to find out if the level of change in the input parameter is able to alter the design flow by the level that is noticeable. For instance, the pressure reading on probe 1 is seen to noticeably changing on 0.4 mm change of the depth while the head change is noticeable only after 1mm change in depth. Output parameters change can be used as the way to quantify the level of uncertainty that is useful for our design. The results from this research is in terms with Salehi et al. (2018) where the geometrical uncertainty had minimal impact in the pump’s head coefficient.

CHAPTER FIVE: CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

Numerical model of the pump is prepared and solution is observed in different speeds. To exploit the OpenFOAM's setup for finding the quick solution of the variation in the pump parameters, pump is chosen and sensitivity solution method is tested. The sensitivity parameters chosen are speed and the flow rate through the pump. Baseline simulation of the base speed and flow rates are performed and in the vicinity of them. Comparing the baseline simulation, it is perceived that the effect of impeller speed is higher in the output parameters compared to flow rate through pump for same relative change.

In sensitivity simulation, it is found that the sensitivity solution for speed changes can be used for speed changes up to 10% for reducing simulation time by half and for 1% speed variation sensitivity simulation produces results in one-fourth time compared to baseline simulations. With flow rate as sensitivity parameter, the speed of convergence is double for up to 15% changes in flow rate and 2% change can be solved in one-fifth time compared to the baseline simulation. Changes up to 10% can be done comfortably in sensitivity solution in shorter time compared to baseline solutions. Comparing both, it can be concluded that sensitivity solution method can be useful for small variations in input parameters. Its use case may be for finding results in short time thus helping to create data sets that can be used in machine learning applications.

For up to 60% increase in the depth of the defect, sensitivity method has provided solution in one-tenth time of baseline simulation. Increase in depth is seen to create no significant change in the simulation time in sensitivity simulation. The change in certain output variables could be measured for change in depth.

The sensitivity solution process for the defect of depth revealed the complexity associated with the creating mesh. Using the method of mesh preparation with Ansys mesh, it is found, change in geometry is difficult to realize keeping the mesh parameters (element count in each bodies and face) same.

Use of the sensitivity methods has helped to create lots of simulation data in short time compared to baseline simulations. These data can be used with machine learning algorithms to help predict the fault levels or predict performance in case of changes in discharge or speed.

5.2 Recommendations

Usage of simulation for solving a problem is frequently limited by the computational resources available. The use of various methods for reducing simulation time aids the situation which is goal of this research. The following recommendations could be pointed out from this research.

- Sensitivity simulation described here could be applied to other type of defect (For instance, Suction and discharge blockage, impeller erosion) to describe their performance or observe fluid flow with minimum exploitation of computational resources once a base case is performed.
- Sensitivity based method discussed here could be useful in number of other pumps to see the effect of a parameter in solution.
- Only one parameter is changed for sensitivity solution in this research. Further research could be done by changing multiple parameters same time so as to see the effect of changing multiple parameters on the convergence.
- Problem involving geometry change was difficult in this research. If geometry/meshing method can make structured meshing where geometry mesh change is anticipated in sensitivity solution, then this could further shorten the time of pre-processing, thus leading to shorter time for overall process.

REFERENCES

Alemi, H., Nourbakhsh, S., Raisee, M., & Najafi, A. (2014). Effects of Volute Curvature on Performance of a Low Specific-Speed Centrifugal Pump at Design and Off-Design Conditions. *Journal of Turbomachinery, 137*(4). doi:10.1115/1.4028766

Bachus, L., & Cusstodio, A. (2003). *Learn and Understand Centrifugal Pumps.* Elsevier.

Bhattrai, S., Baar, J. H., & Neely, A. J. (2018). Efficient uncertainty quantification for a hypersonic trailing-edge flap, using gradient-enhanced kriging. *Aerospace Science and Technology, 80*, 261-268. doi:https://doi.org/10.1016/j.ast.2018.06.036

Caruso, F., & Meskell, C. (2020). Effect of the axial gap on the energy consumption of a single-blade wastewater pump. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy,* . doi:10.1177/0957650920927366

Chen, Y., Yuan, J., Luo, Y., & Zhang, W. (2021). Fault Prediction of Centrifugal Pump Based on Improved KNN. *Shock and Vibration*.

Fu, Y., Yuan, J., Yuan, S., Pace, G., d'Agostino, L., Huang, P., & Li, X. (2014). Numerical and Experimental Analysis of Flow Phenomena in a Centrifugal Pump Operating Under Low Flow Rates. *J. Fluids Eng, 137*(1). doi:https://doi.org10.1115/1.4027142

Huang, S., Wei, Y., Guo, C., & Kang, W. (2019). Numerical Simulation and Performance Prediction of Centrifugal Pump's Full Flow Field Based on OpenFOAM. *Processes, 7*(9). doi:http://dx.doi.org/10.3390/pr7090605

Jasak, H. (2009). OpenFOAM: Open source CFD in research and industry. *International Journal of Naval Architecture and Ocean Engineering, 1*(2), 89-94. doi:10.2478/ijnaoe-2013-0011

Karassik, I. J., & McGuire, T. (2012). *Centrifugal Pumps.* Springer Science & Business Media.

Kumar, D., Dewangan, A., Tiwari, R., & Bordoloi, D. (2021). Identification of inlet pipe blockage level in centrifugal pump over a range of speeds by deep learning algorithm using multi-source data. *Measurements*. doi:https://doi.org/10.1016/j.measurement.2021.110146

Li, W., Yang, C. Y., & You, Y. (2009). Prediction on hydrodynamic performances of propeller by OpenFOAM. *In Proceedings of the ASME 2009 28th International Conference on Ocean, Offshore and Arctic Engineering, Honolulu, HI, USA, 31 May–5 June*, (pp. 633-638).

Liu, H. L., Yun, R. E., Kai, W. A., Wu, D. H., Ru, W. M., & Tan, M. G. (2012). Research of inner flow in a double blades pump based on OpenFOAM. *J. Hydrodyn. Ser. B, 24*, 226-234.

McKee, K., Forbes, G., Mazhar, I., Entwistle, R., & Howard, I. (2011). A review of major centrifugal pump failure modes with application to the water supply and sewerage industries. In A. M. Council (Ed.), *ICOMS Asset Management Conference, May 16 2011. Gold Coast, QLD, Australia: Asset Management Council.*

Nallasamy, M. (1987). Turbulence models and their applications to the prediction of internal flows: A review. *Comput. Fluids, 15*, 151-194.

Rapur, J. S., & Tiwari, R. (2017). Experimental Time-Domain Vibration-Based Fault Diagnosis of Centrifugal Pumps Using Support Vector Machine. *ASCE-ASME J. Risk and Uncert. in Engrg. Sys., Part B: Mech. Engrg., 3*(4). doi:10.1115/1.4035440

Sakran, H. K. (2015). Numerical analysis of the effect of the numbers of blades on the centrifugal pump performance at constant parameters. *Technology, 6*, 105-117.

Salehi, S., Raisee, M., J. Cervantes, M., & Nourbakhsh, A. (2018). On the flow field and performance of a centrifugal pump under operational and geometrical uncertainties. *Applied Mathematical Modelling, 61*, 540-560. doi:10.1016/j.apm.2018.05.008

Shah, S. R., Jain, S. V., Patel, R. N., & Lakhera, V. J. (2013). CFD for Centrifugal Pumps: A Review of the State-of-the-Art. *Procedia Engineering, 51*, 715-720. doi:10.1016/j.proeng.2013.01.102

Shojaeefard, M. H., Tahani, M., Ehghaghi, M. B., Fallahian, M. A., & Beglari, M. (2012). Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid. *Comput. Fluids, 60*, 61-70.

Standingford, D. W., & Forth, S. A. (2003). A Discrete Sensitivity Solver for an Industrial CFD Code via Automatic Differentiation. In S. W. Armfield, P. Morgan, & K. Srinivas (Ed.), *Computational Fluid Dynamics 2002.* Springer. doi:https://doi.org/10.1007/978-3-642-59334-5_8

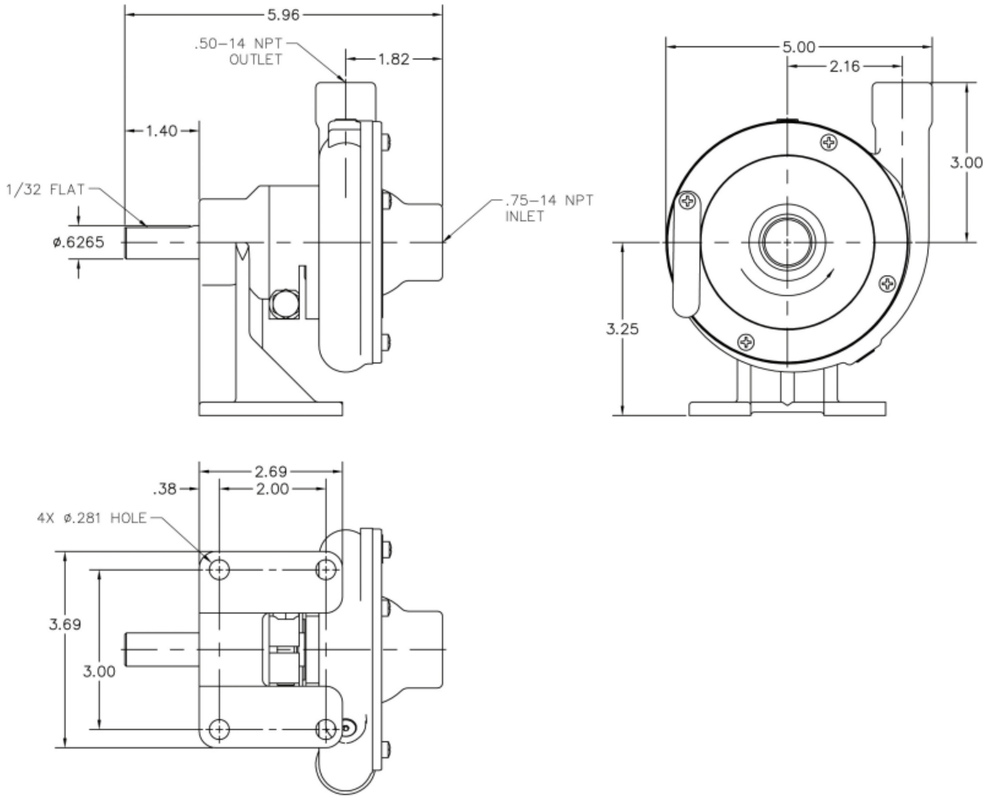
Tiwari, R., Bordoloi, D. J., & Dewangan, A. (2020). Blockage and cavitation detection in centrifugal pumps from dynamic pressure signal using deep learning algorithm. *Measurement*. doi:10.1016/j.measurement.2020.108676

Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method.* London: Pearson Education.

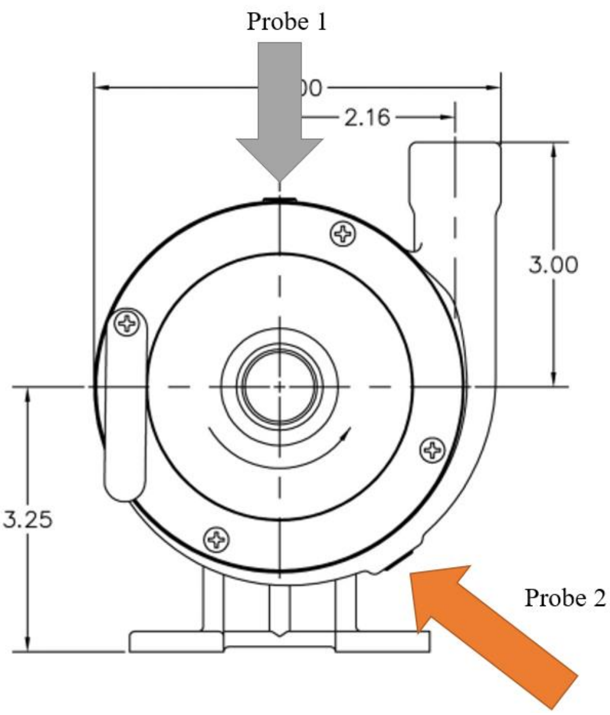
Wang, Y., Wang, X., Chen, J., Li, G., Liu, H., & Xiong, W. (2022). Numerical simulation of the influence of mixed sand on erosion characteristics of centrifugal pump. *Engineering Computations, 39*(6), 2053-2080. doi:https://doi.org/10.1108/EC-10-2021-0602

APPENDIX

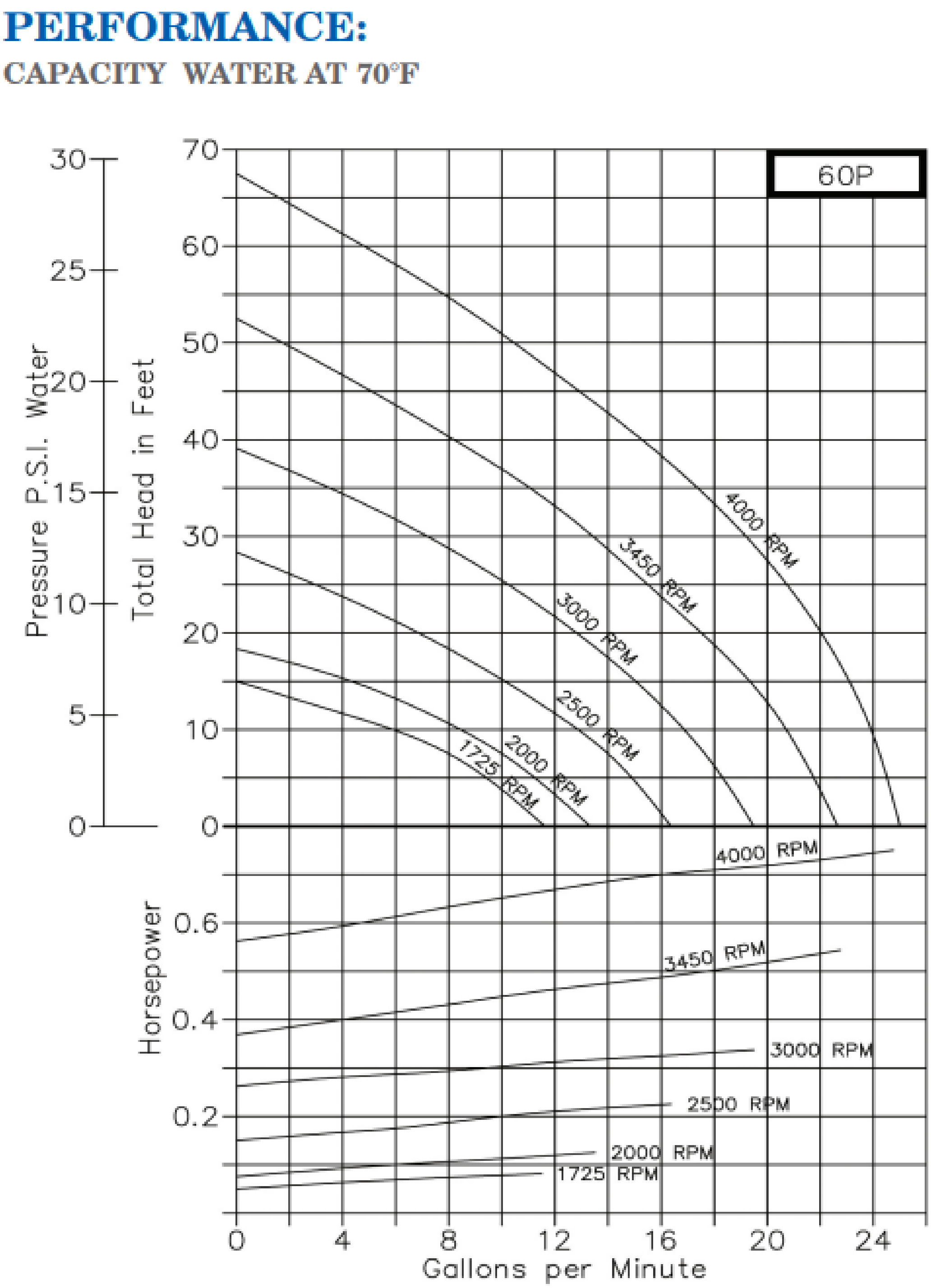
Pump Drawing



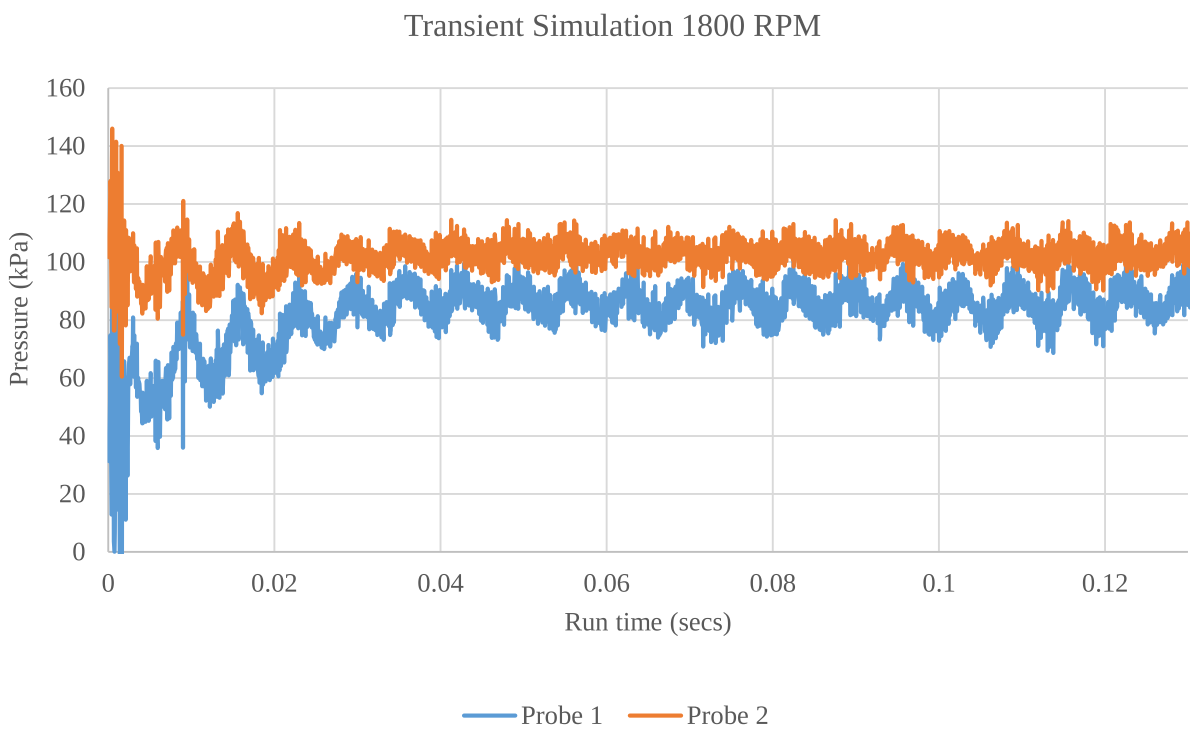
Pressure probe locations



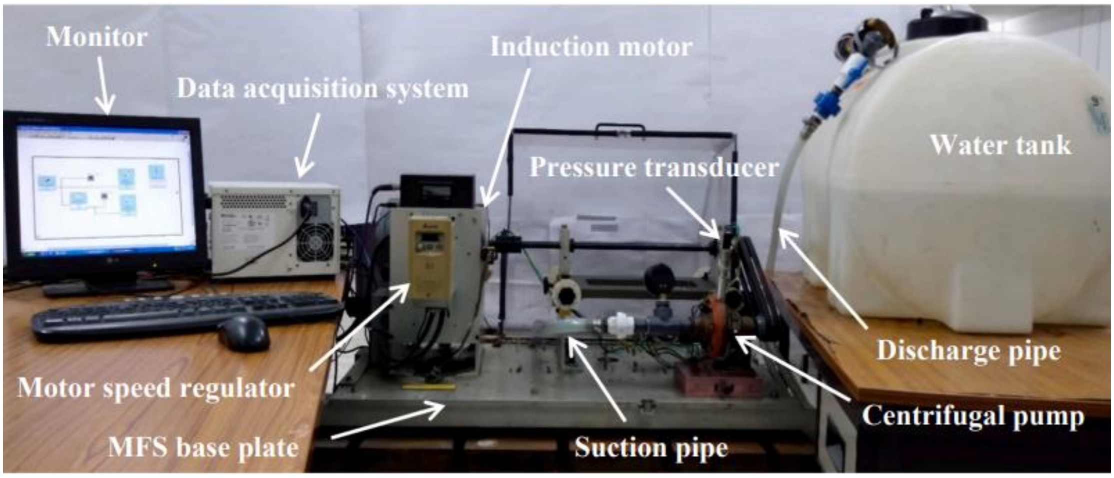
Manufacturer’s performance curve



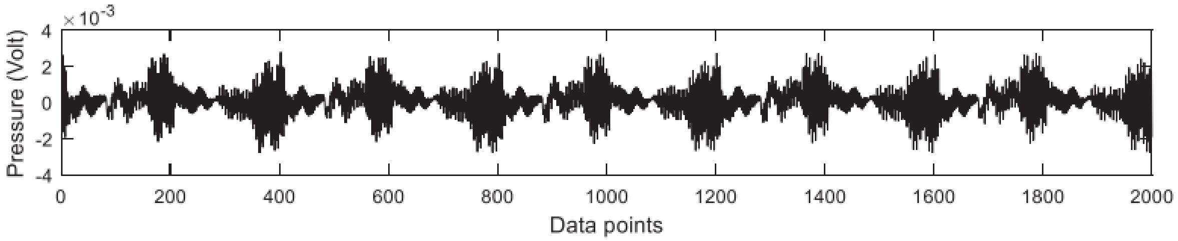
Pressure value at probes in 1800 RPM



Tiwari et al. (2020) and Kumar et al. (2021) experimental setup



Tiwari et al. (2020) pressure plot sample (Pressure in volts, total time = 0.1 seconds)



Flow Rate Sensitivity

Discharge (% of design discharge)	Discharge (lps)	Baseline simulation	Sensitivity simulation	Head (baseline), m	Head (sensitivity solution),m	% error in head
0.8	0.505	2574	1278	10.543	10.526	0.16%
0.85	0.536	2467	1035	10.416	10.42	0.04%
0.9	0.568	2219	704	10.277	10.271	0.06%
0.95	0.599	2452	552	10.1	10.1	0.00%
0.98	0.618	2451	271	9.996	10.009	0.13%
0.99	0.625	2255	137	9.976	9.938	0.38%
1	0.631	2452		9.938		
1.01	0.637	2030	185	9.901	9.906	0.05%
1.02	0.644	2204	488	9.889	9.895	0.06%
1.05	0.662	2349	797	9.82	9.812	0.08%
1.1	0.694	2007	821	9.718	9.71	0.08%
1.15	0.726	1860	865	9.615	9.607	0.08%
1.2	0.757	1711	1058	9.525	9.495	0.31%

Sensitivity for defect on single blade

crack size (mm)	Baseline simulation	iterations when ran from converged solution	Sensitivity Simulation iterations	Head (baseline),m	Head (sensitivity solution),m	probe 1 (baseline)	probe 1 (sensitivity solution)
3	4426			8.853		91.5	
3.2	5262	5277	851	8.858	8.909	91.6	91.3
3.4	5769	5288	862	8.898	8.906	91.1	91.1
3.62	5491	4526	100	8.93	8.889	90.7	91
3.8	4918	4524	98	8.8717	8.891	90.5	91
4	5603	4526	100	8.791	8.884	90.83	91
4.2	3825	4995	569	8.826	8.901	90.08	90.08
4.4	3818	4532	106	8.823	8.891	89.91	90.9
4.6	3504	4536	110	8.841	8.891	89.3	90.9
4.8	5460	4991	565	8.084	8.891	89.6	89.7

Comparison of Simulation results with performance charts

	Head					Power			
RPM	Chart (ft)	Chart (m)	Simulation (m)	% error	error (m)	Chart (hp)	Chart (W)	Simulation (W)	% error
1725	3.8	1.15824	2.509	116.62%	1.35076	0.07	52.2	48.16	7.73%
2000	7.5	2.286	3.516	53.81%	1.23	0.11	82.0	72.45	11.67%
2500	15	4.572	5.853	28.02%	1.281	0.2	149.1	138.45	7.17%
3000	25.5	7.7724	9.047	16.40%	1.2746	0.3	223.7	228.67	2.22%
3450	37	11.2776	12.357	9.57%	1.0794	0.44	328.1	347.84	6.01%
4000	51	15.5448	17.148	10.31%	1.6032	0.66	492.2	541.04	9.93%

Mesh Convergence

No of elements	Total Head (m)	% difference from earlier result	Force (N)	% difference from earlier result	Torque (Nm)	% difference from earlier result
99688	9.274		35.297		0.615	
203493	9.494	2.32%	31.748	11.18%	0.614	0.19%
253972	9.467	0.29%	31.22385	1.68%	0.607	1.11%
302241	9.563	1.00%	31.24	0.05%	0.605	0.39%
331679	9.595	0.33%	31.26	0.06%	0.604	0.10%
386952	9.601	0.06%	31.25	0.03%	0.604	0.05%

त्रिभुवन विश्वविद्यालय
Tribhuvan University
इन्जिनियरिङ अध्ययन संस्थान
Institute of Engineering

डीनको कार्यालय
OFFICE OF THE DEAN

GPO box- 1915, Pulchowk, Lalitpur
Tel: 977-5-521531, Fax: 977-5-525830
dean@ioe.edu.np, www.ioe.edu.np
गोश्वारा पो.ब. नं- १९१५, पुल्चोक, ललितपुर
फोन- ५५२१५३१, फ्याक्स- ५५२५८३०

Date: November 26, 2023

To Whom It May Concern:

This is to certify that the paper titled "***Sensitivity solution of off-design conditions in centrifugal pump using OpenFOAM***" (Submission# **584**) submitted by **Sunil Sharma** as the first author has been accepted after the peer-review process for presentation in the 14th IOE Graduate Conference being held during Nov 29 to Dec 1, 2023. Kindly note that the publication of the conference proceedings is still underway and hence inclusion of the accepted manuscript in the conference proceedings is contingent upon the author's presence for presentation during the conference and timely response to further edits during the publication process.

Bhim Kumar Dahal, PhD
Convener,
14th IOE Graduate Conference

centrifugal pumpp

ORIGINALITY REPORT

7%

SIMILARITY INDEX

PRIMARY SOURCES

1	www.mdpi.com Internet	161 words — 2%
2	Fu, Yanxia, Jianping Yuan, Shouqi Yuan, Giovanni Pace, Luca d'Agostino, Ping Huang, and Xiaojun Li. "Numerical and Experimental Analysis of Flow Phenomena in a Centrifugal Pump Operating Under Low Flow Rates", Journal of Fluids Engineering, 2014. Crossref	121 words — 1%
3	worldwidescience.org Internet	89 words — 1%
4	Rajiv Tiwari, D.J. Bordoloi, Aakash Dewangan. "Blockage and cavitation detection in centrifugal pumps from dynamic pressure signal using deep learning algorithm", Measurement, 2020 Crossref	73 words — 1%
5	openfoamwiki.net Internet	38 words — < 1%
6	uniassignment.com Internet	32 words — < 1%
7	ntnuopen.ntnu.no Internet	30 words — < 1%

8	Lubars, Joseph. "Learning and Adaptation in Graphs, Networks, and Autonomous Systems", University of Illinois at Urbana-Champaign, 2023 ProQuest	19 words — < 1%
9	Si Huang, Yifeng Wei, Chenguang Guo, Wenming Kang. "Numerical Simulation and Performance Prediction of Centrifugal Pump's Full Flow Field Based on OpenFOAM", Processes, 2019 Crossref	19 words — < 1%
10	123dok.org Internet	18 words — < 1%
11	openresearch.lsbu.ac.uk Internet	17 words — < 1%
12	www.dynaflux.com.pe Internet	13 words — < 1%
13	Giancarlo Chiatti, Ornella Chiavola, Silvia Conforto, Manfredi Amalfi. "Engine Block Dynamic Response via Combustion Noise Radiation", SAE International, 2015 Crossref	12 words — < 1%
14	cites.org Internet	12 words — < 1%
15	kawamoto-pump.co.th Internet	12 words — < 1%
16	cfcfd.mechmining.uq.edu.au Internet	9 words — < 1%
17	"Industrializing Additive Manufacturing", Springer Science and Business Media LLC, 2024	8 words — < 1%

Crossref

18	Bennett, Andrew. "Chemical Models of Intermediate Extinction Molecular Clouds", The University of Manchester (United Kingdom), 2023 ProQuest	8 words — < 1%
19	Viand, Roya Ebrahimi. "Molecular Dynamics Simulation of Open Systems Far From Equilibrium", Freie Universitaet Berlin (Germany), 2023 ProQuest	8 words — < 1%
20	repository.president.ac.id Internet	8 words — < 1%
21	www.ncbi.nlm.nih.gov Internet	8 words — < 1%
22	www.researchgate.net Internet	8 words — < 1%
23	Mhd Rashid, Munira Binti. "Perceptions of Cruise Tourism and Its Impact on Tourism-Related Businesses: A Case Study in Penang, Malaysia", University of Exeter (United Kingdom), 2023 ProQuest	6 words — < 1%