International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137

Volume – 09, Issue – 06, June 2024, PP – 22-40

A Systematic Review of Studies on Centrifigal Pumps

Clinton Nachukwu Idibia, Joseph Chukwuma Ofodu,
Ebigenibo Genuine Saturday
Department of Mechanical Engineering, University of Port Harcourt

Abstract: This paper focused on a systematic review of empirical studies carried out on radial blade centrifugal pumps in the last four decades (1980-2023).A focused literature search, based on PRISMA format for document selection and screening process for the systematic review, was adopted in conducting the empirical literatures search on Google scholar and other relevant sites, which are globally considered as most reliable and detailed database for literatures, to assess and download relevant empirical literature on efficiency and performance of centrifugal pumps carried out from 1980 to 2023. The results of the systematic empirical review were grouped into six different sections namely: Parametric Studies on Centrifugal Pump Impeller, the Pump Performance and Flow Field Studies, Design and Optimization of pump preference, Unsteady Flow Studies on Centrifugal Pump, Centrifugal Pump Performance Evaluation Studies and Centrifugal pump losses. The systematic review revealed that centrifugal pumps showed variant process operating conditions which were not always in agreement with the actual design conditions of the Pump. Hence, since the pumps are usually selected for operations within specific range of process conditions, it is usually critical that some field operators and maintenance teams usually may adopt to make certain field modifications in order to ensure and maintain the pumps availability to sustain operation and production, and a well calculated modification of the pump impeller have shown some remarkable operational improvement under some unavoidable changes on the process condition.
Keywords, Centrifugal Pump, Pump impeller, Process condition, Systematic Review.

1. Introduction

One of the major driving forces for innovations in mechanical studies has been the need to improve on the efficiency and performance of mechanical equipment in operation, a trend which has been the present day point of high priority in global energy serving need, which have also necessitated mandatory regulations by many nations and organizations. Rotodynamic pumps also called centrifugal pump are not left out, they are the most common types of pump adopted in several of our present day pump application needs, its application ranges from the Oil & Gas, Water production and treatment, Agricultural, Domestic and Chemical industry application among others, most reasons for the choice to adopt or deploy centrifugal pump on a particular application could not be far from its simple design pattern, broader operating capacity, improved performance and better efficiencies, inherent laminar flow pattern with minimal flow pulsations, easy operation and generous maintenance concept [1].

Pump could be described as a hydraulic or mechanical machines which transfers or converts mechanical energy into another forms of energy known as hydraulic energy. Centrifugal pumps could also be used to describe those pumps in which input mechanical energy, is converted into pressure energy by the process of centrifugal actions of the forces acting on the process fluid. The development of centrifugal pumps could be dated to have started in Europe during the late 1600's, after which the development transcended to the United States during the early 1800's. Several research, studies and development in centrifugal pump have led to both operational advancement and improvement in both the pump operational performance as well as applications of new materials in its production, as well as expansions in the pump application and operating envelope. These studies and improvements in the field of centrifugal pump has presently led to development and productions of pump that have attained over 90% efficiency. Usually, the pump internal structure and the flow pattern, together with the casing is usually designed to receive high velocity fluid departing from the centrifugal pump impeller, then converts the high velocity of the flowing fluid into pressure energy. In most designs, the pump impeller is often designed to discharge the flowing fluid directly into the pump chamber called the volute, the volute is a spiral shaped flow chamber and is usually trapezoidal in shape with a circular cross sectional area. The cross section of the centrifugal pump volute increases gradually around the periphery of the impeller, emanating from the volute tongue and truncated at the volute throat. Centrifugal pump volute tongue usually is designed to guide and directs the liquid flow, gathered from around the centrifugal swirling impeller, and passing same through the section called the throat of the pump diffuser. The centrifugal pump diffuser were applicable, is located between the pump volute throat and the pump discharge flange, depending on the pump design pattern, space availability and the desired flow velocities [2].

A centrifugal pump is mechanical device usually designed for the purpose of transferring of liquid by means of conversion of rotational energy of a moving impeller attached to a driven rotor to the process fluid. Based on the outlet blade angle of the centrifugal pump impeller, one of the classifications of centrifugal pump are: Forward blade centrifugal pump (a) used to describe these centrifugal pump whose outlet tip of the impeller blade curve are in the direction of the blade motion, and the fluid exit angle ($\beta 2$) are usually greater than 90°. (b) Backward blade centrifugal pump is usually adopted to describe those centrifugal pumps whose outlet blade angle curves are in opposite direction to the rotation of the motion of the rotor, and their flow exit angle ($\beta 2$) are usually less than 90°, (b) while our study on this work will be based on Radial blade centrifugal pump, which is used to describe those centrifugal pumps whose flowing fluid exits the impeller blade with relative velocity along the radial direction, and their fluid exit flow angle ($\beta 2$) are usually at 90°, (Ahmed, 2015) as shown in Figure 1

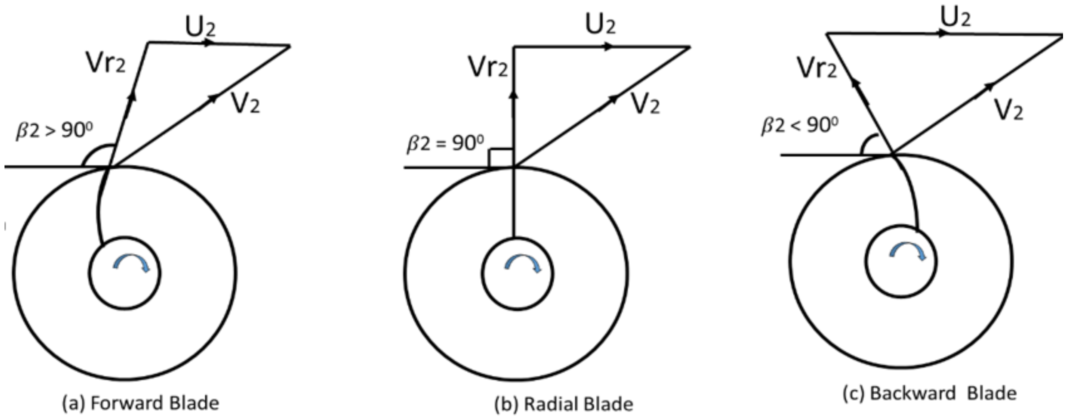

Figure 1, Centrifugal pump outlet blade classification [3]

Over the years, there has been progressive studies in the field of centrifugal pump internal flow mechanism with the sole aim of understanding how to improve on the design and performance of the pump, these advancement have necessitated the possibilities for pump designers to carry out analysis of the various flow conditions that occur internally inside the centrifugal pump [3].

The study by Yuan, *et al.* (2022) [4] revealed that traditionally, fluid flow through centrifugal pumps are exceedingly complex, involving curvatures, system rotation, separations, turbulence, unsteadiness and secondary flows, Gonzalez,et al. (2002) [5] stated that the geometry of the flow through centrifugal pump is often asymmetric due to the volute shape, as a result, these relative motion between the impeller and volute usually generates an unstableness which affects not only the overall performance of the pump, but also responsible for the pressure fluctuations, hydraulic noises and unforeseen hydrodynamic forces. Jafarzadeh, *et al.* (2021) [6] stated that the design of centrifugal pump impeller has to deal with a detail analysis of the expected flow and head requirement, and by selection of a suitable blade configuration for the application, these process involves the selection of the impeller blade and blade features such as the blade width, height, blade angle and blade structure.

Several studies have revealed that the overall characteristic and performance of centrifugal pump is mainly dependent on the process interactions between the internal elements of the pump wet-end, i.e., pump impeller, pump diffuser and the pump volute, these internal elements or parts usually predicts the overall performance and the characteristics of the liquid flow pattern through the centrifugal pump [1, 5-7]. Their studies on the subject revealed that Asymmetric flow structures and transient asymmetric flow force distribution are usually present within the centrifugal pump impeller. It is the Asymmetric flow structure and the transient asymmetric flow that usually leads to the issues of operating point-dependent on hydrodynamic forces of the impeller, and these cannot be counterbalanced by just mechanical imbalance from the systems, the outcome such usually leads to transmission of high level of the flowing fluid transient characteristic to the system, while in operation, it is obvious that these transmitted transient flow characteristics are usually detrimental to the pump operational effectiveness, flow smoothness and the pump efficiency, also to note that the pressure fluctuations can be manipulated to diminish by some calculated modifications of the impeller blade features, and the pump efficiency can be made to improve from the former. However, in actual practice, changes on the process fluid parameter away alter from that considered during the pump design and selection, which may usually present some significant fluctuations on the pressure and velocity flow field even at the design point.

Generally, centrifugal pumps are expected to operate effectively over a wide range of flow application and the flow within the centrifugal pump is seen to be with rotating stalls, characterized by the presence of distinct cells of flow separations within the impeller circumference, rotating at a fraction of the impeller revolution rate, these are seen as a result of the complexity of the flow structure within the impeller- volute body, the part-load flow in radial blade centrifugal pump poses a major challenges for accurate flow and performance prediction. Field experience has also shown that a well calculated modification of the pump impeller feature usually may produce significant influence and improvement on the stability of the system operation. Thus, this current study is a systematic review of empirical studies carried out in the last four decade on the design, efficiency and performance of centrifugal pumps using experimental and numerical approaches

2. Selection of Works for Review

A focused literature search, based on PRISMA (2020) [8] format for document selection and screening process for Systematic Review was adopted in conducting empirical literatures search on Google scholar and other notable sites, which are globally considered as most reliable and detailed database for literatures, to assess and download relevant literature on efficiency and performance of centrifugal pumps carried out from 1980 to 2024. Other relevant publications such as international conferences and project documents were utilized in this study. The keywords used in the search are "factors that affect the efficiency and performance of centrifugal pumps, experimental and numerical approach to the design, operation and maintenance of centrifugal pump". Downloaded documents from the search were properly screened and important ones were selected. Duplicate documents and irrelevant and unrelated ones were removed while some other documents were also eliminated based on some criteria: which includes (i) Documents that did not contain results of interest concerning centrifugal pumps like efficiency of pump, performance of the pump, numerical and experimental approach to ascertaining the performance and efficiency of the centrifugal pumps and factors responsible for improving or enhancing performance or efficiency of the pumps (ii) Documents that are published in other languages other than English language. (iii) Document that are not published in indexed journals are also eliminated or removed. Figure 2 show the schematic diagrams of the sorting process. The documents and articles considered in this review were pre-reviewed, analyzed, and categorized as related to the objective of the review.

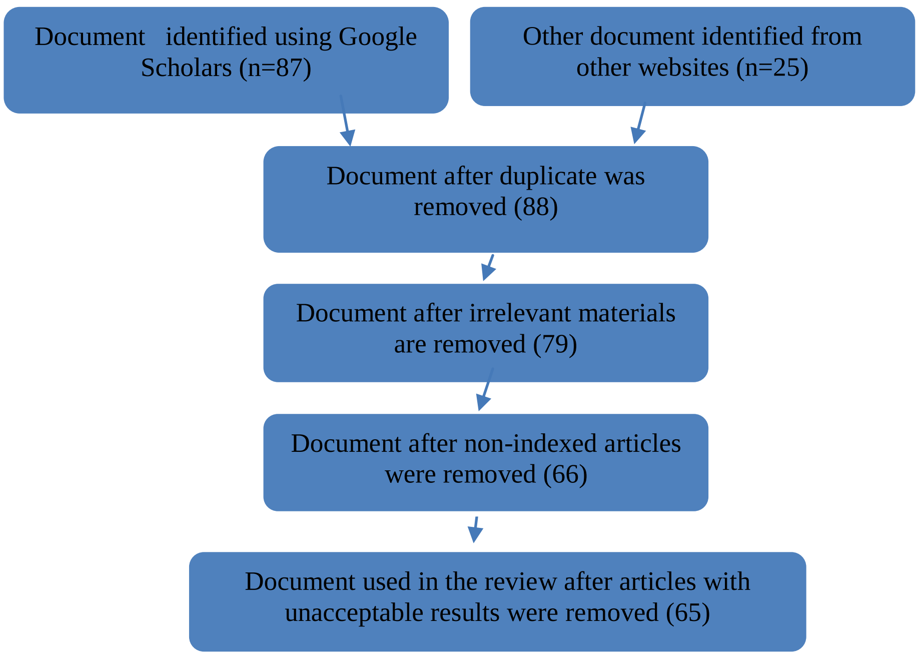

Figure 2: Summary of the document selection/screening process

3. Categories of Works

This systematic review study on centrifugal pump is grouped into different sections namely; Parametric Studies on Centrifugal Pump Impeller, Centrifugal Performance and Flow Field Studies, Pump Design and Optimization, Unsteady Flow Studies on Centrifugal Pump, Centrifugal Pump Performance Evaluation Studies and Centrifugal pump losses.

3.1 Parametric Studies on Centrifugal Pump Impeller

Several parametric studies using computational fluid dynamics (CFD) actually been supportive to pump application engineers, in predicting the various behaviors of centrifugal pump components and the overall performance of hydraulic machines, this has supported the pump design and manufacturing process as well as in the modification of exiting systems, most of the modifications were incorporated into mathematical and numerical models and the pump performance was perfectly evaluated and predicted, before the actual implementation of the required actions, through computational fluid dynamics (CFD), several studies and investigations has helped in independently analyzing the effects of system performance and operating parameters, in additions to forming the non-dimensional groups on pump overhaul performance.

Bacharoudis, *et al*. (2008) [9] studied the internal flow performance and characteristic of centrifugal pump by adjusting the impeller blade angles and retaining the pump discharge port diameter, a three dimensional numerical investigation was conducted using Navier-Stokes incompressible flow energy equations with a basic CFD finite-volume code. It was discovered that at the pump nominal capacity, and by gradual adjustment of the impeller blades angle from 20° to 50°, The attained pump discharge head was observed to have increase over six percent (6%) and the system hydraulic efficiency was observed to have reduced by up to four percent (4.5%). However, it was concluded that, with the pump preference at high flow rate, a positive adjustment of the impeller exit angle will led to a positive increase on the system efficiency and improved hydraulic performance of the pump. while from another study by Anagnostopoulos, (2009) [10] where a parametric study was conducted and developed a model (Numerical) for the summation of turbulent flow within and around the centrifugal pump impeller, it was a 3-D flow model for solving an RANS equation, on this study, the impeller geometry was presented with a number of modifiable design parameters, the system has the capability for varying the shape of the impeller an ascertaining the pump performance at various impeller configurations, the result of this work showed that a significant improvement of pump hydraulic efficiency can be obtained through modification and optimization of the pump impeller geometry in line with the pump present process condition.

On another numerical study by Grapsas, *et al.* (2008) [11] which was conducted on hydrodynamic design of centrifugal pump with the aim of improving the centrifugal pump hydraulic efficiency, through a calculated modification of the blade shape, the aim of the parametric study was to examine the outcome of impeller blade design parameters on the overall hydraulic performance of centrifugal pump impeller, some of the variable parameters considered were the inlet height of the impeller, length of the impeller blade, the inclination angle of the impeller leading edge, and in order to maintain better stability, the test started with lower revolution per minutes and proceeded to a higher RPM of up to a higher value of 3000 revolution per minute, form the outcome of the experimental test result, the system was seen to have converged after 100 number of completed solution from the flow field to the pump optimum impeller condition, and these was for the impeller with inlet height b_1 of 1 and 2, which is with a wrap angle of 74^o, and with the angle of the impeller leading edge inclination set at 2.1^o.

Djerroud, *et al*. (2011) [12] also conducted a numerical investigation study on the influence of the various impeller deign parameters on a steady liquid flow through a 3-dimentional flow centrifugal pump, with respect to varying the impeller diameter, the impeller width, impeller outlet blade angle, impeller blade number and height of the blade, they considered three basic components of the pump, which were the impeller and volute combined , the impeller and diffuser combined, as well as the pump impeller alone, outcome of this investigation confirmed that the head developed by the centrifugal pump and the brake horsepower (hp) of the pump will significantly improve, when the impeller blade height and impeller number is increased, it also revealed that the pump brake horsepower will decrease when the impeller blade width is increased, one of the major resolutions of their study revealed the internal interactions of the pump impeller and the volute which showed that with decrease in impeller outer diameter, while maintaining the volute dimension constant, will result to a reduction on the attained pump head, as well as the brake house power, the study also showed that the overall efficiency of the pump will be influence by any adjustment on the impeller outer diameter.

Chakraborty, *et al.* (2012) [13] conducted a numerical study on the impeller of centrifugal pump having different blade number, while maintain the same impeller outlet diameter and the different blade number was analyzed, with respect to the pump performance, the analysis was aimed at ascertaining the efficiency of the centrifugal pump, the experiment was conducted on a centrifugal pump with 4 to 12 impellers which were molded, the pump performance was obtained with respect to the pump efficiency at various rotating speed (2900 rpm, 3300 rpm and 3700 rpm) the result was analyzed using computational fluid dynamics software. The result of the study showed that an increase on the pump total head was attained with an increase on the pump rotor speed (in rpm) with 4 to 12 impeller blade number, it also revealed an increase in the pump efficiency with rise on the pump operating speed, it also revealed the presence of a low pressure area at the suction side of blade inlet for the pump which is proportional to the pump blade number. Also stated that an increase in blade number of the pump usually will lead to an increase on the attained head by the pump, while also confirming the existence of optimum number of blade for each application, thus the optimum blade number for their test application was 10.

Chakraborty & Pandey (2011) [14] conducted a numerical study of the performance of centrifugal pump impeller, using two pumps which were with identical impeller outer diameter but with different impeller blade number, the study was by evaluation and consideration of the impeller outlet diameter, the impeller blade number and the blade angle, since they were seen as the most vital elements that predicts the output performance of a centrifugal pump, they developed a test pump for operation at rotating speed of 4000 and with a solid impeller having 4 to 12 blade number, the outcome of their study shown that the impeller blade number is a major factor that has significant impact of the pump performance, and that with a positive increase on the blade number indeed reflected a positive increase on the total pressure generated by the system, the study also concluded that with 7 and 10 blade number, the attained efficiency was higher than that obtained when tested with lower blade number impeller.

Lu, *et al.* (2022) [15] conducted a numerical investigations on a centrifugal pump with respect to the whole flow channel while varying the pump flow rate and the pump net positive suction head (NPSH) the outcome of their investigation revealed that cavitation on the pump will occur close to the tongue of the centrifugal pump when considered under large flow capacity condition. Another study by Ref [16] proved that installation of inducer on the centrifugal pump impeller could offset the axial suction force of the centrifugal pump impeller as well as will promote the stability of the pump flow characteristic, while the investigation by Ref [17] revealed that due to impeller blade fracture, and considering a decrease on the pump capacity (Q) the attained head by the pump was reduced by 9.85% with a reduction of the pump efficiency by about 1.06%, they also observed a massive vibration on the pump discharge flanges, all these were attributed to the blade fracture, also observed was a rising amplitude for the vibrating peak value with turbulence at the impeller tongue, while the peak at the pump discharge grew respectively from 4.7% to 9.5%.

Wang, *et al.* (2021) [18] conducted a study using laser Vibro-meter to monitor the level of vibration produced by a centrifugal pump by analyzing the distributions of vibration around the impeller and the pressure distribution around the impeller, they discovered that much of the trailing edge was resulting from fluid separation inside the pump casing. Wu, *et al.* (2021) [17] conducted an experimental analysis on the consequence of blade pressure profile on the pump performance with respect to the dynamic and hydraulic performance of a low speed centrifugal pump, the result of their findings revealed unsteady pressure pulsation as was observed inside the centrifugal pump which can effectively be modulated by effective modification of the pump impeller blade.

Yuan, *et al*. (2022) [4] conducted a numerical study to calculate the flow structure and turbulent flow response, using three impellers with different design parameters by deploying ANSYS and CFX work bench, they discovered that closed impellers had better stability and with better hydraulic performance while split impellers showed poor stability and good hydraulic performance on that application.

Adamkowski, *et al.* (2016) [19] while studying the causes and behaviors of cavitation in centrifugal pump, their study revealed that the flow pattern around the impeller usually deviate drastically from the expected semi normal laminar pattern, thus recirculation may occurs with possible vibrations around the impeller, which will usually lead to abnormal noise around the impeller. Cui, *et al*. (2018) [20] also conducted a numerical study using Fluent software to investigate the pump internal flow structure on a double-volute multi stage centrifugal pump internal, the outcome of their investigation revealed that for the flow 0.6 pump capacity as studied for the nominal pump capacity (0.6Q), at the pump capacity (Q) and at the pump BEP and 1.2Q capacity, the highest radial thrust was observed with the flow rate at 0.6Q capacity, it was then recommended that a balance of the pump radial force during the design of all multi-stage centrifugal pump impeller is recommended, while from the study by Ref [21] which was on the effect of centrifugal pump impeller blade number, pump flow capacity and impeller outlet angle and impeller diameter, on the overall hydraulic performance and dynamic response of centrifugal pump, it was considered that vibration reduction design is a major factor that should be considered, when considering the pump optimum performance with respect to the selected parameters. Generally, these studies presented by the various scholars have predicted the various dynamic characteristic functions of centrifugal pump and have mostly affirmed that cavitation among others will mainly surface around the front and rear side of the centrifugal pump impeller and within the inner surface of the impeller front cover.

3.2 Pump Performance and Flow Field Studies

Various studies conducted on the performance of centrifugal pump and the fluid flow field inside of the centrifuge pump has generally agreed that the performance of centrifugal pump is greatly determined by the geometrical characteristic of the pump internal and the impeller, both when considering reverse and direct operating conditions, the centrifugal pump impeller is agreed to play significant role in detecting and addressing the performance characteristic of centrifugal pumps, various flow conditions such as flow loses and flow instabilities could be traced to the condition of the pump impeller and internal conditions. Appiah, *et al.* (2018) [22] presented that the pump impeller usually detects and predicts the transport and transient characteristics and the overall performance of centrifugal pumps. It is usually seen that the impeller geometry could consist of the shroud, the hub and the impeller blades, which usually impact the rotational energy required to propel the fluid flow. Traditionally, the centrifugal pump impeller components which usually consist of the leading edge and the impeller trailing edge, is usually adopted in describing the impeller inlet and outlet respectively. The impeller leading face of the swirling impeller usually receives the maximum flow pressure for any given impeller radius and is usually termed as the pressure side of the impeller, while the opposite side which is usually called the suction side of the impeller usually receives the lowest pressure. Several geometrical studies has been conducted on centrifugal pump impeller either by Experimental or Numerical approach, and all have affirmed same performance condition, including the theoretical investigation by Zhang. *et al.* (2018) [23].

Singh. (2005) [24] also conducted an experimental investigation to analyze the various methods of modification of centrifugal pump impeller geometry with the aim of attaining improved performance and operational efficiency on a pump as a turbine mode, the test was done without redesigning of the pump main hydraulic parts. The test was done by rounding off the impeller inlet of different impellers of different specific speed and providing a qualitative framework on the impeller rounding effect with respect to the pump internal hydraulic flow performance.

Chappell (1982) [25] performed an experimental investigation on the expected performance of centrifugal pump, when the impeller diameter is been reduced/ redesigned as a method of fine-tuning the pump to attain an operating point with better performance and improved efficiency, it was a special simulation approach only for those systems when an over dimensioned impeller was installed and requires a calculated reduction to attain point of better performance within the new operating envelop. Thome (1989) [26] conducted an experimental investigation on the performance of centrifugal pump impeller, by modifying the impeller diameter to ascertain the pump performance, outcome of the test showed that modifications of the pump impeller could promote an improvement in the pump performance as well as increase in the pump hydraulic efficiency, the study suggested that reduction on the pump impeller could also be adopted as a strategy to reduce some leakages loses around the pump impeller. Milne (1986) [27] conducted further investigations on the concept and revealed that further reduction of the pump impeller diameter may result to creation of secondary flow effect as will lead to fluid flow circulation around the impeller, and these will further lead to a reduction on the attained pump head.

Pandey (2008) [28] conducted a study using a 75inces fixed guide vanes on the pump volute of a centrifugal pump impeller, with the objective to minimizing the internal kinetic energy loses inside the pump casing, the outcome of their test revealed a tangential fluid flow profile along the impeller blade inlet. The study by Singh and Nestmann, (2011) [29] performed a numerical investigation to produce a reference model for a verification on the consequence of impeller shape modification for a mixed and radial flow centrifugal pump, under applications of pump as a turbine mode, the outcome of the investigation reveal that the performance of the pump has allot to do with the process interaction between the impeller and the working fluid. Similar result from the experimental test conducted by Ref [30] which was by modification of the impeller blade tip, for attainment of improved efficiency, materials were chipped off from the inlet end of the impeller tip, and the test was conducted at the pump flow capacity of $0.13m^3$/sec, and at a discharge head of 13m and 0.4kw pump power. The result showed that the pointed-nose shape of the impeller blade as streamlined, as a feature that promoted a reduction in turbulent flow within the pump internal and promoted an increase in the pump hydraulic performance and efficiency of the pump.

Yang, *et al.* (2012) [31] conducted an experimental investigation on the centrifugal pump impeller by modification of the impeller tip, through streamlining of the impeller inlet leading edge of the hub, for a system in pump as a turbine mode, the test was conducted using a radial blade centrifugal pump, the outcome of the test showed that streamlined impeller leading edge usually will improve the pump efficiency. With similar conception, Sun-Sheng and Fan (2012) [32] conducted a test to prove the consequence of impeller splitter blade on the overall hydraulic performance of a pump as a turbine (PAT) mode, the outcome of the test proved that increase in number of splitter blade installed will lead to a proportional improvement on the operational performance. Similar result was also obtained from the study conducted by Jain & Petal (2014)[33] through an experimental investigation of the centrifugal pump impeller geometry and operating parameters, considered parameters includes the diameter of the pump impeller, impeller blade tip roundness, and the rotating speed of the impeller. The study was conducted with a rotating speed ranging from 900 rpm to 1500 rpm when testing for the impeller mode. While the impeller diameter was reduced from about 10% to 20%, for the impeller diameter of 200mm, 225mm and 250mm diameter, then the maximum efficiency observed was at 10% of the trimmed impeller at an operating speed of 1100RPM. The study also developed an empirical correlation and produced a prediction that collaborated the range of 6% to 10% deviation of the efficiency prediction with respect to the experimental result.

Petel, *et al.* (2015) [34] conducted an experimental test through an impeller edged trimming, impeller exit angle trimming, hub rounding, and investigating the pump hydraulic performance from these modifications, under constant speed condition of 1100 rpm, and at various flow capacities of the pump. From the examination of the untrimmed and trimmed impellers with respect to the various performance, it was observed that trimmed impeller proved better hydraulic performance and better efficiency than untrimmed impeller, the best efficiency point form the operation was observed to be at lower flow rate, it was also seen that impellers with rounded edge had better performance, while on another experimental study by Doshi, *et al.* (2017) [35] investigating the effect of impeller blade rounding and inner shroud condition, on the output performance of a pump as turbine mode, from the analysis of the blades, when the rounded blades radius was 0.5 of the blade thickness, 0.5mm radius of the shroud thickness was adopted for the blade inner edge. The outcome of this experimental test on nine pump as a turbine simulations, revealed that a reduction in process internal energy loses were recorded at pump as turbine inlet mode. Whereas, when the rotational speed was varied from 18rpm to 54rpm, there was a reduction in loses recorded under low operational speed of the impeller, while at higher operational speed, a higher fluctuation of the produced torque was recorded with an overall advancement in efficiency, which was seen to be between 1 to 2.5%, revealing that there are higher possibilities of enhanced performance of the systems efficiency through further enlargement of the pump volute inlet. Singh and Nestmann, (2012) [29] also conducted an experimental study, using combination of radial and mixed flow pump, the study objective was to investigate the effect of impeller tip streamlining on the leading and trailing edge of the centrifugal pump impeller, the investigation outcome revealed an overhaul improvement on the pump efficiency and hydraulic performance proceeding from a proper impeller sharp end rounding.

Yang, *et al.* (2012) [31] also conducted a theoretical study and developed an applicable methodology, for forecasting the pump brake house power (BHP) of a centrifugal pump impeller with respect to the pump hydraulic parameters, they developed a system for sizing and selection of a suitable pump to operate on turbine mode, on an applications such as micro-hydro-site, while Ref [36] used the principles of comparing the “area ratio” approach to determine the pump brake horse power (BHP), while using geometric parameter and the pump performance curve to estimate the behaviors of the various parts of the pump, these traditional method of predicting the performance through experimental and theoretical approach, usually poses some limitations with respect to predicting the exact flow pattern within and inside the turbo machineries, and studies have shown that the flow inside the turbo machines are highly complex owing to the already known three dimensional nature of the flow pattern. Thus, the study proved that one of the most reliable method of predicting the flow pattern within the centrifugal pump as a turbine has been by computational fluid dynamics, which is seen as useful tool in predicting the performance of the turbo machinery.

Carravetta, *et al*. (2011) [37] used computational fluid dynamics (CFD) to analyze the complex fluid flow characteristic at the centrifugal pump internal, these approach was seen to be of huge time and cost savings when compared to the traditional approach, several other studies have also been carried out on the characteristics of centrifugal pump impeller, all aimed at predicting the pump performance, Yang, *et al.* (2014) [38] also conducted a numerical study using computational fluid dynamics (CFD) to study same impeller structure effect, using three blade thickness on pump as a turbine mode, their study affirmed that with impeller of higher blade thickness, the overall efficiency of the pump is reduced, and this reduction was attributed to the continuous rise in head drop and shaft power, which leads to more hydraulic loses around the impeller.

On another study by Ref [33] in which they tried to understand on how to fix problems that is associated with impeller flow instability, by manipulation and analysis of the flow pattern through an experimental and numerical approach, with adjustable guide vanes installed on a systems that is configured for pump as a turbine (PAT) operation. Using the adjustable guide vanes (AGV), the systems was seen to produce better performance when operating at the various capacities of the pump, and this was tested at various climatic conditions on a hydro site at a remote location, outcome of the test hinged the performance of the centrifugal pump on the structure of the impeller and the process fluid interaction with the impeller.

Fernandez, *et al.* (2014)[39] conducted a numerical analysis using a single suction centrifugal pump operating at 1750rpm rotating speed, the study compared results from both the numerical and experimental investigation using ANSYS fluent software and produced a result for the pump head, the pump efficiency and the pump power consumption, their findings revealed that under part-load operation, it recorded very massive variation of about 20%, reduction on the deviation, which minimized accordingly with increase in the flow rate, it was concluded that both the experimental and numerical test result shows an increase in the pump head curve with increase in flow rate, these was attributed to the presence of tangential velocity component at the impeller exist angle. There was a good agreement from the numerical and experimental test results, showing that in pump as turbine mode (PAT), the attained best efficiency point (BEP) was seen to be lower than that at the pump best efficiency point (BEP), for the pump mode, and the phenomenon was attributed to the complex flow structure inherent in the internal flow pattern for the turbine mode, which has created the massive flow losses in the operation.

Wen-Guang (2008) [40] conducted an investigate the general behaviors of a centrifugal pump on a viscous oil and water application, using a numerical study approach with a computational fluid dynamics (CFD) software, investigating the performance of the system under part load condition, full load at the pump best efficiency point. The test adopting a general purpose Fluent CFD software, to evaluate the outcome from three dimensional incompressible non laminar flow within the pump internal. The result from their study revealed that the pump volute has significant effect on the pump performance, as the performance of the pump without volute was remarkably difference from the performance of the pump with volute, also revealed a significant impact of the relative position, between the pump volute tongue tip and that of the impeller blade trailing edge. The study also outlined slip factor as a major factor that is highly dependent on the pump capacity, liquid viscosity, roughness of the pump internal and flow rate. The result of the numerical test was closely related to that obtained from the experimental test, showing a sudden rise on attained head, while confirming that the hydraulic loss which occurred at the pump volute is seen to be higher than that occurring within the pump impeller, while the fluid viscosity and other causal factors from the internal roughness was seen to be responsible for the seen sudden head rise effect from the profile, under part load condition, impeller and volute interaction was seen as a very important performance determinant factor in centrifugal pump operation.

Anagnostopoulos (2006) [41] conducted a numerical investigation to determine the impact of centrifugal pump impeller geometry on the performance of hydraulic machines with respect to its hydraulic efficiency, the study considered new meshing techniques adopting Cartesian grid method, this method was able to eliminate the computational constrain and automation of grid generation process, the performance parameters of the centrifugal pump was fully predicted under the various impeller condition as was modified.

Cheah, *et al.* (2007) [7] also conducted a numerical and experimental investigation on the fluid flow and flow field inside the centrifugal pump liquid end, with their adopted numerical and experimental investigation approach, results from both methods were obtained over a wide range of the fluid flow regime, result from both methods were in agreement, showing that at the design BEP point of the pump, the fluid flow velocity vector was visualized to be streamlined and smooth with flow stream aligned to the impeller blade curvature, while confirming the presence of flow separation along the impeller leading edge which is seen to be as a result of non-tangential flow conditions. While the flow structure at the volute casing revealed the presence of single and double vertical flow structures. By operating the pump at off design flow capacity, a significant changes was observed, as there was a gradual deviation from the well-structured flow pattern, showing a new flow pattern with various layers of internal recirculation and presence of eddies, and was seen to be intercepting the flow passage-way, thus stressing the need to pay great attention to the pump internal rotational effect, the causes and possible consequences.

Anagnostopoulos (2009) [10] conducted a numerical study on the methodology of developing a simulation for turbulent flow in two dimensional flow centrifugal pump impeller, using the methodology to develop a simulation considering a turbulent flow condition, the study objective was to investigate the pump characteristic and performance over the full range of the pump operating envelope. The study deployed an advance numerical method approach with Cartesian grid approach, it was observed that grid lines did not align into the normal operating boundaries as were created. The result of the investigation was able to present that the adopted optimization method could be used to predict the performance of the blade angle, stating that smaller blade angle had better performance than a large blade angle impeller under the study condition, and that impellers with smaller blade angle produced smoother flow pattern than the former.

Cui, *et al.* (2011) [42] conducted an analysis using numerical method to study the flow field within the centrifugal pump impeller, the impellers were of long to medium and short blade configuration, the study also conducted an experimental investigation on the pump on the analysis of same parameters, the result form both test revealed that the internal fluid flow within the centrifugal impeller passage way was seen to be complex, and unsymmetrical, these was traced to the hydrodynamic interactions between the pump impeller and the volute surface. And that the flow pattern near the diffuser throat was seen to have been influenced by the forms of the volute, while the relative velocity at the impeller exit is seen to be bigger with higher tendency to make a back flow at the volute throat outlet, under the influence of any obstruction. The result also revealed the presence of static and total pressure rise which is seen to have gradually moved from the impeller inlet point, to the impeller exit point, excluded was the passage near the diffuser throat and the impeller leading edge, it was evidence that the changes in total pressure magnitude was higher than the value for the static pressure at the boundary between the impeller and the volute.

Jafarzadeh, *et al*. (2021) [6] conducted a simulation study, to ascertain the nature of the velocity and pressure field profile within the centrifugal pump internal, using a 3-dimensional turbulent fluid flow model, the study did investigate the consequence of the turbulent models on the performance flow field of the centrifugal pump internal for a high speed centrifugal pump, the study was able to analyze and predict the effect of impeller blade number on the performance of centrifugal pumps, adopting the use of segregated solver which was quite faster and easier in producing a convergence solution. The result on blade number relationship on efficiency shows that impeller with higher blade number of seven, produced higher head coefficient as against the impeller with lower blade number of six and five in all ranges of the applications. The position of the blade with respect the volute tongue was also seen to have significant effect in determine the starting point for the fluid separation inside the centrifugal pump liquid end.

Benigni, *et al.* (2012) [43] conducted a numerical investigation on low specific speed centrifugal pump, the study was considered under part and full load conditions of the pump, the study adopted a 3-D CFD model and considered the impeller front and back cavities, it adopted an impeller with 360^o with a double volute diffuser, also integrated the system suction and discharge piping, it studied the steady state and the transient state conditions, also calculated was the front and back cavity pressure. The result of the findings showed that while neglecting the cavities, the head and efficiency developed followed the average of three clock position namely; zero degree position, twenty four degree position and forty-eight degree position respectively, while under full load condition, the produced head curve showcased a strong increasing tendency across the entire operating range, while the pump efficiency was seen to be at a much higher level. A very significant error margin was observed on the value obtained and was traced to some assumption that were not considered with respect to the impeller cavity, the result of the computational fluid dynamics calculation revealed a steeper head curve structure, than that obtained from the experimental test. Also shows a strong indication of presence of flow instability under the part load operation, while evaluating the transient conditions which showed an almost flat/ horizontal head curve, which was not very stable, in addition to showing and overlapping of head flow curve at the head flow intersection.

Hofmann, *et al*. (2001) [44] conducted a numerical simulation with a view to ascertain the performance of cavitation in turbo-machineries, the study adopted to use multi block mesh approach to analyze blade to blade interactions on a single channel pump, outcome of the test was seen to be in agreement with the findings from the experimental study and the visualization technics, results from both findings show that the model has the capacity to simulate the main features of a three dimensional view of cavitation flow in turbo machineries.

3.3 Pump Design and Optimization.

Several works centred on pump design parameters leading to system modifications with a view to performance optimization. Asuaje, *et al*. (2004a) [45] adopted a numerical optimization approach with the sole aim of performance enhancement, aimed at attaining better performance of the centrifugal pump impeller, their work started with the importation of raw data from one dimensional performance analysis software using Meridional and HELIOX quasi-3D software as Premix, then the new impeller produced a result at off design point with a new flow rate regime, which affirmed the reliability of the adopted procedure, while studying the kinetic field around the design point, the outcome was closely related to the result they obtained from the basic potential calculation, also same with the result obtained from the three dimensional flow simulation, which revealed that at the pump design point, both simulations produced similar trend of result. The outcome for the work promoted the passion to adopt simplified tools as against the traditional use of Computational fluid dynamics software in industrial process analysis. The study also confirmed that with the application of HELIOX software, enormous time can be saved for the applicable analysis as against the use of the traditional three dimensional code approach.

Derakhshan, *et al.* (2013) [46] conducted a study which was said to have produced an approach that successfully developed a centrifugal pump impeller which attained an improved performance and better optimum efficiency for the first time then, using the Artificial Bee-Colony and Artificial Neural Networks algorithm for the optimizations study. The input parameters adopted arbitrary design parameters for the pumps data when generating the impeller geometrical data. Their work revealed that the performance of a centrifugal pump impeller greatly depends on certain elements of the impeller geometry; some of these elements includes

(1) the impeller inlet and outlet angle (2) impeller diameter (3) impeller width (4) impeller hub diameter (5) impeller suction diameter. For the study, the shape of the impeller was designed using Bezier curves, the outcome of the numerical analysis produced a 3.59% positive gain on the pump efficiency with only 6.89m gain on the total pressure difference by the adopted pump. While showing for much improvement on the blade profile, meridian channel as was presented by the new impeller. The new impeller produced substantial improvement on the impeller optimal design with better performance with respect to the adopted ABC algorithms, which implied that the adopted algorithm could be deployed for better pump optimization applications.

Barrio, *et al.* (2011) [47] conducted a test to estimate the total radial force (TRF) on the centrifugal pump impeller using computational fluid dynamics (CFD) approach, the study solved a complete 3-D-URANS model to evaluate various flow rate and flow behaviors ranging from 10% to 130% of the pump rated flow, while validation was made using experimental data obtained from global characteristic as well as data obtained from the pressure distribution within the impeller. A tetrahedral cells method was used to generate the mesh for the analysis due to the complexity of the geometry. While the mesh was refined near specific regions as per those points where remarkable pressure variations and flow turbulent occurred, such regions includes the impeller trailing edge, volute tongue tip and at the impeller leading edge, they also noticed about 0.5mm reduction in the size of tetrahedrons characteristic, the test was conducted at 20% of the pump rated flow rate, while the heat coefficient and the flow coefficient were the adopted reference variables. While from the impeller inner to the outer region noticed a continuous pressure growth as a result of the impulse of the blade, which represents a growth in magnitude with a reduction in flow rate. When evaluating for the impeller blade to tongue relative position at nominal condition of 20% flow capacity, the region near the tongue show the presence of low pressure, while at 130% of the pump rated capacity, same region showed the presence of high pressure, also observed the presence of enormous flow circulation emanating from the pressure side to the suction side of the impeller, which shows the introduction of small low pressure region within the impeller trailing edge.

Asuaje, *et al.* (2004b) [48] did conduct a study to understand the radial thrust on the shaft of centrifugal pumps under different flow capacities, their study adopted the use of a 3-D computational fluid dynamics investigation approach, to fully investigate the performance of the centrifugal pump impeller with respect to: the volute and the casing, using experimental test approach to validate the result from the simulation test for the impeller radial thrust, this was done by ascertaining the pressure fluctuation within the impeller. The result from this turbulence model presented same trend as the result from the simulation test, both results as presented showed that the amplitude of the static pressure was able to reach 27% of the mean pressure generated by the impeller, while at the best efficiency point (BEP), the pump produced the weakest amplitude. When the pump was operated outside the 0.5 capacity of the best efficiency point and at 1.04 capacity of the Best Efficiency Point, the observed pressure pulsation was seen to be able to attain up to maximum of 50 percent of the relative value. While at the impeller periphery, they noticed a non-uniform pressure distribution, which was traced to be resulting from the flow rate condition leading to an unstable radial thrust of a very high amplitude at the impeller tip.

3.4 Unsteady Flow Study on Centrifugal Pumps

Lucius & Brenner (2010) [49] conducted a study using the application of eddy resolved turbulent model to investigate turbo-machinery performance. They adopted and deployed Reynolds average Navier stock model and large eddy simulation approach. The study was able to identify the unsteady rotating stall situations, which was observed to have occurred under partial load condition. The study adopted to deploy ANSYS (2006) for the investigations, while the results were validated using transient measurement data. The integral qualitative and frequency valuation of the transient data was deployed for the correlation of the experimental data. Results from both test were seen to be in agreement with respect to trend of the pressure raise and the efficiency, when correlated with respect to steady state condition at the pump design point, the results from both methods showed similar trend, showing same pressure raise conditions, and presenting that there are some migration of the stall cells with respect to the blading, and they were able to effectively and correctly determine the stall frequency which was usually a very important and complex excitation frequency for the centrifugal pump operation.

Zhang, *et al.* (2018) [23] conducted a numerical study using computational fluid dynamics, with the objective to investigate the performance of a three dimensional incompressible viscous fluid flow on a radial blade centrifugal pump, the test was made to investigate the performance for a quick start-up case of about 0.12 seconds on the pump, the study first analyzed the rotational speed variation of the flow field within the pump impeller using a dynamic slip region approach. Nonconformance grid boundaries and dynamic mesh method was deployed. The performance of transient flow of a swift start centrifugal pump was studied and predicted, by deploying and analyzing the Dynamic Slip Region (DSR) around the impeller, the result from the DSR test was obtained and compared with already existing experimental test results, which affirmed the validity of the adopted DSR method as could be applicable when investigating the instantaneous flow pattern of centrifugal pump under transient operational conditions. Some of the remarkable outcome of the adopted DSR methods was seen to be a perfect predictions of the instantaneous impeller to volute interactions. It was also very clear that the adopted DSR approach could be adopted in resolving unsteady flow simulations in turbo-machineries, especially when reviewing for transient operations, such as during start-up and shut-down operation, variable frequency operations, flow dynamic conditions and several other dynamic operating conditions.

Barrio, *et al.* (2010) [50] conducted a simulation study to investigate the flow through a test centrifugal pump, using computational fluid dynamics tool, they studied the flow pulsation entering a centrifugal pump with respect to the hydraulic interaction between the pump, the analysis was made using Computational Fluid Dynamics model, the result from the analysis shows the existence of flow pulsation at the impeller volute tongue which was as a result of the liquid flow pattern, as was notice from all the data collected from the test at various flow rates ranging from 20% to 160% of the pump rated flow capacity, the test also deployed a numerical test model to investigate the development of flow loses, along and in between the pump impeller tongue gap, and the fluid flow leaving the impeller, through a defined angular intervals of a single blade passage. The flow pattern produced a prismatic cells for the duct, while the presence of tetrahedral shaped cells were seen at the other regions, the presence of no cells were noticed at the region around the leading edge of the impeller as well as around the volute tongue, while 0.000001mm and 0.000003mm sizes of tetrahedrons were the seen size of the tetrahedrons that was visible from the investigation. While also the pressure and velocity production using numerical approach, produced data as was collected from the various critical positions near the tongue region, which revealed that, the flow turbulent from the medium and high discharge capacity was seen to be directly related to the passage of the impeller blade at the front of each of the referenced point. This seen effect was said to be associated to the jet-wake form the associated secondary flow between the suction side of the impeller blade and the internal pressure. While the minimum value for the pressure signal was noticed at the front of the blade pressure side. The presence of vortex flow was eminent with large counter rotation, which was very visible at the outlet of the impeller channel and near the tongue. The presence of stagnation point was noticed close to the blade trailing edge, which was induced by the vortex. Fluid loses through the impeller tongue gap produced fluid pulsation, which was seen to be amplified by about 35% proceeding from 160% flow capacity of the rated flow. Both sides of the impeller tongue edge produced tremendous pulsation, which was emanating from the flow exiting the impeller and was seen to be most critical, and appeared when operating the pump too far from the nominal flow capacity, while the peak-to-peak pulsation of the net flow produced by the operating pump was stable and constant below 0.3% of the pump nominal capacity.

Goonzalez, *et al*. (2002) [5] conducted a numerical simulation and studied the dynamic and unsteady flow effect within a centrifugal pump internal, the study main focus was to investigate the impeller volute hydraulic interaction, while modification of the system static pressure to kinetic energy was performed through flow adjustment. Investigation of the structure near the volute tongue showed the presence of mesh refinement zone. The study adopting a time step of 0.00029 for the unsteady state calculation, while maintaining the two as the second and coherent number, numerical stability and better time accuracy was assured, the presence of spatial pattern at the blade passage frequency was noticed and was attributed to be related to the flow rate as was recorded from both the numerical and experimental approach, fluctuating effect was observed inside the pump volute. The various unstable forces were evaluated using the numerical approach, analysis of the internal flow pattern was done with numerical test approach, and the result showed strong presence of secondary flow, which were prominent in radial position and within the impeller exit, the blade tongue interaction effect was also produced by the pressure fluctuation at the blade pass frequency (BPF), and was seen to be prominent at the impeller exit plane, while the blade tongue interaction clearly magnified the instability level at off design condition.

Majidi, (2005) [51] also conducted a computational fluid dynamic study, with the objective to investigate the three dimensional fluid flow pattern, within the impeller and volute casing of a radial blade centrifugal pump. Outcome of this investigation were adopted to determine the pump impeller and volute interactions, also was applied to predict the liquid pressure distribution within the centrifugal pump impeller and volute chamber, the outcome of the investigation revealed that the flow field within the centrifugal impeller and volute casing, is consistently unstable, also revealed that the hydraulic interactions between the centrifugal pump impeller and pump volute casing is usually predominantly with pressure fluctuations, and that the flow fluctuations are emanating from the impeller outlet and within the volute tongue. While producing a high amplitude of the fluctuation which was seen to be prominent at the impeller exit and at the volute tongue region, these is observed to be fading away as the flow approach the casing, with respect to an increase on advancement angle. The decreasing pressure turbulent was seen to propagate to the discharge nozzle, and is seen to be more visible within the impeller inlet, adopting the dynamic pressure distribution within the pump impeller, the unstable blade loading torque was well predicted, which was seen to be among the most critical reason for vibration evaluation in the centrifugal pump.

3.5 Centrifugal Pump Performance Evaluation Studies

This section will be dealing with the various studies conducted on centrifugal pump performance, either by numerically or experimental approach. Discussing some of the important phenomena that is associated with centrifugal pump performance and operations, some of these will includes factors such as loses and secondary flow within the pump internal, slip factor among others.

Caridad and Kenyery (2005) [52] conducted a numerical study to investigate the slip factor within the centrifugal pump, their test was conducted using a centrifugal pump handling two phase viscos fluid, adopting a study approach of using a 3-D and computational fluid dynamics (CFD) simulation, the result was able to showcase the actual relative fluid angle, which was obtained based on the fluid flow mass averaged, the test was able to present the theoretical head for the finite number of the impeller bade and the value for the slip factor, conclusion that was drown from the test, was that slip factor has a very significant effect on the fluid flow rate as well as the gas volume factor of the two face liquid, while also showing that GVF will increases as the system flow rate reduced, and that a decrease in slip factor will occur with respect to increase in Gas Volume Faction (GVF) for a system with constant flow rate, it was concluded among others that proper understanding of the physical mechanism of the associated flow mechanism, of any system is very vital in any system design, as these will promote reliable and accurate modeling of the system.

Day, *et al.* (2003) [53] conducted a slip factor analysis to investigate a centrifugal pump impeller slip factor mechanism, adopting the impeller outlet flow deviation concept for the correlation study. The study was able to derive a simple kinematic argument concept, with a mathematical structured evaluation, in addition to a refined interpolation for impeller statistical design approach. While also collaborating the result over a wider range of the pump operating envelop with respect to the adopted flow coefficient for the pump, these method was confirmed to be more efficient and accurate in addition with the basic classical formulation approach such as those by Wiesner. Their study also analyzed the application of a 2-D and 3-D impeller modeling using the same concept.

Muel & Nourbakhsh, (2009) [54] conducted an experimental study on the impeller slip factor concept, considering impeller outlet angle with respect to the impellers of five different industrial pumps, at various flow rates, their slip factors value were determined for each of the scenarios; correlating several available methods in determining the values for the adopted slip factor, with main emphasis on the impeller geometrical parameters. The value from the calculated theoretical method was correlated with that obtained from the experimental test, which proved the existence of other factors at off design point which affects the magnitude of the slip factor at such off design conditions. Counted among them includes non-uniform laminar flow, back flow free rotation phenomenon and the non-axisymmetric flow structure among several others. Emphasizing that the slip factor table can to be suitable sufficient and can be adopted when predicting for the slip factor value and the performance at the design point.

Zou, *et al.*(2011) [55] conducted a numerical study to investigate blade thickness effect and slip factor phenomenon, considering the slip factor model with addition of a blockage factor which was able to compensate for the aerodynamic blockage effect of the slip factor. Result from their study was able to comprehend and emphasis the Computational Fluid Dynamics (CFD) result with detailed explanations of the test data, the outcome of their findings revealed those impeller blade factors that could positively affect the slip factor such as aerodynamic blockage of the fluid with respect to the blade thickness. The study revealed that the shroud side thickness of an impeller had more consequential effect on the slip factor than that from the hub side thickness. It also revealed that modifying the thickness of the blade leading-edge will have minimal impact on the slip factor, while addition of more material thickness at the center side of the impeller will reduce the slip factor effect, also revealed that operation with impeller that have very thin trailing edge than another with very tick trailing edge thickness will reduce the impeller slip factor, their model also considered two zone theories and boundary layers, a Qiu's slip factor model was developed with addition of a new blockage factor, the phenomenon adopted in the CFD test was fully explained using the developed model.

In another study by Ji, *et al.* (2011) [56] in which they developed a new slip factor correlation model for a mixed flow and radial blade centrifugal pump impeller, with which inference was made from Euler equation fundamentals for turbo machineries and relative eddy theory. they concluded that beside geometrical parameters, slip factor is a function of systems flow coefficient, thus the study was able to accurately predict the slip factor values from various flow conditions through the system, as well as been adopted for validation of several cases owing to its accuracy level.

Elsheshtawy, (2012) [57] conducted a slip factor investigation study using numerical investigation approach, he studied the various flow parameters which includes the pump discharge capacity, the blade number, impeller exit angle, process fluid type and process fluid properties, the study revealed that earlier adopted method by Busemann was very efficient but not to be applicable in all cases, as they discovered some limitations, because their model did not consider all other associated condition that could affect the slip factor, such as increase of the number of blade on the impeller, which will in practice positively alter the value for the slip factor, also by increasing the value of the energy transfer to the working fluid, but the aforementioned will also reduce the area of the fluid passage way through the pump, thereby restricting the fluid flow channel and reducing the capacity of the impeller, while affirming that one of the most critical factor which affects the value of the slip factor is the value of the blade exit angle, as this will usually predict and determine the slip factor trend with respect to increase or decrease in value, it will also predict the conditions of the impeller operations.

3.6 Centrifugal Pump Loses

Centrifugal pump impeller performance as was generally revealed by Euler pump equation, it represented a simple and loss-free description of the centrifugal pump impeller performance, but in actual operations, owing to several hydraulic, volumetric and mechanical loses, all the actual loses through the centrifugal pump impeller was not fully accounted for by the Euler equation, thus the actual performance of the centrifugal pump is usually lower than that predicted by Euler equation, and several further studies have been conducted in trying to validate by incorporation of all the applicable associated loses encountered by the pump in operation.

Khalil, *et al*. (2008) [58] conducted a study on improvement of centrifugal pump performance and efficiency, their study was performed on a very low specific speed centrifugal pump, two major investigation were conducted, one was a pump performance test investigation, and the second was a free impeller test, on which three different impeller of low specific speed was analyzed, while the performance test adopted the combination of two volute and six impeller combination. The outcome of both findings were correlated with the outcome from the capacity of the 3-D approach, and both findings revealed that, with a very low specific speed impeller pump, the observed disc friction was seen to be very high, and impeller efficiency was seen to be low, in addition to the presence of large flow recirculation within the impeller channel, even more visible at the pump best efficiency point (BEP), which resulted in a very high slip factor, and has promoted higher hydraulic loses which were not also very massive in relation to ordinal impeller. Under very large flow rate region, they observed a reduction in magnitude of the reverse flow and minimal flow circulation around the impeller. The pump efficiency was also seen to have improved along a wider range. They observed the challenges of designing a pump with low specific speed with improved performance and efficiency using the conventional method, emphasizing that the slip factor is directly related to the pump flow rate. Thus they recommended the possibility of adopting the design with narrower volute width, with wider impeller width for attainment of better efficiency. The study recommended that control of volute throat by possible insertion of spacer on the impeller neck region, and that a reduction in the value of impeller width will lead to a gradual drop in the pump efficiency, while a reduction in volute width will drastically reduce the efficiency of the pump.

Wen-Guang (2011)[59] also conducted an experimental investigation on centrifugal pump, analyzing the performance of the pump with water and viscos oil application, the kinematic viscosity of the oil and water were seen to be $48mm^2/sec$ and $1mm^2/sec$ respectively, two dimensional laser Doppler Velocity meter was adopted to accurately measure flow through the pump at the best efficiency point, and the data acquired at part-load condition while in operation for each of the process fluid, investigated during the test includes the effect of viscosity on the pump performance, and the fluid flow pattern around the impeller, the result revealed that some of the causal effect for centrifugal pump performance degradation when handling high viscos fluid, was attributed to the observed increase in disc friction loses around the pump impeller periphery, and those within the impeller shroud and hub, also with proven hydraulic losses along the flow channels. The study also revealed that the viscous fluid disc friction phenomenon will have insignificant effect on the flow pattern within the impeller exit at the pump best efficiency point (BEP), under part load condition. Whereas the disc friction effect as a result of the high viscosity, was seen to have affected greatly the flow panther within the impeller inlet, the blade suction side of the impeller showed the presence of a wide-wake, with the absence of flow-jet near the bade pressure side, these presented a very different flow pattern which was very different from the already known jet-wake model.

Gandhi, *et al*. (2002) [60] conducted a study to investigate the performance of an un-shrouded centrifugal pump impeller on a solid-liquid mixture application, they analyzed the impact of the clearance between the casing and impeller tip, with respect to the concentration of the solid, the liquid density and the impeller diameter with respect to the pump performance, with the various operating conditions, the outcome of the study shown as good correlation and agreement with the result of the experimental test form their former work, the result showed a decrease in pump efficiency with increase in the impeller tip clearance under the present test condition.

Weng-Guang, (2011) [59] also conducted an experimental investigation on the performance of radial blade centrifugal pump on an oil transfer application, the objective was to investigate and determine the correction factor to be adopted from the oil application, with a view to determining the correction factor to be adopted for same pump under water application, the test revealed that the head correction factor was with respect to the pumping system conditions. When analyzing the viscosity factor, its effect and correlation factor, the pump operating condition was seen as a major factor to be considered in determining the correlation factor, a large error of up to 10% was observed when adopting the existing correlation factor from the water application to determine the performance of same pump in oil application. Wen-Guang (2004) [61] conducted an experimental investigation to evaluate the performance of centrifugal pump on a pump application that is handling liquid of variable density and viscosity, the test process adopted the method of replacement of the impeller blade from three number of blade to seven impeller number, using water of 1cSt kinematic viscosity as the process fluid, while the kinematic viscosity of the adopted oil ranges from 29 to 255cSt, the investigation was to ascertain the impact of blade number on the hydraulic performance of the centrifugal pump on the oil application. The results from this test revealed that the number of impeller blade has significant impact on both the hydraulic and mechanical performance of a centrifugal pump, and that an increase on the liquid viscosity will alter the pump general performance. The result of the findings presented that with kinematic viscosity of 200cst, the blade number with optimum performance was seen to be 5, but proved the optimum number of blade to be 3 when working with a fluid of kinematics viscosity of above 200cst, thus fewer number of blade is recommended for pumping of high viscous fluid.

Vlachakis & Baldoukas (2002) [62] conducted a test and was able to develop an empirical equation for the analysis of centrifugal pump performance, with respect to the relationship between the various pump parameters such as the pump head and flow, speed and flow capacity, pump power and flow capacity, net positive suction head and pump capacity for centrifugal pump. They adopted to deploy small number of pump geometric data, and the outcome of the test revealed that the attained performance curve from the experimental test and that obtained from the empirical test showed good correlation, the overall recommendations was that the expected performance of a centrifugal pump can be effectively predicted using mathematical modeling, with not much requirement for a costly experimental test, which will be a cost effective decision with respect the cost and time savings.

Baun & Flack, (2003) [63] conducted an experimental research using a four and a five vane impeller pump, with a view to comparing the performance characteristic of the values obtained from the measured impeller lateral force and the value obtained for the pump hydraulic performance, for both impeller vane numbers, the test was conducted with each impeller on a spiral volute, double volute and a concentric volute, the result was obtained for each of the five vane impeller test, and from each of the three volute type, also correlated was the outcome of the four vane impeller, and the result showed a clear decrease in the shut-off head coefficient, when operating with the five-vane impeller. It also revealed a decrease on the zone of unstable flow as the pump speed decreased below 0.6 of the pump nominal speed, but the attained head for the operation with concentric volute pump increased to an unstable level at a speed below 0.4 of the nominal speed. They also noticed that the head produced by the double volute pump appeared unstable below 0.4 value of the nominal speed, and the five vane impeller was seen to have developed better head compared to the four vane impeller on each of the volute arrangement.

Day, *et el.* (2003) [53] conducted a study on a small centrifugal pump with a view to investigate the effectiveness of the application of traditional affinity law and its suitability, limitations and consequence on viscous fluid application, using Reynolds number concept with respect to the pump performance, the study was conducted through numerical and experimental investigation using a small implantable centrifugal pump model, adopted impeller diameter was 45mm having long spiral volute, the test was conducted at a rotor speed of 500 and 3000RPM, the experimental test results for both the head, and flow values were determined and plotted for the two different viscous fluid applications. From both the numerical and experimental test, the findings proved the effectiveness of the traditional flow modifications as is affirmed by the affinity law for the investigation, and recommended for further studies on the characteristic for evaluating the performance of smaller pumps at lower operating speed (RPM), they also affirmed that Reynolds number is very critical and should be considered for proper scaling of the viscous effect, for the numerical investigation and not applicable for the experimental investigation.

Pazzi, *et al*. (2023) [64] conducted an optimization study on a typical centrifugal pump impeller, with the aim to investigate the impeller performance under low flow conditions, also to use an advance intelligence automatic program (IAP) to investigate the cylindrical blade performance, using feasible sequential quadratic program algorithmic, the adopted system was able to resolve geometrical constrains as it eliminated the various computational constrains on analysis of the pump physical features, the outcome of their work showed that the algorithm is acceptable, as it was capable of predicting the required optimization for the centrifugal pump without much human effort, and the program was recommended to have met the requirement for the general industry application, as an instrument which is capable of designing an equipment component parts without much human effort.

Moreover, Ref [61] conducted some parametric studies, with a view to investigate submicron characteristics within the centrifugal pump impeller when in operation, some of the targeted element includes the pump mechanical efficiency, hydraulic efficiency, volumetric efficiency and slip factor, the evaluation considered the impeller geometry and its characteristics at the best efficiency point of the pump. The findings from the study presented the following observations, that slip factor was seen to have developed, grew and reduced with an increase in the fluid viscosity, this effect was more pronounced when operating the pump with more higher a viscous fluid than the former. And that the hydraulic efficiency of centrifugal pump will reduced with increase in process liquid viscosity, and these was been attributed to the higher energy loses due to friction at the interface of the casing to impeller and the pump volute, the volumetric efficiency was also seen to have increased or remained constant when the viscosity grew, those was linked to be as a result of the reduced recirculation and leak flow within the impeller wearing ring region. While a drop in mechanical efficiency was noticed with increase on the fluid viscosity. Their study recommended that, during the design of centrifugal pumps, disc friction loses needs to be effectively evaluated and considered for reduction on any adopted design, in order to increase the pump overall performance by reduction of the flow circulations within the impeller.

Choi*, et al.* (2006) [65] also conducted an investigative study on centrifugal pump impeller, analyzing the internal flow structure produced by a low speed centrifugal pump of a semi open impeller type, they adopted to deploy two impellers, one with a specific speed of 1.14 and the second with a specific speed of 0.42. The outcome of their findings revealed that the attained head (m) and the ultimate efficiency from a close type impeller, was much higher than the head (m) attained with a semi open impeller. Also very visible was the presence of operational instability, with respect to rising head curve with flow increase with an increase on the impeller tip clearance, leading to a reduction in some level of performance instability in a semi-open impeller, proving that with a well calculated modifications of the impeller tip clearance, the performance of a semi open impeller can be manipulated to overcome the operational instability than a close impeller, the presence of large reverse flow structure was also eminent in semi open impeller, through the tip clearance area close to the outlet portion of the impeller, and returning to the main stream at the impeller passage way. Also noticed was that the reverse flow will considerably decrease the absolute tangential velocity at the impeller outlet, while the tip clearance ratio will determine the absolute tangential velocity at the impeller outlet, and above phenomenon did have a considerable influence of the overall performance of the pump

Khalil, *et al.* (2008) [58] conducted an experimental investigation to ascertain the performance characteristic of the centrifugal pump under variable process applications, under stable and unstable oil-in water emulsion, empirical result were obtained from the pump head and flow capacity at various process fluid temperature, the conducted test correlated the result between the produced head and flow capacity for the various pump, using the emulsified liquid test result and that of water, and considering the various flow rate, and attained head for both process fluid. The result was derived for both stable and unstable flow conditions as well as correlated with that of the water flow. The overall result showed that there was a reduction in head for the oil in water application, same was also observed for the hydraulic efficiency and the flow rate through the centrifugal pump, as the oil-in-water bobble reduces, the fluid flow rate increases, while the temperature decrease, the head also increased, less decrease in load was observed from the unstable oil-in-water emulsion, same decrease in flow rate was observed, larger decrease in produced head was noticed from stable emulsion, when compared with that for normal water. The head and flow rate was also affected when the emulsion was prepared using Sodium Dodecyl Sulfate solution, which presented a system with minor reduction in head and flow rate, when compared with that of normal emulsion, such that with organic acid base and amine base origin, there was a great variation in the rheology of the emulsion in addition to build-ups, the internal energy change within the microstructures were also seen as been responsible for the accelerated loses within the pump, as well as the variation of the pump performance, those were responsible for the reduction in the pump efficiency with increase in the build-up and the decrease in the system internal energy.

On another study by Zhang, *et al*. (2009) [66] on which they conducted a numerical investigation in addition to an experimental examination of centrifugal pump impeller, with respect to a non-overloading design of the impeller, their study considered two impellers with different impeller outlet angle, and another two impellers of different impeller type as was examined, with a view to understanding and presenting the findings on the performance of the sectional area of the pump volute with respect to the performance of a low specific speed pump, findings from their work shows that the performance and efficiency of those pumps with low specific speed is seen to be strongly dependent on the impeller volute characteristic, showing that at operating point below the pump best efficiency point, the volute characteristic superseded the impeller head characteristic. These was seen to be resulting from the instability at the impeller exit, which was induced by the flow circulations and the backflow within the impeller-volute exit, but when operating the pump at a point above the pump best efficiency point, a remarkable decrease on the pump head which was resulting from a large flow lose within the volute during the large flow condition, the study also revealed a signification parts of friction losses, which was responsible for the large volute loss from a low specific seed pump, these losses amplified greatly with rise in flow velocity within the pump volute, it also confirmed changes in pump power consumption with respect to changes on the volute cross sectional area, and when operating with an impeller that has small outlet exit angle, it was observed that the volute cross-sectional area has little or insignificant effect on the overall hydraulic performance of the impeller and the entire pump.

4. Conclusions

Based on the fact that Centrifugal Pumps are expected to work efficiently over a defined range of flow and operating conditions, it become necessary to carry out detail empirical studies needed to evaluate and ascertain the performance and efficiency of the pump under these various ranges of flow and operational conditions. Thus, this systematic empirical review showed that most centrifugal pumps in operation faces variant process operating conditions which are not always in agreement with the actual design conditions of the pump. Hence, since the pump is usually selected for operations within specific range of process conditions, it was recommended that under critical operational exigency, field operators and maintenance teams can adopt to make some calculated field modifications in order to improve the pump performance, ensure and maintain the pump availability to sustain operation and to maintain production. One workable approach commonly adopted by experienced operators and maintenance team to improve on the performance of centrifugal pumps under some drastic changes on the process fluid conditions is modifications of the pump impeller. The outcome of a well calculated impeller modifications under such drastic process fluid change conditions, have actually improved the pump performance via the reduction of some internal cross flow, reduction in excessive power consumption, impeller modifications are usually adopted as one of the remedies to manage a phenomenon of extreme changes in the system process fluid density, thus impeller modification when adopted are usually targeted at reducing the overall system losses and flow instabilities aimed at improvement of the system operational performance and system stability, but with great caution and attention to ensuring a reduction in secondary incidence flows and decrease in backflow areas around the impeller region.

References

[1]. Adamkowski, A., & Lewandowski, M. (2016). Resonance of torsional vibrations of centrifugal pump shafts due to cavitation erosion of pump impellers. *STGM Journal on Engine Failure Analysis17,* 56–72.

[2]. Ahmed, F. (2015). Experimental and Computational Study of Semi-open Centrifugal Pump: *Research gate publication No 293806626*

[3]. Anagnostopoulos, J. (2006). CFD analysis and design effects in a radial pump impeller. WSEAS Transactions in Fluid Mechanics, 1(7), 763-770.

[4]. Anagnostopoulos, J. (2006). Numerical calculation of the flow in a centrifugal pump impeller using Cartesian grid. Proceedings of the 2nd WSEAS Int. Conference on Applied and Theoretical Mechanics, November 10 – 15, 2006.

[5]. Anagnostopoulos, J. (2009). Numerical method for flow analysis and blade design in centrifugal pump impellers. *Computers & Fluids, 38,* 284-289.

[6]. Appiah, D., Zhang, F., & Yuan. (2018). Effects of geometrical conditions on the performance of a side channel pump: a review. *International journal on Energy Research*. *42,* 416–428.

[7]. ArunKumar, P., & Saravanakumar, K. (2014). Analysis and investigation of centrifugal pump impellers using CFD. *EST International Journal,4,* 30 – 37.

[8]. Asuaje, M. F., & Bakir, S. (2002). *Optimize of centrifugal pump in 2D. CFD.* Windsor Publication.

[9]. Asuaje, M., Bakir, F., Kouidri, S. & Rey, R. (2004). Inverse design method for centrifugal impellers and comparison with numerical simulation tools. *International Journal of Computational Fluid Dynamics,18* (2), 101-110.

[10]. Bacharoudis, E., Filios, A., Mentzos, M., & Margaris, D. (2008). Parametric study of a centrifugal pump impeller by varying the outlet blade angle. *The Open Mechanical Engineering Journal*, *2* (2), 75 - 79.

[11]. [11] Barrio, R., Fernandez, J., Blanco, E., & Parrondo, J. (2011). Estimation of radial load in centrifugal pumps using computational fluid dynamics, *European Journal of Mechanics-B/Fluids*, *30* (5), 316-324.

[12]. Barrio, R., Parrondo, J., & Blanco, E. (2010). Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. *Computers & Fluids*, *39* (2), 859-870.
[13]. Baun, O., & Flack, R. (2003). Effect of volute design and number of impeller blades on lateral impeller forces and hydraulic performance, *International Journal of Rotating Machinery,9* (4), 145 -152.
[14]. Benigni, H., Jaberg, H., Yeung, T., Salisbury, O., Berry, T., & Collins, T. (2012). Numerical simulation of low specific speed American petroleum institute pumps in part-load operation and comparison with test rig results. *Journal of Fluids Engineering*, 5 (2), 134 – 139.
[15]. Caridad, A., & Kenyery, F. (2005). Slip factor for centrifugal impellers under single and two-phase flow conditions. *Journal of Fluids Engineering*, *127* (233), 317-321.
[16]. Carravetta, A., Fecarotta, O. & Ramos, H. (2011). Numerical simulation on pump as turbine: mesh reliability and performance concerns. *Proceedings of the international conference on clean electrical power (ICCEP), New York.*
[17]. Chakraborty, S., Pandey, K., & Roy, B. (2012). Numerical analysis on effects of blade number variations on performance of centrifugal pumps with various rotational speeds. *InternationalJournal of Current Engineering and Technology, 3* (4), 143-152.
[18]. Chappell, R., Hickman, W. & Seegmiller, S. (1982). *Pumps as-turbines experience profile.* Publication of Washington, DC, USDE.
[19]. Cheah, K., Lee, T. &Winoto, S. H &Zhao, Z.M. (2007). Numerical flow simulation in a centrifugal pump at design and off-design conditions. *International Journal of Rotating Machinery,* 2007, 1-8.
[20]. Choi, D., Kurokawa, J. & Matsui, J. (2006). Performance and internal flow characteristics of a very low specific speed centrifugal pump. *Journal of Fluids Engineering*, *12* (8), 341-349.
[21]. Cui, B., Li, X., & Rao, K. (2018). Analysis of unsteady radial forces of multistage centrifugal pump with double volute. *Eng. Computational analysis, 35* (2), 1500–1511.
[22]. Cui, B., Lin, Y., & Jin, Y. (2011). Numerical simulation of flow in centrifugal pump with complex impeller, *Journal of Thermal Science,20* (1), 47-52.
[23]. Day, S., Lemire. P., Flack, D. & McDaniel, J. (2003). Effect of Reynolds number on performance of a small centrifugal pump. Proceedings of FEDSM'034TH ASME_JSME Joint Fluids Engineering ConferenceHonolulu, Hawaii, USA, July 6–11, 2003.
[24]. Derakhshan, S., & Nourbakhsh, A. (2008). Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation. *Computers & Fluids*, *16* (32), 1620–1627.
[25]. Derakhshan, S., Pourmahdavi, M., Abdolahnejad, E., Reihani, A. & Ojaghi, A. (2013). Numerical shape optimization of a centrifugal pump impeller using artificial bee colony algorithm. *Computers & Fluids, 81* (131), 145-151.
[26]. Djerroud, D., Dituba, N., & Ghie, W. (2010). Numerical identification of key design parameters enhancing the centrifugal pump performance: impeller, impeller-volute, and impeller-diffuser. *Mechanical Engineering, 1(2), 10 – 15.*
[27]. Doshi, A., Channiwala, S. & Singh P. (2017). Inlet impeller rounding in pumps as turbines. An experimental study to investigate the relative effects of blade and shroud rounding. *Wisden International Journal.7* (82), 333–348.
[28]. Elsheshtawy, H. (2012). Numerical study of slip factor in centrifugal pumps and study factors affecting its performance. *Proceedings of 2012 International Conference on Mechanical Engineering and Material Science*, *Yangzhou, China,* May 2 – 7, 2012
[29]. Fernandez, J., Barrio, R., & Blanco, E. (2010). Numerical investigation of a centrifugal pump running in reverse mode. *IMech E Journal on Power and Energy*, *22* (4), 373–381.
[30]. Gandhi, B.K., Singh, S. N. & Seshadri, V. (2002). Effect of speed on the performance characteristics of a centrifugal slurry pump. *Indian Journal of Hydraulic Engineering,49*(1), 253-233.
[31]. Gonzalez, J., Fernandez, J., Blanco, E. & Santolaria, C. (2002). Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump, transactions. *American society of Mechanical Engineers Journal of Fluids Engineering*, *12* (4), 348-355.
[32]. Grapsas, V., Stamatelos, F., Anagnostopoulos, J. & Papantonis, D. (2008) Numerical study and optimal blade design of a centrifugal pump by evolutionary algorithms. *Knowledge-Based Intelligent Information and Engineering Systems, 2* (2), 26-33.
[33]. Hofmann, M., Stoffel, B., Coutier-Delgosha, O., Fortes-Patella, R. & Reboud, J. (2001). Experimental and numerical studies on a centrifugal pump with 2D-curved blades in cavitating condition.*Journal of Fluids Engineering*, 125(6): 970-978.

[34]. Jafarzadeh, B., Hajari, A., Alishahi, M. & Akbari, M. (2021). Flow simulation of a low-specific-speed high-speed centrifugal pump. *Applied Mathematical Modelling, 35* (1), 242-249.

[35]. Jain, S., & Patel, R.N. (2014). Investigations on pump running in turbine mode. Review of the state-of-the-art. *Renew Sustain Energy, 30* (1), 841–868.

[36]. Parrondo-Gayo, J.L., Gonz- alez-P-erez, J., & Fern Andez-Francos, J. (2002). The effect of operating point on the pressure fluctuations at the blade passage frequency in the volute of a centrifugal pump. *Journal of Fluids Engineering*, *124* (3), 784–790.

[37]. Khalil, F., Kassab, Z., Ismail, A. & Elazab, I, S. (2008). Centrifugal Pump Performance under Stable and Unstable Oil-Water Emulsions Flow. *Proceedings of the 20th International Water Technology Conference, Alexandria, Egypt*., May 18-20.

[38]. Li, J., Zeng, Y., Liu, X., Wang, H. (2022). Optimum design on impeller blade of mixed-flow pump based on CFD. *Journal of Procedural Engineering*, *31* (1), 187-195.

[39]. Lucius. A., & Brenner, G. (2010). Unsteady CFD simulations of a pump in part load conditions using scale-adaptive simulation, *International Journal of Heat and Fluid Flow*, *31*(1), 1113-1118.

[40]. Majidi. K., (2005). Numerical study of unsteady flow in a centrifugal pump. *Journal of Turbo-machinery, 12* (7), 363-371.

[41]. Milne, A. (1986,). *A comparison of pressure distribution and radial loads on centrifugal pumps. In: radial loads and axial thrusts on centrifugal pumps.* Seminar Organized by the Fluid Machinery Committee of the Power Industries Division of Mechanical Engineers, January 4 – 9, 1986.

[42]. Muel, M., & Nourbakhsh, A. (2009). Experimental investigation of slip factors in centrifugal pumps. *Experimental Thermal and Fluid Science, 33*(1), 938-945.

[43]. Nan, G., Chen, E., & Yang, A. (2015). Dynamic characteristics of centrifugal pump induced by fluid and motor excitation forces. *Journal of Vibro-engineering*, 17(1), 3909–3918.

[44]. Pandey, R. (2008). Performance analysis of modified pump used as turbine. *Mechanical Technical Dissertation, AHEC, IIT Roorkee,* Roorkee.

[45]. Patel, J., Barad, G., & Chaudhari, V. (2015). Experimental investigation of centrifugal pump regular and reverse mode. *International Journal of Science and Research Development*, *2*(1), 542–545.

[46]. Pazzi, F., Martelli, S., Michelassi, V., Berghen, F., & Bersini, H. (2003). *Intelligent performance CFD optimisation of a centrifugal impeller.* Fifth European Conference on Turbo-machinery, Prague, CZ, March 24 – 29 2003.

[47]. PRISMA (2020). Format for document selection and screening process for Systematic Review.

[48]. Quan, H., Cheng, J., Guo, Y., Kang, L., & Peng, G. (2020). Influence of screw centrifugal inducer on internal flow structure of vortex pump. *Journal on Fluids Eng.-TransASME, 14* (2), 105 – 117.

[49]. Singh, P. (2005). *Optimization of the internal hydraulic and of system design in pumps as turbines with field implementation and evaluation.* PhD Thesis, University of Karlsruhe, Karlsruhe.

[50]. Singh, P. & Nestmann, F. (2011). Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines. *Experimentations on Thermo Fluid Science, 3* (5), 121–134.

[51]. Suarda, M, Suarnadwipa, N. & Adnyana, W. (2005). *Assessment performance of pumps-as-turbines.* Proceedings of the international conference on energy and environment, Stavanger. October 12–15, 2005.

[52]. Sun-Sheng, Y. & Fan, J. (2012). Numerical research on effects of splitter blades to the influence of pump as turbine. *International Journal on Rotating Equipment*, *2*(2), 19 29.

[53]. Thome, E. (1979). *Centrifugal pumps as power recovery turbines.* Proceedings of the 6th BPMA conference on Pumps, Canterbury, New York: ASME. March 19–28, 1979.

[54]. Vlachakis, N. & Baldoukas, A. (2002). *Centrifugal pumps performance curves estimation using empirical equations.* WSEAS 4th Mechanical Engineering Multi conference, Cancun, Mexico, May 12-16 2002.

[55]. Wang, L., Liao, J, Guo, W., Zhu, P., & Luo, X. (2021). Numerical investigation on vibration and pressure fluctuation characteristics in a centrifugal pump under low flow rate. *Journal of Mechanical Engineering & Science*, *23*(5), 888–899.

[56]. Wen, G. (2004). *Experiment on impeller trimming of a commercial centrifugal oil pump.* International Conference on Hydraulic Machinery and Hydrodynamics. June 10 – 15, 2004.

[57]. Wen-Guang, I. (2008). Numerical study on behavior of a centrifugal pump when delivering viscous oil - part 1: performance. *International Journal of Turbo and Jet Engines*, 25(130), 61-79.

[58]. Wen-Guang, I. (2011). Blade exit angle effects on performance of a standard industrial centrifugal oil pump. *Journal of Applied Fluid Mechanics*, *4*(1), 105-119.

[59]. Yang, H., Liu C., & Tang, F, (2007). Experiment and numerical simulation of centrifugal pump with different wrap angle. *Chinese Journal of Mechanical Engineering*, *43*(10), 166–169.

[60]. Yang, S., Liu, H., & Kong, F. (2014). Effects of the radial gap between impeller tips and volute tongue influencing the performance and pressure pulsations of pump as turbine. *Journal of Fluids Engineering*, *136* (5), 150–164.

[61]. Yang, S., Derakhshan, S., & Kong, F. (2012). Theoretical, numerical and experimental prediction of pump as turbine performance. *Renew Energy*, *48*(2), 507–513.

[62]. Yuan, J., Shi, J., Fu, Y., Chen, H., Lu, R., & Ho, X. (2022). Analysis of fluid-structure coupling dynamic characteristics of centrifugal pump rotor system. *Energies*, 15(6), *1-18.*

[63]. Zhang, F., Appiah, D., & Yuan, S. (2018). Transient flow characterization in energy conversion of a side channel pump under different blade suction angles. *Energy*, 161, 635–648.

[64]. Zou, C., Ji, X., Ruan, P. (2011). A new correlation for slip factor in radial and mixed-flow impellers, Proceedings of the Institution of Mechanical Engineers, Part A: *Journal of Power and Energy*, 22 (5), 114-119.