Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023

MODIFIED DESIGN AND FLOW KINEMATIC ANALYSIS OF AXIAL FLOW PUMP IMPELLER BLADE USING CFD

Mr. K. S. Mohanraj, Assistant Professor, Dept. Of Mechanical Engineering, Sri Shakthi Institute of Engineering and Technology, Coimbatore..
Sabareeshan P, UG Scholar, Dept. Of Mechanical Engineering, Sri Shakthi Institute of Engineering and Technology, Coimbatore.
Shiva Shankar R, UG Scholar, Dept. Of Mechanical Engineering, Sri Shakthi Institute of Engineering and Technology, Coimbatore.
Sivathatchana R, UG Scholar, Dept. Of Mechanical Engineering, Sri Shakthi Institute of Engineering and Technology, Coimbatore.
Yokesh T, UG Scholar, Dept. Of Mechanical Engineering, Sri Shakthi Institute of Engineering and Technology, Coimbatore.

Abstract

If you want your axial flow pumps to function at their best across a variety of industrial uses, you need to do a thorough flow study. Fluid flow through axial flow pumps can be simulated with the help of computational fluid dynamics (CFD), which can then be used to locate potential issue areas such as flow separation, recirculation, or cavitation. In this research, we demonstrate how to use computational fluid dynamics (CFD) analysis to enhance the efficiency and effectiveness of an axial flow pump. Methodology steps include: generating a 3D model of the pump shape; defining acceptable boundary conditions; defining fluid properties; selecting appropriate solver settings; visualising and analysing results; and using post-processing tools. The findings from the CFD study (Steady flow analysis the K-omega governing equation) can be used to refine the axial flow pump's design for maximum efficiency and performance. You can change the shape of the impeller's blades to minimise flow separation and recirculation, or you can rebuild the diffuser to boost pressure recovery and cut down on energy waste. The axial flow pump's potential for cost savings, efficiency gains, and diminished environmental impact stems from its well-optimized design. In conclusion, this research shows how CFD analysis may be used to improve axial flow pumps' designs and outputs, and it provides a helpful methodology for engineers and researchers in the field. The software programme Solidworks 2022 was utilised in order to create the model of the flow pump impller, and the numerical investigation (Flow characterization) was carried out with the help of the CFD dynamic tool Ansys18.0. Finally, to get the optimized design of pump impeller of prototype model by 3D- printing technology the Machine "Flashforge Guider II S" and the slicing software "Ultimaker Cura version 4.12.1" was used.

1. Introduction

Because of their efficiency at pumping huge volumes of fluid at low pressures, axial flow pumps find widespread use in a variety of industrial settings. These pumps see widespread application in HVAC, irrigation, and sewage systems. Axial flow pumps' efficiency is affected by many variables, including the fluid's characteristics and the pumps' impeller and diffuser designs. Flow separation, recirculation, and other flow abnormalities that might lead to inefficiencies or other problems can be modelled and located via CFD simulations of axial flow pumps. Engineers can increase the pump's performance and efficiency by using CFD analysis to get insight into the flow patterns and flow attributes.

The purpose of this research is to conduct a computational fluid dynamics (CFD) flow analysis of an axial flow pump with the aim of optimising the pump's design for greater efficiency. The procedure entails building a 3D model of the pump's shape, followed by mesh development, boundary condition selection, fluid property specification, solver parameter selection, and final result post-processing. This analysis will reveal information about the pump's flow characteristics and flow

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023
patterns, which can be used to enhance the pump's design. Costs may be reduced, efficiency can be increased, and the environmental effect of industrial processes can be lessened all thanks to CFD analysis in the design and optimisation of axial flow pumps. As a result, this discovery is crucial for engineers and scientists engaged in fluid mechanics and pump design.
The axial-flow pump impeller is the heart of a pump device and is considered to be its most significant flow component. The outcome of the design process directly influences how the pump device, as well as the overall pumping station, will perform in its intended capacity. In recent years, about half of the yearly production of axial-flow pump impellers has been utilized to replace items that were trashed due to blade issues. This trend has continued across the industry[1]. a multi-disciplinary resilient design optimization approach based on time-varying sensitivity analysis via a multi-disciplinary optimization design was proposed as a solution for the problem of time-varying uncertainty. In the areas of launch vehicle design optimization, hard rock tunnel boring machine performance design optimization, and all-electric GEO satellite design optimization, MDO offers a viable option[2]. There have been a number of CFD-based research projects focused on the design and performance optimisation of axial flow pumps. The impact of impeller blade shape on the flow patterns and performance of an axial flow pump, for instance, was investigated using CFD. They discovered that changing the blade's angle and curvature led to a notable rise in the pump's efficiency. [3] Diffusion enhancement in axial flow pumps was the subject of [4], another investigation. After running computational fluid dynamics (CFD) simulations through a number of different diffuser shapes, they concluded that the most effective diffuser in terms of both pressure recovery and efficiency was one with a curved form and a narrow input diameter. CFD has been utilised for other purposes besides performance optimisation in axial flow pumps, such as flow analysis and troubleshooting. To find out where high turbulence intensity and pressure fluctuations were generating vibration in an axial flow pump, for instance[5], computational fluid dynamics (CFD) was utilized[5]. CFD has also been used to investigate the effect of different operating conditions on the efficiency of axial flow pumps. The performance of an axial flow pump, for instance, was studied using CFD to determine how factors such as rotational speed, flow rate, and impeller blade angle affected pump operation. They discovered that increasing the blade angle decreased efficiency [6], but increasing the rotational speed and flow rate increased pressure rise and flow rate.
Conventional axial flow pumps aren't the only ones that computational fluid dynamics has been utilised on; it's also been applied to unorthodox designs. For instance, CFD was used to study how well a pump that blends axial and centrifugal flow characteristics (a hybrid axial flow pump) performed. When comparing the hybrid design to a conventional axial flow pump, they discovered that the latter had a higher flow rate and pressure rise [7]. Additional research has used CFD to optimise individual components of the axial flow pump. It has been shown, for instance, that using CFD to optimise the impeller blade tip clearance in an axial flow pump improves pump performance and efficiency[8].
As a whole, CFD's application to the analysis and design optimisation of axial flow pumps is an active study area with room for growth and development. Engineers and scientists can increase the pump's performance, efficiency, and dependability across a wide range of industrial applications by first gaining a thorough understanding of the fluid flow patterns and features within the pump.

1. Aim & Objectives

- To design a high flow rate axial flow pump using Solidworks software
- To develop an impeller for axial flow pump with higher discharge capacity and automate the design process.
- To conduct the flow analysis of axial flow pump blade with CFD and iudentify the suitable design
- To make the prototype with 3D printing technology

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023

2. Materials and methods

Common components and procedures for a CFD analysis of an axial pump are as follows:

2.1 Materials:

The 3D geometry of the pump was modelled using computer-aided design tools.
Pump fluid flow simulation using computational fluid dynamics software.
Tools for running computations, such as a high-performance computing cluster.
Density, viscosity, and temperature are all important physical characteristics of the pumped fluid.

2.2 Methods:

The geometry of the pump is modelled in a CAD programme and exported to a format that can be read by the CFD programme.
Mesh generation: Using the meshing tools in the CFD software, the pump shape is broken down into smaller pieces or cells.
The CFD program's boundary conditions include the fluid's inlet and output flows, pressures, and characteristics.
Convergence criteria, turbulence models, and numerical methods for solving the Navier-Stokes equations are all examples of solver parameters that can be found in CFD software.
In a CFD simulation, the Navier-Stokes equations are solved at each element of the mesh, yielding the flow's velocities, pressures, and other variables.
Extracting data like flow rate, pressure drop, and velocity profiles from the CFD results requires post-processing with visualisation tools.
The overall accuracy and reliability of the results from a CFD analysis of an axial pump are dependent on the materials and methods utilised to conduct the analysis.

3. AXIAL FLOW PUMP

Axial flow pumps are a subset of positive displacement pumps that move fluid by channelling it along the pump's axis with the use of propeller-like blades. Like aeroplane wings, the rotating pump blades create lift to move the fluid along the pump's axis as it enters the pump through an inlet. The fluid is subsequently released through an outlet placed behind the rotors. In large-scale applications like irrigation systems, power plant cooling water transfer, and chemical processing plant fluid transfer, axial flow pumps are frequently employed. They are more energy-efficient than other types of pumps for high-flow, low-pressure applications and can handle enormous volumes of fluid at relatively low pressures.

3.1 Axial flow pumps are made up of the following primary parts:

The inlet of a pump is the part of the pump where fluid first makes contact with the rotating part, the impeller.
The fluid in a pump is pushed along its axis by a revolving component called the impeller. Lift is created by a set of blades or vanes attached to a central hub.
The diffuser is a fixed part of the pump that increases the flow path's cross-sectional area, turning the high-velocity flow produced by the impeller into high-pressure flow.
Pumps normally have an outlet placed downstream of the diffuser, where the fluid is released into the system.
Depending on the needs of the task at hand, axial flow pumps can be built to work at a variety of flow rates and pressures, and can have as few as one stage or as many as several. They can be powered by electric motors, diesel engines, or other sources, and often take up less room during installation than other types of pumps. In general, axial flow pumps are a reliable and effective choice for transferring large volumes of fluid in a variety of industrial settings.

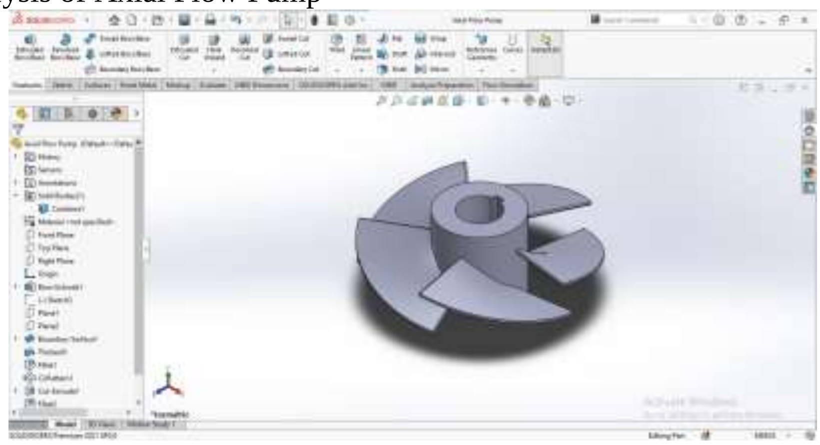
3.2 Need and Necessity

There are many compelling arguments in favour of analysing axial flow pumps.

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023
Improved efficiency and reduced risk of cavitation and other issues can be achieved through computational fluid dynamics (CFD) study of axial flow pumps, which can also be used to optimise the pump design.
CFD analysis can be used to foretell how an axial flow pump would function in response to varying inputs like flow rate, pressure, and fluid viscosity. Pump design optimisation, pump selection for a certain application, and pump troubleshooting can all benefit from this data.
Saving money: CFD analysis can assist cut expenses in areas like energy use, maintenance, and downtime by optimising pump design and anticipating pump performance. Particularly in large-scale industrial applications where pumps are utilised frequently, this has the potential to result in substantial savings over time.
CFD analysis can help make axial flow pumps safer by revealing problems including cavitation, flow instability, and excessive vibration. This data can be used to incorporate additional safeguards into the pump's construction or to alter the environment in which it operates.
CFD analysis can be used to assess the environmental effects of axial flow pumps, such as their ability to harm aquatic life or contaminate water supplies. With this data, we can create environmentally friendlier pump designs or alter current practises to lessen pumps' negative effects.
Axial flow pumps have a broad variety of industrial uses, so understanding how they work is crucial for maximising productivity while minimising negative effects on the environment and saving money.

3.3 Blade Details

Axial flow pumps' impeller blades determine performance. Impeller blade details include these:
Impeller flow depends on blade profile. Streamlined blades reduce turbulence and increase flow rate. Based on pump needs, NACA and circular arc blade profiles are used.
Blade angle: The angle between the impeller blade and its plane of rotation is the blade angle. Blade angle affects flow direction and lift. Pump efficiency is optimised by blade angle.
Blade thickness: Blade thickness affects strength and fluid forces. The thickness is optimised to reduce drag and turbulence while ensuring the blade can bear fluid forces.
Blade curvature determines flow direction and lift. Pump efficiency is optimised by curvature.
Blade skew: The angle between the blade and a plane perpendicular to the impeller axis. Blade skew improves impeller flow and reduces turbulence.
The pump's flow rate, pressure, and efficiency depend on the impeller's blade count. The number of blades is optimised to maximise pump performance and avoid flow interference.
5 Design and Analysis of Axial Flow Pump



Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023

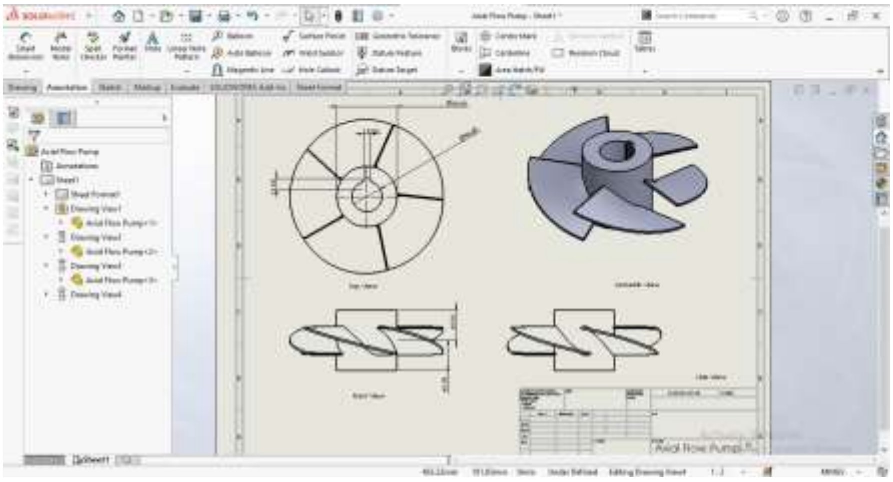


Fig.(5.1) Detailed Drawing

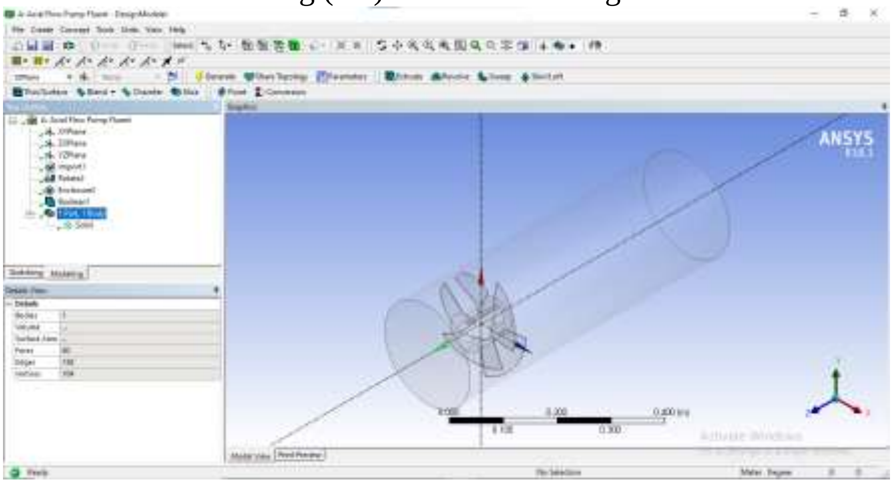


Fig.(5.2) Solid with Cylindrical Enclosure

Computational fluid dynamics (CFD) simulations have the ability to offer information on the flow field inside the cylindrical enclosure. This information can include velocity vectors, streamlines, and turbulence levels.

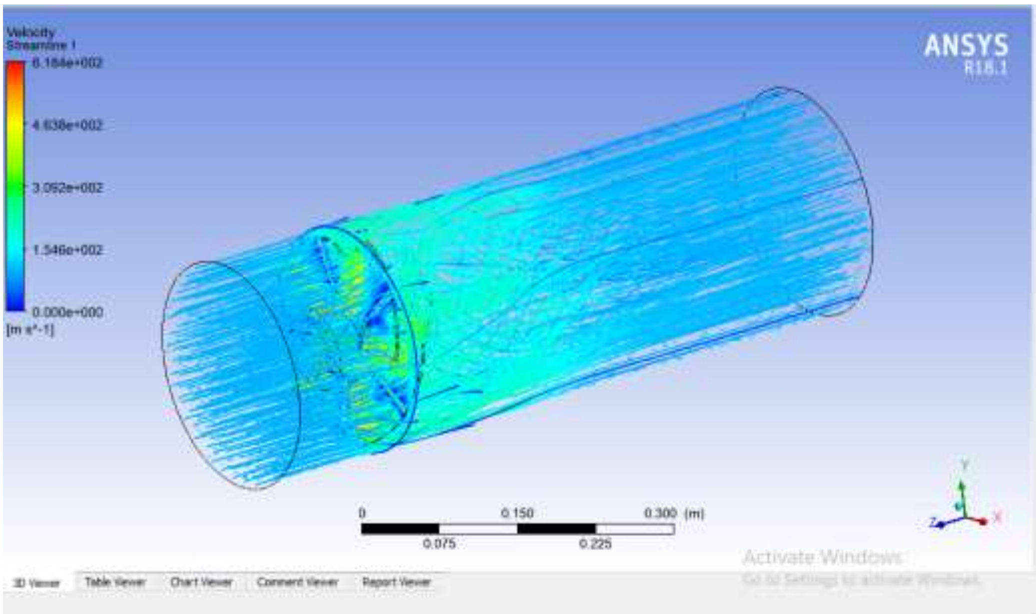


Fig.(5.3) Velocity Streamline

Information on the flow field within an axial flow pump impeller can be gained by computational fluid dynamics (CFD) simulations of the velocity streamlines of the impeller. The CFD model can

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023
show how fast the different parts of the impeller are moving. Streamlines, which show the direction and magnitude of the fluid velocity, are commonly used to depict the velocity distribution. High- and low-velocity regions within the impeller can be pinpointed with the aid of streamlines, allowing for more precise design.

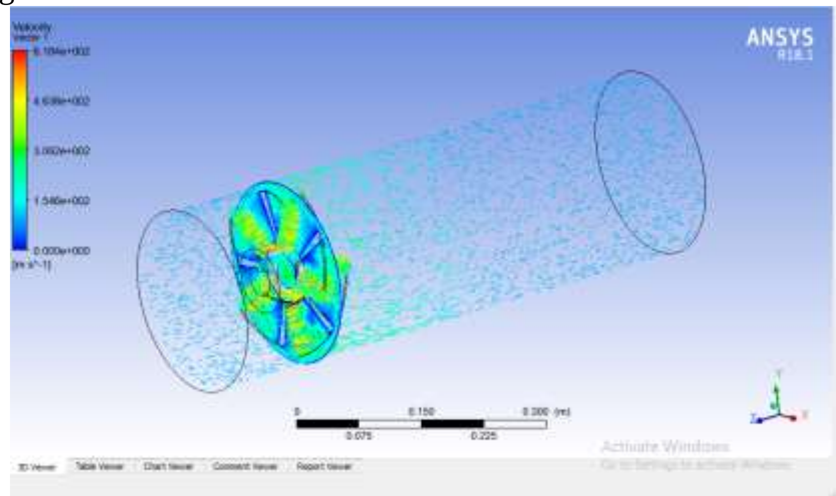


Fig.(5.4) Velocity Vector

velocity vectors provide a visual representation of the distribution inside the impeller. The magnitude and direction of the fluid velocity at a given location in the flow domain are represented by velocity vectors. The velocity vectors can be utilised to better understand where the impeller's velocities are highest and lowest, allowing for more efficient design. The velocity vectors can be used as a visual aid in observing the creation of vortices and turbulence within the impeller. The impeller design can be optimised to reduce turbulence and boost efficiency by studying the flow patterns.

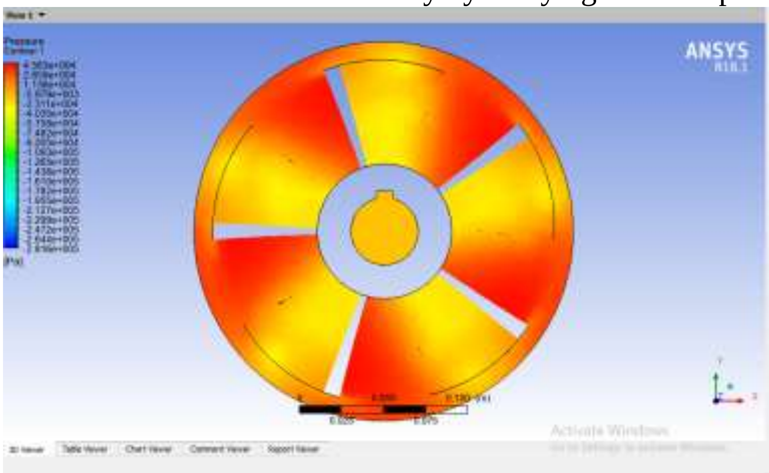


Fig.(5.5) Pressure Contour

The pressure distribution inside the impeller may be seen in three dimensions thanks to the CFD simulation, which can be represented graphically with pressure outlines. Constant pressure lines, or pressure contours, characterize the pressure field at a given location in the flow domain. High- and low-pressure regions within the impeller can be pinpointed with the aid of pressure contours, which can then be utilised to fine-tune the impeller's design and increase its efficiency.

4. Conclusion

In conclusion, computational fluid dynamics (CFD) flow analysis of axial flow pumps is an effective method for enhancing pump efficiency and productivity in commercial and institutional settings. The behaviour of the fluid flow through the pump can be simulated and studied with the use of a 3D model, mesh generation, and suitable boundary conditions.

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 6, No. 5, June : 2023

The pump's performance, efficiency, and environmental effect can all be enhanced by CFD analysis optimisation. The flow separation and recirculation, as well as the pressure recovery, can be decreased and enhanced, respectively, by adjusting the impeller blade geometry or the diffuser design. In conclusion, computational fluid dynamics (CFD) is an indispensable resource for developing and optimising the performance of axial flow pumps, which are widely used across a wide range of industrial applications. Researchers and engineers have been able to save money, work more efficiently, and have less of an impact on the environment by using CFD to build better pumps for a variety of industrial uses.

REFERENCES

[1] Liu Houlin., (2010) "Effects Of Blade Number On Characteristics Of Centrifugal Pumps" Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang Vol. 23,

[2] Anatoliy A. Yevtushenko., "Investigation Of Flow Inside an Axial-Flow Pump of GV – Impeller" Department of Applied Fluid Mechanics, Sumy State University, Ukraine.

[3] I. S. Jung, W. H. Jung, S. H. Baek, S. Kang "Shape Optimization of Impeller Blades for a Bidirectional Axial Flow Pump using Polynomial Surrogate Model" World Academy of Science, Engineering and Technology.

[4] S Kim , Y S Chois, K Y Lee and J H Kim "Interaction of impeller and guide vane in a series-designed axial-flow pump" 26th IAHR Symposium on Hydraulic Machinery and Systems

[5] V. P. Vasandhani, D. S. Hira.,(1975) "Influence Of Volute Tongue Length and Angle on the Pump Performance" IE Journal-ME Volume 56.

[6] Jaroslaw Mikielewicz, David Gardon Wilson, Tak-Chee Chan and Albert L. Goldfinch, (1978), " A Method for Correlating the Characteristics of Centrifugal Pumps in Two-Phase Flow" Journal of fluids Engineering, Vol 10.

[7] J.W Crisswell, ''Practical Problems Associated With Selection and Operation of Slurry Pumps'' Proc.Hydrotransport-8, Paper H1, BHRA Fluid Engineering, pp 317-338

[8] J. Remisz, Dr. eng.,(1983) "Slurry Pumps: Transformation Of Characteristics And Design" Eithth Conference of British Pump Manufacturers Association BHRA Fluid Engineering.

[9] Koji Kikuyama ,Murakami M,Asakuru E,Osuka T,Jinsheng, (1985) ''Velocity Distribution In The Impeller Passages of Centrifugal Pumps" Bulletin of JSME, Vol 28, Issue No. 243.

[10] Mez, W. "The Influence of Solid Concentration, Solid Density and Grain Size Distribution on the Working Behavior of Centrifugal Pumps" Proc.Hydrotransport-9, Paper H1, BHRA Fluid Engineering, pp. 345–358.

[11] Walker, C. I., and Goulas, A., "Performance Characteristics of Centrifugal Pumps When Handling NonNewtonian Homogeneous Slurries" Proc. Instn.Mech. Engrs., 198A, pp. 41–49.

[12] Sheth, K. K., Morrison, G. L., and Peng, W. W. "Slip Factors of Centrifugal Slurry Pumps" ASME J. Fluids Eng, pp. 313–318.

[13] M A Rayan, M Shawky, "Evaluation of Wear in a Centrifugal Slurry Pump" Proc Instn Mech Engrs Vol 203

[14] Dong, R., Chu, S., and Katz, J, "Quantitative Visualization of the Flow within the Volute of a Centrifugal Pump Part A: Technique" ASME J. Fluids Eng., 114 pp. 390–395

[15] Cader, T., Masbernat, O., and Roco, M.C, ''LDV Measurements in a Centrifugal Slurry Pump: Water and Dilute Slurry Flow,'' ASME J. Fluids Eng., 114, pp. 606–615.

[16] Gahlot, V. K., Seshadri, V., and Malhotra, R. C. "Effect of Density, Size Distribution, and Concentration of Solid on the Characteristics of Centrifugal Pumps" J. Fluids Eng., ASME Vol 114, 386–389.

[17] Cader, T., Masbernet, O., and Roco, M. C., "Two-Phase Velocity Distributions And Overall Performance of a Centrifugal Slurry Pump" ASME J.Fluids Engineering Conf., Washington, D, pp. 176–186.