INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 04, Issue 11, November 2024, pp : 192-196
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

ANALYZING THE CENTRIFUGAL PUMP IMPELLER'S PERFORMANCE ON A COOLING SYSTEM THROUGH COMPUTATIONAL FLUID DYNAMICS

Syed Nadeem Ahmed Alvi[1]

[1]Associate Professor Mechanical Engineering Department J. D. College of Engineering and Management, Nagpur
ng.alvi@rediffmail.com

ABSTRACT

The configuration of a centrifugal impeller is necessary to provide mechanically sound, easily constructed, and aerodynamically efficient blades. In addition to satisfying the first two criteria, this kind of blade design may also address stress issues with the right selection of factors. This approach is detailed in this article. The required surface speed distributions aid in the creation of the blade shape, which also includes straight-line components that link sites at the hub and shroud. It is also feasible to produce layouts with radially elemented blades and backward-swept blades using this method. This page gives a brief overview of the background, explains the idea, and shows an example of the design. The MATLAB programming may have a role in the future of pump development.

Keywords: Centrifugal, Pump, Impeller, Blades

1. INTRODUCTION

The centrifugal pump plays an essential role in the operation of cooling systems across a variety of industries, including automotive, aerospace, HVAC (heating, ventilation, and air conditioning), and power generation. These pumps are designed to move fluids efficiently, relying on rotational energy from the impeller to convert mechanical energy into hydraulic energy. The performance of a centrifugal pump is heavily influenced by the design of its impeller, which determines critical parameters such as flow rate, head, and efficiency. As cooling systems are fundamental in regulating temperatures and maintaining optimal operational conditions in machines and industrial processes, improving pump efficiency and performance is a high priority. One method for achieving this is through computational fluid dynamics (CFD), a powerful tool used to simulate, analyze, and optimize fluid flow within such systems.Centrifugal pumps are integral to cooling systems because they ensure the continuous circulation of coolant, which absorbs heat generated by machinery and processes, transferring it to heat exchangers where it is dissipated. This function is vital for maintaining safe operating temperatures, preventing overheating, and ensuring efficient energy use. The performance of a cooling system is therefore dependent on the pump's ability to maintain consistent flow under varying thermal and hydraulic loads. For example, in automotive systems, centrifugal pumps drive coolant through the engine block and radiator, preventing thermal damage and ensuring efficient fuel combustion.In large-scale industrial applications, such as in power plants and refrigeration systems, centrifugal pumps regulate fluid movement to maintain the appropriate thermal balance. If the pump impeller’s performance is suboptimal, it can lead to insufficient coolant flow, increased energy consumption, and potential equipment failure due to overheating. As a result, enhancing the efficiency and reliability of centrifugal pumps is crucial for improving the overall performance and sustainability of cooling systems.

Overview of Centrifugal Pump Impeller Design

The impeller is the most critical component of a centrifugal pump, as it is responsible for imparting velocity to the fluid and converting kinetic energy into pressure energy. The impeller typically consists of a series of blades or vanes, which are arranged radially or semi-radially around a central hub. When the impeller rotates, fluid is drawn into the pump near the axis and expelled radially outward into the pump casing at high velocity, where it is directed through the system.The performance of the impeller is governed by several factors, including its geometry (e.g., blade number, angle, and curvature), size, and the rotational speed of the pump. Variations in impeller design can significantly impact the flow characteristics, including pressure distribution, head generated, and efficiency. For example, increasing the number of blades can improve the head and flow rate, but it may also increase the energy consumption and potential for cavitation, a damaging phenomenon where vapor bubbles form and collapse within the fluid. Thus, finding the optimal balance between these factors is critical for achieving peak performance in a centrifugal pump.In traditional design approaches, empirical methods and prototype testing have been used to improve impeller performance. However, these methods are time-consuming, costly, and often lack precision in predicting real-world behavior under different operational conditions. As a result, engineers and researchers increasingly rely on advanced computational tools, such as CFD, to study and optimize pump impeller designs more efficiently.

Introduction to Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems involving fluid flow, heat transfer, and associated phenomena. By simulating the behavior of fluids within complex systems, CFD provides detailed insight into flow patterns, pressure distribution, turbulence, and other critical aspects of fluid dynamics that are challenging to measure experimentally.In the context of centrifugal pumps, CFD can be used to simulate the flow of coolant through the pump impeller, allowing engineers to visualize and analyze key performance metrics, such as velocity fields, pressure gradients, and energy losses. Unlike physical testing, which often provides limited data at discrete points, CFD enables a comprehensive analysis of the entire flow domain within the pump. This capability allows for more accurate predictions of pump performance under varying operating conditions, facilitating the design of more efficient and reliable systems.By leveraging CFD, engineers can also investigate the impact of design changes on impeller performance without the need for expensive and time-consuming prototyping. For instance, altering the blade shape or spacing can be simulated in a virtual environment, with the resulting effects on flow rate, head, and efficiency evaluated in detail. This enables a more iterative and refined design process, where multiple configurations can be tested and optimized in silico before committing to a final design.

CFD Simulation of Centrifugal Pump Impellers

CFD simulations of centrifugal pump impellers typically involve several key steps. First, a detailed geometric model of the impeller is created using computer-aided design (CAD) software. The geometry is then imported into a CFD software platform, where a computational mesh is generated to discretize the flow domain into small, finite elements. The accuracy of the simulation depends on the quality and resolution of the mesh, as a finer mesh provides better resolution of the flow features but requires more computational resources.Once the mesh is generated, boundary conditions, such as inlet velocity, outlet pressure, and rotational speed, are defined to mimic real-world operating conditions. The CFD solver then uses numerical methods, such as the finite volume method, to solve the governing equations of fluid dynamics—namely, the Navier-Stokes equations, which describe the conservation of mass, momentum, and energy in the fluid.As the simulation progresses, the CFD software calculates the flow velocity, pressure, and turbulence at each point in the flow domain. The results are visualized as contour plots, vector fields, and streamline diagrams, providing a clear picture of how the fluid moves through the impeller and interacts with the pump casing. This data is then used to assess the pump's performance and identify areas where improvements can be made, such as reducing pressure losses, minimizing cavitation risk, or enhancing flow uniformity.CFD simulations can also provide insights into phenomena that are difficult to observe experimentally, such as the development of secondary flows, recirculation zones, or separation regions near the impeller blades. These insights are invaluable for optimizing impeller designs to improve performance and reduce energy consumption.

Applications of CFD in Impeller Performance Optimization

CFD has become a critical tool for optimizing the performance of centrifugal pump impellers, particularly in applications where efficiency, reliability, and precision are paramount. Some key areas where CFD is applied in impeller performance analysis and optimization include:

Hydraulic Efficiency Optimization: By analyzing the flow patterns within the impeller, CFD simulations help engineers identify areas where hydraulic losses occur due to factors such as turbulence, flow separation, or inefficient blade geometry. This enables targeted design modifications to improve the pump's overall hydraulic efficiency.

Cavitation Prediction and Mitigation: Cavitation, a common issue in centrifugal pumps, occurs when local fluid pressure drops below the vapor pressure, causing vapor bubbles to form and collapse. CFD simulations allow for the prediction of cavitation-prone regions within the pump and the evaluation of different impeller designs or operating conditions to reduce the risk of cavitation damage.

Flow Uniformity and Pressure Distribution: Achieving uniform flow and balanced pressure distribution across the impeller is critical for maintaining pump efficiency and minimizing wear. CFD provides detailed visualizations of the flow field, allowing engineers to optimize the blade angles and curvature to ensure uniform flow and balanced pressure gradients.

Thermal Management: In cooling systems, the thermal performance of the pump is just as important as its hydraulic performance. CFD can simulate both fluid flow and heat transfer, enabling engineers to optimize the impeller design for effective heat dissipation and temperature regulation within the cooling system

CONCEPT

For the purpose of carrying out a finite element analysis, the version that we are using need to be segmented into a number of smaller components that are referred to as finite factors. It is possible to label finite element analysis (FEA) as a discretization technique due to the fact that the version is divided into certain discrete components. When it comes to carrying out a finite detail assessment, it is necessary to have a mathematical net or "mesh" in order to do so. In the event that the device under investigation is of a one-dimensional form, we still have the option of using line factors in order to represent our geometry and carry out our study. It is necessary to have a two-dimensional mesh if the difficulty can be specified in terms of dimensions. As a result, we make use of a three-dimensional mesh if the hassle is intricate and a three-dimensional illustration of the continuum is necessary. There are two possible shapes for area factors: triangular and quadrilateral. The intricacy of the geometry and the nature of the problem that is being modeled are the primary considerations that are taken into account when making the decision about the detail form and order. Elements that make up a membrane do not have any thickness. As a consequence of this, they do not possess any bending stiffness; loads may only be carried in the plane of the element. For the purpose of modeling thin walled areas in three-dimensional space, plate and shell factors are used. Because the plate precept is based on the assumption that the load is conveyed by bending, the plate detail is constructed around this precept. Modeling shells, in which there may be an accumulation of flexure and membrane motion, is accomplished by the use of shell factors. Plate elements are regarded important in situations when the out-of-plane distortion is somewhat larger than the thickness of the plate. In addition, there are parts that are one of a kind, which serve to ease the accurate modeling of thick plates. In the event that the deflection is greater than the thickness of the plate, membrane movement must be taken into consideration, and shell factors must be used hence. There are five degrees of freedom available for shell element nodes; however, the in-aircraft rotational flexibility, which is also frequently referred to as the drilling freedom, is what is lacking. There are several sorts of solid components that may be found. In order to provide an explanation for the pass-phase of an axially symmetric factor, axis symmetric components are used. Stress caused by flying. In order to provide an explanation for the segment of lengthy objects (which may contain a shaft or wall cross-phase), factors are used. It is assumed that the strain inside the out-of-plane path is zero, which is consistent with the concept that the strain is contained within a single In order to provide an explanation for the sections of thin items (such as a wrench), plane pressure components are used. Therefore, the strain in the out-of-plane direction is assumed to be zero, which is consistent with the notion that the strain is contained inside a single aircraft

MATERIAL PROPERTIES OF THE PUMP:

The analysis is performed on (i) MS pump Impeller (ii) SS pump Impeller

Material properties of MS pump:

1. Young’s modulus E= 210 GPa
2. Poisson’s ratio NUXY=0.303
3. Mass density =7960 kg/m3
4. Damping co-efficient =0.008

Material properties of SS pump:

1. Yield stress 0.2 % proof minimum- 170
2. Elastic modulus- 193 GPa
3. Mass density-8000 kg/m3
4. Hardness B (HRB) max- 217
5. Elongation (%)- 40 minimum

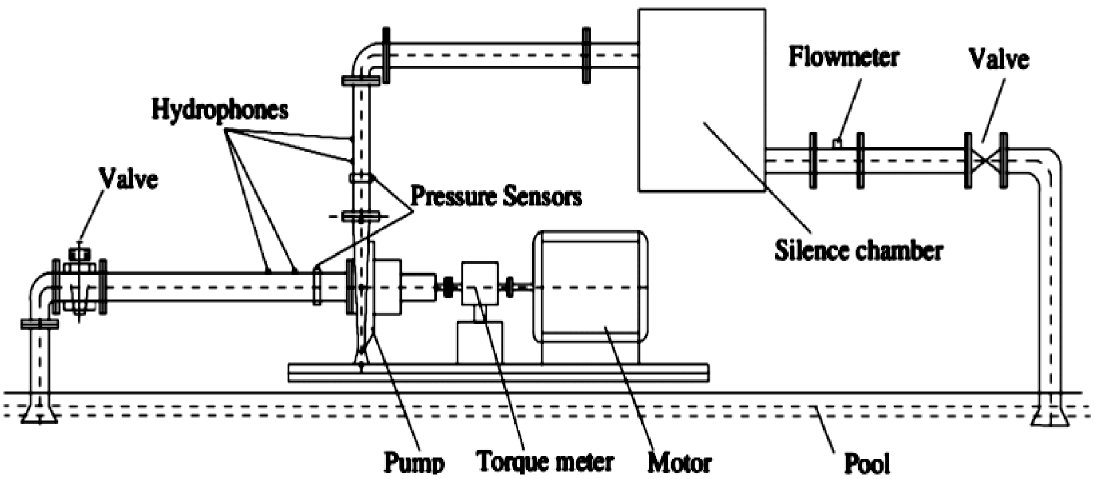

Fig 1 Pump system Centrifugal pump

There are a few other names for centrifugal pumps, including roto-dynamic pumps and dynamic pressure pumps. The concept of centrifugal pressure is the basis for its operation. Through the use of a spinning impeller that is composed of certain vanes that are bent in the opposite direction, this kind of pump causes the liquid to be subjected to a whirling motion. The liquid is released into the casing that surrounds the outside edge of the impeller after entering the impeller at its eye, which is located either in the centre of the impeller.

The upward push that occurs inside the pressure head at any given factor or output of the impeller is proportional to the rectangle of the tangential speed of the liquid at that point. As a result, the rise occurs at the hole of the impeller, which is the location where the radius is greater. There is a possibility that the strain head will be higher, and using a high pressure head, the liquid will be released through the hole. It is possible for the liquid to be raised to a higher level as a result of this very high stress head. Impellers are the most common kind of fluid flow equipment, and they are responsible for converting the electricity generated by the equipment into the fluid strain and kinetic electricity. This type of equipment has been widely employed in industry. The centrifugal pump, which is the most popular kind of pump, has been used in a variety of industrial settings, including water and sewage systems, drainage systems, and chemical manufacturing facilities. As a consequence of this, a great deal of study has been completed for the purpose of designing various types of centrifugal pumps. An optimisation strategy that makes use of mechanical concepts has recently been investigated in response to the needs of the business. This strategy will result in pumps that have improved heads and greater levels of performance. Because the impeller is responsible for the generation of electricity via the flow of fluid through the pump, it is the component of the pump that has the most influence on the overall performance of the device.

The definition of specific pace is "the rate of a really perfect pump geo-metrically just like the real pump, which when walking at this speed will raise a unit of volume, in a unit of time through a unit of head." Specific pace is denoted by the phrase "the rate of a really perfect pump."

The performance of a centrifugal pump may be stated in terms of the pressure at which the pump is operating, the total head, and the amount of flow that is needed. This information may be obtained from the curves that have been released by the pump manufacturer. In order to determine the specific speed, the accompanying formulae angeles are used, and data derived from these curves are utilised at the pump's prime performance component.

The formula for calculating the specific pace (Ns) is as follows: (NxQ 1/2)/H 3/4 N = The velocity of the pump measured in revolutions per minute (rpm).

In both unmarried and double suction impellers, the flow charge is denoted by the letter Q and is measured in litres per minute.

"H" equals the total dynamic head measured in metres.

Radical go with the flow pumps, mixed drift pumps, and axial drift pumps are the three types of pumps that are usually classified. When you look at the chart that is above, you will notice that there is a slow transition from the radial float impeller, which generates pressure primarily through the movement of centrifugal pressure, to the axial go with the flow impeller, which generates the majority of its head with the assistance of the propelling or lifting motion of the vanes at the liquid. When it comes to the particular speed range of around one thousand to six thousand, double suction impellers are used just as often as single suction impellers.

It is possible that the numerical cost of Ns will vary if you substitute other devices with flow and head mechanisms. Revolutions per minute (rpm) is the unit of measurement that is always used to express the speed. If you are using alternative devices for ability and head, the following is a method that you may use to change the Specific Speed range (Ns):

It is a metric................Q is equal to metres per hour, while H is equal to metres.

• For the sake of illustration, we shall do a computation of Ns using both metric and United States units:

• Q equals 120 litres per second. • H equals 100 metres

• Speed equals 1500 revolutions per minute

2. CONCLUSION

The performance of a centrifugal pump impeller is critical to the efficient operation of cooling systems across numerous industries. Given the complexities of fluid dynamics within the pump, traditional design and testing methods are often insufficient for fully optimizing performance. Computational Fluid Dynamics (CFD) offers a powerful alternative, providing detailed insights into fluid behavior that can be used to improve impeller design, enhance pump efficiency, and reduce energy consumption. Through advanced CFD simulations, engineers can explore a wide range of design variations and operating conditions, ensuring that centrifugal pumps are optimized for the specific demands of cooling systems, thereby improving their reliability and effectiveness in managing thermal loads.

3. REFERENCES

[1] T.C. Mohana Klumar and Y.V.N. Rao, Quasi two-dimensional analysis of flow through a centrifugal pump impeller. ASME J. Fluid Eng. 99 (1977) 687–692.

[2] T. Ogawa and S. Murata, On the flow in the centrifugal impeller with arbitrary aerofoil blades (II report: The effect of change of impeller width). Bull JSME. 17 (1974) 723–730.

[3] C.H. Wu, A general theory of three-dimensional flow in a subsonic and supersonic turbo machines of axial, radial and mixed flow types. Trans ASME. 74 (1952) 1363–1380

[4] D. Japikse, Review — progress in numerical turbomachinery analysis. ASME J. Fluid Eng.98 (1976) 592–606.

[5] D. Adler, Status of centrifugal impeller internal aerodynamics (Part 1. Inviscid flow prediction methods). ASME J. Eng power. 102 (1980) 728–737.

[6] Ch. Hirsch and G. Warzee, An integrated quasi-3D finite element calculation programme for turbomachinery flows. ASME J. Eng power. 101 (1979) 141–148.

[7] Y. Krimerman and D. Adler, The complete three-dimensional calculation of the compressible flow field in turbo impellers. J. Mech. Eng. Sci. 20 (1978) 149–158.

[8] A.J. Acosta and R.D. Bowerman, An experimental study of centrifugal pump impellers. Trans. ASME. 79 (1957) 1821–1839.

[9] T.C. Mohanakumar and Y.V.N. Rao, Experimental determination of pressure distributions along the blade surfaces of a centrifugal pump impeller. Proc. XII National Conference on FMFP, Indian Institute of Technology, Delhi (1983) pp. 11–17.

[10] B. Opelt and A.H. Khalifa, Measuring the pressure distribution along the surfaces of the blades in a Francis turbine runner. Proc. Inst. Mech. Eng. 189 (1975) 213–220.

[11] B. Maiti, Analysis of flow through pump impellers by finite element method. Ph.D. thesis, Indian Institute of Technology, New Delhi (1984).

[12] S.K. Ayyubi and Y.V.N. Rao, Theoretical analysis of flow through centrifugal pump impeller by method of singularities. ASME J. Basic Eng. 93 (1971) 35–41

[13] K.H. Huebner, The finite element method for engineers (1st edn). John Wiley & Sons, New York (1975).