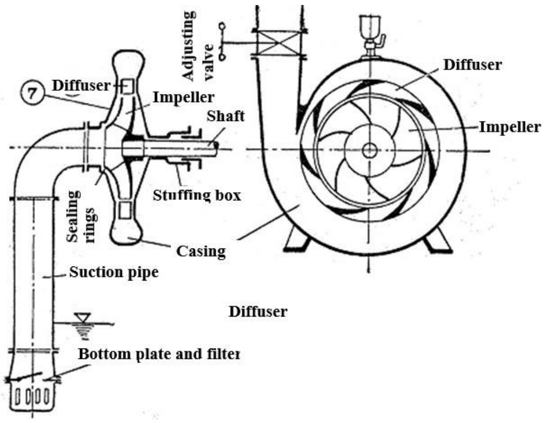
ISSN: 2278-0181
ETEDM - 2021 Conference Proceedings

Design and CFD Analysis of Centrifugal Pump

Dinesh.N [1] Nakul C.J [2], Vishnu.R [2], Vishnu Mohan[2] , Abdul Haleem.A

[21-]Assistant Professor, [2-]UG Scholar
Department of Mechanical Engineering, Hindustan Institute of Technology,
Coimbatore.

***Abstract*:- Centrifugal pumps are a most commonly used in different fields like industries, agriculture and domestic applications. Computational Fluid Dynamics is most commonly used tool for simulation and analysis. 3-D numerical CFD tool is used for simulation of the flow field characteristics inside the turbo machinery. CFD simulation makes it possible to visualize the flow condition inside centrifugal pump. The present paper describes the head, power, efficiency and to evaluate the pump performance using the ANSYS FLUENT, a computational fluid dynamics simulation tool. These simulations of centrifugal pumps are strongly related to flow rate and pressure drop, which may occur in either the rotating runnerimpeller or the stationary parts of the centrifugal pumps. The numerical simulation can be used to detect the performance of centrifugal pump and to get safe range of operating at different flow rate and rotating speeds.**



Top view and front view of centrifugal pump

INTRODUCTION

Centrifugal pump is a machine that imparts energy to a fluid. This energy can cause a liquid to flow or rise to a higher level. Centrifugal pump is an extremely simple machine which consists of two basic parts: The rotary element or impeller and the stationary element or casing. The centrifugal pumps are widely used in the world because the pump is robust, effective and inexpensive to produce. Centrifugal pumps are more economical to own, operate and maintain than other types of pumps. Pumps operate via many energy sources, including manual operation, electricity, engines, or wind power, come in many sizes, from microscopic for use in medical applications to large industrial pumps. Mechanical pumps serve in a wide range of applications such as pumping water from wells, aquarium filtering, pond filtering and aeration, in the car industry for water-cooling and fuel Injection etc. A centrifugal pump essentially consists of an impeller rotating in a casing called volute. Fluid enters the eye (center) of the impeller and exits through the space between the impeller blades to the space between the impeller and casing walls. The velocity of fluid elements is in both tangential and radial directions, as the impeller rotates. The velocity and pressure both increase as the fluid flows through the impeller. Since the rotational mechanical energy is transferred to the fluid, at the discharge side of the impeller, both the pressure and kinetic energy of the water will rise. At the suction side, water is getting displaced, so a negative pressure will be induced at the eye. This low pressure helps to suck a freshwater stream into the system again, and this process continues. The impeller is the most vital part of a centrifugal pump design. Successful impellers have been developed with many years of analysis and developmental work. The vanes (blades) of the best impellers curve backward by design. These backward-curved vanes have the blade angle less than 90 degrees and are the most preferred vane type in the industry due to its self-stabilizing power consumption characteristics. This means that with an increase in flow rate, the power consumption of the pump stabilizes after a limit. In this work, we have analysed a dewatering centrifugal pump by using a technique called CFDComputational Fluid Dynamics. This approach can be used as design validation tool before going for prototyping and physical testing. We can use CFD as tool to make any improvements that can be made in CAD model itself. This will save lot of time and expenses for testing. Here, we have presented the CFD methodology used for the analysis in the chapter 4 and the results are presented and discussed in Chapter 5. The conclusion is made in the Chapter6.

LITERATURE SURVEY

Centrifugal pump is a most common pump used in industries, agriculture and domestic applications. Its impeller design demands a detailed understanding of the internal flow at rated and part load operating conditions. For the cost-effective design of pumps it is very crucial to predict their performance in advance before manufacturing them, which requires understanding of the flow behaviour in different parts of the pump. Experimental model testing is one of the solutions for prediction of performance but it is tedious, time consuming and costly. Conversely, theoretical approach merely gives a value; but it is unable to determine the root cause for the poor performance. In the recent years, CFD started to play a key role for the prediction of the flow through pumps and turbines having successfully contributed to the enhancement of their design. The application of CFD in the design of water pumps and turbines started about 30 years ago. The first steps coincided with the introduction of the finite element method into CFD and were characterized by simplified Quasi-3D Euler solutions and fully 3D potential flow solutions. Over the years the complexity continuously increased in stages: via 3D Euler solutions, to steady Reynolds Averaged Navier- Stokes (RANS) simulations of single blade passages using finite volume methods, extending to steady simulations of whole machines, until today unsteady RANS equations are solved with advanced turbulence models. The most active areas of research and development are now concerned with including the effects of 2-phase flow and fluid —structure interaction. A growing availability of computer power and a progress in accuracy of numerical methods, brought turbo machinery CFD methods from pure research work into the competitive industrial markets. Many softwares are available in the market for numerical analysis of turbo-machines viz. Fluent (UK and US), CFX (UK and Canada), Fidap (US), Polyflow (Belgium), Phoenix (UK), Star CD (UK), Flow 3d (US), ESI/CFDRC (US), SCRYU (Japan) and more recently, Kaupert et al, Potts and Newton and Sun and Tsukamoto studied pump off-design performance using the commercial software CFX-TASCflow, FLUENT and STARCD respectively. Normally, the CFD codes provide three calculation methods for the analysis of turbo machinery flows: the Multiple Reference Frame (MRF), the Mixing Plane and the Sliding Mesh. The first two methods are basically steady flow methods. In MRF method, the rotor is kept at a fixed position and the governing equations for rotor are solved in a rotating reference frame, including Coriolis and centrifugal forces, while for the stator are solved in an absolute reference frame. CFD has been widely used in low specific speed and medium specific speed mixed flow pumps by Takamara and Goto and Goto ; and in radial flow pump by Sedlar and Mensik. Pump designers are continually being challenged to provide machines that operate more efficiently, quietly and reliably at lower cost. Many investigators have applied CFD as a numerical simulation tool to carry out different investigations on centrifugal pumps. This section describes the research work carried out by the researchers in the centrifugal pumps by CFD approach.

Performance prediction at different operating conditions

Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. Few efforts were made to study the offdesign performance of pumps, where the performance of pump deteriorates. With the aid of the CFD approach, the complex internal flows through the different components of pump can be studied at different operating conditions which help in improvement in the performance at off-design conditions. Mentzoset a1. carried out a numerical simulation of the internal flow in a backward curve vaned centrifugal pump. The MRF approach used to take into account the impeller-volute interaction was completely failed, due to its fixed coupling formulation. However, its use was recommended for basic understanding of the flow at various operating points. The transient analysis was suggested as a real tool for understanding of the interaction between impeller and spiral casing. Three-dimensional computational model of centrifugal pump is shown in Fig. 1(a). Mentzos simulated the flow through the impeller of centrifugal pump using finite-volume method along with a structured grid system for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. It was reported that, although the grid size was not adequate to investigate the local boundary layer variables, global ones were well captured. The proposed approach was advocated for the basic understanding of the flow at various operating points.

COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is the use of computers and numerical methods to solve Problems involving fluid flow. CFD has been successfully applied in many areas of fluid mechanics. These include aerodynamics of cars and aircraft, hydrodynamics of ships, flow through pumps and turbines, combustion and heat transfer chemical engineering. Applications in civil engineering include wind loading, vibration of structures, wind and wave energy, ventilation, fire, explosion hazards, dispersion of pollution, wave loading on coastal and offshore structures, hydraulic structures such as weirs and spillways, sediment transport. More specialist CFD applications include ocean currents, weather forecasting, plasma physics, blood flow and heat transfer around electronic circuitry.

1. Basic Principles of CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called discretisation. The fundamental elements of any CFD simulation are: 1. The flow field is discredited; i.e. field variables (, u, v, w, p) are approximated by their values at a finite number of nodes. 2. The equations of motion are discretised (approximated in terms of values at nodes): Control-volume or differential equations (Continuous) algebraic equations (Discrete) The main stages in a CFD simulation are: 2. Pre- processing: Formulation of the problem (governing equations and boundary conditions); Construction of a computational mesh (set of control volumes). 3. Solving: Discretisation of the governing equations; Solution of the resulting algebraic equations. 4. Post-processing: Visualization (graphs and plots of the solution); Analysis of results (calculation of derived quantities: forces, flow rates …) 5. Forms of the Governing Fluid-Flow Equations The equations governing fluid motion are based on fundamental physical principles: Mass: change of mass = 0 Momentum: change of momentum = force Ã— time Energy: change of energy = work + heat In fluid flow these are usually expressed as rate equations; i.e. rate of change = When applied to a fluid continuum these conservation principles may be expressed Mathematically as either: Integral (i.e. control-volume) equations; 6. Differential equations. Integral (ControlVolume) Approach This considers how the total amount of some physical quantity (mass, momentum, energy,) is changed within a specified region of space (control volume). For an arbitrary control volume the balance of a physical quantity over an interval of time is

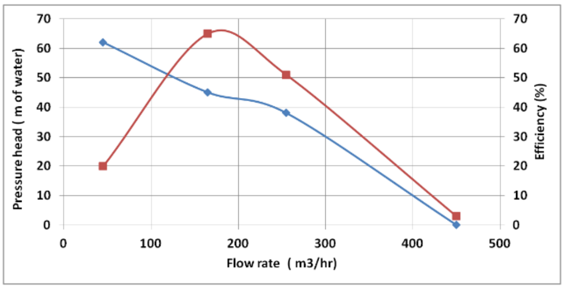
Change = amount in – amount out + amount created In fluid mechanics this is usually expressed in rate form by dividing by the time interval (and transferring the net amount passing through the boundary to the LHS of the equation): The flux, or rate of transport across a surface, is composed of: Advection movement with the fluid flow; Diffusion net transport by random molecular or turbulent motion. The important point is that this is a single, generic equation, irrespective of whether the physical quantity concerned is mass, momentum, chemical content, etc. Thus, instead of dealing with lots of different equations we can consider the numerical solution of a generic Scalar-transport equation Differential Equations In regions without shocks, interfaces or other discontinuities, the fluid- flow equations can also be written in equivalent differential forms. These describe what is going on at a point rather than over a whole control volume. Mathematically, they can be derived by making the control volumes infinitesimally small. The Main Discretization Methods 1. Finite-Difference Method Discretize the governing differential equations; e.g. 1. Finite-Volume Method Discretize the governing integral or control-volume equations;

e.g. 2. Finite-Element Method Express the solution as a weighted sum of shape functions S (x); e.g. for velocity: Substitute into some form of the governing equations and solve for the coefficients,

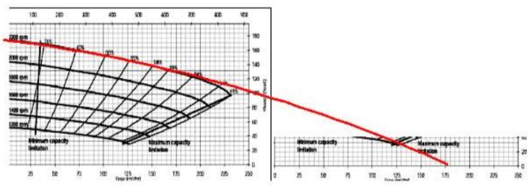
RESULTS AND DISCUSSION

Pump performance curve

The pump performance curve is obtained from the analysis for different flow rates and pressure as shown in the Fig 3. The duty point achieved is 4.3 bar with a flow rate of 165 m3/hr. The maximum efficiency attained is 65% corresponding to the duty point. This is in good agreement with the design expectations as shown in the Figure.



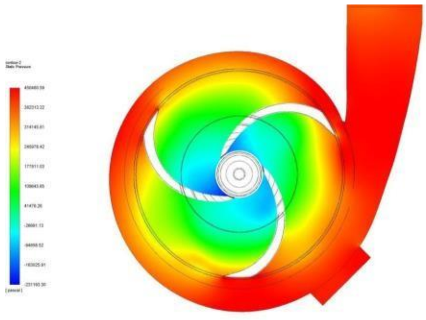
Pump performance curve obtained fromsimulation



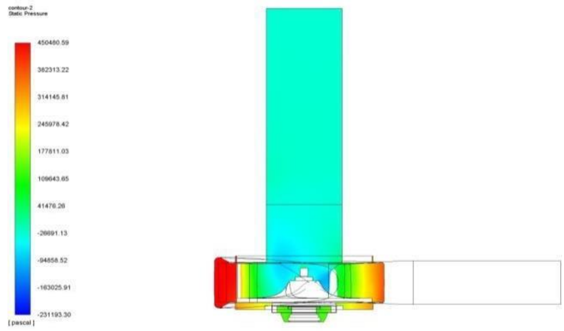
Pump performance curve obtained from designcalculation

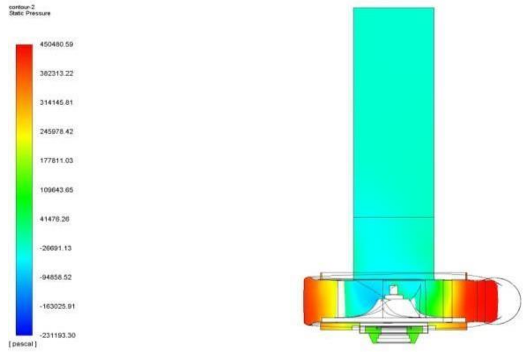
Pressure at Duty point

The pressure achieved at duty point is 4.3 bar at a flow rate of 165 m3/hr. This can be observed in the Fig 5. The pressure distribution at inlet section is shown in the Fig. 6 and near to impeller region is shown in the Fig.7. The pressure development is gradual along the periphery of casing and reached the design pressure.

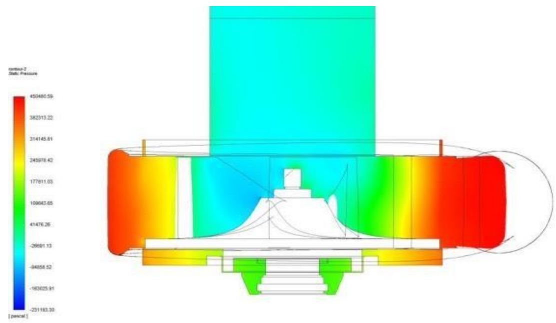


Pressure head developed at duty point





Pressure distribution at the inlet at duty pointcondition

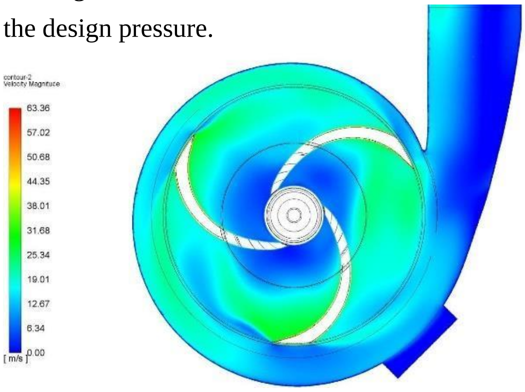


Pressure distribution near to impeller at dutypoint condition

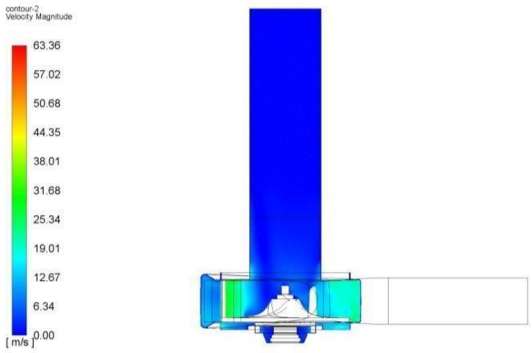
Flow Velocity at Duty point

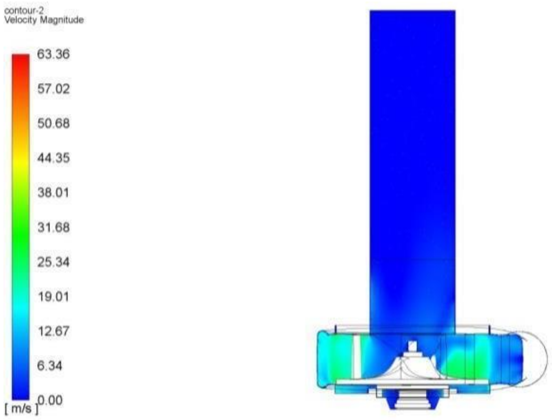
The velocity achieved at duty point is 14 m/sec ata flow rate of 165 m3 /hr. This can be observed in the Fig.8. The velocity distribution at inlet section is shown in the Fig. 9 and near to impeller region isshown in the Fig.10. The pressure development is gradual along the periphery of casing and reached

the design pressure.

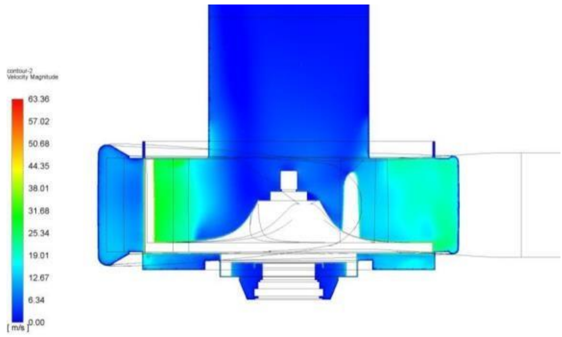


Velocity head developed at duty point





Velocity distribution at the inlet duty pointcondition



Velocity distribution near to impeller at dutypoint condition

CONCLUSION

The design and analysis of the centrifugal pump is done and through analysis we conclude that the deflection on the vane of an impeller is very less and within the design limits and hence the design of the impeller is safe. Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD).It is found that the design and analysis methods lead to completely very good flow field predictions. This makes the methods useful for general performance prediction. In this way, the design can be optimized to give reduced energy consumption n, lower head loss, prolonged component life and better flexibility of the system, before the prototype is even built. The efficiency of the centrifugal pump can be improved by increasing the value of surface finish factor. The impeller is the important component of the centrifugal pump which plays a crucial role in determining the efficiency of the centrifugal pump. The fatigue life of shaft is calculated which is infinite. The centrifugal pump operates at high temperature and high pressure, so selection of mechanical seals becomes of utmost importance in this situation. Hence, the design and performance analysis of the pump components is done and efficiency is found out which is satisfactory.

REFERENCES

[1] S.Rajendran and Dr.K.Purushothaman,―Analysis of a centrifugal pump impeller using ANSYS-CFX,‖ International Journal of EngineeringResearch & Technology, Vol. 1, Issue 3, 2012.

[2] S R Shah, S V Jain and V J Lakhera, ―CFD based flow analysis of centrifugal pump,‖ Proceedings of the 37th National & 4th InternationalConference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[3] P.UshaShriansC.Syamsundar, ―computational analysis on performance of a centrifugal pump impeller,‖ Proceedings of the 37th National& 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[4] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, ―Parametric Study of a Centrifugal Pump Impeller by Varying the OutletBlade Angle,‖ The Open Mechanical Engineering Journal, no 2, 75-83, 2008.

[5] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova, ―Cavitation modelling of a centrifugal pump impeller ,‖ 22nd International Congress of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil, 2013.

[6] Mohammed Khudhair Abbas, ―cavitation in centrifugal pumps,‖ Diyala Journal of Engineering Sciences, pp. 170-180, 2010.[7]

AbdulkadirAman, SileshiKore and Edessa Dribssa, ―Flow simulation and performance prediction of centrifugal pumps using cfd -tool,‖ Journal of EEA, Vol. 28, 2011.

[7] Erik Dick, Jan Vierendeels, Sven Serbruyns and John VandeVoorde, ―Performance prediction of centrifugal pumps with cfd -tools,‖ Task Quarterly 5, no 4, 579–594, 2001.

[8] S. C. Chaudhari, C. O. Yadav and A. B. Damo, ―A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump,‖ International Journal of Research in Engineering & Technology, Vol. 1, Issue 3, 57-64, 2013.

[9] D. Somashekar and Dr. H. R. Purushothama, ―Numerical Simulation of Cavitation Inception on Radial Flow Pump,‖ IOSR Journal of Mechanical and Civil Engineering, Vol. 1, Issue 5, pp. 21-26, 2012.

[10] Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao and Wang Kai, ―Effects of Blade Number on Characteristics of Centrifugal Pumps,‖ Chinese journal of mechanical engineering, Vol. 23, 2010.

[11] Myung Jin Kim, Hyun Bae Jin, and Wui Jun Chung, ―A Study on Prediction of Cavitation for Centrifugal Pump,‖ World Academy of Science, Engineering and Technology, Vol. 6, 2012**.**