International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 2349-3771 Volume 2 Issue 9, pg: 372-376

DESIGN AND CFD ANALYSIS FOR IMPELLER OF A CENTRIFUGAL PUMP

Namdev T. Patil[1], Narayan S. Dharashivkar[2], Swapnil S. Kulkarni[3]
[1]Student, [2]Professor,
Department of Mechanical Engineering, Tatyasaheb Kore Institute of Engineering & Technology, Warananagar. Maharashtra, India
[3]Director-Able Technologies India Pvt. Ltd., Pune, India

ABSTRACT

It is a well-established fact that a number of problems faced by pumps in a pumping station are related to structural instability or failure of impeller/blades. Vibration, Cavitations, rough running, lower than expected efficiency and reduced pump life can generally, be traced to undesirable flow conditions on the process side. Although some design guidelines for configuration of pumps are available, but depending on the fluctuating discharge effects on its structural stability has not been studied so far. The major reason for this being site specific geometrical and hydraulic constraints. The most common solution to solving potential problems in new designs and rectifying problems observed in existing installations is to construct a scaled model in the laboratory, observe the flow (by dye injection) and propose modifications to intake geometry. The limitation of cost and time, inhibit trying out of many variants before reaching an optimized design for an intake with rapid progress in Computational Fluid Dynamics (CFD), numerical simulation is being regarded as an effective tool in solving fluid problems in simulating the flow and pressure distribution inside the pump. It reduces the cost as well as time associated with finalizing the design of a pumping station, by reducing the number of options to be tested in a laboratory.

KEYWORDS: *Finite volume method, Centrifugal Pump Design , Impeller Vane angle , ICEM CFD Tool – Ansys Fluent ,HyperMesh*

I. INTRODUCTION

These studies pertain to development of CFD code for simple centrifugal pump. These codes cannot be used as general software for CFD analysis of any intake structure.

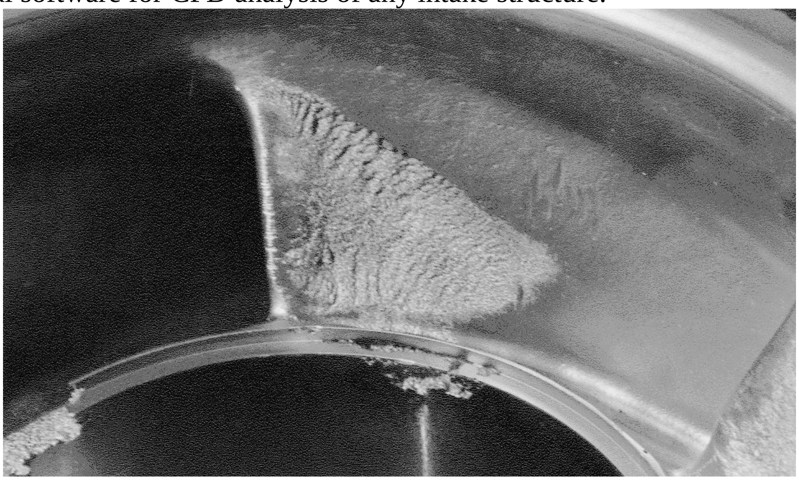

Fig. 1: Damage patterns on impeller

Fig -1 shows, Impellers of this pump suffered from premature wear due to cavitation attack. Within 10000 hrs of operation severe erosion has occurred at the hub and suction side of the impeller vanes just after the leading edges.
Numerical Analysis of Flow inside the centrifugal pump can be done and simulation can be analyzed using number of variables. In this study it has been restricted to the comparison of results based on the pattern of streamlines of flow, velocity vectors, velocity profiles and pressure profiles. The pressure fluctuations obtained from CFD can be used for the analysis of structural stability of impeller blades.

II. LITERATURE REVIEW

F.C. Visseret. al. study has been carried out with the aid of three dimensional computational-fluid-dynamics calculations, employing the potential-flow approximation of the governing equations. The study was conducted because the first-stage impellers of the pump considered appeared to suffer from severe premature wear due to cavitation attack on the vane leading edges, this design had to be improved. The analysis carried out for the existing design produced suggestions for improvement, and based on these suggestions a new first-stage impeller design was developed. Subsequently, this new design was numerically analysed to substantiate its potentially better (cavitation) performance. It appeared that the blade inlet angle of the original impeller design was too excessive at mid span, causing best cavitation performance to occur at 160 percent of the rated flow. The new design has its best cavitation point at the rated flow, and will not suffer from premature wear due to cavitation attack like the existing design. The plan of discussion is first to present the numerical method of cavitation inception analysis employed, including some background on NPSH and cavitation. Next, (operating) details of the existing pump and analysis results for the existing design will be outlined. Then, lastly, the new design and its numerically predicted performance will be discussed. To solve this cavitation problem and reduce the resulting premature wear, a new first- stage impeller design has been presented. This new impeller has its best cavitation point near the rated flow, and is predicted to be free from cavitation for flow rates ranging from 85 till 110 percent of the rated flow. The findings presented have all been based on three- dimensional numerical flow-field calculations, employing the potential-flow approximation of the governing equations. This approach simplifies the flow problem (mathematically), while leaving the (physical) essentials intact.
TAN Lei1 et al. summarizes cavitation model proposed by Kunz et al., Medvitz et al. calculated the cavitating flow in acentrifugal pump at small cavitation numbers. They analysed the reason why the pump head drops at small cavitation numbers for the off-designed condition, but the assumption of the potential flow in their calculations limits their methods used in more application. Coutier-Delgosha et al. experimentally and numerically investigated the pressure distribution and hydraulic performance for a centrifugal pump with two-dimensional curved blade.
A computational model on the basis of the combination of the full cavitation model and the modified re-normalization group (RNG) k- € turbulence model are used to simulate the cavitating flow in a centrifugal pump at a low flow rate. A filter function is introduced for the RNG k-€ turbulence model in order to consider the compressibility in the cavitating flow. The calculated values of the net positive suction head available (H_{NPSHa}) agree well with the experimental measurements. The cavities in the impeller are obtained and their evolvement with H_{NPSHa} are revealed.

III. OBJECTIVES

- To study the existing system
- To carry out CFD analysis for varying discharge conditions
- To carry out structural analysis
- Modified geometry and its CFD and structural analysis

IV. SCOPE

1.Identification of problem and defining the same **-** Identify the precise nature of problem along with the relevant sizes at the suction and the discharge sides while limiting the study over a single variant of the pump (1 Hp).

2.Deploying appropriate Methodology- The problem being in the domain of fluid mechanics, CFD techniques are considered for solving the problem and generating alternatives for the solution. ANSYS Fluent will be deployed as a computational tool for solving the problem while generating alternative solutions.
3. Experimentation / Validation- Benchmarking the problem with a standard or norm followed by the sponsoring company, in general. Seeking to validate the solution for existing problem using miniature or full scale prototype for the delivery (discharge) end of the pipes based on the nature of the study undertaken. In case the experiment is too elaborate to undertake, secure the test data from the sponsoring company towards validation. In such case, validate the solution secured by analytical (CFD) methodology using, in turn, the Mathematical model for the problem.

V. METHODOLOGY

Mathematical Model: Using Mathematical model we are going to calculate the overall efficiency of the system by considering different losses of various combinations.
Analytical Method
In this method we are using FVM (i.e. Finite volume method). Process is carried out in four steps:
Step 1: 3D Modeling
Dimensions of the given system would be received from the sponsoring company. Based on this dimensions fluid domain would be generated using CAD Software.
Step 2: Pre-processor
In pre-processing stage discritization and application of boundary conditions would be carried out using software like Gambit or ICEM CFD. Generally tetrahedral element is used for meshing.
Step 3: Processing
ANSYS Fluent workbench is used for processing. In this workbench typically k-epsilon turbulent model with 2 equations is used for solving purpose.
Step 4 : Post-processing
In post processing we are using ANSYS Fluent workbench. We are going to study the vector plot for identifying the vortices region. We will also calculate the velocity and back pressure.
General Process Flow chart:

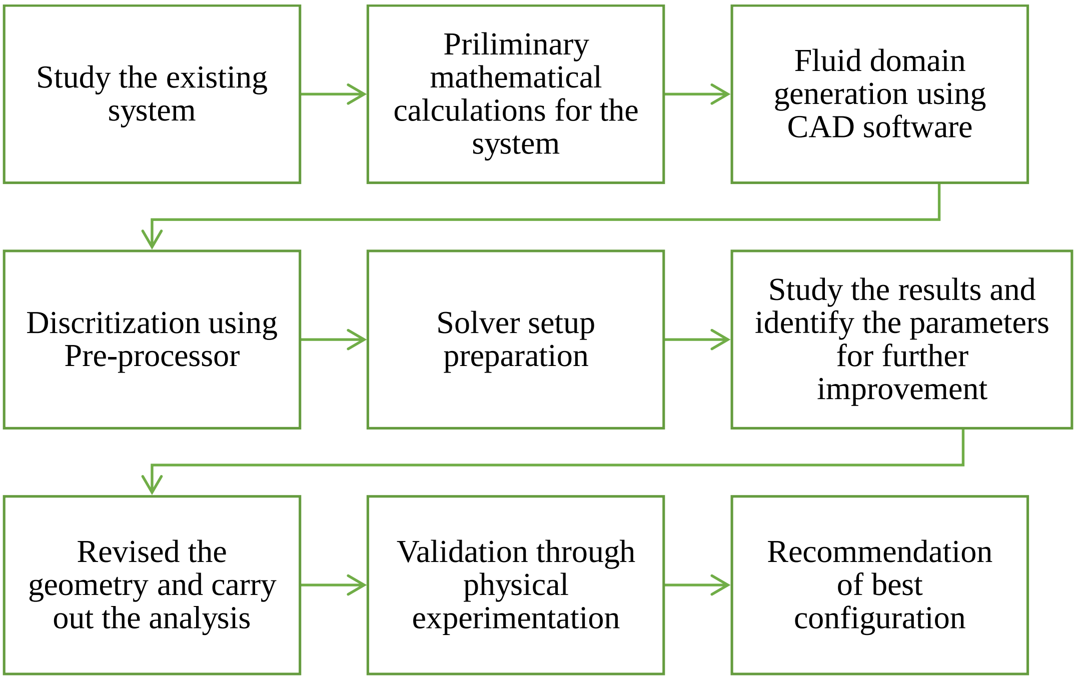

Chart -1

VI. VALIDATION AND CONCLUSION

Looking into the feasibility of making the prototype, the benchmark i.e. the existing pump is going to be used for experimentation. Comparison of the 'discharge' using numerical approach with the physical setup will be the basis for validation. During experimentation, time required for constant discharge is to be determined for the optimum variant. In this process a pump of 1HP is going to be used. Suitable suction head and discharged head will be administered during experimentation.

VII. FUTURE SCOPE

Benchmarking the problem with a standard . Seeking to validate the solution for existing problem using miniature or full scale prototype for the delivery (discharge) end of the pipes based on the nature of the study undertaken. In case the experiment is too elaborate to undertake, secure the test data towards validation. In such case, validate the solution secured by analytical (CFD) methodology using, in turn, the Mathematical model for the problem. And modify the vane angle that means pump gives the higher efficiency with high discharge.

REFERENCES

[1]. F.C. Visser, R.J.H. Dijkers& J.G.H. op de Woerd," Numerical flow-field analysis and design optimization of a high-energy first-stage centrifugal pump impeller" , Flowserve, The Netherlands (1999), PP.103-108.

[2]. TAN Lei, ZHU BaoShan, CAO ShuLiang& WANG YuMing1, "Cavitation flow simulation for a centrifugal pump at a low flow rate", Hydroscience and Engineering, Tsinghua University,Beijing100084,China (2012), PP.949-952.

[3]. Sankaranarayanan Ravi Scott J. Peltier Eric L. Petersen"Analysis of the impact of impeller geometry on the turbulent statistics inside a fan-stirred, cylindrical flame speed vessel using PIV",Berlin Heidelberg(2012), PP.1-16.

[4]. Suthep Kaewnai, Manuspong Chamaoot & Somchai Wongwises,"Predicting performance of radial flow type impeller of Centrifugal pump using CFD", Journal of Mechanical Science and Technology 23 (2009), PP.1620-1627.

[5]. Zhiyi YU, Guoyu WANG &Shuliang CAO ,"Extended two-fluid model applied to analysis of bubbly flow in Multiphase rotodynamic pump impeller", Mechanical Engineering, China (2009), PP.53-59.

[6]. BinCheng, Yonghai Yu "CFD Simulation and Optimization for Lateral Diversion and Intake Pumping Stations", International Conference on Modern Hydraulic Engineering,(2012), PP.122-127.

[7]. Porto D, Larson L A. "Multiphase pump field trials demonstrate practical application for the technology", Proc Society of Petroleum Engineers (SPE) Annual Technical Meeting, Houston, (1996).

[8]. VanOs, M.J., Jonker, J.B., Op de Woerd, J.G.H. "A geometric model for the parametric design of pump impellers", Paper C5-3. In: Proc. Pumpentagung, Karlsruhe, Germany, VDMA-Verlag (1996).

[9]. J. Gonzalez, J. Fernandez-Francos, E. Blanco and C. Santolaria-Marros, "Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump", Transactions of the ASME, Journal of Fluids Engineering, 124 (2002), PP.348-355.

[10]. J. F. Guelich, J. N. Favre and K. Denus, "An assessment of pump impeller predictions by 3D-Navier Stokes calculations", Proceedings of the ASME Fluids Engineering Division Summer Meeting (FEDSM'97), Vancouver, Canada, June 1997,paper FEDSM1997-3341.

[11]. Nitin S. Ghokhale, Sanjay S. Deshpande, Sanjeev V. Bedekar, Anand N Thite, "Practical Finite Element Analysis", Finite to Infinite Publications, Pune, pp1-175.

BIOGRAPHY

Namdev Tanaji Patil Student, ME Mechanical Design Engineering , Tatyasaheb Kore Institute of Engineering &Technology, Warananagar. Maharashtra, India

Narayan S Dharashivkar Professor, Department of Mechanical Engineering, Tatyasaheb Kore Institute of Engineering &Technology, Warananagar. Maharashtra, India Having 5 years industrial experience and 14 years teaching experience in the domain of Machine Design, FEA, Engg. Graphics, Theory of Machines, Mechanical System Design , Experimental Stress Analysis, Process Equipment Design;

Swapnil S.Kulkarni Director, Able Technologies India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM's and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.