International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal www.ijarst.in
IJARST
ISSN: 2457-0362

PERFORMANCE ANALYSIS OF A CENTRIFUGAL PUMP IMPELLER USING COMPUTATIONAL METHODS

C. Syamsundar[1*], G. Venkatasubbaiah[2]

[1]Department of Mechanical Engineering, CMR Engineering College, Hyderabad, Telangana 501 401, India

[2]Department of Mechanical Engineering, MVSR Engineering College, Nadergul, Hyderabad, Telangana 501 510, India

syamsundariitm@gmail.com

Abstract: Centrifugal pump is the most common pump used in industries, agriculture and domestic applications. The design of a centrifugal pump impeller demands a detailed understanding of the internal flow during design and off-design operating conditions. The present paper describes the simulation of the flow in a centrifugal pump impeller at five different flow coefficients viz. 0.0146, 0.0346, 0.0546 (designed flow coefficient), 0.0746 and 0.0946. The numerical solution of the discretized three-dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with the CFD package Ansys-CFX. For each flow co-efficient, performance results, static pressure contours, absolute velocity vectors, blade loading plots, mass averaged total pressure and static pressure, area-averaged absolute velocity, and pressure variation in the blade-to-blade passage are presented.

Keywords: Centrifugal pump impeller, CFD analysis, Flow coefficient, Pressure contours, velocity vectors.

Nomenclature

b Blade width
D_0 Impeller eye diameter
H Pump head
N Speed of the impeller
N_s Specific speed
Q Flow rate
U_1 Tangential velocity at inlet
U_2 Tangential velocity at the outlet
WG Water Gauge
Z Number of blades
ρ Density
η Hydraulic efficiency
β_1 Inlet vane angle β_2 Outlet vane angle
Φ Flow Coefficient

1. Introduction

The complexity of the flow in any turbomachine is due to the three-dimensional blade geometry, turbulence, secondary flows, unsteadiness etc. Computational fluid dynamics (CFD) has successfully contributed to the prediction of the flow through pumps and the enhancement of their design. Various researchers have considerably contributed to revealing the flow mechanisms inside centrifugal pump impellers with spiral volute or vaned diffuser volute aiming at the design of high-performance centrifugal turbomachines.

Computational analysis of a centrifugal pump presented by Bcharoudis et al., 2008 reveals the flow mechanisms inside centrifugal impellers and performance by varying outlet blade angles. They observed a gain in the head of more than 6% with an increase in outlet blade angle from 20o to 50o. Computational analysis of a centrifugal pump for the off-design volume flow rate is done by Khin Cho Thin et al., 2008.

They predicted the performance and calculated impeller volute disc friction loss, slip, shock losses, recirculation losses and other frictional losses. Optimization of blade inlet geometry of centrifugal pump impeller is done by Vasilios et al., 2005. John S. Anagnostopoulos, 2006 solved RANS equations for the impeller of a centrifugal pump and developed a fully automated algorithm for impeller design. Computational analysis of a centrifugal pump handling viscous fluids is presented by Shojaee Fard and Boyaghchi, 2007. They observed performance improvements in the centrifugal pump with an increase in the outlet blade angle due to a decrease in wake formation at the exit of the impeller.

Experimental investigation of centrifugal pump impeller handling both water and viscous oil is done by Wen-Guang, 2006. He observed that the blade discharge angle has a significant influence on the head, shaft power and efficiency of the centrifugal oil pump at various viscosity conditions. The effects of flow behaviour in a non-traditional centrifugal pump, whose diffuser was subjected to three different radial gaps (10%, 15%, and 20%), were investigated numerically by Adnan Ozturk et al., 2009. The last decade of research involves more advanced computational results. Recently, Milan Sedlr et al., 2009, Min-Guan Yang et al., 2007, Timar, 2005, Barrio et al., 2010, TAN Minggao et al., 2010, and Cheah et al., 2007, extended the prediction of the performance at various operating conditions.

Several algorithms have been proposed and developed, targeting the numerical simulation of the flow field of a centrifugal pump impeller. These algorithms apply either pressure-based or density-based methods for the solution of Navier Stokes equations. Weidong Zhou et al., 2003, Miguel Asuaje et al., 2005, Michalis et al., 2005, and Guleren and Pinarbasi, 2004, provide a review technique that is useful to study flow mechanisms inside a centrifugal impeller. Soon-Sam Hong and Shin-Hyoung Kang, 2002 have studied exit flow measurements of a centrifugal pump impeller. This data can be used for a centrifugal pump impeller design and validation of CFD codes and flow modelling. The present paper is concerned with the influence of the flow coefficient on the performance of a centrifugal pump impeller. The computational fluid dynamics analysis is carried out with ANSYS-CFX.

2. Pump Specifications

The specification of the centrifugal pump undertaken in the current analysis is shown in Table No. 1. The centrifugal pump impeller model with six blades is shown in Figure 1.

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
ISSN: 2457-0362
www.ijarst.in
IJARST

Table No. 1: Specifications of Centrifugal Pump Impeller

Specifications		Values
Blade width b	:	20 mm
The diameter of the impeller at the suction side D_1	:	150 mm
The diameter of the impeller at the pressure side D_2	:	280 mm
Outlet blade angle β_2	:	20^o
Pump head H	:	10 m
Speed of the impeller N	:	925 rpm
Flow rate, Q	:	0.0125 m^3/sec
Specific speed N_s	:	18.39
The diameter of the impeller eye	:	43.5 mm
Hydraulic Efficiency	:	83%
Number of Blades	:	6

3. Governing Equations:

The incompressible flow through the rotating impeller is solved with a moving frame of reference with constant rotational speed. 3-Dincompressible Navier-Stokes equations with the rotational force source term are solved to analyze the flow in a centrifugal pump. Turbulence ismodelled with the k-ε turbulent model. The geometry and the mesh of a six-bladed pump impeller domain were generated with Ansys Workbench. Unstructured meshes with tetrahedral cells are used for the domain of impeller as shown in Figure 2.

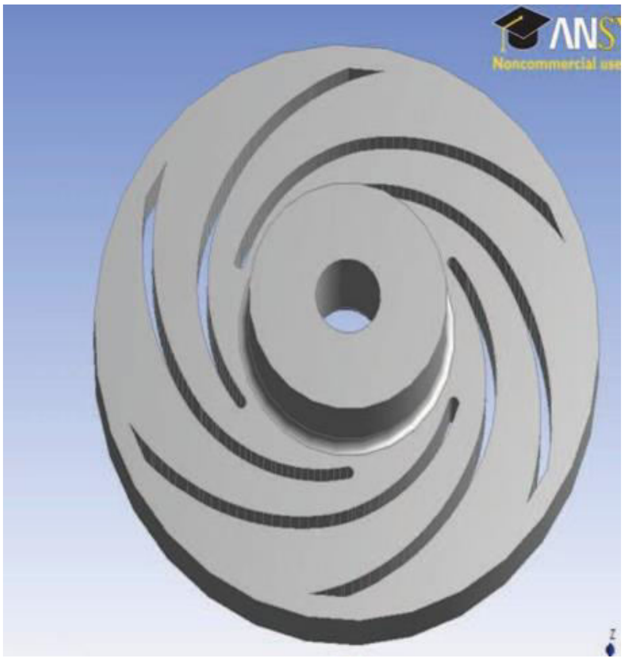


Figure 1: 3D Model of the radial impeller

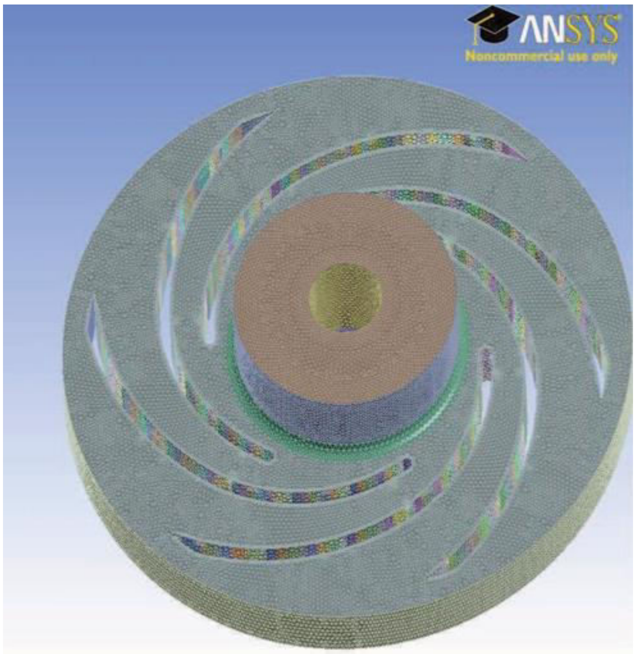


Figure 2: Unstructured mesh of the impeller

International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal
www.ijarst.in
IJARST
ISSN: 2457-0362

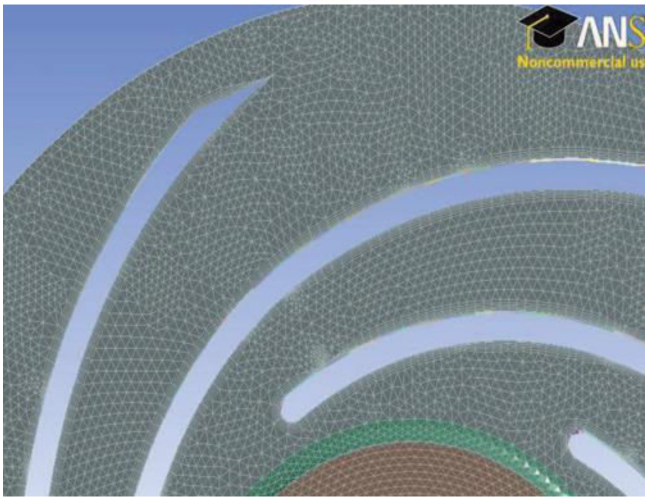


Figure 3: Mesh refinement around the bladesurface and inflated layers

The mesh is refined in the near tongue region of the volute as well as in the regions close to the leading and trailing edge of the blades. Around the blades, structured prismatic cells are generated to obtain better boundary layer details. Figure 3 shows the mesh refinement around the blade surface and inflated layers around the blades. A total of 3,570,268 elements are generated for the impeller domain. Mesh statistics are presented in Table No.2.

4. Boundary Conditions

Centrifugal pump impeller closed type domain is considered as a rotating frame of reference with a rotational speed of 925 rpm anti-clockwise direction. The working fluid through the pump is water at 25°C. The casing is not taken into account for analysis. k-ε turbulence model with a turbulence intensity of 2% is considered. Non-slip boundary conditions have been imposed over the impellerblades, hub and shroud. The roughness of all walls is considered 100μm. Inlet static pressure and outlet mass flow rate are given based on flow coefficients. The convergence precision of residuals was considered as 10^{-5}. Three-dimensional incompressible N-S equations are solved with Ansys-CFX Solver.

Table No.2: Mesh Statistics

Total number of nodes	:746141
Total number of tetrahedral	:3318311
Total number of pyramids	:879
Total number of prisms	:251078
Total number of elements	:3570268

5. Results

The centrifugal pump impeller is solved for five different flow coefficients. The performance results for different flow coefficients are presented in Table No.3. The head obtained by the CFD analysis for the designed flow coefficient is 9.45m WG and static efficiency is 61.22%. These results are compared with the Bcharoudis et al., 2008 experimental results.

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal www.ijarst.in
ISSN: 2457-0362

Table No.3: Centrifugal Pump Performance Results

Performance Results					
Flow Coefficient	**0.0146**	**0.0346**	**0.0546**	**0.0746**	**0.0946**
Head Coefficient	0.1944	0.1667	0.1334	0.0959	0.0522
Mass FlowRate [Kg/s]	3.33	7.91	12.52	17.06	21.64
Head (LE-TE) [m]	11.49	10.15	9.24	6.09	3.54
Head [m]	12.31	10.56	9.45	6.07	3.30
Shaft Power [kW]	2.59	5.21	6.95	7.89	8.04
Power Coefficient	0.0025	0.0051	0.0068	0.0077	0.0079
Static Efficiency %	54.01	60.43	61.22	54.10	31.92

5.1 Static pressure contours on mid-span for different flow coefficients

Figure 4 shows static pressure contours on the mid-span for different flow coefficients. It is observed that the static pressure increases gradually from the impeller inlet to the outlet due to the energy transfer from the rotating impeller. Static pressure on the pressure side is more compared to the suction side. The static pressure contours vary with span. Pressures near the hub are higher than the shroud due to the transition of flow from the axial to the radial direction. The minimum value of the static pressure inside the impeller is observed at the leading edge of the blades on the suction side due to flow acceleration and blade-tongue interaction.

For mass flow rates 3.33 kg/s and 7.91 kg/s which are lower than the designed mass flow rate of 12.52kg/s, the lowest value of pressure is seen near the leading edge and starts to increase to higher values around the trailing edge. Higher values of pressures are observed for lower mass flow rates. The pressure is decreasing with increasing mass flow rates.

When the mass flow rate approaches the designed value of 12.52 kg/s, the pressure gradient is more uniform due to the absence of shock loss.

At higher mass flow rates 17.06 kg/s and 21.64 kg/s, the pressure decreases gradually from its maximum value near the leading and trailing edge for all the blades.

5.2 Absolute velocity vectors near the leading edge at mid-span for different flow coefficients

At designed or more than designed mass flow rates, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortex at the pressure side of the blade and low-pressure regions. Figure 5 shows absolute velocity vectors near the leading edge at the mid-span for different flow coefficients.

International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

When the mass flow rate is less than designed, a recirculation zone is established near the leading edge of each blade. It is observed that the absolute velocity vector distribution is not uniform in the blade passage across the impeller width.

5.3 Blade loading at 50% span location for differentflow coefficients

When the mass flow rate is less than the designed, a gradual increase of pressure is observed with stream-wise increment. High pressures on the pressure side of the blade and low pressures on the suction side of the blade are observed as shown in Figure 6.

When the mass flow rate is designed or more than the designed value, at the leading edge, there is apressure drop on the pressure side is observed. This is due to the presence of a weak vortex at the pressure side of the blade.

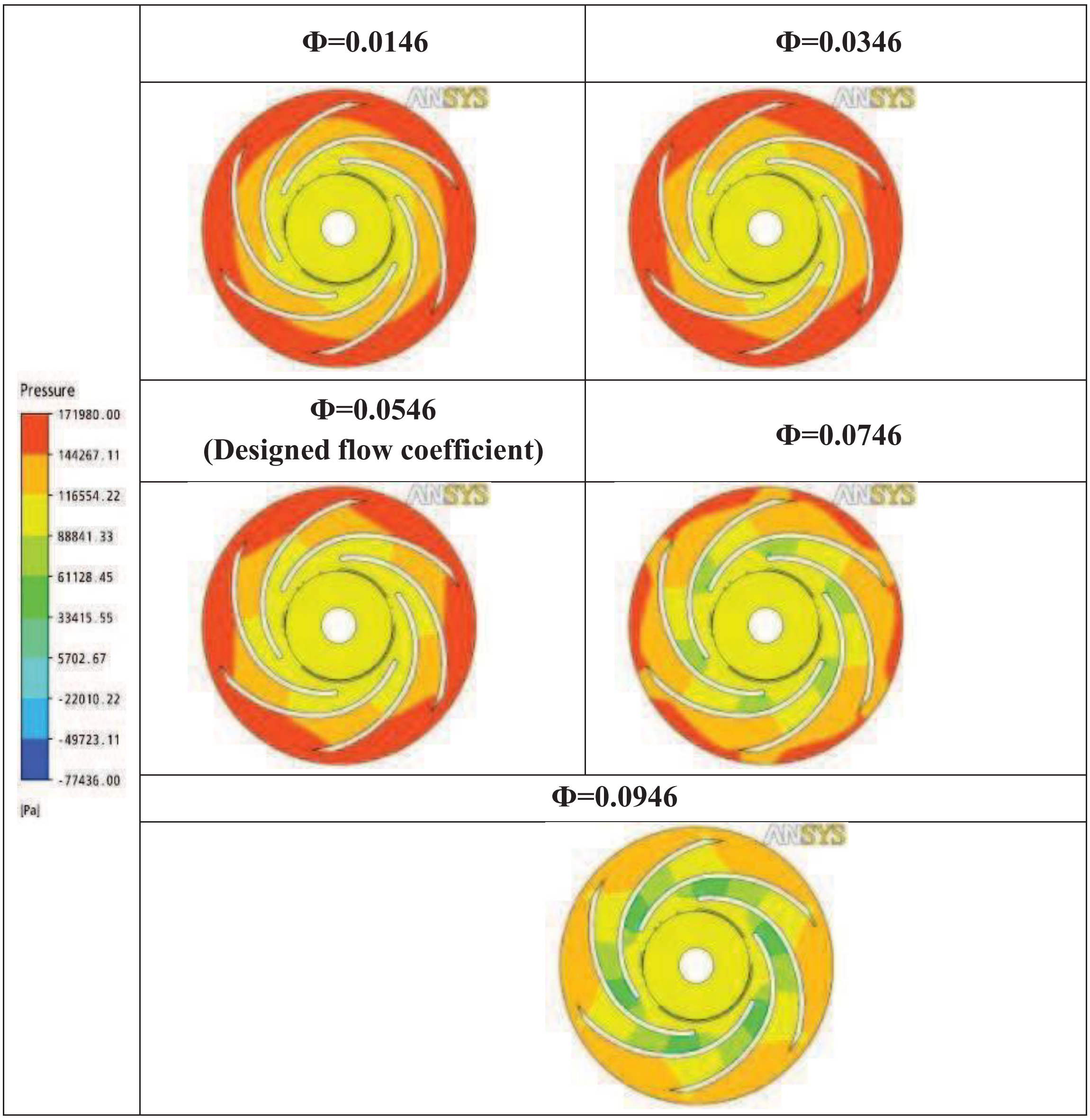


Figure 4: Static Pressure Contours on mid-span fordifferent flow coefficients

International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362
IJARST

5.4 Streamwise variation of mass averaged totalpressure for different flow coefficients.
Streamwise variation of mass averaged total pressure is shown in Figure 7 for different flow coefficients. At 20% streamwise locations, constantmass averaged total pressures are observed due to the inlet duct before the impeller.

For high flow coefficients, a total pressure drop at the leading edge is observed due to the weak vortex.With the increase of flow coefficients drop in total pressure is observed.

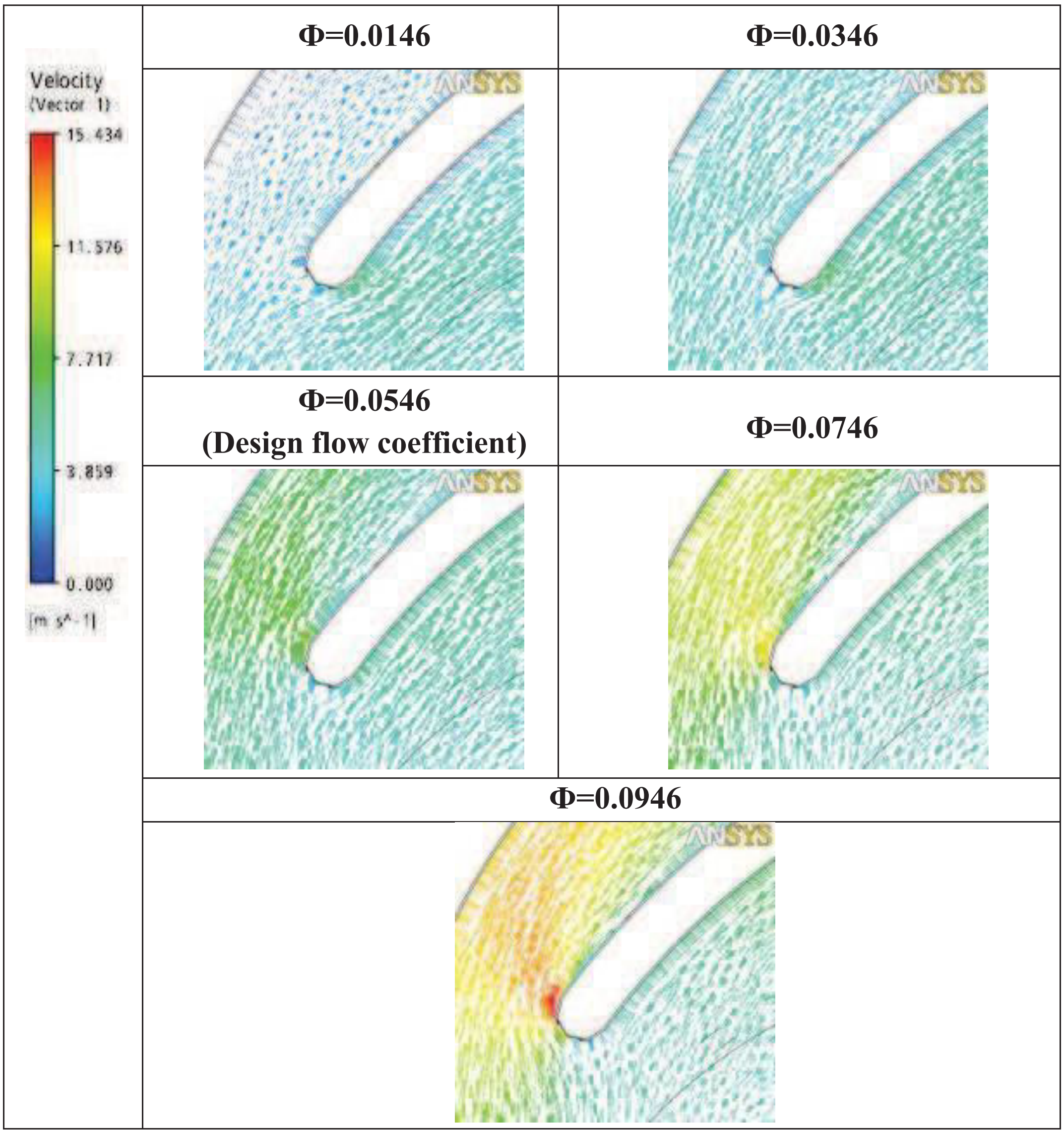


Figure 5: Absolute velocity vectors near the leading edge at themid-span for different flow coefficients

5.5 Streamwise variation of mass averaged staticpressure for different flow coefficients.
At streamwise locations, 0 to 0.2 the mass averaged static pressures are observed constant due to the inlet duct before the impeller.

From a 30% streamwise location, the pressuresare increasing due to the dynamic energy transfer from the rotating impeller to the fluid.

From a 90% streamwise location static pressure increment is observed because of diffusion in the outlet duct. With the increase of flow coefficients, a drop in static pressure is observed.

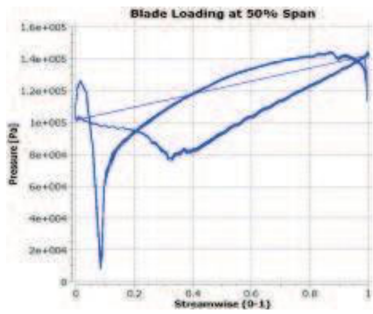
International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

Φ=0.0146

Φ=0.0346

Φ=0.0546

Φ=0.0746



Φ=0.0946

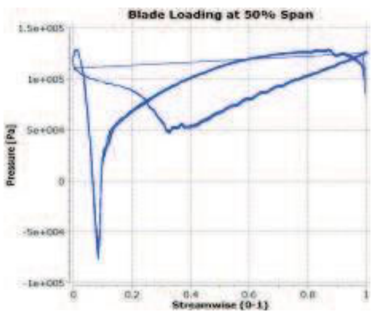


Figure 6: Blade loading at 50% span location for differentflow coefficients

5.6 Streamwise variation of area-averaged absolutevelocity for different flow coefficients

From a 20% streamwise location, the area averaged absolute velocity is increasing with streamwise increment due to the dynamic energy transfer from the impeller to the fluid. From a 90% streamwise location, the area averaged absolute velocity is decreasing because of an increase in pressure at the outlet duct.

For low flow coefficients higher value of absolute velocity is observed. With the increase of flow coefficients, there is a drop in absolute velocity.

International Journal For Advanced Research
In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

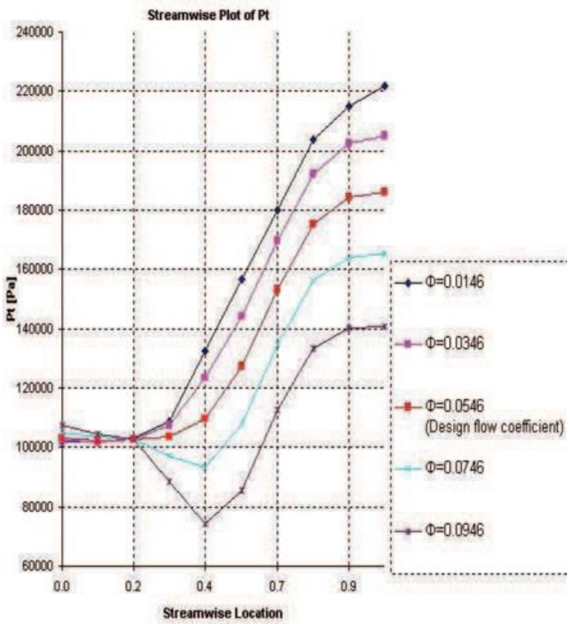


Figure 7: Streamwise variation of mass averaged totalpressure for different flow coefficients

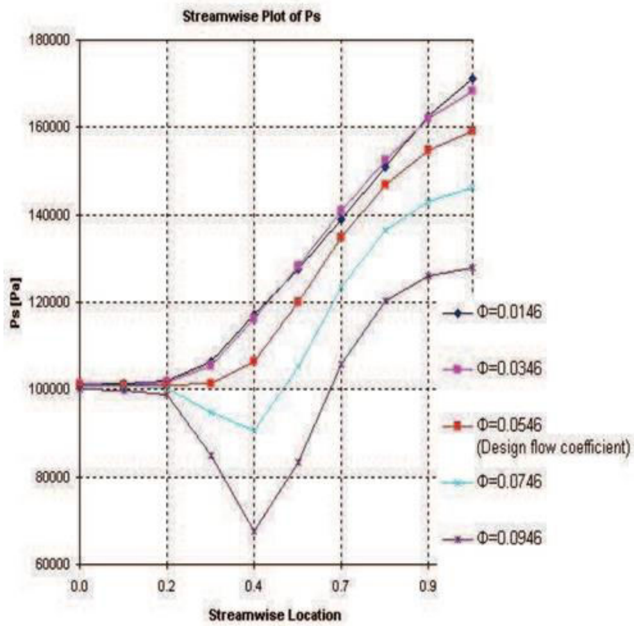


Figure 8: Streamwise variation of mass averagedstatic pressure for different flow coefficients

5.7 Pressure contours on the blade-to-blade plane fordifferent flow coefficients

The pressure increases gradually along a streamwise direction within the impeller blade-to-blade passage and has higher pressure on the pressure surface than the suction surface for each plane as shown in Figure 10.

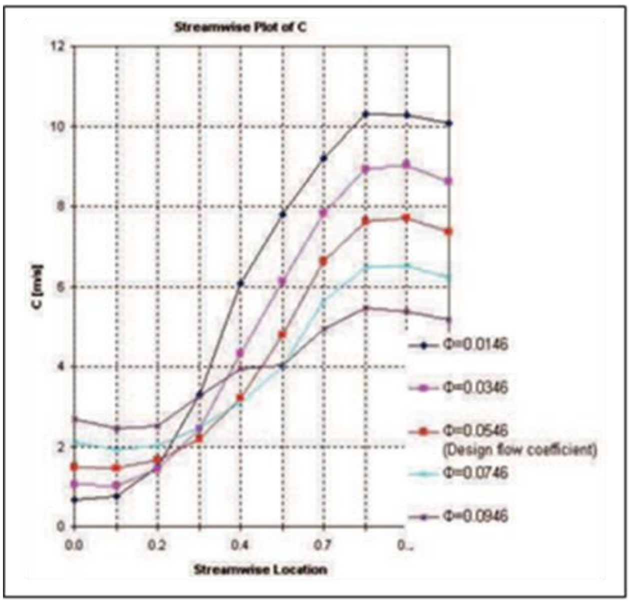


Figure 9: Streamwise variation of area- averaged absolutevelocity for different flow coefficients

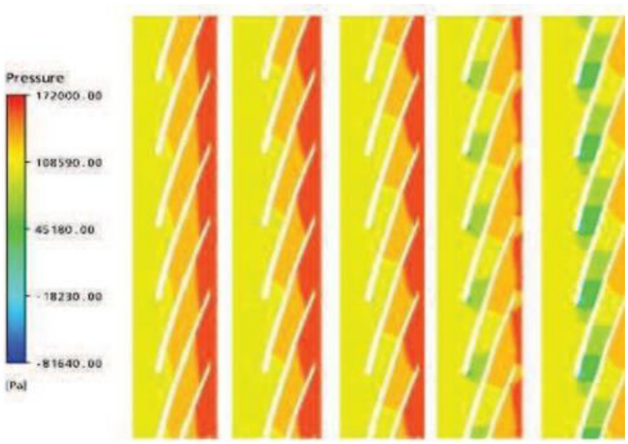


Figure 10: Pressure contours on blade-to-blade plane

6. CONCLUSIONS

A centrifugal pump impeller was designed and analyzed with the aid of computational flow dynamics. The flow patterns through the pump,performance results, static pressure contours, absolute velocity vectors, blade loading charts at 50% span, streamwise variation of mass averaged total pressure and static pressure, streamwise variation of area-averaged absolute velocity, and pressure contours on blade to blade plane are predicted for five different flow coefficients.

The CFD predicted value of the head at thedesigned flow rate is approximately H=9.4528 m. There is a 5.78% difference between the theoretical head and the predicted numerical head.

The increase in the designed flow rate causes a reduction in the total head of the pump. With the increase of mass flow rates drops in static pressures are observed from pressure contours on mid-span for different flow coefficients.

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal www.ijarst.in
IJARST ISSN: 2457-0362

At a designed or more than designed mass flow rate, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortexat the pressure side of the blade. On the pressure side of the blade static pressure drop is observed. At low mass flow rates, a recirculation zone isestablished near the leading edge of each blade.

7. References

1. Adnan Ozturk, Kadir Aydin, Besir Sahin and Ali Pinarbasi, 2009. Effect of impeller–diffuser radial gap ratio in a centrifugal pump, Journal of Scientific & Industrial Research, Vol. 68. pp.203-213.
2. Barrio, R., Fernández, J., Parrondo. J., and Blanco, E., 2010. Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid Dynamics, International Conference on Renewable Energies and Power Quality,(ICREPQ'10), Granada (Spain).
3. Bcharoudis, E.C., ilios, A.E., Mentzos, M.D., and Margaris, D.P., 2008. Parametric study of a centrifugal pump impeller by varying the outlet blade angle, The Open Mechanical Engineering Journal, pp75-83.
4. Cheah, K.W., Lee, T. S., H.Winoto, S., and Zhao, Z. M., 2007. Numerical flow simulation in a centrifugal pump at design and off-design conditions, Hindawi Publishing Corporation, International Journal of Rotating Machinery, Article ID 83641. 8 pages.
5. Guleren, K.M., and Pinarbasi, A., 2004. Numerical simulation of the stalled flow within a vaned centrifugal pump, Journal of Mechanical Engineering Science, Vol. 218, pp 1-11.
6. John S. Anagnostopoulos, 2006. CFD analysis and design effects in a radial pump impeller, Wseas Transactions on Fluid Mechanics.
7. Khin Cho Thin, Mya Mya Khaing, and KhinMaungAye, 2008. Design and performance analysis of centrifugal pump, World Academy of Science, Engineering and Technology, pp46,
8. Michalis, D., Mentzos, Andronicos, E., Filios, A.E., Dionisios, P., Margaris, and Dimitrios, G., Papanikas, 2005. CFD predictions of flow through a centrifugal pump impeller, 1st International Conference on Experiments/Process/System Modeling/ Simulation/ Optimization, 1st IC-EpsMsO Athens.
9. Miguel Asuaje, Farid Bakir, Frank Kenyery, and Robert Rey, 2005. Numerical modelization of the flow in centrifugal pump: Volute influence in velocity and pressure fields, International Journal of Rotating Machinery. pp. 244– 255.
10. Milan Sedlr, Jiri Soukal, and Martin Komarek, 2009. CFD Analysis of the middle stage of multistage pump operating in Turbine regime, Journal of Engineering Mechanics, Vol. 16. pp. 413–421.
11. Min-Guan Yang, Dong Liu and Xiang Dong, 2007. Analysis of turbulent flow in the impeller of a chemical pump, Journal of Engineering Science and Technology, Vol. 2. pp. 218 – 225.
12. Shojaee Fard, M.H., and Boyaghchi, F.A., 2007. Studies on the influence of various blade outlet angles in a centrifugal pump when handling viscous fluids, American Journal of Applied Sciences. pp.718-724.

13. Soon-Sam Hong and Shin-Hyoung Kang, 2002. Exit flow measurements of a centrifugal pump impeller, KSME International Journal, vol No: 16. pp.1147- 1155.
14. TAN Minggao, YUAN Shouqi, LIU Houlin, WANG Yong, and WANG Kai, 2010. Numerical research on performance prediction for centrifugal pumps, Chinese Journal of Mechanical Engineering, Vol.23.
15. TIMÁR. P., 2005. Dimensionless characteristics of centrifugal pump, Department of Chemical and Biochemical Engineering, Faculty of Chemical and Food Technology, Slovak University of Technology. pp. 812-837.
16. Vasilios A. Grapsas, John S. Anag nostopoulos and Dimitrios E. Papantonis, 2005. Hydrodynamic designs of radial flow pump impeller by surface parameterization, International Conference on Experiments / Process / System Modeling / Simulation / Optimization, Athens, 6-9 July.
17. Weidong Zhou, Zhimei Zhao, Lee, T.S., and Winoto, S.H., 2003. Investigation of flow through centrifugal pump impellers using computational fluid dynamics, International Journal of Rotating Machinery. pp 49–61,
18. Wen-Guanf LI, 2006. Blade exit angle effects on the performance of a standard industrial centrifugal oil pump, Department of Fluid Machinery, Lanzhou University of Technology, Lanzhou, P R China.