CFD Analysis of Domestic Centrifugal Pump for Performance Enhancement

Satish M.Rajmane[1], Dr.S.P.Kallurkar[2].

[1]*Research Scholar, WIT Research Center, Solapur University, Solapur, India (satishrajmane79@gmail.com)*
[2] *Principal, Atharva College of Engineering, Mumbai, India (drkallurkar@yahoo.co.in)*

---***---

Abstract- *A centrifugal pump is one of the simplest pieces of equipment in any process plant. Its purpose is to convert the energy of a prime mover (an electric motor or turbine) first into velocity or kinetic energy and then into pressure energy of a fluid that is being pumped. The energy changes occur by virtue of two main parts of the pump, the impeller and the volute or diffuser. The impeller is the rotating part that converts driver energy into the kinetic energy. The volute or diffuser is the stationary part that converts the kinetic energy into pressure energy. The effort of this dissertation work would be to improve the head of the regenerative pump which is available in the market. For this the pump is modeled in modeling software CATIA and the geometry is imported in the ANSYS fluent analysis. By analyzing existing design the results are compared with the analytical solutions.*

Key Words: Regenerative pump, CFD, ANSYS Fluent

1. INTRODUCTION

A pump is a machine used to move liquid through a piping system and to raise the pressure of the liquid. A pump can be further defined as a machine that uses several energy transformations to increase the pressure of a liquid. Regenerative turbine pump is a type of centrifugal pump that uses a rotating impeller to increase the velocity of the fluid.

However, the impeller on a regenerative turbine pump looks quite different from a centrifugal type impeller. The impeller has radially oriented buckets or teeth, which make the impeller look more like a turbine rotor. Turbine pumps are special types of centrifugal pumps which use turbine-like impellers with radially oriented teeth to move fluid. They are also referred to as vortex, periphery, or regenerative pumps. These pumps combine higher discharge pressures of positive displacement or multistage centrifugal pumps with the flexible operation of centrifugal pumps. Additionally, the flow rate of turbine pumps is not extremely variable with large changes in pressure like in most centrifugal pumps. However, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters, the effect of which cannot be directly evaluated. For this reason CFD analysis is currently being used in the design and construction stage of various pump types. It can be noted from the literature that the effect of varying the number of impeller blades and the effect of geometry change on the inlet and outlet flow passage of the pump on its performance has not been the focus of research and hence an attempt has been made to numerically establish the effect of the above geometrical modification on the pump performance.

Figure 1 shows the impeller of a regenerative pump adopted in the analysis. The pump was dismantled after recording its performance parameters and the geometrical dimensions were measured. Using the measured geometric details, the numerical model was constructed and the CFD analysis was carried out with the ANSYS (Fluent) software then validation of the numerical model with calculations is done.

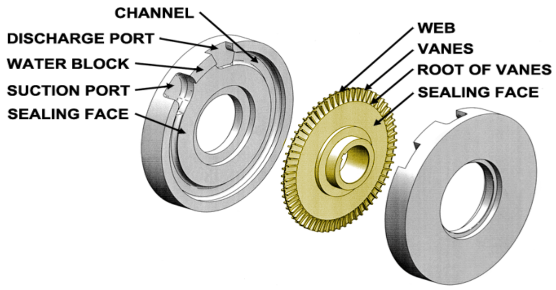


Fig 1: Regenerative turbine pump impeller showing radial vanes on each side of its rim

S. C. Chaudhari [1] suggested optimum inlet and outlet blade angle for the mixed flow pump. Kapil Pandya1 [2] gone through various approaches used in CFD analysis of centrifugal pump and highlight the advantages and application of CFD analysis in turbo industries. P.Gurupranesh[3] to enhance the performance of the centrifugal pump through design modification of impeller. A.Manivannan [6] performed detailed CFD analysis to predict the flow pattern inside the impeller which is an active pump component. Weidong Zhou [8] describes the three-dimensional simulation of internal flow in three different types of centrifugal pumps. Mr. Jekim [7] explained the effect of various operating conditions like Head, Discharge, Power and Speed on the performance of the pump.

1.1 Pump Specifications

Figure 2 shows the photo of the Casing and its CATIA model. Whereas Figure 3 shows the photo of the impeller of actual pump and its CATIA model

Fig 2: Image of Actual Pump Casing and its CATIA Model

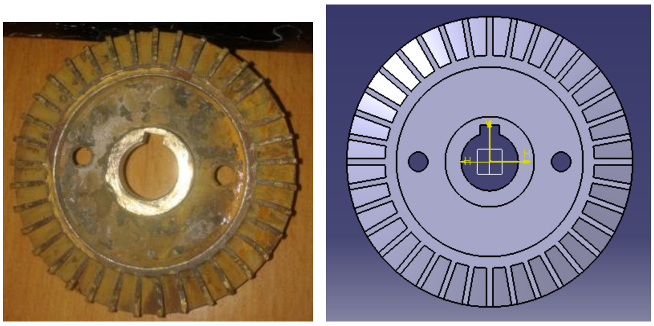


Fig 3: Image of Actual Pump Impeller and its CATIA Model

Table -1: Specifications

Power Input	0.9 kW
Pump kW/HP	0.37/0.5
Maximum Current	2.5 A
Frequency	50Hz
Speed	2880 RPM
Voltage	220 V
Discharge	900 lph
Size	25 x 25 mm
Total Head	15 / 25 m
No. of vanes	36 vanes
Thickness of	7mm
Thickness of vane	1.2 mm
Diameter of impeller	60 mm

After finding the dimensions of the pump body it is modeled in CATIA, and then the fluid region is obtained. Figure 2 shows solid and fluid region of the model

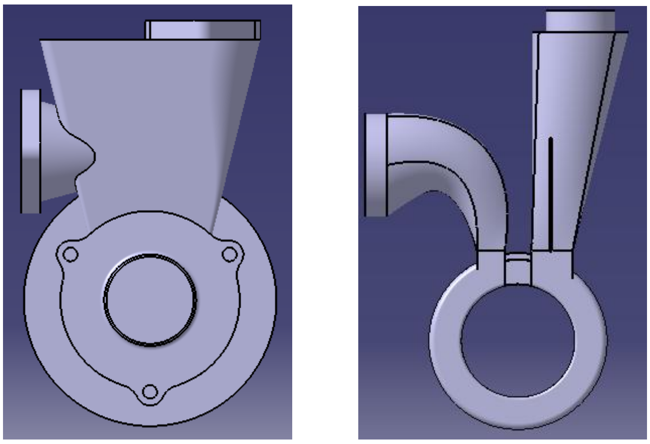


Fig 4: Solid region and Fluid region of the model

2. ANALYSIS OF THE PUMP

Meshing

The meshing of the model is done in ANSYS workbench itself. Before meshing interface surfaces are created in order to separate the fluid regions into stationary and rotating. Figure 3 shows the three dimensional mesh of the pump fluid region

Table -2: Meshing

Meshing Type	3D
Type of Element	Tetrahedral
No. of Nodes	161505
No. of Elements	801993

Fig 5: 3D Mesh of fluid region

Boundary Conditions

A centrifugal pump impeller domain is considered as the rotating frame of reference with a rotational speed of 2880 RPM. The working fluid through the pump is water. K-omega viscous turbulence model with turbulence intensity of 5% is considered.

3. RESULTS

Velocity Stream Contour

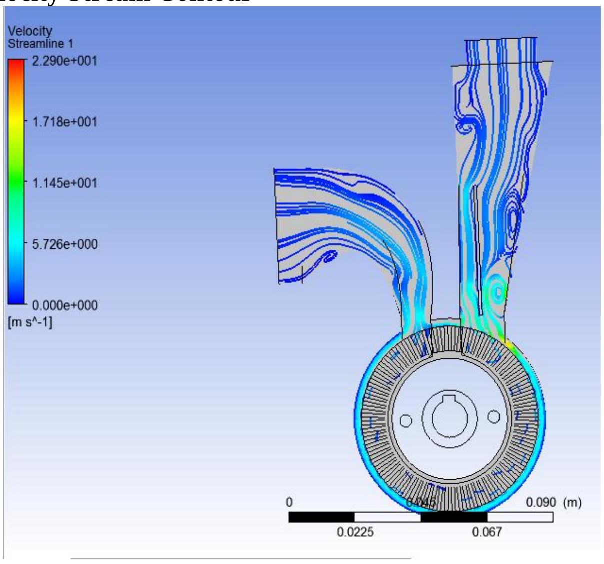


Fig 6: Streamlines at the central plane

Pressure contour

Inlet

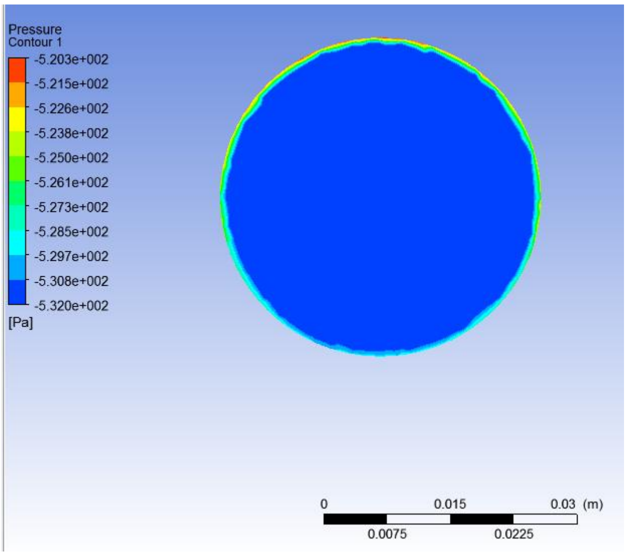


Fig 7: Pressure contour at the inlet

Outlet

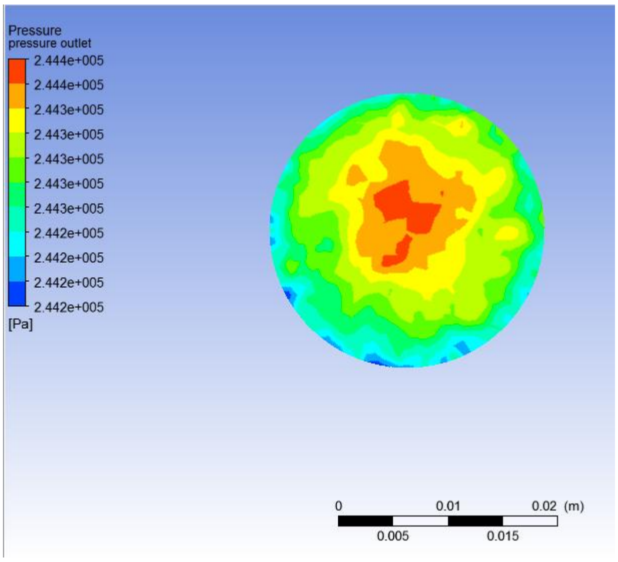


Fig 8: Pressure contour at the outlet

Figure 4 shows the stream lines of velocity from inlet to the outlet. Figure 5 shows the pressure distribution at the inlet of the pump whereas figure 6 shows the pressure distribution at the outlet of the pump.

$$\text{The head generated} = \frac{\text{Difference in pressure}}{\rho \cdot g}$$

$$\text{Head generated} = \frac{(245549-(-532))}{9.81*998.2}$$

$$= \mathbf{25.12\ m}$$

Table 3: Variation of Head

Sr.No	Description	Original Pump
1	Head (from specifications)	22m
2	Head (from CFD)	25.12m
3	Percentage Variation	14.18%

4. OPTIMIZATION

Many alternatives were made in the geometry of the pump to improve the head of the pump out of which following modifications showed some increase in the head.

Modification 1
Providing additional splitter in the outlet passage

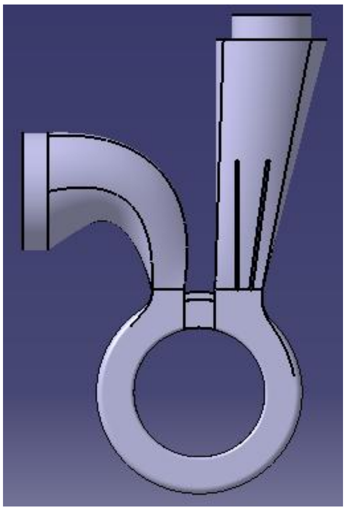


Fig 9: Model with additional splitter

Modification 2
Increasing Number of Vanes

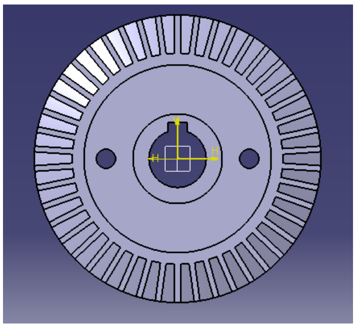


Fig 10: Impeller with 46 vanes

Modification 3
Inclining the Vanes

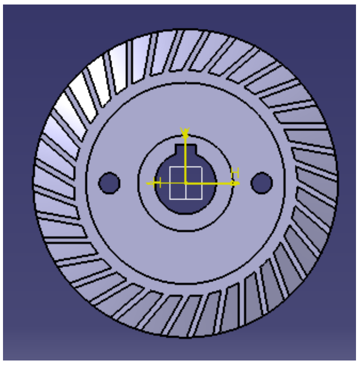
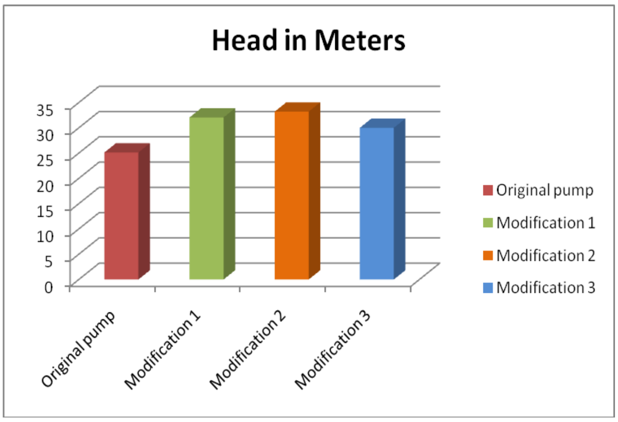


Fig 11: Impeller with 30° inclination

5. RESULTS OF THE MODIFIED MODEL

Sr.No	Pump	Head
1	**Original Pump**	25.12 m
2	Modification 1	32.05 m
3	Modification 2	33.20 m
4	Modification 3	30.00 m



5. CONCLUSIONS

There are a number of conclusions which may be drawn with regard to effectively matching the regenerative pump CFD model with the result from calculations. A good match or concurrence would indicate a strong basis for validation of the work. The error found between the analysis and the calculations is within permissible limit. CFD results produced a reasonable representation of the flow in a regenerative pump and are being utilized to focus investigation for unit performance improvement. As the capabilities of CFD continue to develop, it is to be expected that the uncertainties associated with CFD prediction should also reduce. At the very least it is to be expected that there will be a continuing growth in processing power for the foreseeable future, which will reduce and perhaps remove the geometric simplifications which have to currently be made.

i) By first modification we found an increase of 6 m in head of the pump which would increase the efficiency of the pump.

ii) By increasing number of vanes we found 7 m head rise in the pump.

iii) By giving inclination to the vanes we found 5m rise in head.

ACKNOWLEDGEMENT

Authors are also thankful to Mr.Umesh.D.Sulakhe who is M.E student (Second Year) of Solapur University.

REFERENCES

[1] S. C. Chaudhari, C. O. Yadav & A. B. Damor, A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump, International Journal of Research in Engineering & Technology (IJRET) ISSN 2321-8843 Vol. 1, Issue 3, Aug 2013, 57-64.

[2] Kapil Pandya, Chetankumar M.Patel, A Critical review on CFD Analysis of centrifugal pump impeller, International Journal of Advance Engineer ing and Research Development (IJAERD) Volume 1,Issue 6,June 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406

[3] P.Gurupranesh, R.C.Radha, N.Karthikeyan, CFD Analysis of centrifugal pump impeller for performance enhancement, IOSR Journal of Mechanical and Civil Engineering (IOSR- MCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 33-41

[4] S R Shah, S V Jain, V J Lakhera ,Cfd based flow analysis of centrifugal pump, Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December 16-18, 2010, IIT Madras, Chennai, India.

[5] K.W Cheah, T.S. Lee, and S.H Winoto, Unsteady Fluid Flow Study in a Centrifugal Pump By CFD Method, 7th ASEAN ANSYS Conference Biopolis, Singapore 30th and 31st October 2008

[6] A.Manivannan, Computational fluid dynamics analysis of a mixed flow pump impeller, International Journal of Engineering, Science and Technology Vol. 2, No. 6, 2010, pp. 200-206

[7] Mr. Jekim J. Damor, Prof. Dilip S. Patel , Prof. Kamlesh H.Thakkar , Prof. Pragnesh K. Brahmbhatt, Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study

[8] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H.Winoto, Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics. International Journal of Rotating Machinery, 9(1): 49–61, 2003 Copyright °c 2003 Taylor & Francis 1023-621X/03 DOI: 10.1080/10236210390147380

[9] Michalis D. Mentzos , Andronicos E. Filios, Dionisios P. Margaris, Evangelos Bacharoudis and Michalis Gr. Vrachopoulos, The use of cfd for flow analysis and performance prediction of centrifugal pumps, South-eastern europe fluent event 2005 11 - 13 May 2005, Porto Carras Grand Resort, Halkidiki, Greece

BIOGRAPHIES

He is having ME from Solapur University.He is Research Scholar for WIT Research Center, Solapur University, Solapur, India. He is having more than 10 years teaching experience .His area of interest is in Fluid machinery, FEM, Design engineering.

He is having PhD from NITIE Mumbai. Presently working as Principal in Atharva College of Engineering, Mumbai, India. He is Research Guide for WIT Research Center, Solapur University, Solapur, India . He is having more than 25 years teaching experience.

