# PERFORMANCE IMPROVEMENT OF EXTERNAL GEAR PUMP THROUGH CFD ANALYSIS

**P. S. Mali[1], G. S. Joshi[2], I. A. Patil[3]**

*[1]PG Student, DKTE's Textile and Engineering Institute, Ichalkaranji, India.*
*[2]Professor, DKTE's Textile and Engineering Institute, Ichalkaranji, India.*
*[3]Technical Director, Adroit Engineers, MIDC Area Shiroli Kolhapur, India.*

---***---

**Abstract –** *External gear pumps are the largest used hydraulic machine. However, the performance of external gear pump is generally not very good. Improving efficiency is a major challenge. The performance of an external gear pump is highly dependent on its geometric parameters. Computational fluid dynamics (CFD) has been found to be very good tool for numerical analysis of flow through complex system, including external gear pumps. This study involved the performance analysis of an external gear pump designed for external gear pumps using ANSYS CFX 16.0 software to deliver 0.85 lpm of oil at 2500 rpm. Modeling the external gear pump has been done using CATIA V5 software.*

*The results shows that for experimental analysis, the efficiency of the pump is 17.29% & head is 10.670 m. however, for CFD analysis the efficiency of the pump raised to 17.30% & head is 13.4086 m thus showing an improvement in performance of external gear pump.*

***Key Words*****: Computational Fluid Dynamics Analysis, External gear pump, Overall efficiency, ANSYS CFX, Pump performance.**

## 1. INTRODUCTION

A pump is a mechanical device that increases the pressure of a liquid pressure. The gear pump is a sturdy and simple positive displacement pump. Gear pumps are used in many different applications in the industrial and technical fields. However, their design and performance prediction process is still a difficult task. Therefore, CFD analysis is currently being used in the design and construction phases of various pump types. Complex flow in the gear pump casing can be analyzed by computational fluid dynamics (CFD) analysis software and advanced post-processing tools. The performance of the external gear pump is greatly affected by the geometry of the gear pump.

Computational Fluid Dynamics (CFD) is one of the Computer Aided Engineering (CAE) tools. CFD has recently been used as an alternative to studying complex fluid flow phenomena in pumps. It is rapidly becoming an important tool for analysis and design of hydraulic engineering.

CFD simulations can show flow conditions in external gear pumps and provide valuable information about the hydraulic design of external gear pumps. The simulation results are used to calculate or predict the performance of the external gear pump to replace or reduce the experiment in the pump design process. It will save a lot of labor and facilities and help shorten the design cycle. Therefore, significant improvements in the design of external gear pumps must be achieved through CFD analysis.

## 2. ANALYSIS OF PUMP

Modeling of the external gear pump casing is done in CATIA V5 software and then the CFD analysis of the external gear pump is performed. It is shown in fig1.

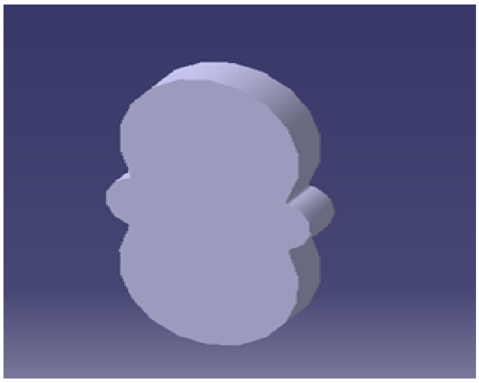

**Fig1.** 3D model of external gear pump casing

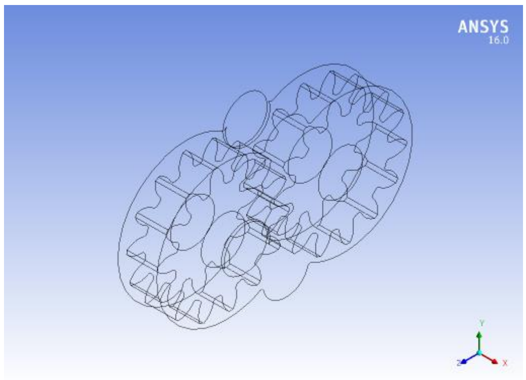


**Fig2.** Iso View of Wireframe

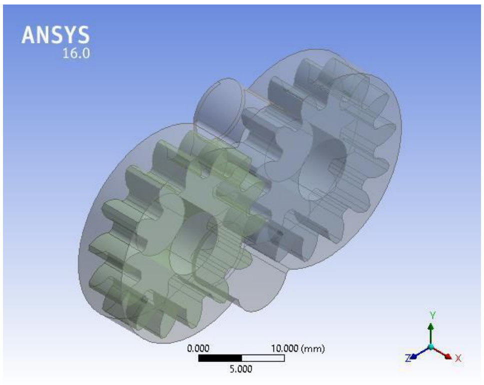


**Fig3.** Pump geometry for CFD analysis

## Meshing of the pump

Meshing is a part of modeling. After complete geometry mesh will be applying on geometry. Meshing geometry shown in fig4.

**Table1.** Mesh information of pump assembly

| Meshing type | 3D |
|---|---|
| Types of Element | Tetrahedral |
| No. of Nodes | 138594 |
| No. of Elements | 212419 |

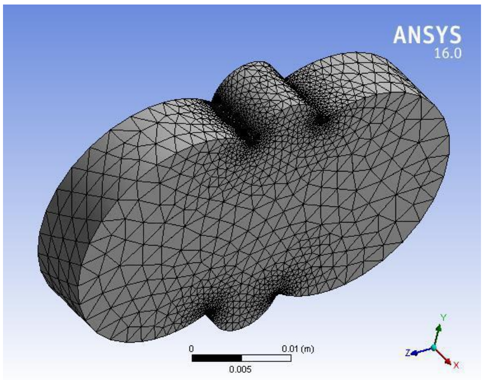


**Fig4.** Meshing of the pump

## Boundary conditions

An external gear pump domain is considered as the rotating frame of reference with a rotational speed of 2500 rpm. The boundary condition was 1 atm. at inlet and $1.4166 \times 10^{-5}$ $m^3/s$ at outlet. The working fluid in the pump is oil at 20°c. SST turbulence model with turbulence intensity of 5% is considered.

## Pressure contour

Inlet -

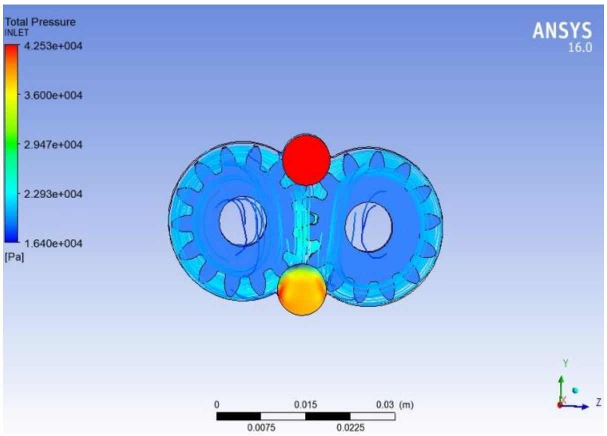


**Fig5.** Pressure contour at inlet

Outlet -

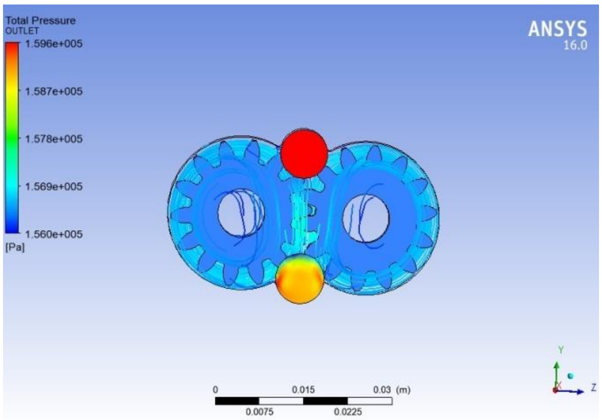


**Fig6.** Pressure contour at outlet

Velocity stream line contour-

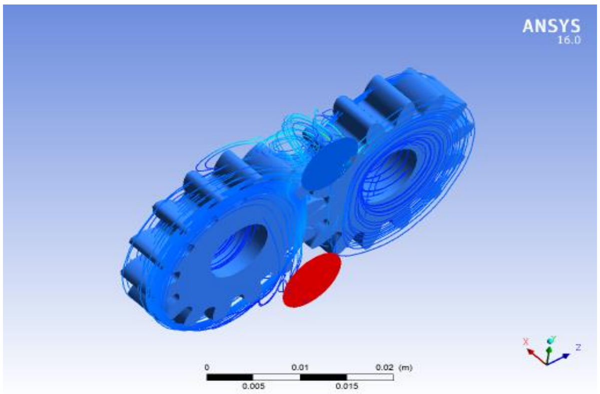


**Fig7.** Streamline flow in external gear pump

## 3. RESULTS

Inlet power = 2πNT/60*1000

= 2π*2500*0.0366/60*1000

= $9.5818*10^{-3}$ KW

Outlet power = ρ .Q. H/75

= 890. $1.4166*10^{-5}$. 13.41/ 75

= $2.2540*10^{-3}$ H.P

Or

Outlet power = $P_o$- $P_i$*Q/1000

= ($4.253*10^5$ - $1.596*10^4$).$1.4166\times10^{-5}$ / 1000

= $1.6584*10^{-3}$ KW

Overall efficiency = (outlet power/ inlet power)

= $1.6584*10^{-3}$/ $9.5818*10^{-3}$

= 17.30 %

**Table2.** Variation of efficiency

| Sr. no. | Description | Efficiency |
|---|---|---|
| 1. | Experimental analysis | 17.29 % |
| 2. | CFD analysis | 17.30 % |
| 3. | Percentage variation | 0.05783 |

Head generated = outlet pressure – inlet pressure/ ρ. g

= $1.596*10^5$-$4.253*10^4$/ 890×9.81

= 13.4086 m

**Table3.** Variation of head

| Sr. no. | Description | Head |
|---|---|---|
| 1. | Head (from experimental analysis) | 10.670 m |
| 2. | Head (from CFD) | 13.4086 m |
| 3. | Percentage variation | 25.67 % |

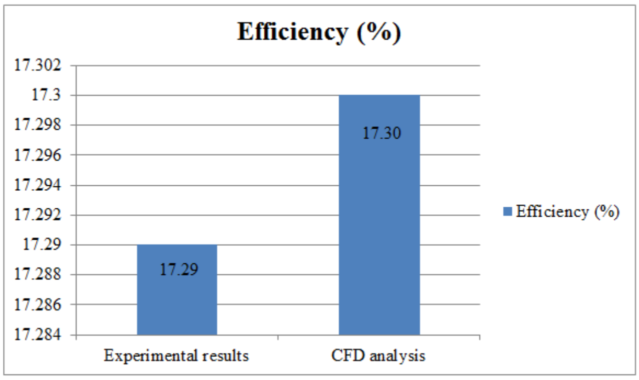


**Fig8.** Performance chart of external gear pump for efficiency

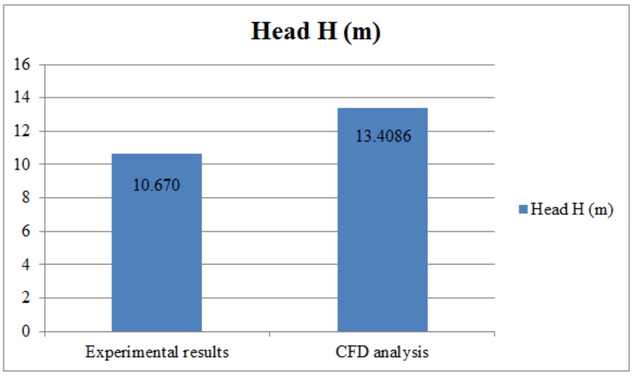


**Fig9.** Performance chart of external gear pump for head

## 4. CONCLUSION

CFD analysis of the pump is carried out to check the performance and efficiency of the pump. Efficiency of the pump from CFD results is coming 17.30% and by actual test efficiency is coming 17.29%, by which it is confirmed that CFD analysis is clearly validated. Efficiency predicted by CFD analysis is higher than the test result. Head predicted by CFD analysis is higher than the test result.

This shows that CAD and CAE tools are very useful in hydraulic design.

## 5. REFERENCES

1.Mr. Dipen Rana, Mr. Nirmal Kumar, "Experimental and Computational Fluid Dynamic Analysis of External Gear Pump", International Journal of Engineering Development and Research, volume 2, (2014), pages 2474-2478.

2.R. M. Pande, S. U. Kandharkar, "Computational Fluid Dynamics (CFD) of Centrifugal Pump to Study the Cavitation Effect", International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME), Volume 4, (2015).

3.S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX," International Journal of Engineering Research & Technology, Vol 1, (2012).

4.Raghavendra S Muttalli, Shweta Agrawal, Harshla Warudkar, "CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX ", International Journal of Innovative Research in Science, Engineering and Technology, vol 3, (Aug 2014).

5.Satish M.Rajmane, Dr.S.P.Kallurkar, "CFD Analysis of Domestic Centrifugal Pump for Performance Enhancement", International Research Journal of Engineering and Technology (IRJET), Volume: 02, (May-2015), pages 984-988.

6.Tilahun Nigussie, Edessa Dribssa, "Design and CFD analysis of centrifugal pump" International Journal of Engineering Research and General Science Volume 3, (May-June, 2015), pages 668-677.

7.P.Gurupranesh , R.C.Radha , N.Karthikeyan, "CFD Analysis of centrifugal pump impeller for performance enhancement", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), PP 33-41 .

8.Ankurkumar. H. Vyas, "Performance, Optimization and CFD Analysis of Submersible Pump Impeller", International Journal for Scientific Research & Development, vol. 1, (2013), Pages 859-861.

9.Krishna Kumar Yadav, V.K. Gahlot, "Performance improvement of mixed flow pump impeller through CFD analysis", International Journal of Research in Engineering and Technology, vol.4, (July 2015), Pages 243-247.