International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 15 (2018) pp. 11970-11973

CFD Simulation of External Gear Pump using ANSYS CFX

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.
Corresponding Author

Abstract

Use CFD technology to analyze and predict external gear pump performance. In this paper, Experimental investigations were conducted on external gear pump with a 24 mm outlet pump diameter, suction pressure 2 kg/cm^2, speed 2500 r.p.m and flow rate 0.85 lit/ min to assess the effect of various operating conditions like Head, Discharge , Power on the performance of the pump. Further the pump casing modeled using CATIA V5 software and computational fluid dynamics (CFD) analysis is carried out using ANSYS CFX software on the developed model of gear pump casing to predict the performance virtually and to verify with the experimental result of the pump and helps us rapidly to establish the efficiency of the pump over various operating conditions.

Keywords: Gear pumps, ANSYS CFX, CFD, Experiment of external gear pump test rig and performance curves.

INTRODUCTION

Computational Fluid Dynamics (CFD) is one of the Computer Aided Engineering (CAE) tools. Computational fluid dynamics (CFD) analysis is increasingly used for the design of external gear pumps. Using the CFD method, it is possible to well predict the complex internal flow within the gear pump casing. This is not fully understood and can speed up the pump design process. Therefore, CFD is an important tool for pump designers. At present, the use of CFD tools in the turbo machinery industry is very common. Computational fluid dynamics has now become an industrial design tool that helps shorten design time and improve overall process engineering.

Over the past 25 years, CFD has been increasingly used in various engineering applications. In the beginning, the use of these technologies was just a practice in the aviation and nuclear fields. Subsequently, this use has been extended to various products, physical conditions and manufacturing processes. Experimental and theoretical studies take a long time. In experiments, the actual physical model of the prototype actually needs to be manufactured, which is expensive. Due to this reason CFD analysis can also be carried out on the prototype. The CFD tool avoids every physical modeling and testing. Better and faster pump design and analysis can shorten the design cycle.

As a basic hydraulic pump, the gear pump has a simple structure, low cost and long service life. It is a widely used factor in mining machinery, engineering mechanics, metallurgical equipment, engineering machinery, agriculture and forestry machinery and other fields. In terms of design and performance prediction, CFD has proven to be a very useful tool for analyzing these turbo machines.

The purpose of this study was to conduct numerical studies on the external gear pump casing, taking into account the entire three-dimensional geometry and flow instability. It has been completed using the commercial software package CFX.

Working of gear pump

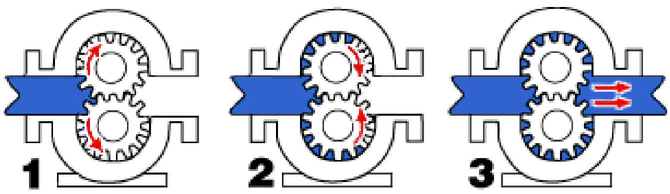

Figure 1. Working of external gear pump

The gear pump is made of two or more gears that rotate inside a closed casing. The motion of the drive gear is generated by the motor, and the motion of the driven gear is generated by the meshing of the teeth of the two gears. When the gear starts to rotate, the teeth come into contact with each other and disengage. When the teeth leave the contact area, a vacuum is generated. The liquid entering this space to fill the vacuum must be supplied through the inlet of the pump. Once filled with fluid, the fluid enters the pocket between the teeth and is trapped in place due to the sealed housing until it reaches the outlet port of the pump.

EXPERIMENTAL WORK

All of these experimental works was done in Adroit Engineers, Shiroli MIDC, Kolhapur, India. The experimental setup was successfully developed and assembled without any leakage problems. Experiment with oil as working fluid.

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 15 (2018) pp. 11970-11973

Figure 2. Experimental device

The experimental device consists of a gear pump, in which a vacuum gauge is mounted on the suction side and a pressure gauge is mounted on the discharge side for measuring the transmission head. The schematic diagram of the external gear pump test rig is shown in Figure 2. The test rig consists of a motor, oil sump, voltmeter, ammeter, tachometer, flow control valve, discharge measurement tank with stopwatch, matching gear, pedestal bearing block and belt pulley assembly. In the external gear pump test rig, flexible pipes are used for side suction and discharge. The main parameters observed from the test rig are the rotational speed of the gear pump, the vacuum pressure of the inlet oil (kg/cm^2), the discharge pressure of the outlet oil (kg/cm^2), the measured discharge flow (lit/min) and the head (Meter).

Test Methods

First all the varies is connected to the main current supply line. Through varies connection attached with ammeter and motor of single phase. After this work is done, the power switch is turned on and the voltage is changed by the tachometer to set the required motor speed. Set 2500 rpm with the aid of a tachometer and measure the inlet pressure and outlet pressure at this speed with a pressure gauge. When setting 2500 rpm, also measure amps to understand the actual load of the motor required for the speed. So we collect data on the motor speed 2500 rpm. The collected data is shown in the observation table. At 2500 rpm, the discharge flow was measured using a 10 liter volumetric flask. Start to set up and keep running. The oil depth of the tank was measured with the help of the scale. We performed six readings over a period of time. The entire experimental data and experimental results are shown in the table below.

Observation

Speed (N) - 2500 rpm

Suction pressure (P1) - $2kg/cm^2$

Flow rate (Q) - 0.85 lit/min

Table1. Experimental Observation Table

Sr. No.	Discharge pressure (P2) (kg/cm^2)	Head (m)	Efficiency (%)
1.	1.05	10.670	17.29 %
2.	0.85	12.916	
3.	1	11.232	
4.	0.85	12.916	
5.	1	11.232	
6.	1	11.232	

COMPUTATIONAL FLUID DYNAMICS METHODOLOGY

To model an external gear pump, catiav5 software was used to create a solid model as shown in fig.3. The solid model of Pump casing completed by Catiav5 is saved as a .CAT product format, and then is import into ANSYS workbench.

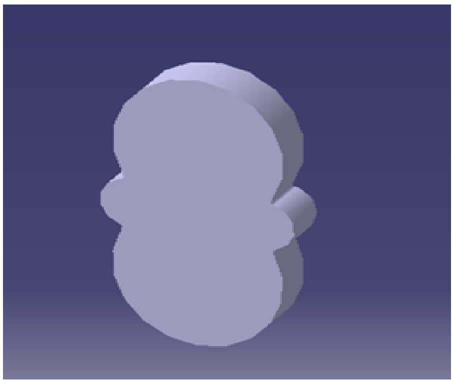

Figure 3. 3D geometric model of pump casing created by CATIA V5

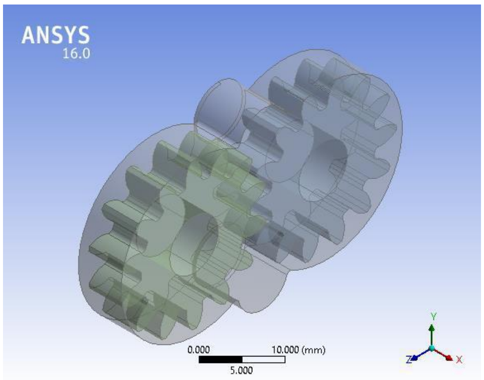

Figure 4. Pump geometry for CFD analysis

Meshing

In order to perform CFD analysis, meshing can be done in CFD software tool. There is one in the meshing process the important point is that the model should be used as an import Surface model. Then make a volume mesh.

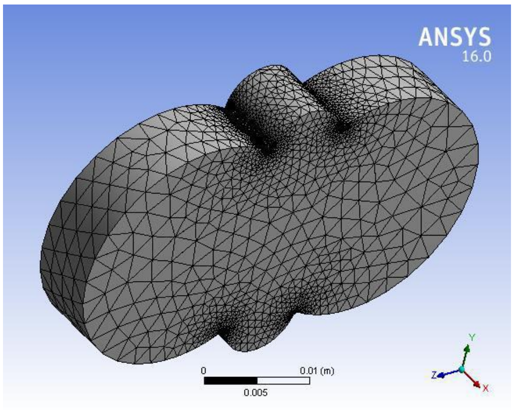

Figure 5. Meshed geometry

Meshing type	3D
Type of Element	Tetrahedral
No. of Nodes	138594
No. of Elements	212419

Boundary conditions

After meshing, apply boundary conditions and boundary conditions are the set of conditions specified for the behavior of the solution to a set of differential equations at the boundary of its domain. Mathematical solution is determined with the help of many boundary conditions physical problems. These conditions specify flow variables on the boundary of a physical model. External gear pump casing field is considered to be rotate the reference system at 2500 rpm. The boundary condition at the inlet is 1 atmosphere. The liquid in the pump is oil at 20°C. SST turbulence model Consider 5% turbulence intensity.

Then the resulting file is imported into CFX-Post and the results are obtained. The streamline flow in the casing of the pump is shown in the fig. below,

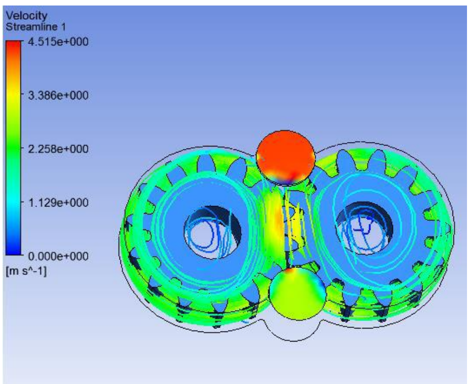

Figure 6. Streamline flow in gear pump casing

Pressure contour

Inlet -

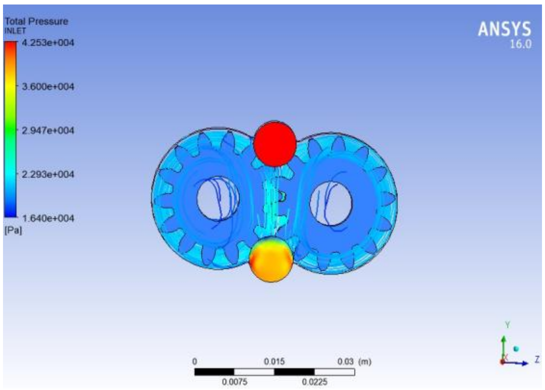

Figure 7. Pressure contour at the inlet

Here minimum outlet pressure is $1.640*10^4$ Pascal and maximum outlet pressure is $4.253*10^4$ Pascal.

Outlet -

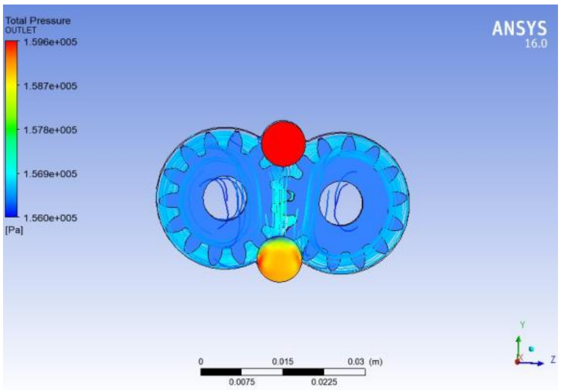

Figure 8. Pressure contour at the outlet

Here minimum outlet pressure is $1.560*10^5$ Pascal and maximum outlet pressure is $1.596*10^5$ Pascal.

COMPARISON BETWEEN CFD ANALYSIS AND EXPERIMENTAL TESTING

The comparison between the CFD analysis and the hydraulic performance test of the pump is shown in Table 2 and the performance curve of the pump is shown in fig 9.

Table 2. Comparison of CFD Analysis and Experimental Testing

Sr.	Parameters	CFD analysis	Experimental results
1.	Discharge (m^3/s)	1.4166×10^{-5}	1.4166×10^{-5}
2.	Head (m)	13.4086	10.670
3.	Pump input (KW)	9.5818×10^{-3}	7.6314×10^{-3}
4.	Pump efficiency (%)	17.30 %	17.29 %

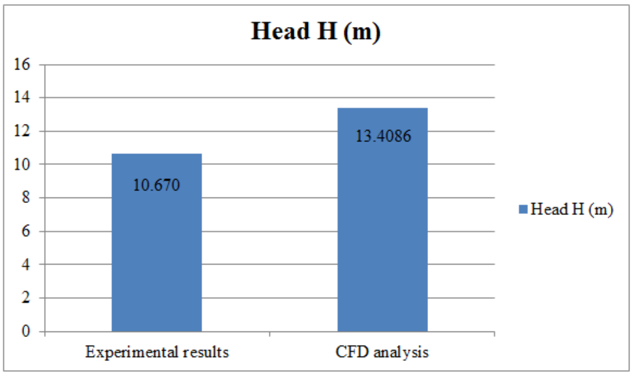

Figure 9. Chart showing performance curve of pump for total head

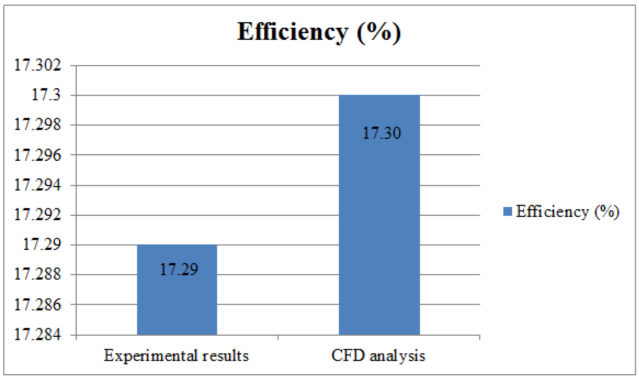

Figure 10. Chart showing performance curve of pump for efficiency

CONCLUSION

The CFD analysis of the external gear pump is based on the requirements to determine the performance of the external gear pump at a specific temperature and pressure. The theoretical results are compared with the CFD results, so we can say that the quantitative results match the desired results. The change in CFD results is due to the fact that surface finish factors play a very important role in determining the efficiency of the external gear pump. By CFD analysis, the obtained efficiency is 17.30 %, which is satisfactory.

The external gear pump efficiency can be increased by increasing the surface smoothness factor. The pump casing is an important part of the external gear pump and it plays a crucial role in determining the efficiency of the external gear pump. Therefore, the design and performance analysis of the pump assembly has been completed and it has been found that the efficiency is satisfactory.

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 .