Study on Performance of Plunger Pump at Various Crank Angle Using CFD

R.Ragoth singh
Assistant Professor, Kathir college of Engineering,
PhD Scholar, Anna University of Technology,
Coimbatore, India

Dr.M.Nataraj
Associate Professor, Government college of Technology, Coimbatore, India

Abstract - ***Reciprocating pumps are in use where precise amount of liquid is to be delivered in addition to higher delivery pressure. These pumps are normally used for pushing viscous liquids and injecting chemicals/additives. This paper presents the use of a general-purpose Computational Fluid Dynamics (CFD) to simulate the flow within the triplex pump and compare the results with experimental data. The results also demonstrate reasonable agreement with the trial investigational test. The parameters are currently being used to optimise key geometric features to augment pump efficiency. The performance of the pump such as discharge, percentage of slip and volumetric efficiency is validated consequently with pump theory for the modelled triplex pump. This paper summarizes the effect of volume flow rate considering at various crank angles considering water as working fluid for the investigation methods were revealed. The theoretical pump discharge and CFD results were compared.***

I. Introduction

During the last two decade there has been a progressive increase in the use of commercial CFD software to model the flow through turbo machinery. CFD makes it possible to evaluate velocity, pressure, temperature, and species concentration of fluid flow throughout a solution domain, allowing the design to be optimized prior to the prototype phase. Following this line of work, this paper presents an investigation on the ability of one such commercial code, ANSYS to predict the flow behaviour through a triplex pump. The study mainly considered the performance characteristics of the pump at different crank angles and also some specific features such as pressure, velocity and turbulence contours. Results are presented in terms of the designed flow rate using CFD. The performance of the pump flow rate has been compared with the CFD and experimental values.

II. Triplex pump

The triplex plunger pump is chosen for the analysis, because it is the most useful fluid machine which is widely used in chemical industry, automobile industry and pharmaceutical industry. The performance evaluation of plunger pump is taken to improve the efficiency by reduce the losses such as turbulence loss, shock losses, plunger friction losses and recirculation losses and also power consumption. Experimental investigations are generally carried out on pumps which are expensive, time consuming and limited to some extent. To reduce the number of experimental works, virtual analysis can be carried out on different pump models with the use of CFD packages and pump performance can be predicted [1, 3, 6]. The flow pattern created inside a plunger pump by the motion of plunger is deceptively complex, despite the simple geometry of the pump. The flow must be generated from a stationary fluid. Therefore, the transient flow analysis must be employed to predict this essentially steady motion of the fluid flow against the very high adverse pressure distribution. Research on existing commercial CFD software reveals that FLUENT, CFD-ACE, and ANSYS are most suitable software to be used as a research tool as in [3,6]. Consequently, ANSYS, a leading software package with moving dynamic meshing capability was selected to investigate the flows in plunger pump. Although the complexity of analysis is inherent in all positive displacement pumps, the plunger pump poses an exceptional challenge in numerical modeling that requires a single fluid domain. The fluid domain in the physical model is actually separated into two domains by the rotating crank shaft with plunger which must be in contact with each other all the times. The study and analysis presented in this paper will address those problems associated with establishing an acceptable preliminary investigation on the plunger pump flow by documenting a procedure for the modeling and the results of the numerical analysis. The simulation on 3-D model was performed on a Intel core 2 QUAD processor with DDR 2 - 2 GB RAM.

III. Modeling and Analysis

A. Model Description And Computational Methods

The triplex pump consists of basic components like Crank case, Plunger, Crankshaft, Connecting Rod, and Crankshaft Support Bearings shown in Fig. 1.

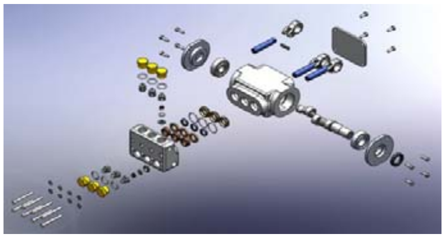

Fig.1. 3D model of the pump

The dimensions selected for this study of the triplex pump [5, 7] as shown in Table I. The input details are 9LPM discharge, 60Bar pressure. Fig. 2 shows the 3-D model and the exploded assembled model of the pump. CFD is a significantly young branch of engineering in which solutions of flow variables are computed by numerical methods [6]. ANSYS is one of the commercial CFD packages for fluid flow modeling in complex geometries, for mesh generation or grids for flow geometry, a way to set up models without hard coding, reasonably robust solvers to get converged numerical solutions and post processing tools to analyze the results as in [1,3]. 3D model for the pump is modeled using solid works and converted the model into igs file, then imported in ICEM CFD. Surface cleanup is done initially and topology is checked.

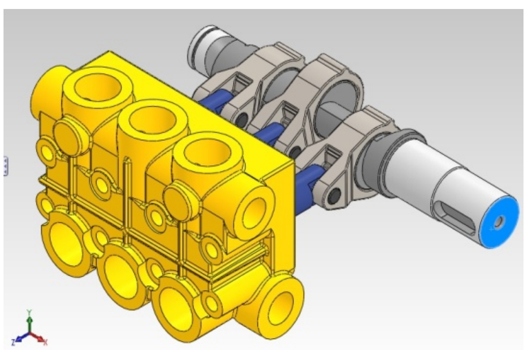

Fig.2. 3D model of the pump

The fluid domain is extracted and the fluid flow path is made air tight for the triplex pump, due to the complexity of analyzing triple cylinder pump, single cylinder domain has been extracted as shown in the Fig. 3 for the performance analysis.

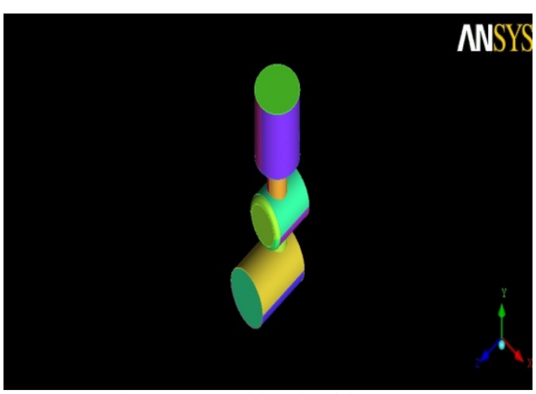

Fig. 3. Extracted domain of the pump

B. Grid generation

The extracted model is subdivided into number of smaller parts and is converted into non dimensional coordinates for grid generation. The pump is divided into three regions, Plunger, cylinder (surface wall) and cylinder head (top cover) [5, 7, 8]. Each region is discretized independently; Triangular surface mesh is generated on the surface wall, tri mesh is created on the plunger and Quad mesh of single layer is developed in the top cover of the plunger for layering of mesh during dynamic meshing using ANSA.

In ANSA, the system automatically chooses the most suitable elements for complex geometry. The fluid volumes were defined by tetrahedral/hybrid elements. Volumetric mesh is created using TGRID to achieve better skewness [6] as shown in the Fig. 4. Grid independency tests were performed on the model. The difference of the outlet flow rate caused by further increasing grid size was below 1% and the grid size was taken for the model to perform all simulations. Skewness value for surface mesh is kept as 0.6 and 0.89 for volumetric mesh as per industrial standards [4, 6].

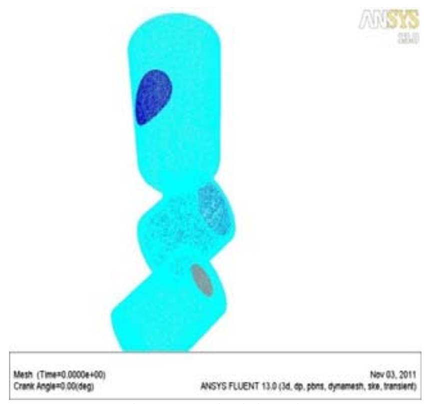

Fig. 4. Meshed domain

C. Dynamic meshing

The operation of the reciprocating plunger pump is similar to IC engines. A single layer of hex mesh is generated below the top of the plunger. Layering concept is used to simulate the plunger movements with the inputs speed, stroke length and degree of crank period given in the table II. The inlet and outlet valves are given user defined functions (UDF) for simulation.

TABLE I: GEOMETRICAL PARAMETERS OF THE PUMP

Value	Description
9.6 lpm	Discharge
1450 rpm	Speed
1hp	Power
60 bar	Pressure
18mm	Bore diameter
15mm	Piston diameter
68.5mm	Connecting rod length

1.5mm	Clearance between TDC & manifold
1.5mm	Radial clearance
12.5mm	Stroke
3	Number of plungers
360^o	Crank rotation

TABLE II: INPUTS FOR DYNAMIC MESHING

1450rpm	Speed
360^o	Crank period
12.5mm	Piston stroke
68.5mm	Connecting rod length

D. Boundary conditions

In the present study, mass flow rate and pressure outlet boundary conditions were used for the inlet and outlet, respectively. Simulation was made with assumptions, Incompressible flow, Non slip boundary condition, Gravity effects are negligible and Fluid properties are not functions of temperature [6]. The boundary conditions used for the flow simulation are summarized in table III

E. Solver

In order to calculate the flow field a commercial CFD code, ANSYS 13 package which has FLUENT solver is used. The governing integral equations for the conservation of mass, momentum and when appropriate, energy and other scalars such as turbulence were solved. FLUENT solves the Navier-Stokes equation using the finite volume method (FVM), which has been applied widely in fluid mechanics and engineering applications. FLUENT 3d, double precession, transient conditions with dynamic meshing is applied to solve the problem.

TABLE III: BOUNDARY CONDITION

Parameters	Simulating conditions
Grid	Structured
Fluid	Water at standard condition
Inlet	Total pressure=101325(Pa)
Outlet	Mass flow rate = Variable (kg/s)
Turbulance model	standard k-ε model
Discretizatiom	Second order

IV. RESULTS AND DISCUSSIONS

In the present study, we intend to investigate the effect of volume flow rate considering the turbulence model as standard k-ε model. Successive iterations were done in the solver to obtain the flow rate and water is selected as working fluid for the investigation. The flow rates at various crank angles such as 180^o, 210^o, 240^o, 270^o, 300^o, 330^o and 360^o are found out as listed in the table IV and maximum flow rate is observed at crank angle of 300^o.

The flow rate is maximum at 300^o because of the dynamic nature imparted on the fluid by the plunger. The static pressure, velocity, turbulence intensity and velocity vector contour plots are drawn at the maximum discharge position and further investigations are carried out.

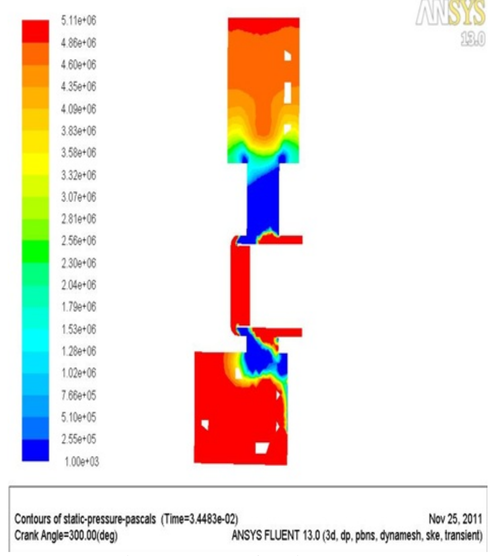

Fig. 4: Contour of Static pressure

The static pressure is found to vary from 5.10e6 Pascal to 1.00e3 Pascal as shown in the Fig. 4. The static pressure is maximum at cylinder top dead centre because the fluid is under highest compression force imparted by the plunger. Similarly the pressure is maximum at inlet chamber because of the closed inlet valve so that the movement of the liquid is arrested.

TABLE 3: FLOW RATES AT VARIOUS CRANK ANGLES

Crank Angle (in degree)	Volume Flow Rate X 10^{-5} (in m^3/s)
180	0.928
210	2.426
240	5.501
270	7.466
300	7.506
330	5.312
360	1.643

Velocity magnitude contours are also investigated as shown in Fig 5. Magnitude of velocity is found to be very high at the gap between plunger and cylinder wall because the liquid is squeezed out of the gap between piston and cylinder top. Velocity is very less at inlet and outlet chambers because the fluid is incompressible and has no movement.

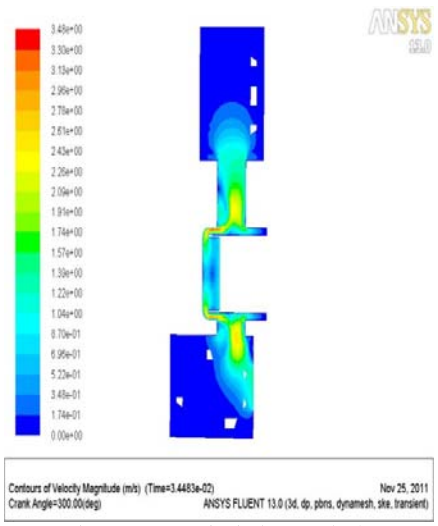

Fig. 5: Contour of velocity magnitude

Turbulence intensity is investigated for to find the region of maximum turbulence as shown in Fig. 6. Turbulence is generally higher in reciprocating pumps because their dynamic nature. Turbulence is high at the corners of cylinder top because at that crank positions liquid has to escape trough that small volume. Generally velocity is also high in that zone. Turbulence is less pronounced in the outer regions of inlet and outlet chamber because of no movement of liquid in those regions.

Velocity vectors are drawn to find the regions of reversed flow which are the main reasons for reduction in pump efficiency. In the current study, reversed flow as shown in Fig. 7 occurs at the exit region from working volume to the delivery valve. Similarly at the bottom corners of the exit chamber also reversed flow occurs which are the major regions of concern.

It is observed from CFD analysis that the flow rate in single cylinder of a triple plunger pump is 4.966×10^{-5} m^3/s (average value). Hence the total discharge of the single pump is three times of flow rate in one cylinder i.e. $3 \times 4.966 \times 10^{-5}$ m^3/s equal to 14.899 x10-5 m^3/s .The corresponding value in liters per minute is 8.9388 lpm. The volumetric efficiency of the pump using CFD is found to be 91% and slip is around 8.98%.

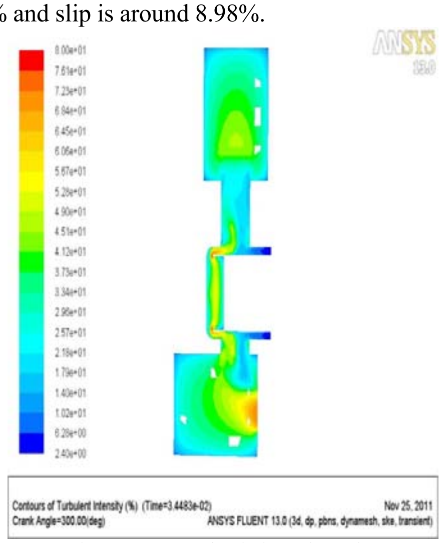

Fig. 6: Contour of turbulence intensity

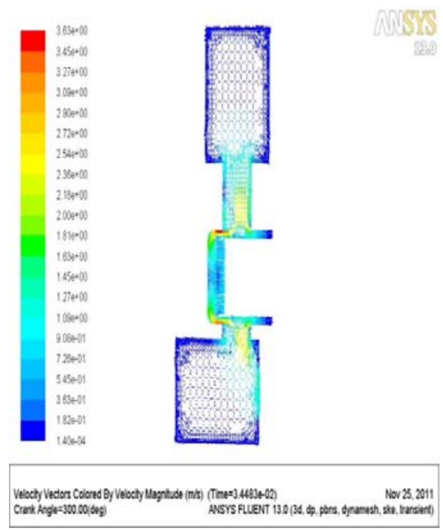

Fig. 7: Contour of velocity vector

Further the pump has been fabricated for the design specification and confirmation test have been conducted on performance of the pump. The actual fabricated pump discharge was found to be 8.1 lpm. Further the pressure, velocity and turbulence contours for better understanding were revealed for better performance of the pump investigated.

V. CONCLUSION

The effect of volume flow rate at various crank angles on the characteristics of triplex pump are clarified experimentally and theoretically. CFD analysis was carried out using ANSYS 13 software to compare the performance of the virtual model with the modified base pump. Pump discharge, slip and volumetric efficiency of the pump are validated in accordance with the pump theory for the developed triple cylinder plunger pump. The theoretical pump discharge was 9.6 lpm while the CFD results gave 8.9 lpm. However the actual discharge of the pump was around 8.1 lpm. To improve the performance of the pump optimization of design parameters is carried out in future.

REFERENCES

[1] Hui-min FAN, Fang-wen HONG, Guo-ping ZHANG, Liang YE and Zhong-min LIU, "Applications of CFD Technique in the Design and Flow analysis of Implantable Axial Flow Blood Pump," Journal of Hydrodynamics, Ser. B, Volume 22, Issue 4, Pp518-525, August 2010

[2] Timothy D. Spegar, Shi-Ing Chang and Sudhakar Das, Eugene Norkin and Robert Lucas, "An Analytical and Experimental Study of a High Pressure Single Piston Pump for Gasoline Direct Injection (GDi) Engine Applications". SAE International, 2009

[3] Sinclair, Rory., Strauss, Tim., Schindler, and Peter., "Code Coupling, A New Approach To Enhance CFD Analysis Of Engines", SAE Paper, 2000-01- 0660, 2000.

[4] Rosli A. Bakar, Semin and Abdul R. Ismail, "Development of Intake and Exhaust Stroke Flow Simulation in an Engine Cylinder Using CFD Model". 2007.

[5] Igor J. Karassik, Joseph P. Messina, Paul Cooper Charles C. Heald, "Pump Handbook," Third Edition, 2000.

[6] John D. Anderson, "Computational Fluid Dynamics the basics with application, " International Edition, 1995.

[7] William K. Chaplis and Frederic W. Buse, "Power Pump Design and Construction", Cader Pump Handbook.

[8] Yunus A.Cengel, John M. Cimbala, Fluid Mechanics, Second reprint 2011.

AUTHORS PROFILE

R. Ragoth Singh is currently pursuing Ph.D degree program from Anna University of Technology, Coimbatore, India. He is currently Assistant Professor in Department of Mechanical Engineering, Kathir College of Engineering, Coimbatore. His research interest is in Design and optimization of fluid flow machines.
E-mail: ragothsingh@rediffmail.com

Dr.M. Nataraj, received PhD from Bharathiyar University, Coimbatore. He is curently Associate Professor of Mechanical Engineering, Government College of Technology, Coimbatore.