Three-Dimensional CFD Analysis of a Hydraulic Gear Pump

Ms. Mamta patil[1], Dr.S.B.Rane[2]
[1, 2] Dept of Mechanical Engineering
[1, 2] DepartmentSardar Patel College of engineering, Mumbai, India

Abstract- *Hydraulic machines are faced with increasingly severe performance requirements. The need to design smaller and more powerful machines rotating at higher speeds in order to provide increasing efficiencies has to face a major limitation CFD analysis is applying for to know flow behavior inside the pump that cannot be captured in physical testing. When applying simulation with the experimental inlet boundary condition and rotation, it has generated profile. This profile was very accurate with measure quantity so, simulation validate accurately for this 2.5D meshing is used. This gives good agreement with experimental result.*

Keywords- CFD, gear pump, volumetric efficiency, 2.5D meshing

I. INTRODUCTION

1.1 Gear pump

The pump is the heart of the hydraulic system. Like a heart in a human body, a hydraulic pump generates a flow by moving the fluid in an environment with an adverse pressure gradient. The pumps are generally categorized in two distinct groups, positive-displacement pumps and kinetic pumps. The kinetic type pumps, like centrifugal pumps, transfer the mechanical power input to kinetic energy and transforms the kinetic energy into static pressure. These pumps are mainly used to generate a high rate of fluid flow with a relatively small pressure rise. Gear rotor pumps provide high volumetric efficiency and smooth pumping action. Further, they work well with a wide range of fluid viscosities. Due to manufacturing tolerances, flow leakages do occur through the tip clearances of the gear teeth. To limit the working pressure, excess fluid is re-circulated to the inlet cavity through a pressure relief valve system. In theory, the pumping action is only a function of the gear rotational speed, and discharge is constant regardless of the operating pressure. In practice, however, leakage through the gaps is formed by the meshing gears which increase with increasing in pressure.

Following are the specifications and operating conditions of the pump.

Type : External gear pump
Pump Displacement : 50 cc/rev
Speed : 500 to 5000 rpm
Pressures : up to 3500 psi
Flow rate : up to 41 gpm @ 3000 rpm
Ambient Temp : -40oC to + 70oC
Dimensions : Inlet port: 38.1 mm
Outlet port : 25.4 mm
Center distance between gears : 50.8 mm
Gear width : 32.9 mm

II. METHODOLOGY

2.1 Fluid used and properties of fluid required are as follows:

Table -1 Properties of fluids

Biodiesel	Density (Kg/m3)	Viscosity (Kg/m-s)
Cotton seed biodiesel (c-10)	0.830	38.84
Green solvent (d-limonene)	0.834	8.3
Isovg 32	0.857	3.2

2.2 Modeling the gear pump:

Creating Geometry Using Ansys Design Modeler:For the geometry of fluid flow analysis, create geometry in ANSYS Design Modeler, or import the appropriate geometry file. If The geometry in ANSYS Design Modeler is not created then import itfrom pre-existing geometry byright-clicking the Geometry cell and selecting the Import Geometry option from the context menu. From there,browsers file system to locate the geometry.iges geometry file.In this project the fluid geometry model had been created in ANSYS Design Modeler only.The fluid geometry of the Gear Pump is being created as an assembly made up of three parts with a thickness of 10 mm each.

The various fluid parts are as follows:

1) Inlet fluid Volume
2) Fluid Volume surrounding both the Gears
3) Outlet Fluid Volume.

This above 3 fluid volumes is being created separately so that can give rotary motion to the boundaries of Gear1 and Gear2 -one in clockwise and another one in anticlockwise direction respectively.

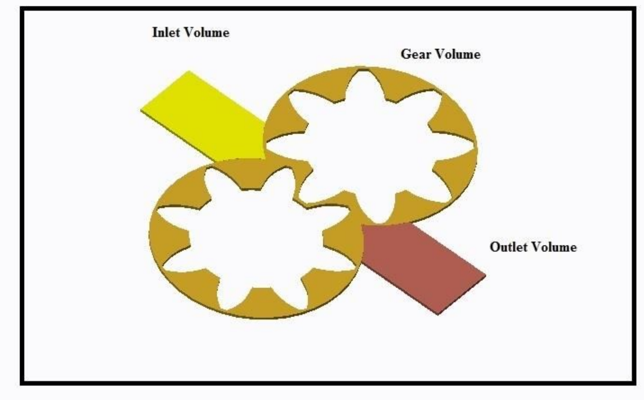


Figure 1 Shows modeling assembly of Inlet Volume, Gear VolumeAnd Outlet Volume

2.3 Meshing Using 2.5D Model

For 3D simulations only, select the 2.5D model under Models in the Dynamic Mesh Parameters panel. This model allows for a specific subset of smoothing and remeshing techniques.

The 2.5D mesh essentially is a 2D triangular mesh which is expanded, or extruded, along the normal axis of the specific dynamic zone that you are interested in modeling. The triangular surface mesh is remeshed and smoothed on one side, and the changes are then extruded to the opposite side. Rigid body motion is applied to the moving face zones, while the triangular extrusion surface is assigned to a deforming zone with remeshing and smoothing enabled. The opposite side of the triangular mesh is assigned to be a deforming zone as well, with only smoothing enabled.

Note that in the Smoothing tab of the Dynamic Mesh Parameters panel the 2.5D model allows to change only the Boundary Node Relaxation value and the Number of Iterations. Also note that the Remeshing tab of the Dynamic Mesh Parameters panel automatically has Face Remeshing enabled.

The 2.5D model only applies to mapable (i.e., extrudable) mesh geometries such as pumps, as in Figure. Only the aspects of the geometry that represent the "moving parts" need to be extruded in the mesh.

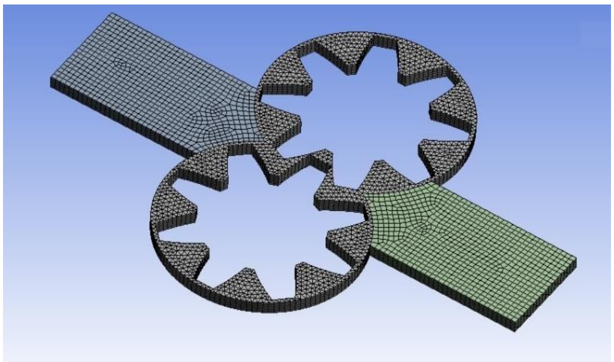


Figure 2shows -2.5 D Extruded Gear Pump Geometry

It must only apply smoothing to the opposite side of the extruded mesh, since **FLUENT** requires the geometry information for the dynamic zone. **FLUENT** projects the nodes back to its geometry after the extrusion. Without this geometry information, the dynamic zone tends to lose its integrity.

In parallel, a partition method that partitions perpendicular to the extrusion surface should be used. For example, if the normal of the extrusion surface points in the x-direction then Cartesian-Y or Cartesian-Z would be the perfect partition methods.

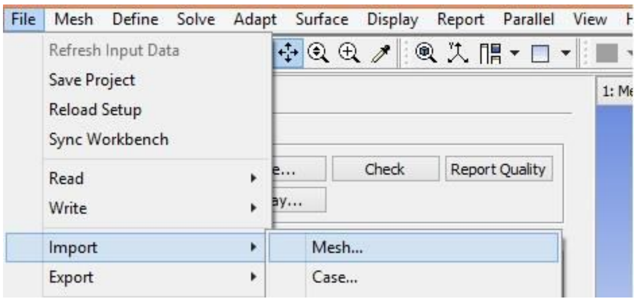
The 2.5D model is used in combination with a DEFINE_GRID_MOTION UDF (User Defined Feature). This UDF is associated with the extrusion surface that is adjacent to the cell zone, in turn applying the same deformation to the entire cell zone. This approach is particularly useful when modeling gear pumps that are predominantly extruded hexahedral meshes

2.3 NAMED SELECTION:

In order to give the boundary condition named selection has to be given in the meshing stage itself. The same named selection are being seen in ANSYS Fluent. The various names given to the various sides of the geometry are inlet, outlet, symmetry1_gear, symmetry2_gear, symmetry1_inlet, symmetry2_inlet, symmetry2_outlet, boundarywall_outlet, boundarywall_inlet, Gear1 and Gear2.

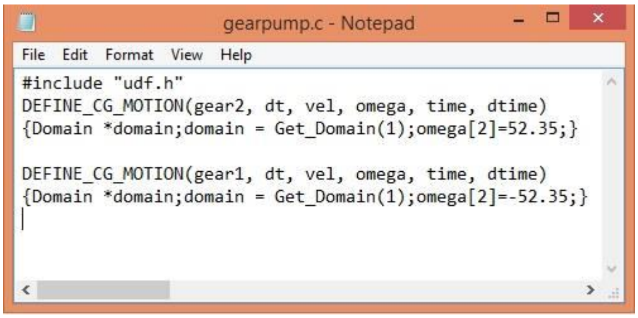
2.4 PERFORMING THE ANALYSIS IN ANSYS FLUENT:

a) The first step after getting into fluent is to get the model which has been already meshed and named selection done on it. Go to file Import Mesh.



Browse the required file FFF.set file and say open, the meshed file will get loaded with the named selection to the various boundaries.

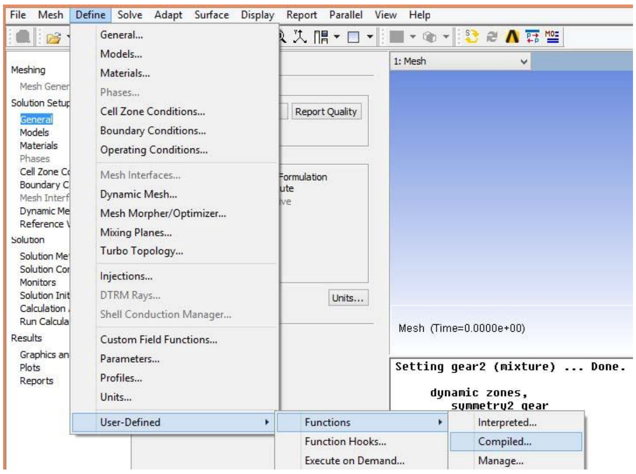
b) Creating the UDF (User Defined Feature) and Hooking the same: A UDF is a program written in C to control the moment of Dynamic Mesh. Here in this case of making two gears to move in opposite direction the program is given below:



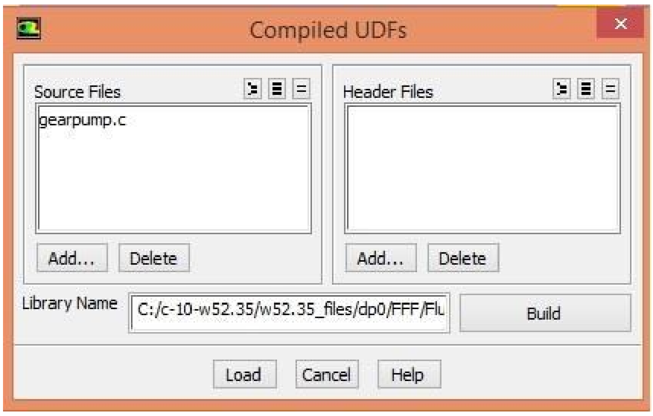
```
#include "udf.h"
DEFINE_CG_MOTION(gear2, dt, vel, omega, time, dtime)
{Domain *domain;domain = Get_Domain(1);omega[2]=52.35;}

DEFINE_CG_MOTION(gear1, dt, vel, omega, time, dtime)
{Domain *domain;domain = Get_Domain(1);omega[2]=-52.35;}
```


The above program need to be compiled and the hooked in ANSYS Fluent.



Browse the gearpump.c file by pressing the Add tab and Build the same.



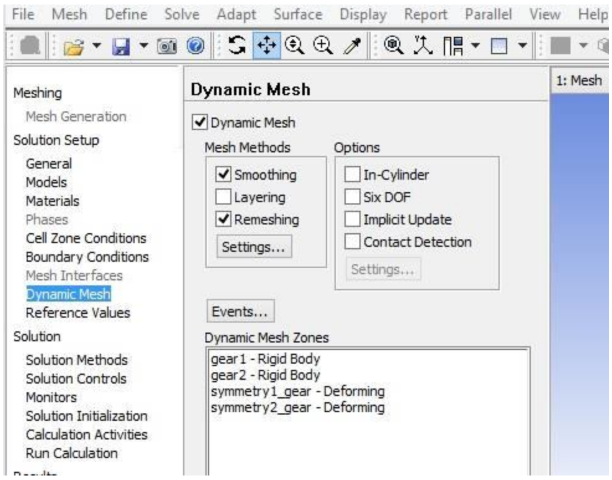
c) In the solver type select the pressure based, absolute (velocity Formulation) and Transient (Time) options.

d) Go for viscous flow with Realizable K-epsilon and Standard wall function.

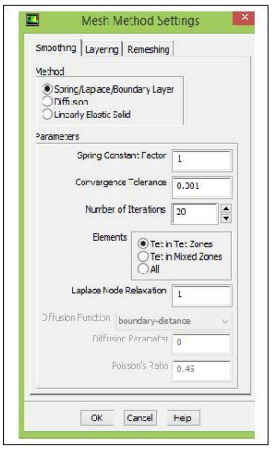
e) Create the material / Fluid (in this case Green Solvent)

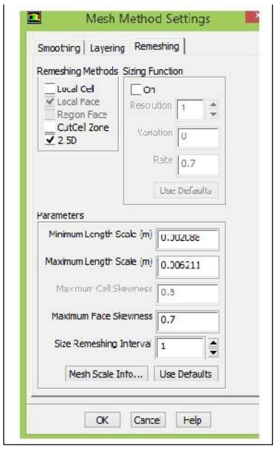
f) Define the inlet boundary condition –as pressure inlet with a Gauge pressure of 101325 Pascal's

g) Dynamic mesh Setting as given below:



Press the Setting Tab

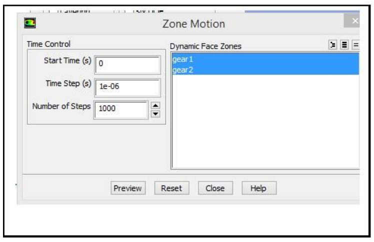


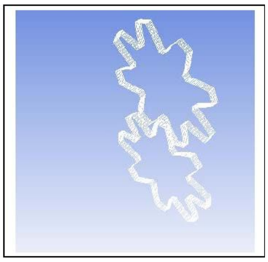


Press the Remeshing Tab Select the Meshing Type to be 2.5D and give the remaining details.

h) Linking the Dynamic Mesh Zones to the Named selections:

a) Assign Gear1 to be Rigid Body and link it with Motion UDF as Gear1:libudf and fill the other required details.
b) Similarly assign Gear2 to be Rigid Body and link it with Motion UDF as Gear2:libudf and fill the other required details.
c) Assign Symmetry1_gear & Symmetry2_gear as Deforming type and fill the Geometry definition and M eshing Options respectively.
d) Click on Display zone motion.





Wherein we can verify the motion of both the gear boundaries. i) Set the required monitor'si.e.mass flow rate at the outlet.

a) Go for standard solution initialization with keeping the gauge pressure as 101325 Pascal and Turbulent KineticEnergy as 0.1 m2/s2.
b) Run the calculation keeping the Time step Size 1e-6 and Number of Time step as 3000.

III. RESULTS

Following observation table is obtained after the simulation of gear pump in fluent package of ANSYS. For the analysiswe have constrained the output pressure to1bar respectively. And speed of gear pump changing from 500 RPM to 5000RPM. Also flow is calculated analytically and output volume flow taken after the simulation.

3.1 Theoretical & Ansys results for C-10

Table 2 mass flow rate & discharge pressure for C-10

Sr.No.	RPM	Mass flow rate (Kg/s) by Ansys	Mass flow rate(Kg/s) by theoretical	Pressure (Pascal)
1.	500	0.2090	0.1693	0.05480
2.	1000	0.3751	0.3387	0.17318
3.	1500	0.5329	0.5081	0.34892
4.	2000	0.7005	0.6775	0.60446
5.	2500	0.8915	0.8468	0.97740
6.	3000	1.08815	1.0163	1.4544
7.	3500	1.2039	1.1856	1.8042
8.	4000	1.3852	1.3550	2.3615
9.	4500	1.6576	1.5243	3.2039
10.	5000	1.7366	1.6937	3.7052

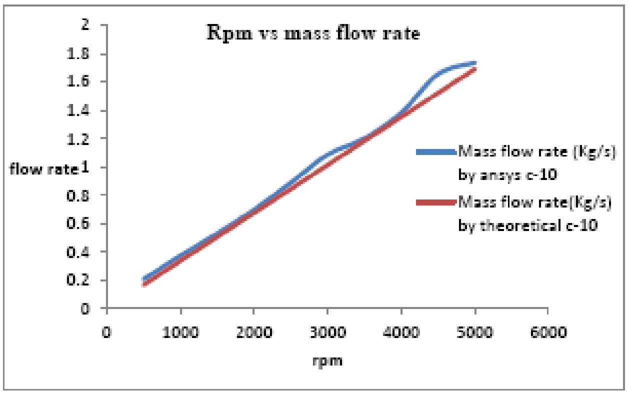


Fig 4 Comparison of mass flow rate between theoretical & ansys for C-10

Fig 4 shows comparative study between actual and simulated mass flow rate for C-10 .it shows flow increases withincreases in speed of pump.

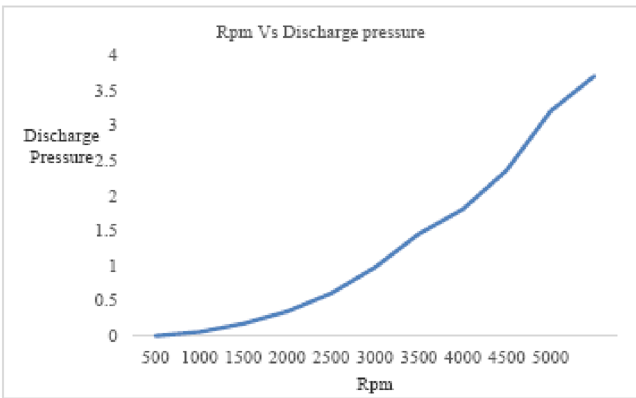


Fig 5 Relation between rpm Vs discharge pressure by ansys

Fig 5 shows relation between discharge pressure and pump speed for c-10.it shows discharge pressure increases withincreases in speed of pump

3.2 Theoretical & Ansys results for Green solvent

Table 3 mass flow rate & discharge pressure for Green solvent Green solvent

Sr.No.	RPM	Mass flow rate (kg/s) by ansys	Mass flow rate(kg/s) by theoretical	Pressure(Pascal)
1.	500	0.2100	0.1707	0.03529
2.	1000	0.3769	0.3403	0.18022
3.	1500	0.5349	0.5106	0.3857
4.	2000	0.7039	0.6807	0.6481
5.	2500	0.8957	0.8509	1.0155
6.	3000	1.0933	1.0212	1.5543
7.	3500	1.2097	1.1913	1.9935
8.	4000	1.3919	1.3615	2.4339
9.	4500	1.6656	1.5317	3.3039
10.	5000	1.7450	1.7013	3.8072

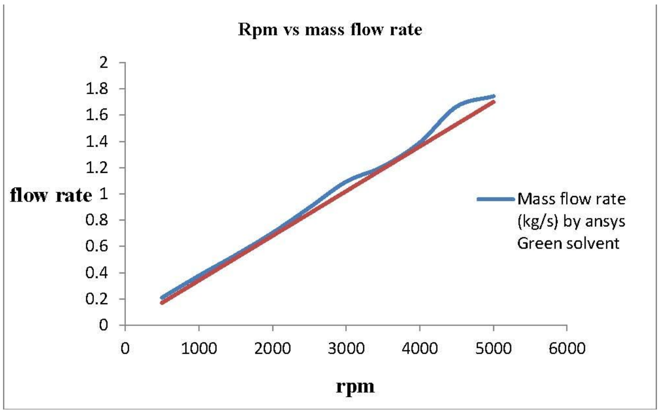


Fig 6 Comparison of mass flow rate between theoretical & simulated for Green solvent

Fig 6 shows comparative study between actual and simulated mass flow rate for Green solvent .it shows flow increases with increases in speed of pump.

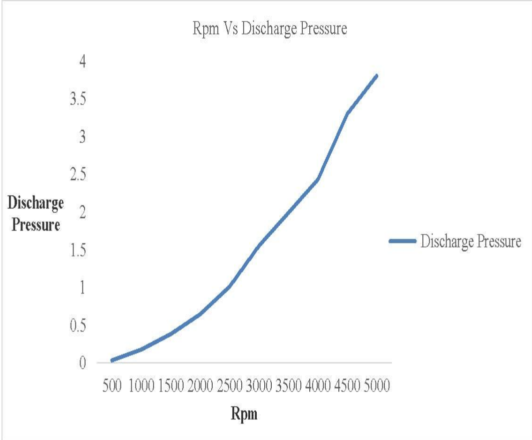


Fig 7 Relation between rpm Vs discharge pressure by ansys for green solvent

Fig 7 shows relation between discharge pressure and pump speed .it shows discharge pressure increases with increases in speed of pump. The pressure contours are also obtained from the simulation providing details of pressure variation in the fluid domain of the gear pump. The pressure contours are captured at every time step to observe pressure variation in the fluid zone with respect to time at different speeds for different fluids this contours shows that discharge pressure increases with increases in speeds.

For Green solvent

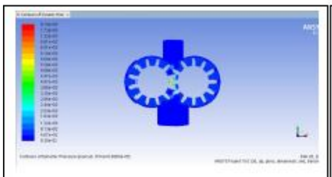


Figure 8 – Pressure Contour Plot showing Pressure variation on different speed (500 rpm)

Figure 9 -Pressure Contour Plot showing pressure variation on different speed (1000 rpm)

For C-10

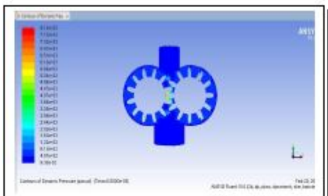


Figure 10 – Pressure Contour Plot showing Pressure variation on different speed (500 rpm)

Figure 11 – Pressure Contour Plot showing pressure variation on different speed (1000) rpm

3.3 Comparative study between C-10, Green solvent & IsoVg32

Table 4 volume flow rate & discharge pressure for Green solvent Fig.12 comparison of rpm Vs mass flow rate between three fluids C-10, Green solvent & Isovg32 by ansys

rpm	Mass flow rate for c-10	Mass flow rate for green solvent	Mass flow rate for isovg 32
900	0.3751	0.3152	0.607
1500	0.5329	0.5349	1.022
3000	1.088	1.0933	2.078

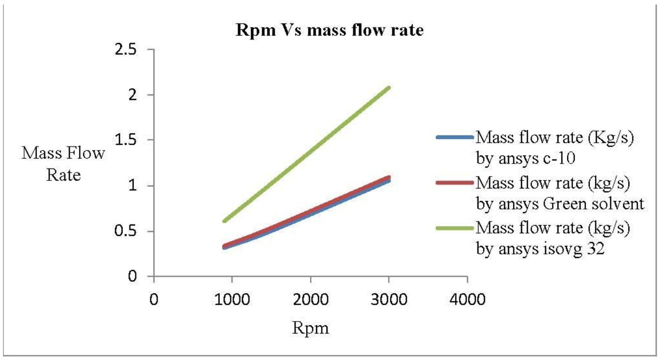


Fig 12 shows comparative study of rpm vs. mass flow rate between three different fluids i.e. C-10, Green Solvent & Isovg 32 by ansys. It shows flow rate increases with increases in speed of pump. Due to similarity in fluid properties nature of graph of c-10 and green solvent is same as that of isovg 32 fluid.

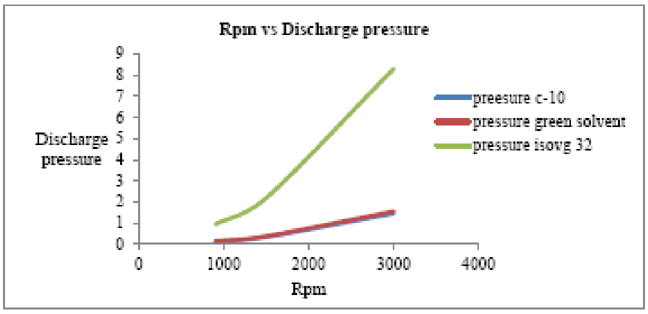


Fig.13 comparison of Rpm vs. discharge pressure between two fluid C-10, Green solvent & isovg 32

Fig 13 shows comparative study of rpm vs. discharge pressure between three different fluids i.e. C-10, Green Solvent &Isovg 32 by ansys. It shows discharge pressure increases with increases in speed of pump. Due to similarity in fluidproperties nature of graph of c-10 and green solvent is same as that of isovg 32 fluid.

3.4 Experimental & ansys results for Isovg 32

Table 5 volume flow rate & discharge pressure for Green solvent

rpm	Volumetric flow rate experimental (lpm)	Volumetric flow rate ansys(lpm)	Discharge pressure pascal
900	42.50	44.952	0.9442
1500	71.60	68.418	2.1903
3000	145.5	133.035	8.2951

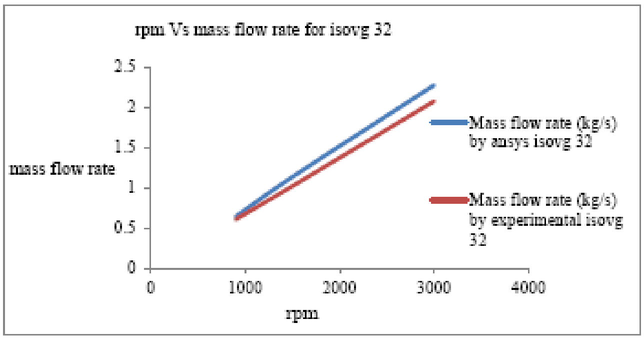


Fig.14 comparison of Rpm Vs mass flow rate by experimental &ansys

Fig 14 shows comparative study between experimental and ansys mass flow rate for isovg32 .it shows flow increaseswith increases in speed of pump.

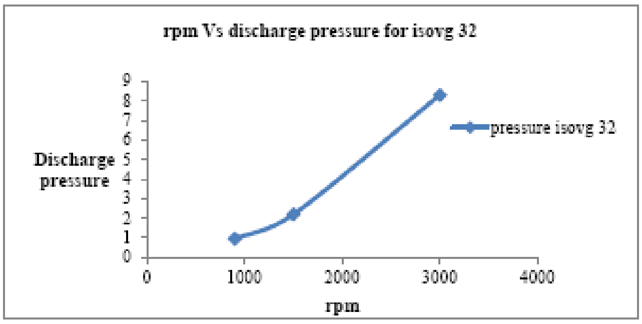


Fig.15 comparison of Rpm Vs discharge pressure for isovg 32

Fig 15 shows relation between discharge pressure and pump speed .it shows discharge pressure increases with increasesin speed of pump.

Figure 16 – Pressure Contour Plot showing Pressure variation on different speed (900 rpm)

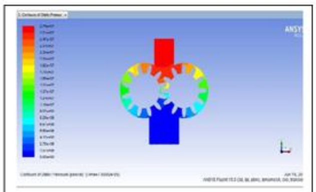


Figure 17 – Pressure Contour Plot showing Pressure variation on different speed (1500 rpm)

Figure 18 – Pressure Contour Plot showing Pressure variation on different speed (3000 rpm)

The pressure contours are also obtained from the simulation providing details of pressure variation in the fluid domain of the gear pump. The pressure contours are captured at every time step to observe pressure variation in the fluid zone with respect to time at different speeds for different fluids this contours shows that discharge pressure increases with increases in speeds.

3.5 Discussions:

It is clearly seen from the above two; the results obtained from manual calculation for the discharge of two different oils matches with the software results obtained from ansys fluent with 5% difference, which is well accepted.

This method can be used by gear Pump manufacturing companies while designing gear pumps for a specific fluid having specific density & viscosity, thereby predicting its volume flow rate at a specific rpm.& giving a new design of gear pump.

IV. CONCLUSION

This study focused only on the Comparative Study of oil flow in the suction pipe to delivery pipe with help of three different fluids to investigate performance of gear pump such as efficiency, flow rate, discharge pressure using CFD analysis. The experimental results given in pump manual are verified with the simulation results as seen both fall in place. This project work derives a method of carrying out computer simulation for designing various gear pumps required to discharge a given volumetric flow at given pressure. This work also explains the pattern of how the volumetric flow increases with increase in gear rpm as it uses an udf (user defined feature) which acts as a control for increasing & decreasing the speed of gear. This methodology saves the cost of manufacturing many prototypes as the complete simulation is done on computer, before actually manufacturing the gear pump. Hence a gear pump manufacturing company can easily adopt this methodology & complete its design work & give various options to client before actually manufacturing the pump saving lot of money, material &manpower. Various designs can be verified by changing the rpm, pcd of gear; height of gear etc. & an optimized design can be concluded.

V. FUTURE SCOPE

Future research will be extended to investigate the possibilities of cavitation, noise generation, and air entrainment in the suction line, inlet ports, and chambers, and their effects on the pump performance. It is imperative that the amount of turbulence and entrained air is kept to a minimum. Entrained air can cause a reduced efficiency as well as vibration, noise, and/or accelerated corrosion. The challenge was to ensure that as much oil as possible would be fed evenly into the pump, even when operating at high speeds. By performing a series of CFD simulations, the design of the intake channel was optimized.

REFERENCES

[1] James Sullivan, “FluidPower, Theory andApplications, 4thEd.”,Prentice-Hall, 1998.

[2] Haworth,D.C.,Maguire,J.M.,Matthes, W .R.,Rhein,R.,El Tahry,S.H., “DynamicFluidFlow Analysis of Oil Pumps, ” S AEPaper 960422, 1996.

[3] SureshPatil, “Numerical Simulation of Multi-Dimensional Flowsina GearPump”, Youngstown State University, 2006.

[4] Fluent, Inc., “Gear PumpSolution, Tutorial 4”,Fluent Inc., USA

[5] Fluent, Inc., “User’s Manual”,Fluent Inc., USA.

[6] Hyun Kim, Hazel Marie, Suresh Patil, “TWO-DIMENSIONAL CFD ANALYSIS OF A HYDRAULIC GEAR PUMP”AC 2007-821.

[7] Hart DP (1998) The Elimination of Correlation Errors in PIV Processing, 9th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon.Hart DP (1999) Super-Resolution PIV by Recursive Local-Correlation, Journal of Visualization (10)

[8] Iyoi H and Ishimura S (1983) χ-Theory in gear geometry, Transaction of ASME Journal of Mechanisms, Transmissions, and Automation in Design 105, pp 286–290.

[9] Moore J (2007) Dry sump pump bubble elimination for hydraulic hybrid vehicle systems, Master thesis in the department of Mechanical engineering, The University of Michigan.

[10] Nogueria J, Lecuona A, and Rodriguez PA, (1997) Data validation, false vectors correction and derived magnitudes calculations on PIV data, Meas. Sci. Technol. (8), 1493-501.

[11] Frosina, E.; Senatore, A.; Buono, D.; Stelson, K.A.; Wang, F.; Mohanty, B.; Gust, M.J. Vane pump power split transmission: Three dimensional computational fluid dynamic modeling. In Proceedings of the ASME/BATH 2015 Symposium on Fluid Power and Motion Control, Chicago, IL, USA, 12–14 October 2015.

[12] Frosina, E.; Buono, D.; Senatore, A.; Stelson, K.A. A modeling approach to study the fluid dynamic forces acting on the spool of a flow control valve. J. Fluids Eng. 2016, 139, 011103. [CrossRef]

[13] Pellegri, M.; Vacca, A.; Frosina, E.; Senatore, A.; Buono, D. Numerical analysis and experimental validation of gerotor pumps: A comparison between a lumped parameter and a computational fluid dynamics-based approach. Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci. 2016, 1989–1996, 203–210. [CrossRef]

[14] Schleihs, C.; Viennet, E.; Deeken, M.; Ding, H.; Xia, X.; Lowry, S.; Murrenhoff, H. 3D-CFD simulation of an axial piston displacement unit. In Proceedings of the 9th International Fluid Power Conference, Aachen, Germany, 24–26 March 2014; pp. 332–343.

[15] Vacca, A.; Franzoni, G.; Casoli, P. On the analysis of experimental data for external gear machines and their comparison with simulation results. In Proceedings of the IMECE2007 ASME International Mechanical Engineering Congress and Exposition, Design, Analysis, Control and Diagnosis of Fluid Power Systems, Seattle, WA, USA, 11–15 November 2007.