Study of Performance Analysis of Reciprocating Pumps using CFD

[1]Suhas D. Thorat, [2]A. B. Kanase-Patil

[1,2]Department of Mechanical Engineering, SCOE, Pune 411041, India
Email: [1]suhas31190@gmail.com, [2]abkanase.scoe@gmail.com

Abstract- Reciprocating pump is positive displacement pump. The energy is periodically added to generate flow. It is used in application where low discharge is required at high pressure. The analytical process for designing the pump and flow prediction is very difficult, time consuming and costly. Computational fluid Dynamic (CFD) analysis is a best tool for analysing the flow patterns inside the reciprocating pump and predicting the behaviour of the pump under different operating conditions. In this seminar, the CFD analysis of the effect of grooves cut along the length of piston of the axial piston pump is to be studied. The effect of number of grooves, their position and their orientation on the outlet delivery is analysed. Also collision characteristics of reciprocating pump is to be studied. CFD analysis of plunger pump is to be investigated. The case study for the CFD analysis of 2 D geometry of simplex pump has been performed. The result shows contours of static pressure, velocity magnitude and density

Keywords: Computational fluid Dynamic, positive displacement, Reciprocating pump

NOMENCLATURE

θ	valve disc cone angle
H	valve disc rises to certain height
V	velocity
CFD	Computation fluid dynamics

I. INTRODUCTION

Pump transports fluid from one place to the other. There are many types of pumps available in the market like centrifugal pumps, gear pump, reciprocating pump etc. The selection of pump depends on many conditions like operating pressure, overall efficiency, discharge, cost effectiveness etc. Reciprocating pump is one of the most important types of pump used in many industrial applications and also in some of the domestic applications. In reciprocating pumps, the fluid enters the pump body through the inlet port provided and gets compressed by the application of the piston force to particular desired pressure. The outlet port delivers the compressed fluid to the required height or place. Designing the pump and also visualisation of inside fluid of the pump is very difficult as well as time consuming and costly. Computational Fluid Dynamic (CFD) analysis is a best tool available now a days for analysing the flow patterns inside the reciprocating pump and predicting the behaviour of the pump under different operating conditions. CFD also helps to optimise the design parameters of the pump by giving nearly correct flow patterns along with more effective working of the pump.

II. LITERATURE REVIEW

Tsui and Lu [1] evaluated the performance of a valvelessmicropump by CFD. They also carried out lumped system analysis. In multidimensional simulation of CFD model, Navier-strokes equation has been solved by using finite volume method. They only considered half part of pump for calculation due to symmetrical geometry. In lumped system, pumping process was divided into three mode such as pump mode, supply mode and transition mode. In pump mode, the fluid emerges from the chamber due to the high chamber pressure. In supply mode, the fluid is forced to flow into the chamber. In transition mode, the fluid outside the outlet flows into the chamber and the fluid inside the chamber flows out through the intake element. The plots of streamlines and pressure contours in nozzle/diffuser regions have been studied. The result indicate that at 2000 Pa back pressure simulation, pump mode and supply mode efficiency was 4%, 9% and 15% respectively. Kumar and Bergada [2] studied the effect of piston grooves performance in an axial piston pumps via CFD analysis. In this study, Reynolds equation of lubrication in Cartesian coordinate has been applied to the piston cylinder clearance. This equation also considers the piston eccentricity and the relative tangential movement between piston and barrel. Different grooves configurations have been studied. The result indicates that grooves placed towards the inner edge of the piston produces 1.5% higher force on piston surface with respect to the non-grooved piston. Fan et al [3] studied computational fluid dynamic analysis and design optimization of jet pumps. The compressible flow within the jet pump has been modelled using Navier-Stokes equation. In this study, the standard $k - \varepsilon$ turbulence model has been used. The comparison between analytical and CFD prediction of Mach number have been studied. The CFD prediction of pump efficiency for original and optimized designs at a range of entrained flow rates ranging from 0 L/min to 200 L/min has been studied. The result indicates that the pump efficiency increases from 29 % to 33 %. The energy requirement of the pump is reduced by 20%.

Alves et al [4] studied analytical and CFD modelling of the fluid flow in an eccentric tube centrifugal oil pump for hermetic compressors. In CFD modelling, the fluid flow in eccentric tube oil pumping system has been solved numerically using FLUENT package. The analytical model for the pick-up tube and shaft channel were developed independently and were coupled via a numerical procedure to determine the steady state volume flow rate assembly. The steady state results have been verified against a CFD model. The contours of volume oil fraction as a function of time has been studied. The result indicates that the maximum flow rate was observed for an inclination angle of 68°. Luckmann et al [5] studied the analysis of oil pumping in a reciprocating pump. In this study, CFD analysis of the oil pumping system of a reciprocating compressor has been done. The commercial CFD package Fluent has been used and the two-phase free surface gas-liquid flow has been resolved through the volume of fluid method. The calculated oil flow rate has been compared with an experimental datum. Results show in terms of phase fraction contours representing the volumetric flow rate of the oil as a function of time. It is observed that the climbing time is of the order of 0.6 s. Kumar et al. [6] studied analysis of axial piston pump grooved slipper by CFD simulation. In this study, static and dynamic characteristics of a piston pump slipper with groove were investigated. Three dimensional Navier Stokes equations in cylindrical coordinates have been applied to the slipper gap, including the groove. Fluid momentum interchange inside the groove has been studied. Vorticity inside the groove has been analysed under several working conditions. A test rig has been built to compare experimental and CFD results. The result indicates that the slipper leakage at 10 MPa inlet pressure for 17 microns true clearance is 0.10 l/min and 17 microns CFD clearance is 0.082 l/min. Pei et al. [7] studied collision characteristics for reciprocating pump using FEA and experimental. In this study, the valve disc motion parameters have been measured directly by displacement and acceleration sensor, where both are mounted on the valve disc. The test data has been used as valve motion parameters to simulate the collision process by ANSYS/LS-DYNA software. Valve disc vibration displacement and velocity signal has been studied for different strokes. The simulation and experimental results have been used for studying pump valve failure mechanism, optimization of design and then improving pump services life. Approximation theory has 2.12% less displacement of valve disc as compare to U.Adolph theory for stroke of 200(times/min). Kumar et al. [8] studied optimization of connecting rod parameter using CAE tools. The model has been developed in Pro/E wildfire 5.0 and then imported in ANASYS workbench. In this study, only static FEA of connecting rod has been performed. During analysis two cases have been analyzed one with load applied at the crank end and restrained at the piston end, and the other with load applied at the piston end and restrained at the crank end. The analysis has been carried out under the axial and buckling loads. The result indicated that the weight of the connecting rod was reduced by 0.004 Kg. The percentage reduction of weight was 3.05 %.Parkash et al. [9] studied optimization of design of connecting rod under static and fatigue loading. In this study, the mathematical model of connecting rod has been developed. The model of connecting rod was developed in CATIA. During analysis two cases analyzed, one with load applied at the crank end and restrained at the piston end, and the other with load applied at piston end and restrained at the crank end. The comparisons for static and fatigue loading have been studied under same boundary condition. The result indicated that the weight of connecting rod was reduced by 0.005 Kg. The percentage reduction of weight was 0.23 %.Singh and Nataraj [10] studied the performance of plunger pump at various crank angles using CFD. The triplex plunger pump has been used for analysis. Due to the complexity of analyzing triple cylinder pump, single cylinder domain has been extracted. The extracted model has been subdivided into number of smaller parts and converted it into non dimensional coordinates for grid generation. In his investigation the effect of volume flow rate considering the turbulence model as standard K- ε model. The maximum flow rate was observed at crank angle of 300°. The volumetric efficiency of the pump using CFD was found to be 91% and slip 8.98%. Samad and Nizamuddin [11] studied flow analysis inside jet pump used for oil wells. In this study, Reynold-averaged Navier Stokes (RANS) equation has been solved. The turbulence k-ε model has been used for simulation. The numerical study has been performed for different primary and secondary fluids. The effect of the area ratio and the throat length on the performance of the jet pump has been studied. The result indicated that the efficiency of the jet pump increased up to 20% to 30%. Hubacher and Groll [12] studied crankshaft bearing analysis of a single stage, Semi-Hermetic Carbon Dioxide compressor. In this study, the friction loss analysis has been conducted, which was used previously obtained compressed performance data as inputs. In the first step, the force loads acting on the two crankshaft bearing have been predicted based on the operating conditions. In the next step, the predicted force loads have been used in the bearing loss analysis. The result indicated that the frictional losses of the two crankshaft bearings contribute with approximately 19 to 43% to the total frictional losses. Deng et al. [13] carried out analysis of oil discharge rate in rotary compressor by using CFD. The lubricant oil plays an important role in a hermetic compressor. It provides lubrication to the part and fills in the sealed parts to reduce gas leakage. Also it removes heat generated from mechanical friction of sliding parts. The flow field of rotary compressor has been calculated by Volume of fluid method. The turbulent K- Epsilon model has been used. Oil pumping height for different scheme has been studied. The result indicates that the maximum oil circulation ratio is 2.85%. Oil circulation ratio is used to denote oil discharge rate of rotary compressor.Diany and Bouzid [14] evaluated the stresses and displacement of stuffing box packing based on a flexibility analysis. An analytical model based on thick cylinder theory has been developed to model stem-packing-housing interaction. The model gave reliable contact pressure distributions at the interfaces and predicted the deformation involved. The model suggested that the contact pressure ratio was close to one. The interface contact pressure depends on several parameters including geometry, material and friction. The model has potential in predicting contact stress as a function of geometry, material and friction coefficient. The result indicated that for the same axial stress the friction coefficient was less than the 10%.

III. EFFECT OF PISTON GROOVES ON THE PERFORMANCE OF AXIAL PISTON PUMPS

In axial piston reciprocating pumps, some manufacturer uses pistons having grooves cut on along the length. Grooves are meant to stabilize the piston. The number of grooves needed for special application and where it is to be located is an ultimately design consideration. Figure 1 shows the piston with grooves cut along its length. This piston is considered for the CFD analysis. The grooves cut along the length by grooving operation.

Figure 1: Grooved piston [2].

Initially, by considering all the governing equations, a mathematical model of grooved axial piston pump is prepared. After preparation of mathematical model, the solution technique is defined. The solution technique involves solution of various numerical equations that are required to be solved for the CFD analysis. After defining the methodology for solving the equations, the simulation via CFD is done [2].

A. CFD Analysis for Different Grooved Configurations

Kumar and Bergada [2] represented paper to understand the effect of number of grooves and their position, three different types of pistons are studied with different groove configurations. Figure 2 shows the pressure distribution inside the cylinder of the reciprocating pump for the forward stroke. The variation in the pressure for the complete forward stroke is observed from figure 2. The pressure peak appearing in the figure produces a negative y-directional torque. This torque is trying to restore the piston eccentric displacement. Which arises from the differences in the friction ball-cup and piston cylinder pairs.

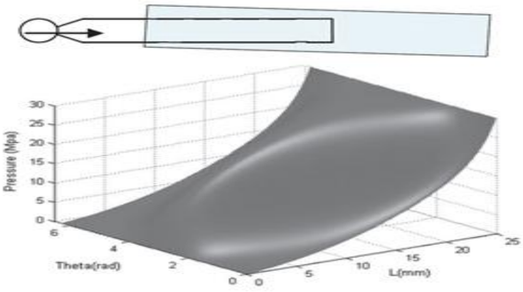

Figure 2: Pressure Distribution for Non-Grooved Piston [2].

Figure 3 shows the pressure distribution for piston with grooves on both sides. From figure it is cleared that two separate pressure waves are seen. This is almost at stability level.

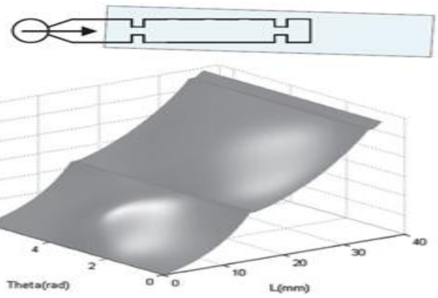

Figure 3: Pressure distribution for piston with grooves on both sides [2].

Figure 4 shows the pressure distribution for piston with one outer groove. Grooves located towards the edge of the piston for more stability. It also gives more stable pressure distribution.

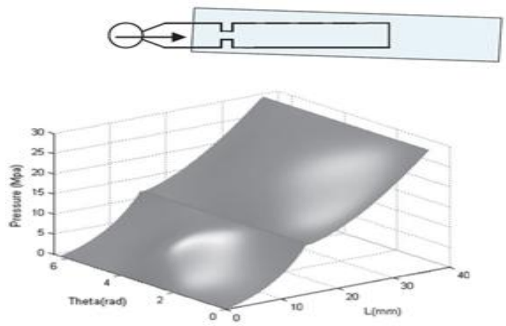

Figure 4: Pressure distribution for piston with one outer groove [2].

IV. COLLISION CONTACT CHARACTERISTICS FOR RECIPROCATING PUMP

During the closing of valve, the valve disc suddenly stops moving. The kinetic energy of the valve disc is instantaneously converted into impact energy. An impact is produced due to the dynamic contact motion between valve disc and its seat. The kinetic energy of disc is large. Due to this contact surface of the valve disc and seat were out quickly. Hence, it leads to an increase in pump valve leakage.

A. Collision Description and Valve Parameters Measuring

A Dynamic model of pump valve is as shown in Figure 5. The valve seat is mounted on pump body of reciprocating pump. θis the valve disc cone angle. Consider the example of discharge valve. The valve disc rises to certain height (h) under the thrust of

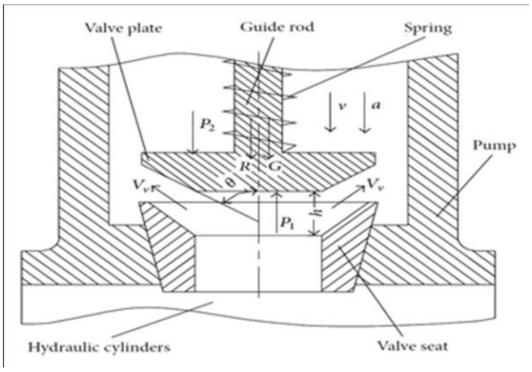

Figure 5: Dynamic model of the pump valve [7].

liquid during discharge process and discharge valve is open. When the discharging process is completed,

the suction process starts. The discharge valve disc drops down from a certain height (h) to its seat with a certain velocity (v). This makes a collision between the discharge valve disc and its seat [7].

Approximation theory [7] and U.Adolph theory [7] are most widely used theories on pump valves motion calculation. The approximation theory assumes that the valve disc is mass less. The spring force (R) is keeps constant. In the inelastic parts of pump incompressible fluid flows. Hydraulic cylinders are fully charged with continuous flow during the valve discs movement. In U.Adolph theory, the valve disc inertia force, disc mass, spring force and the changes of the flow velocity in the valve gap is taken into account. The maximum displacement and closing speed between valve disc and seat during the operation are measured directly by the displacement and an acceleration sensor. Sensors are mounted on the valve disc [7].

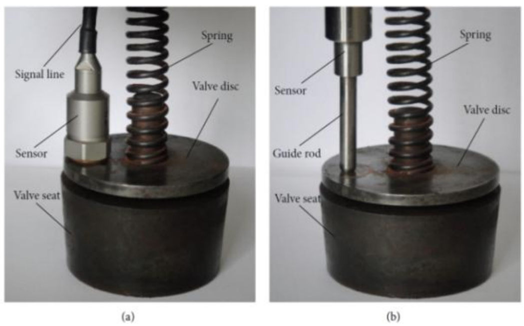

Figure 6: Sensors (a) Acceleration and (b) Displacement [7].

B. Collision Model and Numerical Solution:

ANSYS/LS-DYNA is the most commonly used software in analyzing the collision problems. The data of maximum distance and closing speed between valve disc and seat is obtained through repeated test. The data is used as the initial condition values needed in the software calculation. 3D model of pump has been developed by using Pro/E. Then the model is imported to the ANSYS/LS-DYNA. Due to the symmetrical structure of pump valve quarter part of pump is used for analyses [7].

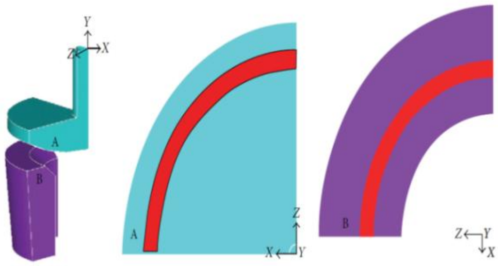

Figure 7: Finite element model of the pump [7].

C. Results of Analysis

Figure 8 shows the contact Von Mises stress vs mass of valve disc. In the analysis, mass of disc varies from 0.185 to 1.2 kg. The mass of disc increases the contact stress increases proportionally. The minimum stresses obtained at 0.185 kg mass of disc. The maximum stresses obtained at 1.2 kg mass of disc.

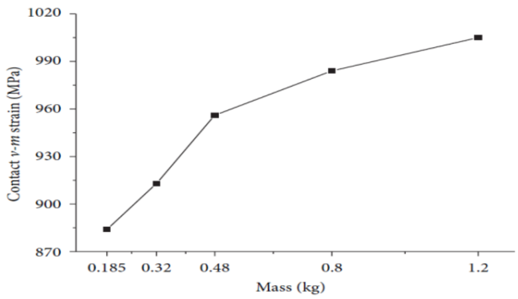

Figure 8: Von Mises stress vs Mass [7].

Figure 9 shows the contact Von Mises stress vs diameter of valve disc. In the analysis, diameter of disc varies from 62 to 102 mm. The diameter of disc increases the contact stress increase proportionally. The minimum stresses obtained at 62 mm diameter of disc. The maximum stresses obtained at 102 mm diameter of disc.

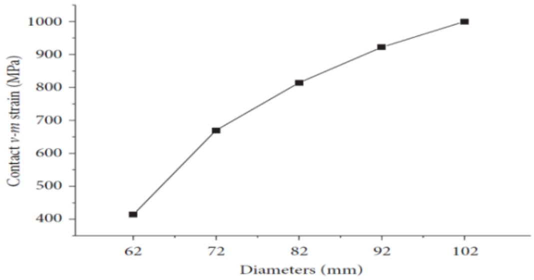

Figure 9: Von Mises stress vs Diameter [7].

Figure 10 shows the contact Von Mises stress vs taper angle of valve disc. In the analysis, taper angle of disc varies from 30^0 to 70°. As the taper angle of disc increases, the contact stress decreases proportionally. The minimum stresses are obtained at 70° taper angle of disc. The maximum stresses obtained are at 30° taper angle of disc.

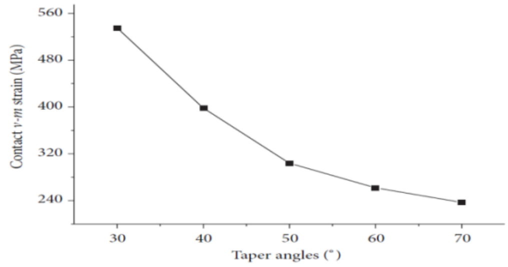

Figure 10: Von Mises stress vs Taper Angle [7].

V. CFD ANALYSIS OF TRIPLEX PUMP

Triplex plunger pump is widely used in chemical industry, automobile industry and pharmaceutical industry. The triplex pump consists of basic components like crankshaft, connecting rod, and crankshaft support bearings.

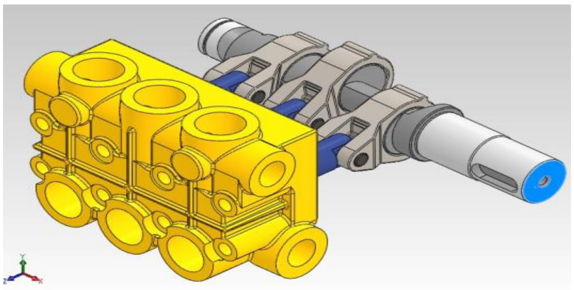

Figure 11: 3D model of pump [10].

3D mode of pump is modeled using solid works and converted the model into .igs file then imported in ICEM CFD. The fluid domain is extracted and the fluid flow path is made air tight for the triplex pump. The analysis of triple cylinder is very complicated. Hence single cylinder domain has been extracted [10].

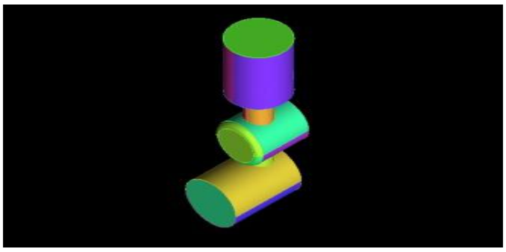

Figure 12: Extracted domain of the pump [10].

A. Grid Generation

The extracted model is sub divided into number of smaller parts. Then the parts are converted into non dimensional coordinates for grid generation. The pump is divided into three regions, plunger, cylinder (surface wall), and cylinder head (top cover. Each region is discretized independently. Triangular surface mesh is generated on the surface wall. Tri mesh is created on the plunger. Quad mesh of single layer is developed in the top cover of plunger for layering of mesh during dynamic meshing using ANSA. Layering is used to simulate the plunger movements with the inputs speed, stroke length and degree of crank period [10].

B. Boundary Condition

Parameter	Simulating condition
Grid	Structured
Fluid	Water at standard condition
Inlet	Total Pressure = 101325 Pa
Outlet	Mass flow rate = Variable Kg/s
Turbulence model	Standard K-ε model

C. Result:

Figure 13 shows contour of static pressure. The static pressure is varying from 5.10e6 pa to 1.00e3 Pa. The maximum static pressure is at cylinder top dead center because the fluid is under highest compression force imparted by plunger. The maximum pressure is observed at inlet chamber because of the closed inlet valve so that the movement of the liquid is arrested.

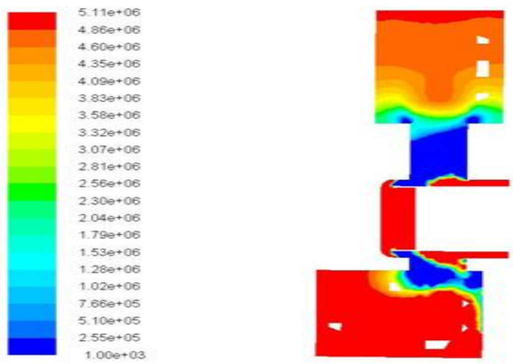

Figure 13: Counter of static pressure [10].

Figure 14 shows contour of velocity magnitude. Magnitude of velocity is 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 chamber because the fluid is incompressible and has no movement.

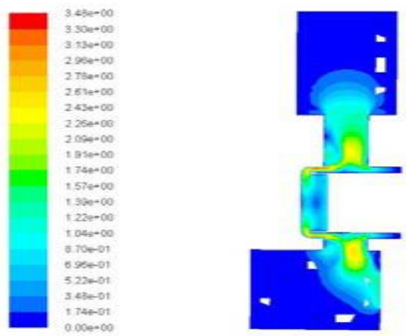

Figure 14: Contour of velocity magnitude [10].

Figure 15 shows contour of turbulence intensity. Turbulence is high at the corners of cylinder top. Turbulence is less in the outer region of inlet and outlet chamber because of no movement of liquid in those regions.

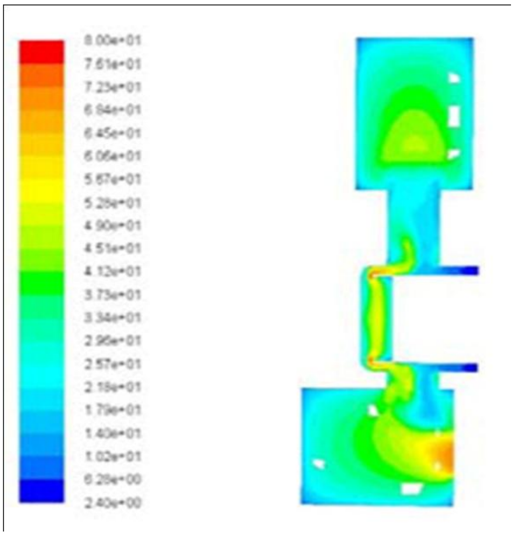

Figure 15: Contour of turbulence intensity [10].

6. CASE STUDY

6.1 CFD Analysis of Simplex Reciprocating Pump

Consider 2 D diagram of simplex pump, having 15 mm diameter and 100 mm height. Water is used as working fluid. Inlet velocity is .5 m/s. Outlet pressure is 10 bar.

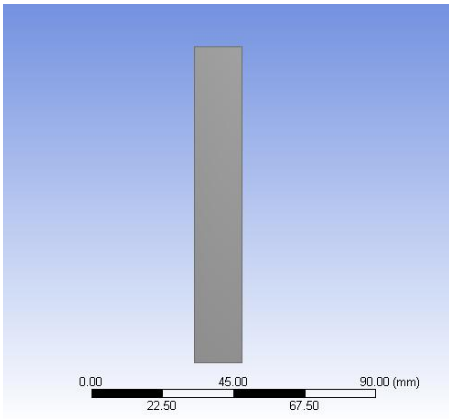

Figure 16: 2 D Diagram of Simplex.

6.2 Meshing

In meshing, geometry is divided into small number of element.

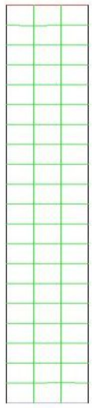

Figure 17: Meshing of geometry.

6.3 Results

The result shows the contour of static pressure, velocity magnitude, and density. Figure 18 shows contour of static pressure. Static pressure is high at inlet. Static pressure is less just away from inlet. Maximum static pressure observes is 7.94e+07 Pa.

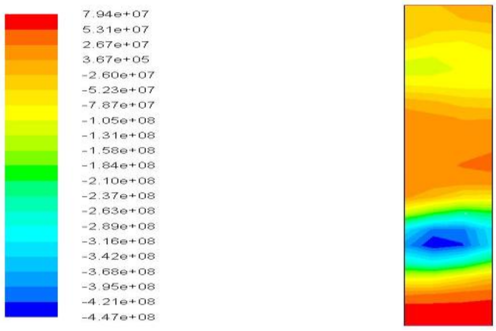

Figure 18: Contours of static pressure.

Figure 19 shows contour of density. Density is constant throughout the pump. The density of fluid is 1.23 Kg/m^3.

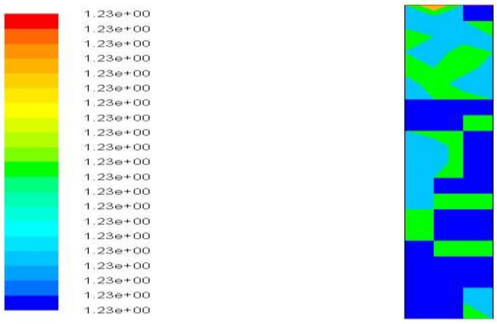

Figure 19: Contour of density.

Figure 20 shows contour of velocity magnitude. Velocity magnitude is zero along the wall. Velocity magnitude is high just away from inlet. Maximum velocity magnitude is observed is 1.44e+04 m/s.

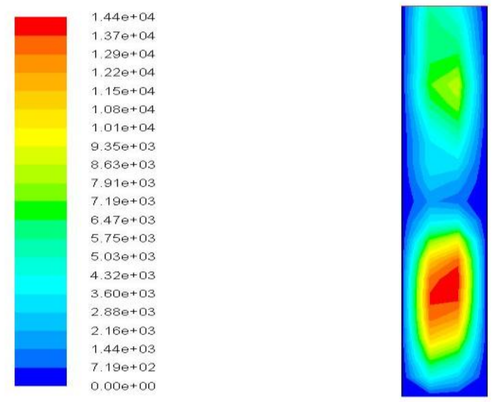

Figure 20: Contour of velocity magnitude.

6. SUMMARY

The effect of grooves on the piston of an axial piston pump is discussed in this seminar. The effect of number of grooves cut long the length of piston, their position, different patterns of the grooves on the performance of pump is studied. CFD analysis is used to predict the most stable model for the axial piston pump by specifying number of grooves and their orientation. The new experimental approach is acquire the true and accurate velocity, displacement data during pump valve collision and then combining the data to calculate the maximum Von Mises stress by ANSYS/LS-DYNA. Pump valve is vulnerable due to the residual stain generated during collision. In order to overcome the problem, high abrasion resistance, high fatigue strength and high strength material is selected. The CFD analysis of plunger pump, result shows contour of static pressure, velocity magnitude, turbulence intensity. In case study, 2 D geometry of simplex reciprocating pump is analyses by using CFD. The result shows contour of static pressure, density and velocity.

REFERENCES

[1] Tsui Y.Y. and Lu S.L., (2008), “Evaluation of the performance of valvelessmicropump by CFD and lumped system analyses”, Sensors and Actuators, Vol.148, Issue 1, pp.138-148.

[2] Kumar S. and Bergada J.M., (2013), “The effect of piston grooves performances in an axial piston pump via CFD analysis”, International Journal of Mechanical Sciences, Vol. 66, pp. 168-179.

[3] Fan J., Eves J., Toropov V.V., Kapur, Copley D. and Mincher A., (2011), “Computational fluid dynamic analysis and design optimization of jet pump”, Computers and Fluids, Vol. 46, Issue 1, pp. 212-217.

[4] Alves M. C., Barbosa J.R. and Prata A. T., (2013), “Analytical and CFD modeling of the fluid flow in an eccentric-tube centrifugal oil pump for hermetic compressors”. International Journal For Refrigeration, Vol. 1, Issue 7, pp. 1905-1915.

[5] Lucmann A.J., Alves M.V.C. and Barbosa J., (2009), “Analysis of oil pumping in reciprocating compressor”, Applied Thermal Engineering , Vol. 29, Issue 14-15, pp. 3118-3123.

[6] Kumar S., Bergada J.M. and Watton J., (2009), “Axial piston pump grooved slipper analysis by CFD simulation of three dimensional NVS equation in Cylinder Coordinates”, Computers and Fluid, Vol. 38, Issue 3, pp. 468-663.

[7] Pei J., Hai C., Lv M., Huang X., Shen K. and Bi K., (2013), “Collision Contact Characteristics for reciprocating pump using FEA and experiments”, Advances in Mechanical Engineering, Volume 2013.

[8] Kumar A., Grover K. and Budania B., (2012), “ Optimization of Connecting Rod Parameters using CAE Tools”, International Journal of Latest Treads in Engineering and Technology, Vol. 1, Issue 3, pp. 98-104.

[9] Parkash O., Gupta V. and Mittal V., (2013), “Optimizing the Design of Connecting Rod under Static and Fatigue Loading”, International journal of research in Management, Science and Technology, Vol. 1, Issue 1, pp. 39-43.

[10] Singh R.R. and Nataraj M., (2012), “Study on Performance of Plunger Pump at Various Crank angle using CFD”, Engineering Science and Technology: An International Journal, Vol. 2, Issue 4, pp. 549-553.

[11] Samad A. and Nizamuddin M., (2013), “Flow Analyses Inside Jet Pumps Used for Oil Wells”, International Journal of Fluid Machinery and Systems, Vol.6, Issue. 1, pp. 1-10.

[12] Hubacher B. and Groll E., (2004), “Crankshaft Bearing Analysis of a Single-Stage, Semi-Heremetic Carbon Dioxide Compressor”, International Compressor Engineering Conference.

[13] Deng L., Liang S., Liu Q., Wu J. and Xu J., (2012) “CFD Analysis of Oil Discharge Rate in Rotary Compressor”, International Compressor Engineering Conference , 14-19 July 2012 at Purdue University, West Lafayette, INDIANA, USA.

[14] Diany M. and Bouzid A., (2009), “Analytical Evaluation of Stresses and Displacements of Stuffing-box-packing based on a Flexibility Analysis”, Tribology International, Vol. 42, Issue6,pp.980-986.