AC 2007-821: TWO-DIMENSIONAL CFD ANALYSIS OF A HYDRAULIC GEAR PUMP

Hyun Kim, Youngstown State University

Hyun W. Kim, Ph.D, P.E. Hyun W. Kim is a Professor of Mechanical Engineering and Director of Fluid Power Research and Education Center at Youngstown State University. He has been teaching and developing courses and research projects in the fluid thermal area. He is a registered Professional Mechanical Engineer in Ohio and is currently conducting applied research in fluid power control and computational fluid dynamics with local industries. Dr. Kim received a B.S.E. degree from Seoul National University, a M.S.E. from the University of Michigan, and a Ph.D. from the Univ. of Toledo.

Hazel Marie, Youngstown State University

Hazel Marie, Ph.D.

Dr. Hazel Marie is currently an Assistant Professor of Mechanical Engineering at Youngstown State University. In 2004, Dr. Marie received the university-wide Distinguished Professorship Award for Teaching and the Outstanding Teaching Award from the Northeast Ohio Council on Higher Education. Dr. Marie's current research is focused on CFD modeling of solid-fluid interaction of thin film lubrication in rotordynamic seals coupled with the overall dynamic modeling of adaptive seal – rotor assemblies in turbine engines. Prior to entering academia, Dr. Marie worked as a materials engineer, primarily investigating high temperature copper alloys, and as an automation process engineer both at Packard Electric in Warren, OH. In addition, she currently offers consulting services to local industries.

Suresh Patil, Youngstown State University

Suresh Patil Suresh Patil is currently working at Youngstown State University as a Research Associate. He received a B.S. degree in Mechanical Engineering from India and a M.S. in Mechanical Engineering from Youngstown State University in Fall 2006. His research interest is in computational fluid dynamics.

Two-dimensional CFD Analysis of a Hydraulic Gear Pump

Abstract

Development of a productive research and educational program through a strategic focus on technology development in emerging areas such as controls and computing has been one of the primary goals of the mechanical engineering program. Several main initiatives to facilitate the successful development have been implemented. A number of collaborative research and educational projects have been conducted or are in progress, including computational fluid dynamics (CFD) projects and simulation projects for intelligent hydraulic systems, a valve design project and a piston pump-based dynamometer project. This gear pump analysis is one of CFD projects that have been developing in the last few years.

The physical model chosen for this analysis represents P-640 series external gear pumps manufactured by a local fluid power industry. FLUENT, commercially available CFD software, was used to analyze the two-dimensional flow of this industrial gear pump. The investigation produced significant information on flow patterns, velocity and pressure fields, and flow rates. The data clearly showed the phenomena regarding extremely high and low internal pressures that design engineers have encountered in the past. As expected, the accuracy of the computed flow rate for a given model increases as the outlet pressure decreases. It also increases as the speed increases. The results also confirm that the fluid gap size is the most pertinent parameter affecting the pump capacity.

The development of computational projects and research positively affect undergraduate and graduate education in this small mechanical engineering program. The use of commercial CFD software enhances students' learning and understanding of complex flow phenomena. The experience obtained through this analysis will be incorporated by expanding the computer use in undergraduate design courses and graduate courses.

1. Introduction

Development of a productive research program through a strategic focus on technology development in emerging areas such as controls and computing has been one of the primary goals of the mechanical engineering program at this University. In order to accomplish this goal and expedite the development, the program has been striving to develop a niche area in fluid power and control. The first initiative was to establish a hydraulics research and education center through a partnership with a local hydraulic industry. It acquired a couple of hydraulic and pneumatic trainers for instructional use. However, with a moderate amount of internal and external funding, it was difficult to purchase expensive equipments for research such as hydraulic test cells. Therefore, in order to jump start research program, a decision was made to focus on computational fluid dynamics and motion control simulation. A number of projects have been developed in cooperation with the local fluid power industry, including flow analysis of pumps and simulation projects for intelligence hydraulic systems, and a piston pump-based dynamometer project. In addition, the use of the CFD and motion control software for undergraduate and graduate instructional purpose was promoted vigorously to expedite the skill development and expand the basis of research pool. This gear pump analysis is one of the on-going CFD projects that have been investigating the flow inside external gear pumps for the last several years. The analysis is also developed as a master's thesis project. The project was not externally supported. This paper summarizes technical results obtained from the CFD analysis.

2. 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[1]. All pumps used in fluid power systems are of the positive displacement type that includes gear, vane, and piston pumps. The gear pump is made of two or more gears rotating inside a closed casing. The driving gear motion is produced by a motor, while the driven gear motion occurs through the meshing of the teeth of the two gears. As the gears start to rotate, the teeth are in and out of contact with each other. As a tooth leaves the contact region, a vacuum is created. The liquid that runs into this space to fill this vacuum has to be supplied through the pump's inlet port. Once filled with the fluid, the fluid follows in pockets between the teeth, trapped in place because of the sealed housing, until it reaches the pump's outlet port. In other words, the gear pump works like a rotating conveyor belt that moves pockets of liquid between the teeth of the gears.

The hydraulic pump has been used for a long time since the ancient Egyptians invented water wheels with buckets mounted on them to move water for irrigation. Despite its long history of usage and numerous researches on related subjects[2], no work has been reported in the literature that examined the behavior of transient flows of a gear pump with the exception of a technical note[3]. The flow pattern created inside a gear pump by the motion of two gears rotating in opposite directions 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. Consequently, FLUENT, a leading software package with moving dynamic meshing capability was selected to investigate the flows in gear pumps. Although the complexity of analysis is inherent in all positive displacement pumps, the gear 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 two rotating gears which must be in contact with each other all the times.

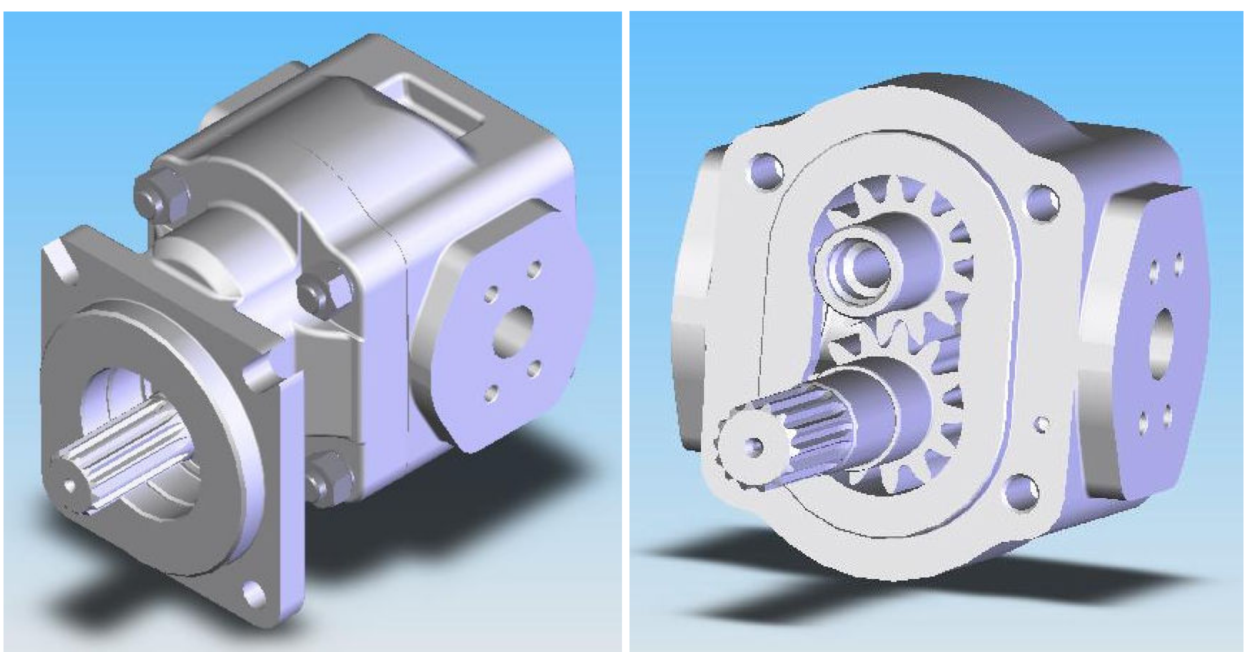
The study and analysis presented in this paper will address those problems associated with establishing an acceptable preliminary investigation on the gear pump flow by documenting a procedure for the modeling and the results of the numerical analysis. The simulation on 2-D models was performed on a Dell Precision 470 workstation with dual 4 GB processors. It is expected that the results obtained in this report provide crucial information for the extension to improved two dimensional and three dimensional analyses.

3. Modeling

One of the PGP 640 series external gear pumps manufactured by Parker Hannifin Corporation was selected for this study. This heavy duty, external gear pump is made of stainless steel for gear rotors and cast iron for gear housing and consists of four basic components, casing, driving gear, driven gear, and end plates. The pump operates at 2500-3500 psi continuously in the range between 400 and 3000 rpm. Figures 1(a) and (b) show the SolidWorks 3-D model of the pump.

Following are the specifications and operating conditions of the pump.

Type	:	External gear pump	
Pump Displacement	:	50 cc/rev	
Speed	:	500 to 3500 rpm	
Pressures	:	up to 3500 psi	
Flow rate	:	up to 41 gpm @ 3000 rpm	
Ambient Temp	:	- 40°C to + 70°C	
Dimensions	:	Inlet port: 38.1 mm	
		Outlet port: 25.4 mm	
		Center distance between gears: 50.8 mm	
		Gear width: 32.9 mm	
Properties of the oil	:	Density:	878.3 kg/sec.
		Viscosity:	0.005972 kg/m-sec



(a) (b)

Figure 1 –SolidWorks Model of Gear Pump

Following are the assumptions incurred on the present analysis:
1. The fluid is Newtonian and is incompressible.
2. The fluid is initially stationary.
3. The flow is two-dimensional.
4. Body forces acting on the fluid are negligible.
5. The fluid is isothermal and has constant properties.

By considering above assumptions, the governing equations in the Cartesian co-ordinate system, with its origin at the center of the driving gear, can be expressed as follows.

The continuity equation is

$$\frac{\partial \rho}{\partial t} + \nabla . \rho V = 0 \qquad \text{..... (1)}$$

Two scalar equations of the Navier-Stokes equation may be simplified as

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \text{.....(2a)}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \qquad \text{.....(2b)}$$

Initial condition,

at time $t \leq 0$: $\boldsymbol{V} = 0$

Boundary conditions,

on the casing wall: $\boldsymbol{V} = 0$
on the gear surfaces: $\boldsymbol{V} = \boldsymbol{Vs}$
at inlet port: $P = P_i$
at outlet port: $P = P_o$

The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The transport equations are based on assumptions that the flow is fully turbulent and the effect of the molecular viscosity is negligible.

Transport Equations for the standard k-ε model

$$\frac{\partial}{\partial t}(\rho \kappa) + \nabla(\rho \kappa V) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \qquad \text{.....(3.a)}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla(\rho \varepsilon V) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \qquad \text{.....(3.b)}$$

where the turbulent viscosity, $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$ **,** and the model constants are $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 0.3$ and $\sigma_\varepsilon = 1.3$

Many commercial CFD software packages are available in the market, which include FLUENT, CFD-ACE and Star-CD. FLUENT is the state-of-the-art commercial computational fluid dynamics software for modeling fluid and heat transfer problems in complex geometries. The numerical algorithm is based on the finite volume method. FLUENT is written in C language and has mesh flexibility including the ability to solve the flow problems using unstructured meshes in complex models. Hence, it is the most versatile software suitable for solving a problem like the gear pump. FLUENT, like almost all commercially available CFD software, utilizes a solution procedure[4] that can be described as follows:

- Preprocessor **–** Geometry creation and meshing by Gambit.
- Solver **–** Physical model defining, boundary conditions, equation solving.
- Postprocessor **–** Analysis and visualization of results.

Gambit is a preprocessing software package designed to import, build, and mesh models for FLUENT and other scientific applications. Gambit performs fundamental steps of building, meshing, and creating zone types in a model. Gambit's combination with CAD interoperability, geometry cleanup, decomposition, and meshing tools makes it one of the easiest, fastest, and most straightforward preprocessors. Most models can be built directly in Gambit or imported from major CAD systems. A comprehensive set of highly automated and size-function driven meshing tools ensures that the best mesh can be generated, whether structured, multi-block, unstructured or hybrid.

As in many numerical simulation using commercial software, success of the analysis by FLUENT largely depends on mesh quality of the model. Three successful models have been created through numerous trials and errors. Some of the pertinent information, techniques and skills developed during the modeling are summarized as follows:

- The geometry of the fluid domain is made simple such that it is suitable for the cooper mesh technique due to its unique advantage of having a simple algorithm for dynamic meshing.
- The length of inlet and outlet fluid passage is just sufficient to simulate the actual inlet/outlet conditions with reasonably accuracy.
- The model is enlarged, generally 10 times, for better meshing in narrow gaps and then scaled down in FLUENT back to its original size.
- More layers of elements are accommodated in fine gaps ensuring convergence of the solution. The number of layer of elements is limited to 3 due to small gap size and limitations of dynamic meshing.
- The skewness of the mesh is controlled by dividing the fluid domain in different faces and meshing it separately with interface between two faces.
- The density of mesh is controlled by a sizing function in particular regions of the fluid domain where the gradient of variables such as pressure and velocity is high; the purpose is to make the mesh denser in those areas; especially within narrow gaps and near walls.
- The skewness of the 2-D mesh is controlled below 0.6 in order to prevent premature collapse of cells.

For this study, a 3-D CAD model provided by the manufacturer was imported in the Gambit. The CAD model was reduced to a simplified 2-D model consisting of gears, casing and part of inlet and outlet. The simplification of the geometry of the inlet and outlet regions reduces number of cells, data storage, and processing time. A justification is that the scope of study is limited to the fluid domain near gears rather than inlet and outlet region, which may have less impact on pump performance. The simplified 2-D model is then meshed with triangular cells keeping mesh skewness below 0.6. A major problem with gear pump modeling is the creation of very fine mesh in extremely small gaps between the gears. These are the areas, which are vulnerable to fail during a transient simulation. Two earlier simulations[5, 6] reported results of the analysis for models having much wider gaps. Figure 2 shows the meshed 2-D model with triangular meshes. The number of cells and their maximum skewness are shown in Table 1.

Table 1 – Mesh Details of 2-D Model

Gap between the gears	Number of cells	Skewness
15 µm	451,409	< 0.548
22 µm	464,961	< 0.543
30 µm	457,855	< 0.543

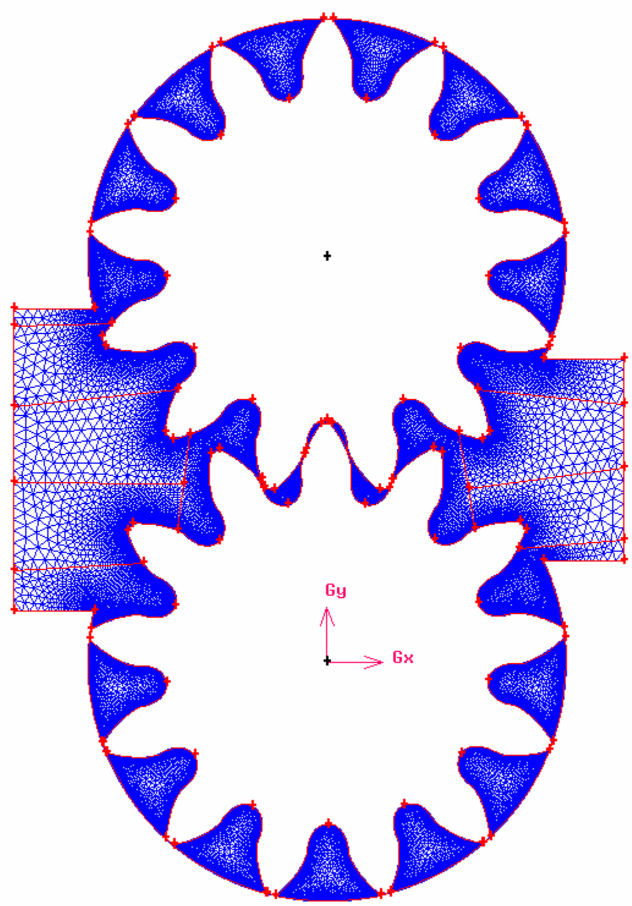


Figure 2 – Meshed 2-D Model

4. Analysis

FLUENT is a solver and also a postprocessor to review the results of the analysis. It has a unique feature of a Moving Dynamic Mesh (MDM) capability that facilitates solving transient flow problems with changing fluid domain due to motion of the domain boundaries. The motion can either be a prescribed motion or an un-prescribed motion where the subsequent motion is determined based on the solution at the current time. The update of the volume mesh is handled automatically by FLUENT at each timestep based on the new position of the boundaries. To use the dynamic mesh model, it is needed to provide a starting volume mesh and the details of the motion of any moving zones in the model by using either boundary profiles, user-defined functions (UDFs), or the Six Degree of Freedom solver (6DOF). FLUENT facilitates to specify the description of motion on either face or cell zones. If the model contains moving and non-moving zones, it is needed to identify these zones by grouping them into respective face or cell zones. The motion of the rigid body was specified by the linear and angular velocity of the center of gravity. Velocities can be specified by profiles or user-defined functions (UDF). UDF can be written for linear velocity, angular velocity or both as a function of time. In this case, angular velocity was specified to both gears by UDF. For the gear pump, it is essential to use UDF to define the motion of gears as the geometry of fluid domain changes with time. A UDF code was written in “C” language. The computational setup for a 2-D model is shown in Table 2.

The results presented here are from 2-D flow analysis performed for three different models having gaps of 30 μm, 22 μm and 15 μm between gears. The simulation was performed for 25 cases with different operating conditions such as outlet pressure and speed, as shown in Table 3. Each simulation is initialized with inlet pressure as 0 psi.

Table 2 – Computational Setup for 2-DAnalysis

Computation Domain:	
Inlet	stationary
Outlet	stationary
Casing	stationary
Gears	rotation at four different speeds
Boundary Conditions:	
Inlet	pressure inlet
Outlet	pressure outlet
Casing	Wall
Gears	Wall
Solver:	
Solver	Segregated
Formulation	Implicit
Space	2-D, double precision
Time	Unsteady

Viscous model:	
Model	k-epsilon, standard
Model constant	$C_{\mu} = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.44$
Solution Control:	
Pressure-Velocity Coupling	SIMPLE
Discretization	Pressure – Standard Momentum – First Order Upwind Turbulence Kinetic Energy – First Order Upwind Turbulence Dissipation Rate – First Order Upwind
Time step	1e-06 sec.
Convergence criteria	Residual : 1e-06
Dynamic Mesh Modeling:	
User Defined Function	Suitable for 2-D model – g3000.c
Dynamic Mesh zone	CW Gear : Type : Rigid body Motion : clockwise CCW Gear : Type : Rigid body Motion : anticlockwise

Table 3 – Operating Conditions for 2-D Simulation

Gap between Gears	Speeds	Outlet Pressures
• 15 µm • 22 µm • 30 µm	• 3500 rpm • 3000 rpm • 2500 rpm • 2000 rpm	• 3500 psi • 3000 psi • 2500 psi

There is no universal metrics for judging convergence of solution. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problem. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as mass flow rate. For most problems the default convergence criterion in FLUENT is sufficient. It is possible that if the initial guess is poor, the initial residuals are so large that a three-order drop in residual does not guarantee convergence. These are true for k and ε equations where the initial guess is difficult. The convergence criteria for this analysis is 1e-6 for residual and the size of time step is 1e-6 sec. Figure 3 shows convergence of solution at each time step.

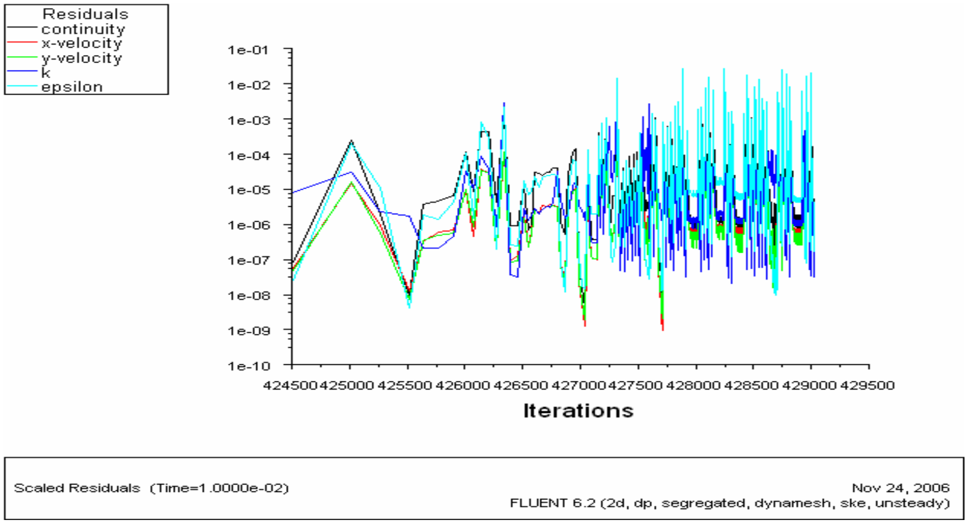


Figure 3 – Convergence of Solution

5. Results of 30 µm gap model

The 2-D analysis for 30 µm gap model was performed for 10,000 time steps, i.e. 180 deg. of rotation of gears. The mass flow rate through suction and pressure at the two points are monitored periodically. Also pressures contours and velocity vector plots are captured at every 25th time step. The mass flow rate as a function of time is sinusoidal and steady, as shown in Figure 4 and 5. No change has been observed in the nature of variation of mass flow with time.

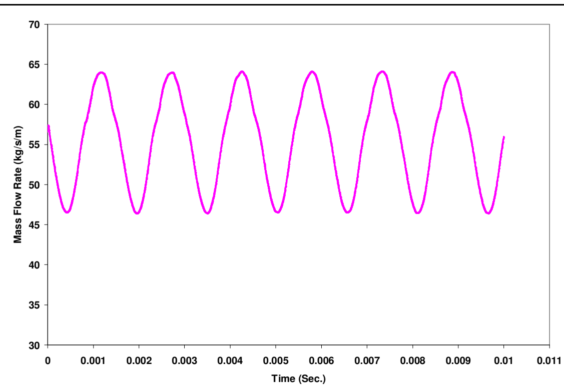


Figure 4 – Mass Flow Rate Results from Simulation (for 3000 rpm & 3000 psi)

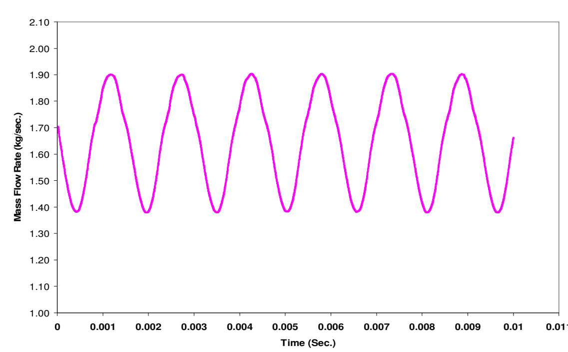


Figure 5 – Converted Mass Flow Rate for Actual Pump. (for 3000 rpm & 3000 psi)

The results also provide data on the effect of speeds and outlet pressures on mass flow rate. Table 4 shows that the mass flow rate decreases with increase in outlet pressure due to high resistance to flow. Figure 6 shows that the flow rate increases with speed. The increase in flow rate is in proportion to the speed increase. Table 5 and Figure 7 show the relation between speed and flow rate at different outlet pressures. The mass flow rates are less than the theoretical flow rates for all the cases, which is due to the slip through the gaps between casing and gear as well as between gears from the high to low pressure regions.

Table 4 – Mass Flow Rate for Different Speeds and Outlet Pressures.

Case No	Speed (rpm)	Outlet Pressure (psi)	Mass Flow Rate (kg/sec) Max.	Min	Ave
1	3500	3500	2.22	1.63	1.92
2		3000	2.24	1.68	1.96
3		2500	2.25	1.74	2.00
4	3000	3500	1.88	1.34	1.61
5		3000	1.90	1.38	1.64
6		2500	1.92	1.44	1.68
7	2500	3500	1.55	1.04	1.30
8		3000	1.57	1.09	1.33
9		2500	1.58	1.14	1.36
10	2000	3500	1.21	0.75	0.98
11		3000	1.24	0.80	1.02
12		2500	1.25	0.85	1.05

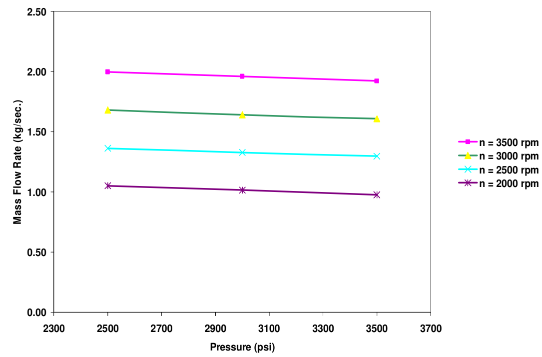


Figure 6 – Effect of Outlet Pressure on Mass Flow Rate at Different Speeds

Table 5 - Theoretical and Numerical Mass Flow Rates

	Theoretical Mass Flow Rate (kg/sec.)	Numerical Mass Flow Rate (kg/sec.)		
Outlet Pressure (PSI)		3500	3000	2500
Speed (rpm)				
3500	2.56	1.92	1.96	2.00
3000	2.20	1.61	1.64	1.68
2500	1.83	1.30	1.33	1.36
2000	1.46	0.98	1.02	1.05

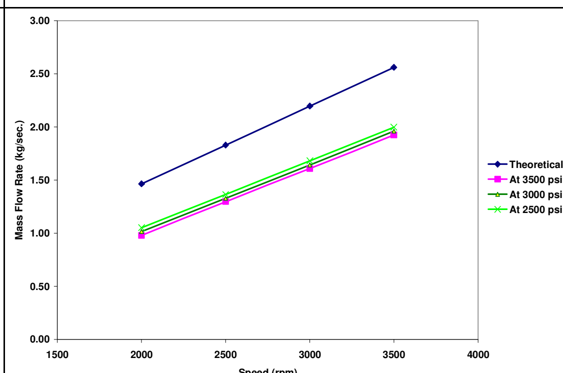


Figure 7 – Effect of Speed on Mass Flow Rate

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 25th time step to observe pressure variation in the fluid zone with respect to time. In this case with 3000 rpm and 3000 psi outlet pressure, the pressure reached to 4220 psi at 1.4e-3 seconds and falls to a minimum of -8440 psi at 8.75e-4 seconds, as shown in Figure 8 and 9. The negative pressure is observed in the gap between the gears teeth where the pressure region changes from high to low and generates high velocity flow through narrow gaps. It must be noted that the negative pressure is unrealistic and is the result of the extremely high local velocity explained above. It may also indicate that the cavitation generated due to extremely low local pressure invalidates the governing equations for a single phase fluid used in this analysis. Further examinations on this phenomenon will be needed in the future.

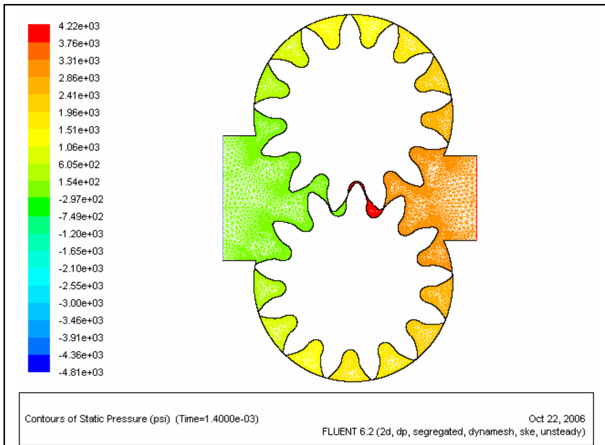


Figure 8 – Pressure Contour Plot showing Maximum Pressure (for 3000 rpm & 3000 psi)

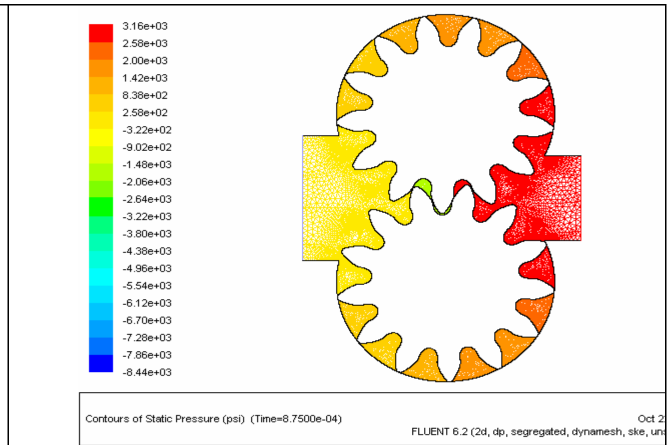


Figure 9 – Pressure Contour Plot showing Minimum Pressure (for 3000 rpm & 3000 psi)

Also, the pressure at two points in the fluid region is monitored. Two points, one in the suction region and the other in delivery region, are located for monitoring, as shown in Figure 10.

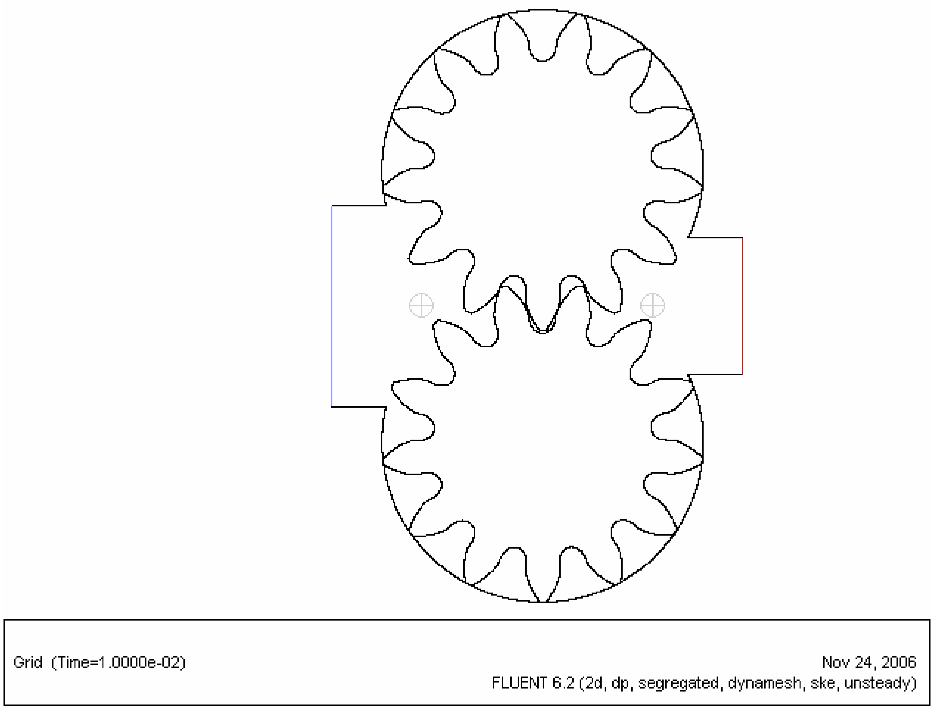


Figure 10 – The Point Locations for Pressure Monitoring

The pressure in the suction region falls below the atmospheric pressure which causes the suction of fluid in this region, as shown in Figure 11. The pressure fluctuates between -9.5 psi to 3 psi in this region[2]. On the high pressure or delivery side, the pressure varies between 2990 psi to 3010 psi as shown in Figure 12. The variation of pressure at both the points is cyclic and steady in nature. This is due to the meshing of the teeth.

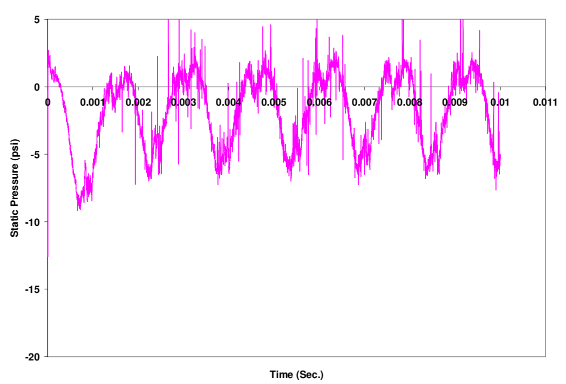


Figure 11– Pressure Variation at Point in Suction Region (for 3000 rpm & 3000 psi)

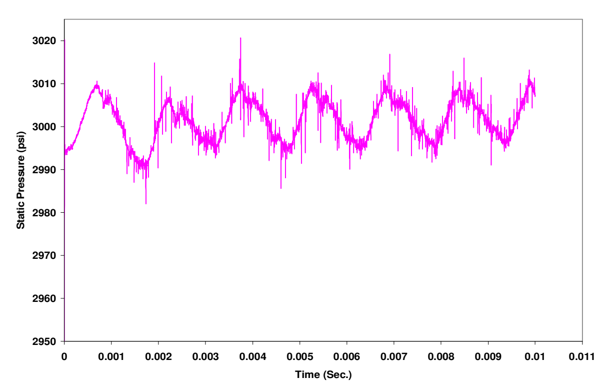


Figure 12 – Pressure Variation at Point in Delivery Region (for 3000 rpm & 3000 psi)

The maximum pressure in the fluid domain is observed between meshed gear teeth on the delivery side. It increases with outlet pressure and speed. Figure 13 shows the relation among the maximum pressure, speed and outlet pressure. Figure 14 shows minimum pressure in the fluid domain decreases with increase in speed.

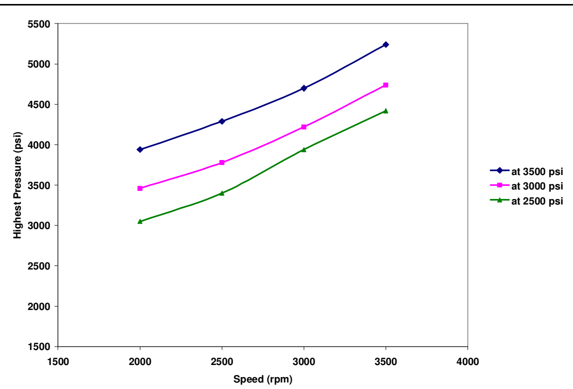


Figure 13 – Comparison Between Maximum Pressure in Fluid Zone and Speed

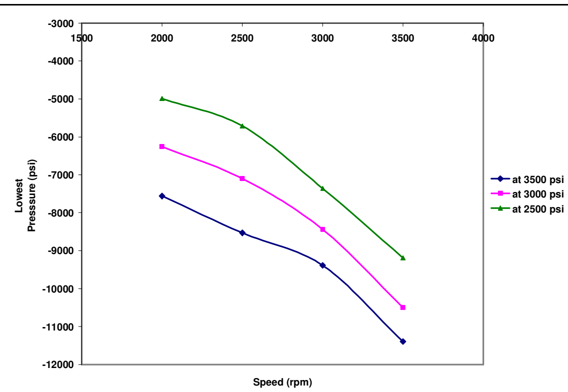


Figure 14 – Comparison Between Minimum Pressure in Fluid Zone and Speed

The velocity in fluid domain is also monitored. The maximum velocity of 388 m/s is observed in the gap of gears teeth between high and low pressure region, which creates the minimum pressure in the fluid region during a whole cycle of one tooth movement. Figure 15 and 16 show the high velocity of flow between the smallest gaps.

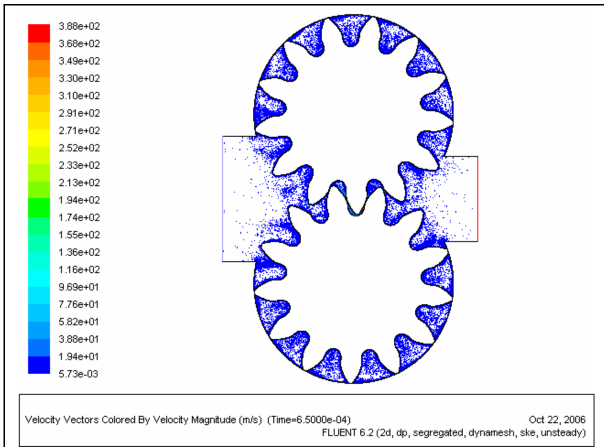


Figure 15 – Velocity Vector Plot (for 3000 rpm & 3000 psi)

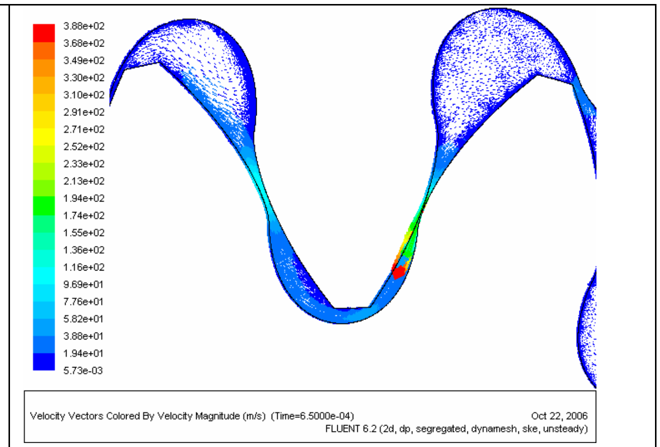


Figure 16 – Enlarged View of Velocity Vector Plot (for 3000 rpm & 3000 psi)

The maximum velocity in the fluid domain is also monitored. The maximum velocity in the fluid domain increases with speed as well as with outlet pressure. Figure 17 highlights the relation between the maximum velocity and speed, and also the effect of outlet pressure on velocity.

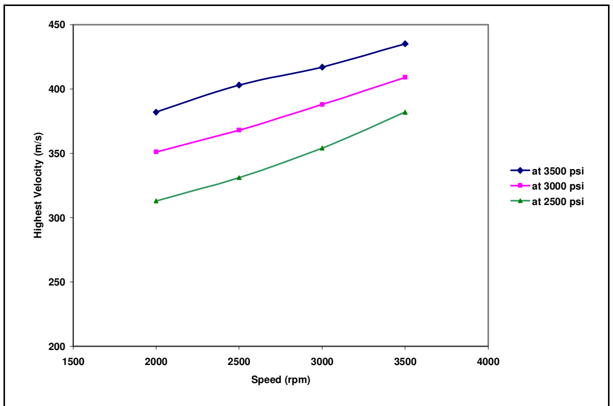


Figure 17 – Comparison between Maximum Velocity in Fluid Zone and Speed

6. Analysis of 15 µm gap model

The 2-D analyses for 15 µm gap model has been performed with same external conditions as the 30 µm model for 6000 time steps, i.e. 108 deg. of rotation of gears. The data shows that the simulation results of this model are closer to the theoretical value than the 30 µm model.

The general trend of the mass flow rate with respect to time is similar to that of the 30 µm model. However, the mass flow rate in this case is higher and closer to the theoretical mass flow rate, as shown in Figure 18 and 19 for the case with 3000 rpm and 3000 psi. Other trends such as change in mass flow rate with respect to speed and pressure are similar.

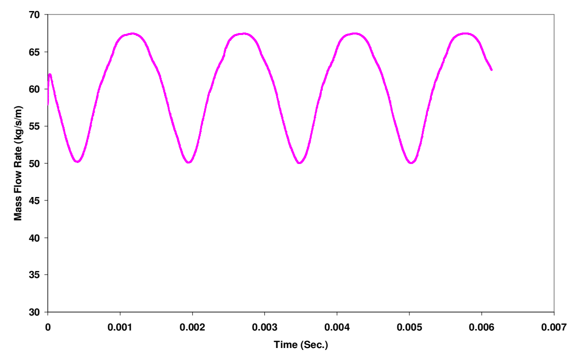


Figure 18 – Mass Flow Rate with Time for 15µm Simulation (for 3000 rpm & 3000 psi)

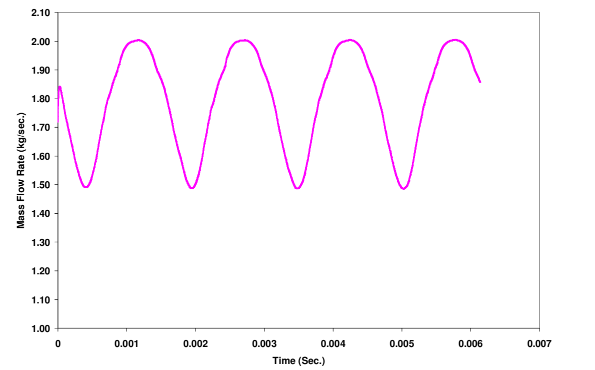


Figure 19 – Converted Mass Flow Rate with Time (for 3000 rpm & 3000 psi)

Table 6 – Mass Flow Rate at Different Speeds and Outlet Pressures.

Case No.	Speed (rpm)	Outlet Pressure (psi)	Mass Flow Rate (kg/sec.)		
			Max	Min	Ave
1	3500	3500	2.34	1.75	2.04
2		3000	2.34	1.79	2.07
3		2500	2.35	1.83	2.09
4	3000	3500	2.00	1.45	1.72
5		3000	2.01	1.49	1.75
6		2500	2.01	1.54	1.77
7	2500	3500	1.66	1.16	1.41
8		3000	1.67	1.20	1.43
9		2500	1.68	1.24	1.46
10	2000	3500	1.30	0.81	1.06
11		3000	1.33	0.91	1.12
12		2500	1.34	0.95	1.14

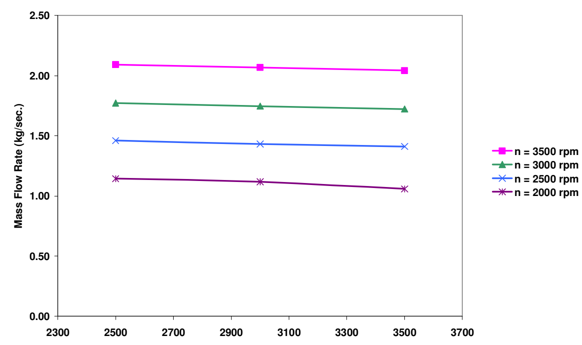


Figure 20 – Effect of Outlet Pressure on Mass Flow Rate at Different Speeds

Figure 21 and 22 show the pressure contours of this model with the maximum and minimum pressure. As for the velocity and mass flow rate, its behavior is similar to that of the 30 µm model. The maximum pressure in this case is higher than the 30 µm model due to smaller gaps between the teeth. Also the drop in the minimum pressure in the 15 µm model is less due to the low maximum velocity in narrow gaps.

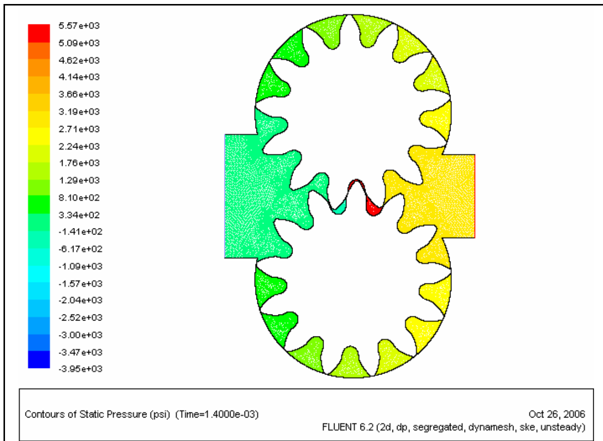


Figure 21 – Pressure Contour Plot with Maximum Pressure (for 3000 rpm & 3000 psi)

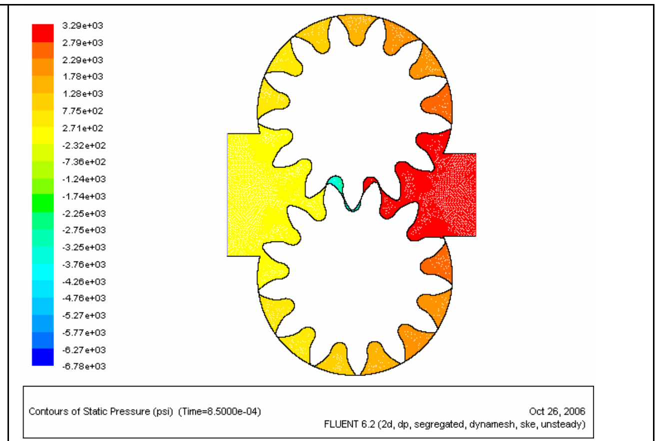


Figure 22 – Pressure Contour Plot with Minimum Pressure (for 3000 rpm & 3000 psi)

The nature of variation in velocity with respect to speed and outlet pressure is also similar to the 30 μm model. The velocity vector diagram is shown in Figure 23 and 24. It is interesting to note that the maximum velocity is lower, which may attribute to the smaller gap between teeth.

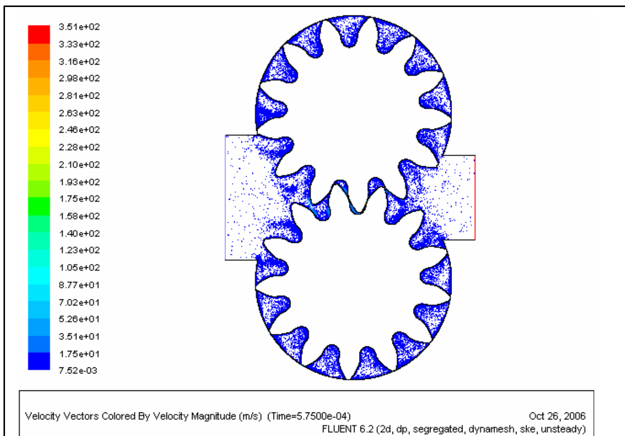


Figure 23 – Velocity Vector Plot (for 3000 rpm & 3000 psi)

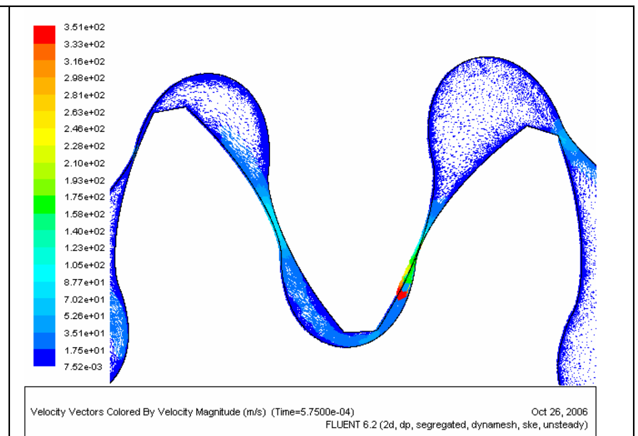


Figure 24 – Enlarged view of Velocity Vector Plot (for 3000 rpm & 3000 psi)

7. Effects of gap

Figure 25 through 28 show the effects of the gap between gears on the mass flow rate, maximum and minimum pressure, and the maximum velocity. As expected, the mass flow rate increases with finer gaps in the gears. The mass flow rate approaches the theoretical value of 2.20 kg/sec at 3000 rpm as the gap size decreases. The maximum pressure in the fluid domain increases with finer gaps between the gears. This is basically due to low slip through the gap. However, the minimum pressure decreases with increase in gap size. Figure 26 and 27 show the relation. The maximum velocity in the fluid domain increases with the gap between the gears, as shown in Figure 28. This causes the lowest pressure drop as mentioned above.

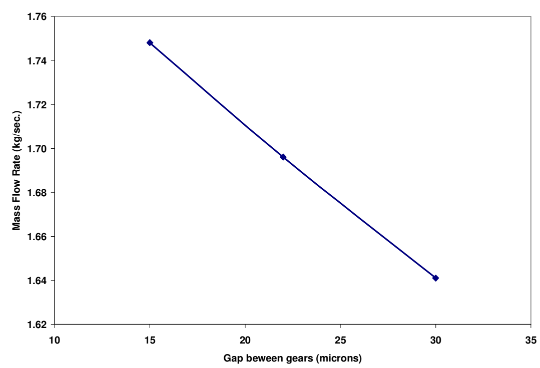


Figure 25 – Effect of Gap on Mass Flow Rate (for 3000 rpm & 3000 psi)

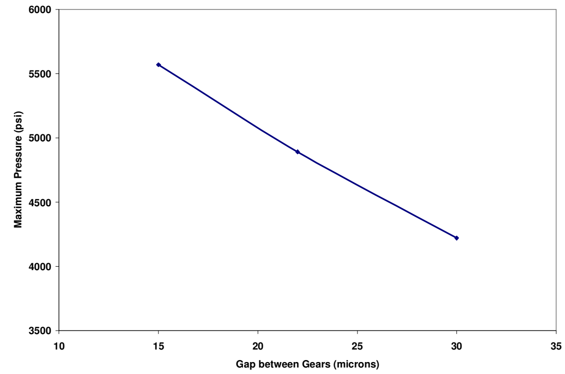


Figure 26 – Effect of Gap on Maximum Pressure (for 3000 rpm & 3000 psi)

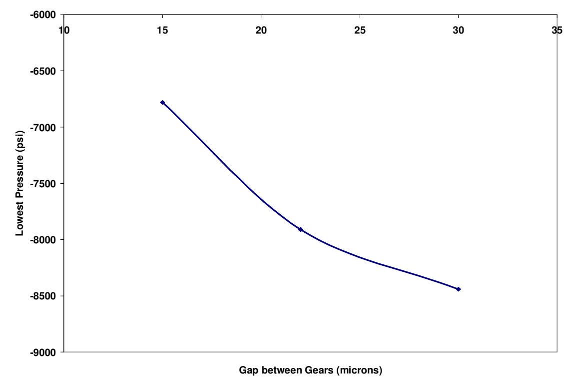


Figure 27 – Effect of Gap on Lowest Pressure (for 3000 rpm & 3000 psi)

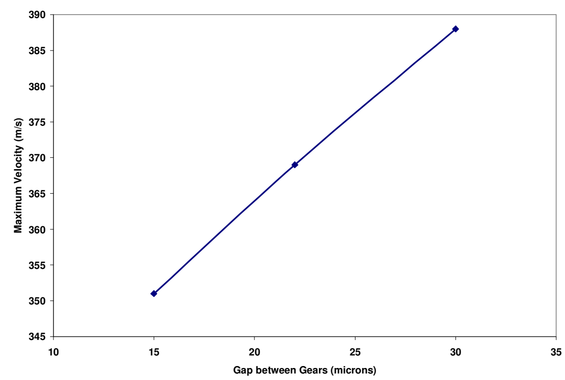


Figure 28 – Effect of Gap on Highest Velocity (for 3000 rpm & 3000 psi)

8. Observation and Conclusion

The leading commercial CFD software package, FLUENT, was used to analyze flows in an external gear pump. The simulation on 2-D models for transient flow analyses was performed on a Dell Precision 470 workstation with dual 4 GB processors. The 2-D simulation included 25 case studies for three different models with gap sizes of 15 µm, 22 µm, and 30 µm. Three different outlet pressures of 2500 psi, 3000 psi and 3500 psi were applied for each model at four different speeds of 2000 rpm, 2500 rpm, 3000 rpm, and 3500 rpm. As expected, the accuracy of the computed flow rate for a given model increases as the outlet pressure decreases. It also increases as the speed increases. The results also confirm that the gap size is the most pertinent parameter affecting the pump capacity.

The average mass flow rates of the 15-µm 2-D model at 3,000 rpm and 3,000 psi outlet pressure is 1.75 kg/s, compared to the theoretical flow rate of 2.2 kg/s. This means that the volumetric efficiencies of the 2-D flow model are 80%. The manufacturer's empirical data[7] of the pumps indicate that the volumetric efficiency would be slightly above 90%. The difference between the actual and computational data will be reduced by further narrowing the gap. The gap reduction and the sensitivity analysis on the improved 2-D models as well as 3-D models[8] will continue.

The development of computational projects and research positively affects undergraduate and graduate education in this small mechanical engineering program, such as fostering undergraduates' interest in fluid thermal sciences, invigorating undergraduate research, and the increment of master's thesis production. The use of commercial CFD software enhances students' learning and understanding of complex flow phenomena. The experience obtained through this analysis will be incorporated in expanding the computer use in undergraduate design courses.

Bibliography

1. James Sullivan, "*Fluid Power, Theory and Applications*, 4thEd.", Prentice-Hall, 1998.
2. Jaroslav Ivantysyn and Monika Ivantysynova, "*Hydrostatic Pumps and Motors*", Tbi, 2003.
3. Fluent, Inc., "*Gear Pump Solution, Example x219*", Fluent Inc., USA.
4. Fluent, Inc., "*User's Manual*", Fluent Inc., USA.
5. Yogendra Panta, "*Numerical Flow Analysis of Gear Pump*", Youngstown State University, 2004.
6. Jyotindra Killedar – "*CFD Analysis of Gear Pump*", Youngstown State University, 2005.
7. Parker Hannifin, *"Pump performance, PGP 640 Series"*, unpublished.
8. Suresh Patil, "*Numerical Simulation of Multi-Dimensional Flows in a Gear Pump*", Youngstown State University, 2006.