CFD ANALYSIS OF GEAR PUMP

Jyotindra S. Killedar

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT

FOR THE DEGREE OF

MASTER OF SCIENCE IN ENGINEERING

IN THE

MECHANICAL ENGINEERING

PROGRAM

YOUNGSTOWN STATE UNIVERSITY
AUGUST, 2005

CFD ANALYSIS OF GEAR PUMP

Jyotindra S. Killedar

I hereby release this thesis to the public. I understand that this thesis will be made available from the Ohio Link ETD center and the Maag Library circulation Desk for public access. I also authorize the University or other individuals to make copies of this thesis as needed for scholarly research.

SIGNATURE

JYOTINDRA S. KILLEDAR
DATE 08/08/2005

APPROVALS

8-8-2005

DR. HYUN W. KIM, THESIS ADVISOR
DATE

8-8-2005

DR. DANIEL H. SUCHORA COMMITTEE MEMBER
DATE

08-08-2005

DR. HAZEL PIERSON, COMMITTEE MEMBER
DATE

8/9/05

DR. PETER J. KASVINSKY, DEAN OF GRADUATE SCHOOL
DATE

DEDICATED TO MY PARENTS

MR. SHANKARRAO KILLEDAR & MRS. UMADEVI KILLEDAR

ACKNOWLEDGEMENTS

With Sincere gratitude I thank Dr. H.W. Kim for guiding me in my graduate course work on CFD analysis of Gear Pump, and for being my thesis advisor. Dr. Kim's encouragement and guidance led me through times when I may have missed the path in achieving my goal. I am one of the great beneficiaries of his immense knowledge. I will always admire and appreciate his knowledge and sharing of it with me without which this thesis would not have seen the light of the day.

I thank Dr. D. H. Suchora and Dr. Hezel Pierson for serving on my committee and guiding me from time to time. Their valuable time spent reviewing the manuscript and their constructive comments were greatly appreciated.

I would like to thank Mr. Tushar Sambharam, Mr. Nilesh Gandhi and Ashish Kulkarni of Fluent. Inc. for their valuable guidance on resolving the difficulties faced during the simulation using the Gambit and Fluent software.

I would like to thank Mr. Burke Davis who helped me in preparing the solid works model. Additional thanks go to all the colleagues. Through fun times and hard times, it felt as though we were a family, without which the experience would be empty.

I would like to thank department secretary Karen Pomponio for her cooperation and help rendered to me during my studies

My wife Dr. Smruti and my son Amogh, receive my greatest gratitude. For without their love, encouragement and boundless patience I would have never completed this endeavor.

Finally, I would like to give thanks to the Department of Mechanical and Industrial Engineering and Graduate School for providing financial support for my studies.

CURRICULUM VITA

2005	M.S. DEPARTMENT OF MECHANICAL & INDUSTRIAL ENGINEERING. YOUNGSTOWN STATE UNIVERSITY
1991	B.S. INDUSTRIAL ENGINEERING INDIAN INSTITUTE OF INDUSTRIAL ENGINEERING , MUMBAI, INDIA
1988	MASTERS IN MANAGEMENT SCIENCE.(M.M.S.) PUNE UNIVERSITY, PUNE, INDIA.
1986	DIPLOMA IN BUSINESS MANAGEMENT PUNE UNIVERSITY, PUNE, INDIA.
1984	B.S. –MECHANICAL ENGINEERING COLLEGE OF ENGINEERING, PUNE UNIVERSITY , PUNE, INDIA
1982	A.M.I.E – INSTITUTION OF ENGINEERS, KOLKATA, INDIA
1979	D.M.E. GOVERNMENT POLYTECHNIC, PUNE, INDIA

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

$\bar{A}$	Surface Area Vector.
$\bar{A}_f$	Area of face $f \lvert A \rvert = \left(\lvert A_x \hat{i} + A_y \hat{j} \rvert \right)$ in 2D .
$C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}, C_{\mu}$	Constants.
$\hat{e}_r, \hat{e}_\theta$	Unit vectors.
ε	Kinetic energy dissipation rate.
G	The transformation matrix that decides the choice of θ
G_k	The generation of turbulence kinetic energy (mean velocity gradients).
G_b	The generation of turbulence kinetic energy due to buoyancy.
k	Kinetic energy
$\nabla\Phi$	Gradient of Φ
∇	Del operator
Γ_ϕ	Diffusion coefficient for ø
ρ	Density
μ	Viscosity
u, v, w	Linier velocity as function of time
$\bar{V}$	Velocity vector
σ_k	Turbulent Prandtl number for k
σ_ε	Turbulent Prandtl number for ϵ
T	Temperature
V	Cell volume
P	Pressure
g_x, g_y, g_z	Gravity in x, y and z direction
$S_{\Phi,} S_k and S_\varepsilon$	User Defined source terms
V	Cell Volume
$(\nabla\Phi)_n$	Magnitude of $\nabla\Phi$ normal to f

ABSTRACT.

This thesis presents the development of computational fluid dynamics model of the external gear pump for the purpose of the analysis of flow, pressure, velocity and cavitation with an actual 2-D model. Analysis was done for various pressure outlet conditions and different speed range. A well-known CFD package –Fluent was used to perform analysis with control volume technique.

The CFD analysis results were compared with the theoretical data of the P315 gear pump manufactured by Parker Hannifin Corporation and reasonable agreement is found in terms of volumetric flow rate. This analysis is expected to be a useful reference to design and development engineers and they will be in position to use this work to obtain the variety of flow parameters of the gear pump.

This work also shows how effectively moving dynamic mesh can be used in analysis of the turbulent flow of the external gear pump. An exercise was also done in performing cavitation analysis as part of the CFD analysis of the gear pump. This work will be useful in setting up the cavitation problem in similar environment.

Lastly this two dimensional analysis will prove as stepping stone for three dimensional analysis of the gear pump.

CHAPTER 1

INTRODUCTION

In modern world hydraulic systems are often used to deliver and /or transmit power in operating machines, manufacturing goods and transporting materials. At the heart of almost every hydraulic system lies a pump. The pumps are broadly classified under two major categories (1) Positive displacement pumps and (2) Dynamic pumps (Kinetic pumps), Fig 1.1. A rotary Gear Pump belongs to a family of positive displacement pumps that are main choice for pumps of many fluid power or hydraulic systems. There are two different types of gear pumps depending upon the arrangement of gears. They are external and internal gear pumps. The pump under study is an external gear pump.

The gear pump generates the flow by moving /pushing the flow in forward direction. A known constant amount of fluid is generated in each revolution by the gear pump. Thus the flow of the gear pump is directly proportional to its speed [1] and the flow rate is obtained by (Q_{flow}) =k x N. The coefficient of proportionality "k" is better known as unit flow. The pressure in the system is merely a function of the system resistance of a hydraulic circuit under consideration. The pump is better known by its characteristic curve which is plot of flow rate as function of pressure (head generated by pump).

The invention of the first gear pump is credited to Pappenheim whose first pump was made in 1636 with gears having 6 teeth on each gear. This is the oldest form of chamber-

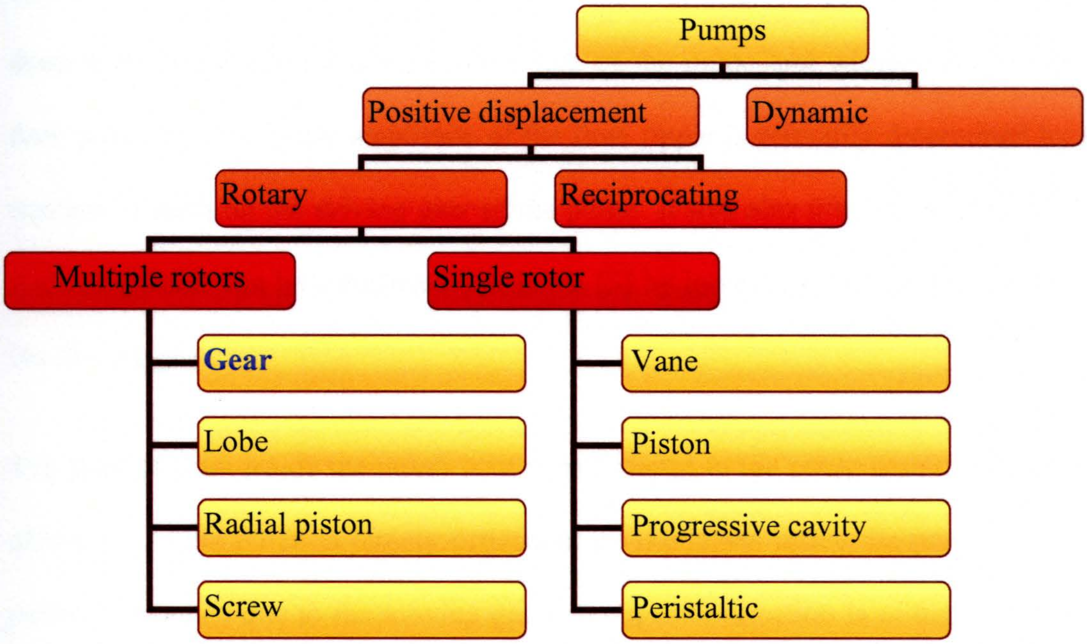


Fig 1.1 Classification of the pumps

wheel-gear, and the forerunner of modern gear pumps.

In 20th centaury, as the industrial growth took place, new techniques for gear manufacturing were invented and consequently advanced gear pumps were manufactured. One of the pioneers in the area of modern gear pump research is Prof. Borghi. He initiated a number of research projects dealing with gear pump analysis and simulations in cooperation with CASAPPA Spa., one of the major Italian gear pump manufacturers. His contributions were towards establishing the pressure distribution in the clearance between gears and the bearing blocks [2]

The research on theoretical flow ripple of an external gear pump of similar size, using different number of teeth on the driving and the driven gears was done by Noah D. Manring and Suresh B. Kasaragadda. They observed that reducing the number of teeth on the driven gear may reduce the center distance and physical pump size. This can be done without altering the average flow rate of the pump and without increasing the flow pulsation. The pulse amplitude of the flow ripple is primarily determined by the number of teeth on the driving gear of the pump. It was also discovered that the flow pulse amplitude can be significantly reduced [3] by increasing the number of teeth on the driving gear.

The flow of fluid inside the pump is of more interest to the pump manufacturers. The efficiency of the pump is largely dependent on the losses inside the pump. The flow pattern generated due to the moving gears in opposite direction is of highly turbulent nature and difficult to examine. The turbulent flow also creates more heat and the part of input energy will be lost in turbulence. The unsteady and turbulent flow inside the pump has been experimentally investigated by the industry with a limited success due to poor accessibility of probes into the pumps. Therefore, the industry mainly has been relying on macroscopic experimental data obtained from the test bed or empirical engineering data.

Experimental fluid dynamics has played an important role in validating and delineating the limits of the various approximations to the governing equations. The wind tunnel, for example, as a piece of experimental equipment, provides an effective means of simulating real flows. Traditionally this has provided a cost effective alternative to full scale measurement. However, in the design of equipment that depends critically on the flow behavior, full scale measurement as part of the design process is economically impractical. This situation has led to an increasing interest in the development of numerical methods.

The improvement in the speed of computers and the available memory size since the 1980s has led to the emergence of computational fluid dynamics. This branch of fluid dynamics complements experimental and theoretical fluid dynamics by providing an alternative cost effective means of simulating real flows. As such it offers the means of testing theoretical advances for conditions unavailable on an experimental basis.

With the rapid advancement in computing hardware and software, engineers turned their focus towards using CFD as a powerful tool for the flow analysis. In a very simple language the "Computational fluid dynamics" is simply the use of computers and numerical techniques to solve problems involving fluid flow. It involves obtaining the solutions to problems involving fluid mechanics. This is more commonly known as Fluid Flow Simulation. Consequently CFD is now the preferred means of testing alternative designs in many engineering fields before final, if any, experimental testing takes place.

Two important concepts in CFD concern verification and validation. These concepts are closely related to the final results where the connection with theory and experiment is sketched. Verification is a process that confirms the existence of an approximate solution converging to the exact solution of a given governing differential equations. Validation is a process that compares the solution of the governing differential equation with experimental outcomes.

Despite the sheer volume of CFD analysis published in recent years, very few results of successful CFD application to gear pump have been reported in literature. A CFD analysis of internal gear pump (gerotor pump) was carried out by *AEA Technology Engineering Software Ltd. in Waterloo, Ontario, Canada* [4]. In this study, there are nine lobes on the outer gear and eight lobes on the inner gear. The outer gear maximum diameter is approximately 45 mm and the inner gear maximum diameter is approximately 41 mm. The rotor thickness is 10 mm and the outer gear is revolved at 1000 RPM. The working fluid is oil and the flow is turbulent. The gerotor pump was modeled using the moving mesh capability in CFX-TASC flow. When the moving mesh feature is used additional terms are included in the governing equations to account for the movement of the grid. These terms account for the velocity of each grid node, since the position of the grid nodes change with time. The grid topology and number of nodes remain constant whereas the nodal position and velocity change each time step. A CFX-TASC tool macro was written to create the initial structured mesh in the rotor using gear profiles that were imported from a file. The rotor mesh was updated on-the-fly in the CFX-TASC flow solver by a user-accessible subroutine. This subroutine adjusts the rotor node positions at the start of each time step. Minimum grid skew was about 20 degrees, while maximum aspect ratio was about 24:1. A minimum clearance of 0.5 mm (500 microns) was used between the inner and outer gears. However, the actual clearance was approximately an order of smaller magnitude. CFX-HEXA was used to create the simple intake and outlet port grids. A non-matching grid interface was used to connect the intake and outlet port grid to the rotor grid. The grid interface is updated by CFX-TASC flow at the start of each time step after the rotor grid has been moved to its new position. A specified total pressure inlet and static pressure outlet were used to define the flow boundary conditions. The pressure differential between the inlet and outlet ports was specified as 10 psi. The resulting pressure and velocity patterns were generated and studied.

A CFD analysis of an external gear pump using Fluent was made by Panta [5]. He was successful in obtaining pressure and flow field inside the pump with different sets of boundary conditions. However, a close examination of his results revealed that the flow domain in the fluent was much larger than actual flow domain. A similarity analysis was required to find correlation of the results from such a large model to real model. To avoid similarity analysis, CFD analysis of actual size model was necessary. His study was part of on going research and development activities taking place at Youngstown State University in recent years. The scope of research activity in the field of hydraulics at YSU cover solid modeling, computational fluid dynamics analysis, interactive flow and stress analysis, interactive motion control and simulation.[6]

The current study presented in this thesis is an extension of Panta's work. A significant improvement in modeling, meshing, input data handling and running the model has been achieved. In consequence, the analysis produced much better results that compare very well with the theoretical values.

CHAPTER 2

MODEL OF GEAR PUMP

2.1 The Pump Model P-315:

The P-315 is proprietary model and is currently manufactured by the Parker Hannifin Corporation, Gear Pump division, Youngstown, Ohio, USA. The solid model was prepared in solid works based on the dimensioned drawings of the Pump P315. The external and internal views of the model are shown in Figs 2.1, 2.2, and 2.3. The alignment of the gears in the solid works itself is very important as it saves the considerable time while handling the model in the Gambit.

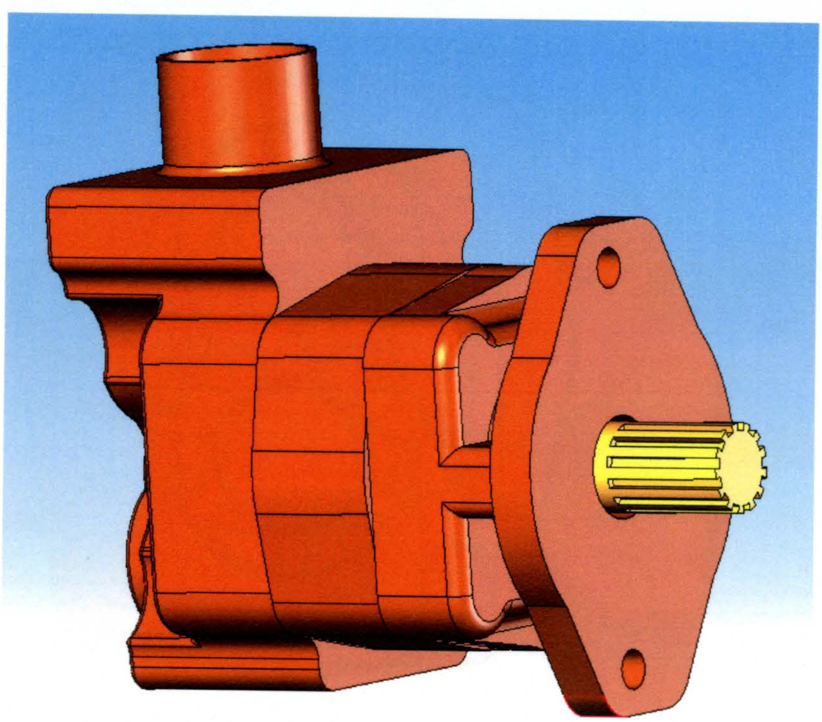


Fig 2.1. - The 3-D view of the P-315 Gear Pump

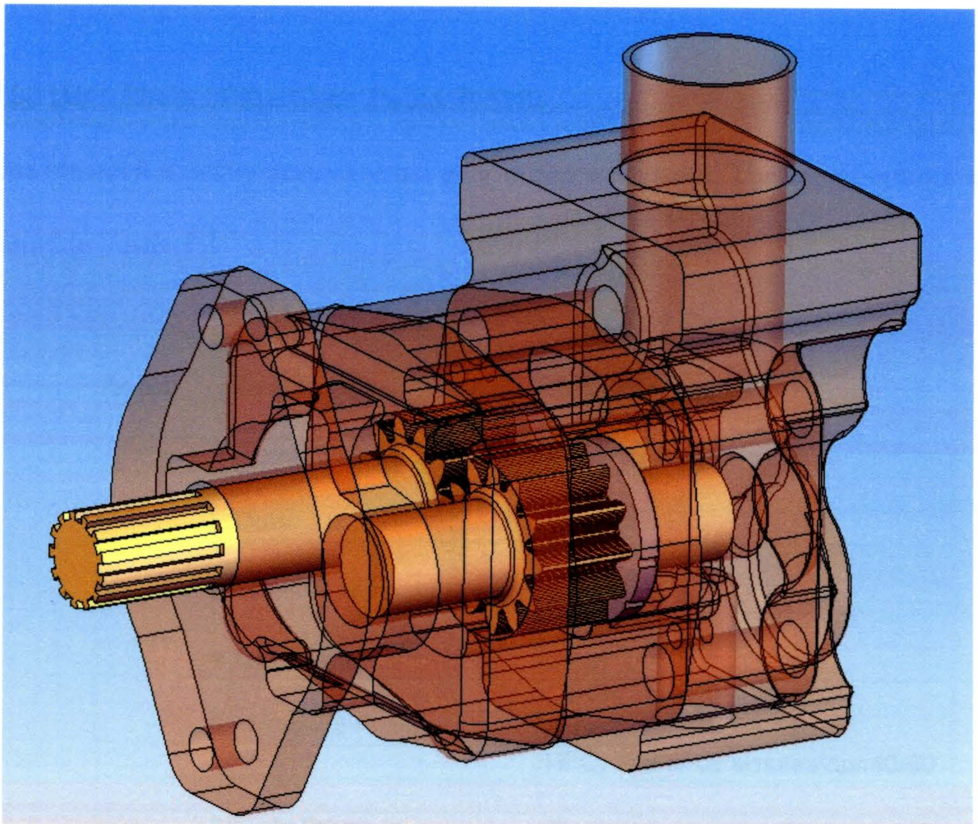


Fig 2.2 Wire frame view showing internals of the Gear Pump

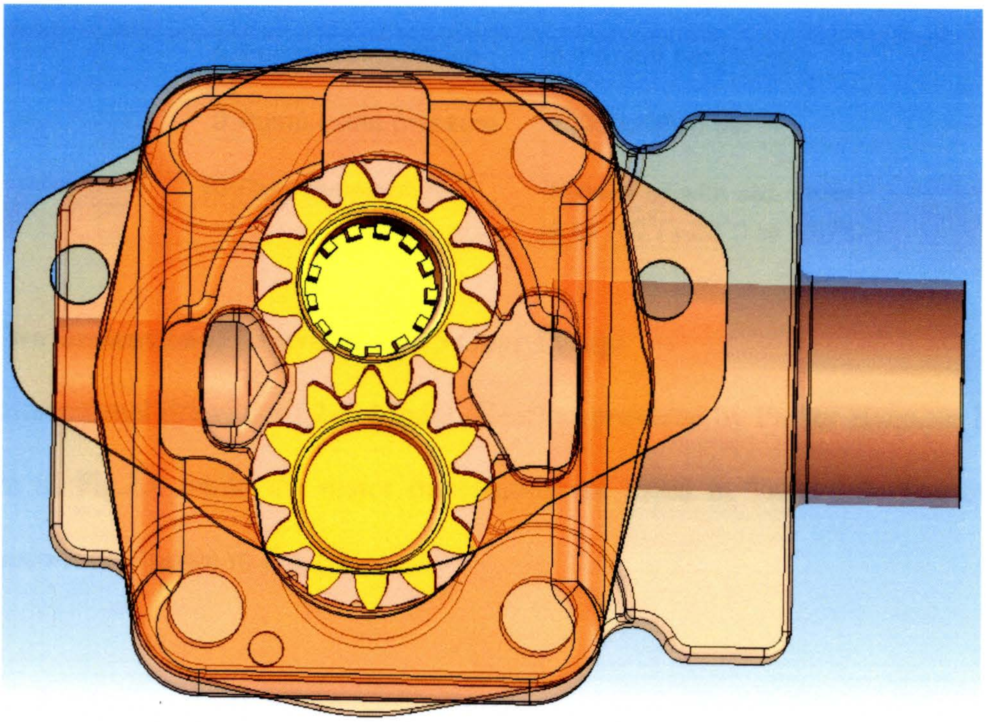


Fig. 2.3 Front view showing aligned gears

2.2 The Data Sheet of the Gear Pump P-315:

For this research a heavy duty external gear pump were used. The specifications are presented in Table 2.1

Table -2.1-Specifications of Gear Pump

Sr.No.	Description	Specifications
1	Pump type	External Gear Pump
2	Duty	Heavy
3	Material-Housing	Cast Iron
4	Gears	Steel
5	Hydraulic Fluid	Mineral Oil,Engine Oil HFC, Water oil emulssions40/60
6	Gear Drives	Clockwise, counterclockwise,doub
7	Speed range	400-3000 rpm
8	Pump Inlet pressure	0.8 to 2.0 bar (30 psi)
9	Pump outlet pressure	2500 to 3500 psi
10	Fluid temperature	Mineral oil with std. seals: 0 to 180^0F(-20^0C to $+80^0$C)

2.3 Two Dimensional Drawings P-315 Gear Pump:

The dimensional drawing of 2-D Model (Gear) as provided by Parker Hannifin Inc. is shown in Fig.2.4 while the major dimensions are listed in Table 2.2. The overall dimensions are shown in Fig.2.5.

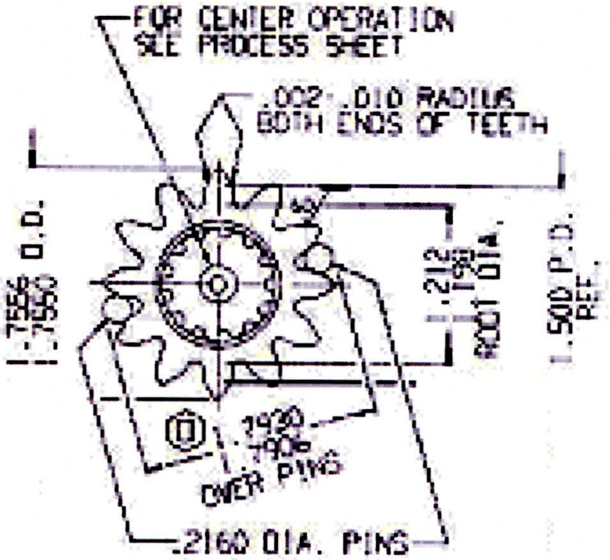


Fig 2.4 Gear of P315

Table 2.2 -Dimensions of gear

Weight	18 lbs
Gear width	1.25”
No. of teeth/gear	12
Center to center distance	1.49”
Radius of gear	0.877”
Inlet port size	1.2”
Outlet port size	0.6”

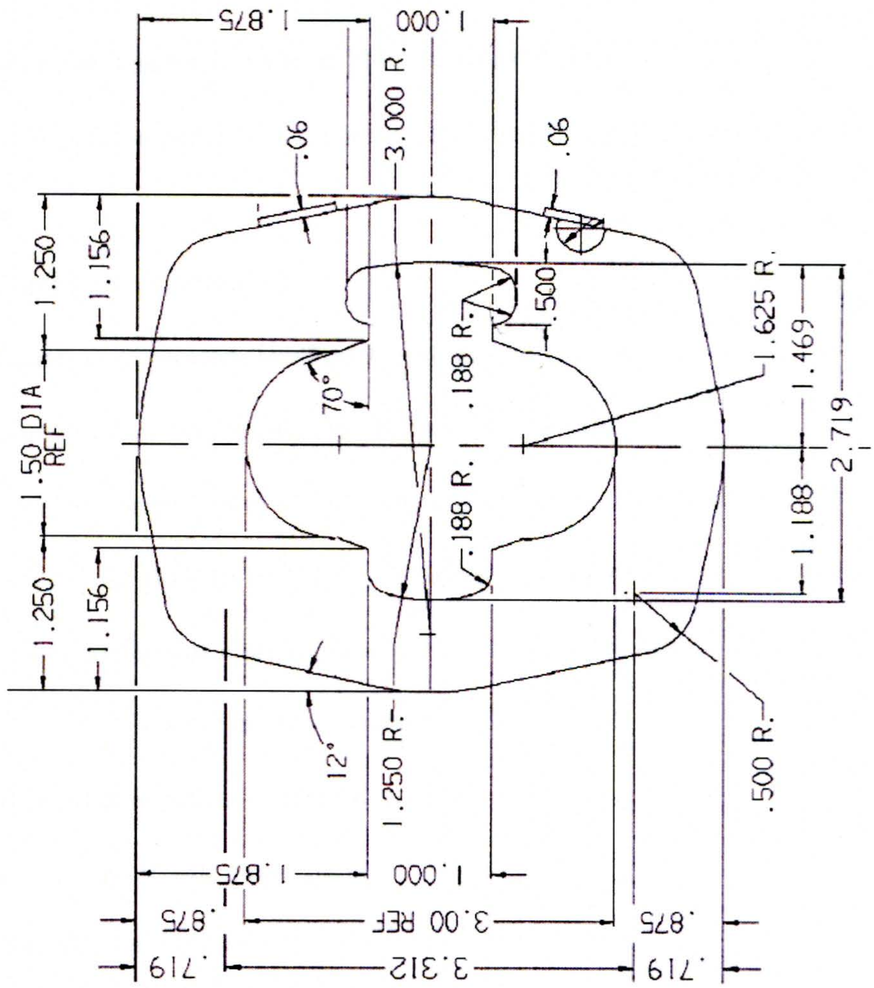


Fig 2.5 Casing of P315

CHAPTER 3

FORMULATION OF GOVERNNG EQUATIONS

3.1 Differential Analysis of Fluid Flow in Gear Pump

The governing differential equations for flows inside a gear pump, as in all other fluid flows, are:

1. Continuity equation
2. Equation of motion-The Navier- Stokes equation.
3. Energy equation –First Law of Thermodynamics.

The flow in the gear pump is generated by motion of the gears. This rapid radial motion causes the flow to be highly turbulent. Thus two more equations are added to properly address the turbulence in the pump flows. These equations are

1. Turbulent Kinetic energy equation
2. Rate of dissipation of Kinetic energy.

The following assumptions are incurred on this problem.

- The fluid is Newtonian.
- The fluid is incompressible.
- The flow is two dimensional.
- Body forces are negligible.
- No viscous heating is considered.

The coordinate system used is Cartesian co-ordinates and the origin is set at the center of the CCW gear. The co-ordinate system, boundaries and types of boundary conditions are indicated in the Fig 3.1. The governing equations are defined below. [7]

3.2 Continuity Equation

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}.(\rho \vec{V}) = 0 \text{ ....Equation 3.1}$$

For the incompressible flow ρ is constant therefore the equation 3.1 simplifies as:-

$$\vec{\nabla}.(\vec{V}) = 0 \text{ ---------Equation 3.2}$$

The above equation can also be expressed as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \text{ --------- Equation 3.3}$$

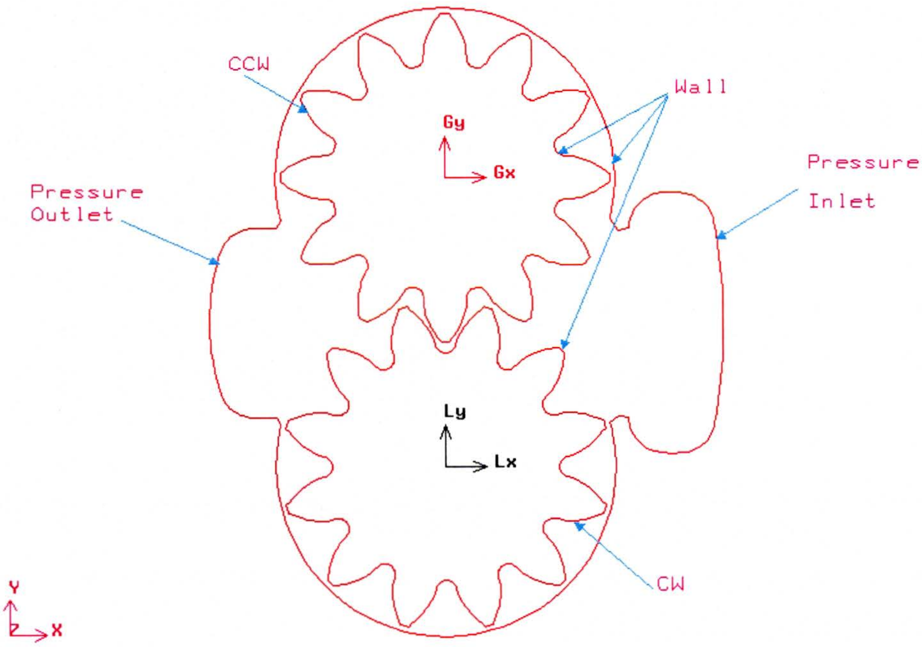


Fig 3.1 Gear Pump model

3.3 Navier Stokes Equations in Cartesian Coordinates:

The vector form of the Navier stokes equation for Incompressible flow is as follows.

$$\rho\left(\frac{\partial \vec{V}}{\partial t}+\left(\vec{V}.\vec{\nabla}\right)\vec{V}\right)=-\vec{\nabla}P+\rho\vec{g}+\mu\nabla^2\vec{V}$$ ------------------------------Equation 3.4

$x - component\ of\ the\ Navier - Stokes\ equation$

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$ ----- $Equation\,3.5$

------Equation 3.5

$y - component\ of\ the\ Navier - Stokes\ equation$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$ ------- $Equation\,3.6$

$z - component\ of\ the\ Navier - Stokes\ equation$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$ ------ $Equation\,3.7$

The model being two dimensional the z- component of Navier Stoke's equation is not applicable to present problem.

Initial Condition:-

At $t \leq 0 : \vec{V} = 0$

3.3 Navier Stokes Equations in Cartesian Coordinates:

The vector form of the Navier stokes equation for Incompressible flow is as follows.

$$\rho\left(\frac{\partial \vec{V}}{\partial t}+\left(\vec{V}.\vec{\nabla}\right)\vec{V}\right)=-\vec{\nabla}P+\rho\vec{g}+\mu\nabla^2\vec{V}$$ -----------------------------Equation 3.4

x – component of the Navier – Stokes equation

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$ ----- *Equation* 3.5

------Equation 3.5

y – component of the Navier – Stokes equation

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$ ------- *Equation* 3.6

z – component of the Navier – Stokes equation

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$ ------*Equation* 3.7

The model being two dimensional the z- component of Navier Stoke's equation is not applicable to present problem.

Initial Condition:-

At $t\leq 0:\vec{V}=0$

Velocity Boundary Condition:-

On the casing wall: $\vec{V} = 0$

On the Gear Surfaces: $\vec{V} = \vec{V}_s$

Pressure Boundary Condition:-

At the Inlet Port : $P=P_i$

At the Outlet Port: $P=P_o$

3.4 Energy Equation (First Law of Thermodynamics)

$$\rho c\left(\frac{\partial T}{\partial t}+V_x\frac{\partial T}{\partial x}+V_y\frac{\partial T}{\partial y}+V_z\frac{\partial T}{\partial z}\right)=-k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)+\underbrace{2\mu\left[\left(\frac{\partial V_x}{\partial x}\right)^2+\left(\frac{\partial V_y}{\partial y}\right)^2+\left(\frac{\partial V_z}{\partial z}\right)^2\right]}_{\text{Not applicable as no viscous heating}}$$

$$\underbrace{\mu\left\{\left(\frac{\partial V_x}{\partial y}+\frac{\partial V_y}{\partial x}\right)^2+\left(\frac{\partial V_x}{\partial z}+\frac{\partial V_z}{\partial x}\right)^2+\left(\frac{\partial Vy}{\partial x}+\frac{\partial Vx}{\partial y}\right)^2\right\}}_{\text{Not applicable as no viscous heating}} \text{-------Equation 3.8}$$

Nondimensionalized form of Energy Equation.

$$\frac{\partial(c\rho T)}{\partial t}+\nabla.\left(\rho\vec{V}cT\right)=\nabla\left(k\nabla T\right)$$ ------------------------------------Equation 3.9

where

$$\nabla = del\ operator = \frac{\partial}{\partial x}\hat{i}+\frac{\partial}{\partial y}\hat{j}+\frac{\partial}{\partial z}\hat{k}$$

Temperature Boundary Condition:-

At the Inlet Port : $T=T_i$

At the Outlet Port: $T=T_o$

On the Gear sureface : $\frac{\partial T}{\partial n} = 0$

3.5 The Turbulence Kinetic Energy k-ϵ Equation

In addition to above equations for modeling turbulent flow, two more equations are required.

$$\boxed{\frac{\partial}{\partial t}(\rho k)+\nabla(\partial k\vec{V})=\nabla\left[\left(\mu+\frac{\mu_i}{\sigma_k}\right)\nabla k\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k}$$ --------- Equation 3.10

$$\boxed{\frac{\partial}{\partial t}(\rho \varepsilon)+\nabla(\partial \varepsilon\vec{V})=\nabla\left[\left(\mu+\frac{\mu_i}{\sigma_k}\right)\nabla \varepsilon\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_k} - Equation\,3.11$$

The turbulent (Eddy) viscosity, μ_i is calculated by combining k and ϵ as follows:

$$\boxed{\mu_i=\rho C_\mu\frac{k}{\varepsilon}}$$ -- Equation 3.12

Typically, these values are initially set to:

$C_{1\varepsilon}=1.44,\ C_{2\varepsilon}=1.92,\ \sigma_k=1.0,\ \sigma_\varepsilon=1.3$

Navier-Stokes equations have been known for over a century. However, the analytical investigation of these equations, even in their reduced or simplified forms, is still an active, vital area of research. The equation of continuity and two Navier-Stokes equations for velocity components *u, v,* are required to be solved (for two dimensional incompressible flow analysis) for three unknowns i.e. velocity components *u, v* and pressure P .The equations are coupled, meaning that some of the variables appear in all three equations. Hence the set of differential equations must be solved simultaneously for all three unknowns. The boundary conditions for all variables must be specified at all boundaries of the flow domain, including inlets, outlets, moving wall and stationary wall. In addition to this the flow is “Unsteady flow”, as such the time variable must be taken into account as flow field changes with respect to time.

CHAPTER 4

MODELING USING GAMBIT

4.1 Mesh Generation

The 3-D model of gear pump was made in Solid Works. After aligning the gears properly a file was created in *.step format. This *.step file was imported in Gambit and unwanted extra lines and faces were deleted and a 2 D model was generated using various gambit commands. Once the model was created a meshing operation was carried out .The boundary conditions and fluid continuum was defined. The meshed model is shown in Fig. 4.1.

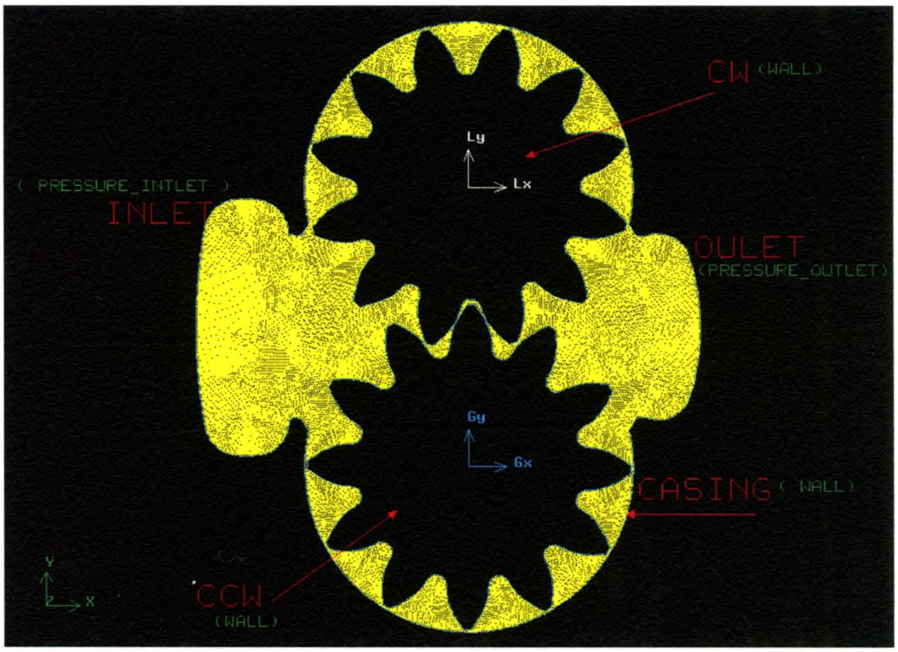


Fig.4.1 –Meshed model with boundary conditions

The text in the red indicates the boundary names and the text in green color indicates the boundary type. The 2-D Model was created with a large gap between CW and CCW gears as well as annulus gap between gears and the casing. The model with large gap (250 micron) is shown in Fig.4.2. The mesh of the large gap model was exported to fluent and flow analysis was done. It was noted during the analysis that the flow was reversed with a RPM range from 500 to 3000. This means that the reversed flow was created solely by the strong adverse pressure gradient, not by motion of gear. Trials were conducted for various inlet pressures, outlet pressures and RPM values and the flow reversal was continually observed.

In order to correct and overcome this fatal error, the gap between the driving and driven gears and the gap between gears and the casing was reduced repeatedly and the model was remeshed for processing in Fluent to generate a motion driven flow.

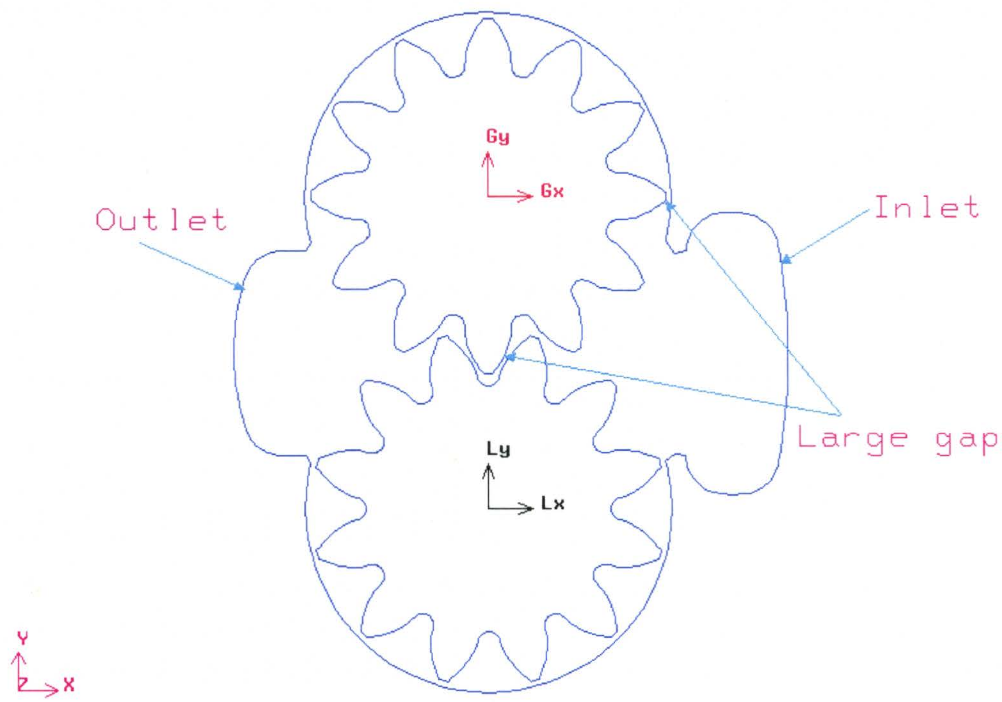


Fig 4.2- Two Dimensional Model with large gap

A successful model was made with the gap reduced to 50 micron as discussed in detail in Chapter 6.

The grid can be structured or unstructured cells and shape of the cells can be:-

- Triangular
- Quadrilateral
- Tetrahedral
- Hexahedral

A structured grid consists of planner cell with four edges (2-D) or volumetric cells with six faces (3-D). An unstructured grid consists of cells of various shapes, but typically they are triangles (2-D) and tetrahedrons or hexahedrons (3-D). A triangular mesh scheme with mesh "Interval size" of 0.01 was applied.

4.2 Boundary Conditions and Fluid Continuum

While solving the equations of motion using the CFD, prescribing appropriate boundary conditions are required to obtain accurate results. The boundary conditions were assigned inlet, outlet and stationary wall and gears in Gambit as follows.

Wall Boundary Conditions

Since the fluid can not pass through the solid wall, the normal component of the velocity is set to zero, relative to wall boundary condition. Also due to no slip condition, the tangential component of velocity at a stationary wall is set to zero. All edges of the model are set as wall. The moving walls are assigned a motion that is defined by user defined function. The user defined function assigns the angular motion to the gears.

Inlet Boundary Condition

A pressure inlet boundary condition was selected in gear pump analysis. Under this condition total pressure is specified along the inlet face through which flow is coming into the computational domain from pressurized source. No Inlet velocity was prescribed.

Outlet Boundary Condition

The fluid flows out of the computational domain through the pressure outlet. A static pressure of 2500 psi and 3500 psi was specified was pressure outlet boundary condition. No outlet velocity is prescribed.

Fluid Continuum

It is a continuous fluid media and fluid domain is defined as fluid zone. The fluid continuum is defined in the Gambit before the mesh is exported to CFD solver. The properties of fluid are defined in CFD solver. The fully defined model is solved using the solver and then results are interpreted for their correctness.

4.3 Quality Check of Mesh

The model was checked for mesh quality and the report from gambit is reproduced below:-

Command>face check "CAS" quality

Summarizing EQUISIZE SKEW of 2D elements measured for 1 meshed face:

Face CAS meshed using Triangle scheme and Interval count size of 0.01. The smaller the interval count size, finer the mesh will be.

Table-4.1 Mesh Quality data

From value	To value	Count in range	% of total count (59026
0	0.1	58018	98.29
0.1	0.2	798	1.35
0.2	0.3	148	0.25
0.3	0.4	49	0.08
0.4	0.5	12	0.02
0.5	0.5	1	0.00
0.6	0.7	0	0.00
0.7	0.8	0	0.00
0.8	0.9	0	0.00
0.9	1.0	0	0.00
0	1.0	59026	100.00

- Measured minimum value: 4.30276e-008
- Measured maximum value: 0.500746
- 0 out of 1 meshed face failed mesh check for skewed elements (EQUISIZE SKEW > 0.97).
- 0 out of 1 meshed face failed mesh check for inverted elements.

The details of the flow domain are shown in Table 4.2

Table -4.2 Flow domain

Domain Extents: (Scales Model)
x-coordinate: min (m) = -3.735255e-002, max (m) = 3.170445e-002
y-coordinate: min (m) = -2.247898e-002, max (m) = 6.057898e-002
Volume statistics:
minimum volume (m3): 1.252917e-008
maximum volume (m3): 5.133250e-008
total volume (m3): 1.739831e-003

The details of the analysis are being presented in forgoing chapters.

4.4 The equivalent lengths of inlet and outlet port

The equivalent lengths were calculated based on inlet and outlet dimensions of gear pump P-315 and assuming the unit depth (2D effect) which will give the identical cross sectional areas between the model and actual pump

Table 4.3 Inlet and outlet dimensions

Description	Length
Inlet	1.0724”
Outlet	0.3975”

While making the model in gambit the above lengths were assigned to inlet and outlet respectively. Once the quality mesh was generated it was exported for reading and further analysis in Fluent.

CHAPTER 5

METHODOLOGY OF NUMERICAL SOLUTION

Computational Fluid dynamics software (CFD) performs the fluid flow analysis after the mesh generated is imported in to the Fluent. The boundary conditions, selection of the fluid, refining the grid, checking the grid and executing the solution are performed in CFD software (Fluent 6.2.16). For the analysis of the gear pump flow, Solid works was used to construct the model. A Gambit 2.2.30 was used for preprocessing and mesh generation. Fluent was used as CFD solver.

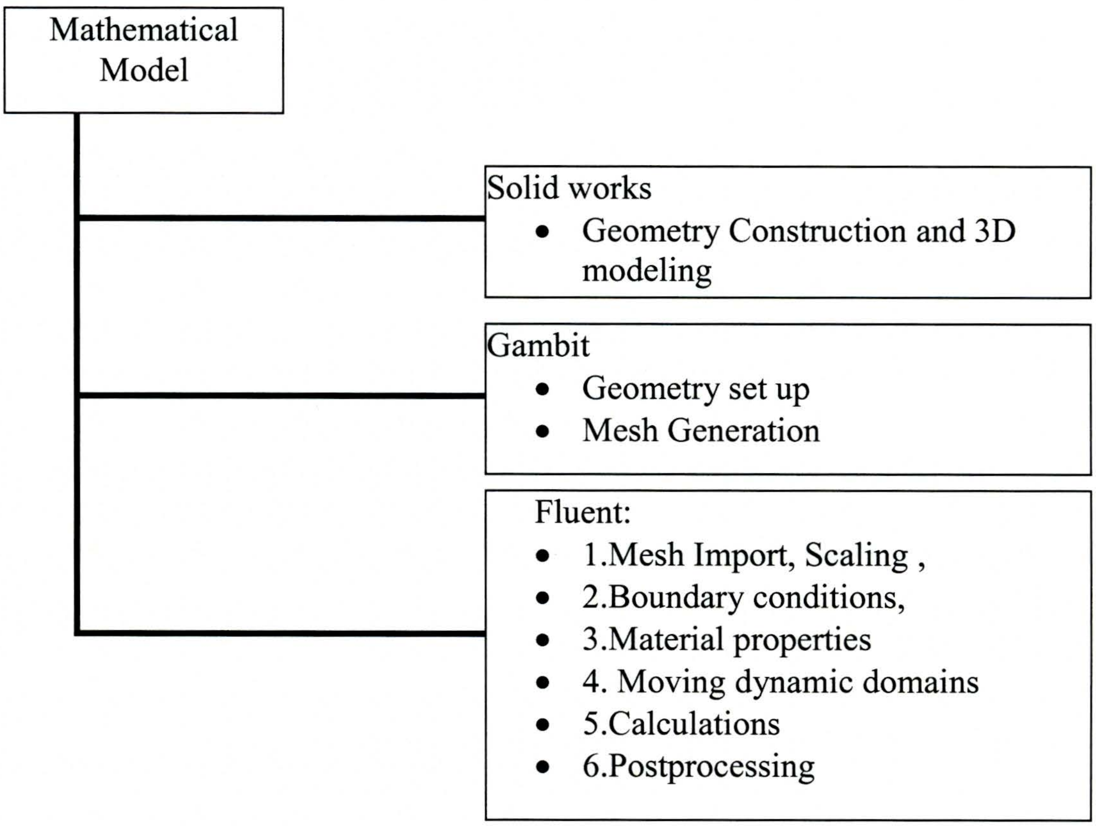


Figure 5.1- Steps in CFD analysis.

5.1 CFD Solver –Fluent

The Fluent is finite volume based algorithm for the simulations and modeling of fluid flow and heat transfer. The Gear pump problem involves unsteady flow. The viscous flow model, unsteady solver was selected for the problem set up.

User defined function (UDF) was used to control the moving dynamic mesh of driven, driving gears and stationary casing. Smoothing and re-meshing techniques were used for the gears as well as casing. The rotation of the gears was defined and controlled by UDF as a rotating rigid body that utilizes a macro specific to the dynamic mesh model. The casing was defined as stationary type boundary (wall Boundary).

5.2 Solution Method Adopted in Fluent.

The segregated solver was used in fluent for solving the gear pump problem [8]. The segregated solver is a solution algorithm used by FLUENT. Using this approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. An educated guess for initial values will definitely reduce the number of iterations and guarantees faster convergence.

In addition, the accuracy of the solution depends on:

- How accurate is the physical model
- Grid resolution
- Problem parameters set.

The equations are expressed through integral formulation. The flow domain is discretized into small cells. On each small cell the fluid equations are applied to calculate the flow field variables at nodes of cell.

5.3 Linearization of Equations

In both the segregated and coupled solution methods the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized may take an '*Implicit*' or '*explicit*' form with respect to the dependent variable (or set of variables) of interest. In 2D solver only implicit option is available and hence it is used. By implicit or explicit we mean the following [9]:

- Implicit: For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

- Explicit: For a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

In the segregated solution method each discrete governing equation is linearized implicitly with respect to that equation's dependent variable. This will result in a system of linear equations with one equation for each cell in the domain. Because there is only one equation per cell, this is sometimes called a "scalar" system of equations. A point implicit (Gauss-Seidel) linear equation solver is used in conjunction with an algebraic multi grid (AMG) method to solve the resultant scalar system of equations for the dependent variable in each cell. For example, the ***x***-momentum equation is linearized to produce a system of equations in which ***u*** velocity is the unknown. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated ***v***-velocity field.

5.4 Dicretization and First Order Upwind Scheme:

When first-order accuracy is desired, quantities at cell faces are determined by assuming that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell; the face quantities are identical to the cell quantities. Thus when first-order up winding is selected, the face value Φf is set equal to the cell-center value of Φ in the upstream cell.

Fluent uses the control volume technique to convert the governing differential equations into algebraic equations. In the control volume technique governing equations are integrated over each control volume (cell). This reduces the differential equation into discrete equation which conserves each quantity on a control volume basis.

Integral form for a control volume

$$\oint \rho\phi\bar{\upsilon}.d\bar{A} = \oint \Gamma_{\Phi}\nabla\Phi.d\bar{A} + \int_{v} S_{\phi}dV ------------------------- Equation\, 5.1$$

where,

$\rho = Density$

$\bar{V} = Velocity\, vector(u_{\hat{i}} + v_{\hat{j}} \rightarrow in\, 2D)$

$\bar{A} = Surface\, area\, Vector$

$\Gamma_{\Phi} = Diffusion\ coefficient\ for\, \Phi$

$\nabla\Phi = Gradient\ of\, \phi = \left(\frac{\partial\phi}{\partial x}\right)\hat{i} + \left(\frac{\partial\phi}{\partial y}\right)\hat{j}\, in\, 2D$

$S_{\phi} = source\, of\ \phi / unit\, volume$

The above equation is applied to control volume (cell) in the computational domain. The discretization on a given cell gives:-

$$\sum_{f}^{N_{faces}} \rho_f \bar{V}_f \phi_f \bar{A}_f = \sum_{f}^{N_{faces}} \Gamma_{\phi} (\nabla\Phi)_n \bar{A}_f + S_{\phi}V --------------------- Equation\, 5.2$$

Where N_{faces} = Number of faces enclosing the cell

$\phi_f = value\ \phi\ of\ convected\ through\ face\ \mathrm{f}$

$\rho_f \overline{v}_f \overline{A}_f = mass\ flux\ through\ the\ face$

$\overline{A}_f = Area\ of\ face, f\ |A|\left(=\left|A_x\hat{i} + A_y\hat{j}\right|\right) in\ 2D$

$(\nabla\phi)_n = Magnitude\ of\ \nabla\phi\ normal\ to\ face\ f$

$V = Cell\ volume$

5.6 Under Relaxation Factor

While solving the nonlinear equations the Fluent changes the variable **Φ**. This is achieved by under relaxation factor. Under –relaxation factors slow down the rate at which solution changes during the iteration and in transient solutions increase the Courant number. This means convergence is slowed down depending upon the value of α.

$$\phi_{new} = \phi_{old} + \alpha\left(\phi_{calc} - \phi_{old}\right) \text{-------------------------- } Equation\ 5.3$$

$Where\ \alpha = Under\ relaxation\ factor$

Courant number (C) is the ratio of a time step to a cell residence time.

$$C = \frac{\Delta t}{\Delta x_{cell} / u_{fluid}} \text{-------------------------------- } Equation\ 5.4$$

CHAPTER 6

DISTRIBUTION OF PRESSURE AND VELOCITY

6.1 Pressure and Velocity Analysis.

The gear Pump was run for different speeds and outlet pressure settings. Pump was run at following conditions:

1. Case 1-1500 rpm, Outlet pressure 2500 psi, Inlet pressure=30 psi
2. Case 2-1500 rpm, Outlet pressure 3500 psi, Inlet pressure=30 psi
3. Case 3-2000 rpm, Outlet pressure 2500 psi, Inlet pressure=30 psi
4. Case 4-2000 rpm, Outlet pressure 3500 psi, Inlet pressure=30 psi
5. Case 5-2500 rpm, Outlet pressure 2500 psi, Inlet pressure=30 psi
6. Case 6-2500 rpm, Outlet pressure 2500 psi, Inlet pressure=30 psi
7. Case 7-3000 rpm, Outlet pressure 2500 psi, Inlet pressure=30 psi
8. Case 8-3000 rpm, Outlet pressure 3500 psi, Inlet pressure=30 psi

Initially a three dimensional model was also prepared and tetrahedral and cooper meshing methods were tried on the model with 250 micron annular and 200 micron gap between teeth. This meshing technique could generate skewness of 0.85. The model failed while analyzing in Fluent as it generated the Negative cell volume. The model was remeshed in gambit with different cooper meshing scheme and cell size (0.02) and better skewness of 0.75 was achieved. The model mesh was exported to fluent and simulation was run. The model could run without generating any negative volume, however, the flow reversal was observed. As an additional experiment the above mentioned gaps were reduced to bear minimum. However, it was very difficult task to accommodate the proper mesh. As gaps were narrowed a fine mesh count was used to mesh the model. The total cell count of this model was more than 20 million cells and the mesh quality reported in gambit was of 0.90 skewness. The mesh file generated is too large to be handled on computers available. Considering these difficulties and time required to generate a good mesh quality with reasonable mesh file size, analysis was completed with two dimensional model.

With the simulation of two dimensional model in Fluent, it was observed that flow was reversed for all above cases. Initially large annular gap of 200 microns was maintained between the casing and the gears and 150 micron gap between the engaging teeth of gears. As an experiment the annular gap between the casing and gear was reduced to 150 microns and model was meshed with triangular mesh of 0.002. This revised mesh also failed in giving the flow in forward direction. The annular gap was further reduced to 150 micron and model was meshed with 0.001 size triangular mesh. With this mesh also reversed flow was observed. The close inspection of the velocity vectors reveled that the large flow reversal through the gap between the gear teeth engagement. Based on this inspection it was decided to reduce the gap between the engaging gear teeth. This gap was brought down to 50 micron and model was meshed with different triangular mesh sizes. A mesh with triangular cell size of 0.01 gave a good mesh quality. This mesh generated two elements in the gap where teeth of gears are engaging. This is a minimum requirement of elements in a gap for running MDM (Moving Dynamic Mesh) and re-meshing option in Fluent software. Another method of meshing was tried for accommodating more layers of meshing in both the gaps i.e. annular gap and the gap between the teeth. The flow domain was divided in number of faces and fine triangular mesh size of 0.001count was generated. However this gave a large mesh count of 473931 elements and it does not result in flow in right direction.

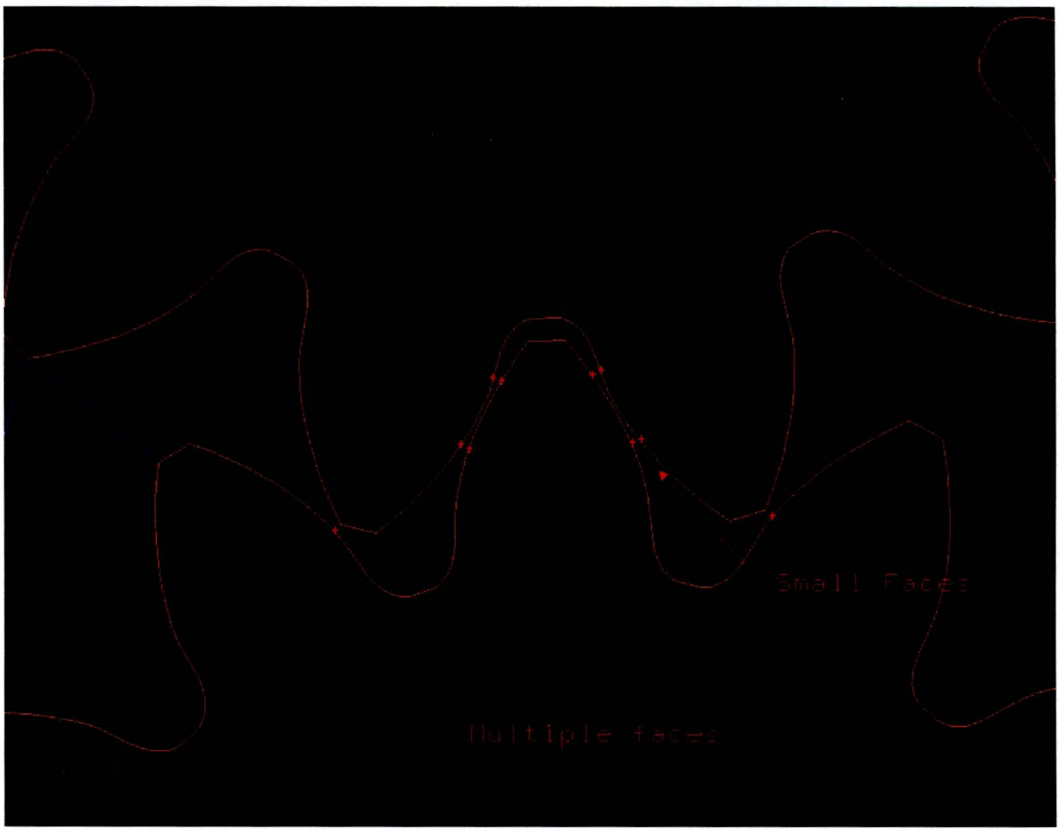


Fig 6.1 Multiple faces

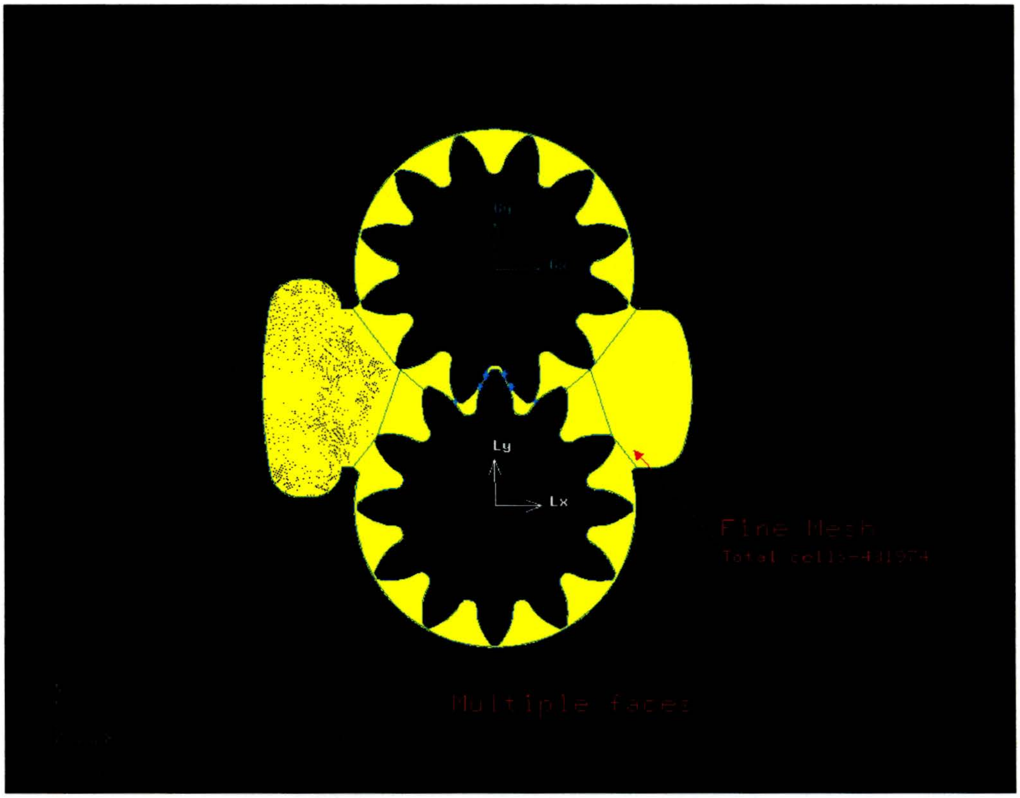


Fig 6.2 Mesh with multiple faces.

The flow reversal was observed at 1500 rpm and outlet pressure of 3500 psi. It was a strange phenomenon and primarily it looks like Fluent can not generate the flow at lower rpm by the motion of the gears and flow is governed due to adverse pressure gradient and hence the flow is from high pressure to low pressure i.e. from outlet to inlet.

To strengthen the above argument, the pump was run at 1500 rpm and the pressure was decreased to 2500 psi and the flow direction was changed to normal i.e. from inlet to outlet. The Pressure contours and velocity vectors and the average velocity plots are presented in forgoing pages.

The pressure patterns and velocity patterns were observed for any possible cavitation phenomena. The average velocity was recorded to check the flow through pump as described in the last chapter.

Table 6.1: – Summary of Pressure and velocities.

Outlet Pressure (psi)	Rotational speed of Gears rpm N	Pressure Contour range	Average velocity (m/s)
2500	1500	-163 to 2640	1.46
	2000	-75.21 to3220.43	1.96
	2500	-385 to2730	2.35
	3000		2.96
3500	1500	-130.04 to 3619.37	Flow reversed
	2000	-166.18 to 3768.87	1.96
	2250	- 22.83 to 4360.5	2.2
	2500	-334.19 to 3701.51	2.35
	3000	-83.17 to 7327.97	2.96

6.2 CASE 1: 1500 RPM, INLET=30 PSI, OUTLET=2500 PSI, FLOW FORWARD

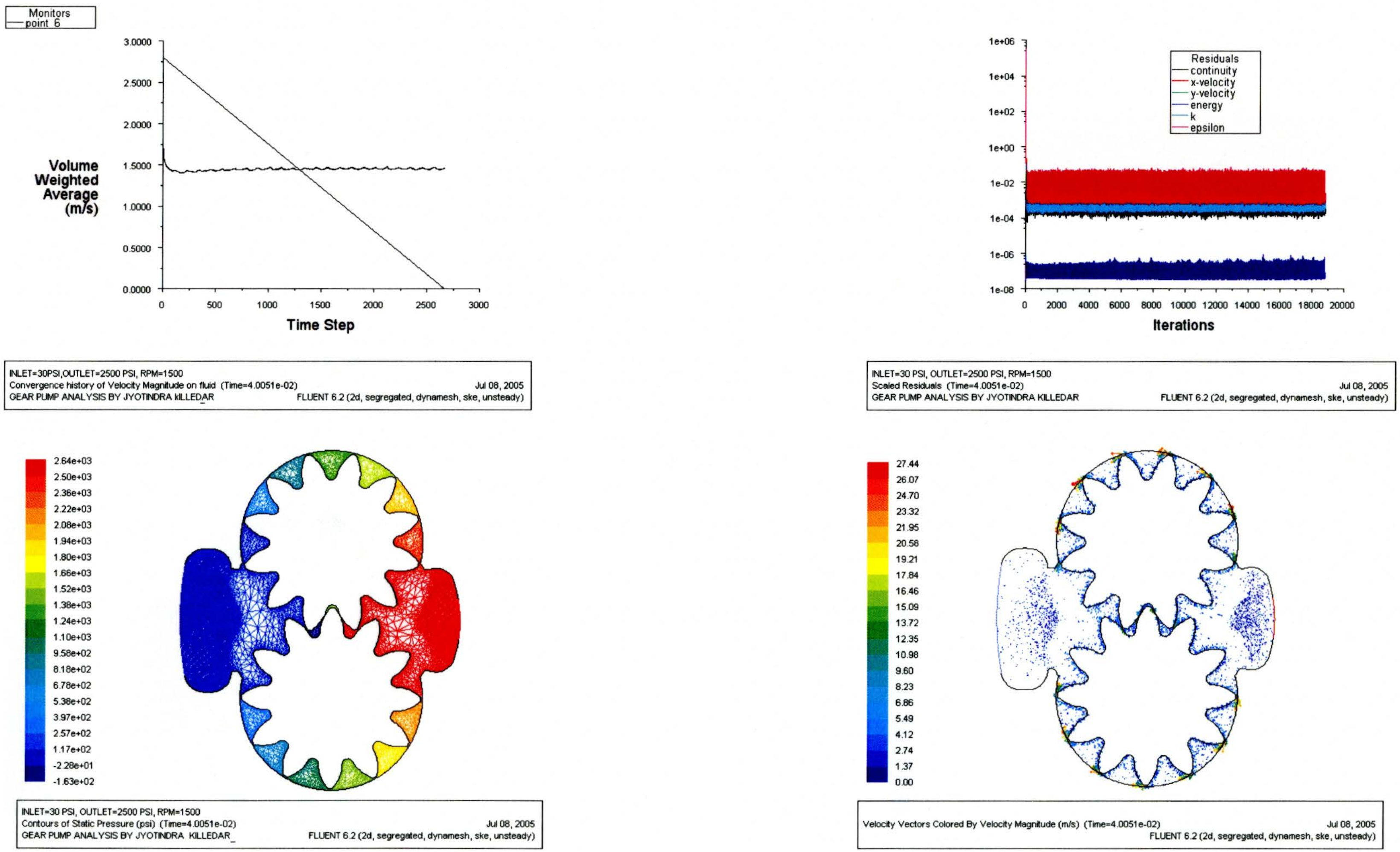


Fig 6.3 Plots of Case 1

6.3 CASE 2: 2000 RPM, INLET=30 PSI, OUTLET=2500 PSI, FLOW FORWARD

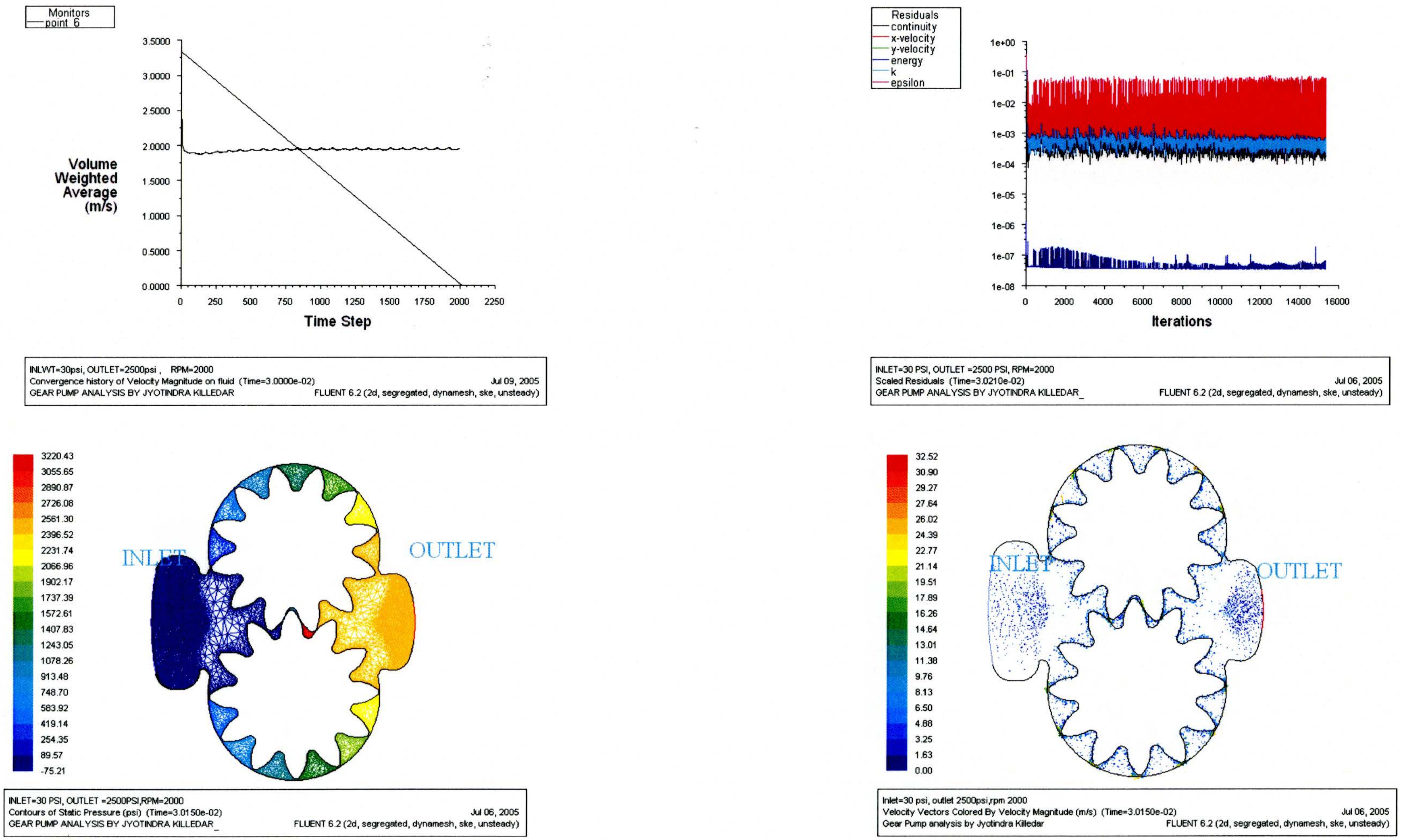
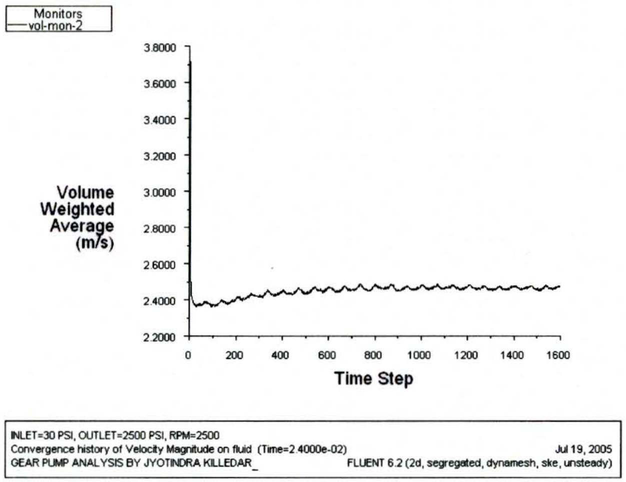
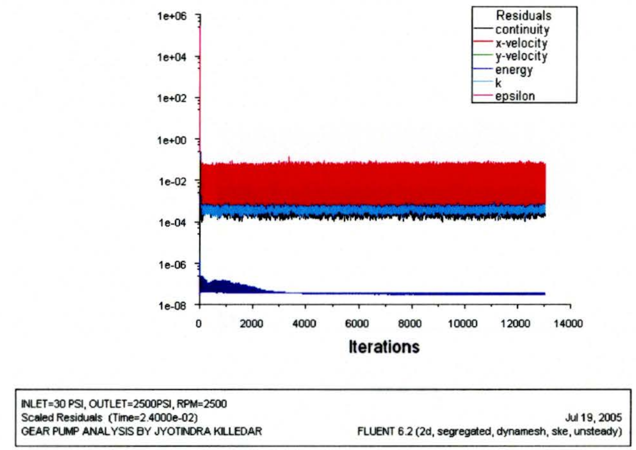
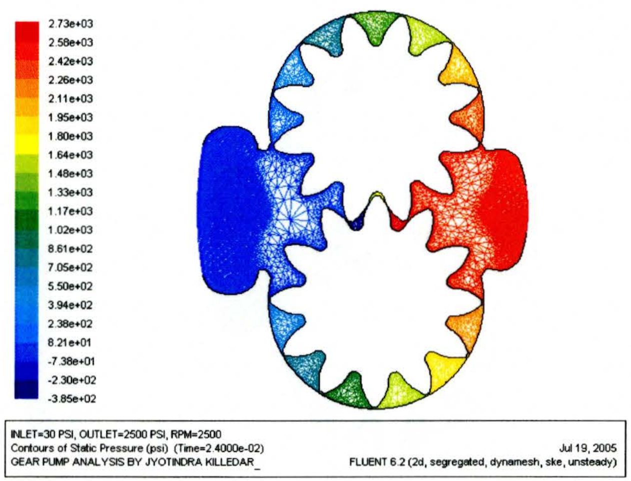


Fig 6.4 Plots of Case 2

6.4 CASE 3: 2500 RPM, INLET=30 PSI, OUTLET=2500 PSI, FLOW FORWARD







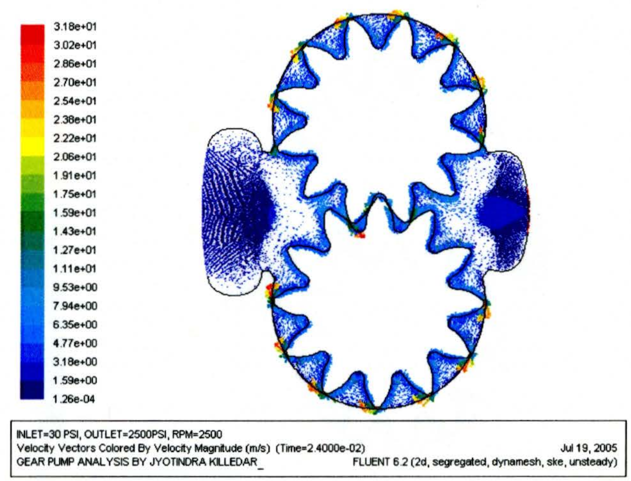


Fig 6.5 Plot of case 3

6.5 CASE 4: 3000 RPM, INLET=30 PSI, OUTLET=2500 PSI, FLOW FORWARD

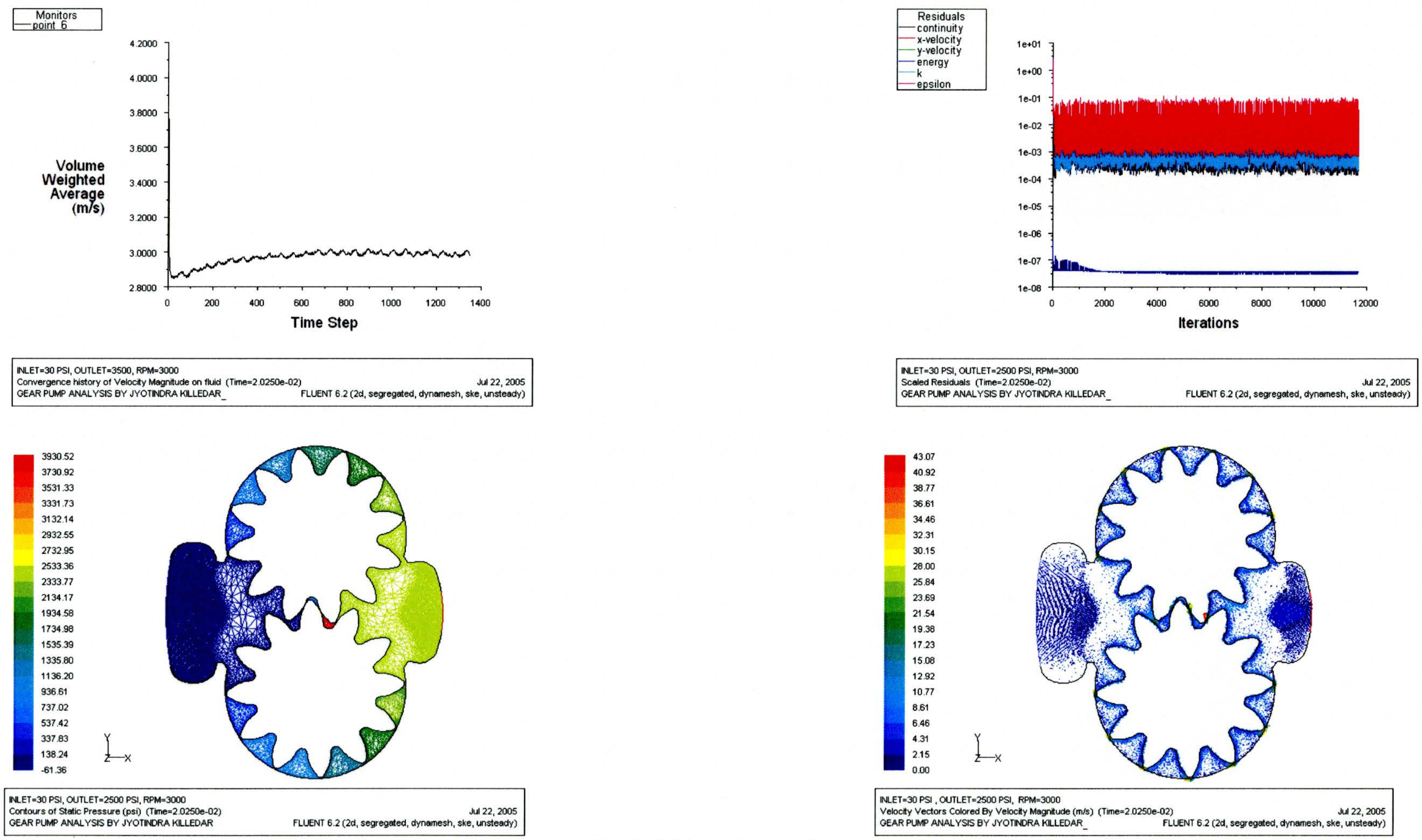


Fig 6.6 Plot of case 4

6.6 CASE 5: 2000 RPM, INLET=30 PSI, OUTLET=3500 PSI, FLOW FORWARD

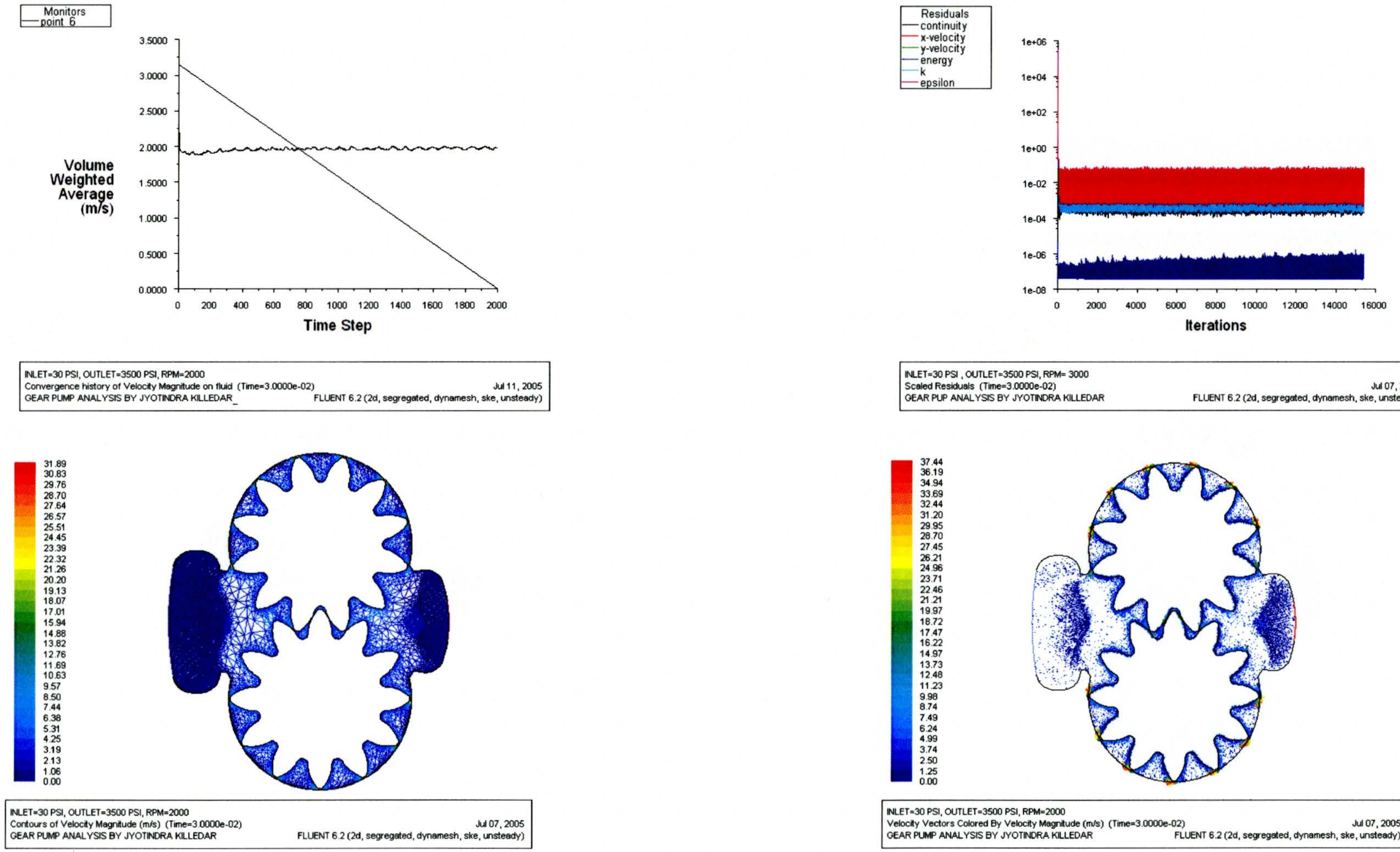
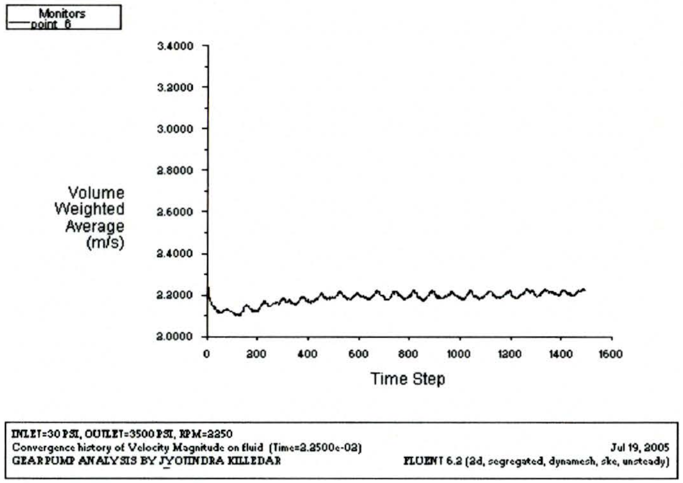
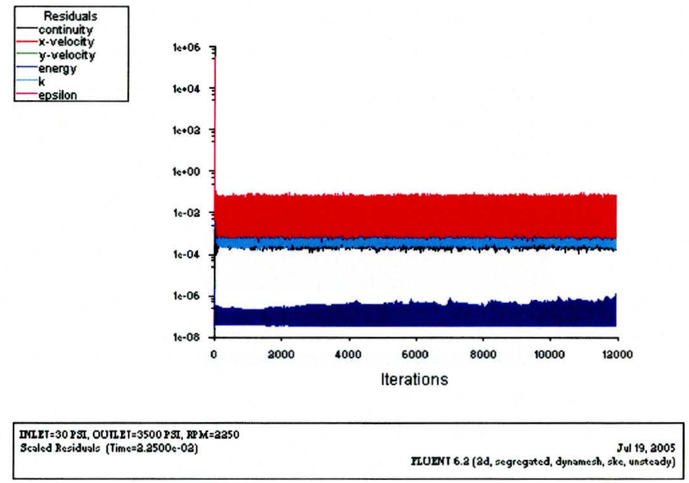
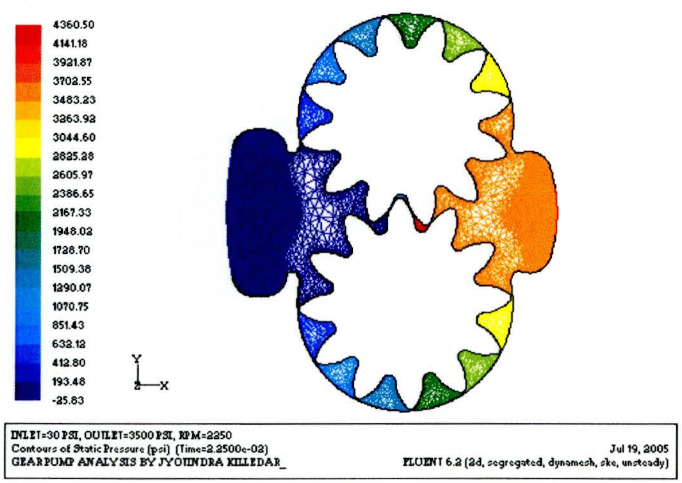


Fig 6.7 Plot of case 5

6.7 CASE 6: 2250 RPM, INLET=30 PSI, OUTLET=3500 PSI, FLOW FORWARD







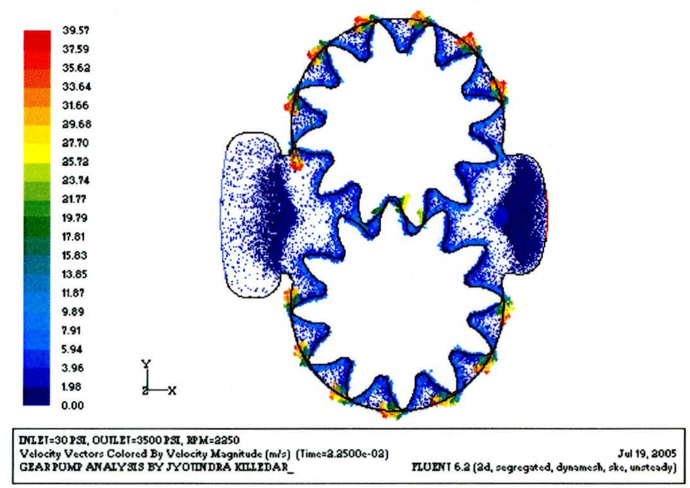


Fig 6.8 Plot of case 6

6.8 CASE 7: 2500 RPM, INLET=30 PSI, OUTLET=3500 PSI, FLOW FORWARD

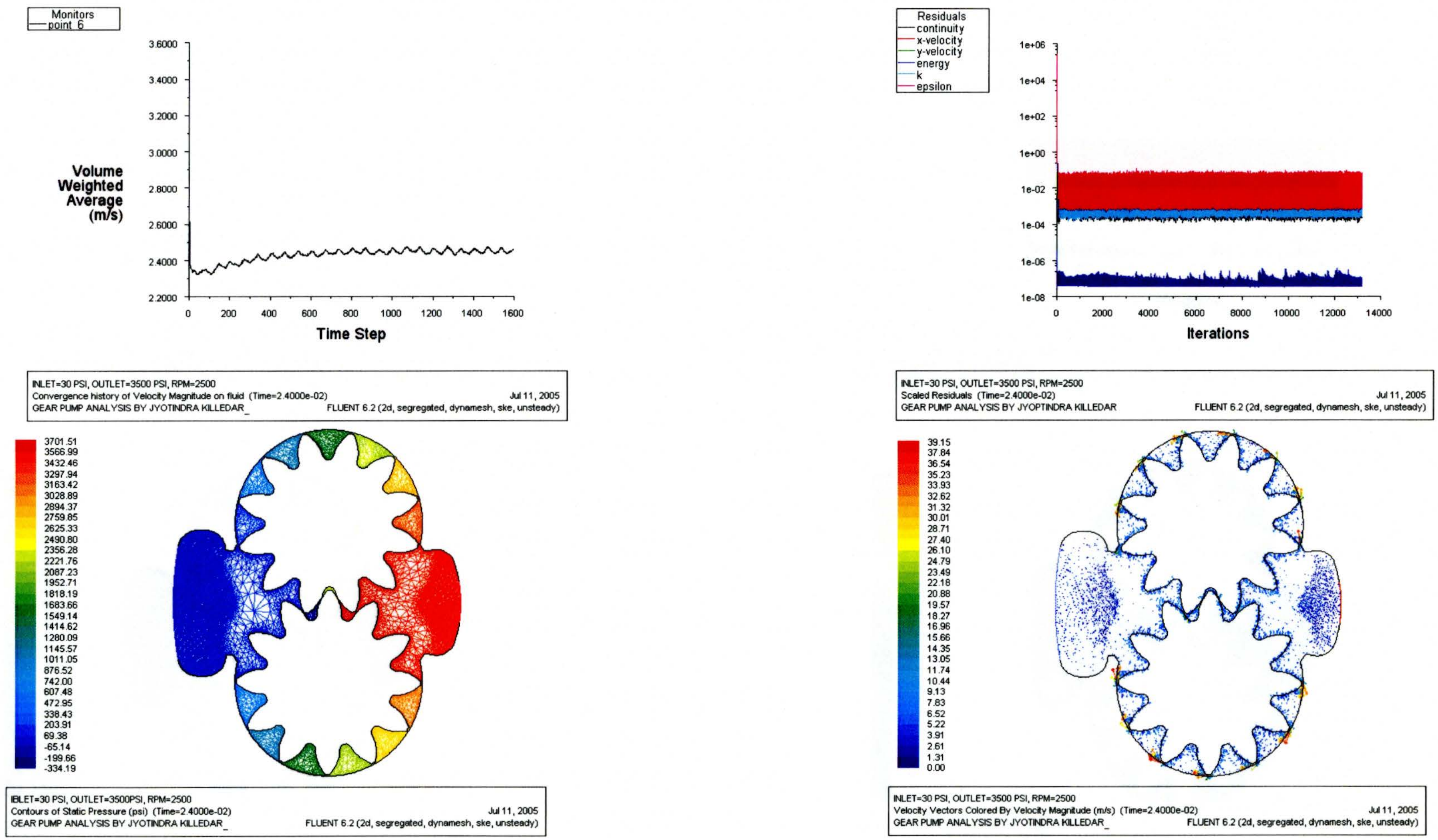


Fig 6.9 Plot of case 7

6. 9 CASE 8: 3000 RPM, INLET=30 PSI, OUTLET=3500 PSI, FLOW FORWARD

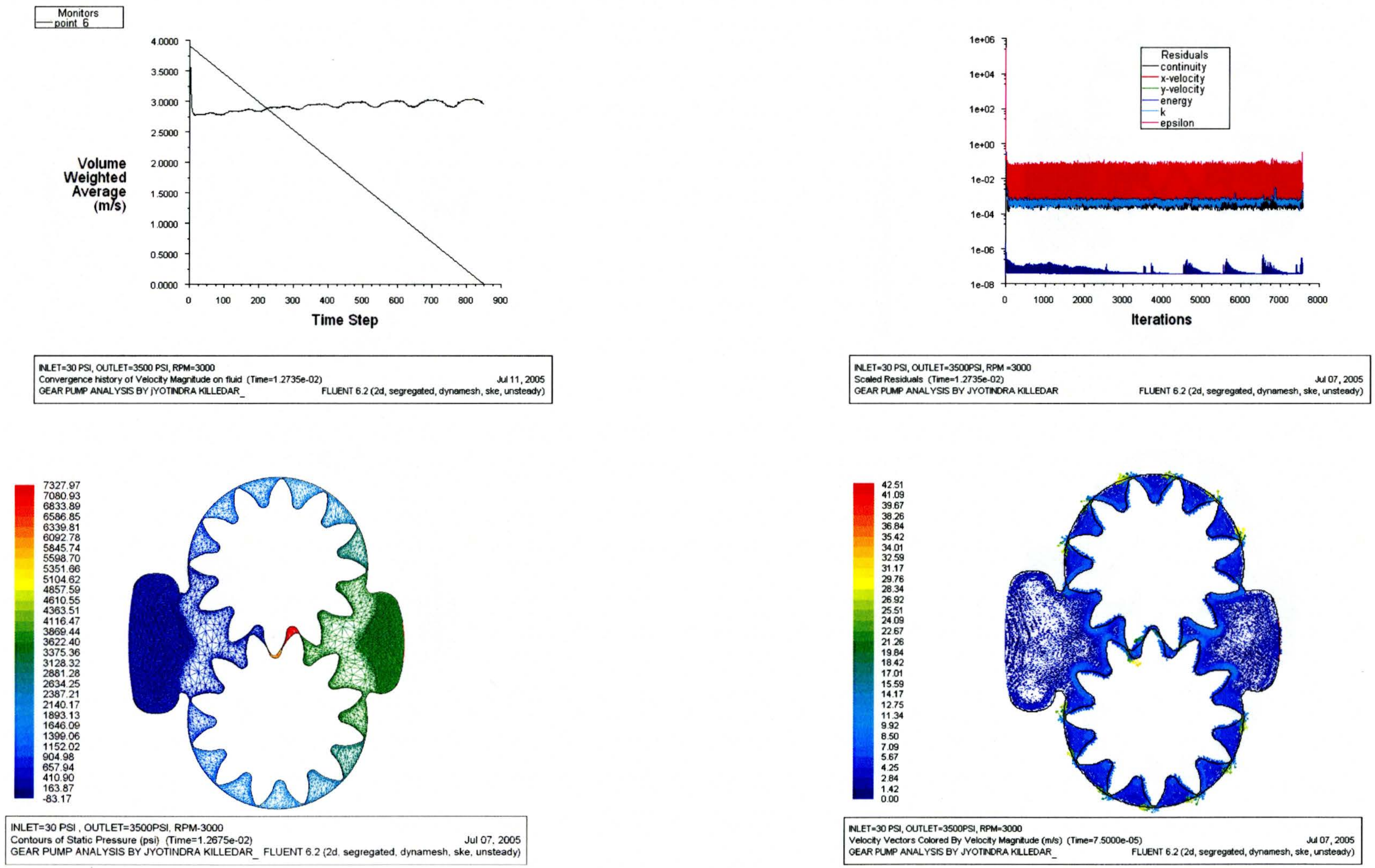


Fig 6.10 Plot of case 8

6. 10 Velocity contours fro rpm range 1500 and 2000

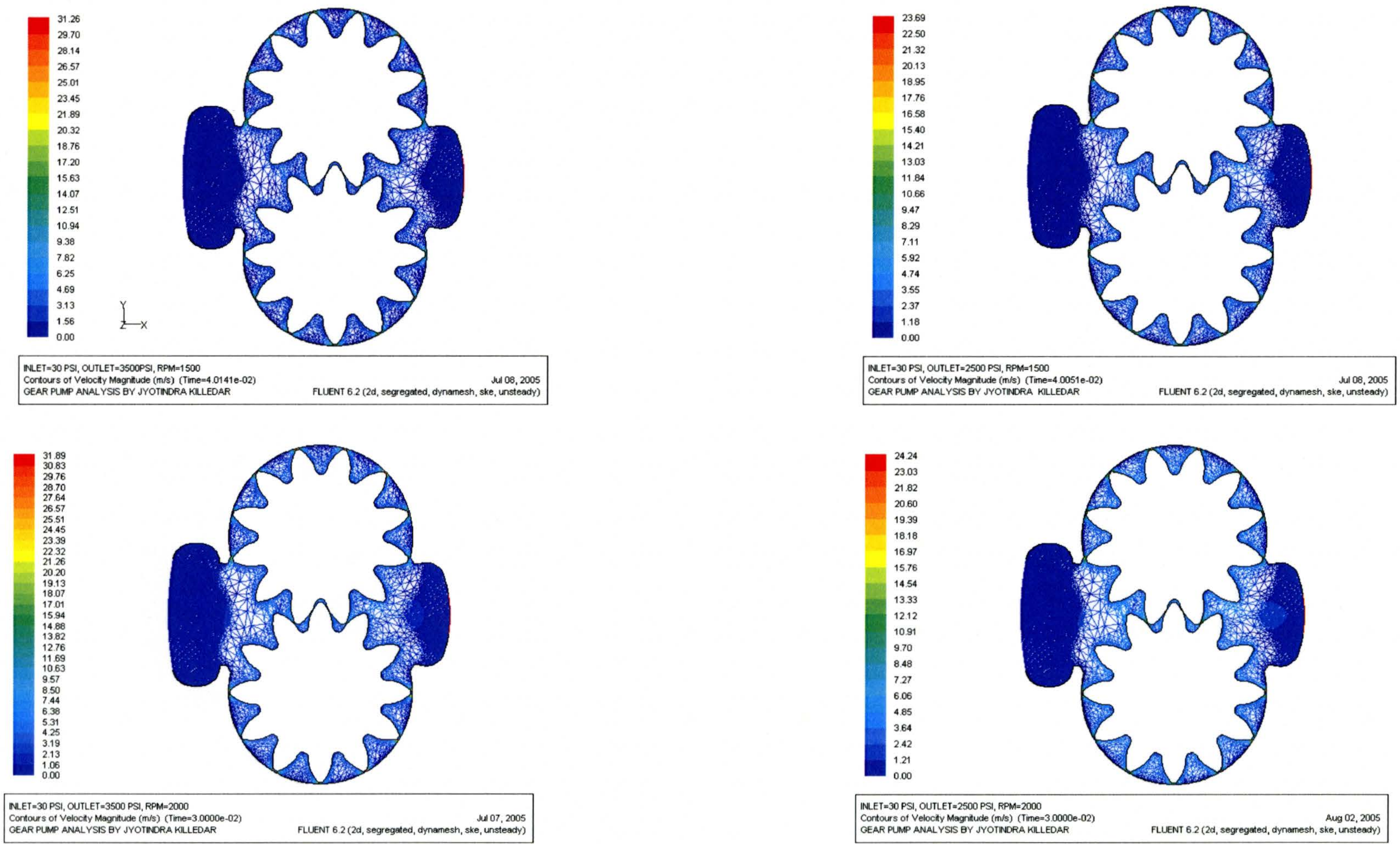


Fig 6.11 Velocity Contours for rpm 1500 to 2000

6. 11 Velocity contours fro rpm range 2500 and 3000

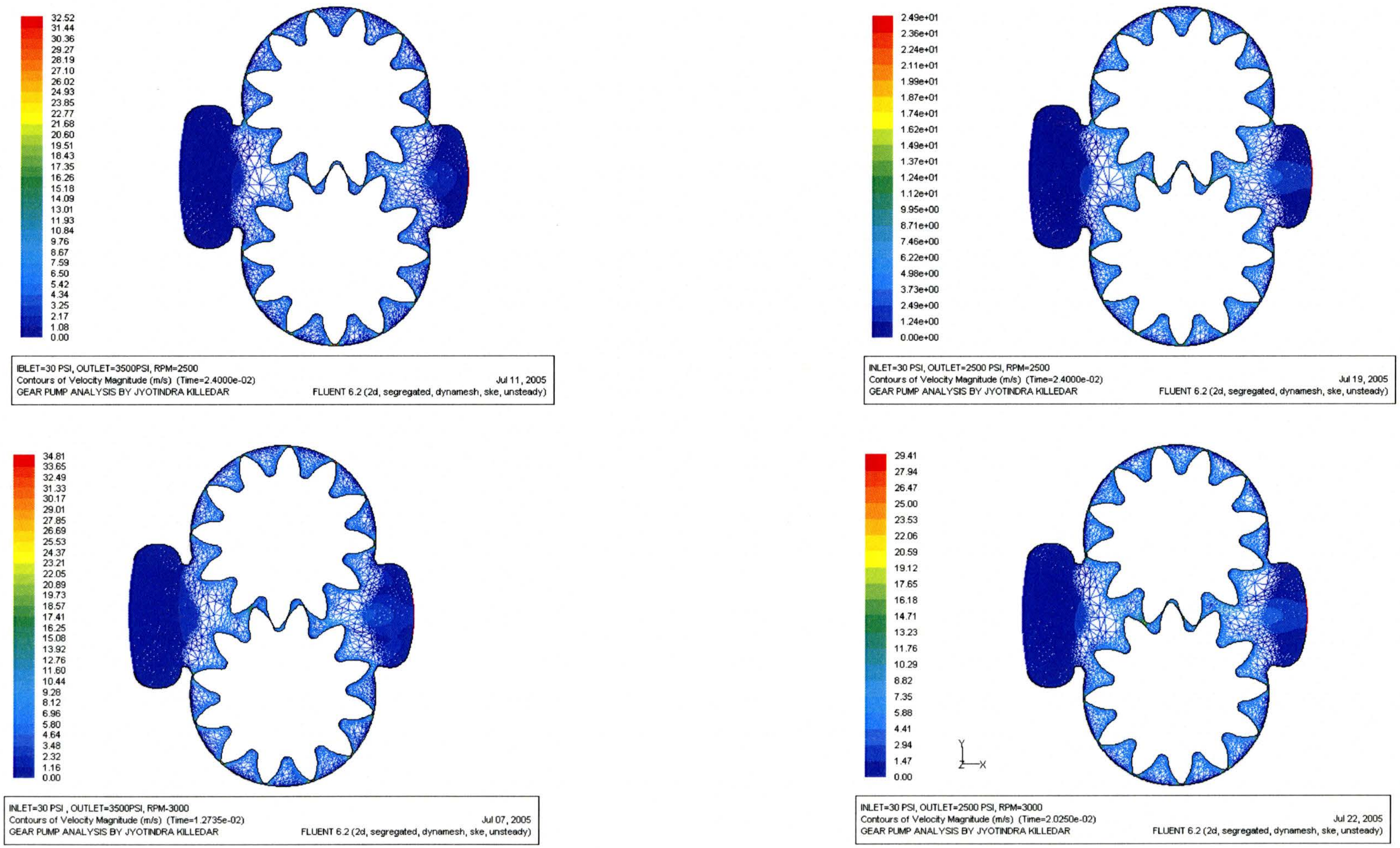


Fig 6.12 Velocity Contours for rpm 2500 to 3500

6.12 Interpretation of Results

The pressure contours and velocity vectors were studied and compared for the same rpm and different outlet pressures such as 2500 psi and 3500 psi. The velocity at 2000 rpm and 2500 psi at gear tip is in the range of 3.25 -4.88 m/s where as the velocity at 2000 rpm and 3500 psi at gear teeth surface is in the range of 3.74-4.99 m/s. The change in velocity is negligible. Even at higher speed of 3000 rpm and 2500 psi velocity is in the range of 6.53-7.83 m/s, at 3000 rpm and 3500 psi it is in the range of 7.09 -8.50 m/s measured at outer surface of gear.

Following are the observations on velocity vector.

1. It can be concluded that the velocities at the tip of gears are independent of the outlet pressure and dependent on speed of the pump at which it rotates.
2. Velocity at location close to casing wall is zero.
3. Fluid is pushed towards outlet port as the gear teeth starts closing.
4. The magnitude of velocity vectors change with respect to time domain.
5. As seen from above the velocities are independent of the output pressure hence outflow is also independent of output pressure.
6. With increase in pressure at output the fluid tries to escape through the small gap between the two meshing teeth as such velocities are high at this location.

Pressure distributions are significantly different in magnitude but indicate the similar patterns as the outlet pressure changes for the cases with similar operating speed of the gear pump.

The gear pump flow domain can be broadly divided into four zones

1. Inlet zone.
2. Zones encased by casing wall and driving gear (CCW) and driven gear (CW).
3. Zone between near where teeth of both gears mesh.(Area prone to cavitation)
4. Outlet zone.

It was observed that negative pressure exists in the zone where the meshed teeth are opening (inlet side).This is sign of the cavitation taking place. This happens when local pressure of the fluid is lower than the vapor pressure of the fluid. This results in vaporization of liquid forming vapor bubbles. The process continues and the vapor bubbles forms a liquid jet flowing into collapsing bubble. This implosion results in very high negative pressures. [11]

A symmetrical pressure distribution was observed around the gears and casing. This is attributed to both the gears having equal number of teeth. Highest pressure observed is at closing of the gear for all the cases run. The high pressure zone initiates from outlet port to the closing of the gear teeth.

6.13 Negative Pressure

It was interpreted from the above results that the cavitation is occurring at the fluid zone on the inlet side where two teeth of gears are meshing and where negative pressure contours are observed. To confirm whether Cavitation really occurs in the Gear Pump it was decided to use the “cavitation model” of Fluent. The cavitation analysis and outcome are discussed in the Chapter 8.

CHAPTER-7

FLOW ANALYSIS OF GEAR PUMP

The total output flow rates are computed based on the average velocity at the outlet port. The average volume monitor was set in fluent while analyzing the other parameters such as pressure and velocity.

7.1 Velocity Measurements.

The velocity of the outgoing fluid was measured and flow was calculated based on the outlet area.

Table 7.1 Velocity Measurement

Sr.No.	Outlet Pressure(psi)	Speed of the Pump	Velocity (m/s)	Outlet Area (in^2) A_o
1	2500	1500	1.2	0.3975
2		2000	1.96	0.3975
3		2500	2.35	0.3975
4		3000	2.96	0.3975
5	3500	2000	1.96	0.3975
6		2250	2.2	0.3975
7		2500	2.35	0.3975
8		3000	2.96	0.3975

7.2 Flow Calculation

The sample calculation for the reading above is provided below.

$$Flow(Gpm) = \frac{v(m/s)39.37(in/m).A_o(in^2)60(s/\min)}{231(in^3/gal)} --------Equation\,6.1$$

$V = velocity$

$A_o = Outlet\ area$

$$Flow(gpm)_{@3500\,psi,2000\,rpm,} = (1.9)(39.37)(0.3975)(60)/231 = 7.723$$

7.3 Speed and Flow Relation

The flow calculations were performed and a graphic relation was established between flow and speed.

Table-7.2- Flow results

Sr.No.	Outlet Pressure. (psi)	RPM N	Velocity m/s.	Unit displacement (q) in^3/in-rev	Width (w) in.	Outlet Area in^2	FLOW Gpm.	Theoretical Flow rate
		1500	1.96	1.24	1.0	0.3975		8.051
1	3500	2000	1.96	1.24	1.0	0.3975	7.97	10.735
2	3500	2250	2.2	1.24	1.0	0.3975	8.94	12.077
3	3500	2500	2.35	1.24	1.0	0.3975	9.55	13.419
4	3500	3000	2.96	1.24	1.0	0.3975	12.03	16.103

Sr.No.	Outlet Pressure(psi)	RPM N	Velocity m/s.	Unit displacement.(q) in^3/in-rev	Width (w) in.	Outlet Area in^2	FLOW Gpm.	Theoretical Flow rate
1	2500	1500	1.2	1.24	1.0	0.3975	4.88	8.051
2	2500	2000	1.96	1.24	1.0	0.3975	7.97	10.735
3	2500	2500	2.35	1.24	1.0	0.3975	9.55	13.419
4	2500	3000	2.96	1.24	1.0	0.3975	12.03	16.103

The theoretical flow is calculated by using following formula.

$$Flow(gpm) = \frac{q(in^3/in-rev).w(in).N(rpm)}{231(in^3/gal)}$$

$q = Unit\ displacment$

$w = Gear\ width$

$N = RPM\ of\ the\ Pump$

The results of the flow derived from analysis by using Fluent differ by 27% compared to the flow obtained from theoretical results. This variation can be accounted to following attributes.

The Effect Of The Two Dimensional Model.

The outlet width a length of 0.3975 in. was assigned based on the actual flow area of the pump P315 and assuming unit depth for 2 D model while preparing the model in Gambit.

Velocity Readings

The velocities considered for calculation purpose are average readings and read from volume monitor output file.

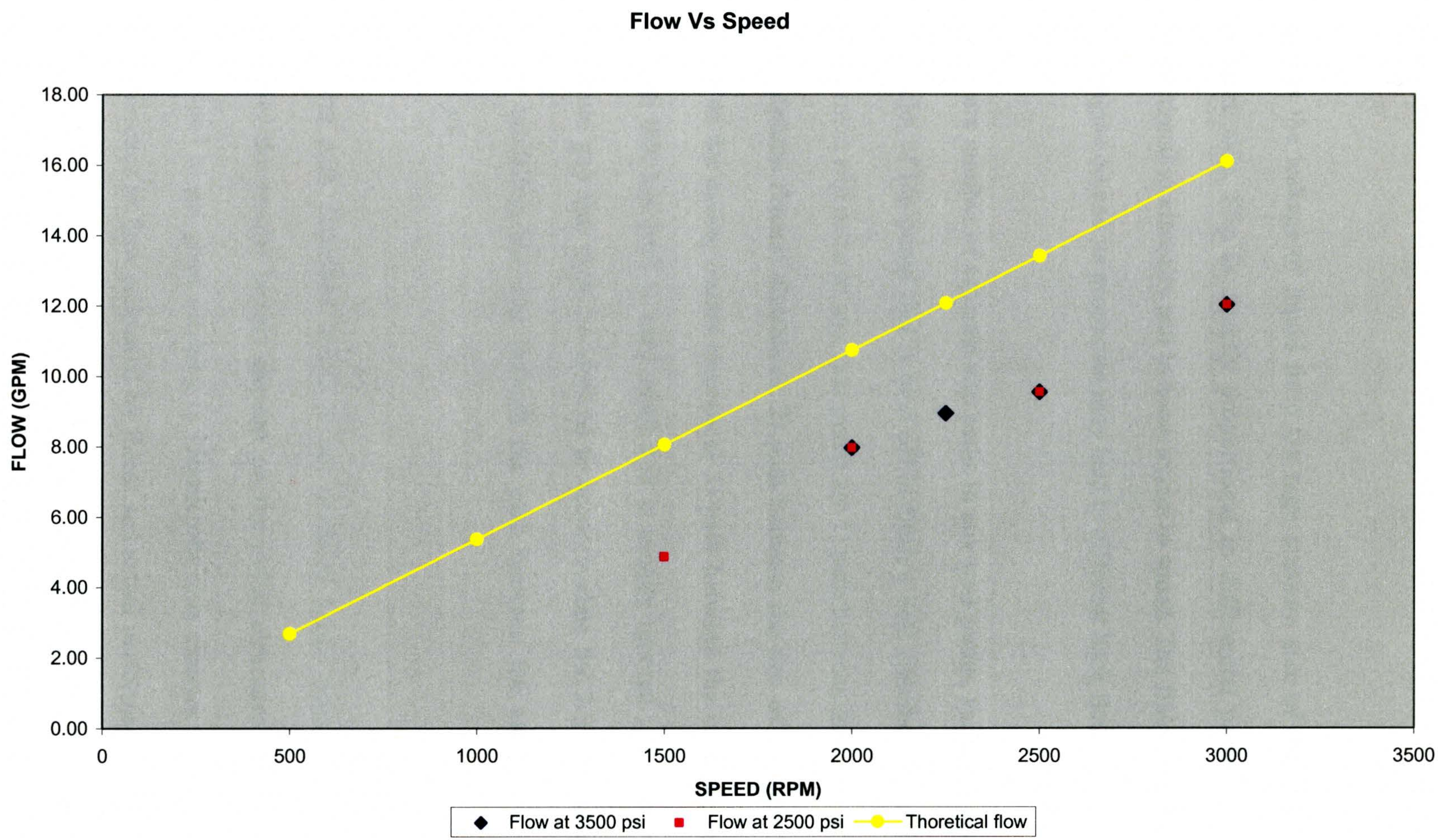


Fig 7.1 Flow and Speed relation

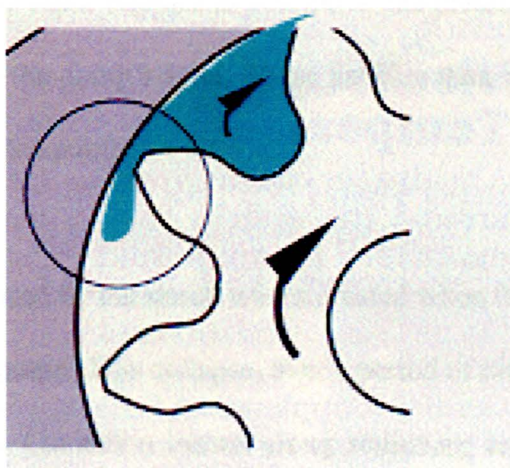
Slip Flow

Slip is the leakage of liquid from the high pressure side of the pump back to low pressure side. Slip is directly proportional to differential pressure. It is inversely proportional to viscosity and to some extent to speed. The fluid used during modeling was engine oil and its properties may lead to different back flow.

There are number of separate slip paths in any gear pump, including and liquid from the outlet of the pump that is bled off to flush a seal chamber or lubricate bearing. Three main slip paths in any gear pump are 1) path between the ends of the gears and the endplates (lateral clearance), 2) path between the tips of the gear teeth and the inside of the casing (radial clearance) 3) path between the meshing teeth. The slip through this last path is very small and is usually ignored [1]. However, due to a minimum gap that must be allowed to accommodate the 2-3 elements of mesh for Fluent modeling, the slip through the path between the teeth is expected to be significant.

The first path described above is not modeled hence does not pose a problem. However the results largely depend on the radial clearance. The radial clearance maintained to get good meshing is 150microns.This clearance probably account for the difference in flow estimated by fluent and actual performance curve [9]. The slip through this last path is very small and is usually ignored [1]. However, due to a minimum gap that must be allowed to accommodate the 2-3 elements of mesh for

Fluent modeling, the slip through the path between the teeth is expected to be significant.



Slip leakage occuring between tip of gear tooth and casing wall

Fig. 7.2- Slip leakage

7.4 Comments On The Fluid Flow

1. The flow readings are fairly accurate and can be improved by more precise 3D modeling. Fluent solver seems to be fairly accurate for flow measurement.

2. flow is independent of outlet pressure i.e. flow at 2500 psi and 3500 psi outlet pressure is the same at same speed.

CHAPTER 8

CAVITATION ANALYSIS

Cavitation is formation of vapor bubbles in a liquid as the pressure drops below the vapor pressure of the liquid in the pump's inlet. These bubbles then collapse when they reach the high pressure side of the pump.

Cavitation damage is caused by the shock waves created when the vapor cavities collapse near the elements in the pump. This collapse, over a period of time, can damage the pump and erode hard surfaces. Cavitation occurs along stationary and moving elements in a pump [9].

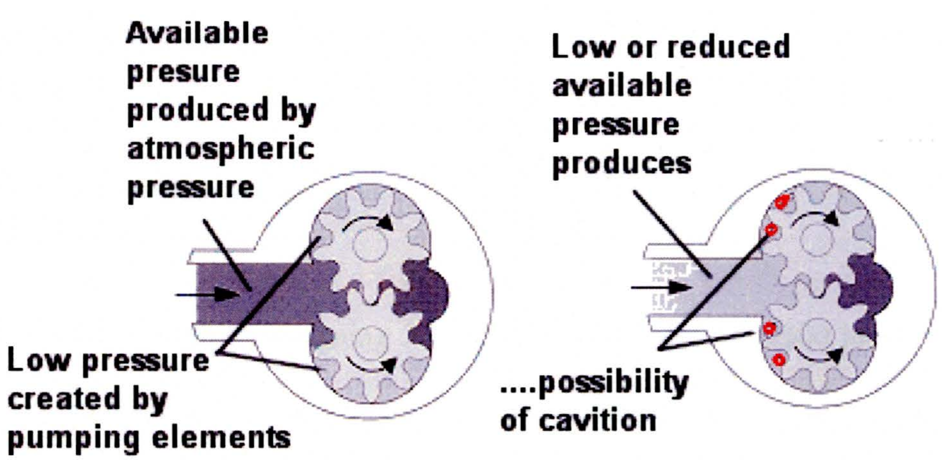


Fig 8.1 Cavitation in Gear Pump

8.1 Cavitation Modeling in Fluent.

The gear pump model used for pressure and flow analysis was used for the cavitation analysis. The "Basic Cavitation model" of Fluent was used for the analysis. In this standard two phase cavitation model following assumptions are made:

1. The system under investigation involves only tow phases (a liquid and its vapor), and a certain fraction of separately modeled non condensable gases.
2. Both bubble formation (evaporation) and collapse (condensation) are taken into account in the model.
3. The mass fraction of non condensable gases is known in advance.

The cavitation model can be used with Fluent's available turbulence model and it is compatible with dynamic mesh. The cavitation model accounts for the mass transfer between single liquid and its vapor.

8.2 Cavitation Problem Set-up:

The problem was set up by assigning 30 psi pressure at inlet and 2500 pressure at inlet and 3000 rpm rotation. The high pressure difference between inlet and outlet influenced the numerical stability of the solution and the flow reversal was observed. The outlet pressure was reduced in steps from 2500 psi to 2000 psi and 1500 psi keeping the speed of the pump constant at 3000rpm, the reversed flow pattern existed. Finally the flow was in right direction for the outlet pressure at 1000 psi.

8.3 Observations.

The Pressure, vapor volume fraction and velocity vector plots are presented below. It was observed from the "Pressure contours" and "Volume fraction-Phase 2(vapor) contours" that the pump does not cavitate under the conditions set. However the pump could not be tested for cavitation effects for higher outlet pressures and running at 3000 rpm due to software capabilities. The Fluent document highlights "In practical applications of the cavitation model, several factors greatly influence numerical stability. For instance, high pressure difference between inlet and exit (outlet), large liquid to vapor density and near zero saturation pressure all cause unfavorable effects on solution convergence"[10]. The outlet pressure in present case varies from 2500 to 3500 psi which makes the high pressure difference between inlet and outlet.

8.4 Comments on Cavitation Model

The cavitation model could not run above 1000 psi due to the limitations described above. The Fluent software 6.2.16 version was used for the analysis. The Fluent-6.2.16 capabilities for gear pump analysis (more precisely for the motion driven flow) are limited as described below:

1. The flow reversal was observed at 2000 rpm and 3000 rpm with pressure range of 2500 psi to 3500 psi while running the cavitation model. It seems that in the software algorithm the flow is governed more by pressure gradient across the inlet and outlet rather than angular momentum of gears.
2. The cavitation model does not work above 1000 psi outlet and 30 psi inlet condition at any rpm ranging from 1000 to 3000 rpm.

8.6 Problem set up Parameters for Cavitation Analysis P-317 (2D)

rpm	Rps	Rad/sec	time/rev	time step size	time steps for 1(one) revolution	time step for 10 deg rotation	max face skewness exceeds 0.95 at	**negative volume generated at**
3000	50.00	314.00	0.02	1.50E-05	1333	37		nil

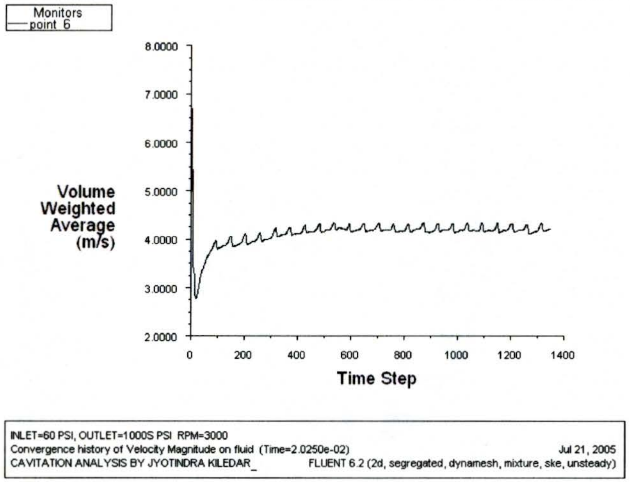
Gambit file	**C:\jyotindra\Indra.dbs(48128Kb)**	**Parameters**	
Mesh file	**IndraM.msh(3307kb)**	Inlet	30 psi
Case file	**IndraM.cas 1840**	Outlet	1000psi
		Rpm	3000

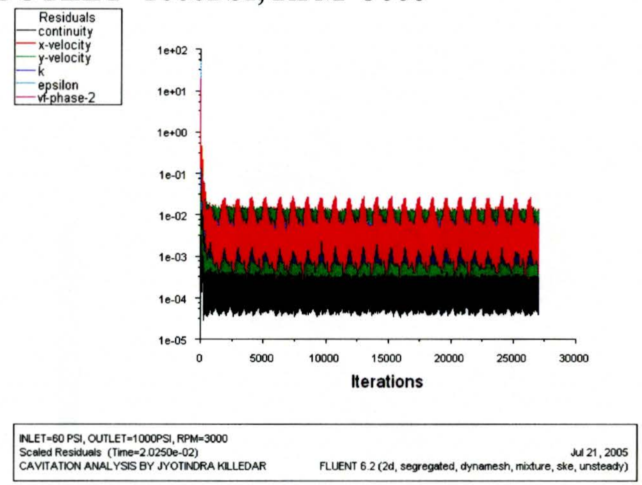
parameters set under dynamic mesh				**Boundary conditions**	***Cavitation parameters***	
		Mesh Methods				
Dynamic mesh	selected	**under smoothing**		**inlet**	TKE	0.02
		Boundary Node Relax	0.3		TDRate	5%
				outlet	BTKE	0.02
		Number of iterations =	50		BTDRate	10%
				controls solution	***Under Relaxation factors***	
Iterate		**Under Remeshing**			pressure	0.4
					Density	0.4
Time step size:1.5*e^-5					Body force	1.0
		Minimum Length scale (in)=	0.005208		momentum	0.8
No Time steps:355	500				Vap.Mass	0.4
		Maximum Length Scal (in)=	39370		vapor	0.3
Iterations/time step=50					TKE	0.1
		Maximum face skewn	0.6		TDE	0.1
					TV	0.1
		Size remesh interval	1			

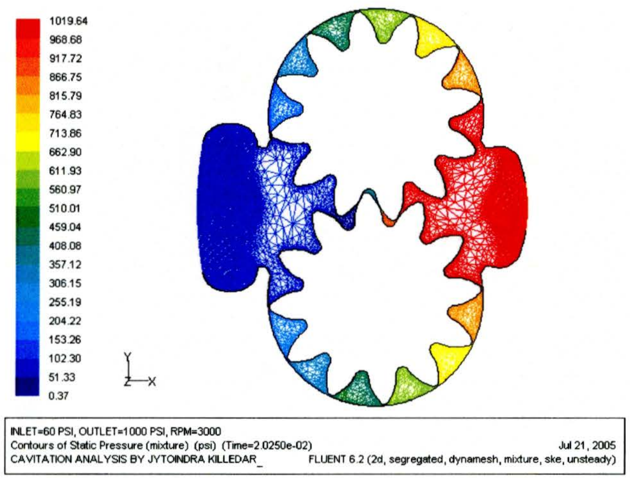
Dynamic Zones				**Descreatization**
Casing	stationary			PRESS
			CW	***Body Force weighted***
Clockwise gear	rigid motion	**x=0.00**	udf	**PRE-VEL COUPLING**
		y=1.50	attached	SIMPLEC
		z=0		
			CCW	**SKEWNESS**
Anticlokwise gear	rigid motion	x=0	udf	**CORRECTION=0**
		y= 0	attached	**MONITOR**
		z=0		***CONVERG. CRITERIA***

MATERIALELECTED		***CONTINUITY***	0.00010
		X	0.001
		Y	0.001
LIQUID	***FUEL- OIL***	***K***	0.00001
VAPOR	***FUEL-OILVAPOR***	***ε***	0.00001

CASE:-CAVITATION ANALYSIS, INLET=60 SPI, OUTLET=1000PSI, RPM=3000







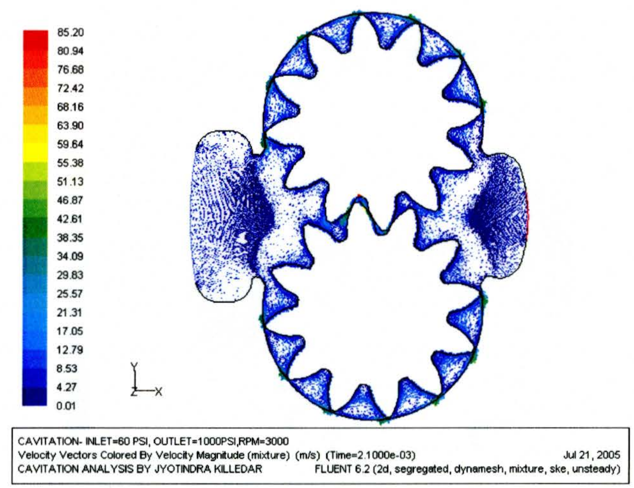


Fig 8.2 .Cavitation- Pressure and velocity plots

CASE:-CAVITATION ANALYSIS, INLET=60 SPI, OUTLET=1000PSI, RPM=3000

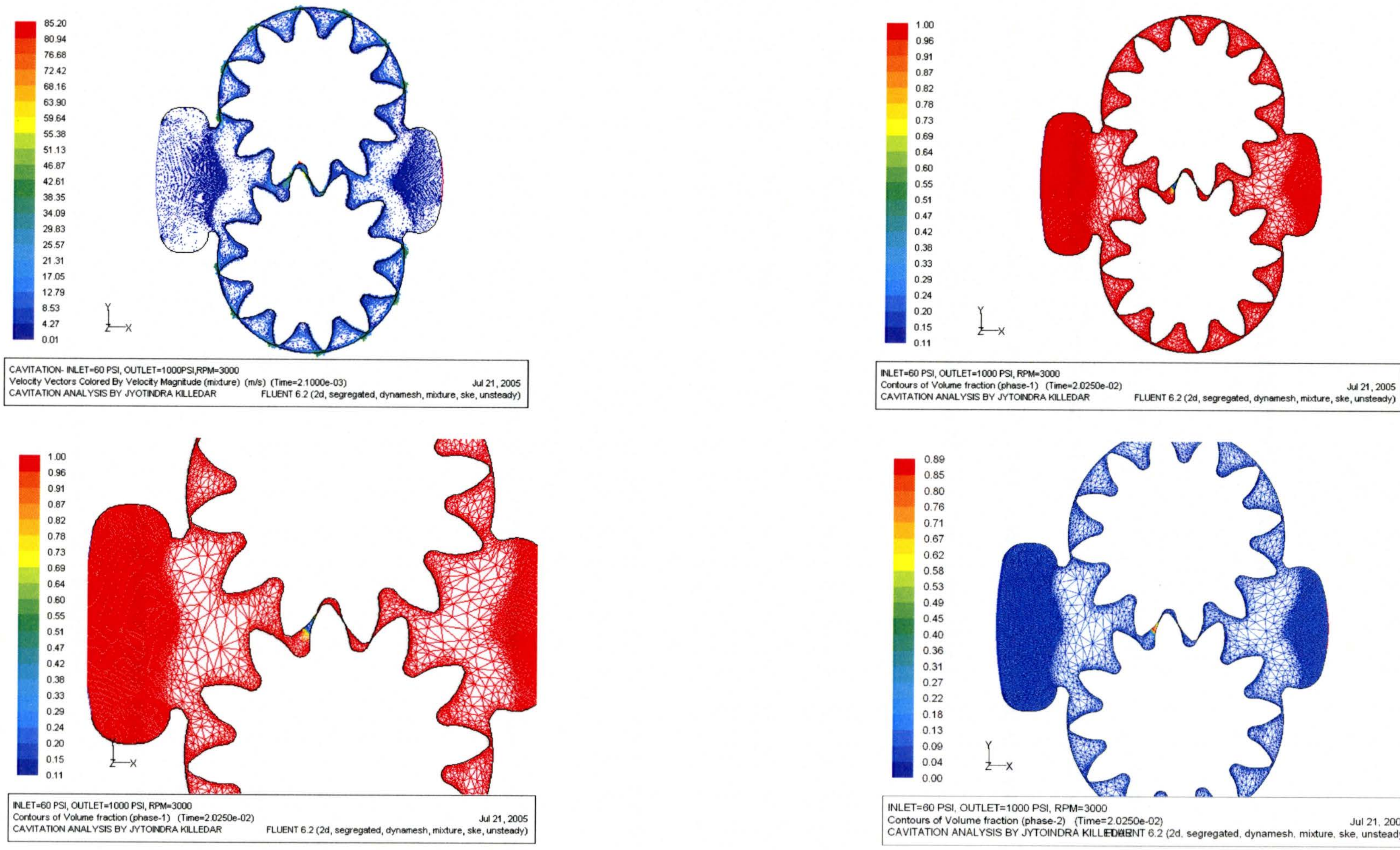


Fig 8.3 Volume Fraction contours

CHAPTER-9

CONLCUDING REMARKS

The gear pump was analyzed by using the state of the art "Computational Fluid Dynamics" software, Fluent. The turbulent flow analysis was carried out. A two dimensional true scale model with simplification was selected for the analysis. The flow was modeled as unsteady, turbulent flow for the analysis by Fluent.

The model consisted of moving zone (gears) and stationary zones (casing). This requires "Moving Dynamic Mesh". Moreover, due to the direction of both gears being opposite, the creation of mesh was a real complicated task. The mesh between the areas where gear teeth engage is very critical. When the gears start rotating the mesh gets pulled in two different directions. This will accelerate the tendency of cell collapsing, which wills prematurely terminate computation. A careful selection of time step size and re-meshing option must be made.

The speed and pressure selected in this thesis range from 1500 to 3000 rpm and 2000 psi to 3500 psi respectively. The analysis showed the correct flow patterns except for the analysis of the gear pump at 1500 rpm and 3500 psi. This particular case revealed the "reversed flow phenomenon". It was observed that at lower rpm range and very high pressure differential considered in the present problem, Fluent (6.2.16) was not capable of giving flow patterns expected in the gear pumps. This observation was confirmed from the communication with Fluent technical staff.

Extensive trials were conducted. The velocity, pressure and volume flow rate were monitored. The convergence of the numerical integration was achieved in a reasonable time steps. The examination of the graphical output shows results fairly close to the theoretical volume flow rate. The results also show that the flow is independent of the output pressure and is a function of speed.

Negative pressure was observed in the region where two gears are opening on inlet side. This was an early indication of the cavitation. It was decided, based on these results, to use Fluent's cavitation model with two phases. However, Fluent's cavitation model did not work for higher pressure differential i.e. outlet pressure higher than 1000 psi at 3000 rpm. The model does not show proper numerical stability at very high pressure differential.

Based on the output of volumetric flow rate, it is concluded that results obtained for selected model are fairly accurate.

In addition, this analysis provides reasonably acceptable results and significant information on modeling and problem set up for solving problems related to flow and pressure of the gear pump. The gap between the two meshing gear teeth and the annulus gap between the gears and casing plays important role and further reduction of this gap will deliver better results.

BIBLIOGRAPHY

[1] Nelik, Lev, Operating conditions and comparisons between chemical duty pumps and specialized high pressure gear pumps- World Pumps December, 2001

[2] Prof. Massimo Borghi, *Fluid Power Research group, DIMECH, department of mechanical and civil Engineering, Faculty of Engineering in Modena, university of Modena and Reggio Emilia, Modena, Italy* , International Journal of Fluid Power jointly published by FPNI and TuTexh, Vol,3,NO1, April2002

[3] Manring, Noah D and Kasaragadda, Suresh B, *The Theoretical Flow Ripple of an External Gear Pump*, Transaction of the ASME Vol. 125, September,2003.

[4] Broberg, Rob, *CFD analysis of Internal Gear Pump using CFX*,AEA Technology Engineering Software Ltd. in Waterloo, Ontario, Canada.

[5] Panta, Yogendra M., *Numerical flow Analysis of Gear Pump*, Master's thesis, Youngstown state University, Youngstown, Ohio 44555, 2004

[6] Kim, H.W., *Development of Fluid program in Engineering and technology*, Proceedings of the ASEE annual Conference, Salt Lake City, Utah, 2004.

[7] Cengel, Yunus A., Cimbala, John M., - *Navier-Stokes equations applied to Incompressible flow*, Chapter-9, Fluid Mechanics -Fundamental and Applications, McGraw-Hill,

[8] Fluent tutorial -26.1.1 Segregated Solution Method Fluent 6.2.16 documentation, Fluent inc., USA.

[9] Fluent tutorial -26.1.3 Linearization: Implicit vs. Explicit, Fluent inc., USA.

[10] Fluent User Guide Article no 24.6.4 -Mass Transfer through Cavitation and tutorial No- 17 Modeling Cavitation, Fluent inc., USA.

[11] Brennen, Christopher Earls, *Cavitation and Bubble Dynamics* © Oxford University Press 1995.

[12] Okita, Kohei; Matsumoto, Yoichiro; Numerical analysis for unsteady cavitating flow in Pump Inducer. Fifth International Symposium on Cavitation (CAV2003) Osaka, Japan, November 1-4, 2003

[13] Thompson, J. F., *Grid Generation Techniques in Computational Fluid Dynamics,* AIAA Journal, Vol. 22, pp. 1505-1523, 1984

[14] Rogers, S.E., Kwak, D. and Kiris, C., *Steady and Unsteady solutions of the Incompressible Navier-Stokes Equations,* AIAA, journal, vol.29, No.4, pp. 603-610, 1991.

[15] Sibley school of Mechanical and Aerospace Engineering, Cornell University, *Fluent Tutorials*, Ithaca, NY, 2002