NUMERICAL FLOW ANALYSIS OF GEAR PUMP

by

Yogendra M. Panta

Submitted in Partial Fulfillment of the Requirements

for the Degree of

Master of Science in Engineering

in the

Mechanical Engineering

Program

YOUNGSTOWN STATE UNIVERSITY

August, 2004

NUMERICAL FLOW ANALYSIS OF GEAR PUMP

Yogendra M. Panta

I hereby release this thesis to the public. I understand that this thesis will be made available from the OhioLINK 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: ______________________ 08-04-04

Yogendra M. Panta, Student Date

Approvals: ______________________ 8-4-04

Dr. Hyun W. Kim, Thesis Advisor Date

______________________ 8/4/04

Dr. Daniel H. Suchora, Committee Member Date

______________________ 08-04-04

Dr. Hazel Pierson, Committee Member Date

______________________ 8/6/04

Dr. Peter J. Kasvinsky, Dean of Graduate Studies Date

ABSTRACT

The objective of this study was to develop a numerical solution method for flow analysis of a gear pump with various boundary conditions and rotational speeds of gears. Flow variable contours and plots were obtained for fluid flow inside a gear pump subject to pressure inlet and pressure outlet conditions using the numerical control volume method in the commercial package FLUENT. Gears with different magnitudes of pressure outlets and rotational speeds were analyzed. Currently, no solutions that report the numerical turbulent flow analysis of gear pumps are available in the literature.

It is important to the design engineer that accurate methods are available to determine various flow parameters such as pressure, velocity, turbulence, flow rate etc. for the flow analysis of the gear pump. The method developed in this work was compared with the general data of P315 external gear pump manufactured by Parker Hannifin Corporation, as a validation of the numerical model.

An engineer will be able to use the solution method developed in this work for the analysis of flow parameters inside a gear pump in order to achieve its optimum design. Another benefit of this work is to demonstrate to practicing engineers that FLUENT has the capability of solving turbulent flow analysis of gear pumps by using its Moving Dynamic Mesh method. Students can follow this thesis to solve the dynamic mesh problem step by step. This work should be considered a small subset of numerical analysis of flow in gear pumps. It is expected that the results presented in this research provide important information for the extension to a three dimensional analysis of a similar problem.

Dedicated to my Parents

Mr. Gehendra M. Panta & Mrs. Bed Kumari Panta

ACKNOWLEDGMENTS

I would like to thank my advisor Dr. Hyun W. Kim for his guidance and insight throughout this thesis. His own work motivated some of the questions studied here and helped me to organize the results in a more logical manner. I would also like to thank him for his help in proofreading.

I would like to thank Dr. Daniel H. Suchora and Dr. Hazel Pierson for being in my thesis committee, helping me in proofreading, and providing important suggestions. I would like to thank Mr. Thomas Calko for doing most of the modeling of the gear pump. I would also like to thank Mr. Ashish Sudhakar Watve of FLUENT Inc., who gave me valuable guidance during the simulation work in FLUENT. In addition, I am grateful to Parker Hannifin Corporation, Gear Pump Division, Youngstown, OH, for providing physical data of the P315 Gear Pump.

During the work, I received very encouraging suggestions and help from my dear friends Manju and Narayan. I would like to thank my brothers Nagendra and Purushottam for their support of me as I completed this work. I would like to express my deepest gratitude to my fellow graduate students and friends, Ankur and Nassib. Finally, I would also like to thank Lois Romito, Administrative Assistant of the Graduate School, and especially Department Secretary Karen Pomponio, for their unwavering strength and friendship.

VITA

1995................................ Bachelor of Science, Physics, Mathematics and Statistics, Tribhuvan University, Kathmandu, Nepal.

2000................................ Bachelor of Engineering, Mechanical Engineering, Tribhuvan University, Kathmandu, Nepal.

2004................................ Master of Science, Mechanical Engineering, Youngstown State University, Youngstown, Ohio.

2002- 2004..................... Graduate Assistant, Department of Mechanical & Industrial Engineering, Youngstown State University, Youngstown, Ohio.

FIELDS OF STUDY

Major: Mechanical Engineering

Minor: Physics, Mathematics, and Statistics

UNDERGRADUATE SENIOR PROJECT

Detail Design of Pelton Turbine for 500 kW (with Pattern)

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

ρ: Density

μ: Viscosity

$\vec{V}$: Velocity Vector

$\vec{V}_s$: Velocity Vector on the gear surfaces

$\vec{P}$: Pressure Vector

P_i: Pressure at the inlet port

P_o: Pressure at the outlet port

$\hat{i}, \hat{j}, \hat{k}$: The unit vectors

μ_t: The turbulent viscosity

$\vec{A}$: Surface Area Vector

Γ_{ϕ} : Diffusion Coefficient for ϕ

∇_{ϕ} : Gradient of Φ

∇ : Del Operator

k: Kinetic Energy

ϵ: Kinetic Energy Dissipation Rate

G_k: The generation of turbulence kinetic energy the mean velocity gradients

G_b: The generation of turbulence kinetic energy due to buoyancy

Y_M: The fluctuating dilatation in compressible turbulence to the overall dissipation rate

σ_{κ}: Turbulent Prandtl number for k

σ_{ϵ}: Turbulent Prandtl number for ϵ

S_{Φ}, S_{κ} and S_{ϵ} : User-defined source terms.

σk: Turbulent Prandtl number for k

σ_{ϵ}: Turbulent Prandtl number for ϵ

$C_{1\epsilon}$, $C_{2\epsilon}$, $C_{3\epsilon}$, C_{μ}: Constants

N_{faces} : Number of faces enclosing cell

Φ_f : Value Φ of convected through face f

$\rho_f \vec{v}_f . \vec{A}_f$: Mass flux thorugh the face

$\vec{A}_f$: 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 f

V: Cell volume

$\vec{x}^{n+1}_{c.g.}$: Position of the center of gravity

$\vec{\theta}^{n+1}_{c.g.}$: Orientation of the center of gravity

$\vec{v}_{c.g.}$: Linear velocity and of the center of gravity

$\vec{\Omega}_{c.g.}$: Angular velocity of the center of gravity

G : The transformation matrix that defines the choice of $\vec{\theta}$

(u, v, ω): Linear velocity as a function of time

(ω_x, ω_y, ω_z) : Angular velocity as a function of time

$\hat{e}_r$, $\hat{e}_\theta$: The unit vectors

CHAPTER 1

INTRODUCTION

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 from a low-pressure region to a higher-pressure region. The pump does not create system pressure, although the pump is often described in terms of its limitation of pressure. The pressure that exists at the outlet port of the pump is a result of system load that was created by a resistance to the flow. Therefore, the outlet flow rate (or the displacement) and the pressure are proper terms to characterize and describe the pump. The pumps are generally categorized in two distinct groups, positive-displacement pumps and kinetic pumps. All pumps used in hydraulic systems are of the positive displacement type that includes gear, vane, and piston pumps. This means that the fluid will be pushed forward from the inlet to the outlet by the direct motion of solid bodies in a positive displacement action. The gear pump is the most robust and rugged type of all positive displacement pumps and is one of the simplest hydraulic pumps [1]. The kinetic type 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 and include centrifugal pumps [2].

Two years ago, Youngstown State University began to develop a collaborative research and educational programs in hydraulics with Parker Hannifin Corporation and established the Center for Hydraulics Research and Education in Fall 2003 [3]. Parker Hannifin, headquartered in Cleveland, Ohio, is a world class business and manufacturing leader in hydraulics. Two of its divisions, Gear Pumps and Mobile Cylinders, are at local plants in Youngstown. As part of that development, the Mechanical Engineering Program is working toward establishing a state-of-the-art computational laboratory that includes solid modeling, computational fluid dynamics analysis, interactive flow and stress analysis, interactive motion control, and simulation. Computational fluid dynamics can help to optimize the design of the pump and improve the efficiency of the pump, other components, and the entire hydraulic system. A flow analysis for an external gear pump by FLUENT, one of the most versatile CFD software packages currently available in the market, was selected as one of the initial research projects in the developmental effort.

The flow pattern created inside an external gear pump by the motion of two gears rotating in opposite directions is deceptively complex despite the simple geometry of the gear pump. The flow cannot be analyzed, based on a steady-state assumption that is usually employed to analyze turbo-machinery despite the fact that the flow is essentially steady. Only the time-dependent, unsteady, dynamic meshing can predict the motion of the fluid flow against the very high adverse pressure distribution. Although the complexity of analysis is inherent in all positive displacement pumps, gear pumps pose an exceptional challenge due to the fact that there are two rotating components which must be in contact with each other all the time housed within a stationary three-dimensional casing. The study and analysis presented in this thesis will deal with those problems to make an acceptable preliminary investigation on the gear pump flow and document the step by step procedure from the description of a physical model to the results of the numerical analysis.

The hydraulic pump has been used for a long time. The Ancient Egyptians invented water wheels with buckets mounted on them to move water for irrigation. In the 200's B.C. Ctesibius, a Greek inventor, made a reciprocating pump for pumping water. Around the same time, Archimedes, a Greek mathematician, invented a screw pump made of a screw rotating in a cylinder, now known as an Archimedes screw. Thousands of years later, pumps still operate in the same basic way. The modern day development of the gear pump began in 1588. In 1588, Ramelli, an Italian engineer invented a water pump which continues to be used in oil pumps and compressors. In 1636, Pappenheim, a German engineer invented the gear pump used to lubricate engines. In 1799, one of James Watt's co-workers, Murdock, adapted Pappenheim's gear pump to create a rotary piston steam engine. In 1859, Jones, modified Pappenheim's gear pump and produced a double rotor with only two teeth per gear [5].

Since the introduction of the gear pump in the 1930s, gear pump technology has been steadily improved. Gear pumps have been widely used in polymer manufacturing plants and transportation industries. Thus, research of the gear pump is one of the older fluid power research fields and is considered as a mature technology. More recent classical research activity, initiated by Professor Borghi at the end of the 1980's, is still active and includes major industrial research in cooperation with the CASAPPA Fluid Power (1995 - present), one of the important Italian manufacturers in this field [6].

Luc Machiels (1997) performed research on different randomly forced turbulent systems, by direct numerical simulation. Luc showed the high degree of randomness produced intrinsically in three-dimensional incompressible Navier-Stokes turbulence and the numerical method for the simulation of randomly forced turbulent systems [7]. Spyros Gavrilakis found the effects of rotation on the turbulent field inside a straight square duct in his research on numerical simulation of turbulent flows. He included the decay of turbulence even at low Reynolds numbers [8].

Noah D. Manring and Suresh B. Kasaragadda have done research on the theoretical flow ripple of an external gear pump of similar size, using different numbers of teeth on the driving and driven gears. According to them, an external gear pump design with a large number of teeth on the driven gear and a fewer number of teeth on the driving gear would be better for high performance. The results showed that the driving gear dictates the flow ripple characteristics of the pump while the driven gear dictates the pump size [9]. Jaakko Myllykylä, Tampere University of Technology, Tampere, Finland has done research on the suction capability of an external gear pump. He showed that higher rotational speeds can lead to cavitations. His research collected data on a number of gear pumps to study the suction characteristics. He formulated a theory for suction of external gear pumps with two gears. According to his research, he mentioned that the biggest uncertainty factor is the mathematical expression for the effect of the housing shape on the suction characteristics [10]. Lev Nelik, used the basic equations of pump flow and volumetric efficiency and verified the results with experiments. He found three important things: volumetric efficiency is lower at higher differential pressures, viscous fluids result in higher volumetric efficiency, and volumetric efficiency appears higher at higher speeds [11]. Y.P. Marx as the chief of research team for Faculty of Engineering Sciences and

Techniques (STI) at the Swiss Federal Institute of Technology - Lausanne (EPFL), developed numerical tools for simulating generalized Newtonian flows in gear-pumps. The computer program code SAGARMATHA, developed by Marx, with contributions from T. Jongen, O. Byrde and M.L. Sawley, was applied on several gear pump inlets for the simulation of generalized Newtonian flows at very low Reynolds numbers. With the proposed formulation, a numerical principle was used to solve the Stokes and the potential equations. This was demonstrated by the ease in transforming a Stokes solver into a potential flow solver. The code incorporates modern numerical methods and computational techniques for the numerical simulation of large-scale incompressible flows. The code is also used for the simulation of flows involving both stationary and rotating sub domains. Computational meshes with blocks that slide across each other can be employed [12]. Using the Mesh Superposition Technique in POLYFLOW software, Fluent Inc. has done a simulation for the molten polymer flow analysis of the gear pump. A research result obtained by the company's internal team showed that a high pressure region is formed where the gear teeth are closing in the outlet port [13].

Despite its long history of usage and numerous researches on gear pumps related subjects [14-39], no work has been reported in the literature that examined the numerical turbulent flow analysis of gear pump. Therefore, this research has been initiated using FLUENT, a commercial finite volume CFD software package, to investigate two dimensional turbulent flow analysis of a gear pump. It is expected that the results obtained in this report provide important information for the extension to a three dimensional analysis.

CHAPTER 2

PHYSICAL MODEL OF GEAR PUMP

2.1 Description of Model

P300 series pumps are the production of Parker Hannifin Corporation, Gear Pump Division, Youngstown, OH. The physical model for the fluid flow analysis was chosen from Parker Corporation's existing product; P315 external gear pump (refer to Figures 2.1, 2.2, 2.3, 2.4, 2.5).

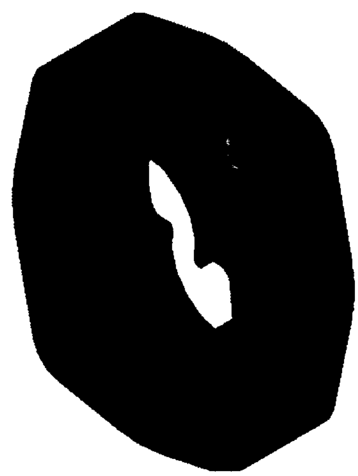


Figure 2.1 Middle Casing 3D View

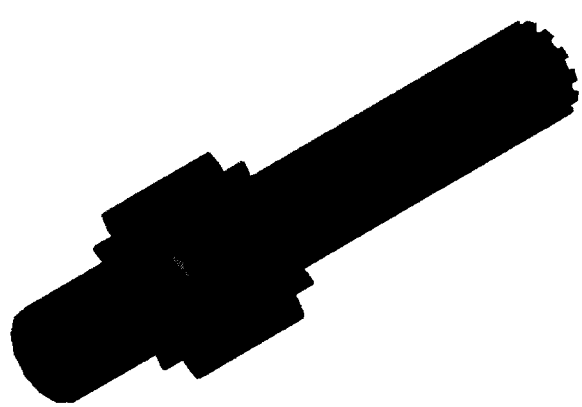


Figure 2.2 Driving Gear 3D View

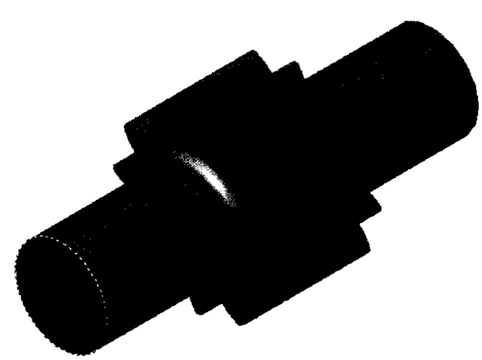


Figure 2.3 Driven Gear 3D View

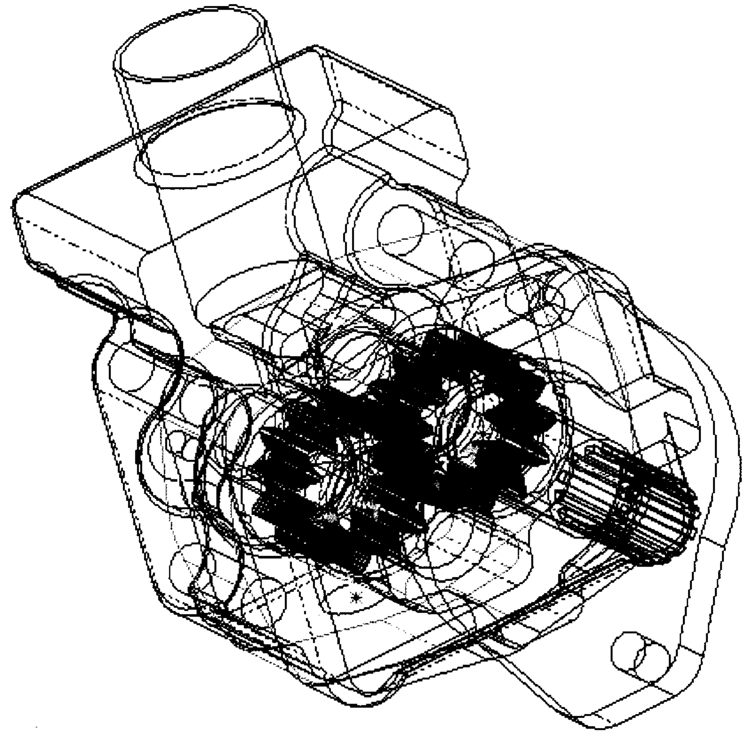


Figure 2.4 Wire Frame View of the Assembled Gear Pump

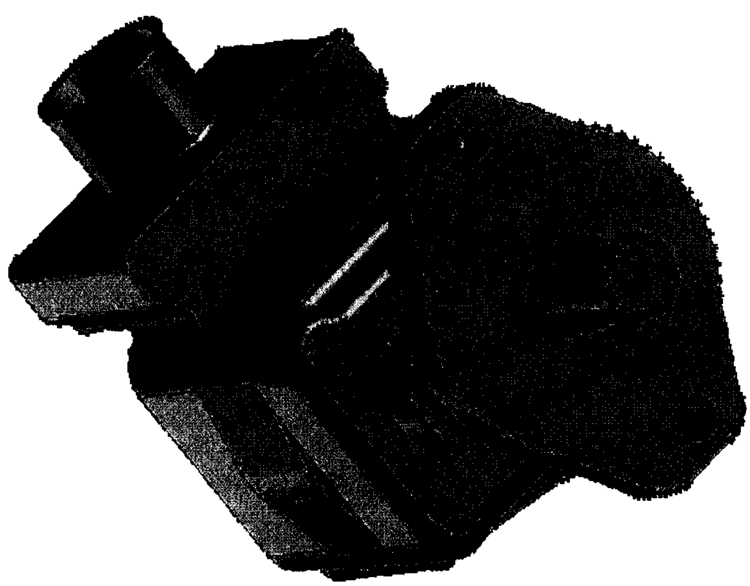


Figure 2.5 Shaded View of the Assembled Gear Pump

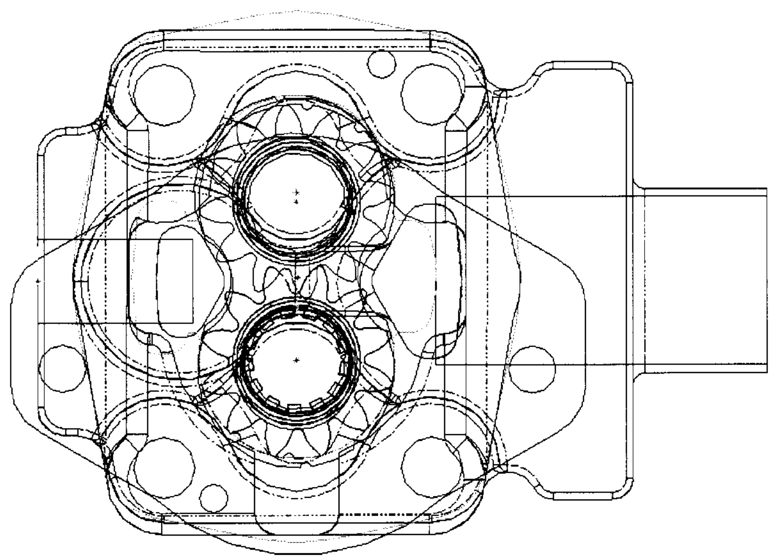


Figure 2.6 Top View of the Assembled Gear Pump

2.2 General Data of P 315 External Gear Pump

After careful study of the general data of the P315 gear pump in PGP/PGM300 Series available and published by Parker Hannifin Corporation, the following data are taken for the analysis of gear pump:

Pump Type

External gear pump, heavy duty

Solid Material:

Cast iron for gear housing

Stainless steel for gear rotors

Hydraulic Fluid:

Mineral oil, water glycol, HFC; water oil emulsions 60/40, HFB

Gear Drives

Clockwise, counterclockwise, double

Speed Range

From 400 to 3000 rpm

Pump Inlet Pressure

30 psi/ 0.8 to 2.0 bar at operating temperature

Outlet Pressure Range

Continuous 2500-3500 psi, intermittent 4000 psi

Fluid Temperature

Mineral oil with standard seals: 0° to 180° F (-20°C to $+80^{\circ}$C)

Basic Dimensions of Gear Pump:

Weight: 18 lbs

Gear widths: 0.5 to 2.5 in.

Total no. of gear teeth in each rotor: 12

Center to center distance of gears: 0.98 in.

Gear outside diameter: 1 in.

Gear inside diameter: 0.75 in.

Inlet port size: 0.25 in. curved

Outlet port size: 0.75 in. curved

2.3 Working Fundamentals of the Gear Pump

The gear pump is made of two or more gears rotating inside a closed casing (Figure 2.4). The driver 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 (refer to Appendix 3). 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. The fluid stays in place between the teeth until it passes to the other open chamber of the gear mesh, on the outlet side. At this point, the teeth of the gears continue rotating and thus come back into contact, and the liquid there is forced out. Since there are sealed bushing around the gears, the displaced liquid must move forward to the pump's outlet port. The pump works like a rotating conveyor belt, with a lot of pockets of liquid between the teeth of the gears moved forward by the rotating motion.

CHAPTER 3

MATHEMATICAL MODELING OF GEAR PUMP

The gear pump problem to be considered is shown schematically in Figure 3.1. Low pressure oil enters through the inlet port and rotates with the rotational motion of gears, creating pockets of fluid, and finally discharges from the outlet port with high pressure. Creation of a 2D gear pump model, setup parameters, and solution procedure to solve the flow inside a gear pump is presented in the following sub topics.

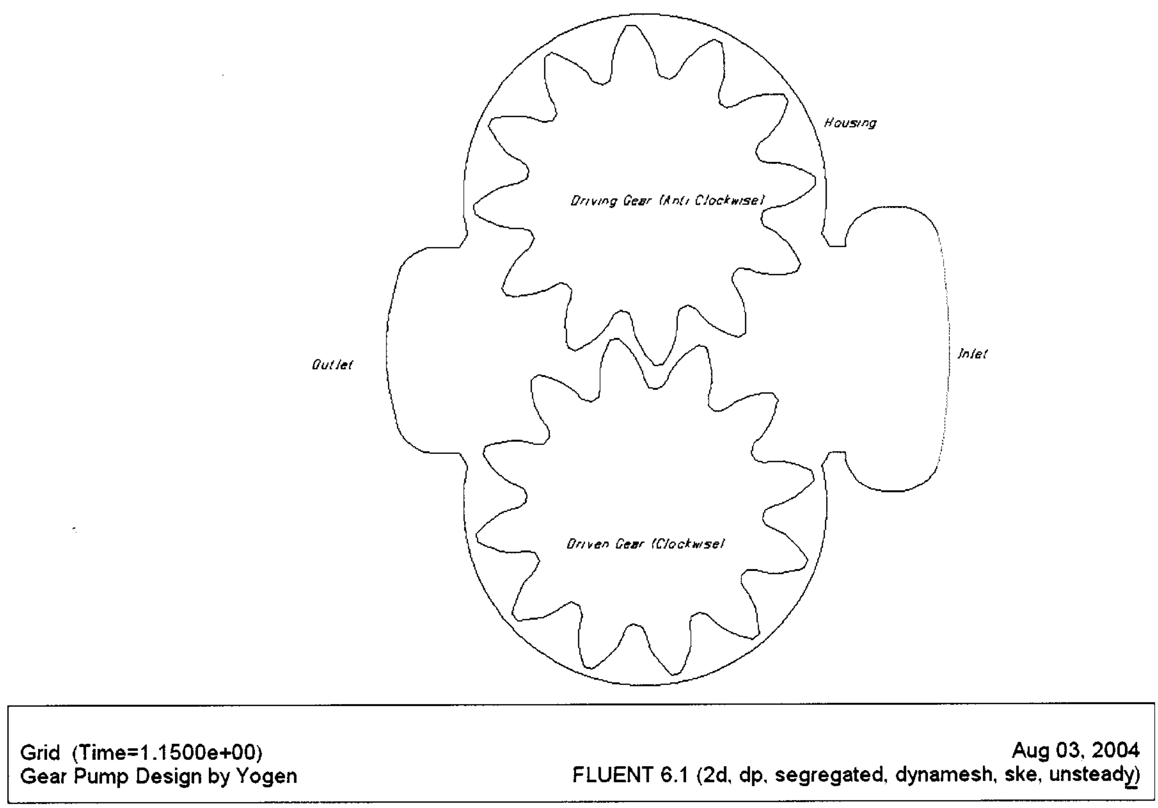


Figure 3.1 2D Gear Pump Model

3.1 Governing Equations of Fluid Flow for Gear Pump

The fluid motion generated by two fast rotating gears inside a gear pump can be described by the Navier-Stokes equation; however, the flow is expected to be highly turbulent. Therefore, the fluid motion and transport characteristics are governed by not only three conservation equations, but also by two additional equations for the turbulent kinetic energy and the rate of dissipation of kinetic energy.

Assumptions incurred on this flow analysis may be stated as follows:

1. The fluid is a Newtonian, incompressible fluid.
2. The fluid is originally stationary.
3. The flow is two dimensional.
4. Body forces are negligible.
5. Viscous heating is considered.

The coordinate system is chosen such that the origin of the Cartesian coordinate is at the center of the driven gear. Upon imposing the above stated conditions, the governing equations, boundary conditions, and initial conditions can be expressed as follows:

Continuity Equation

$$\nabla \cdot \vec{V} = 0 \qquad \text{Equation 3.1}$$

Momentum Equation

$$\rho\left(\frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V}\right) = -\nabla \cdot \vec{P} + \mu \nabla^2 \vec{V} \qquad \text{Equation 3.2}$$

Initial condition:

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

With boundary conditions of velocity:

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

On the gear surfaces: $\vec{V} = \vec{V}_s$

Also, the boundary conditions of pressure:

At the inlet port: $P = P_i$

At the outlet port: $P = P_o$

Energy Equation

$$\frac{\partial(\rho c T)}{\partial t} + \nabla \cdot \left(\rho \vec{V} c T\right) = \nabla\left(k \nabla T\right) + S_\phi \text{ .... Equation 3.3}$$

Where,

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

Boundary conditions of temperatures:

At the inlet port: $T = T_i$

At the outlet port: $T = T_o$

On the gear surfaces: $\frac{\partial T}{\partial n} = 0$

3.2 The Turbulence Kinetic Energy k-ϵ Equations

As previously stated, two additional equations are necessary in order to characterize the turbulent flow.

$$\frac{\partial}{\partial t}(\rho \kappa) + \nabla(\rho \kappa \vec{V}) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \text{ .... Equation 3.4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla(\rho \varepsilon \vec{V}) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \text{ .... Equation 3.5}$$

The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ϵ as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \text{ .... Equation 3.6}$$

Generally, these values are initially set to:

$C_{1\epsilon}$=1.44, $C_{2\epsilon}$=1.92, C_μ =0.09, σ_k =1.0, σ_ϵ =1.3

CHAPTER 4

NUMERICAL MODELING OF GEAR PUMP

4.1 Numerical Simulation

Computational fluid dynamics (CFD) performs numerical analysis of a fluid flow field once a finite volume grid has been created. Setting boundary conditions, defining fluid properties, executing the solution, refining the grid, and viewing and post processing the results are generally performed within the chosen CFD code. For this research, Solid Works was used for model construction, GAMBIT was used for preprocessing & grid generation, and FLUENT was used for the CFD solver.

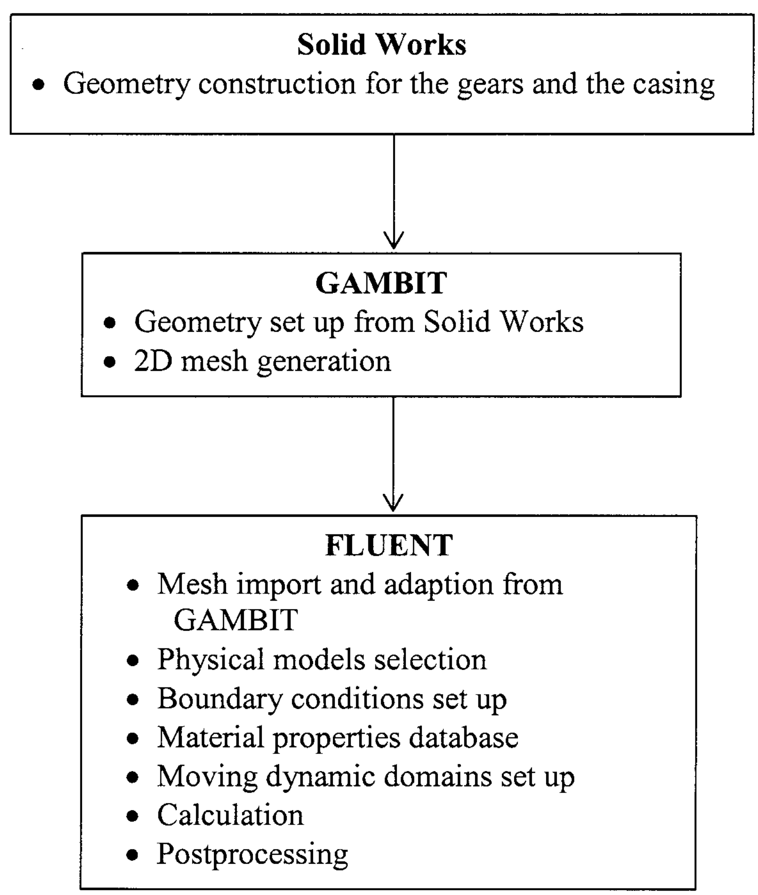


Figure 4.1 Basic Program Structure in Numerical Simulation

4.2 CFD Analysis in FLUENT

FLUENT uses an unstructured algorithm for mesh generation in order to simplify the geometry and mesh generation process, model more complex geometries and adapt the mesh to resolve the flow-field features. FLUENT can also use body-fitted, block-structured meshes. The software is capable of handling triangular and quadrilateral elements in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. The initial mesh is first generated using GAMBIT, and then it can be adapted in FLUENT in order to resolve large gradients in the flow field.

4.2.1 GAMBIT, the Preprocessor for Geometry Modeling and Mesh Generation, Postprocessor for Results Viewing

GAMBIT is a software package designed to build, import and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT gets user input by graphical user interface (GUI). Then the GUI performs the fundamental steps of building, meshing, and creating zone types in a model. The most important operations are associated with the **Geometry**, **Mesh**, **Zones**, and **Exporting files for Postprocessing** command buttons, which will be described here.

(i) Creating Geometry in Solid Works and Importing it into GAMBIT

File → Import → IGES ….the files were imported from Solid Works source files.

(ii) Meshing of the Model

Mesh command button on the **Operation** tool pad in GAMBIT opens the **Mesh** sub pad which contains command buttons that performs mesh operations involving boundary layers, edges, faces, volumes, and groups. In the particular problem of 2D gear pump model, this was used for face meshing (refer to section 5.3/step 7/page 32)

(iii) Specifying Zone Types

The domains of the model at its boundaries and in some specific regions can be defined by using zone-type specifications. There are two classes of zone-type specifications:

- Boundary types: Wall for gears and casing; Pressure Inlet for the inlet and Pressure Outlet for the outlet.
- Continuum types: Fluid for the meshed face.

(iv) Exporting Results Files from GAMBIT

To display post processing results for any given solution, two files must be imported into GAMBIT:

- A results database
- A neutral file

The results-database file contains the results data for the solution. The neutral file contains coordinate and connectivity information for the model.

4.2.2 FLUENT, the CFD Solver

FLUENT is a finite volume based code for the simulations and modeling of fluid flow as well as heat transfer. Since FLUENT is written in the C computer language, it makes full use of the flexibility and power offered by the language. FLUENT is ideally appropriate for incompressible and compressible fluid flow simulations in complex geometries. The following considerations are intricate in order to solve a gear pump problem in FLUENT:

- **Modeling Goals:** One must identify the expected specific results and the degree of accuracy required from the model.
- **Geometrical Parameters Processing:** This addresses the computational domain, where one must specify boundary conditions used at the boundaries of the model, dimension of the model (2D or 3D), type of grid topology best suited for the model.

- **Flow and Dynamic Data Processing:** The flow type of viscous model must be specified: inviscid, laminar, or turbulent, unsteady or steady flow over the time, heat transfer, incompressible or compressible fluid.
- **Solution Procedure:** The solver formulation, solution parameters and the result convergence need to be decided upon and specified.

For the Moving Dynamic Mesh (MDM), user-defined functions (UDFs) were used to control the dynamic mesh of two rotating gears and a stationary casing. Smoothing and re-meshing were used for the rotating gears and casing. The rotation of the gears was defined and controlled by UDF as a rigid body type that utilizes a macro specific to the dynamic mesh model while the casing was taken as a stationary type boundary. Previously created gear pump housing with two gears was imported into GAMBIT from a Solid Works 2D source file to create the 2D geometry for the gear pump. A fine mesh was generated (refer to Figure 5.4). Finally, the mesh file was exported to FLUENT for the further flow analysis. Once the meshed model was in FLUENT, the following major steps are summarized for the flow analysis:

1. Dynamic mesh capability of FLUENT was used to solve rotating gears rigid body motion problem.
2. A compiled user-defined function (UDF) was used to specify the rigid body motion of rotating gears.
3. The dynamic mesh was previewed before starting the flow computation.
4. A solution using segregated solver was calculated.

Program Capabilities of FLUENT

The FLUENT solver has capabilities of solving the following model. Note that the bold type indicates the features utilized in this work.

- **2D planar**, 2D axisymmetric, 2D axisymmetric with swirl and 3D flows.
- Quadrilateral, **triangular**, hexahedral (brick), tetrahedral, prism (wedge), pyramid, and mixed element **meshes**.
- Steady-state or **transient flows**.

- **Incompressible** or compressible flows.
- Inviscid, laminar, and **turbulent flows**.
- **Newtonian** or non-Newtonian flows.
- Heat transfer of all kinds and radiation.
- Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models.
- Free surface and multiphase models for gas-liquid, gas-solid, & liquid-solid flows.
- Lagrangian trajectory calculation for dispersed phase (particles/droplets/bubbles), including coupling with continuous phase.
- Cavitation model.
- Phase change model for melting/solidification applications.
- Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and porous-face pressure jump conditions.
- Lumped parameter models for fans, pumps, radiators, and heat exchangers.
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames.
- Multiple reference frame (MRF) and sliding mesh options for modeling multiple moving frames.
- Mixing-plane model for modeling rotor-stator interactions, torque converters, and similar turbo machinery applications with options for mass conservation and swirl conservation.
- **Dynamic mesh model for modeling domains with moving and stationary mesh (MDM).**
- Volumetric sources of mass, momentum, heat, and chemical species.
- **Material property database.**
- **Extensive customization capability via user-defined functions (UDF).**

4.2.3 Problem Solving Steps

Once after determining the important features of the particular problem of gear pump, the following basic procedural steps were taken.

Geometrical Modeling and Meshing (in **Solid Works** and **GAMBIT**)

1. Create the model the geometry and grid.

Preprocessing (in FLUENT)

2. Start the appropriate solver of **FLUENT for 2ddp** modeling.
3. Import the grid: **File** Menu
4. Check the grid : **Grid** Menu
5. Select the solver formulation : **Define** Menu
6. Choose the basic equations: laminar or turbulent (or inviscid) etc. : **Define** Menu
7. Specify material properties : **Define** Menu

Dynamic Mesh Zones: Moving Dynamic Mesh

8. Specify dynamic mesh zones & boundary conditions: **Define** Menu
9. Adjust the solution control parameters : **Solve** Menu
10. Initialize the flow field : **Solve** Menu

Calculation:

11. Calculate a solution : **Solve** Menu
12. Examine the results : **Display, Plot, Report** Menu

Postprocessing:

13. Save the results : **File** Menu

4.4 Solution Approach in FLUENT

The approach involves a numerical modeling to assess the use of CFD to model and predict the flow patterns of the flow inside the gear pump. FLUENT is used to complete a series of simulations for flow from the pump's intake to its outlet.

For the gear pump model, a **segregated solver** was used**.** Using this method, **FLUENT** solves the governing equations of Navier-Stokes for the conservation of mass and momentum, and (when appropriate) for energy and other scalars such as turbulence. In segregated solver method, a control-volume-based technique is used which consists of:

- Division of the domain into discrete control volumes using a computational grid.
- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables such as velocities, pressure, temperature, and conserved scalars.

- Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

FLUENT is also capable of a coupled solution numerical method which employs a similar discretization process (finite-volume); but the approach used to linearize and solve the discretized equations is different than the segregated solver. Since all of the simulations were done using the segregated solver, the coupled solution method is not included.

4.4.1 Segregated Solution Method

Using this approach, the governing equations are solved sequentially (i.e., segregated from one another). As the governing equations are non-linear, several iterations of the solution loop must be done before a converged solution is achieved.

1. Fluid properties are updated each iteration based on the current solution.
2. The **u, v,** and **w** momentum equations are solved individually using the current iterative values for pressure and face mass fluxes to update the velocity field.
3. Since the velocities obtained may not satisfy the continuity equation locally, the pressure correction equation is then solved to get the corrections for the pressure and velocity fields that do satisfy the continuity equation.
4. Equations for scalars, e.g. turbulence, are solved using the updated values of the other variables.
5. If inter phase coupling is to be included, the source terms in the appropriate continuous phase equations are updated with a discrete phase trajectory calculation.
6. Convergence for the equation set is checked each iteration until the convergence criteria is met.

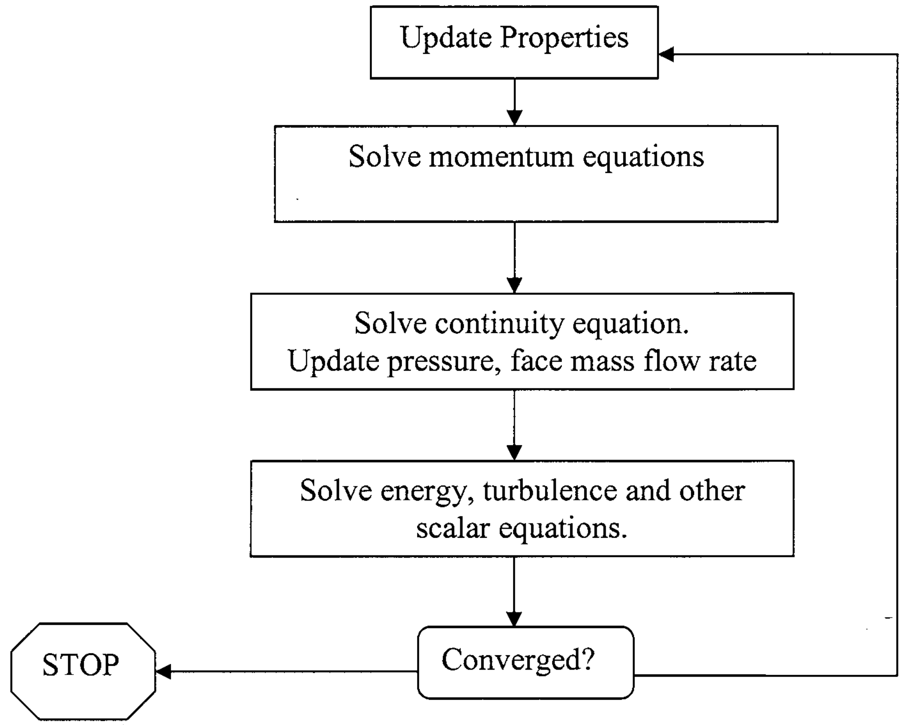


Figure 4.2 Program Algorithm in Segregated Solution Method

4.4.2 Linearization: Implicit

In the segregated solution method, the governing equations are linearized to create a system of equations for all dependent variables in each computational cell. The resultant linear system is then solved for an updated flow-field solution. The implicit option is the only one available in the segregated solver, thus it was the solution method for the gear pump. In the segregated solution, each discrete governing equation is linearized implicitly with its dependent variables. A point implicit (Gauss-Seidel) linear equation solver is used with an algebraic multi-grid (AMG) simultaneously to solve the resultant scalar system of equations for the dependent variable in each and every cell. So, the segregated approach solves a single variable field (e.g., **v**) by considering all cells at once. Then it continues to solve for the next variable field again by considering all cells at once, and so on.

4.4.3 Discretization: First Order Upwind Scheme

FLUENT uses a **Control Volume (CV) Technique** to convert the governing equations into algebraic equations. The control volume technique is the integration of the governing equations in each control volume, yielding discrete equations which conserve each quantity on a control-volume basis.

Integral form for a control volume **V**:

$$\oint \rho\phi\vec{v}.d\vec{A} = \oint \Gamma_\phi \nabla_\phi .d\vec{A} + \int_V S_\phi dV \quad \text{.......... Equation 4.1}$$

Where,

ρ = Density

$\vec{v}$ = Velocity vector ($u\hat{i} + v\hat{j}$ =in 2D)

$\vec{A}$ = Surface area vector

Γ_ϕ = Diffusion coefficient for Φ

∇_ϕ = Gradient of Φ = $\left(\partial\phi/\partial x\right)\hat{i} + \left(\partial\phi/\partial y\right)\hat{j}$ in 2D

S_Φ is source of Φ per unit volume.

This is applied in each control volume, or cell, in the computational domain. Discretization of the above equation on a given cell yields

$$\sum_f^{N_{faces}} \rho_f \vec{v}_f \phi_f .\vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi (\nabla\phi)_n .\vec{A}_f + S_\phi V \quad \text{.......... Equation 4.2}$$

Where,

N_{faces} = Number of faces enclosing cell

Φ_f = Value Φ of convected through face **f**

$\rho_f \vec{v}_f . \vec{A}_f$ = Mass flux through the face

$\vec{A}_f$ = 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

The first-order discretization generally yields better convergence than the second-order scheme, although it generally will yield less accurate results, especially on triangular and tetrahedral grids. This is because the flow is never aligned with the grid. More accurate results could be obtained by using the second-order discretization. However, for moving dynamic meshes, first order upwind is used, and thus used in the gear pump model, as the moving dynamic mesh will be more stable.

4.4.4 Under-Relaxation Factors

As the nonlinearity of the equations is solved by FLUENT, it needs to control the change of a variable Φ, which can be achieved by under-relaxation. This reduces the change of Φ generated during an iteration. So, the new Φ within a cell depends upon the old, Φ_{old} and only a portion, α, of the $\Delta\Phi$ occurring during an iteration.

$\Phi = \Phi_{old} + \alpha \, \Delta \, \Phi$ .... Equation 4.3

4.4.5 Improving the Grid by Smoothing and Swapping

Smoothing and face swapping are basic tools to complement grid adaption, usually increasing the quality of the final numerical mesh. Smoothing relocates the nodes and face swapping modifies the cell connectivity to achieve these improvements in quality.

4.4.6 Judging Convergence

- Crude initial guesses for mean and turbulence quantities can cause the solution to diverge. The iteration solution was started from inlet pressure conditions in this problem. Since the inlet boundary conditions are given in FLUENT, it automatically shows the initial guess for the iteration.
- Reasonable initial guesses for the k and ϵ fields can give a quick convergence. Here, the default criterion was changed from 10^{-3} to 10^{-6} for the precision.

4.5 Dynamic Mesh Modeling: Moving Dynamic Mesh

The dynamic mesh model in FLUENT can be used to model flows where the shape of the domain is continuously changing with time due to motion on the domain boundaries. The motion can either be a prescribed motion (e.g., in this research, angular velocities are specified about the center of gravity of two gears with time) or an un-prescribed motion where the subsequent motion is determined through a user-defined function [38]. The volume mesh updating is done automatically by FLUENT at every time step based on the new positions created in the boundaries. In order to use the dynamic mesh model, it is needed to provide an initial volume mesh and the details of the motion of any moving domains.

As stated previously, the unsteady flow created by two rotating gears inside a gear pump is complex and can be analyzed only by properly accommodating constantly changing boundaries. FLUENT can perform such tasks through its Moving Dynamic Mesh capability.

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 (u, v, ω) as a function of time or angular velocity (ω_x, ω_y, ω_z) as a function of time or both as needed. UDFs were used to specify angular velocities of gears for gear pump. For this problem, UDF code written in "C" has angular velocity inputs for the rotating gears. The starting location of the center of gravity and orientation of the rigid body must also be specified. FLUENT automatically updates the center of gravity position and orientation at every time step (refer to Figure 4.3) such that:

$$\vec{x}_{c.g.}^{\,n+1} = \vec{x}_{c.g.}^{\,n} + \vec{v}_{c.g.}\Delta t \quad \text{Equation 4.4}$$

$$\vec{\theta}_{c.g.}^{\,n+1} = \vec{\theta}_{c.g.}^{\,n} + G\vec{\Omega}_{c.g.}\Delta t \quad \text{Equation 4.5}$$

Where,

$\vec{x}_{c.g.}^{\,n+1}$ and $\vec{\theta}_{c.g.}^{\,n+1}$ are the position and orientation of the center of gravity, $\vec{v}_{c.g.}$ and $\vec{\Omega}_{c.g.}$ are the linear and angular velocities of the center of gravity, and G is the transformation matrix that defines the choice of $\vec{\theta}$. By default, G is taken to be the identity matrix. Typically, $\vec{\theta}$ is chosen to be an appropriate set of Euler angles. In this case, the rigid-body motion must be specified using a user-defined function (DEFINE_CG_MOTION) where the appropriate form of G can be speficied.

The position vectors $\vec{\Omega}_{c.g.}$ on the rigid body are updated based on rotation about the instantaneous angular velocity vector. For a finite rotation angle $\Delta\theta = \left|\vec{\Omega}_{c.g.}\right|\Delta t$, the final position of a vector $\vec{x}_r$ on the rigid body with respect to $\vec{x}_{c.g.}$ can be expressed as:

$$\vec{x}_r^{\,n+1} = \vec{x}_r^{\,n} + \Delta\vec{x} \quad \text{Equation 4.6}$$

Where,

$$\Delta\vec{x} = \left|\vec{x}_r^{\,n}\right|\left[\sin(\Delta\theta)\hat{e}_\theta + (\cos(\Delta\theta)-1)\hat{e}_r\right] \quad \text{Equation 4.7}$$

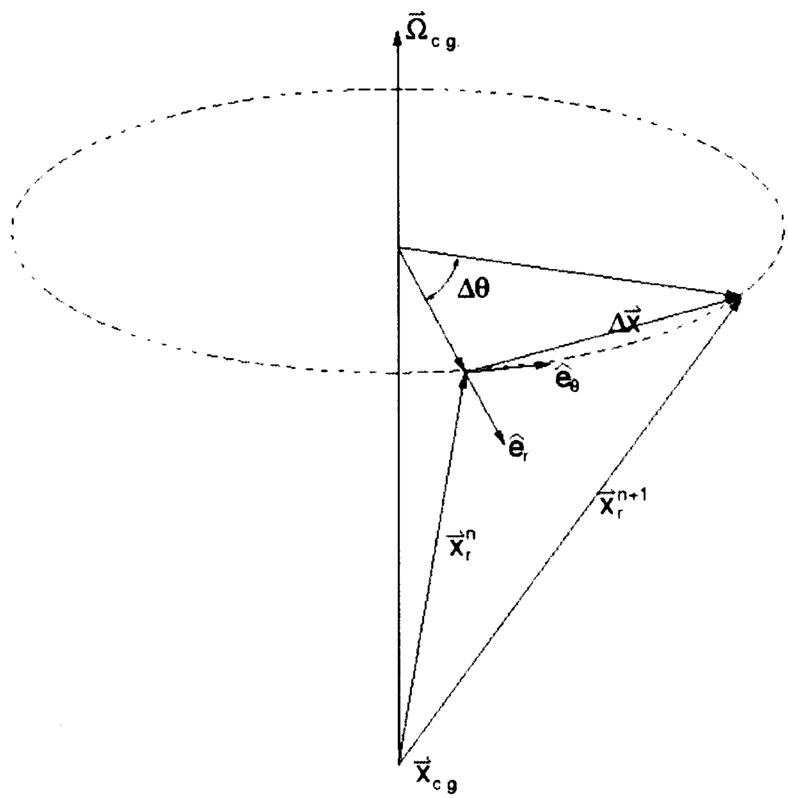


Figure 4.3 Rigid Body Rotation Coordinates

The unit vectors $\hat{e}_r$ and $\hat{e}_\theta$ are defined as

$$\hat{e}_r = \frac{\vec{\Omega}_{c.g.} \times \vec{x}_r}{\left|\vec{\Omega}_{c.g.} \times \vec{x}_r\right|}$$
$$\hat{e}_\theta = \frac{\hat{e}_\theta \times \vec{\Omega}_{c.g.}}{\left|\hat{e}_\theta \times \vec{\Omega}_{c.g.}\right|} \qquad \text{Equation 4.8}$$

If the solid body is also translating with, $\vec{v}_{c.g.}$ the n+1 position vector on the rigid body can be expressed as: $\vec{x}^{n+1} = \vec{x}^{n}_{c.g.} + \vec{v}_{c.g.}\Delta t + \vec{x}^{n+1}_r$ .... Equation 4.9

Where, $\vec{x}^{n+1}_r$ is given by above Equation 4.6. In the gear pump problem, however, the solid body only underwent rotation not translation.

CHAPTER 5

SOLUTION OF 2D GEAR PUMP FLOW USING BY FLUENT

5.1 Two Dimensional Gear Pump Model

For the flow analysis inside the gear pump, two dimensional (2D) gear pump model was used for the purpose of computational analysis in FLUENT. A three dimensional (3D) model was not considered, due to the complexity of a 3D moving mesh, the run time, and the size of files created after the simulation. The 2D simplification in the modeling, however, is expected to produce a reasonably accurate numerical analysis that will provide vital information for 3D modeling work in the future. A full sectional view of the gear pump is as shown in Figure 3.1.

5.2 Case Studies

A 2D gear pump model is proposed for flow analysis, and the commercial CFD software FLUENT is used to analyze the flow inside the gear pump. Using FLUENT, numerous parameters were found for different cases. The velocity vector, pressure contours, turbulence and flow rate are some of them. Among the issues deserving further investigation is the emulation of other camera functions like zooming and rotation as well as the rendering of perspective.

Over six different cases were examined for flow analysis in FLUENT (refer to Table 5.1) and two more cases were used for simulation to test the flow pattern with some geometry change in the casing. The computational analysis includes pressure contours, velocity vectors, and XY plot animation. From each case, the variables were analyzed and were compared with the other cases. The results are presented in Tables 6.1, and 6.2. The following cases were examined in FLUENT for the flow analysis.

Table 5.1 Case studies

Case	Rotational speed (rpm)	P_{out} (psi)	P_{in} (psi)	Casing geometry
1	3000	3500	30	Original
2	2000	1000	30	
3	2000	2500	30	
4	2000	3000	30	
5	2000	3500	30	
6	2000	3500	30	Modified

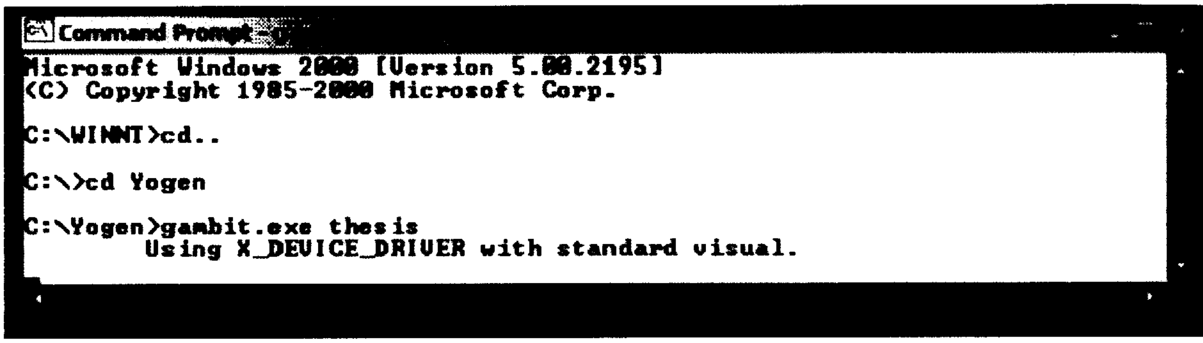
The following variables were analyzed from model:

1.Grid motion

2. Contours of velocity
   (a) Stream function
   (b) Velocity magnitude
3. Contours of pressure
   (a) Static pressure
   (b) Dynamic pressure
   (c) Total pressure
4. Path lines
5. Contours of wall fluxes (Wall shear stress)
6. XY plot :
   (a) Static pressure vs. position of interior
   (b) Velocity magnitude vs. position of interior
   (c) Static and dynamic pressure vs. curve length of driven gear
   (d) Static and dynamic pressure vs. curve length of driving gear
   (e) Velocity vs. curve length of both gears
   (f) Static pressure vs. direction vector of casing
7. Mass flow rate

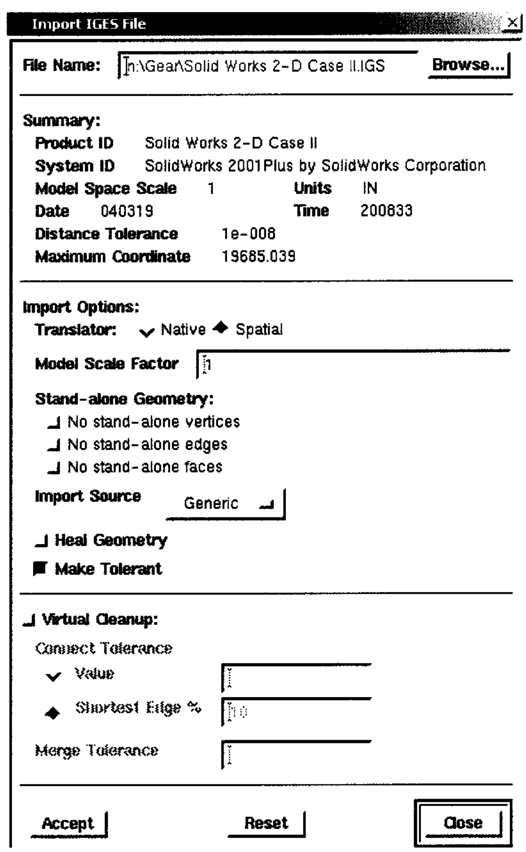
5.3 Geometry Setup and Mesh Generation in GAMBIT

Step 1: Start GAMBIT with ID as sub initial by using DOS command.



Step 2: Import IGES files from Solid Works source file for gears and casing.

File —> Import —> IGES ….. then browsing the right location for the IGES file and accept the three Solid Works IGES files of two rotors and casing to upload in GAMBIT.



Step 3: Selecting a Solver.

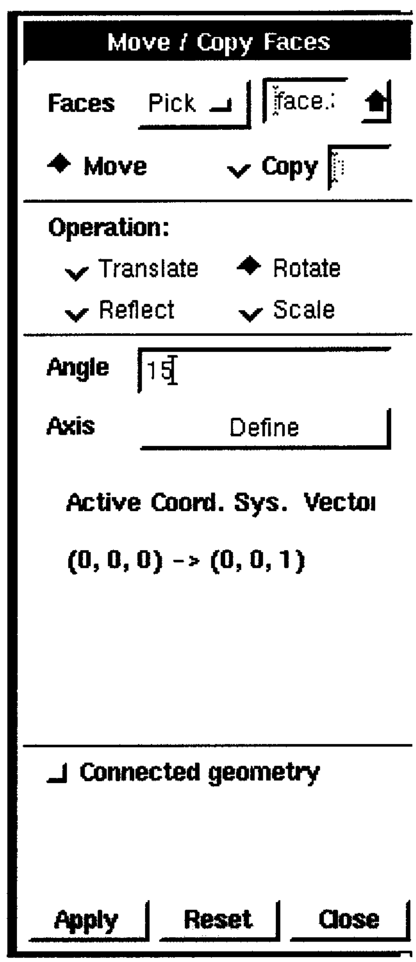
Choosing the solver for CFD calculation by selecting the following from the main menu bar:

Solver —> FLUENT 5/6

The solver currently selected can be seen at the top of the GAMBIT *GUI.*

Step 4: Scaling down, Rotating and translating faces.

GEMOETRY —> FACE —>Move/Copy Faces



First, the mesh file (from GAMBIT) and the C file (specifying dynamic mesh) were uploaded in the FLUENT to see whether the geometry of the gear pump needed revisions. Once the UDF was compiled, the geometry appeared acceptable. However, during the mesh motion preview, it was noticed that after some time steps, the gear teeth contacted each other. This is invalid as far as CFD simulation is concerned. This is due to fact that when gear teeth touch each other, there is no space left between the two gears to put mesh in it. Thus, for gear pump dynamic mesh simulations, it is recommended to avoid contact between gear teeth. There are two possible ways to achieve this:

1. Increase the center to center distance between the two gears. This is only possible in GAMBIT or any CAD package, where the geometry is created.

2. Keep the center to center distance constant but reduce the addendum, pitch circle diameters of the gears by some factor. This also is only possible in GAMBIT or any CAD package, where the geometry is created.

The second method was chosen in this research to avoid the gear teeth contact. First, driven gear face was translated to the origin of the axis and was rotated 15° about the origin. The faces of both rotors were then scaled down to 0.96 to create more clearance between the gear teeth. Finally, the driven gear was translated back to its original position. This is shown in Figure 5.1.

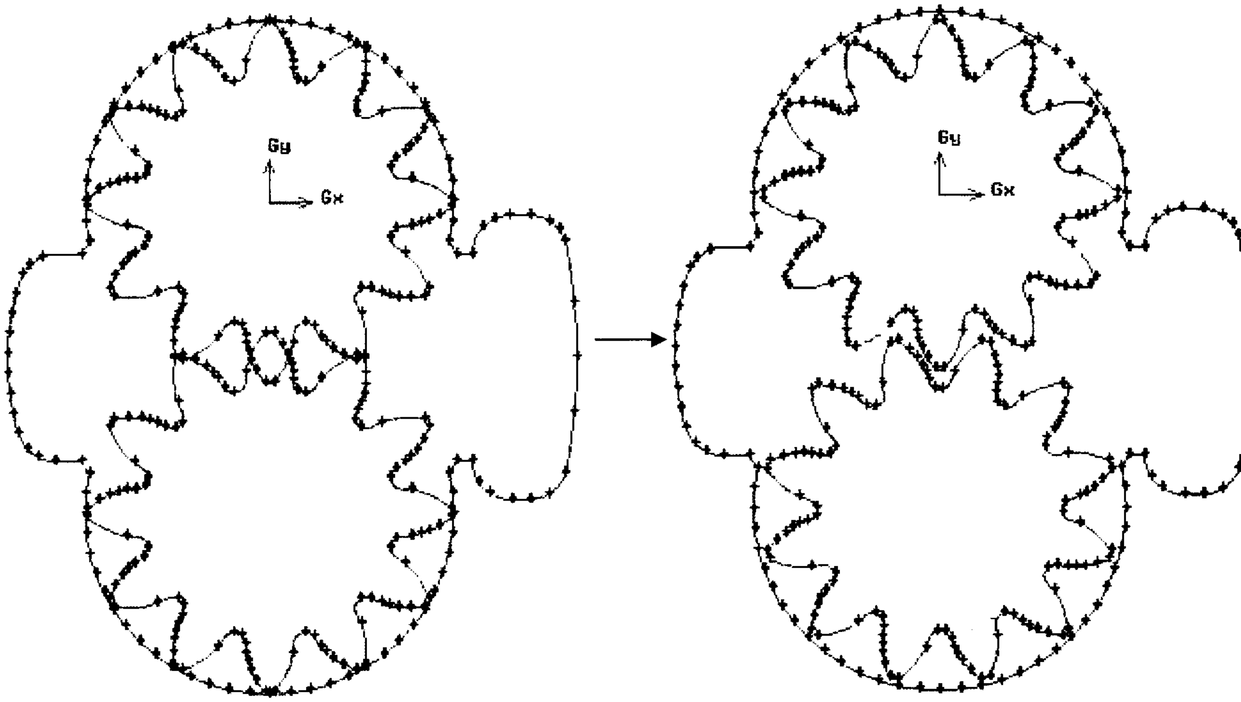
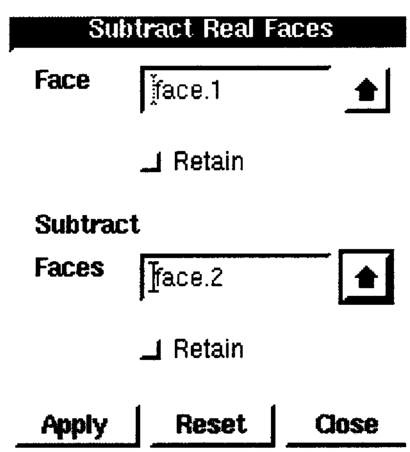


Figure 5.1 Translation, Rotation, and Scaling down of Faces in Gear Pump Model

Step 5: Subtracting faces to specify the fluid face

GEOMETRY  —> **FACE**  —> **Subtract Real Faces** 



The faces of the two gears were subtracted from the face of casing to specify the space enclosed between the inside the casing and the outside the gears. After subtracting, there was only one face enclosed between gears and casing, the fluid zone.

Step 6: Creating groups of edges

GEOMETRY —>GROUP —> Create Group

Five Groups were created to specify Inlet, Outlet, Casing, Driven Gear and Driving Gear using the edges. The respective group were named as in, out, cas, pos and neg as shown in Figure 5.2.

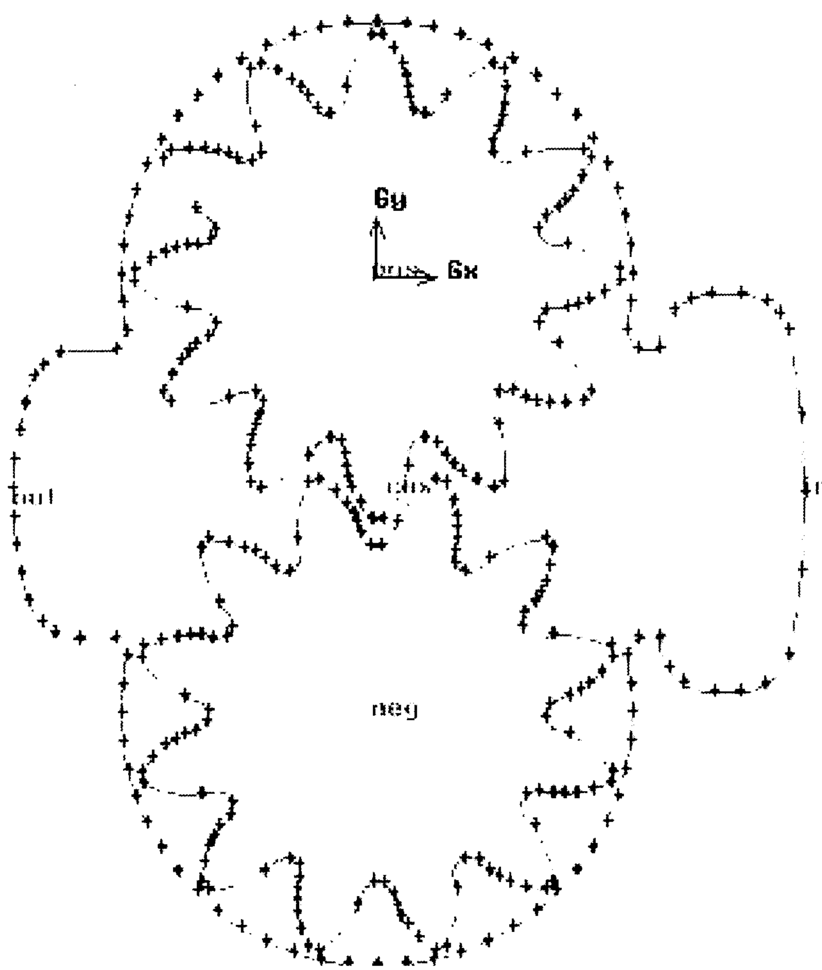
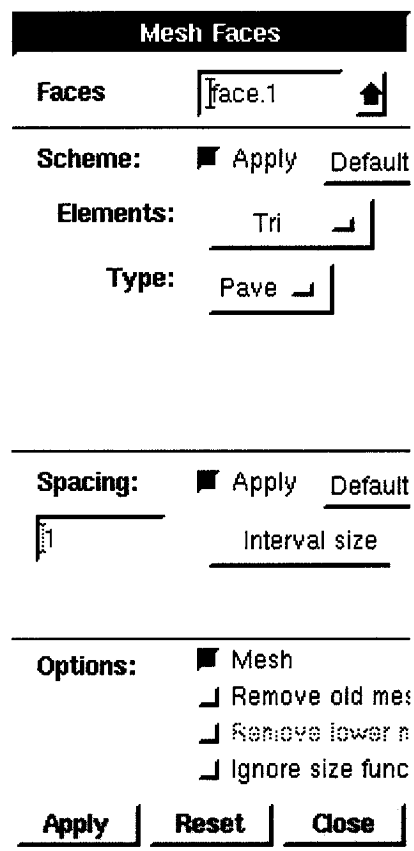


Figure 5.2 Graphics of Gear Pump showing Groups

Step 7: Creating meshes on fluid face

MESH —> FACE —> MESH FACES

This command sequence opens the Mesh Faces form. The interval size of the mesh was taken as the default value and the mesh was tri-pave meshing scheme for the moving dynamic mesh (MDM) setting up. This is shown in Figure 5.3.



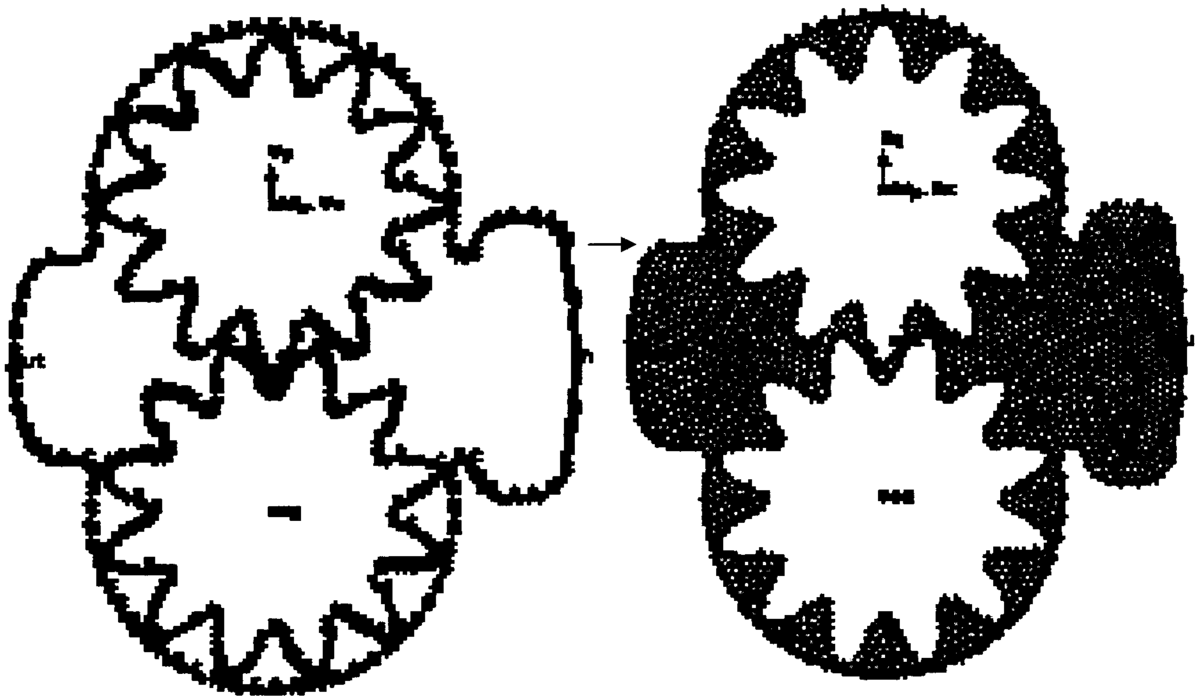


Figure 5.3 Meshes on the Gear Pump's Fluid Face

Step 8: Set Boundary Types

1. Hiding the mesh from the display before setting the boundary types makes it easier to see the edges and faces of the geometry. The mesh is not deleted, but rather removed from the graphics window.

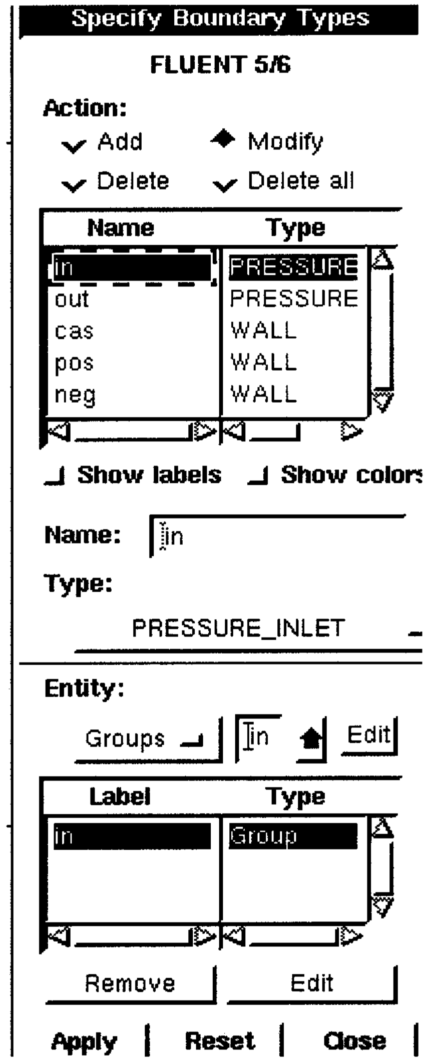
Click the **SPECIFY DISPLAY ATTRIBUTES** command button at the bottom of the **Global Control** tool pad.

b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

c) Click **Apply** and close the form.

2. Set boundary types for the entire gear pump.

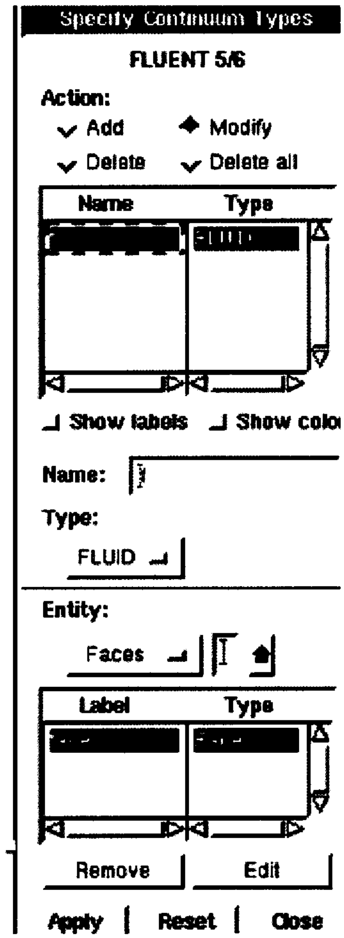
ZONES —> **SPECIFY BOUNDARY TYPES**



3. Set continuum zone types:

ZONES  **—>SPECIFY CONTINUUM TYPES** 

This opens the specify continuum window:



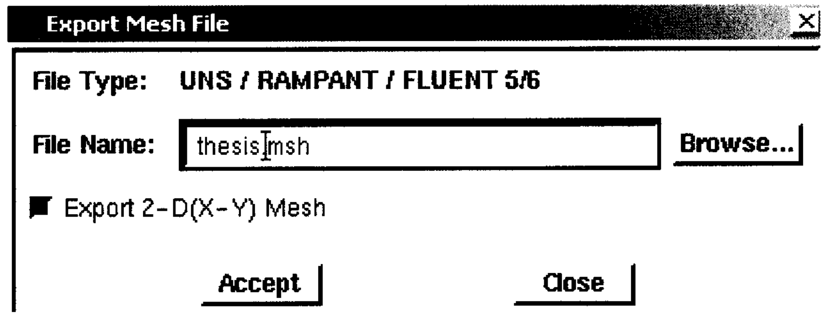
Here, "f" was specified which was taken as fluid flowing in the face1.

Step 9: Exporting the Mesh and Saving the Session

1. Export a mesh file for the gear pump.

File —> Export —> Mesh…

This command sequence opens the **Export Mesh File** for providing that the **File Type** is **Structured FLUENT 5/6 Grid**.



a) Enter the **File Name** for the file to be exported (`2-D thesis.GRD`).

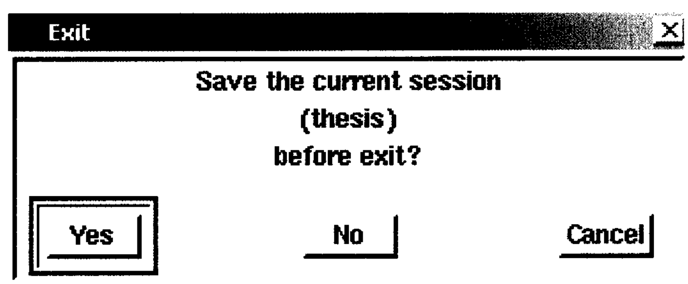
b) Click **Accept**.

The file will be written to the working directory.

2. Saving the GAMBIT session and exiting GAMBIT.

File —> **Exit**

GAMBIT *will ask whether to save the current session before you exit.*



Click **Yes** to save the current session and exit GAMBIT.

In summary, the IGES files from Solid Works source were imported to GAMBIT and manipulated using the necessary steps for mesh generation in the fluid face, including boundary types set up, and specification of the Fluent 5/6 solver. A final mesh file ready for CFD processing was the outcome.

5.4 Dynamic Mesh Setting and Calculating Results in FLUENT

1. Make sure that the user-defined function (UDF) syntax file to define the rigid-body motion of gear is included in the same working directory. This function was already named as "motion.c". This file will be needed to compile it within FLUENT.
2. Start the 2ddp version of FLUENT either using DOS command prompt from the working directory or simply going through the start menu.

Step 1: Grid

1. Read the grid file "thesis.msh".

File →Read →Case... browsing the right location of mesh file in the working directory.

```
     Welcome to Fluent 6.1.22

     Copyright 2003 Fluent Inc.
     All Rights Reserved

Loading "C:\Fluent.Inc\fluent6.1.22\lib\fl_s117.dmp"
Done.
Loading "N:\/.cxlayout"
Done.

> Reading "N:\Thesis1\Thesis.msh"...
    2382 nodes.
     200 mixed wall faces, zone  3.
     198 mixed wall faces, zone  4.
     198 mixed wall faces, zone  5.
       6 mixed pressure-outlet faces, zone  6.
      30 mixed pressure-inlet faces, zone  7.
    5885 mixed interior faces, zone  9.
    4134 triangular cells, zone  2.

Building...
     grid,
     materials,
     interface,
     domains,
     zones,
        default-interior
        in
        out
        cas
        pos
        neg
        f
     shell conduction zones,
Done.
```

2. Check the grid.

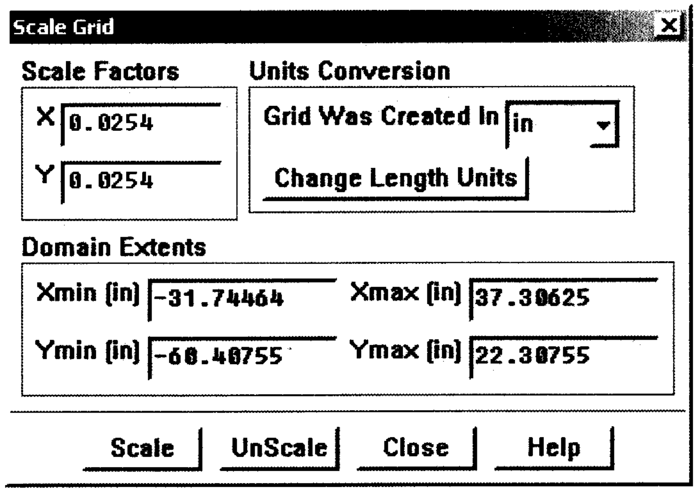
Grid →Check

While checking grids, just make sure that the cell volume does not detect negative volumes, as FLUENT can not do a calculation for a negative cell volume. If a negative volume is found, it would need to be re-meshed in order to make it non negative. Checking the grid shows the following display in the FLUENT window screen:

```
Grid Check

 Domain Extents:
   x-coordinate: min (m) = -3.174464e+01, max (m) = 3.730625e+01
   y-coordinate: min (m) = -6.040756e+01, max (m) = 2.230755e+01
 Volume statistics:
   minimum volume (m3): 1.729073e-01
   maximum volume (m3): 8.153766e-01
     total volume (m3): 1.926298e+03
 Face area statistics:
   minimum face area (m2): 5.114121e-01
   maximum face area (m2): 1.638899e+00
 Checking number of nodes per cell.
 Checking number of faces per cell.
 Checking thread pointers.
 Checking number of cells per face.
 Checking face cells.
 Checking bridge faces.
 Checking right-handed cells.
 Checking face handedness.
 Checking element type consistency.
 Checking boundary types:
 Checking face pairs.
 Checking periodic boundaries.
 Checking node count.
 Checking nosolve cell count.
 Checking nosolve face count.
 Checking face children.
 Checking cell children.
 Checking storage.
Done.
```

3. Scale the grid.

Grid →Scale...



(a) "**in**" was selected under **Units Conversion** from the drop-down list for **Grid Was Created In in** (inches).

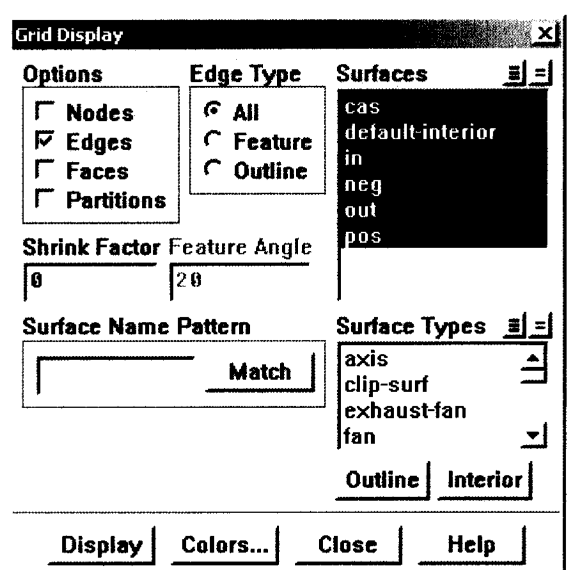
(b) Click **Scale** to scale the grid.

(c) Click **Change Length Units** as the working units for length now becomes inches, and then click close.

It must be noted that the numbers shown in the Domain Extents reflect the size of magnified model drawn in Solid Works. Therefore, the numbers should be further scaled down by the geometric scale factor of 39.18.

4. Display the grid [Figure 5.4].

Display →Grid...



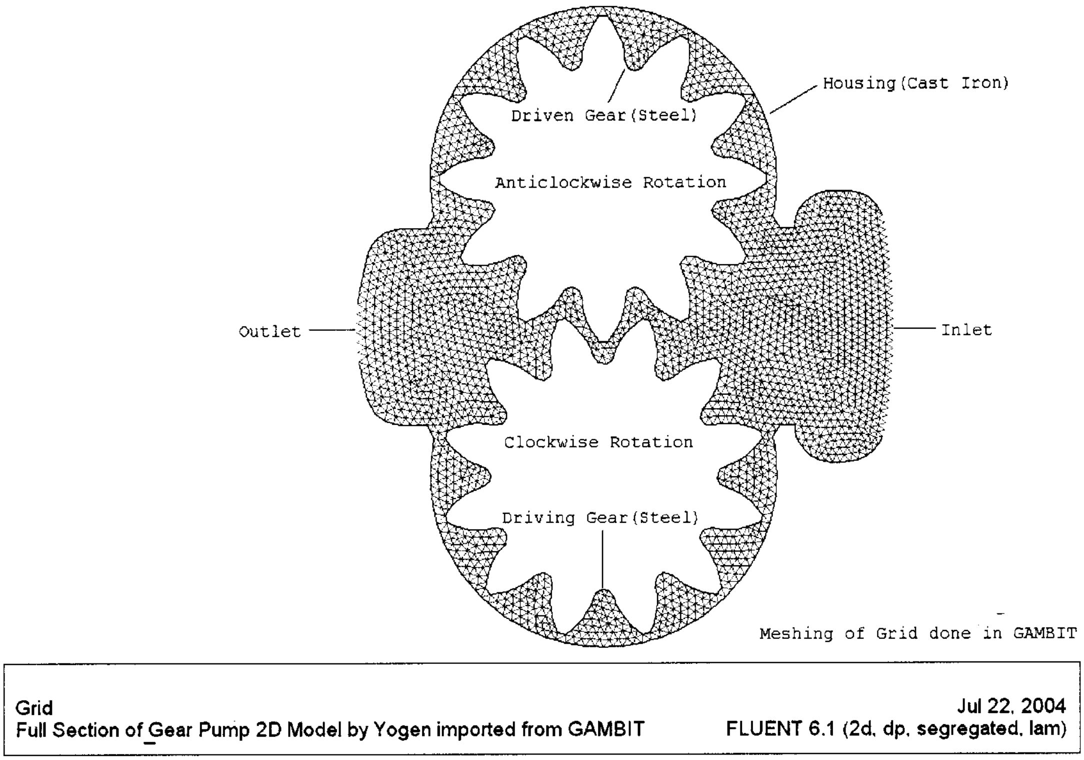


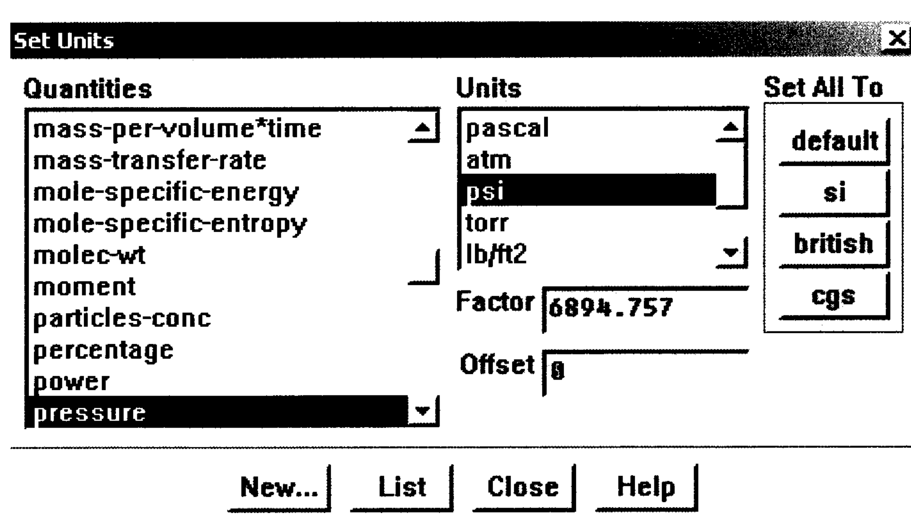
Figure 5.4 Initial grid displayed by FLUENT

Step 2: Units

1. For convenience, define new units for pressure and mass flow.

   Pressure, length, and temperature are specified in psi, inch, and Fahrenheit respectively. The units for length were already changed while scaling the grid.

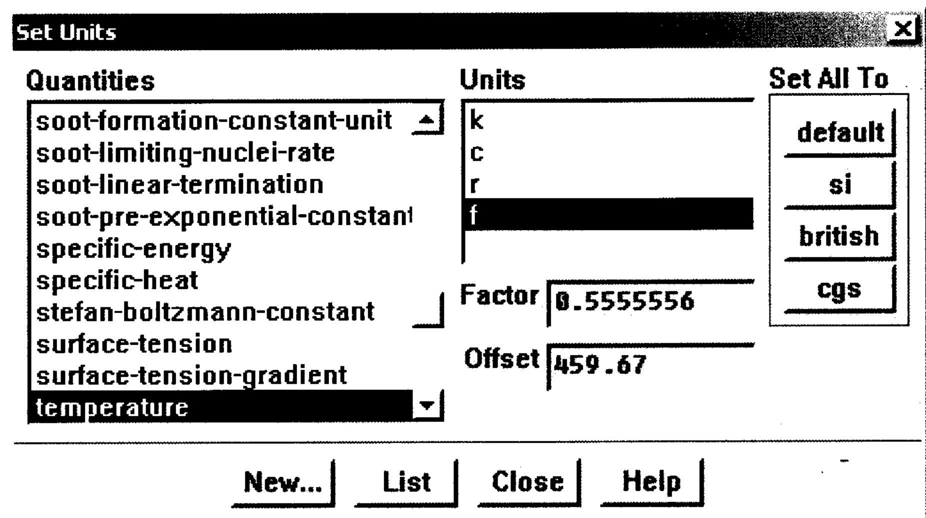
Define →Units...



(a) Select **pressure** under **Quantities**, and **psi** under **Units**.

(b) Select **temperature** under **Quantities**, and click **f** under **Units**.

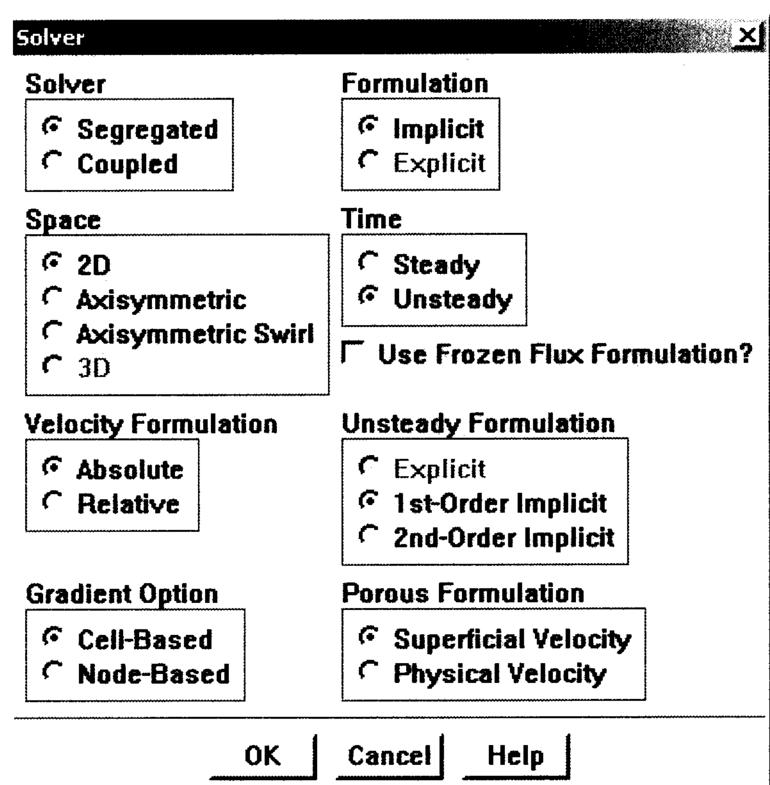
The **Define Unit** panel will be displayed as:



Step 3: Models

1. Enable a 2D time-dependent calculation.

Define →Models →Solver...



(a) Under **Space**, click on **2D**.

(b) Click on **Unsteady** under **Time**.

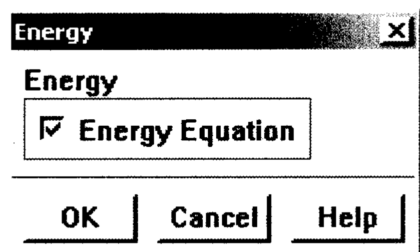
(c) Keep the default **Unsteady Formulation** option of **1st-Order Implicit**.

Keep in mind that dynamic mesh simulations work only with first order time variant for tri/tetra mesh, re-meshing, and smoothing.

(d) Click on **OK**.

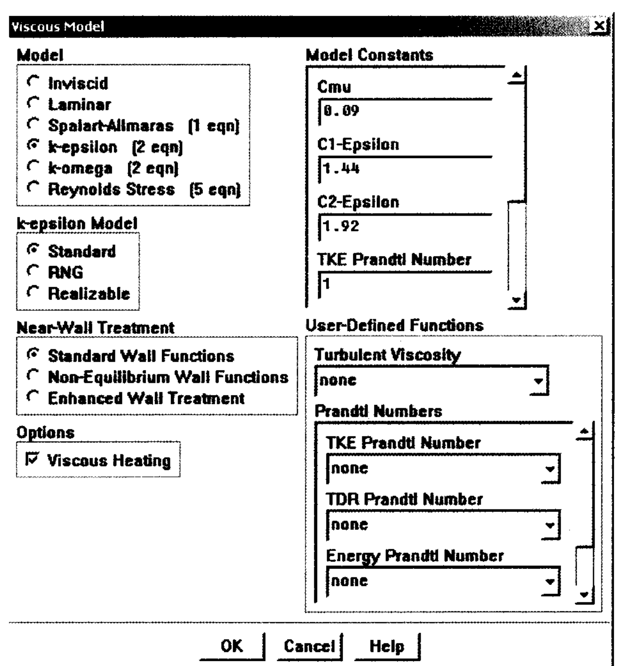
2. Turn on the Energy Equation for viscous heating properties for oil.

Define →Models →Energy...



3. Turn on the standard **k-** ϵ turbulence model.

Define →Models →Viscous...



(a) Check on **k-epsilon** as the **Model**, and use the default setting of **Standard** under **k-epsilon Model.**

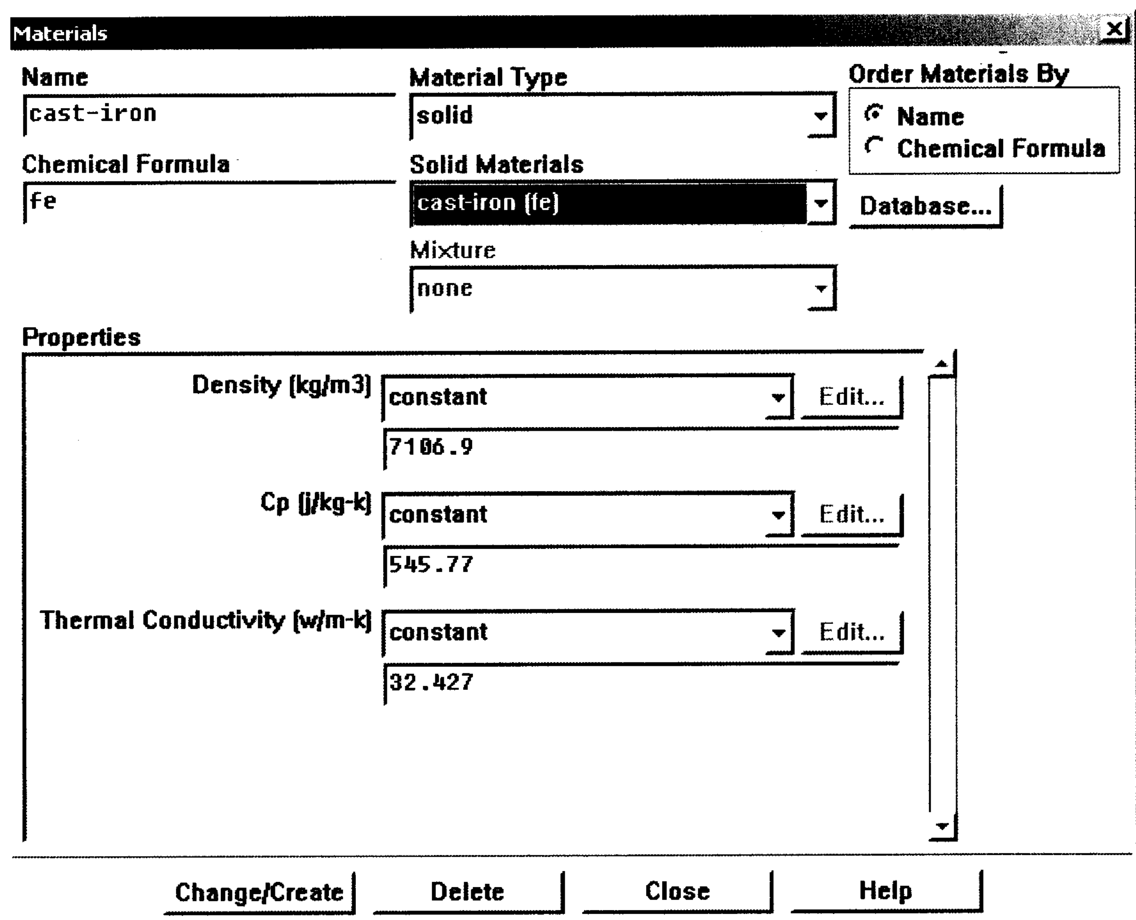
(c) Check on **Viscous Heating** under Options.

(d) Click on **OK**.

Step 4: Materials

A new material called cast iron was created for housing material. Engine oil and steel were loaded from the FLUENT database for the fluid and solid material properties used to represent the fluid face and the gears.

Define →Materials



The physical and thermal properties of cast iron were taken from ALGOR database.

(a) In the **Material Type** field, select **solid** form the drop down list.

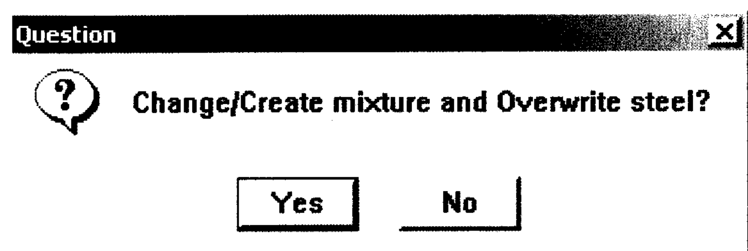
(b) In the **Name** field, enter **cast iron** and type **fe** in **Chemical Formula** field.

(c) Specify 7160.9 for the **Density**.

(d) Specify 545.77 for $\mathbf{C_p}$.

(e) Click **Change/Create**.

(f) Click **No** for not to overwrite steel on aluminum.

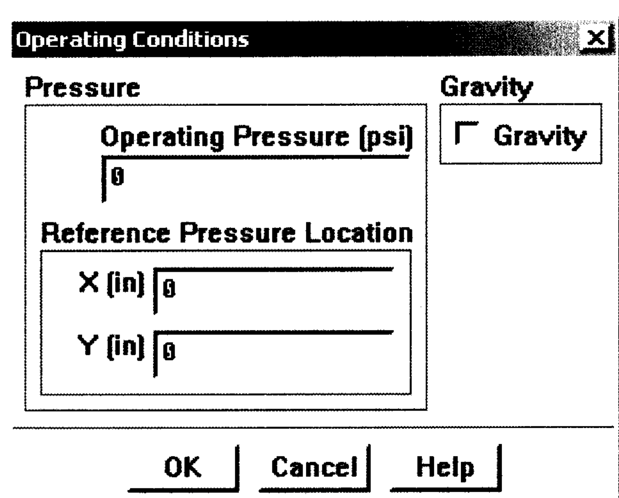


Two other materials engine oil as fluid and steel as solid were uploaded from the FLUENT Database.

Step 5: Operating Conditions

Set the operating pressure to 0 psi.

Define →Operating Conditions...

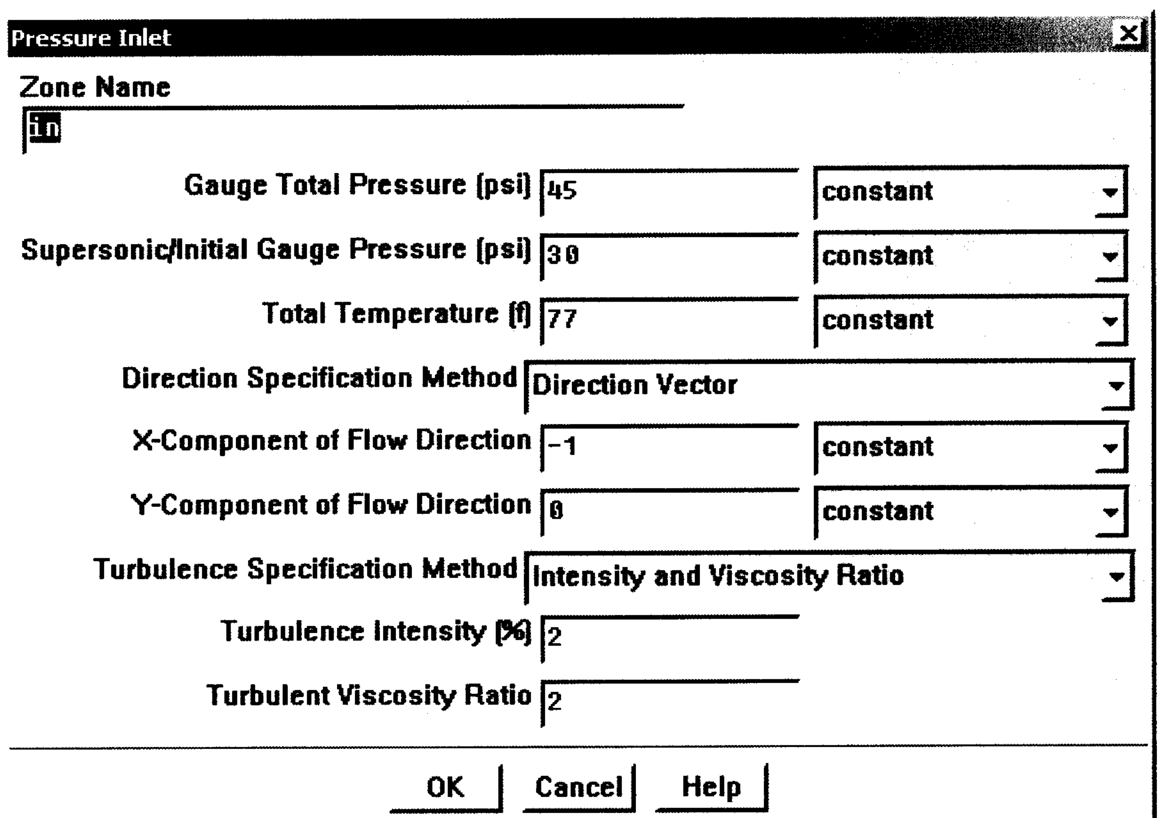


Step 6: Boundary Conditions

Dynamic mesh motion and all related parameters are specified using the items in the **Define/Dynamic Mesh** submenu, not through the **Boundary Conditions** panel. Here, Pressure inlet at "in" zone and pressure outlet at "out" zone were set up as well as "steel" was chosen as the material for the driving and driven gears. "Cast iron" was picked up as the material for the housing and "engine oil" for the fluid meshed face. Inlet zone temperature was assumed as 77° F (25°C) and back flow total temperature from the outlet zone was assumed as 113° F (45° C). Turbulence intensity (%) and turbulent viscosity ratio for inlet zone were assumed 2% and 2 respectively. For the outlet zone, backflow turbulence intensity (%) and backflow turbulent viscosity ratio were taken as 10 % and 10 respectively. The pressure inlet and pressure outlet were used 30 psi and 2000 to 3500 psi respectively.

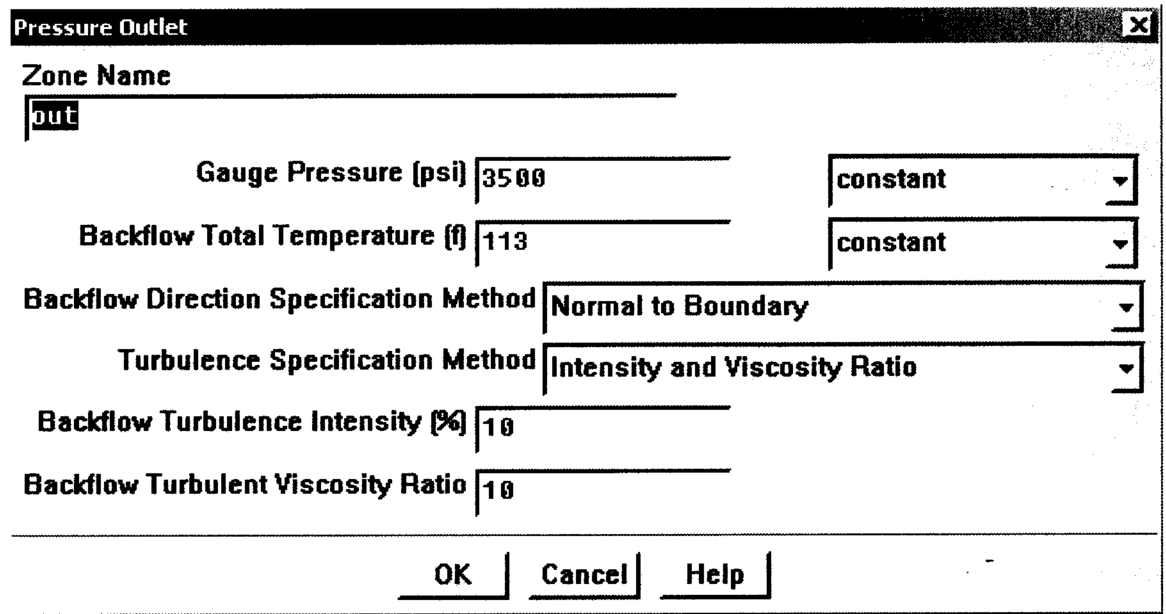
Define → Boundary Conditions...

Set the conditions for the pressure inlet (**inlet**) as shown in the following figure.



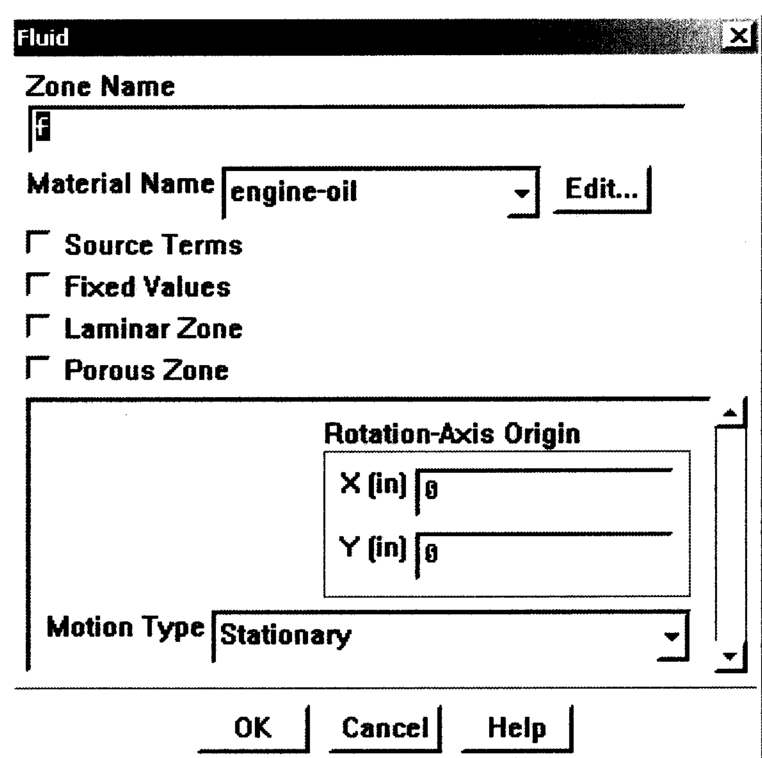
2. Click **OK**.

3. Set the conditions for the exit boundary (**outlet**) as shown in the following figure.

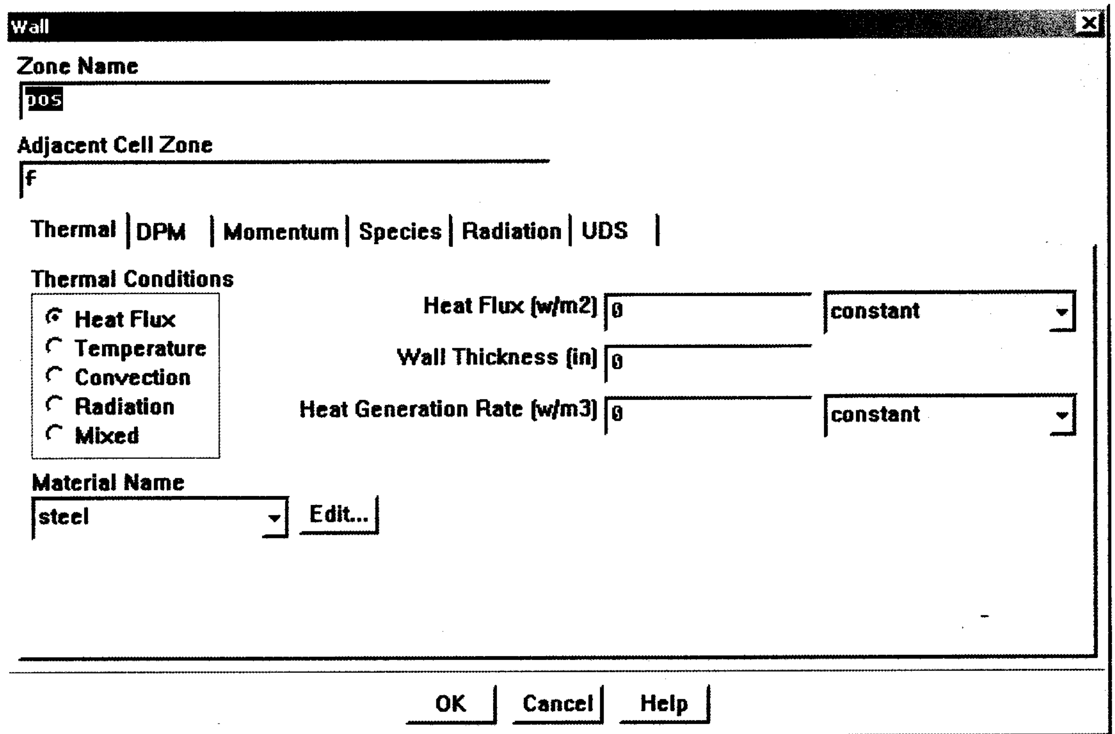


4. Click **OK**.

5. Set the fluid material "engine oil" for the grid face as shown below:



6. Get engine oil selecting from the drop down list in **Material Name**. Click OK.

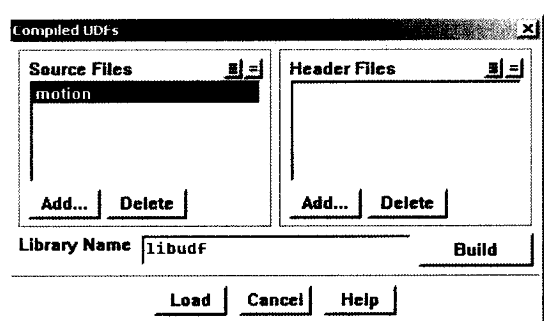


7. Choose steel as the material for driven gear "pos" as shown above and similarly for driving gear "neg" too. Choose cast iron as the material for housing "cas" in the similar way. Click OK for every three wall windows.

Step 7: Mesh Motion

1. Read in and compile the user-defined function (UDF).

Define →User-Defined →Functions →Compiled...



(a) click **Add...** under **Source Files**,

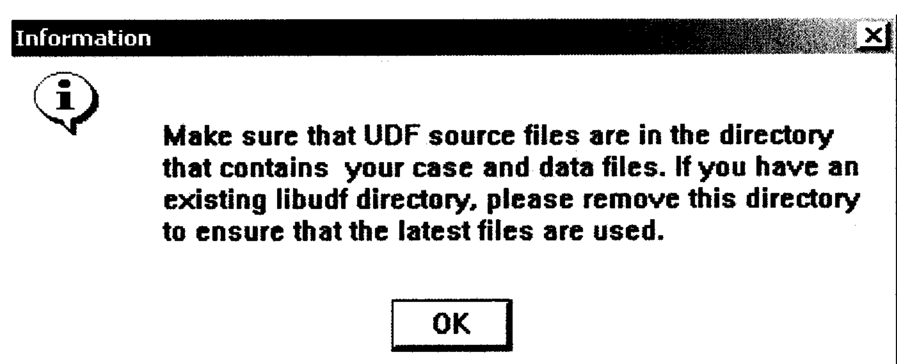
A **Select File** panel will show up. (**Appendix 1**)

(b) Choose the source code **motion.c** in the **Select File** panel, and click on **OK**.

(c) Click on **Build** in the **Compiled UDFs** panel, The user-defined function was already been defined. Compiling the UDF in FLUENT creates a library with the default name "libudf" in the working directory.

```
        1 file(s) copied.
(system "move user_nt.udf libudf\ntx86\2ddp")0
(system "copy C:\Fluent.Inc\fluent6.1.22\src\makefile_nt.udf libudf\ntx86\2ddp\makefile")        1 file(s) copied.
0
(chdir "libudf")()
(chdir "ntx86\2ddp")()
motion.c
# Generating udf_names.c because of makefile motion.obj
udf_names.c
# Linking libudf.dll because of makefile user_nt.udf udf_names.obj motion.obj
Microsoft (R) Incremental Linker Version 6.00.8447
Copyright (C) Microsoft Corp 1992-1998. All rights reserved.

   Creating library libudf.lib and object libudf.exp

Done.
```

(d) Click on **OK** in the dialog box which will show up as shown below. It is just a warning to remove any other "libudf" directory existing in the working directory.

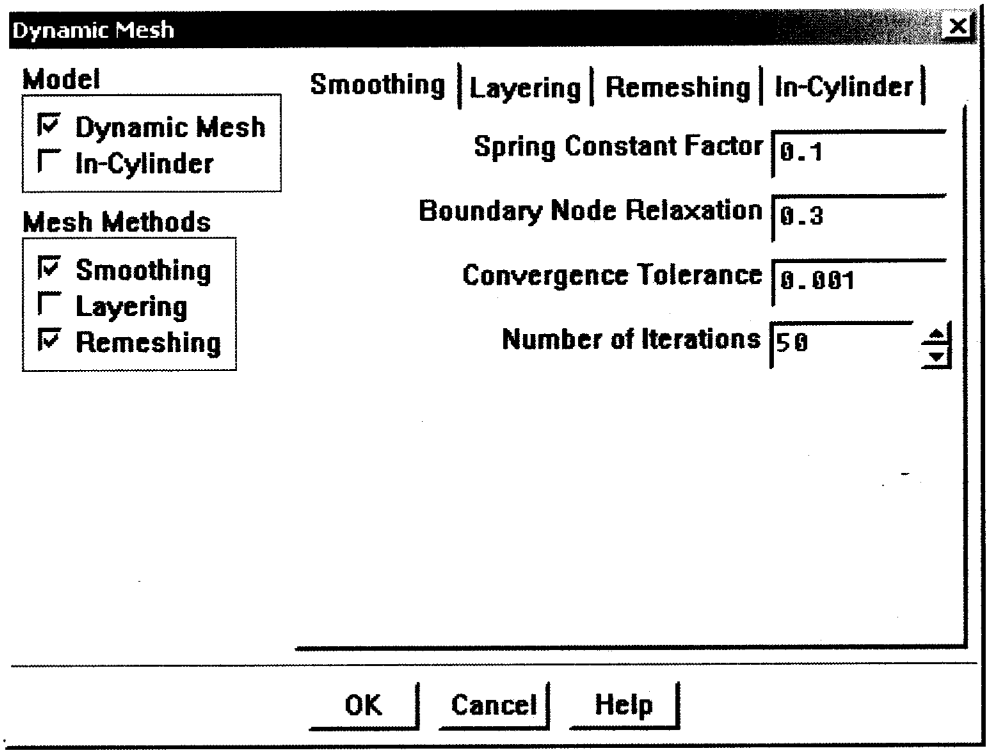


(e) Click on **Load** to upload the user-defined function library for the iteration.

```
Opening library "libudf"...
Library "libudf\ntx86\2ddp\libudf.dll" opened
        clock_wise
        anticlock_wise
Done.
```

2. Activate dynamic mesh motion and specify the associated parameters.

Define →Dynamic Mesh →Parameters...



(a) Select **Dynamic Mesh** under **Model**.

(b) Under **Mesh Methods**, select **Smoothing** and **Re-meshing**.

FLUENT automatically re-meshes and smoothes the existing mesh zones for use of the different dynamic mesh methods where applicable.

(c) Setting up the parameters under **Smoothing** as follows:

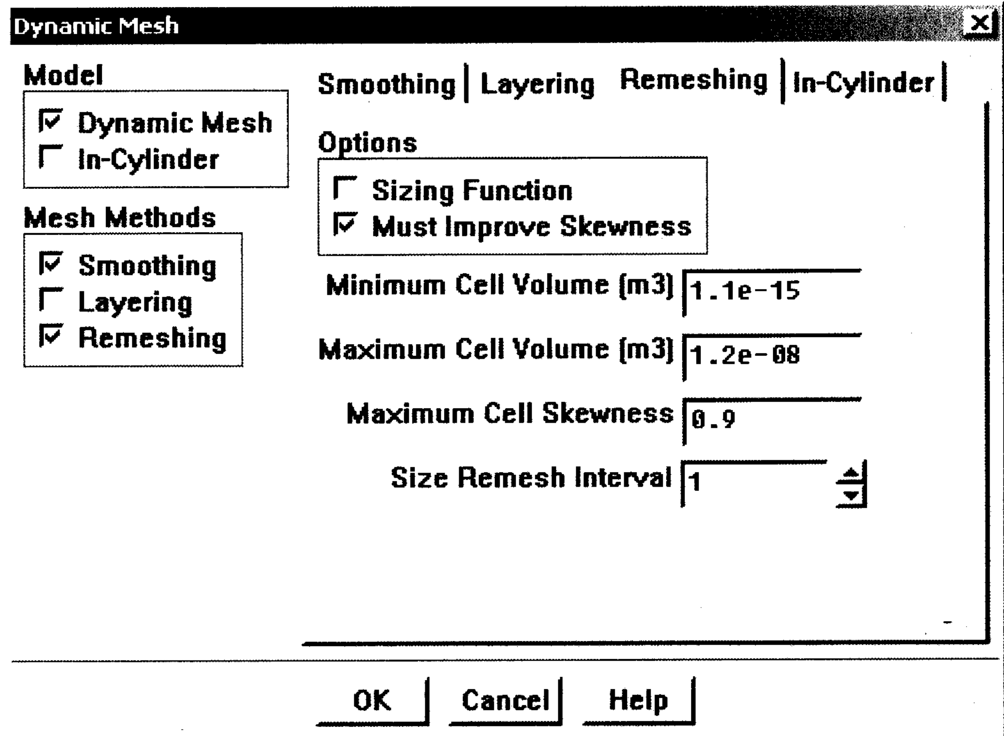
i. Specifying 1 for the **Spring Constant Factor**.

ii. Specifying 0.3 for the **Boundary Node Relaxation**.

iii. Keeping up the default specification of 0.001 for the **Convergence Tolerance**.

iv. Specifying 50 for the **Number of Iterations.**

(d) Setting up the parameters under **Re-meshing** as follows:



i. Under **Options**, be sure that the **Must Improve Skewness** option is selected.

ii. Specify 1.1e-15 m^3 for the **Minimum Cell Volume**.

iii. Specify 1.2e-08 m^3 for the **Maximum Cell Volume**.

iv. Keeping up the default value of 0.9 for the **Maximum Cell Skewness**.

v. Specify 1 for the **Size Re-mesh Interval**.

Any cells exceeding these limits will be re-meshed and smoothed automatically.

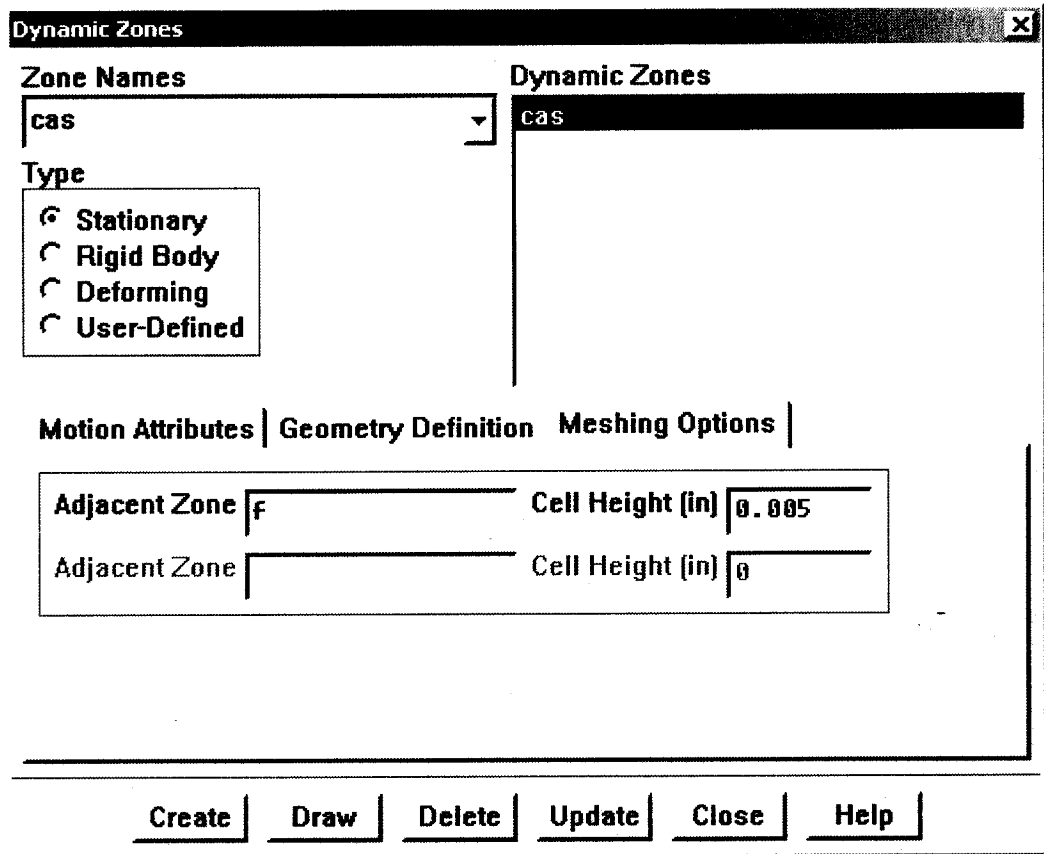
(e) Click on **OK**.

3. Specify the motion of the gears and the housing.

The motion of two gears and the stationary wall "housing" were specified by using UDFs.

Define →Dynamic Mesh →Zones...

(a) Specify the motion of the stationary housing named as 'cas'.



i. Selecting housing '**cas**' in the **Zone Names** drop-down list.

ii. Selecting **Stationary** under **Type**.

iii. Click the **Meshing Options** tab.

vi. Specify `0.005` in for **Cell Height**.

vii. Click on **Create**.

(b) Specify the rigid body motion of the driving gear

i. Select the **driving gear** named as **"pos"** in the **Zone Names** drop-down list,

ii. Under **Type**, keep the default selection of **Rigid Body**.

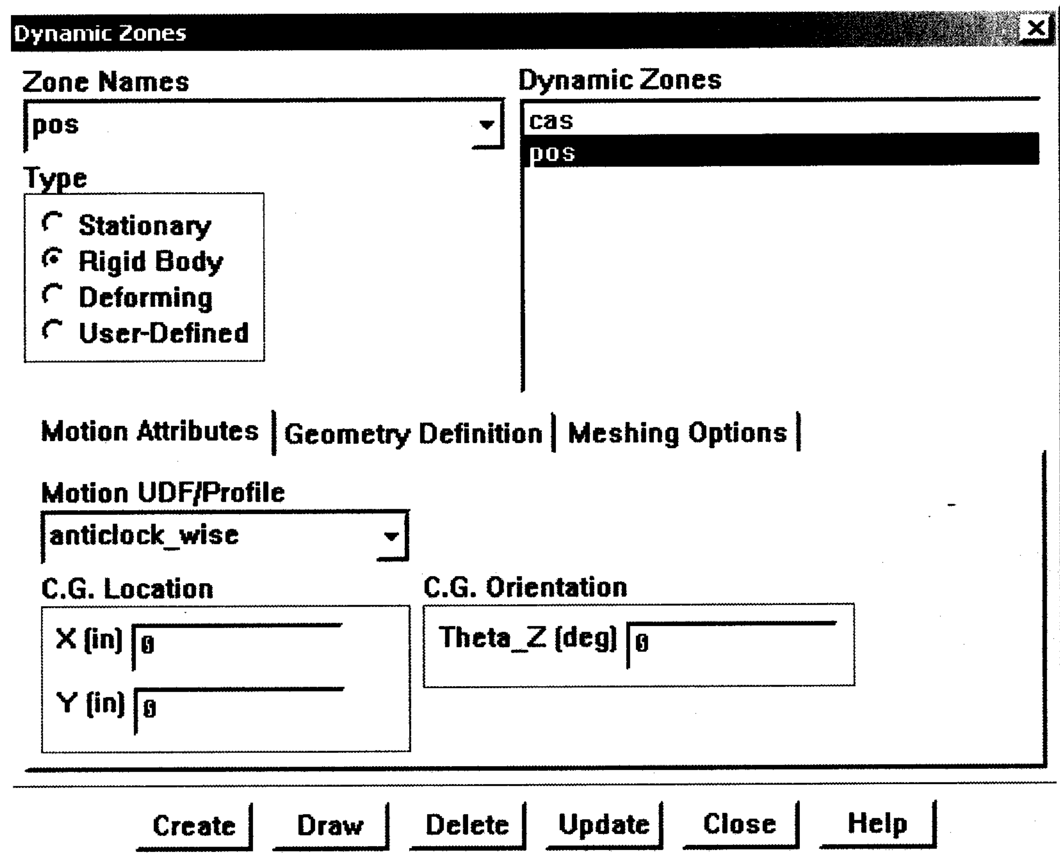
iii. Under **Motion Attributes**, select **anticlock_wise** in the **Motion UDF/Profile** drop-down list.

iv. Keep the default values of (0, 0) m for **C.G. Location**, and 0 for **C.G. Orientation**. The position of the CG will be updated automatically by **FLUENT** based on the input of motion.

v. Click on the **Meshing Options** tab.

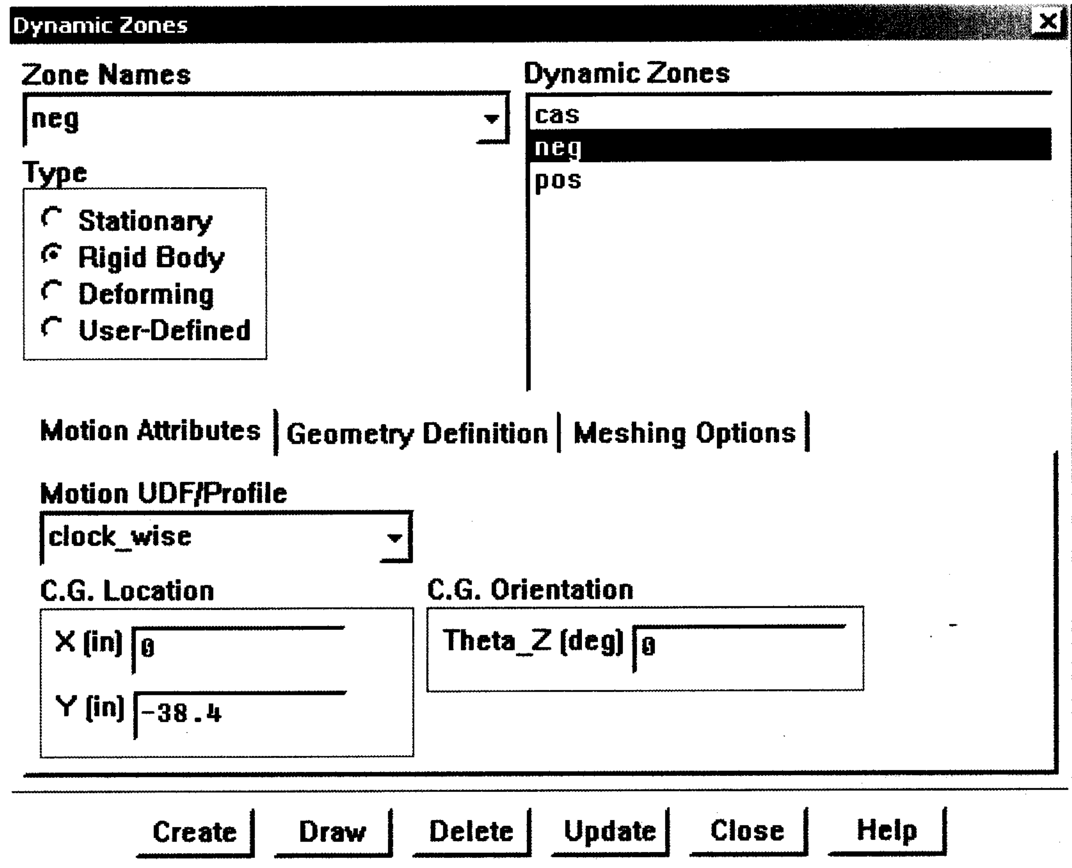
vi. Specifying 0.005 in for **Cell Height**.

vii. Click **Create**.



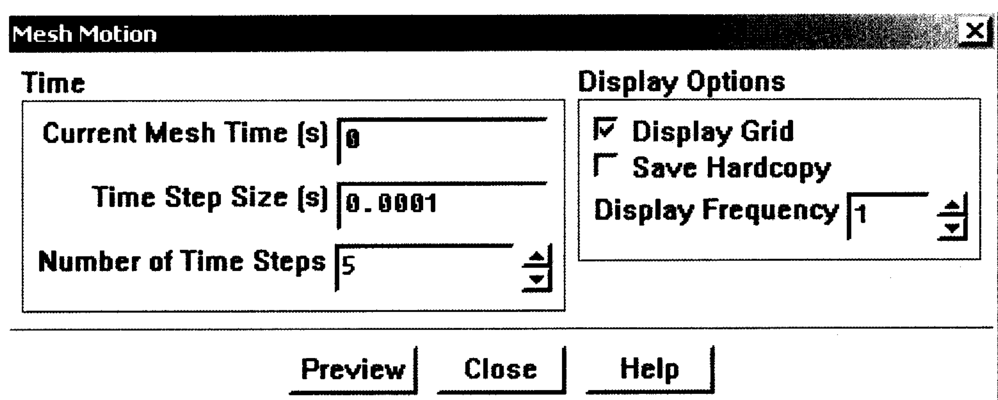
(c) Specify the motion of the driven gear

All of the steps were taken in the similar way as of the driving gear. There was a change in CG location; the CG location of the **driven gear "neg"** was taken as (0, -38.4 in.). Since the geometry is magnified in Solid Works as stated on page 39, the CG location of the driven gear of the model used in this research is (0, -38.4 in.). In fact, the CG location of the driven gear in the actual physical model is (0, -0.98 in.). All other settings were unchanged from that of the driving gear "pos". The Dynamic Zones panel for the driven gear "neg" is as follows:



(d) Preview of **Mesh Motion**

Solve →Mesh Motion...

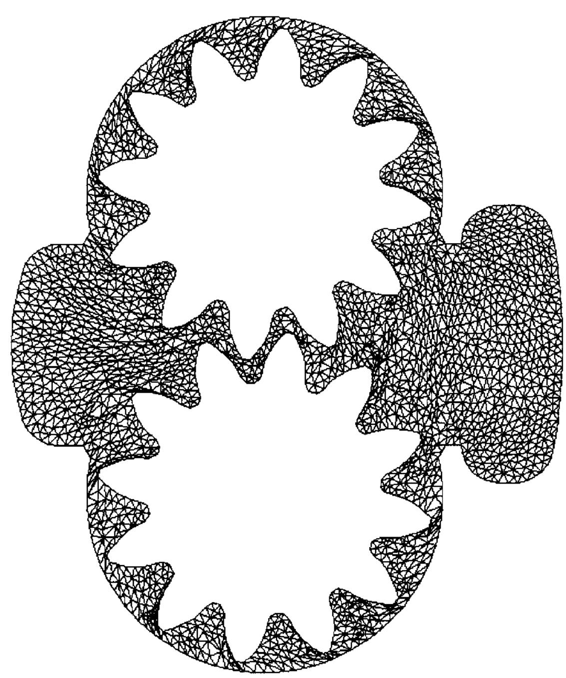


(i) Type 0.0001 in Time Step Size and 5 in the Number of Time Steps.

(ii) Check **Display Grid** under Display Options to see the mesh motion in grid.

(iii)Click on **Preview** to view the mesh motion in gear pump model with rotational motions in both gears. The graphics window of mesh motion in the grid was displayed with the following message in the FLUENT window screen. Also the stretching, skewing, and collapsing of cells were seen in grid graphics (refer to Figure 5.5)

```
Updating mesh to time 2.63500e-01 (step = 02635)
This is Anti Clockwise Gear

CG_Omega for anticlock_wise: 0, 0, 210

CG Position for anticlock_wise: 0, 0, 1.32136e-306

CG Orientation for anticlock_wise: 0, 0, 1.32136e-306

This is Clockwise Gear

CG_Omega for clock_wise: 0, 0, -210

CG Position for clock_wise: 0, -0.97536, 1.32136e-306

CG Orientation for clock_wise: 0, 0, 1.32136e-306

Mesh Statistics:
Min Volume    = 1.78291e-05
Max Volume    = 1.04916e-03
Max Cell Skew = 9.23842e-01 (cell zone 2) Done.
```



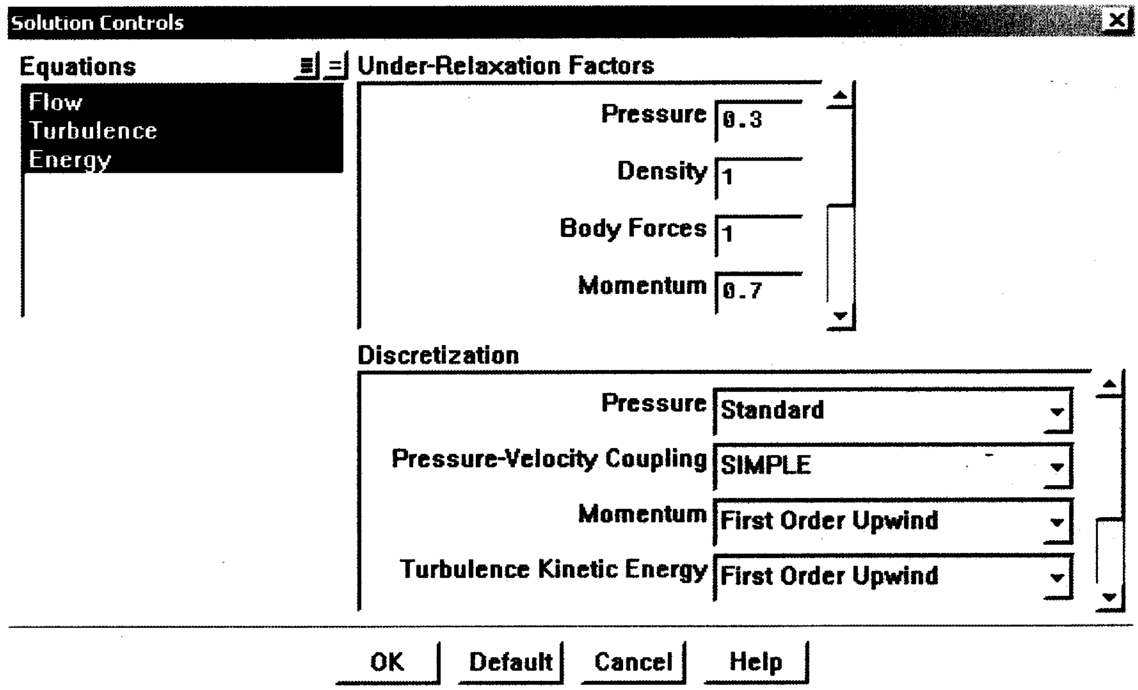
Gear Pump Model by Yogen (Mesh Motion Preview)
Jul 30, 2004
FLUENT 6.1 (2d, dp, segregated, dynamesh, ske, unsteady)

Figure 5.5 Mesh Motion Preview

Step 9: Solution

1. Set the solution parameters.

Solve → Controls→ Solution...

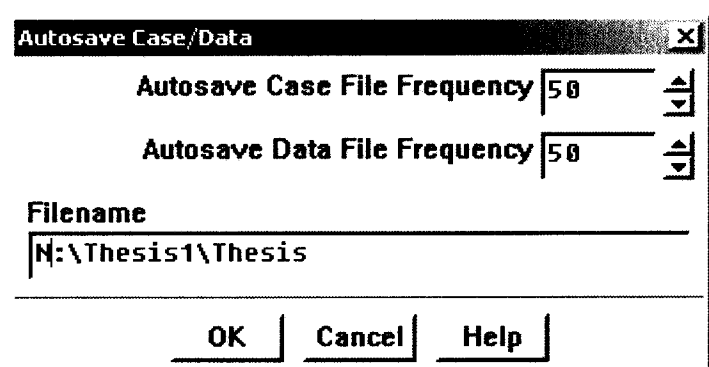


(a) Keep all default discretization methods and values for under-relaxation factors.

(b) Click on OK.

2. Request that case and data files are automatically saved every 50 time steps.

File →Write →Autosave...



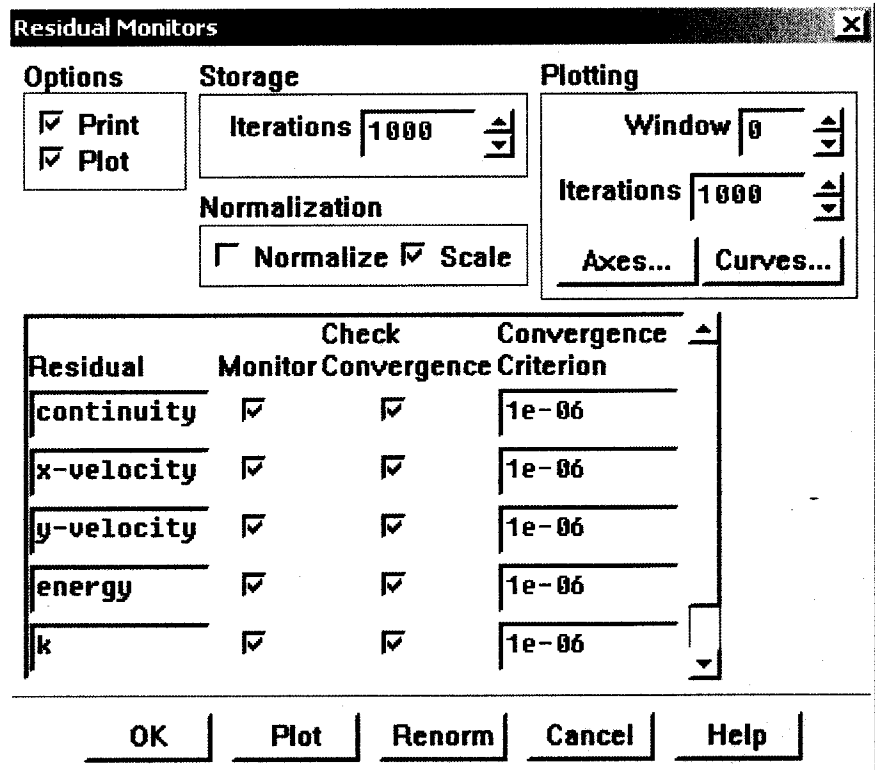
(a) Set both **Autosave Case File Frequency** & **Autosave Data File Frequency** to 50.

(b) In the **Filename** field, enter thesis.

(c) Click **OK**.

3. Enable the plotting of residuals during the calculation.

Solve →Monitors →Residual...



4. Initialize the solution.

Initializing the flow field at this point will display contours and vectors that can be used to define animations.

Solve →Initialize →Initialize...

(a) In Solution Initialization panel, select "**in**" from the drop down lists of **Compute from** menu. Then, FLUENT will automatically set up the following parameters from the inlet "in" boundary conditions as shown below:

(i) **Gauge Pressure** to 30 psi.

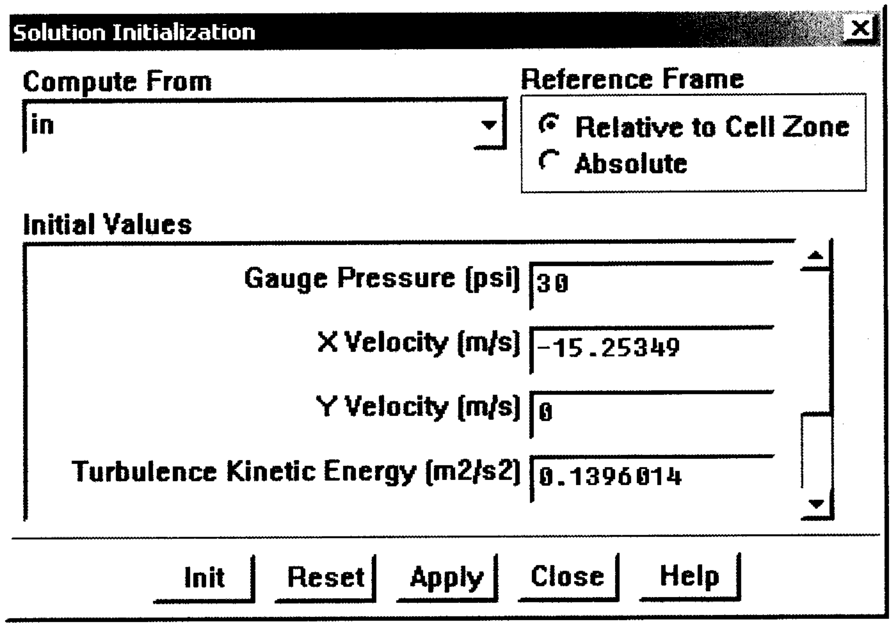
(ii) **X Velocity** to -15.25349 m/s

(iii) **Y Velocity** to 0 m/s.

(iii) **Turbulence Kinetic Energy** to 0.1396014.

(iv) **Turbulence Dissipation Rate** to 16.8147.

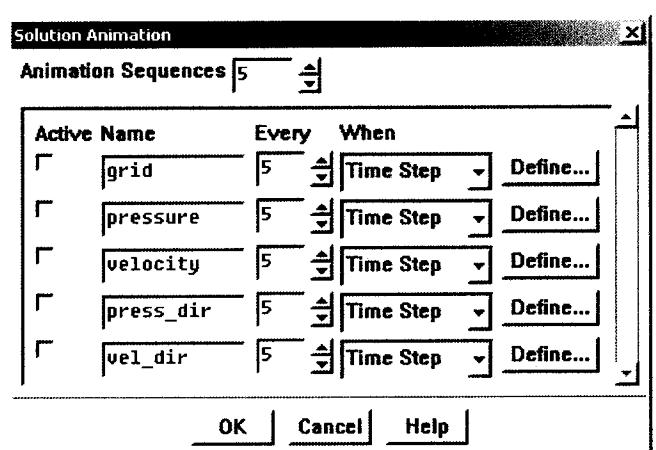
(v) Click **Init**, **Apply** and **Close**.



5. Create animation sequences for the static pressure contour plots and velocity vectors plots in the gear pump.

Solution animation features of **FLUENT** were used to save contour plots of the particular animation sequence in every 5 time steps. After completing the calculation, solution animation playback feature could be used to view the animated sequence plots over time. Basically, there are five types of animation sequence displays over the time; they are grid, contours, vectors, XY plot etc. For this problem, grid animation, velocity and pressure contours, velocity and pressure direction vectors were picked for animation sequences over 5 time steps.

Solve →Animate →Define...



(a) Increase the number of **Animation Sequences** to 5.

(b) Under **Name**, enter grid for the first animation, and pressure for the second one, velocity for the third, pressure_dir and vel_dir for the latter two.

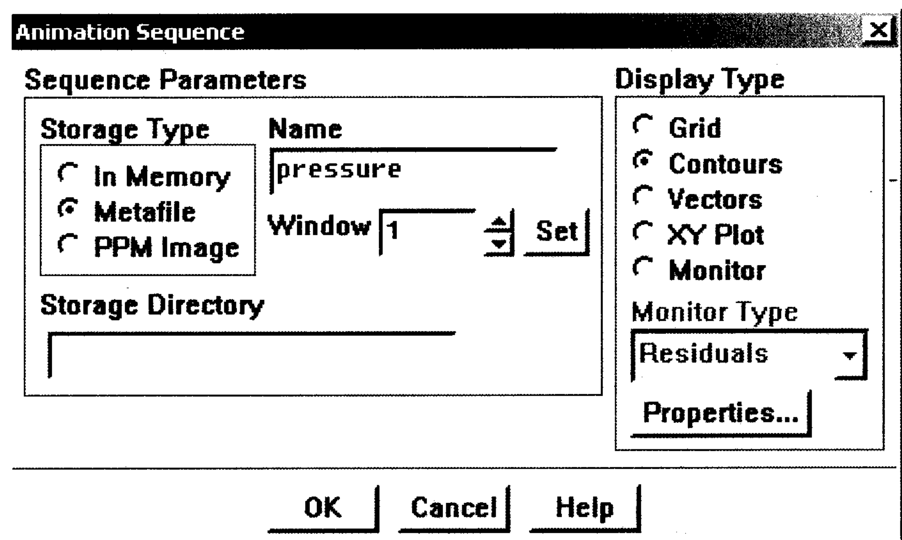
(c) Under **Every**, increase the number to 5 for all five sequences.

(d) In the **When** drop-down list, select **Time Step**.

(e) Define the animation sequence for the second animation sequence pressure.

i. Click **Define...** on the line for pressure to set the parameters for the sequence.

The **Animation Sequence** panel will open as:



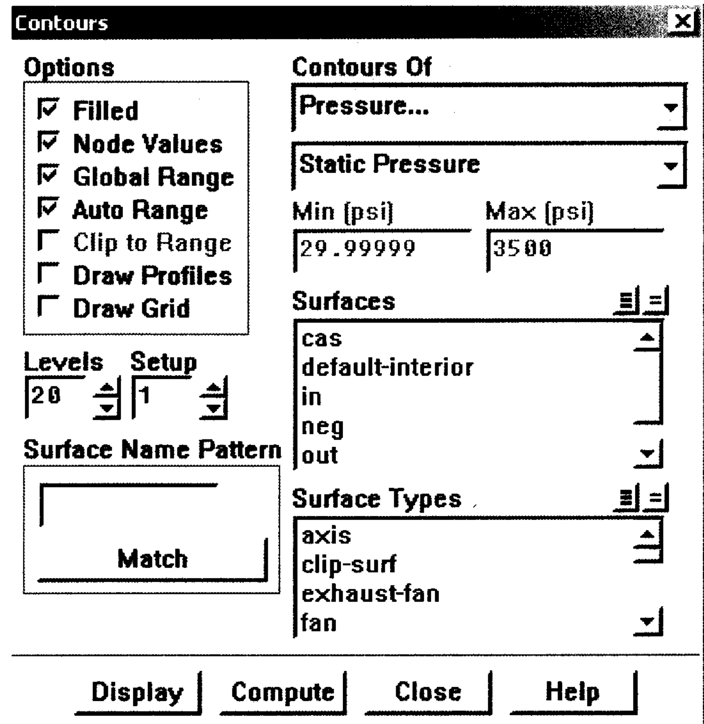
ii. Below the **Storage Type**, keep the default selection of **Metafile**.

iii. Increase the **Window** number to `1` and click **Set**.

Graphics window number 1 will be displayed.

iv. Under the **Display Type**, select **Contours**.

The **Contours** panel will be displayed.



v. Under **Options**, turn on **Filled**.

vi. In the **Contours Of** drop-down lists, select **Pressure...** and **Static Pressure**.

vii. Click **Display** (refer Figure 5.6)

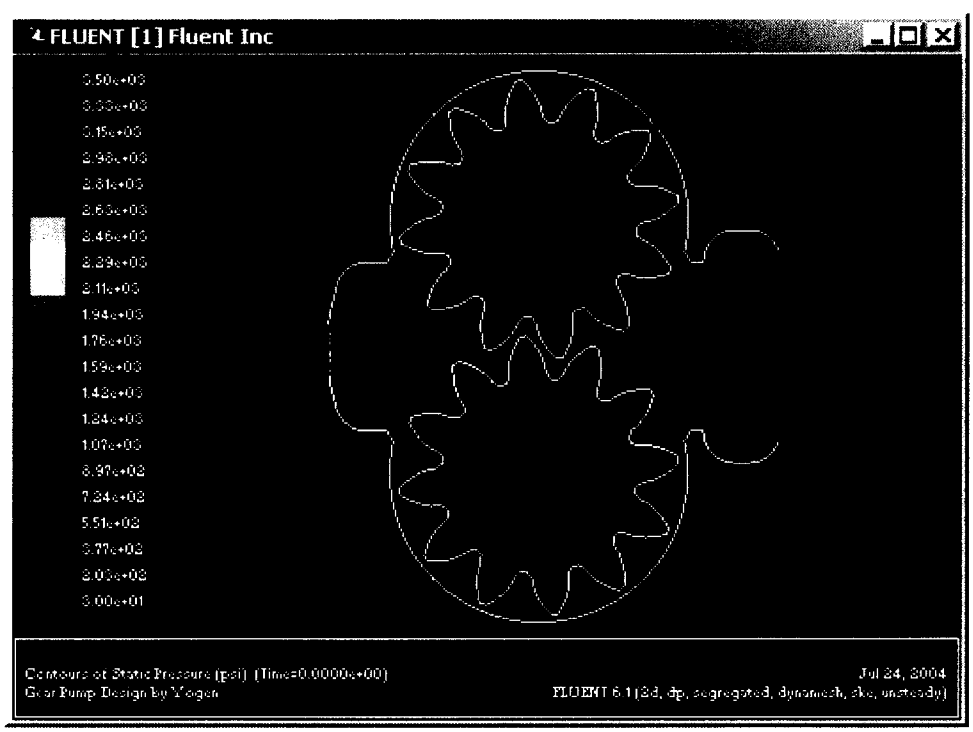


Figure 5.6 Static Pressure Contours at t = 0 s

viii. Click **OK** in the **Animation Sequence** panel.

The **Animation Sequence** panel will close, and the checkbox in the **Active** column next to pressure in the **Solution Animation** panel will become selected.

ix. Click **OK** in the **Solution Animation** panel.

(f) Define the animation sequence for the velocity vectors.

i. Click **Define...** on the line for vel_dir to set the parameters for the sequence.

The **Animation Sequence** panel will open. It is same as pressure contour panel.

ii. Under **Storage Type**, keep the default selection of **Metafile**.

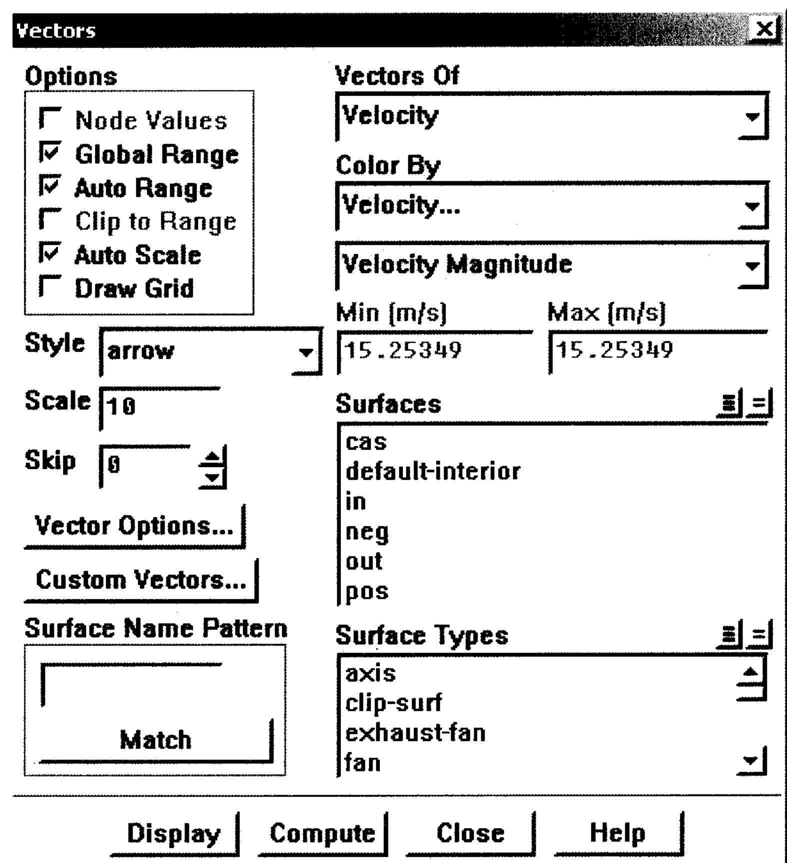
iii. Increase the **Window** number to 2 and click **Set**.

Graphics window number 2 will open.

iv. Under **Display Type**, select **Vectors**.

v. Type 10 in the **Scale** panel to see the velocity vectors distinctive.

The **Vectors** panel will open and it will show Min(m/s) and Max (m/s) 15.25349 when clicking on **Display**.



v. Click on **Display** in the **Vectors** panel (refer Figure 5.7)

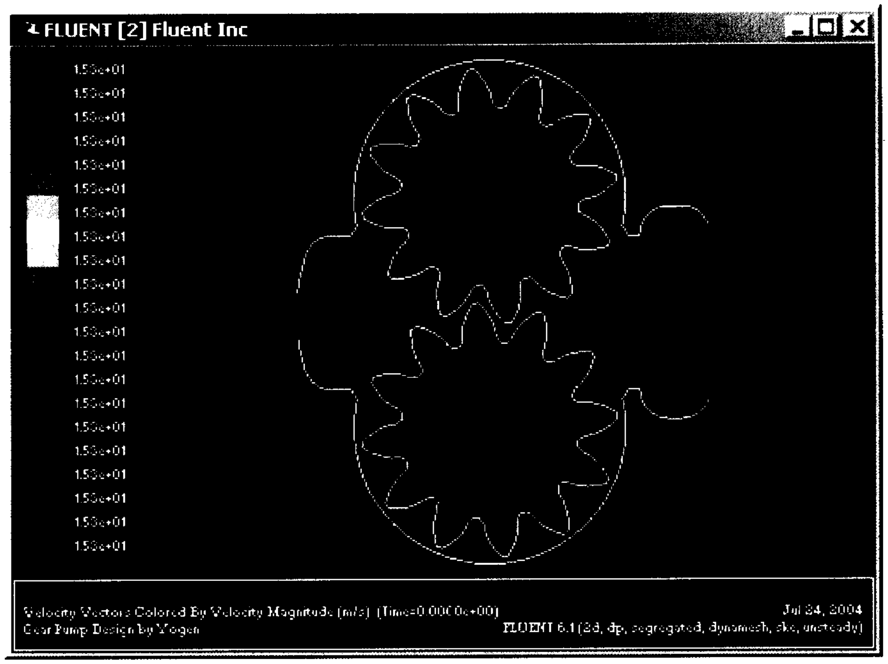


Figure 5.7 Velocity Vectors at t = 0 s

vi. Click **OK** in the **Animation Sequence** panel.

The **Animation Sequence** panel will be closed, and the checkbox in the **Active** column next to vel_dir in the **Solution Animation** panel will become selected.

vii. Click **OK** in the **Solution Animation** panel.

6. Set the time step parameters for the calculation.

Solve →Iterate...

(a) Set the **Time Step Size** to 0.0001 s.

(b) Increase the **Max Iterations per Time Step** to 200.

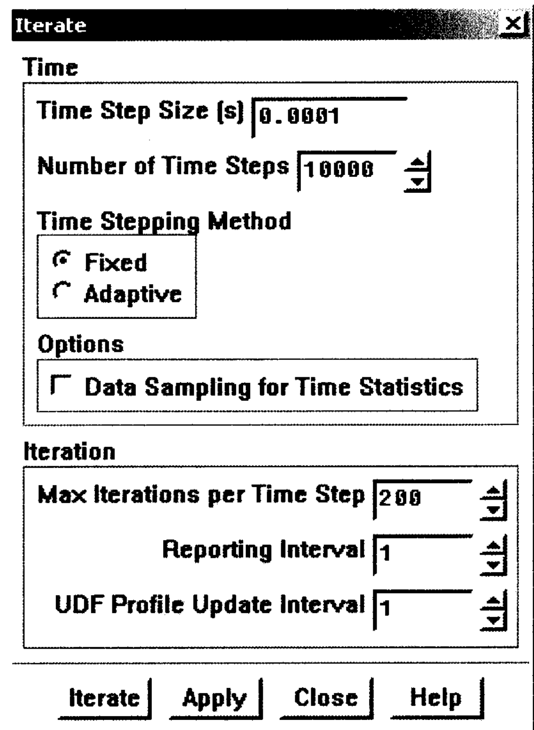
(c) Click **Apply**.

7. Save the initial case and data files (thesis.cas and thesis.dat).

File →Write →Case & Data...

8. Request 100 time steps. For better results at total time t = 1 s, 10,000 time steps were set up.

Solve →Iterate...



Max Iterations per time step, reporting interval & UDF profile update interval can be changed. The following graphics window of iteration (refer to Figure 5.8) was after time t =1.15 sec for the case of gear pump with N = 2000 rpm and P_{in} = 30 psi, P_{out} = 3500 psi.

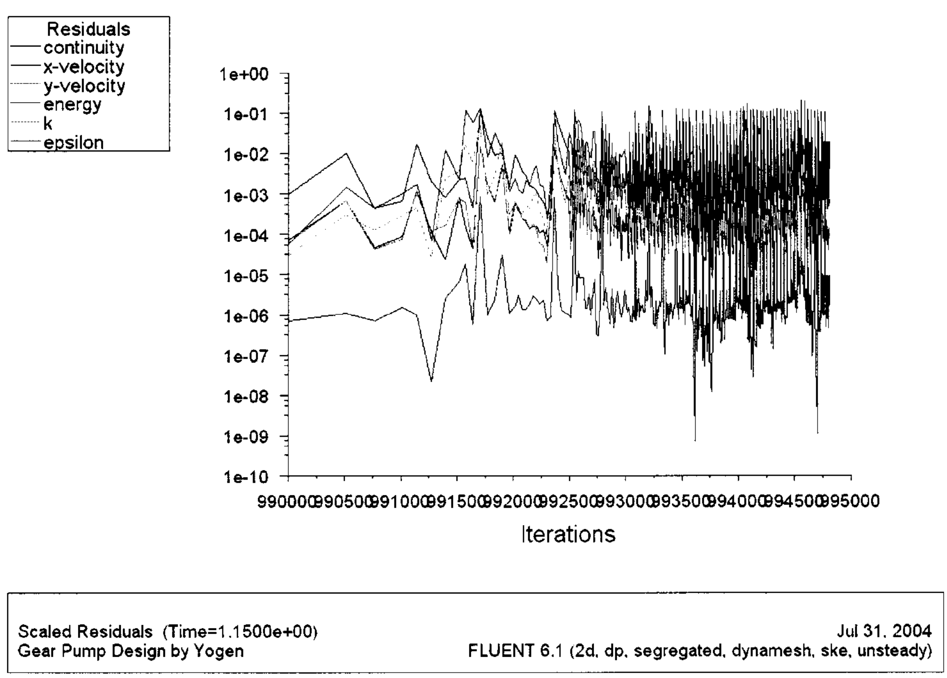


Fig 5.8 Iterations until time t = 1.15 sec for the case: N = 2000 rpm & P_{out} = 3500 psi

Step 10: Postprocessing

1. Inspect the solution at the final time step.

   (a) Observe the contours of static pressure in the gear pump (Figure 5.9).

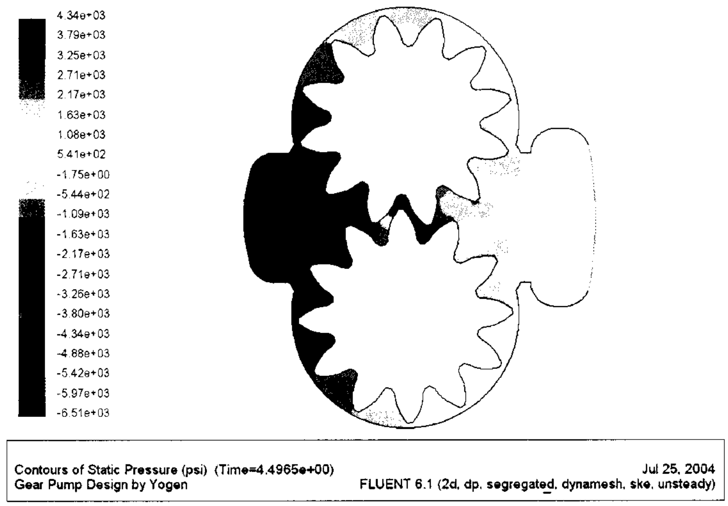


Fig 5.9 Static Pressure Contours at t = 4.4965 s

   (b) Observe the velocity vectors in the gear pump (Figure 5.10).

2. Optionally, inspect the solution at different intermediate time steps.

   (a) Read in the corresponding case and data files (thesis.cas and thesis.dat).

   File →Read →Case & Data...

   (b) Display the desired contours and vectors.

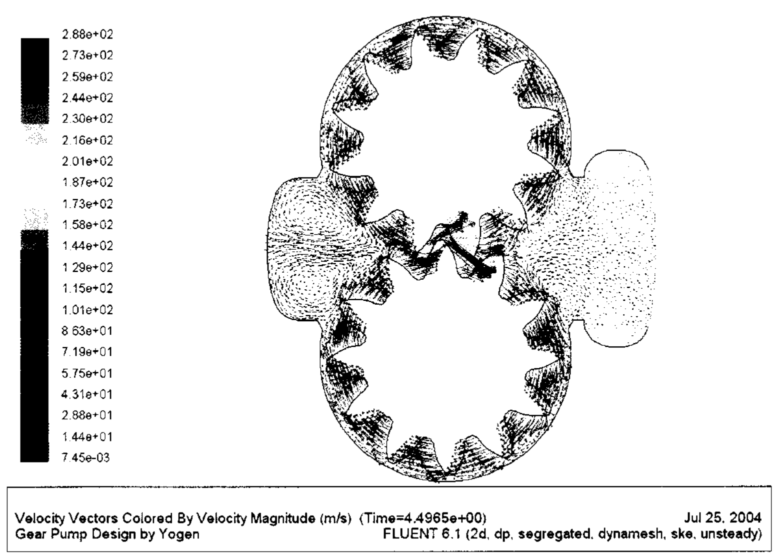


Fig 5.10 Velocity Vectors at time t = 4.4965 s

3. Play back the animation of the pressure contours. A plot at time t = 4.4965 s is shown in Figure 5.11.

Solve → Animate → Playback...

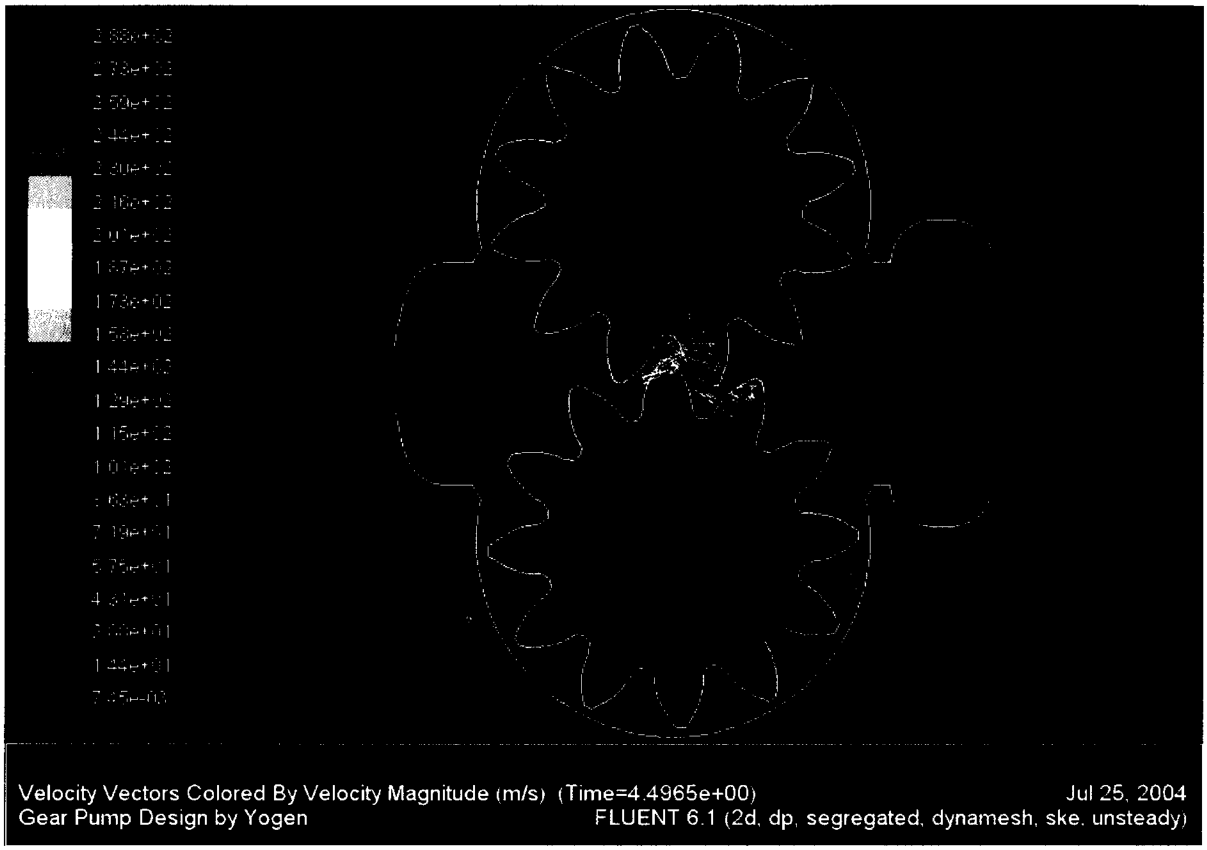


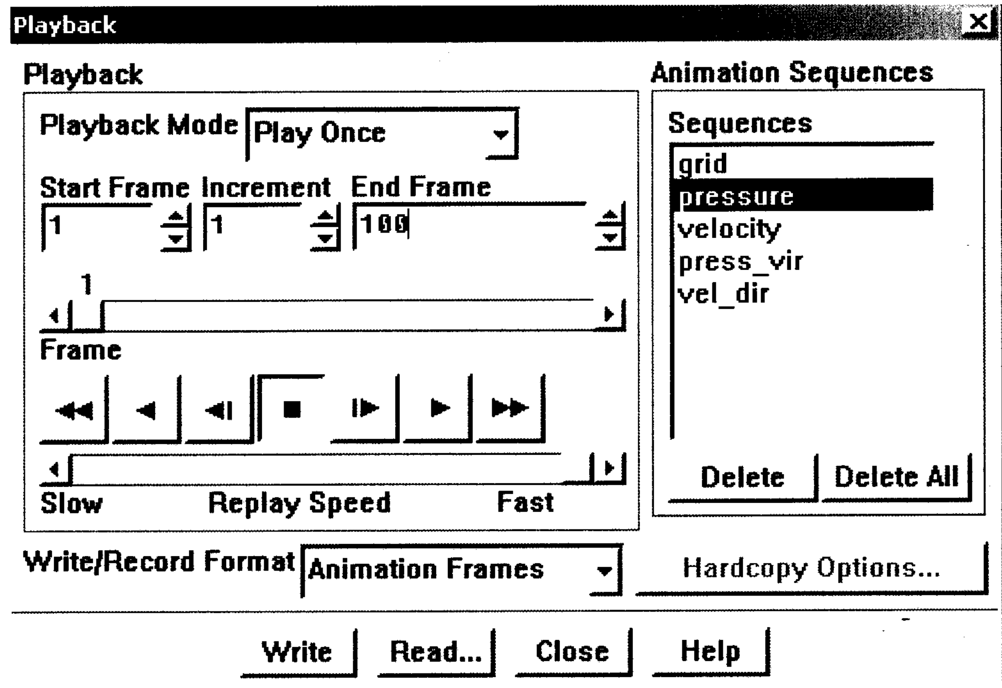
Fig 5.11 Exact Graphic of Velocity Vectors shown at t = 4.4965 s

(a) In the **Sequences** list, select **pressure**.

The playback control buttons will become active.

(b) Set the slider bar above **Replay Speed** about halfway between **Slow** and **Fast**.

(c) Keep the default settings in the rest of the panel and click the play button (the second from the right in the group of buttons under **Playback**).



4. Play back the animation of the velocity vectors.

Solve →Animate →Playback...

(a) In the **Sequences** list, select **vel_dir**.

(b) Keep the default settings in the rest of the panel and click the play button.

Results of graphs and plots are included in chapter 6.

CHAPTER 6

RESULTS AND INTERPRETATIONS

6.1 Results for Various Rotational Speed, N

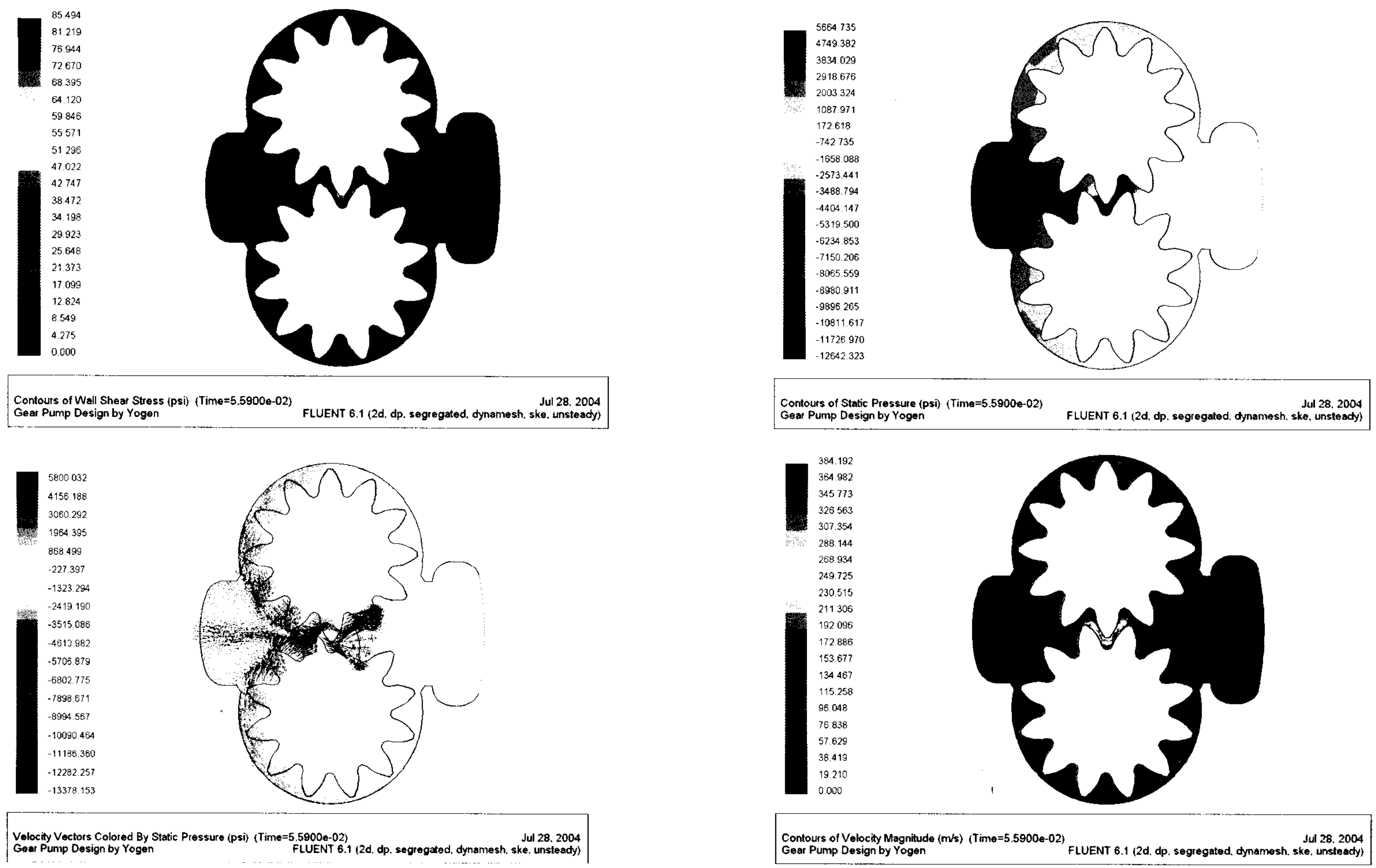


Figure 6.1 Flow Contours for rotational speed of gears = 3000 rpm; P_{out} = 3500 psi, P_{in} = 30 psi

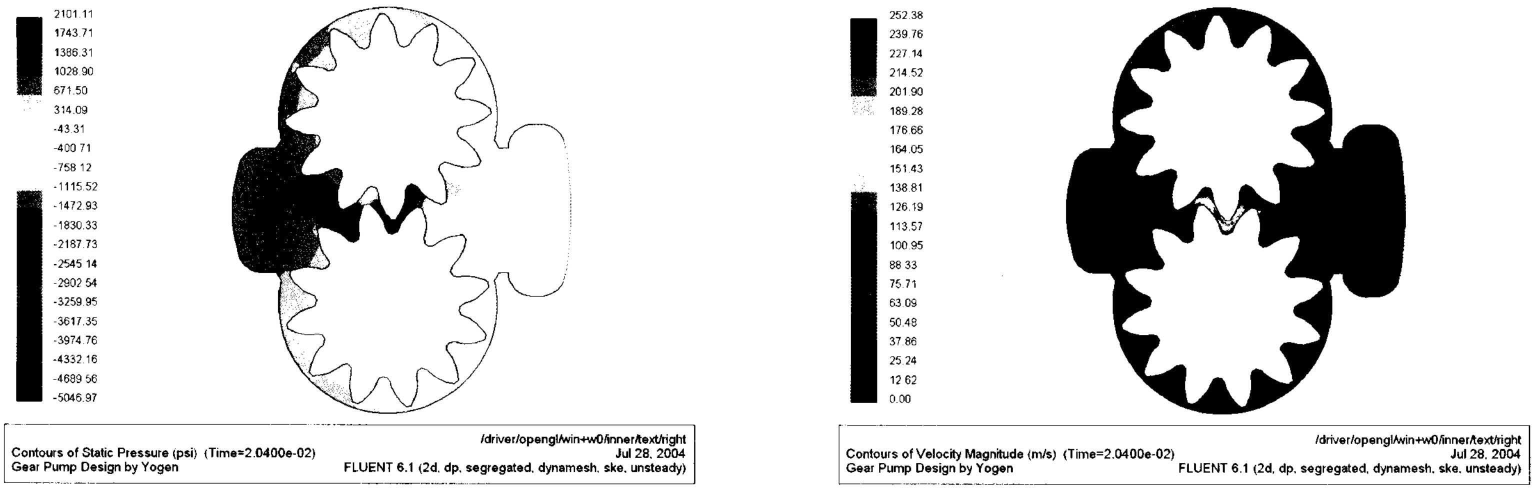


Figure 6.2 Flow Contours for rotational speed of gears = 2000 rpm; P_{out} = 1000 psi, P_{in} = 30 psi

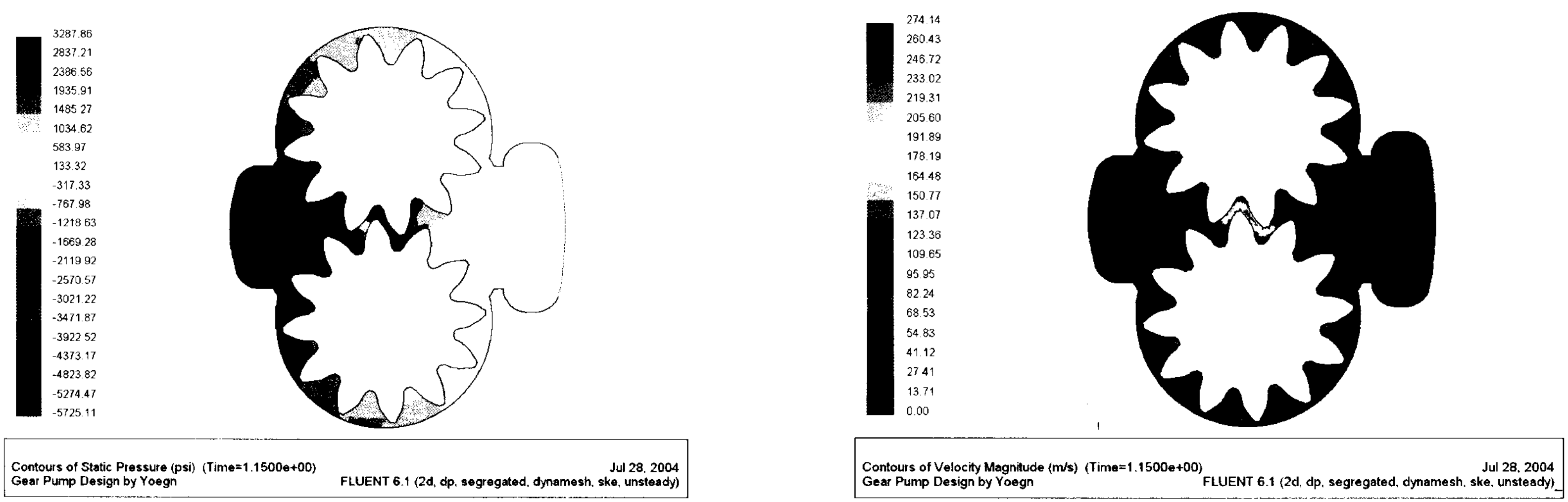


Figure 6.3 Flow Contours for rotational speed of gears = 2000 rpm; P_{out} = 2500 psi, P_{in} = 30 psi

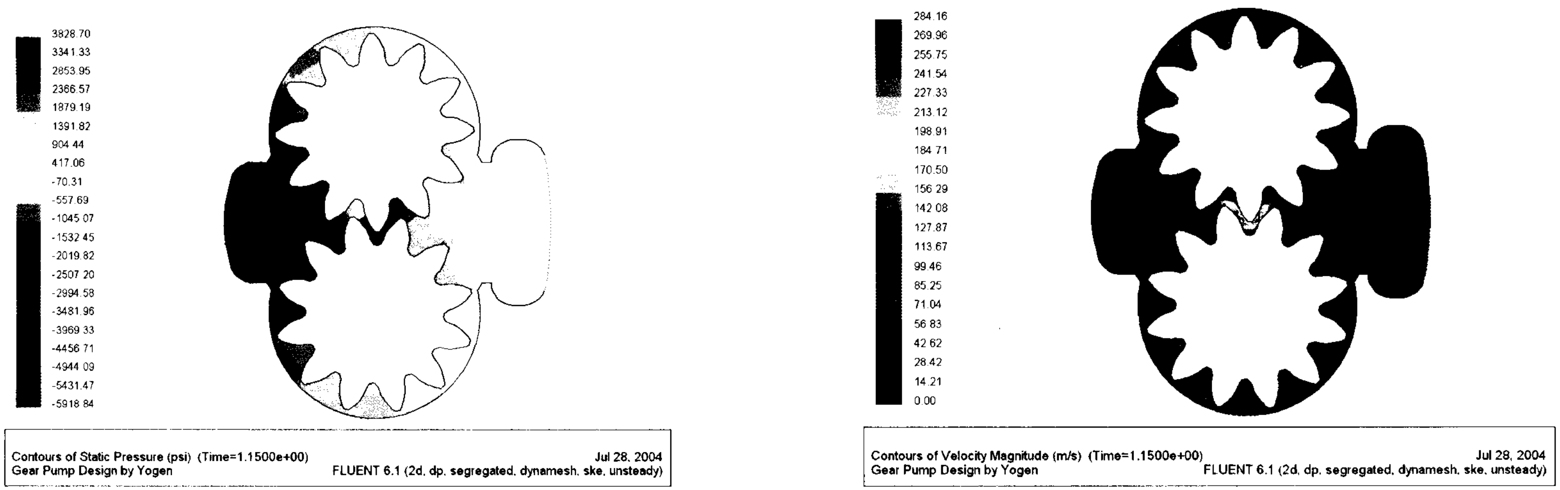


Figure 6.4 Flow Contours for rotational speed of gears = 2000 rpm; P_{out} = 3000 psi, P_{in} = 30 psi

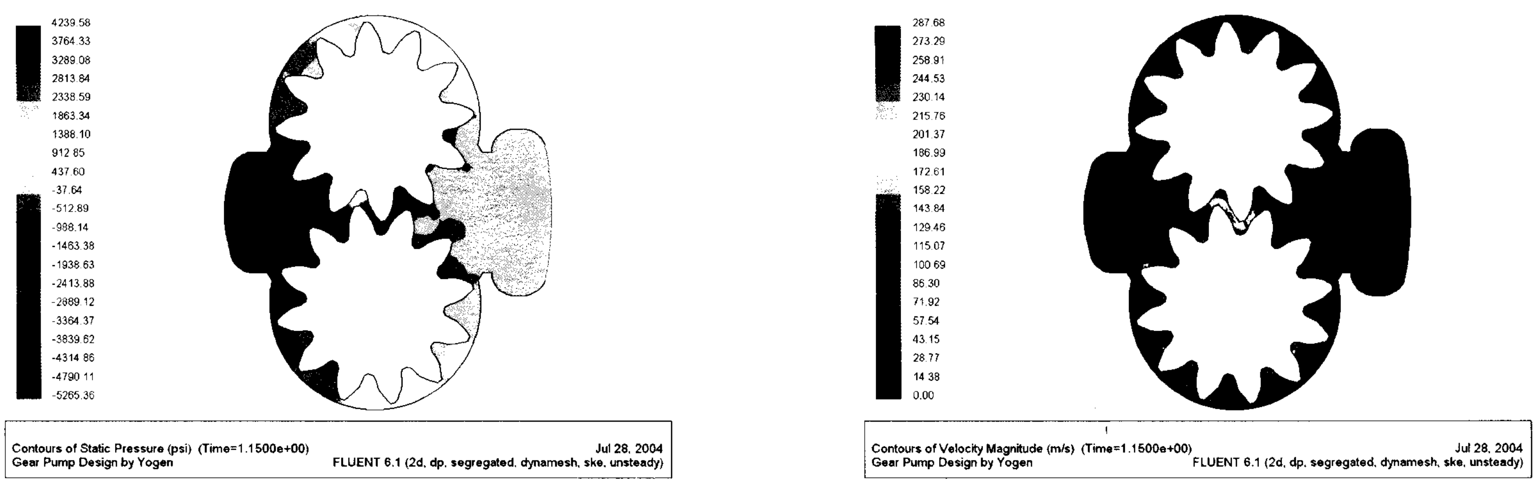


Figure 6.5 Flow Contours for rotational speed of gears = 2000 rpm; P_{out} = 3500 psi, P_{in} = 30 psi

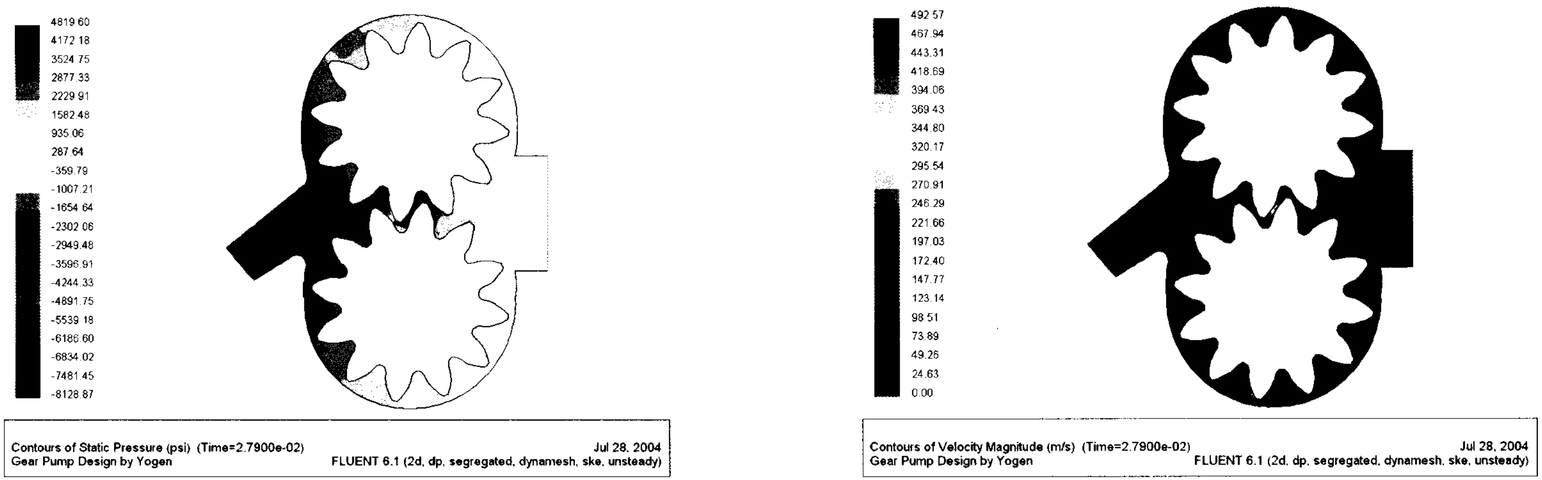


Figure 6.6 Flow Contours for rotational speed of gears = 2000 rpm; P_{out} = 3500 psi, P_{in} = 30 psi (Modified casing)

6.2 Comparative Graphics for Various Flow Parameters

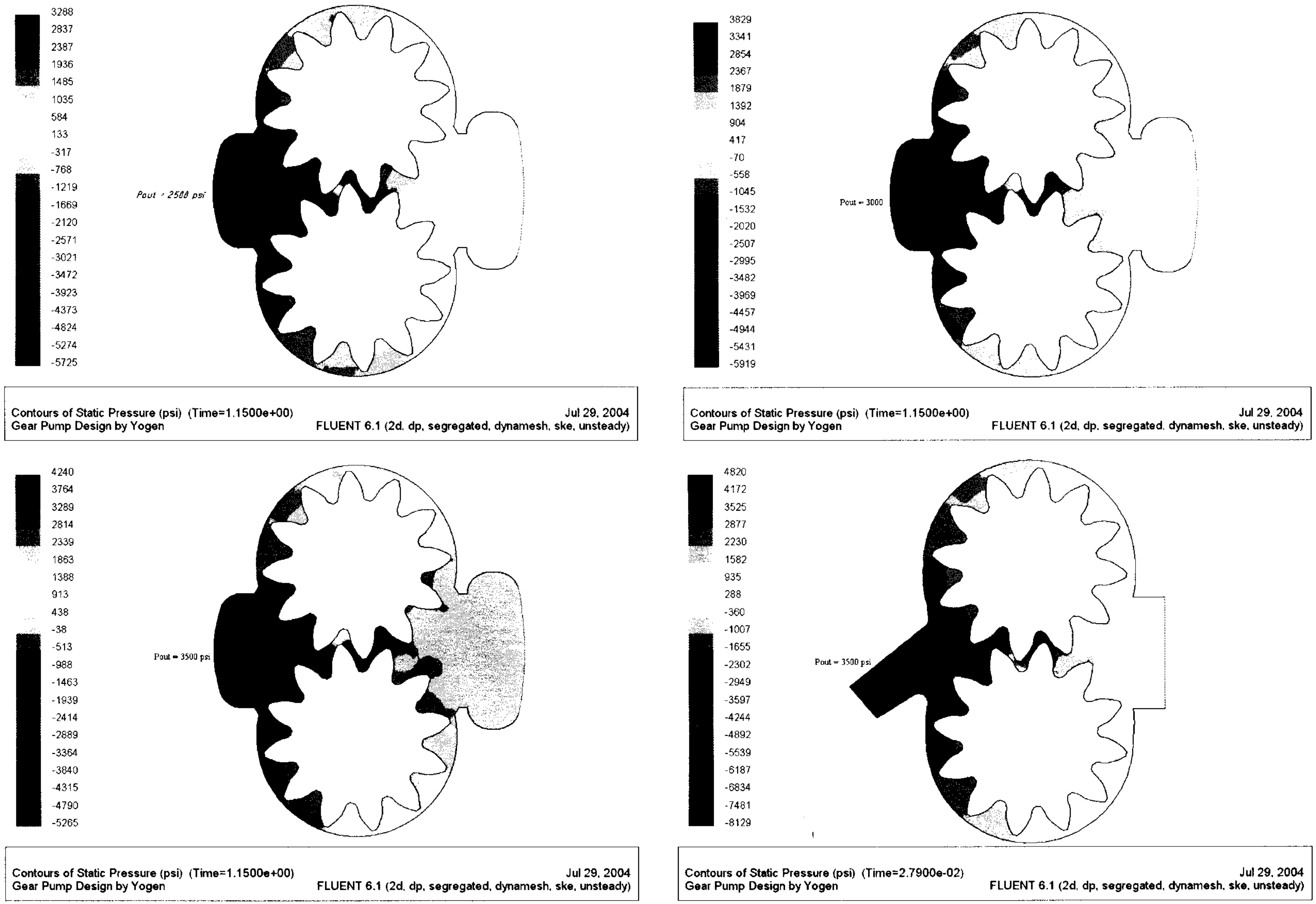


Figure 6.7 Static Pressure Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

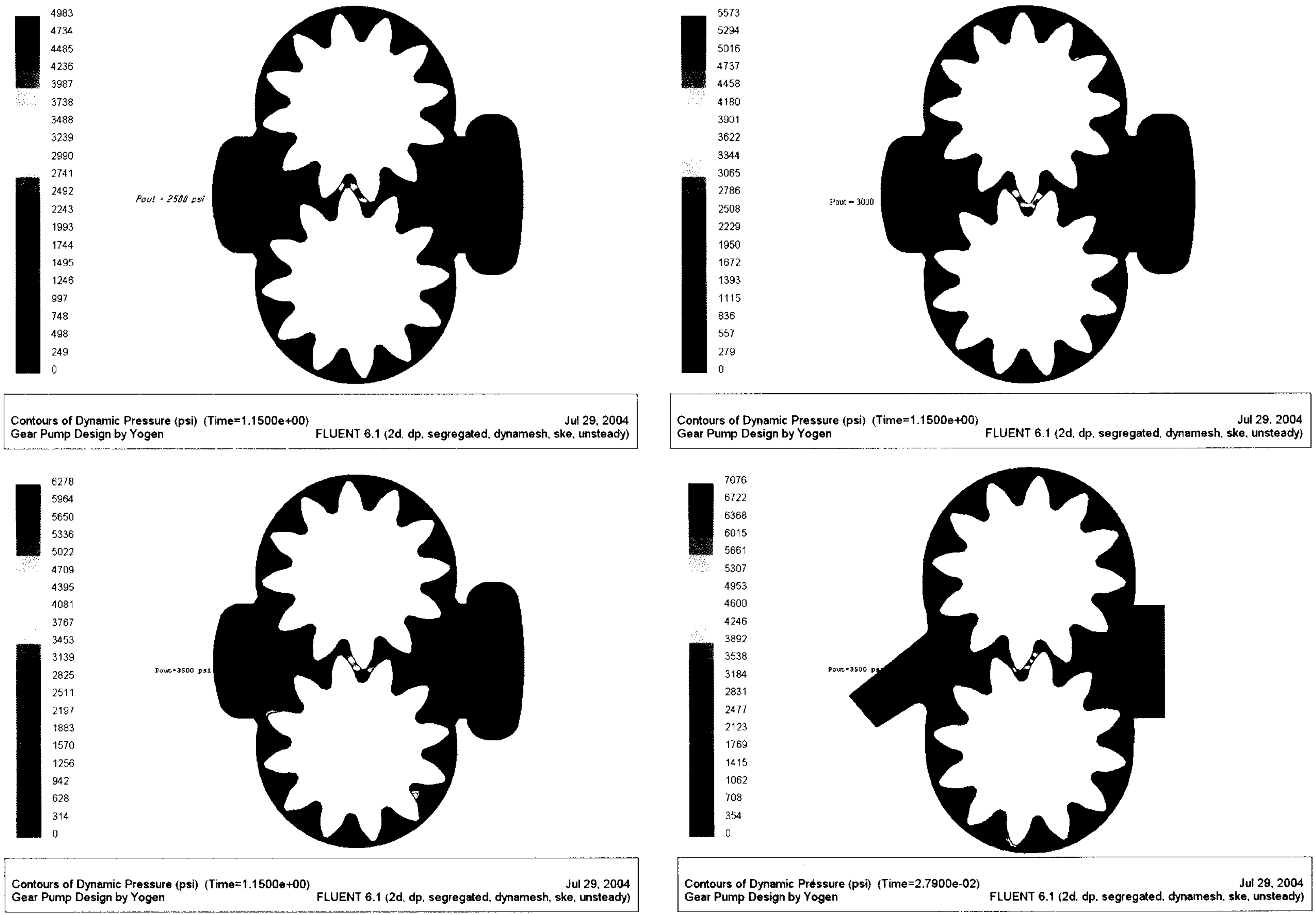


Figure 6.8 Dynamic Pressure Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

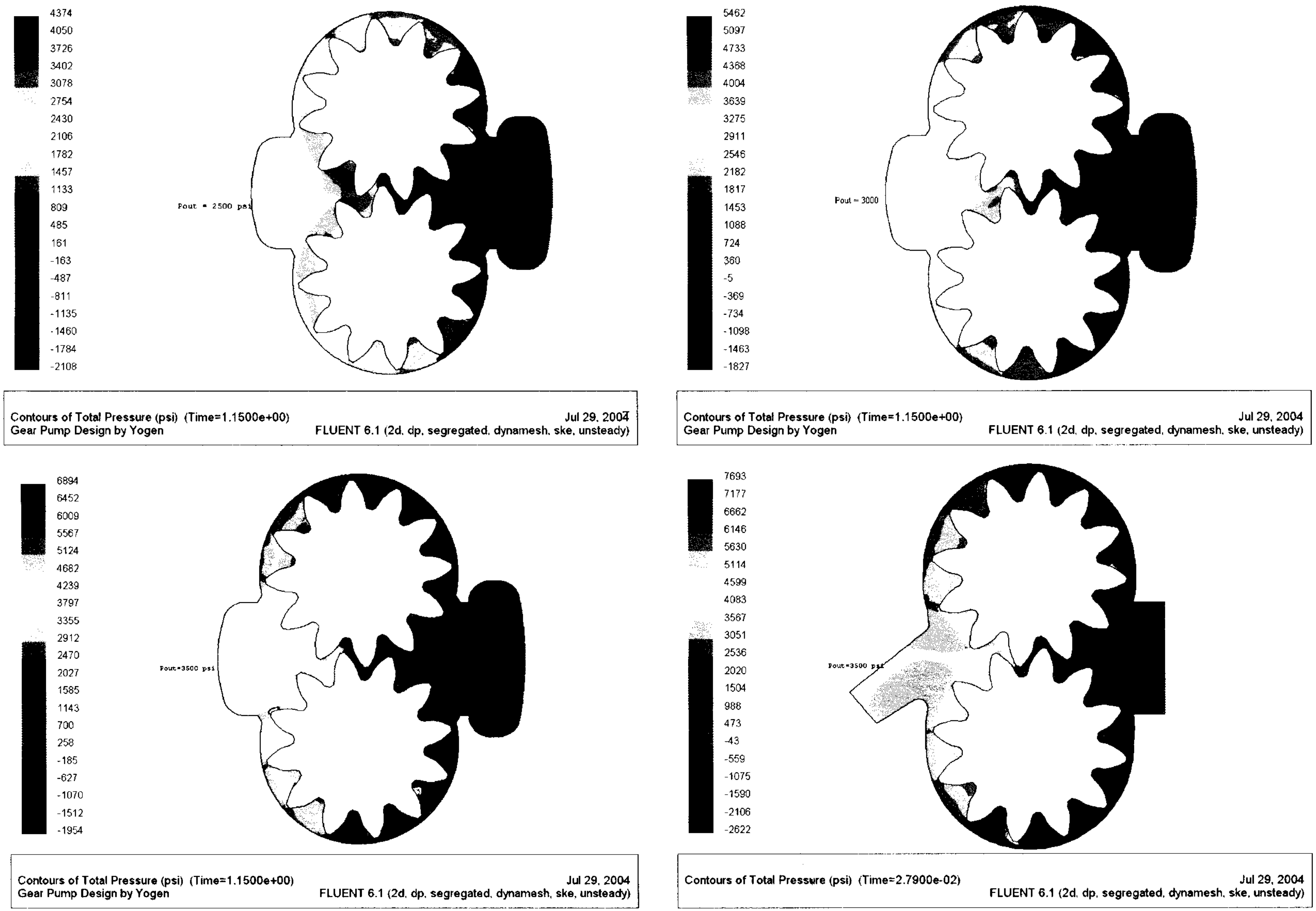


Figure 6.9 Total Pressure Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

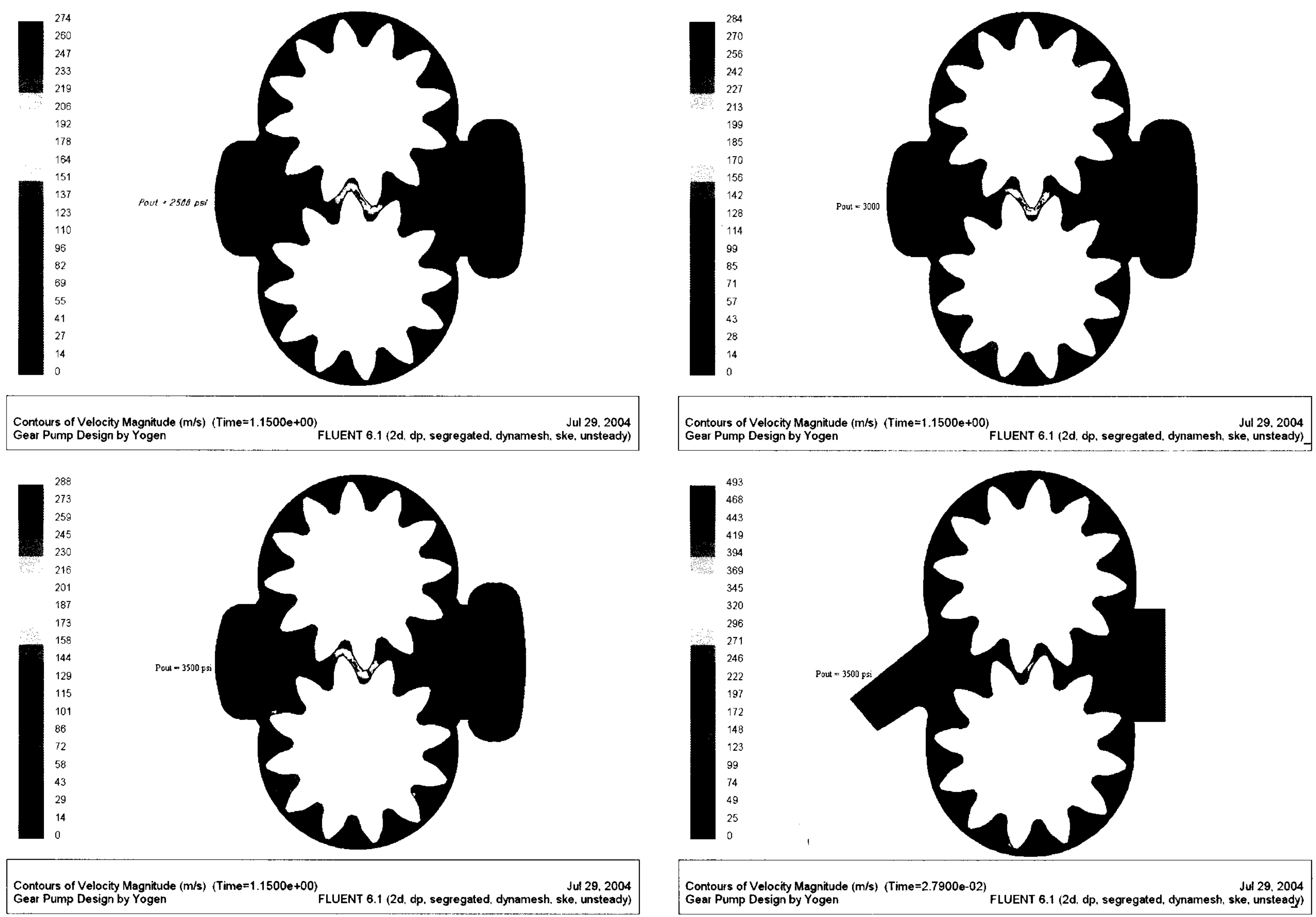


Figure 6.10 Velocity Magnitude Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

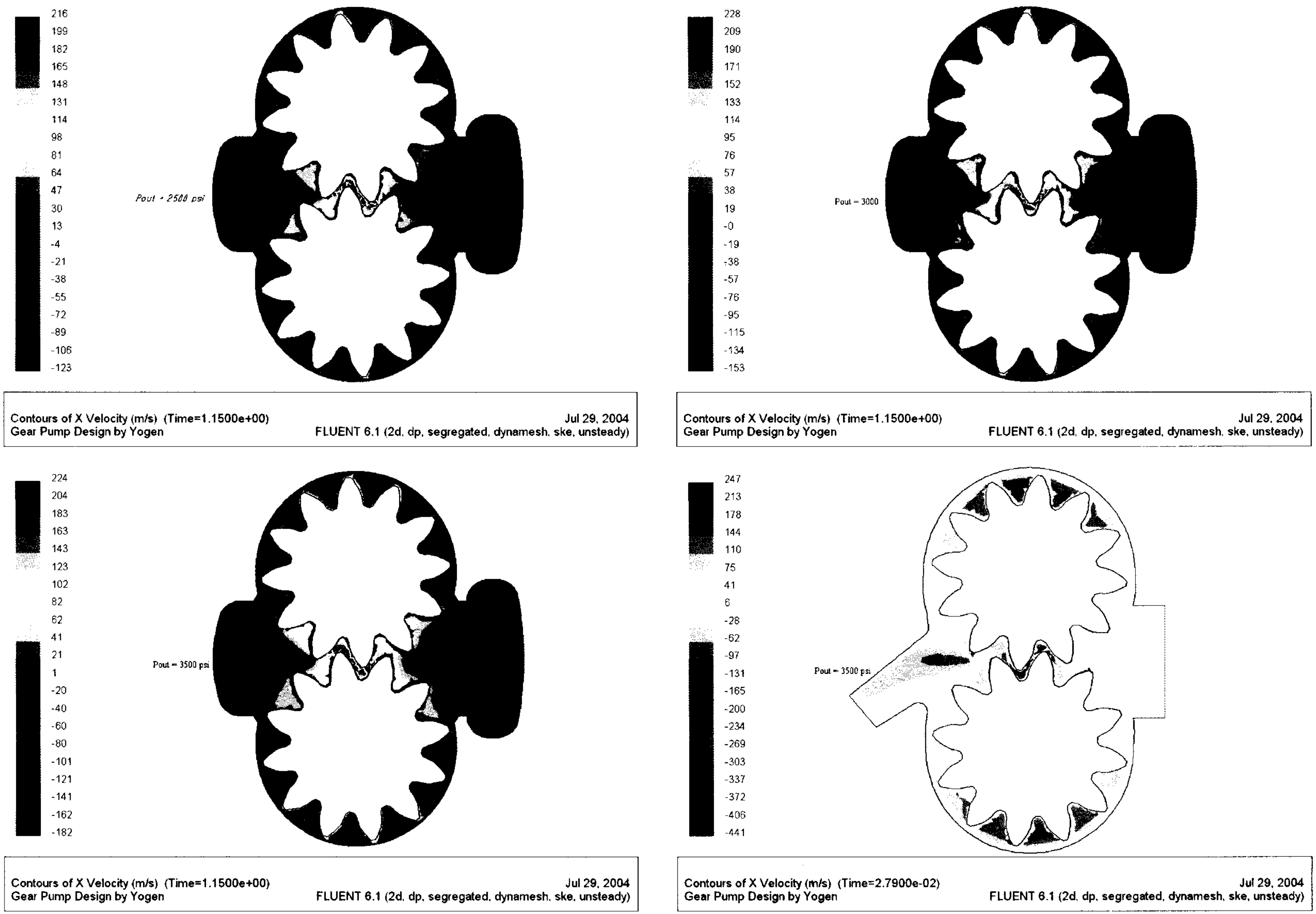


Figure 6.11 X- Velocity Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

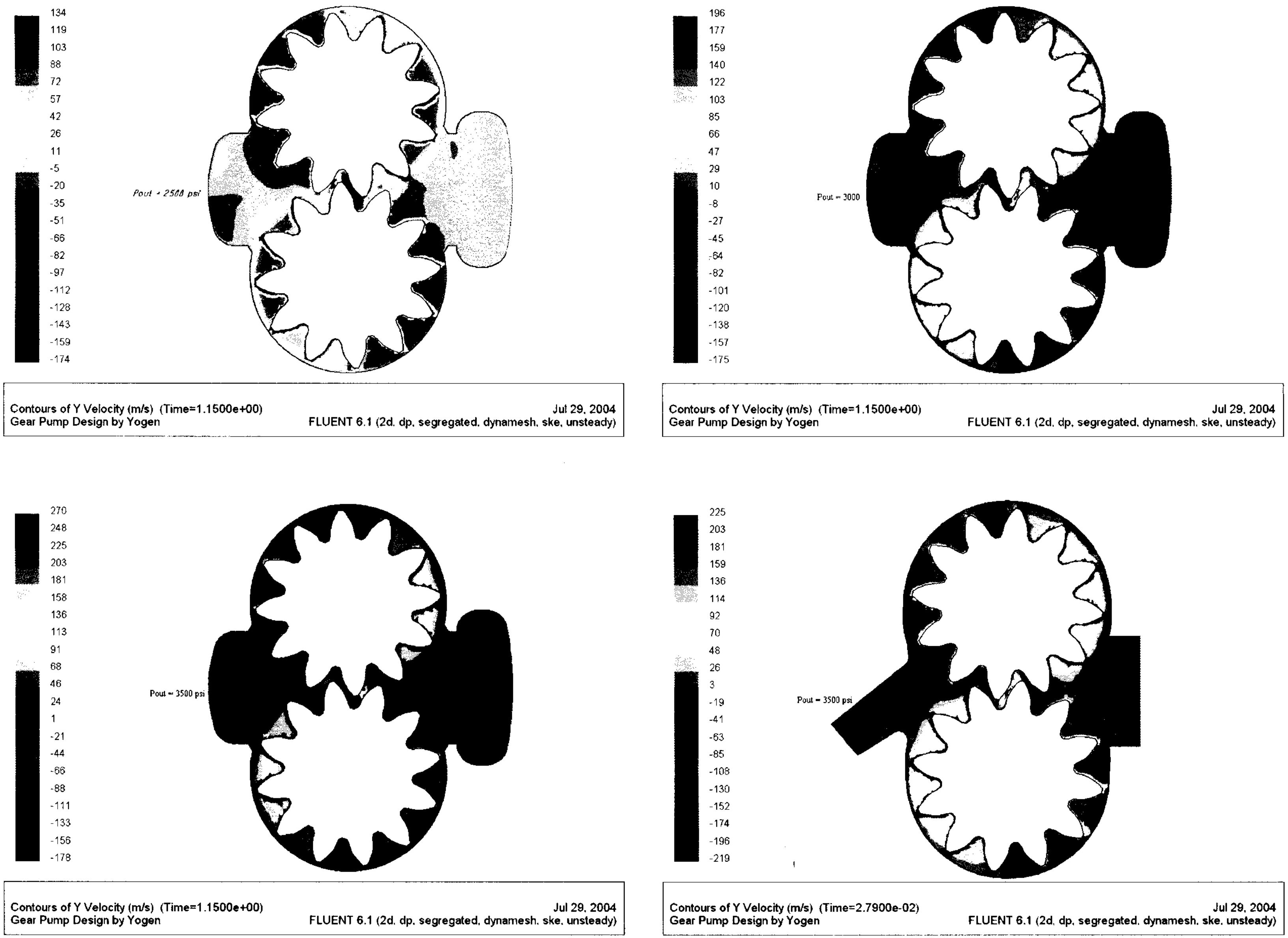


Figure 6.12 Y- Velocity Contours* for gears rotating at 2000 rpm (at Time t = 1.15 s)

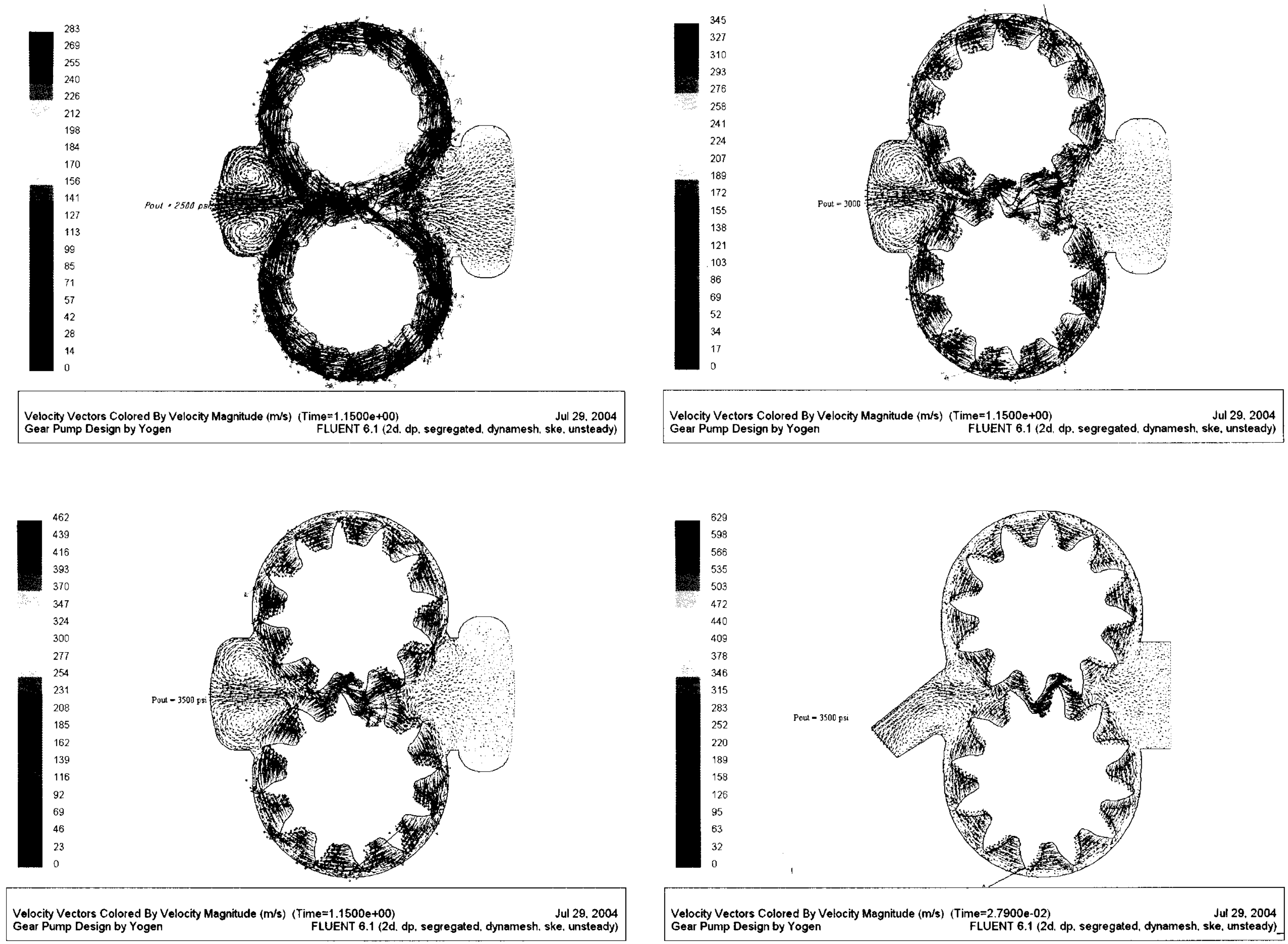


Figure 6.13 Velocity Vectors* colored by Velocity Magnitude for gears rotating at 2000 rpm (at Time t = 1.15 s)

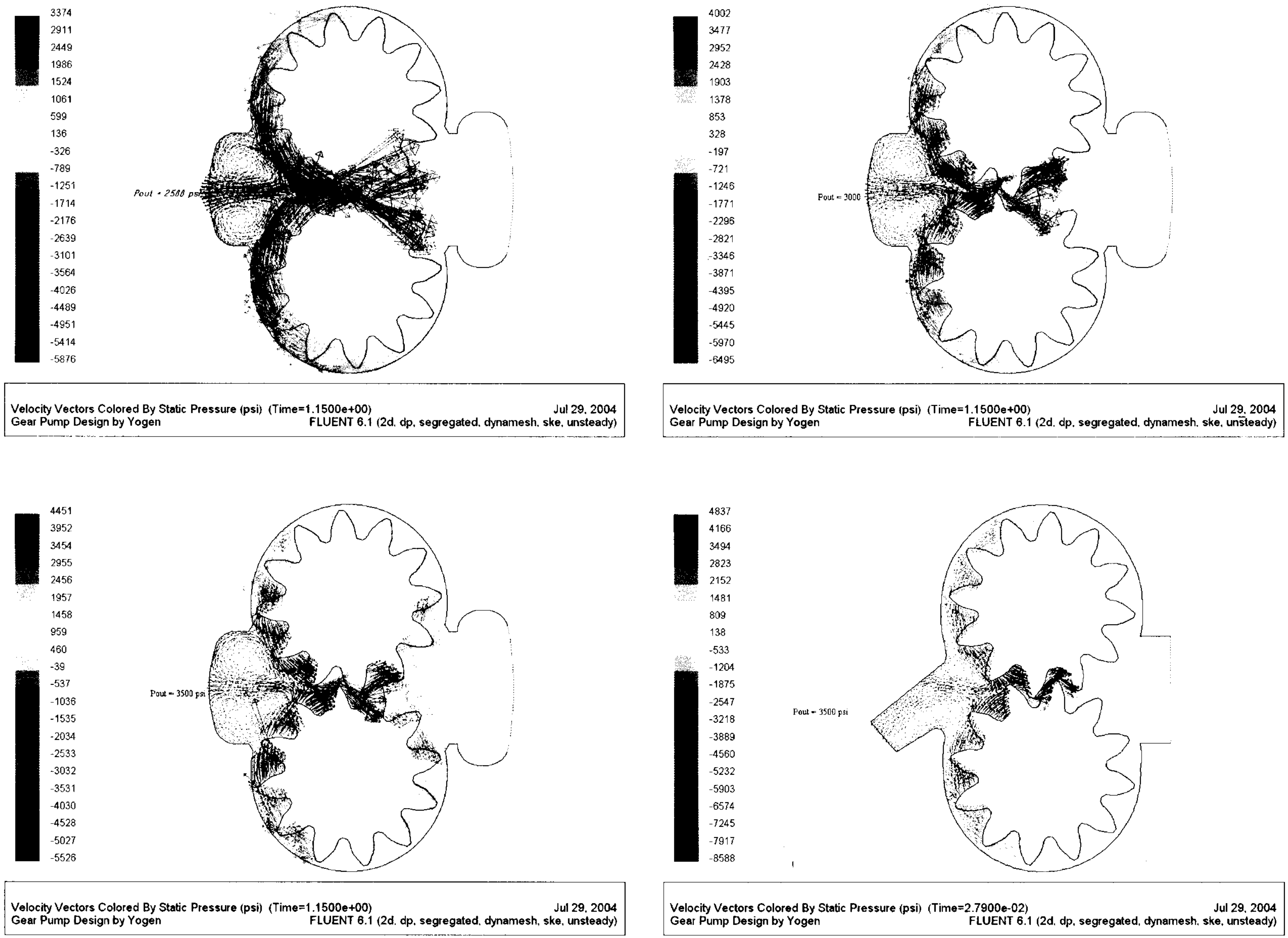


Figure 6.14 Velocity Vectors* colored by Static Pressure for gears rotating at 2000 rpm (at Time t = 1.15 s)

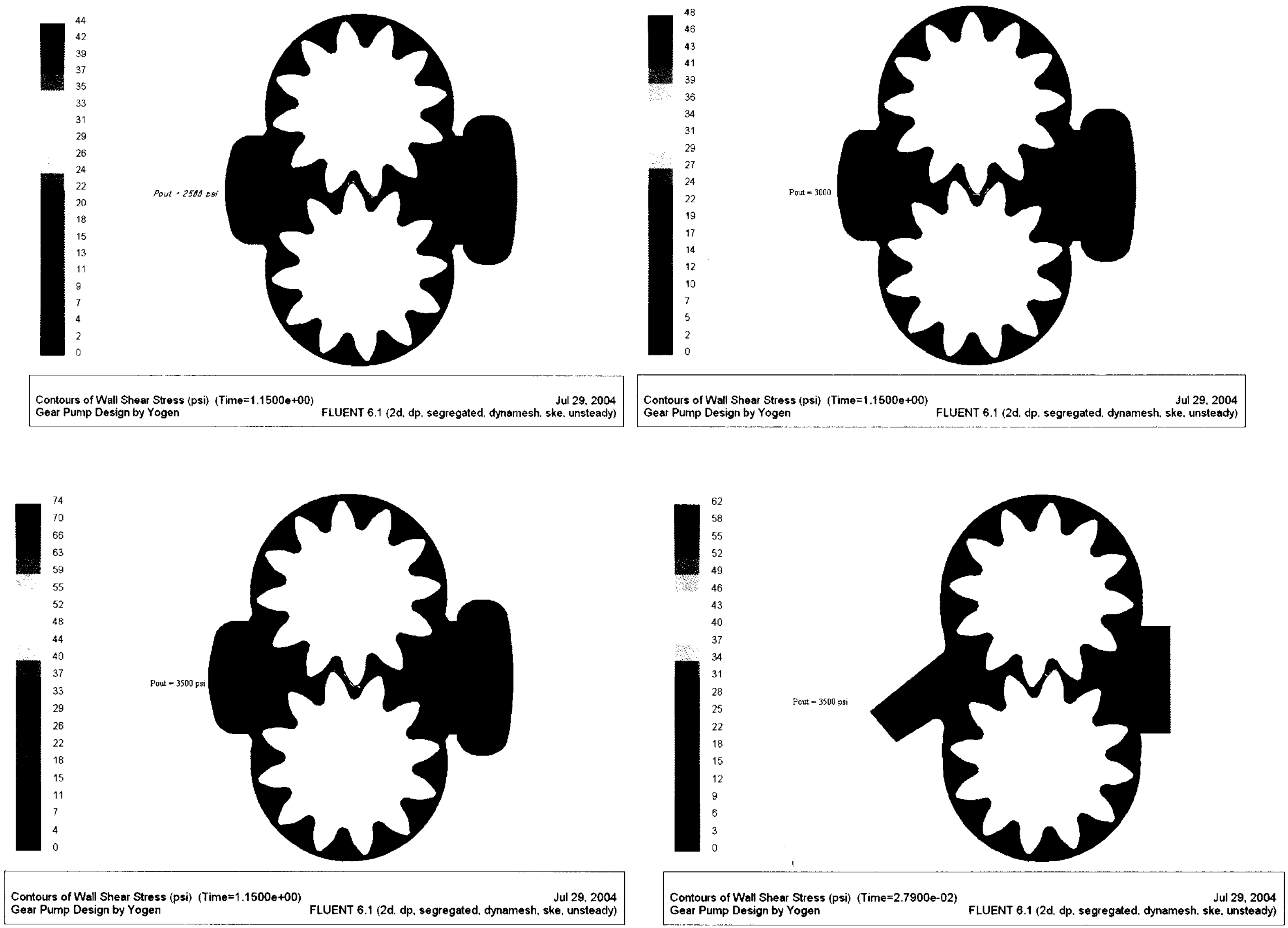


Figure 6.15 Contours of Wall Shear Stress* for gears rotating at 2000 rpm (at Time t = 1.15 s)

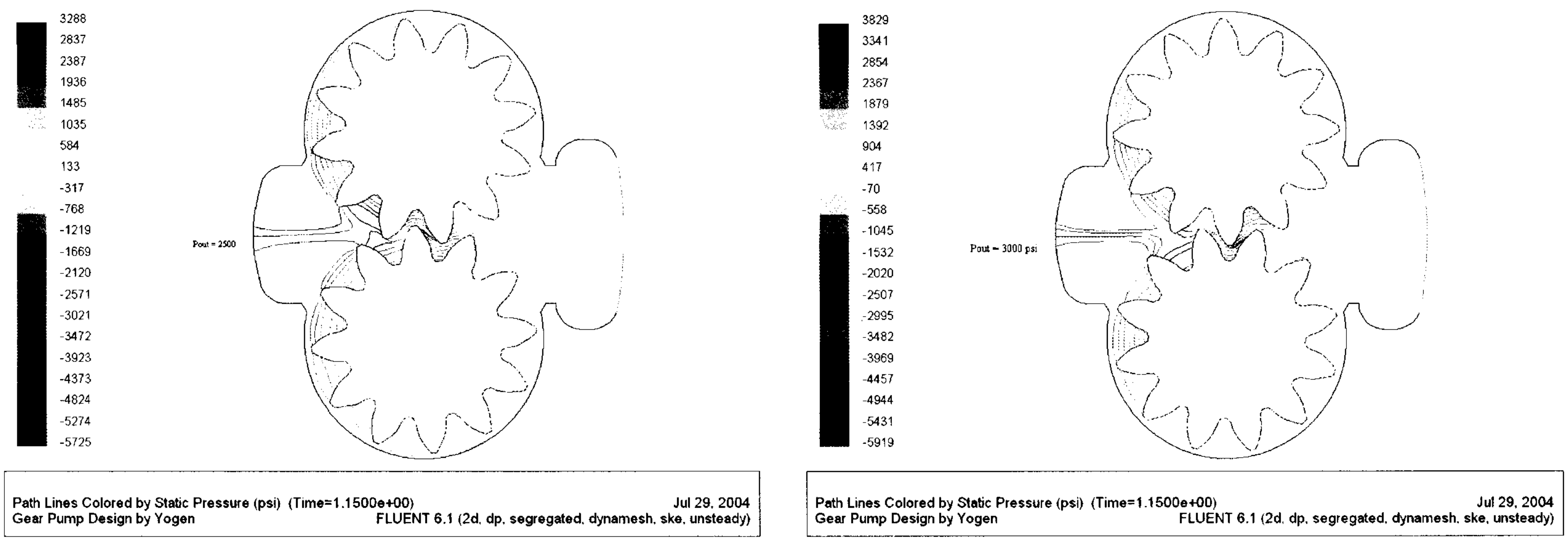


Figure 6.16 Path Lines* colored by Static Pressure for inlet, driven gear, driving gear, and outlet

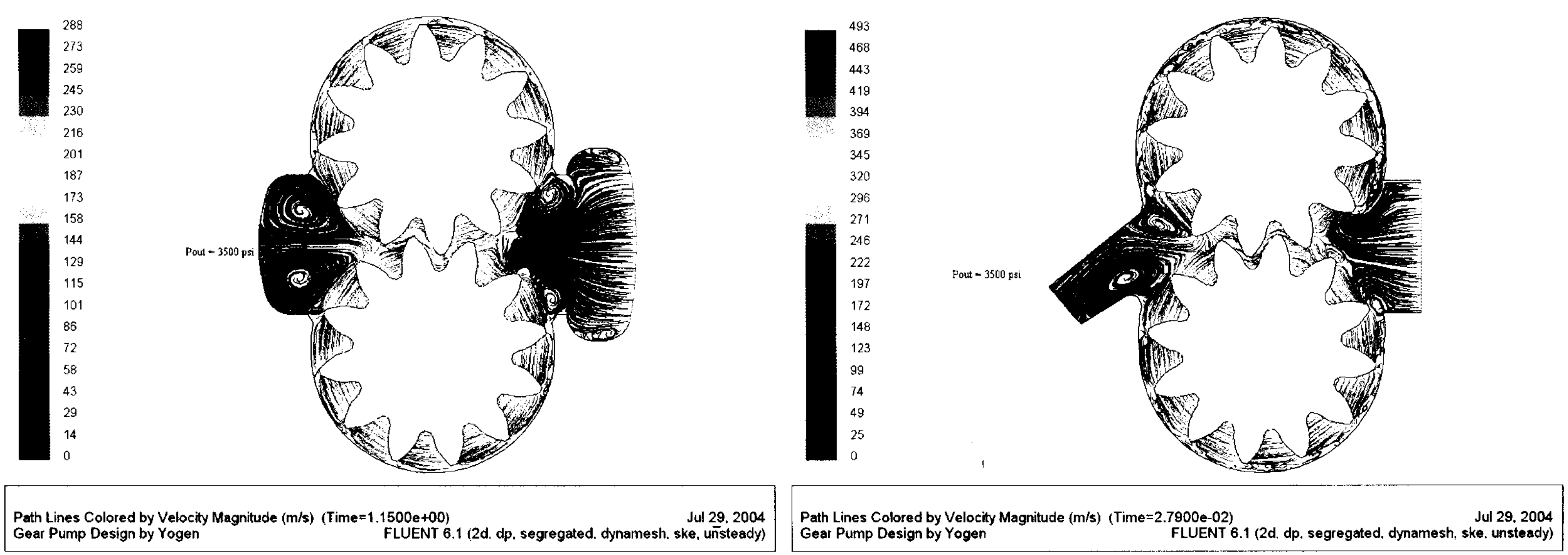


Figure 6.17 Path Lines* colored by Velocity Magnitude in the default interior from inlet to outlet

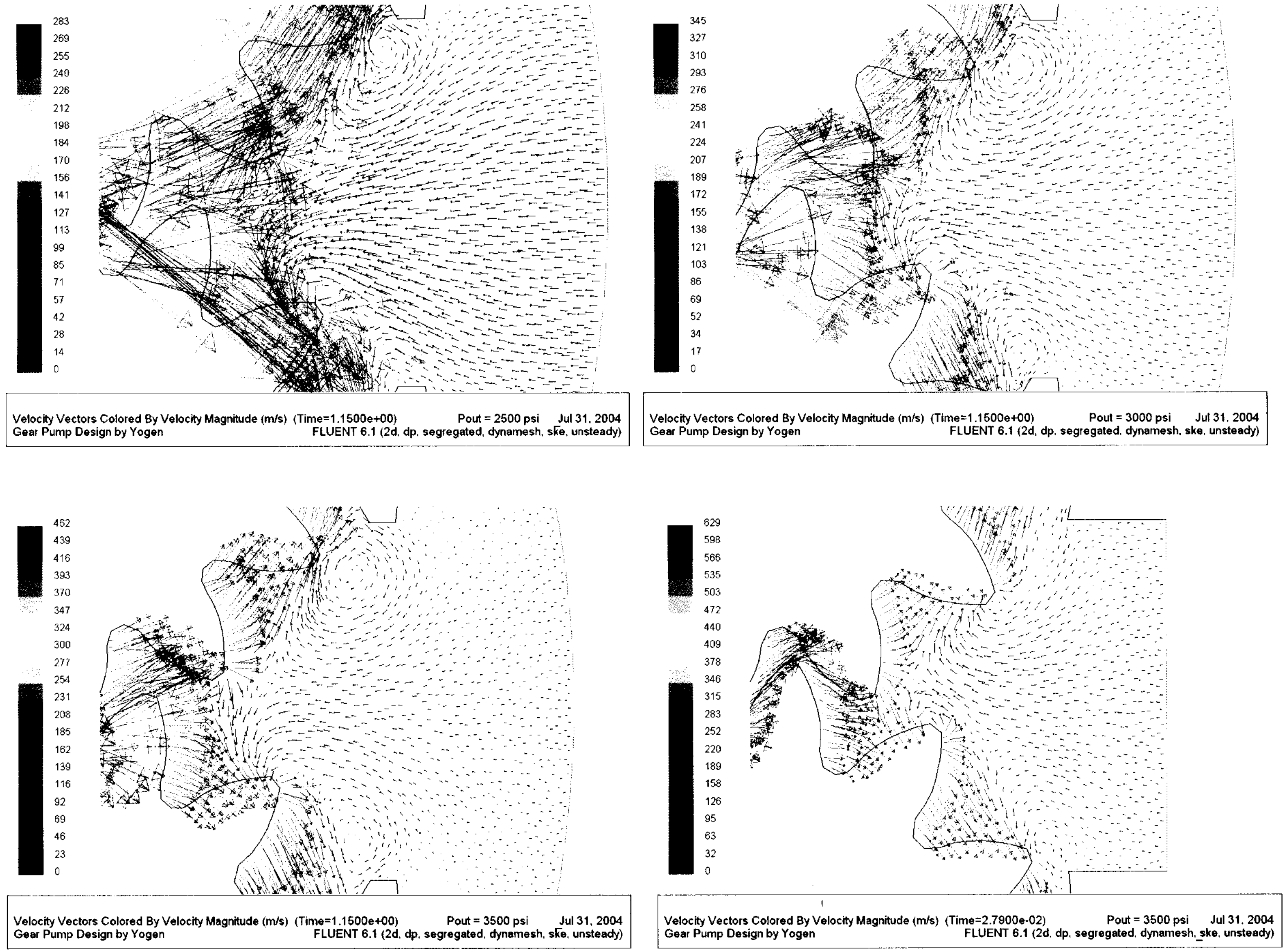


Figure 6.18 Velocity Vectors* at the inlet port:

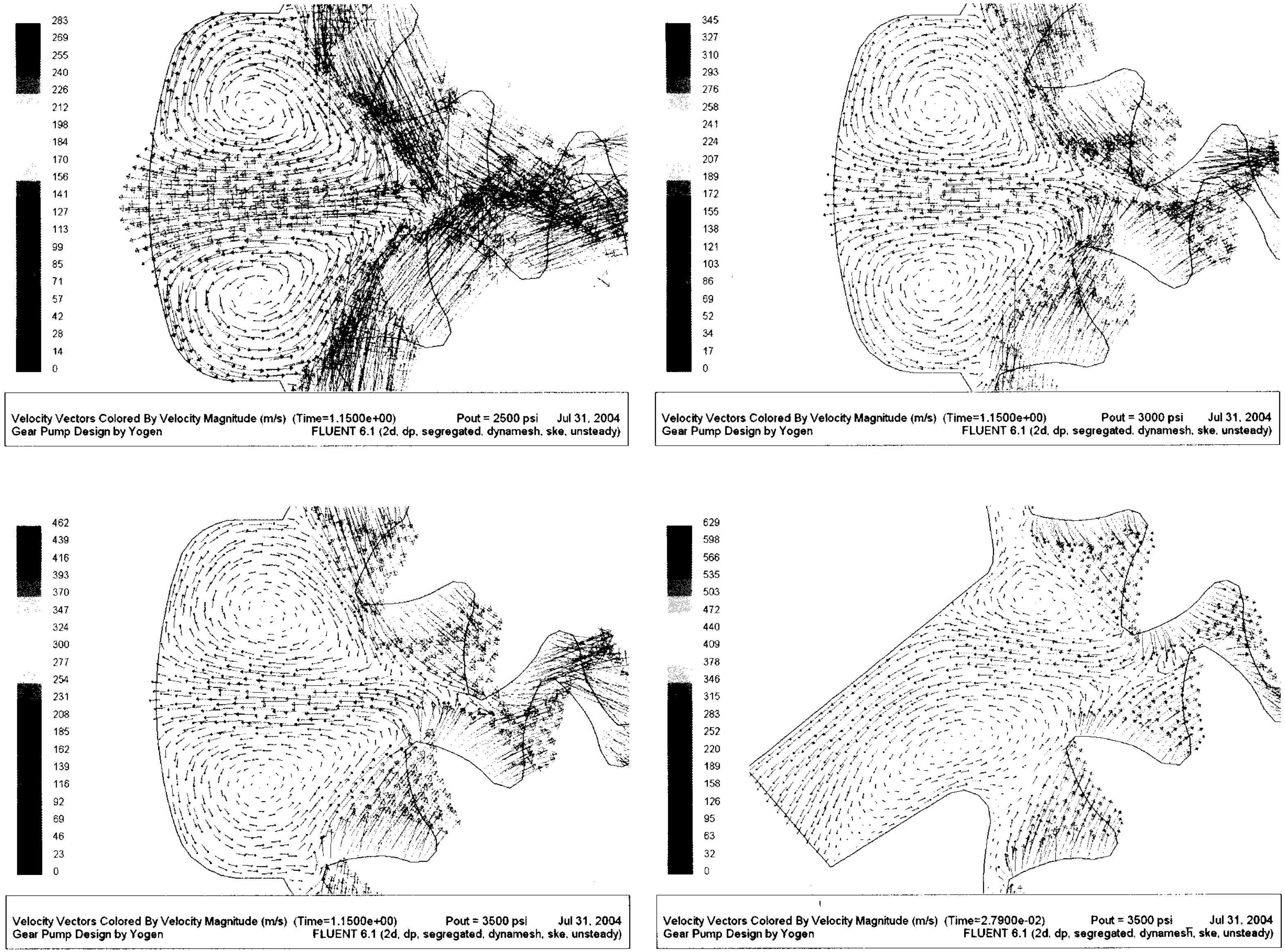


Figure 6.19 Velocity Vectors* at the outlet port:

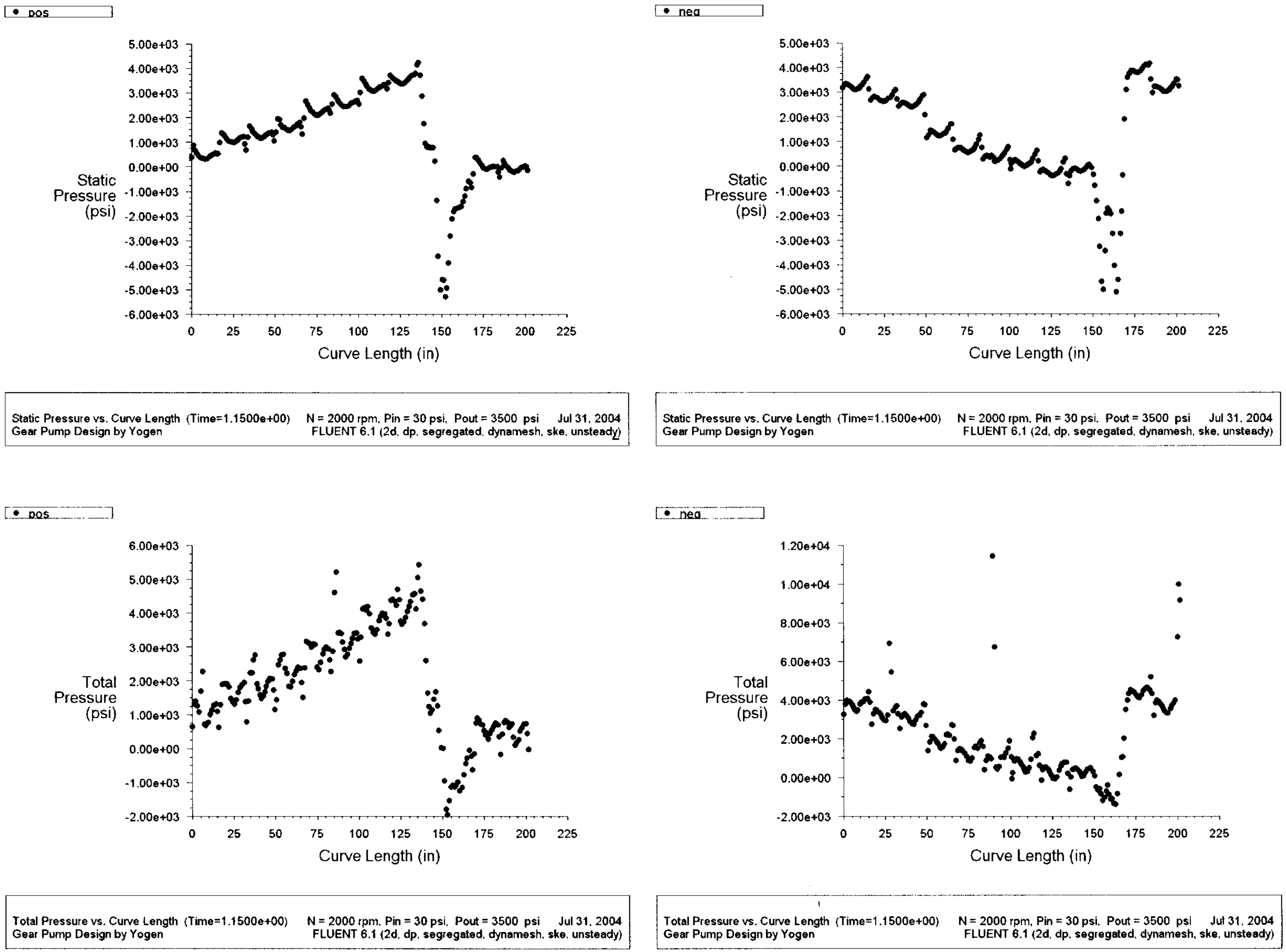


Figure 6.20 Pressure Vs. curved length of gears

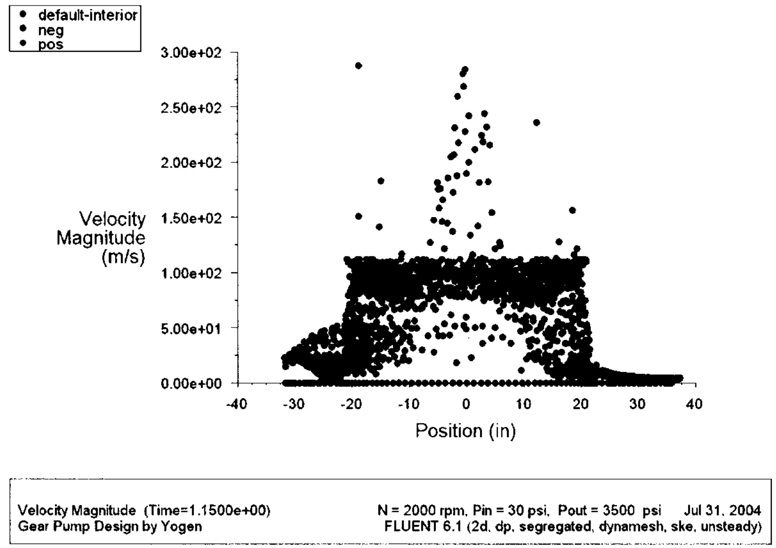


Figure 6.21 Velocity magnitude* vs. position of gears and default interior

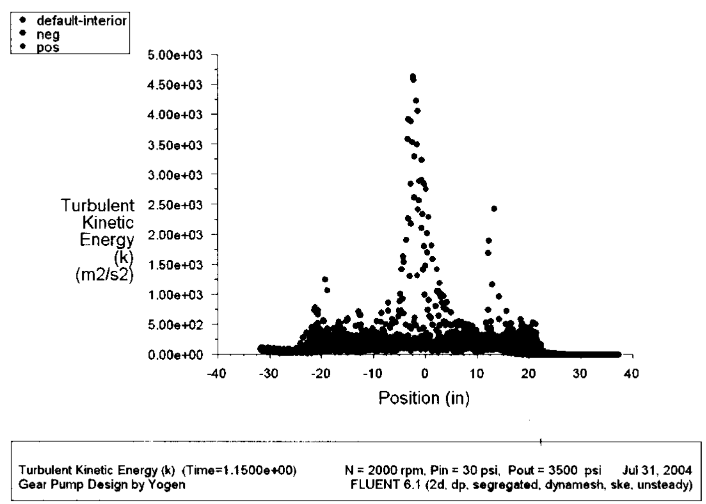


Figure 6.22 Turbulent Kinetic Energy* vs. position of gears and default interior

6.3 Comparison: Results of Flow Parameters

Table 6.1 Pressure and Velocity

Rotational Speed of gears (N), rpm	Pressure Outlet P_{out}, psi	Velocity Range V, m/s Scaling down factor = 39.18 (Page 39)	Static Pressure Range P_{stat}, psi (from the magnified geometry model)		Dynamic Pressure Range P_{dyna}, psi Dynamic Scaling down factor= 1535.36
2000*	**3500**	0 – 12.58	-8129	4820	0 – 4.61
3000	**3500**	0 – 9.80	-12642	5665	0 – 6.19
2000	**3500**	0 – 7.35	-5265	4240	0 – 4.09
	3000	0 – 7.25	-5919	3829	0 – 3.63
	2500	0 – 6.99	-5725	3288	0 – 2.59
	1000	0 – 6.43	-5046	2101	0 – 2.11

Table 6.2 Mass Flow Rate

N, rpm	Pressure Outlet P_{out}, psi	Mass flow rate kg/s		Mass flow rate gpm		Net mass-flow, kg/s
		Inlet	Outlet	Inlet	Outlet	
2000*	**3500**	3.7	- 3.7	68	-68	0
2000	**3500**	1.5	- 1.5	27	-27	0
	3000	1.9	- 1.9	35	-35	0
	2500	2.4	- 2.4	44	-44	0
	1000	3.8	- 3.8	70	-70	0

Negative and positive sign in mass flow rate indicates the mass of fluid coming in and going out from the gear pump.

*Modified Casing Design at t = 2.279e-2 s.

6.4 Interpretation of Results

The figures exhibiting velocity distributions show a slight change in velocity magnitude as the outlet pressure increases for the same rotational speed. However, when the rotational speed is changed from 2000 rpm to 3000 rpm, the maximum velocity increases significantly from a 7.35 m/s to 9.84 m/s (refer to Table 6.1) at the identical pressure boundary conditions. As noted on page 39, the velocity and the dynamic pressure distributions shown in the figures must be reduced by the geometric scaling factor of 39.18 and the dynamic scaling factor of 1535.36, respectively. The velocity is about zero close to the stationary casing wall as the fluid becomes stationary. The velocity contours show the patterns that were expected around the gears. As clearly seen in the figures showing the velocity vectors, the fluid is pushed forward to the outlet port as the teeth of two gears close. The graphs also show almost symmetrical recirculation flows forming near the outlet port. When each gear pushes the fluid towards the outlet port by force, the fluid in the region near the highly curved portions of the outlet pocket tends to stay in place and become separated from the main outlet flow. Subsequently, the fluid flows in a nearly circular motion that can be seen more clearly in the figure showing velocity vectors. There can be small recirculation flows even in the inlet port due to the similar mechanism.

The X-Y plots for the velocity components show very reasonable results. The magnitudes of X-component of the velocity are larger near the top of the gear since the tangential component is in phase of X coordinate. A similar conclusion can be drawn on the Y-component of the velocity showing the large magnitude in the right side of the gear. The magnitude of velocity vectors changes with respect to time domain. As the differential pressure (difference between inlet and outlet pressure) decreases, recirculation flows become more distinct. On the contrary, if the differential pressure increases, the flow rate will decrease due to added resistances to the outgoing flow and the reverse flow through the gap existing between two gears. Typically, fluids seek the path of least resistance; consequently, the higher the differential pressure, the more fluid that will be forced back through the clearances, resulting in decreased flow rate.

The wall shear stress is due to the velocity gradient of flow; as expected due to the same reason already mentioned, the highest wall shear stress was found in the high velocity region where teeth of two gears meet.

As seen in the figures, the pressure distributions are significantly different in magnitude, but show similar patterns as the outlet pressure changes for the cases with constant rotational speed and inlet pressure. The static and dynamic pressures increase as the outlet pressure increases. Static pressure contours inside the gear pump have four regions roughly, the first one is in the inlet port side, the second includes almost two identical contours around the top and the bottom of two gears close to the casing, the third is in between the gear meshes where the cavitation occurs and the fourth is in the high pressure side close to the outlet. The high values of negative static pressure in a very small region near the suction side where the gear teeth come out of the mesh shows cavitation effect in the gear pump. This phenomenon is created by the local pressure being lower than the vapor pressure, resulting in vaporization of the liquid to vapor bubbles. The subsequent collapse of the vapor bubbles forms a liquid jet flowing into collapsing bubble. This implosion results in highly negative values of pressure. M.S. Plesset and R.B. Chapman were able to observe and measure up to 30,000 psi of the bubble collapse pressure [39]. It should be noted that the cavitation may not happen in actual flows in gear pumps. The cavitation effects shown in this analysis are probably due to the high velocities of the fluid passing through the narrow gap between the two gears. The dynamic pressure is due to the velocity of fluid. Almost the concentric contour of pressure around the gear teeth is the main feature of dynamic pressure, which confirms the uniform velocity profiles. Total pressure contours show the combined effect of the static pressure and the dynamic pressure. In all of the cases, the high pressure region starts from the outlet port. The pressure is the maximum at the closing of gear teeth, and decreases toward the inlet port.

CHAPTER 7

CONCLUDING REMARKS

A method of numerical analysis for gear pump flows was developed. A two dimensional, unsteady, turbulent flow model was selected for the analysis by FLUENT. FLUENT has the capability of solving problems that are needed a moving dynamic meshing scheme. This scheme is used to analyze the gear pump flow. Several cases of the gear pump flows with various boundary conditions and different rotational speed of gears were analyzed.

A detailed step by step process of numerical modeling and analysis are presented in order to facilitate modeling for future problems. The results of the analysis showed reasonably expected distribution of the velocity and pressure except the region of a narrow gap between the two gears. The gap was created larger in order to avoid collapsing the moving dynamic meshes that will cause the termination of computing. In future analysis, this gap must be closed to better represent actual models.

A number of graphical outputs for the velocity and pressure distributions versus time for each case study were presented. The maximum time allowed was 5 seconds. The convergence of the numerical integration was obtained fairly quickly. The results showed qualitatively anticipated outcome for a variety of flow characteristics. The results should be used carefully for the purpose of preliminary investigations only since the accuracy of the analysis is low due to the simplification of flow and the modification of the dynamic mesh model. The reverse flow existing between two gears and the subsequent cavitation shown in this analysis may not occur in the same location in an actual flow.

However, despite all shortcomings, this study provides reasonable outcomes and substantial information on modeling and executing FLUENT for solving gear pump flows. A further refining in modeling and analysis will surely produce better results that will be used for design and performance improvements.

BIBLIOGRAPHY

[1] Sullivan, James A., *Fluid Power, Theory and Applications*, Prentice Hall, 4th Edition, 1998.

[2] Wright, T., *Fluid Machining, Performance, Analysis and Design*, CRC Press, 1999.

[3] Kim, H.W., *Development of Fluid Program in Engineering & Technology*, Proceedings of the ASEE Annual Conference, Salt Lake City, Utah, 2004.

[4] Neff, Darby B., *High Pressure Pumps and Their Controls*, American Brake Shoe Company, Columbus, Ohio.

[5] Novak, Julia and Gates, Kimberly, *The World of Pumps*, Civil Engineering Department, Virginia Tech University, Blacksburg, VA, 2002.

[6] Borghi, Massimo, *Fluid Power Research Group, DIMEC, 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 TuTech, Vol. 3, No. 1, April 2002.

[7] Machiels, Luc, *Simulation and theory of randomly forced turbulence,* Thèse No. 1724, Ecole Polytechnique Fédérale de Lausanne (*EPFL scientific report),* 1997.

[8] Gavrilakis, Spyros, *Direct Numerical Simulation of Turbulence*, Laboratoi re d'Ingénierie Numérique (LIN) Projet No 1998.1, Ecole Polytechnique Fédérale de Lausanne (*EPFL scientific report),* 2000.

[9] Manring, Noah D& Kasaragadda, Suresh B, *The Theoretical Flow Ripple of an External Gear Pump*, Journal of Dynamic Systems, Measurement, and Control, Volume 125, Issue 3, pp. 396-404, September 2003.

[10] Myllykylä, Jaakko , *Semi-Empirical Model from the Suction Capability of an External Gear Pump*, thesis 271, Tampere University of Technology 1999.

[11] Nelik, Lev, *Operating conditions and comparisons between chemical duty pumps and specialized high pressure gear pumps*, World Pumps, Dec 2001.

[12] Marx, Y.P., *Numerical simulation of gear-pump flows*, Laboratoire d'ingénierie numérique (LIN) Projet No 1999.2, Ecole Polytechnique Fédérale de Lausanne (*EPFL scientific report),* 2000.

[13] FLUENT Solutions: *Gear pump Solution, Example X 219,* FLUENT inc., USA.

[14] Panta, Yogendra M., Adhikari, S., Panthee, P., Bhattarai, R.and Gyawali, P., *Detail Design of Pelton Turbine for 500 kW (with Pattern)*, Undergraduate Senior Project in Mechanical Engineering submitted to Mechanical Engineering Department, Pulchowk Campus, Tribhuvan University, 2000.

[15] Panta, Yogendra M., *Rural Energy Technology*, E-Vision, Mechanical Engineering Department, Pulchowk Campus, Tribhuvan University, 2nd Edition, 1999.

[16] Murakami, Kobayashi, Yoshizawa, Kato, S., Taniguchi N., Hamba F., Kato C., Oshima M., Ooka R., *Numerical Simulation of Turbulent Flow Research Commemorative Symposium,* Numerical Simulation of Turbulence (NST), Institute of Industrial Science, University of Tokyo & TSFD(Turbulence Simulation and Flow Design) Research Group in IIS, 2003.

[17] Roy, Trina M., *Physically-Based Fluid Modeling using Smoothed Particle Hydrodynamics*, Thesis to University of Illinois at Chicago, Chicago, Illinois, 1995.

[18] Lal, Jagdish, *Hydraullic Machines*, Metropolitan Book Co. Pvt. Ltd., 6th Ed., 1997.

[19] Ferziger, J. H., *Numerical Methods for Engineering Application*, pp.195, John Wiley &Sons, 1981.

[20] Press, W. H., Teukolsky S. A., Vetterling W. T. and Flannery, B. P., *Numerical Recipes in C*, Cambridge University Press, 1992.

[21] Thompson, J. F., Warsi, Z. U. A. and Mastin, C. W., *Numerical Grid Generation - Foundations and Applications*, Elsevier Science Publishing Co., Inc, 1985.

[22] Rogers, S. E., Kwak, D. and Kiris, C., *Steady and Unsteady Solutions of the Incompressible Navier-Stokes Equation*, AIAA J., Vol. 29, No.4, pp. 603-610, 1991.

[23] Barth, T. J., *Analysis of Implicit Local Linearization Techniques for Upwind and TVD Algorithms*, AIAA Paper 87-0595., 1987.

[24] Yang, R. J., Chang, J. L. C. and Kwak, D., *Navier-Stokes Flow Simulation of the Space Shuttle Main Engine Hot Gas Manifold*, J. of Spacecraft & Rockets, Vol. 29, No. 2, pp. 253-259., 1992.

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

[26] *Hydraulic Institute Standards for Centrifugal, Rotary Reciprocating Pumps*, Hydraulic Institute Publication, *Parsippany, NJ, 1994.*

[27] *Centrifugal and Rotatory Pumps: Fundamentals with Applications*, CRC Press, Boca Raton, FL, 1999.

[28] Nelik, L., *Extending the Life of Positive Displacement Pumps*, Pumps & Systems magazine, April, 1999.

[29] Hatzikonstantinou, P. M., Sakalis, V. D., *A numerical-variational procedure for laminar flow in curved square ducts*, International Journal for Numerical Methods in Fluids, Volume 45, Issue 12, 30 August 2004.

[30] Sibley School of Mechanical and Aerospace Engineering, Cornell University, *FLUENT Tutorials*, Ithaca, NY, 2002.

[31] Murthy, Jayathi, ME *608: Numerical Methods in Heat, Mass and Momentum Transfer*, Schools of Engineering, Mechanical Engineering, Purdue University, West Lafayette, Indiana, 2004.

[32] FLUENT Inc., *GAMBIT 2.1 & FLUENT 6.1 Documentation*, FLUENT Inc., 2001.

[33] Parker Hydraulics, *PGP/PGM Bushing Design 300/400 Series*, Parker Hannifin Corporation, Gear Pump Division, Youngstown, OH, 2001.

[34] Iaccarino, Gianluca, *Numerical Methods in Fluid Dynamics Using Commercial CFD Codes*, Center for Turbulence Research, Stanford University, 2004.

[35] Iaccarino, Gianluca, *An approach for local refinement of structured grids*, Journal of Computational Physics, 2001.

[36] Pipaloff, Alexander, *A Step Forward in Rotary Fluid Machine Technology*, Paper Presented at the International Fluid Power Exposition and Technical Conference, 24-26 March 1992.

[37] Balashov, M.M. and Chistov, V.I., *Flow of Thermoplastic Melts in a Gear Pump and Waste Processor*, Chemical and Petroleum Engineering,Vol.38, Nos. 3-4, 2002.

[38] Fluent Tutorial- *Modeling Flows in Moving and Deforming Zones*, FLUENT 6.1 Documentation, Page 9-44, January 28, 2003.

[39] Plesset, M.S. and Chapman, R.B., *Cavitation Bubble Collapse of an initially Spherical Vapor Cavity in the neighborhood of a Solid Boundary*, Fluid Mechanics, P283, 1971.

APPENDIX 1

User Defined Function (UDF) Syntax for "motion.c" for gears at N = 2000 rpm

```
/*This code is written in "C"*/
/*UDF starts for both the gears*/
#include "udf.h"
#include "dynamesh_tools.h"
/*UDF starts for Rotational Speed of 2000 rpm for Driving Gear*/
DEFINE_CG_MOTION(anticlock_wise, dt, vel, omega, time, dtime)
{
        NV_S (vel, =, 0.0);
        NV_S (omega, =, 0.0);
/*Linear Velocity for Driving Gear*/
                vel[0] = 0.0;
                vel[1] = 0.0;
                vel[2] = 0.0;
/*Angular Velocity for Driving Gear */
                omega[0] = 0;
                omega[1] = 0;
                omega[2] = 210;
/*Messages for Display*/
Message("\nThis is Anti Clockwise Gear\n");
Message("\nCG_Omega for anticlock_wise: %g, %g, %g\n",  omega[0], omega[1],
omega[2]);
Message("\nCG Position for anticlock_wise: %g, %g, %g\n", NV_LIST(DT_CG(dt)));
Message("\nCG Orientation for anticlock_wise: %g, %g, %g\n",
          NV_LIST(DT_THETA(dt)));
}
/*UDF ends for Rotational Speed of 2000 rpm for Driving Gear*/
```

Continued...

```
/*UDF starts for Rotational Speed of 2000 rpm for Driven Gear*/
/*Rotational Speed of 2000 rpm for Driven Gear*/
DEFINE_CG_MOTION(clock_wise, dt, vel, omega, time, dtime)
{
        NV_S (vel, =, 0.0);
        NV_S (omega, =, 0.0);
/*Linear Velocity for Driven Gear*/
                vel[0] = 0.0;
                vel[1] = 0.0;
                vel[2] = 0.0;
/*Angular Velocity for Driven Gear */
                omega[0] = 0;
                omega[1] = 0;
                omega[2] = -210;
/*Messages for Display*/
Message("\nThis is Clockwise Gear\n");
Message("\nCG_Omega for clock_wise: %g, %g, %g\n", omega[0], omega[1],
         omega[2]);
Message("\nCG Position for clock_wise: %g, %g, %g\n", NV_LIST(DT_CG(dt)));
Message("\nCG Orientation for clock_wise: %g, %g, %g\n", NV_LIST(DT_THETA(dt)));
 }
/*UDF ends for Rotational Speed of 2000 rpm for Driven Gear*/
/* UDF ends for both the gears*/
```

APPENDIX 2

Moving Dynamic Mesh: DEFINE_CG_MOTION

DEFINE_CG_MOTION macro is used to specify the motion of a particular dynamic domain in FLUENT to provide linear and angular velocities at every time step.

Macro: DEFINE_CG_MOTION (name, dt, vel, omega, time, dtime)

Argument types: Dynamic_Thread *dt

real vel[]

real omega[]

real time real

dtime

Function returns: void

APPENDIX 3

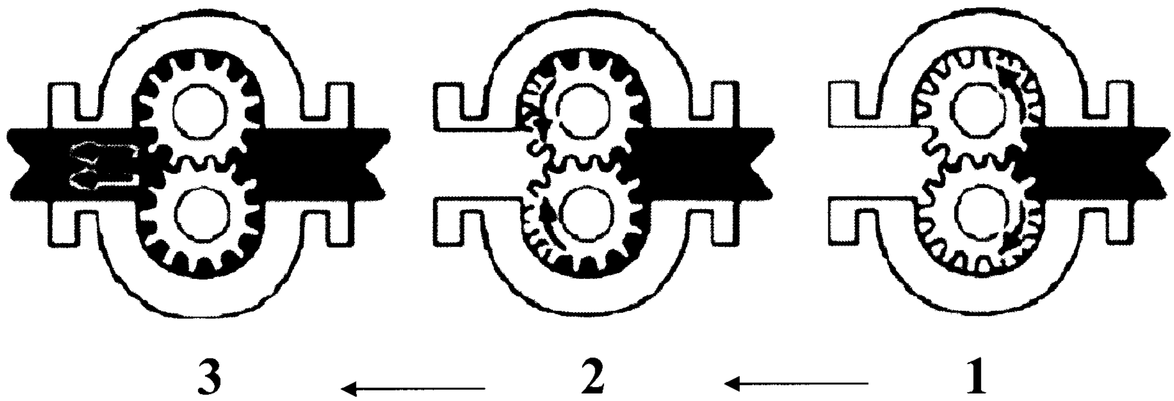


Figure A 3.1 Pumping action in an external gear pump

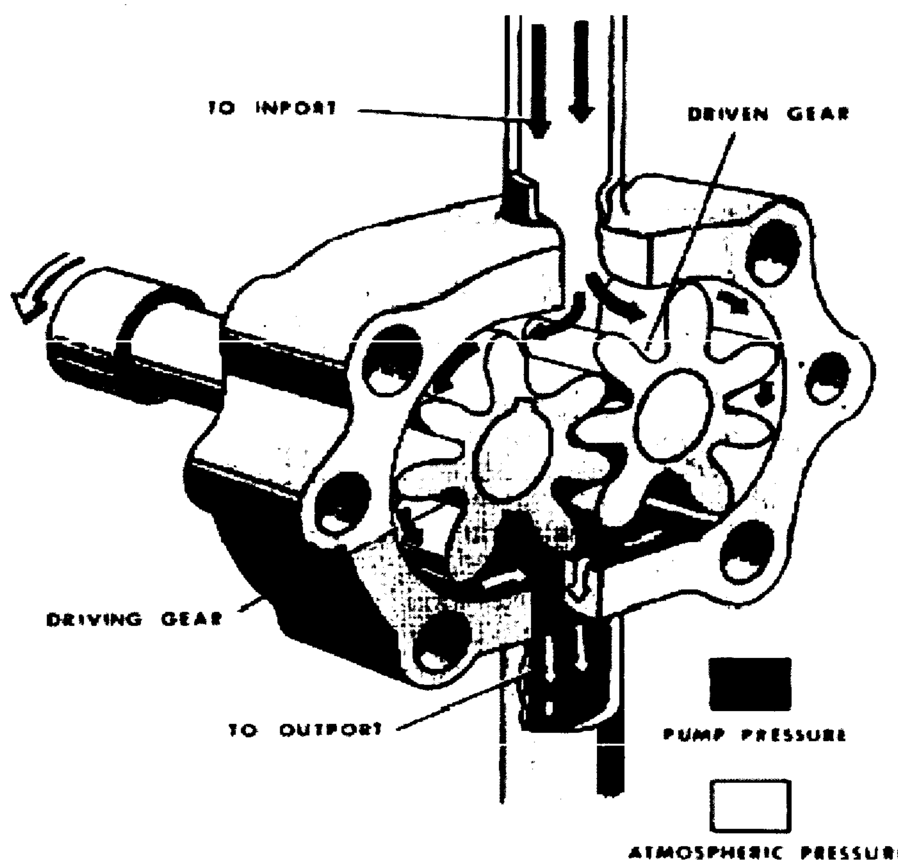
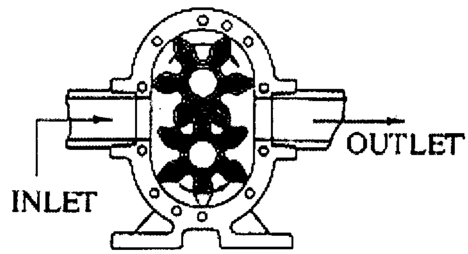
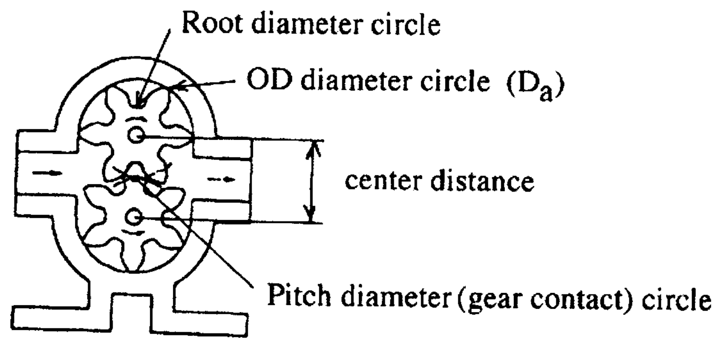


Figure A 3.2 Gear Type Rotary Pump





Addendum = radial distance from the pitch circle and OD circle $\left(a = \frac{D_a - D_p}{4}\right)$

Dedendum = radial distance from the pitch circle to the root circle $\left(b = \frac{D_p - D_r}{4}\right)$

Figure A 3.3 Geometry of an external gear pump