CFD Analysis of Internal Combustion Engine Processes

By

NOEL MECWAN

12MMET35

DEPARTMENT OF MECHANICAL ENGINEERING

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY

AHMEDABAD-382481

MAY 2014

CFD Analysis of Internal Combustion Engine Processes

Major Project

Submitted in partial fulfillment of the requirements

For the Degree of

Master of Technology in Mechanical Engineering (Thermal Engineering)

By

Noel Mecwan (12MMET35)

Guide By

Prof N K Shah

Prof A M Lakdawala

NIRMA UNIVERSITY
INSTITUTE OF TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

AHMEDABAD-382481

MAY 2014

Declaration

This is to certify that

1. The thesis comprises my original work towards the degree of Master of Technology in Thermal Engineering at Nirma University and has not been submitted elsewhere for a degree.

2. Due acknowledgment has been made in the text to all other material used.

Noel Mecwan

12MMET35

Undertaking for Originality of the Work

I, Noel Mecwan, Roll. No. 12MMET35, give undertaking that the Major Project entitled "CFD Analysis of Internal Combustion Engine Processes" submitted by me, towards the partial fulfillment of the requirements for the degree of Master of Technology in Mechanical Engineering (Thermal Engineering) of Nirma University, Ahmedabad, is the original work carried out by me and I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will result in severe disciplinary action.

____

Signature of Student

Date: ________________

Place: ________________

Endorsed by

(Signature of Guide)

Certificate

This is to certify that the Major Project entitled "CFD Analysis of Internal Combustion Engine Processes" submitted by Mr Noel Mecwan (12MMET35), towards the partial fulfillment of the requirements for the of Degree of Master of Technology in Mechanical Engineering (Thermal Engineering) of Institute of Technology, Nirma University, Ahmadabad is the record of work carried out by him under our supervision and guidance. In our opinion, the submitted work has reached a level required for being accepted for examination. The result embodied in this major project, to the best of our knowledge, have not been submitted to any other University or Institution for award of any degree.

Prof N K Shah
Guide
Assistant Professor,
Department of Mechanical Engineering,
Institute of Technology, Nirma University,
Ahmedabad.

Prof A M Lakdawala
Co- guide
Assistant Professor,
Department of Mechanical Engineering,
Institute of Technology, Nirma University,
Ahmedabad.

Dr R N Patel
Head,
Department of Mechanical Engineering,
Institute of Technology, Nirma University,
Ahmedabad.

Dr K Kotecha
Director,
Institute of Technology,
Nirma University,
Ahmedabad.

Acknowledgments

I take this opportunity to express deep sense of gratitude and sincere thanks for the invaluable assistance that I have received during the course of dissertation work at the worthy hands my honorable and learned guide Prof N K Shah and Prof A M Lakdawala of Mechanical Engineering Department, Nirma Institute of Technology, Ahmedabad. He is the constant source of encouragement and momentum that any intricacy becomes simple. I gained a lot of invaluable guidance and prompt suggestions from him during my thesis work. I remain indebted of him forever and I take pride to work under him.

My sincere thanks and due respect to Dr R N Patel, (HOD, Department of Mechanical Engineering, IT, NU) and Dr K Kotecha (Director, IT, NU) who have directly or indirectly helped me during this dissertation work. I also thank Mr Ritesh Vedawala (Lab Assistants IT, NU) for their valuable assistance.

I am very much thankful to all the faculty members of Mechanical Engineering Department for their expertise knowledge and guidance throughout my study, which has provide me valuable insight in number of areas.

I thank all of my friends who are there for me all the time, during my course work.

Noel Mecwan

12MMET35

Abstract

Day by day the pollution of exhaust gases created by vehicles on the road increases. One way to reduce the pollution is to use an alternative fuel. This solution is long-term solution. The short –term solution to reduce the pollution is to make engines more efficient and cleaner. To make engines efficient and cleaner the knowledge of processes happening inside an internal combustion engine is required. This knowledge of processes inside in an internal combustion engine can be obtained by two ways. One way is to perform experiment on an engine, which is difficult task. Another way is the numerical experiment which is not difficult task because it do not require an expensive and time consuming measurement set-up. Due to increased computing power today it is possible to model every processes happening inside an internal combustion engine.The aim of this project is to model the processes happening inside an internal combustion engine with the help of Computational Fluid Dynamics (CFD) package Fluent 6.3. The flow field inside an engine is obviously turbulent in nature. There are various turbulent models available in Fluent which can be used to model the turbulent processes inside an internal combustion engine. The simulation was performed on a single cylinder engine with inlet and exhaust valves having different combustion chamber configuraions. The simulation was performed on four different configurations of engine.The running engine simulations give realistic results which are used for better understanding of processes happening in an internal combustion engine. The way is open for future work in this topic.

List of Figures

List of Tables

Contents

Chapter 1

Introduction

The day by day the population increases due to which roads are becoming crowded. Due to which the pollution created by exhaust gases developed by engines of vehicles increases. Scientists are finding ways to reduce emission levels of engines. One way is to look for alternative fuel like hydrogen. But it will take a long time to switch from oil to hydrogen. Another short-term solution to this is to make engines more efficient and cleaner. To do so the knowledge of processes happening inside an internal combustion engine is required. The internal combustion engine is complex mechanical system due to which it is difficult to perform experiment on it and to carry out measurements. While the numerical approach has advantage that it do not require expensive and time consuming experimental set-up. Today due to increased power of computing it is easy and possible task to model processes happening inside an internal combustion engine. Compuattional Fluid Dynamics (CFD) is a very useful numerical tool to simulate different internal combustion engine processes. FLUENT offers different turbulnence models to simulate turbulence inside an internal combustion engine

1.1 Internal Combustion Engine

The purpose of internal combustion engines is the production of mechanical power from the chemical energy contained in the fuel. In an internal combustion engines, this energy is released by burning or oxidizing the fuel inside the engine. The fuel-air mixture before combustion and the burned products after combustion are actual working fluids. The work transfer which provide the desired output occur directly between these fluids and the mechanical components of the engine.The internal combustion engine are sprk-ignition engines and compression-ignition engines. Because of their simplicity, ruggedness and high power/weight ratio, these two types of engine are widely used in transportation and power generation. During past decades, new factors for change have become important and now significantly affect engine design and operation. These factors are, first, need to control the pollution created by engine and, second, need to make engines more efficient than before. [1]

1.2 Computational Fluid Dynamics

Analysis of the system having flow of fluid, transfer of heat and chemical reaction by using computer simulations is called CFD. CFD is very powerful technique and it used in a wide range of industrial and non-industrial application. Some examples are: [2]

- Aerodynamics
- Hydrodynamics
- Power plant
- Turbomachinery
- Chemical process engineering
- External and internal environment of buildings
- Biomedical engineering

CFD techniques are used in design and R & D of internal combustion engines from 1960s. Automobile industries uses CFD to predicts drag & lift forces, flow of air over car body. [2]

1.2.1 CFD Analysis of Internal Combustion Engine Processes

To develope engine which are more efficient and cleaner, knowledge of thermodynamic processes occuring inside engine is required. By knowledge of these processes one can modify the engine design to achieve better performance of engine. CFD is cheap and time saving than experimental procedure. Various CFD softwares are now available to carry out analysis of internal combustion engine.

Chapter 2

Literature Review

The modelling of engine processes is developing day by day as knowledge of different phenomenon increases as the capability of computers to solve complex equations increases. Modelling of an internal combustion engine gives major contribution in the design development of it.

2.1 Governing Equations

The description of flow is based on the three basic fundamental physical laws. These laws are: [3]

- Conservation of mass
- Conservation of momentum (Newton's Second Law)
- Conservation of energy

The first law gives continuity equation (Eq. 2.1). This equation represents conservation of mass in control volume for compressible flow

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial u_i}{\partial x_i} = 0 \tag{2.1}$$

In here ρ is the density and u_i is the velocity in direction i. The symbols t and x_i represents time and the position in the direction i respectively. The momentum equation is derived using Newton's second law. This equation is known as the Navier-Stokes equation.

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \vartheta \frac{\partial^2 u_i}{\partial x_j^2} \tag{2.2}$$

Where p represents the pressure and ϑ the kinematic viscosity. The second term on the left-hand side is the convective term and the second term on the right-hand side is the diffusion term. It says that the net force on a fluid element equals the mass times the acceleration of the element. The forces acting on the fluid element can be divided into Body and Surface forces. The last fundamental physical law results in energy equation

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p T \frac{\partial u_i}{\partial x_j} = \lambda \frac{\partial^2 T}{\partial x_j^2} \tag{2.3}$$

Where T is the temperature, c_p the heat capacity coefficient at constant pressure and λ the thermal conductivity coefficient. [3]

2.2 Turbulence Modelling

2.2.1 Turbulence

All flows occuring in engineering practice both simple ones and complicated ones such as jet, wake, pipe, etc., become unstable above a certain Reynolds number. The flows at low Reynold number are called laminar flows and the flows occuring at high Reynold number are called turbulent flows. In turbulent flow a chaotic and random state of motion develops in which the velocity and pressure change continuously with time. The results of experiments on fluid systems shows that at values below the critical Reynolds number the flow is smooth and adjacent layers of fluid slide past over each other in an orderly fashion. At values of the Reynolds number above critical Reynolds number a complicated series of events take place which eventually leads to a radical change of the flow character. In the final state the flow behaviour is random and chaotic.

In an internal combustion engine extreme fluid velocities are involved. Due to these high velocities the Reynolds number is also high which shows that the flow field inside an internal combustion engine is turbulent. Turbulent flows are characterised by fluctuating velocity fields. Transported quantities such as momentum, energy and species concentration are mixed due to these fluctuations. Presence of turbulence plays an important role in modelling combustion process, increase in turbulence results in better mixing of air and fuel in case of non-premixed combustion.

Modelling of turbulence is a difficult task when solving practical flow problems. To solve the governing equations exact to smallest scales more computational effort is required. Due to which these equations are Reynolds averaged. There are various turbulence models available, but none of them is generally accepted to describe all the processes in turbulent flow. [3]

2.2.2 Standard $k - \varepsilon$ Model

This is a two equation model and simplest one. It is generally used in simulation of turbulence due to its applicability, robustness and economy. The two transport equations for the kinetic energy and dissipation rate are solved to form a characteristic scale for both turbulent velocity and length. These scales represent the turbulent viscosity. The equations for the kinetic energy (2.4) and dissipation rate (2.5) are given below. [3, 4, 5]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_x}{\sigma_k})\frac{\partial k}{\partial x_j}] + G_k + G_b - \rho_\varepsilon - Y_m + S_k \tag{2.4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho k \varepsilon u_i) = \frac{\partial}{\partial x_i}[(\mu + \frac{\mu_x}{\sigma_t})\frac{\partial \varepsilon}{\partial x_j}] + C_{1\tau}\frac{\varepsilon}{k}(G_k + C_{3\tau}G_b) - C_{2\tau}\rho\frac{\varepsilon^2}{k} + S_\tau \tag{2.5}$$

In these equations, the generation of turbulence kinetic energy because of mean velocity gradients, G_b is the generation of turbulence kinetic energy because of buoyancy; represents the contribution of the fluctuating dilation is represented by Y_m in compressible turbulence to the overall dissipation rate;$C_{1\tau}$, $C_{2\tau}$ and $C_{3\tau}$ are constants; σ_k and σ_t are turbulent Prandtl for k and ε", respectively. S_k and S_τ are user defined source terms. The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε" as follows:

$$\mu_t = \rho C \mu \frac{k^2}{\varepsilon} \tag{2.6}$$

Where C_μ is a constant. [3, 4]

2.2.3 Standard $k - \omega$ Model

It includes modifications due to effects of low Reynolds number, compressibility and shear flow spreading. It is a two equation semi-empirical turbulence model. The $k - \omega$ model is comparable with the equation of transport for the kinetic energy. The equation of dissipation rate is different. In $k-\varepsilon$ model per unit mass dissipation is used while in $k-\omega$ model specific dissipation is used. It can be seen as ratio of ε to k. The transport equations for kinetic energy and specific dissipation are [3, 4]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k \bar{\mu_i}) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}] + G_k - Y_k \tag{2.7}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega \bar{\mu_i}) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\omega})\frac{\partial \omega}{\partial x_j}] + G_\omega - Y_\omega \tag{2.8}$$

Table 2.1: Turbulence model constants

Model	Constants				
Standard$k-\varepsilon$	$C_{1\varepsilon}=1.44$	$C_{2\varepsilon}=1.92$	$C_\mu=0.09$	$\sigma_k=1.0$	$\sigma_\varepsilon=1.3$
RNG $k-\varepsilon$	$C_{1\varepsilon}=1.42$	$C_{2\varepsilon}=1.68$	$C_\mu=0.0845$	$\alpha_0=1.0$	$\eta_0=4.38$
	$\beta=0.012$				
Standard $k-\omega$	$\alpha^*_\infty=1.0$	$\alpha_\infty=0.52$	$\alpha_0=1/9$	$\beta^*_\infty=0.09$	$\beta_i=0.072$
	$R_\beta=8$	$R_k=6$	$R_\omega=2.95$	$\zeta^*=1.5$	$M_{t0}=0.25$
	$\sigma_k=2.0$	$\sigma_\omega=2.0$			

Where G_k is the generation of kinetic energy and $G\omega$ is the generation of specific dissipation rate. Y_k and Y_ω shows dissipation of k and ω. The turbulent viscosity μ_t is obtained by

$$\mu_t = \alpha^* \rho \frac{k}{\omega} \tag{2.9}$$

Where α^* is low Reynolds correction factor. The $k-\omega$ model offers good results for wall bounded flows and free shear flows so this model is also very suitable for internal combustion engine simulations. Table 2.1 shows values of different constants used in turbulence models. [3]

2.2.4 RNG $k-\varepsilon$ Model

The RNG $k-\varepsilon$ model is obtained from the Navier-Stokes equations using a technique called "renormalization group".The analytical derivation results in a model with different constants than those in the standard $k-\varepsilon$ model. Also additional terms and functions will apear in the transport equation for the kinetic energy and dissipation. Equation (2.10) and (2.11) are transport equations of RNG $k-\varepsilon$ model [3, 4]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k \bar{u_i}) = \frac{\partial}{\partial x_j}(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}) + G_k - \rho\varepsilon \tag{2.10}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon \bar{u_i}) = \frac{\partial}{\partial x_j}(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}) + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon \tag{2.11}$$

The main difference with standard $k-\varepsilon$ model is an additional term R_ε which can be shown as

$$R_\varepsilon = \frac{C_\mu \rho \eta^3 (1-\eta/\eta_0)}{1+\beta\eta^3}\frac{\varepsilon^2}{k} \tag{2.12}$$

2.3 Cold Flow Simulation

In cold flow simulation an internal combustion engine is modelled without fuel combustion. All the processes like intake, compression, expansion and exhaust are simulated except fuel combustion in cold flow simulation. Port injection can modelled and evaporation of fuel droplets can be simulated. The interaction of the fuel spray with the intake valve is modelled through the wall film modelling feature available in FLUENT.

2.4 Combustion Modelling

Combustion is one of the most important processes in engineering, which involves turbulent fluid flow, heat transfer, chemical reaction, radiative heat transfer and other complicated physical and chemical processes. Typical engineering applications include internal combustion engines, power station combustors, aeroengines, gas turbines combustors, boilers, furnaces etc. CFD lends itself very well to the modelling of combustion. Various combustion models are now available such as [7]

- Gas jet model
- Spray breakup model

Combustion processes are governed by basic transport equations for fluid flow and heat transfer with additional models for combustion chemistry, radiative heat transfer and other important sub-processes. Combustion is a complex subject, and combustion modelling requires a considerable amount of knowledge and experience.

There are many types of combustion processes. Gaseous fuel combustion, liquid fuel combustion, spray combustion, solid fuel combustion, pulverised fuel combustion are a few of the many other processes used in a wide variety of application areas. In an internal combustion engines generally two types of combustion processes are involved depending on the type of fuel used. First is the premixed combustion for engines using petrol as a fuel and second type is non- premixed combustion for engines using diesel as a fuel. [2, 6, 8]

2.4.1 Premixed Combustion Modelling

As mentioned earlier, fuel and air are mixed prior to combustion in premixed combustion. The strength of mixture may be expressed by the equivalence ratio. During combustion in a premixed flame, the flame front propagates at a certain speed and leaves burnt products behind the flame front. In premixed combustion, laminar and turbulent flame speeds and a parameter known as the reaction progress variable are used to formulate models. If we define T_u as the temperature of unburnt gas, T_b as the temperature of burnt gas and T as the flame temperature, then the reaction progress variable c is defined as [4]

$$c = \frac{T - T_u}{T_b - T_u} \tag{2.13}$$

2.4.2 Non- premixed Combustion Modelling

In non- premixed combustion situations such as furnaces, fuel and air streams are mixed by fluid flow and turbulence, and the resulting combustion temperatures, species concentrations and distribution of species are very much controlled by fluid flow. Equations governing turbulent non- premixed combustion requires averaging and modelling. The first problem that needs be addressed is the fact that strong and highly localised heat generation in combusting flows causes the density to vary as a function of position in combusting flows. There will also be density fluctuations if the flow is turbulent. The Reynolds decomposition of general flow variable is as follows [4]

$$\phi = \bar{\phi} + \phi' \tag{2.14}$$

For the variables in a reacting flow this yields

$$u_i = u_i \bar{+} u_i' \tag{2.15}$$

$$p = \bar{p} + p' \tag{2.16}$$

$$\rho = \bar{\rho} + \rho' \tag{2.17}$$

$$h = \bar{h} + h' \tag{2.18}$$

$$T = \bar{T} + T'' \tag{2.19}$$

$$Y_k = Y_k + \bar{Y}_k \tag{2.20}$$

2.5 Simulation of Engine Using Software

Fluid flow problems can be solved using CFD codes which are structured on numerical algorithm. User interfaces are included in CFD packages to give input parameters of problem and to analyze results. CFD codes have main three components: [2]

(i) Pre- processor,

(ii) Solver

(iii) Post- processor.

General program structure of GAMBIT and FLUENT packages. [9]

- Modeling the engine using GAMBIT
- Analyzing the engine in FLUENT

2.5.1 Pre- processor

Pre-procesing consists of giving input to CFD computer program by user interface and transform this input into a form suitable for use by the solver. The pre- processor involves following activities: [10]

- Definition of the geometry of the region of interest: the computational domain
- Grid generation: the sub division of the domain into a number of small sub domains
- Selection of the physical and chemical phenomenon that need to be modelled
- Definition of fluid properties
- Specification of appropriate boundary conditions

The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside every cell. The accuracy of result obtained by CFD program depends on the number of cells. In general the larger the number of cells, the better the solution accuracy. Both the accuracy of a results and its cost in terms of necessary computer hardware and calculation time are dependent on the health of the grid. Optimal meshes are often non- uniform: finer in areas where large variations occur from point to point and coarser in regions with little change. [2]

2.5.2 Solver

There are three distinct types of numerical solution techniques: finite difference, finite element and finite volume methods. Most CFD packages concerned central to the finite volume method. Numerical algorithm consists: [10]

- Integration of the governing equations involved by fluid flow of all control volume of domain

- Disctretisation- conversation of the obtained integral equations onto a system of algebraic equations
- Obtaining the solution of algebric equations by iterative method

The resulting statements express the exact conservation of relevant properties for each finite size cell. This clear relationship between the numerical algorithm and the underlying physical conservation principle forms one of the main attractions of the finite volume method and makes its concept much simple to understand by engineers than the finite element and finite difference methods. The conservation of a general flow variable φ, e.g. a velocity component or enthalpy, within a finite control volume can be expressed as a balance between the various processes tending to increase or decrease it. [2]

2.5.3 Post- processor

As in pre- processing, a huge amount of development work has recently taken place in post-processing field. Due to increased popularity of workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualisation tools like: [10]

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Colour PostScript output

More recently these facilities include animation for dynamic result display, and in addition to graphics all codes produce trusty alphanumeric output and have a data export facilities for further manipulation external to the codes. [2]

2.5.4 Dynamic Meshing

As the geometry of the internal combustion engine is deforming the mesh must also deform according to the geometry for good quality results. FLUENT offers this facility of deforming mesh which is called as dynamic meshing. In dynamic meshing the various parameters affects the deformation of the initial mesh. The dynamic meshing is done with smoothing, layering and remeshing. In smoothing the factor to be set is spring constant factor. In remeshing the factors to be set are minimumm length scale, maximumm length scale and maximum cell skewness. In layering the factors are to be set are split- factor and collapse factor. The various factors discussed before are set according to the grid size applied to the meshing of internal combustion engine. After the setting of these parameters the various events such as opening and closing of inlet and exhaust valve are defined in the "Events" section of the dynamic meshing in the FLUENT. These events are defined according to valve timing diagram of the internal combustion engine used. [11]

2.5.4.1 Smoothing

Smoothing is a dynamic mesh update method. FLUENT offers three types of smoothing methods which are

(i) Spring based smoothing

(ii) Laplacian smoothing

(iii) Boundary layer smoothing

In the spring-based smoothing method, the edges between any two mesh nodes are idealized as a network of interconnected springs. The initial spacings of the edges before any boundary motion constitute the equilibrium state of the mesh. A displacement at a given boundary node will generate a force proportional to the displacement along all the springs connected to the node.Spring based smoothing method can be used for any cell or face zone whose boundary is moving or deforming. [11]

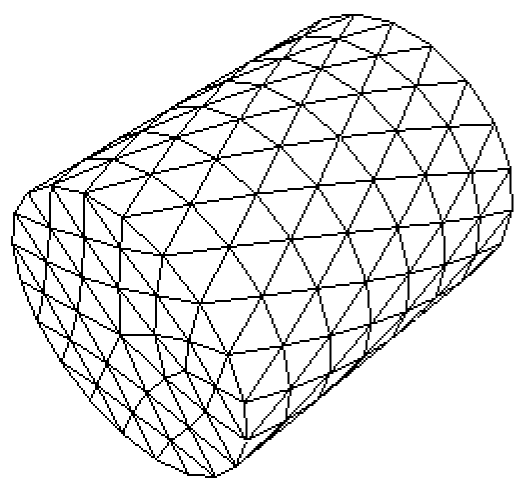

Figure 2.1: Spring based smoothing on interior nodes: Start [11]

Laplacian smoothing is the most commonly used and the simplest mesh smoothing method. This method adjusts the location of each mesh vertex to the geometric center of its neighboring vertices. This method is computationally inexpensive but it does not guarantee an improvement on mesh quality, since repositioning a vertex by Laplacian smoothing can result in poor quality elements. To overcome this problem, FLUENT only relocates the vertex to

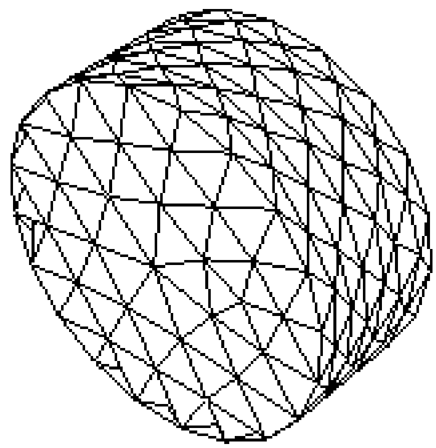

Figure 2.2: Spring based smoothing on interior nodes: End [11]

the geometric center of its neighboring vertices if and only if there is an improvement in the mesh quality (i.e., the skewness has been improved).

The boundary layer smoothing method is used to deform the boundary layer during a moving-deforming mesh simulation. For cases that have a Mesh Motion UDF applied to a face zone with adjacent boundary layers, the boundary layer will deform according to the UDF that is applied to the face zone. This smoothing method preserves the height of each boundary layer and can be applied to boundary layer zones of all mesh types (wedges and hexahedra in 3D, quadrilaterals in 2D).

2.5.4.2 Layering

Dynamic layering can be used to add or remove layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface.The dynamic mesh model in FLUENT allows you to specify an ideal layer height on each moving boundary.With the constant height option, the cells are split to create a layer of cells with constant height, while with the constant ratio option, the cells are split such that locally, the ratio of the new cell heights is exactly same everywhere [11]

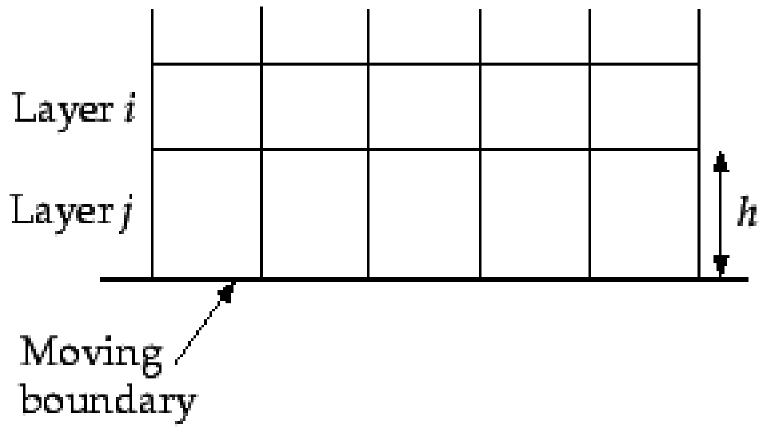

Figure 2.3: Dynamic Layering [11]

2.5.4.3 Remeshing

When the boundary displacement is large compared to the local cell sizes, the cell quality can deteriorate or the cells can become degenerate. This will invalidate the mesh (e.g., result in negative cell volumes) and consequently, will lead to convergence problems when the solution is updated to the next time step. To overcome this problem, FLUENT agglomerates cells that violate the skewness or size criteria and locally remeshes the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells. Otherwise, the new cells are discarded. FLUENT includes several remeshing methods that include local remeshing, local face remeshing (for 3D flows only), face region remeshing, and 2.5D surface remeshing (for 3D flows only) [11]

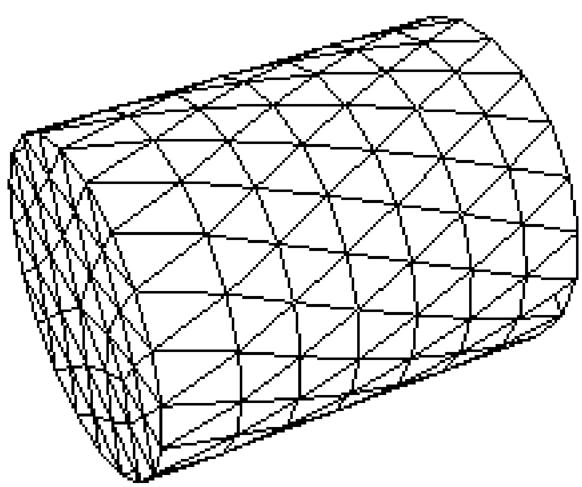

Figure 2.4: Expanding Cylinder Before Region Face Remeshing [11]

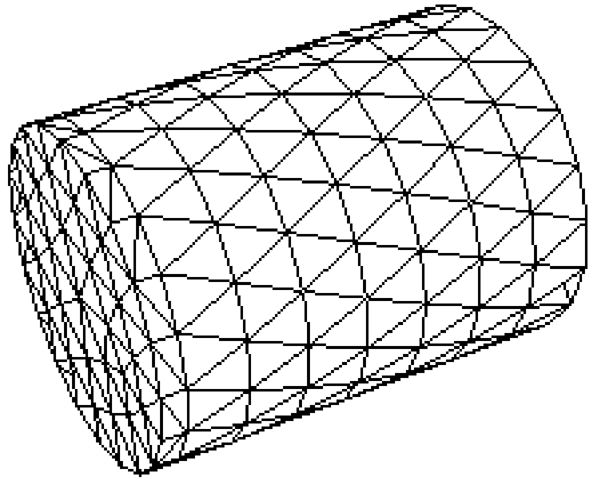

Figure 2.5: Expanding Cylinder After Region Face Remeshing [11]

(i) Local remeshing:

Using the local remeshing method, FLUENT marks cells based on cell skewness and minimum and maximum length scales as well as an optional sizing function.

(ii) Local face remeshing:

FLUENT marks deforming boundary faces for remeshing based on moving and deforming loops of faces.

(iii) Face region remeshing

FLUENT marks the region of faces on the deforming boundaries at the moving boundary based on minimum and maximum length scales.

2.6 Engine Process Modelling

The processes occurring in the working fluid of an internal combustion engine are numerous and very complex. Therefore an accurate numerical modeling of the processes is required. For this task, two basic approaches have been developed. As mentioned before in introducing the simulation tools these can be mainly categorized as thermodynamic or fluid dynamic in nature, depending on whether the implemented equations are based only on energy conservation (Zero dimensional) or on a full analysis of the fluid motion (Multi- dimensional). An engine simulation tool independently from its approach is actually a collection of various engine process models (physical, chemical and thermodynamic phenomena or just control/actuation models like: injector model, spark ignition model that in a successive step generate a phenomenon).

Although the conservation equations that set the balance of the thermodynamic engine-processes are well known and evident like in any other thermodynamic system, the procedure towards a reliable and appropriate process modeling does not follow a unique way and still represents a most complicated and controversial task. Due to the complexity of engine processes, the insufficient understanding at fundamental level and very often still the limited computational resources, it is not possible to model engine processes that describe all important aspects starting from the basic governing equations alone. First of all, in order to govern the complexity, each modeled process must be limited to its relevant effects on engine behavior that have to be analyzed, then the formulations of the critical features of the processes have to be based on a keen combination of assumptions, approximations, phenomenological and eventually empirical relations. This procedure permits both to bridge gaps in our phenomena understanding and to lessen the computational time by reducing the number of required equations. [6]

2.7 Research papers reviewed

G. Sureshbabu et al. studied pressure and temperature of combustion chamber of engine and obtained results show that pressure and temperature distribution is not uniform in the near by region of inlet and exhaust valves. This non- uniform pressure and temperature distribution results in abnormal combustion and unburnt fuel remains in combustion chamber [6]. S. Gavudhama Karunanidhi et al. analyzed combustion in diesel engine and obtained that by chaning injection timing the peak pressure and temperature can be changed and it can be used to find optimal injection timing [8]. G. Sucharitha and A. Kumaraswamy studied internal combustion engine with three different types of piston shapes and obtained results shows that the piston shape does not play important role during intake stroke, the effect of piston shape is observed at top dead center [9]. Benny Paul and V. Ganesan studied engine having different types of manifolds and results shows that the helical-spiral geometry generates high velocity magitude [10].

2.8 Motivation

There mainly two types of internal combustion engine namely spark-ignition engine and compression ignition engine developed. Since the introduction of engine a lot of research work is going on the different engine processes. Today the main aim of the internal combustion enigne research is to obtain best performance and lowest possible emissions. The quality of internal combustion engine is improved and transportation system became more reliable than before. In present era there is a lack of oil resources and demand of oil is increased, this lead to develope more efficient and cleaner engine. This can be obtained by conventional method such as by performing experiment on it, but it is time consuming and costly. Due to well develped computational facility there is a new way to design and analysis an internal combustion engine called the numerical analysis which saves time and costs also.

For better design of engine the knowledge of actual processes happening inside an internal combustion engine is necessary. In the processes happening inside combustion chamber the fluid dynamics and chemical interaction are involved. Modification in combustion chamber is one way to increase efficiency and reduce emissions of an internal combustion engine. To do so a better knowledge of in-cylinder fluid dynamics is very important. Analysis of engine can be done with different models available ranging from zero dimensional to multi-dimensional. Many softwares are available to model the in-cylinder processes to obtain knowledge about different in-cylinder phenomenon.

From the invention of the engine lots of research work have been done to know in-cylinder fluid flow and its effect on engine performance. The fluid motion inside cylinder of engine depends on various parameters such as combustion chamber design, intake system, engine speed, etc. Obviously the fluid flow inside engine cylinder is turbulent and non-stationary. This turbulence inside engine cylinder can be determined by both experiment and modern numerical approach. Performing experimnets is time consuming and costly, while numerical approach is easy to adopt due to increased compuattion power.

2.9 Objectives

- To simulate different processes of an internal combustion engine i.e. intake, compression, expansion and exhaust without combustion happening inside engine cylinder in 2D.
- To know effect of combustion chamber geometry on in-cyinder turbulence.

2.10 Problem Definition

The engine geometry used for the simulation is a two valve engine cylinder with inlet and exhaust manifolds. The simulation of engine's inlet, compression, expansion and exhaust stroke will be done with and without combustion happening inside in engine cylinder for the analysis.

Chapter 3

Mathematical Modeling and Numerical Approach

3.1 Governing Equations

In simulation of an internal combustion engine the equations solved by solver are

(i) Mass conservation

(ii) Momentum conservation equation

(iii) Energy equation

(iv) Turbulence

3.2 Turbulence Models

Turbulent flows are characterised by flucyuating velocity fields. These fluctuations mix transported quantities such as momentum, energy and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are computationally too expensive to simulate. Instead the governing equations can be time- averaged, ensemble- averaged otherwise manipulated to remove small scales, resulting in modified sets of equations. Modified sets of equations contain unknown variables and turbulence models are required to determine these variables. Different turbulence models available in the FLUENT are [4, 3, 8]

(i) Spalart- Allmaras model

(ii) $k - \varepsilon$ models

(iii) $k - \omega$ models

(iv) Reynolds stress model (RSM)

(v) Detached eddy simulation model (DSM)

(vi) Large eddy simulation model (LES)

In present simulation work the standard $k - \varepsilon$ model was used as turbulence model.

3.3 Pre- processing

Gambit is a pre-processor by which a virtual model is created for the given problem. Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. GAMBIT was used for modeling and meshing.

The geometry of engine was meshed with grid sizes of 0.25mm, 0.5mm, 1mm, 2mm and 3mm in the combustion chamber. The meshed file was exported for further work to be done in the FLUENT software. Figure (3.1) and (3.2) shows geometry meshed with different grid sizes

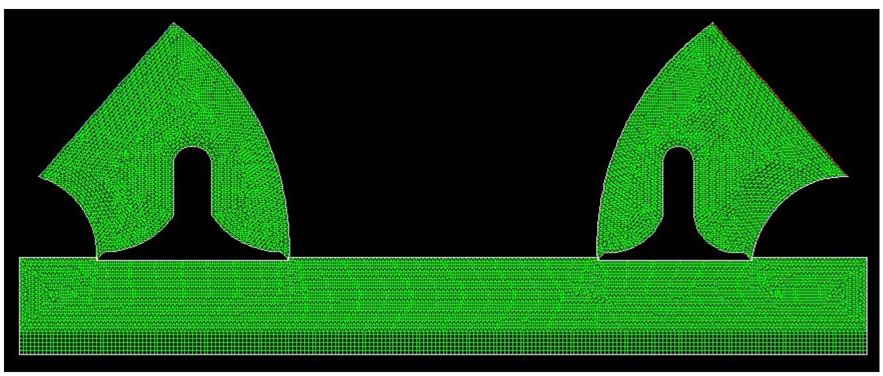

Figure 3.1: Meshed Geometry (0.5mm grid size)

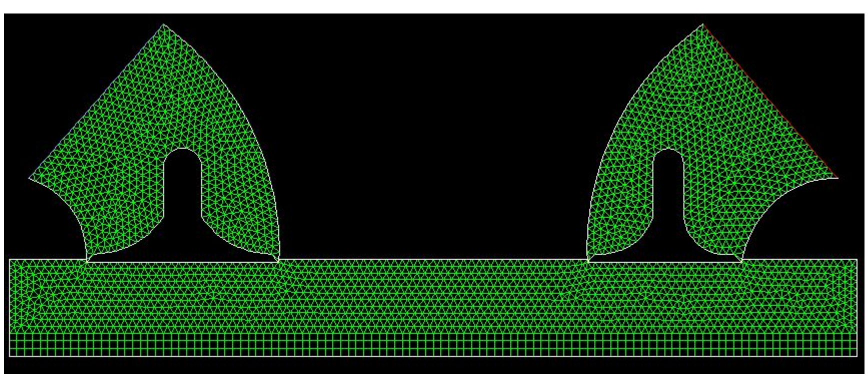

Figure 3.2: Meshed Geometry (1mm grid size)

3.3.1 Meshing Geometry

As discussed above the geometry was meshed with different grid sizes. The flow inside an internal combustion engine is turbulent. Turbulence occurs in total cylinder, so it is very important region of interest. The grid size of cylinder is changed for simulation. The clearance region of cylinder is meshed with triangular mesh with different grid sizes of 0.25mm, 0.5mm, 1mm, 2mm and 3mm. The swept region by piston of cylinder is meshed with rectangular mesh with different grid sizes of 0.25mm, 0.5mm, 1mm, 2mm and 3mm. The cylinder, inlet manifold and exhaust manifold regions is meshed with triangular mesh. The meshed geometry is exported in .msh format for its further use in FLUENT.

3.4 Computational Procedure in FLUENT

All the analysis in FLUENT were done with two dimensional double precision mode (2DDP). The computational procedure used is briefly explain here. First of all mesh file was loaded and its grid was checked. A segregated, unsteady solver with 1^{st} order implicit formulation was selected. Controls for solution were set to SIMPLE first order upwind differencing scheme. Standard $k-\varepsilon$ model was used for turbulence modelling. Model was initialized while residuals and force monitors were set to track the appropriate data. Animation for each time step was applied for contours. At every 5 time steps case and data file was saved. Total no of time step are 720 for the total simulation crank angle period of 720 degree. After successful completion of simulation post processing work was done like writing a hard copy of vorticity contour, pressure contour, hard copy of turbulent kinetic energy (k) versus time-steps.

3.4.1 Discretization Scheme

Several discretization schemes are available for steady and unsteady analysis i.e. PISO, SIMPLE or SIMPLEC. PISO algorithm to solve the flow problem of internal combustion engine. The standard pressure interpolation scheme is used. SIMPLE (Semi- Implicit Method for Pressure Linked Equation) was used in these simulations, which uses relationship between pressure and velocity. 1st order upwind scheme was used for momentum, turbulent kinetic energy, turbulence dissipation rate and energy equation.

3.4.2 Boundary Condition

The inlet is set as pressure inlet and exhaust is set as pressure outlet. Pressure inlet is set with 101325 pascal gauge pressure, 1% turbulent intensity, 318K temperature. Pressure outlet is set with 101325 pascal gauge pressure, 1% turbulent intensity, 318 K temperature.

Under relaxation factors and convergence criteria used are mentioned in table 3.1 and 3.2 respectively.

Table 3.1: Under relaxation factors

Factors	Value
Pressure	0.5
Density	1
Body forces	1
Momentum	0.01
Turbulent viscosity	1
Turbulent kinetic energy	0.8
Turbulent dissipation rate	0.8
Energy	0.8

Table 3.2: Convergence criteria

Convergence criteria	Value
Continuity	0.0001
X velocity	0.001
Y velocity	0.001
Energy	1e-06
Turbulent kinetic energy	0.0001
Turbulence dissipation rate	0.0001

The solution is initialized with following parameters.

- 99000 Pascal for pressure
- 0 m/s for X and Y velocity
- 0.1 m^2/s^2 for turbulent kinetic energy
- 0.1 m^2/s^3 for turbulent dissipation rate
- 318 K for temperature

The simulation is setup to iterate 720 timesteps with 2500 maximum iterations per timestep.

Chapter 4

Results and Discussion

4.1 Simulation of different cases in 2D

Following different cases are soved for parametric study

1. Flat cylinder head, flat piston with inlet valve diameter 25mm and exhaust valve diameter 20mm with smoothing, layering & remeshing technique, 1mm grid size, atmospheric inlet boundary condition.

2. Flat piston, pentroof combustion chamber with inlet valve diameter 30mm & canted exhaust valve diameter 28mm with smoothing, layering & remeshing technique, 1mm grid size, atmospheric inlet boundary condition.

3. Flat piston, pentroof combustion chamber with flat cylinder head with inlet valve diameter 30mm & canted exhaust valve diameter 28mm with smoothing, layering & remeshing technique, 1mm grid size, atmospheric inlet boundary condition.

4. Flat piston, wedge type cylinder head with canted inlet valve diameter 30mm & canted exhausst valve diameter 28mm with smoothing, layering & remeshing technique, 1mm grid size, atmospheric inlet boundary condition.

Figure 4.1, 4.2, 4.3 & 4.4 represents computational domains for cases 1, 2, 3 & 4 respectively.

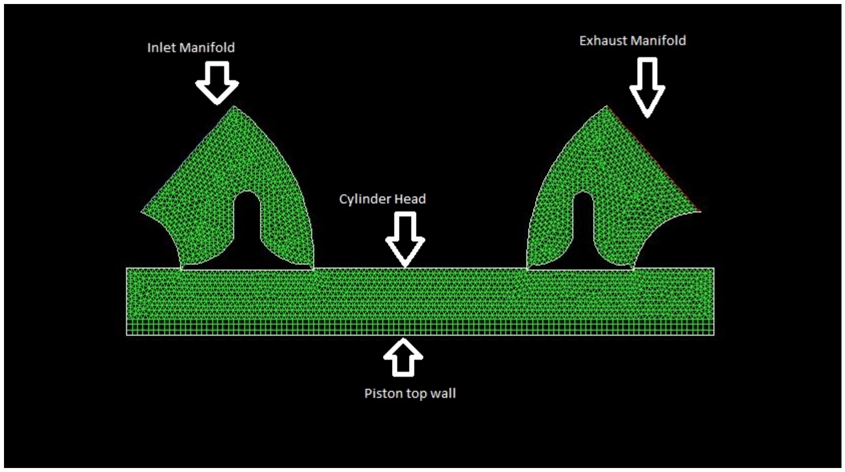

Figure 4.1: Computational domain of case1

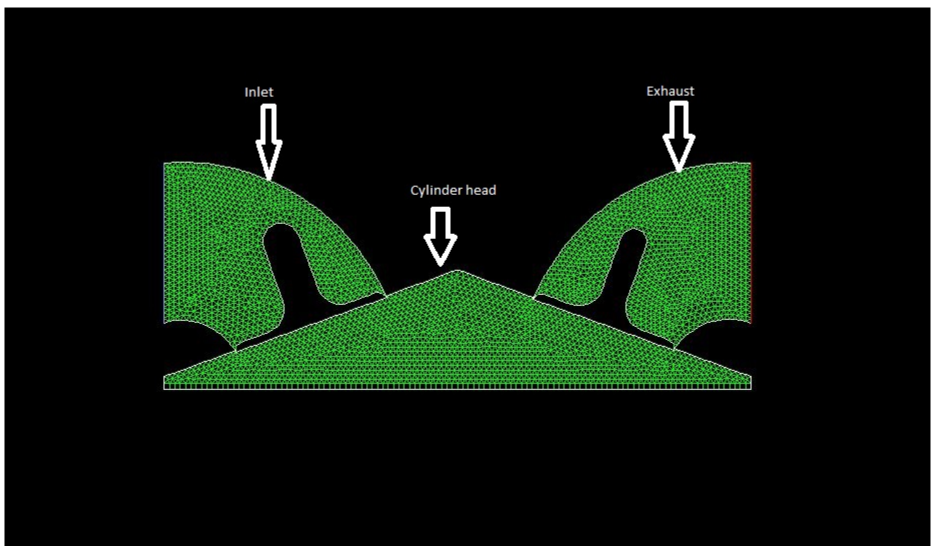

Figure 4.2: Computational domain of case2

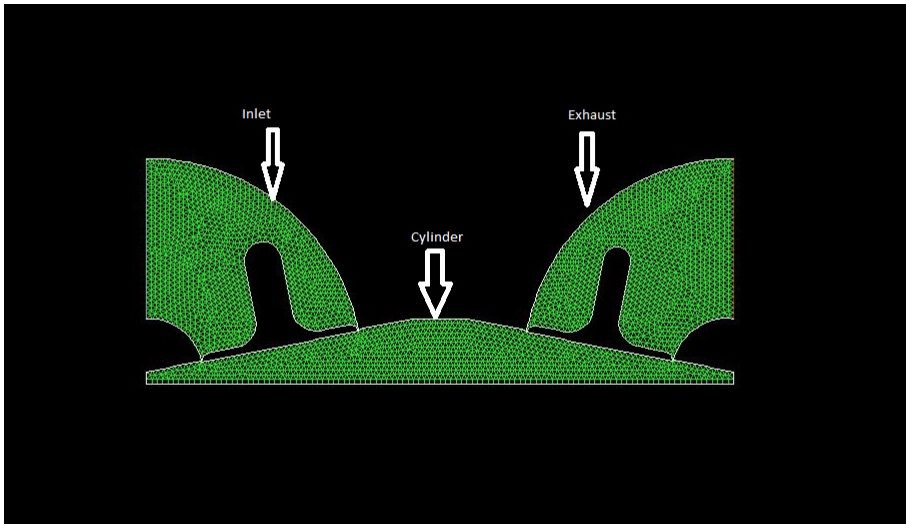

Figure 4.3: computational domain of case3

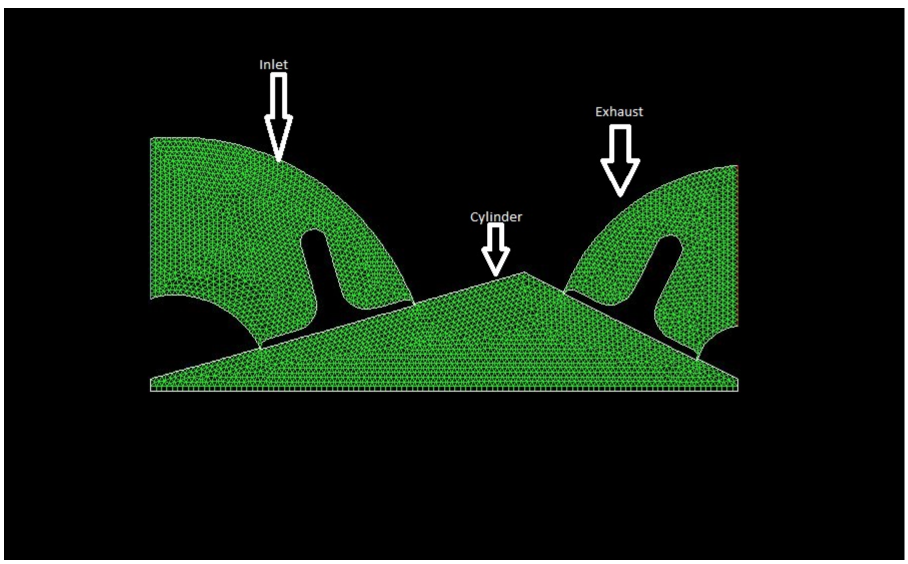

Figure 4.4: Computational domain of case4

4.2 Grid Independence

The computational domain includes both intake and exhaust ports and valves, the cylinder and piston. The engine geometry is meshed with different grid sizes varying from 0.25mm, 0.5mm, 1mm, 2mm & 3mm. The connection of different sub-domains is created by using interfaces in between them. For grid size of 3mm the simulation time is 4 hours approximately. For 2mm grid size the simulation time is approximately 7 hours. For 1mm grid size the simulation time is 12 hours. Thus as the grid size decreases the compuational time increases.

In order to ensure grid independence five different simulations were performed. The results of each simulations are compared in graph of turbulent kinetic energy versus crank angle as shown in figure (4.5). It is clear from the comparison of results that as the grid size decreases the value of the volume averaged turbulent kinectic energy is increasing during intake and compression stroke. During intake stroke due to valve motion, a jet of fluid is coming in combustion chamber because of which turbulent kinetic energy increases. During compression stroke due to wall reaction turbulent kinetic energy decreases. At the end of compression stroke sufficient amount of turbulence is required for better mixing of air and fuel. It is observed from figure (4.5) due to piston movement turbulent kinetic energy decreases due to wall interactions. At the last in exhaust stroke due to valve motion again the turbulent kinetic energy increases slightly.

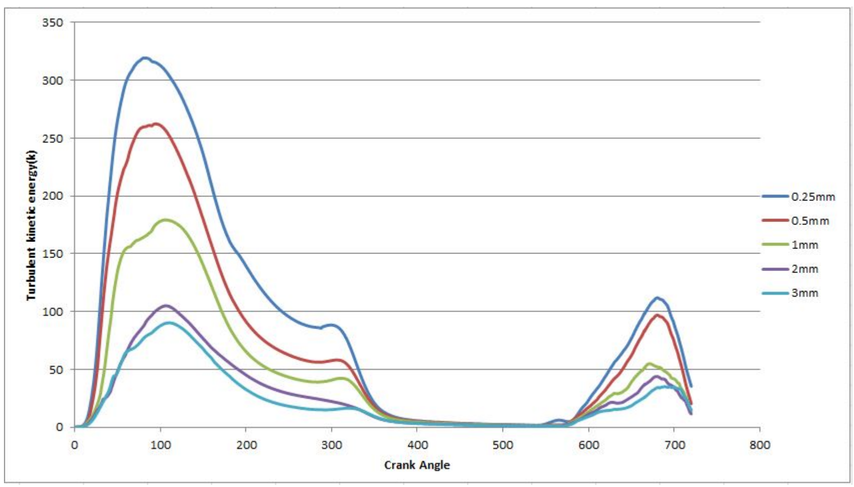

Figure 4.5: Volume average turbulent kinetic energy versus crank angle for different grid sizes

4.3 Effect of combustion chamber configuration

The simulation is performed on four different cases of combustion chamber configurations. Figure 4.6 shows turbulent kinetic energy comparison of different combustion chamber geometries used for simulation . Higher turbulent kinetic energy is observed for wedge shaped combustion chamber geometry. Turbulent kinetic energy for pentroof with flat cylinder head combustion chamber geometry is similar to flat cylinder head combustion chamber geometry. The peak in turbulent kinetic energy is achieved at different crank angle for different combustion chamber geometries. Pentroof combustion chamber geometry gives minimum turbulent kinetic energy. Wedge shaped combustion chamber geometry is better from intake turbulence point of view.

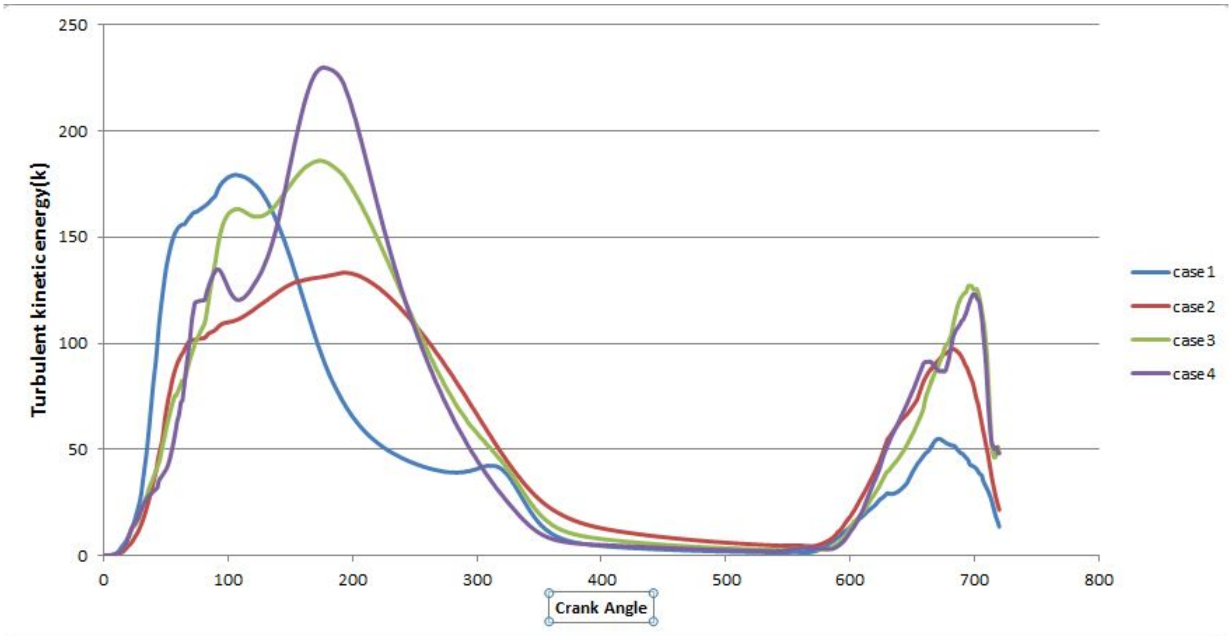

Figure 4.6: Volume average turbulent kinetic energy versus crank angle for different combustion chambers

Figure 4.7 shows pressure versus crank angle diagram for cases 2,3 & 4. The pressure remains constant in intake stroke and continue to increase in compression process and maximum pressure is achieved at the end of compression process after that it decreases in expansion and exhaust process. As combustion is not involved in simulation, peak pressure is achieved nearer 18 bar.

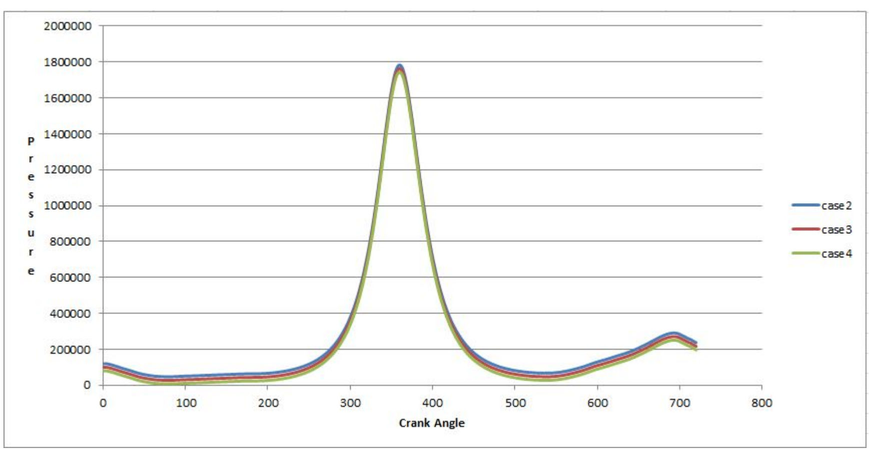

Figure 4.7: Pressure versus crank angle diagram of different cases

Figure 4.8, 4.9, 4.10 & 4.11 shows contours of turbulence for cases 1, 2, 3 & 4 at crank angles 60, 120, 180, 240, 300 & 360 degrees. As discussed in previous sections during intake stroke due to valve motion turbulence is generated from incoming jet of fluid. Turbulence

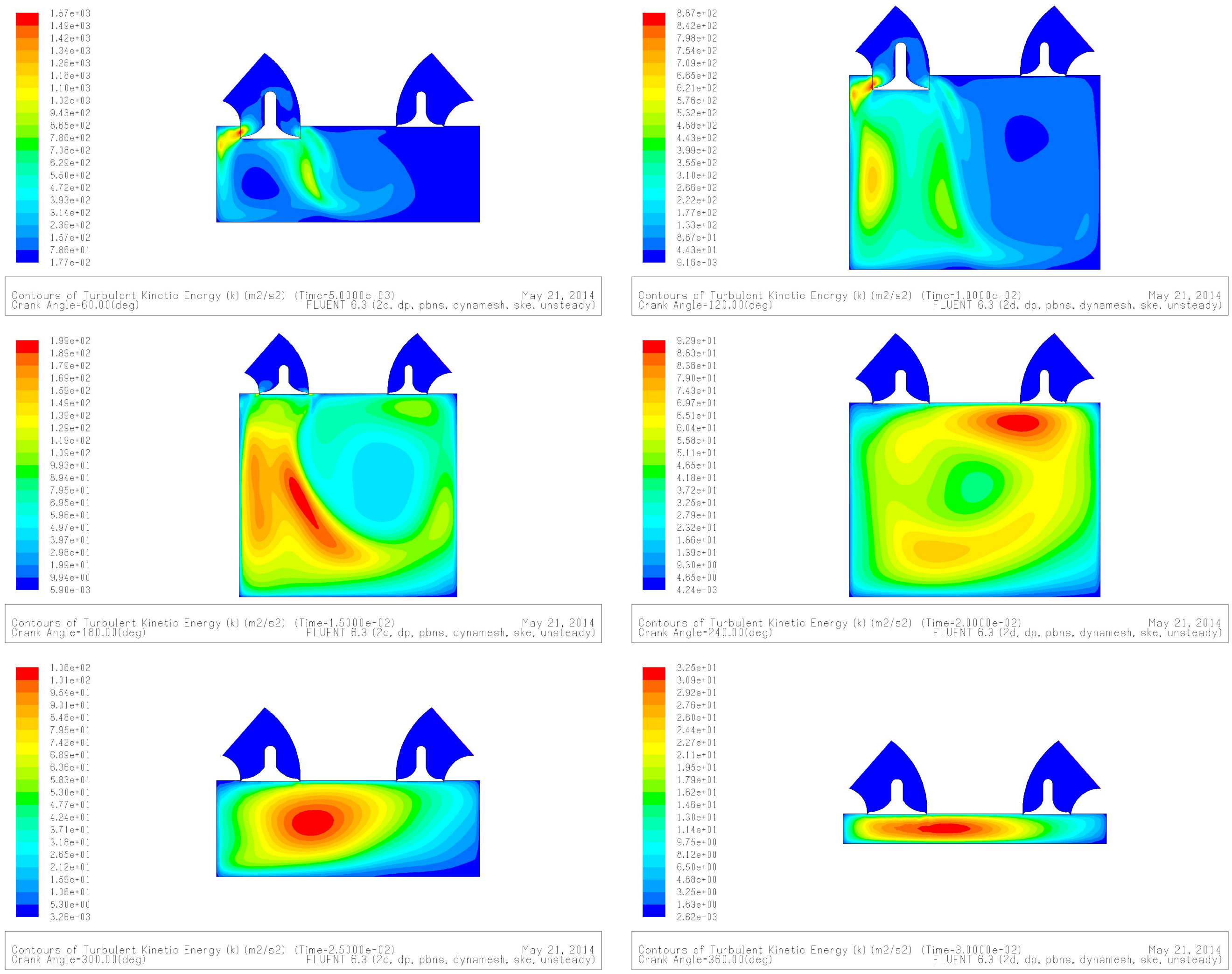

Figure 4.8: Contours of turbulence for case1

is maximum nearer to 180 degree crank angle and it dacays after the inlet valve is closed. In compression process tubulence decreases due to wall interactions. It can also be observed that flow in the cylinder of engine is highly turbulent in nature. The turbuence is highest in the wedge shape combustion chamber geometry.

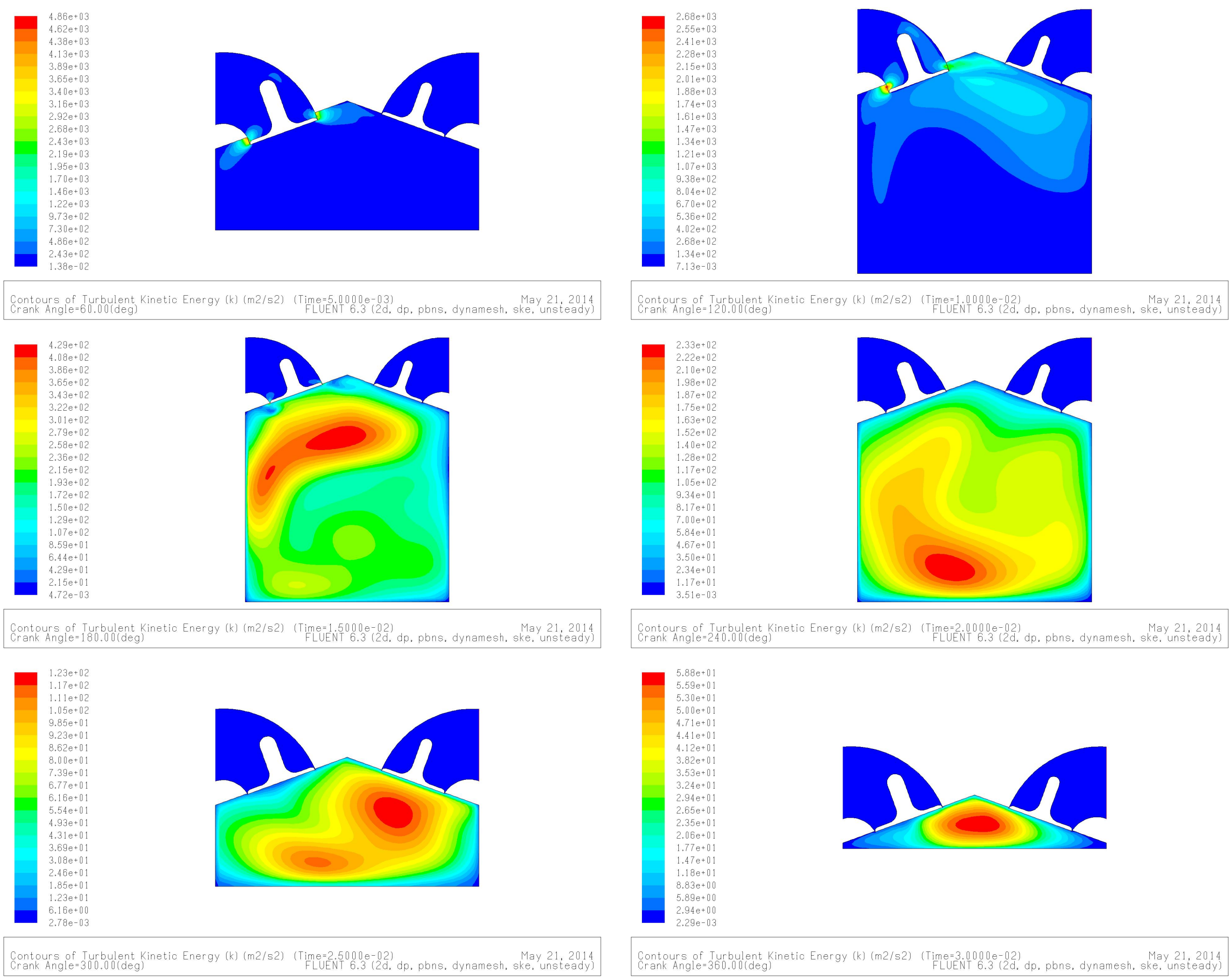

Figure 4.9: Contours of turbulence for case2

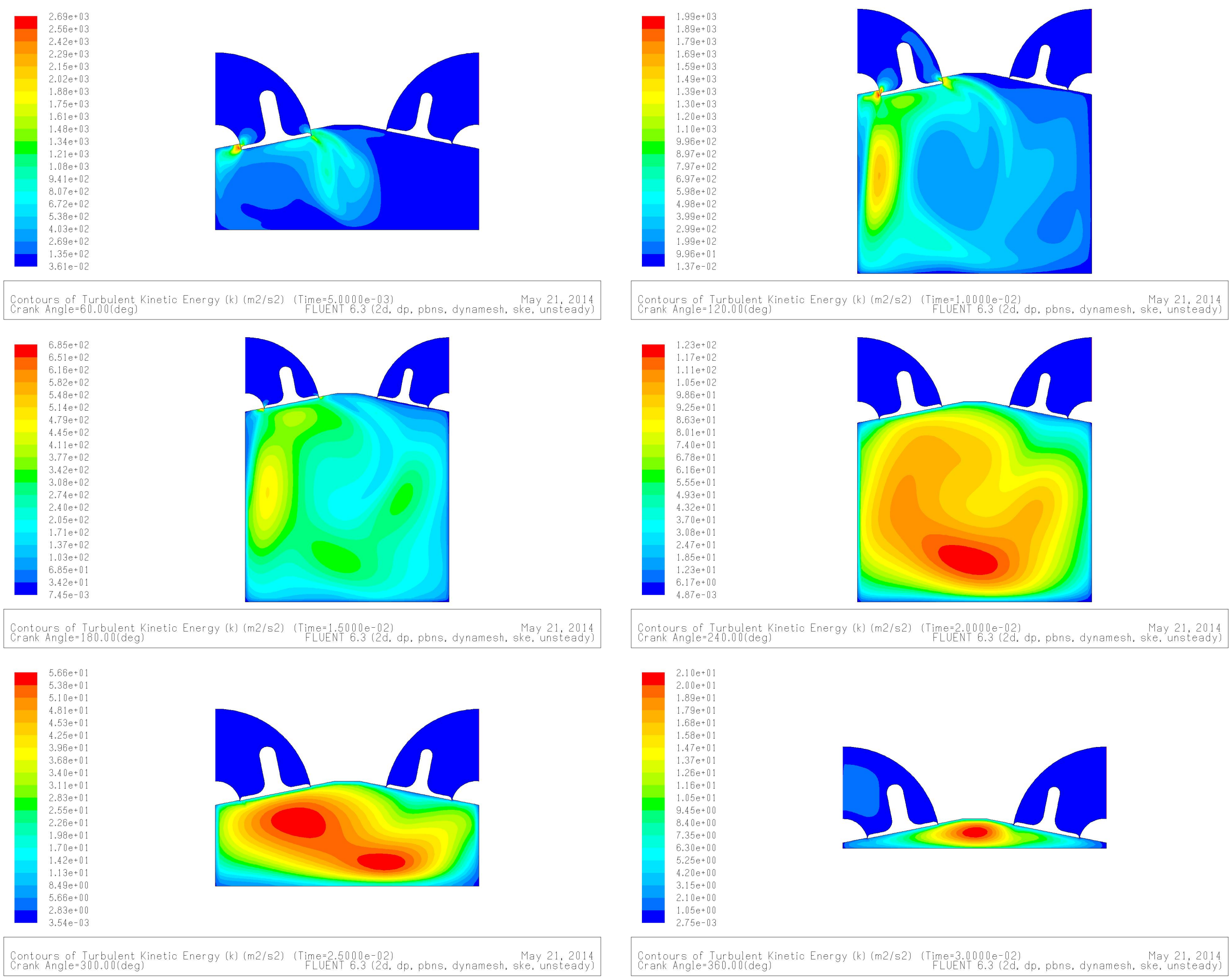

Figure 4.10: Contours of turbulence for case3

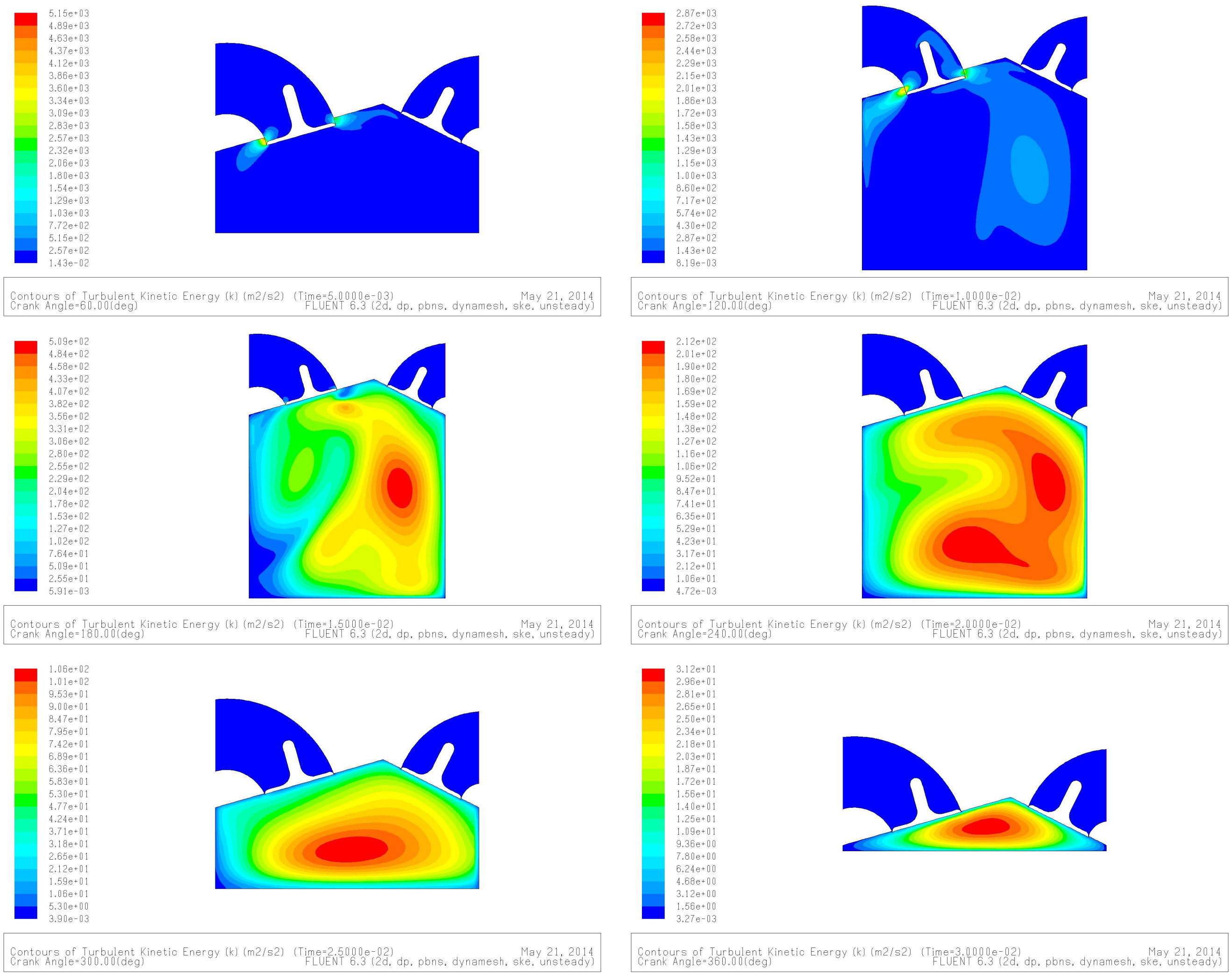

Figure 4.11: Contours of turbulence for case4

4.4 Conclusion

From the results it is clear that during the intake stroke of engine the high turbulence is set because of in intake stroke high velocity jet enters in the cylinder. A shear layer is devloped due to high velocity inlet and this increases the turbulence. The turbulence increases as the piston speed increases. As the inlet valve closes there is a decrease in the turbulence. In compression stroke turbulence decreases due to wall interactions. At the end of compression the turbulence decays. Turbulence increases at the exhaust stroke because after compression stroke the fluid flow in engine reverses and this devlopes a shear which increases turbulence. After exhaust stroke the turbulence further decreases. Based on simulations it was observed that wedge shaped combustion chamber geometry is better.

4.5 Future Work

The present work of simulation of an internal combustion engine describes the analysis of engine in motored case (Without combustion) in 2D. In future the simulation can be done with combustion happening inside engine. The 3D simulation of an internal combustion engine can also be done.

Bibliography

[1] John B Heywood, " Internal combustion engine fundamentals", McGraw Hill Publications; 1998.

[2] H K Versteeg, W Malalasekera, " An introduction to computational fluid dynamics- The finite volume method", Pearson; 2013.

[3] J. J. M. Smits, " Modelling of a fluid flow in an internal combustion engine", Report number WVT 2006.22, Department of Mechanical Engineering, Eindhoven University of Technology; 2006.

[4] Vishal S. Soni, " Multidimensional modelling of internal combustion engine processes", M. Tech Thesis, Nirma University; 2011.

[5] S. M. Jameel Basha, P. Issac Prasad, K. Rajagopal, "Simulation of in-cylinder processes in a DI diesel engine with various injection timings", ARPN Journal of Engineering and Applied Sciences, Vol. 4, No. 1; 2009.

[6] G. SureshBabu, S. D. V. S. Jagadeesh, U. B. Saicharan, P. R. S. Praneeth, " Analysis of a single cylinder combustion engine using CFD", International Journal of Innovative Technology and Exploring Engineering, Volume-2, Issue-5; April 2013.

[7] Shijin Shuai, Neerav Abani, Takeshi Yoshikawa, Rolf D. Reitz, Sung Wook Park, "Evaluation of the effects of the injection timing and rate- shape on diesel low temperature combustion using advanced CFD modelling", Elsevier, Fuel 88 p. 1235-1244; 2009.

[8] S. Gavudhama Karunanidhi, Melvinraj C. R, Sarath Das K. P, G. Subba Rao, "CFD studies of combustion in diesel engine", International Journal of Engineering Research and Applications, Vol. 3, Issue 4, p. 827-830; 2013.

[9] G. Sucharitha, A. Kumaraswamy, "Analysis on three dimensional flow of direct-injection diesel engine for different piston configurations using CFD", India Journal of Science and Technplogy, Vol- 6 (5s); May 2013.

[10] Benny Paul, V. Ganesan, "Flow field development in a direct injection diesel engine with different manifolds", International Journal of Engineering, Science and Technology, Vol. 2, No. 1, p. 80-91; 2010.

[11] FLUENT user guide, 2005.

[12] R. Simpson, S. Gao, "Computational modelling and analysis of transient flow in an IC engine with generic inlet track", International Journal of Engineering and Technology, Vol. 5, No 5, October 2013.