ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

CFD Modeling and Simulation of Progressive HCCI Engines


Soubhagya Ranjan Das
M.Tech (Thermal Engineering) Student
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali.

B.V.V. Prasada Rao
Assistant Professor
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali.

ABSTRACT

At present, it is highly required from the automobile sector to develop clean technologies with lower fuel consumption for ambient air quality improvement, green house gas reduction and energy security. Furthermore, due to continuously stringent emission legislation and the fast depletion of the primary energy resources, the development of new highly efficient and environment friendly combustion systems becomes of paramount importance and hence research need to be done in this area. One such combustion system is Homogeneous Charge Compression Ignition (HCCI) technology, which has the potential to reduce oxides of nitrogen (NOx) and particulate matter (PM) simultaneously maintaining the thermal efficiency at par with that of conventional diesel engine combustion.

However, some issues such as combustion phasing control, controlled auto-ignition, operating range, homogeneous charge preparation, cold start, pressure rise rate and noise and emissions of unburned hydro carbon (UHC), and carbon monoxide (CO) need to be solved for successful operation and therefore commercial application of HCCI engine. Other similar combustion concepts, which can be considered as the extension of HCCI, are Low Temperature Combustion (LTC) and Stratified Charge Compression Ignition (SCCI).

In the present study the Computational Fluid dynamics (CFD) code FLUENT is used to model complex combustion phenomenon in compression ignition (CI) engine. The experiments were accomplished on single cylinder and DI engine, with full load condition at constant speed of 1500 rpm. Combustion parameters such as cylinder pressure, rate of pressure rise and heat release rate were obtained from experiment. The numerical modeling is solved by unsteady first order implicit, taking into account the effect of turbulence. For modeling turbulence Renormalization Group Theory (RNG) k- ε model is used. The sub-models such as droplet collision model and Taylor Analogy Breakup (TAB) model are used for spray modeling. The wall-film model is used to assess spray-wall interaction. Modeling in-cylinder combustion, species transport and finite-rate chemistry model is used with simplified chemistry reactions. The results obtained from modeling were compared with experimental investigation. Consequences in terms of pressure, rate of pressure rise and rate of heat release are presented. The rate of pressure rise and heat release rate were calculated from pressure based statistics. The modeling outcome is discussed in detail with combustion parameters. The results presented in this paper demonstrate that, the CFD modeling can be the reliable tool for modeling combustion of internal combustion engine

1. INTRODUCTION

1.1 Overview

It is highly required from the automobile sector to develop clean technologies with lower fuel consumption for the improvement of the ambient air quality, reduction of the green house gases, security of the primary energy resources and fulfilment of the increasingly stringent emission norms. Therefore, the fuel and the engines used in transportation sector have to cope up with two major challenges of improving efficiency and reducing emissions in a highly competitive era.

Compression Ignition (CI) and Spark Ignition (SI) combustion are two primary technologies with established use in automobile sector. SI and CI engines use fossil fuels and both have their own merits and demerits. The characteristic feature of traditional SI engine is flame propogation for combustion. A conventional SI engine uses a homogeneous fuel/air mixture, which is prepared in the intake port and then undergoes induction compression. This homogeneous mixture is ignited by the spark discharge given theoretically at the end of the compression stroke. The start of combustion in SI engines is governed by varying the spark discharge timing. As the SI engines use fixed air/fuel ratio, therefore the engine load regulation is made possible only by governing the air mass flow into the combustion chamber. The throttle used for this purpose results in pumping losses and reduction in efficiency. It produces extremely low soot emissions because it uses premixed charge with stoichiometric fuel-air ratio (λ=1), but it also has lower thermal efficiency due to pumping loss and a lower compression ratio (generally in the range of 8 to 12), which is limited by knocking. Hence, SI engines with accurate control of air-fuel ratio and three way catalytic convertors are very clean power producing machines but their efficiency is limited because of throttling, knocking and the lean flammability limits On the other hand, a conventional CI engine uses a heterogeneous fuel/air mixture. The air alone is sucked in the suction stroke. And the fuel is injected directly and rapidly into the combustion chamber theoretically at the end of the compression stroke. This injected fuel is then auto-ignited after a short delay time. The processes, which take place between the fuel injection and the start of combustion events are cumbersome, which include droplet formation, collisions, break-up, evaporation. The rate of combustion is affected by these processes. In CI engines, only a fraction of the air and fuel is premixed and burns fast, whereas for the larger part of the fuel, the time scale of evaporation, diffusion etc. is more than the chemical time scale. Hence, the air- fuel mixture within the combustion chamber can be divided into two regions- the high fuel concentration regions and the high temperature flame regions. In the fuel rich regions, the rate of soot formation is high due to absence of oxygen (O2). Though, some soot may be oxidized with the increase in in-cylinder temperature. NOx is produced at high rates in the high temperature regions. In a traditional diesel engine, the cylinder temperature is about 2700 K. In the CI engines, there is less pumping loss and the higher efficiency due to higher compression ratio (generally in the range of 12 to 24). CI engines are very efficient power producing machines, but they have a constraint in the form of trade-off between oxides of nitrogen (NOx) and Particulate Matter (PM) emissions. Therefore, it is necessary to keep the maximum cylinder temperature low in order to minimize the NOx emission and also to promote better fuel-air mixing in order to reduce the smoke emissions . However, CI engines are widely used for personal as well as commercial transportation due to their excellent fuel economy. Both CI and SI engines are major contributors to urban air pollution. Carbon mono-oxide (CO) and unburnt hydrocarbons (UBHC) emissions from these engines contribute to global warming. Oxides of nitrogen and hydrocarbons emitted from these engines react in atmosphere to form photochemical smog, particulate matter emitted from diesel engines causes asthma and respiratory episodes. Due to adverse impact of these pollutants on human health, increasingly stringent emission legislations are being enforced throughout the world, which require simultaneous reduction of PM and NOxemissions . It has been indicated by some of the researchers that the fuel economy of the conventional

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

piston engines has the possibility for further improvement by 25% or more while lowering the harmful emissions to near zero level by applying advanced combustion technologies

Fundamentals of HCCI Combustion

HCCI principle

In HCCI mode of combustion, the fuel and air are mixed prior to the start of combustion and the mixture is auto-ignited spontaneously at multiple sites throughout the charge volume due to increase in temperature in the compression stroke. In this mode, the combustion process is arranged in such a way that the combustion takes place under very lean and dilute mixture conditions, which results in comparatively lower bulk temperature and localized combustion temperature, which therefore, considerably reduces the NOx emissions. Furthermore, unlike conventional CI combustion, in HCCI mode the fuel and air is well mixed (homogeneous). So, the absence of fuel rich regions in the combustion chamber results in considerable reduction in PM generation. Therefore, due to absence of locally high temperatures and a rich fuel-air mixture during combustion process, the simultaneous reduction of NOx and PM emissions is made possible.

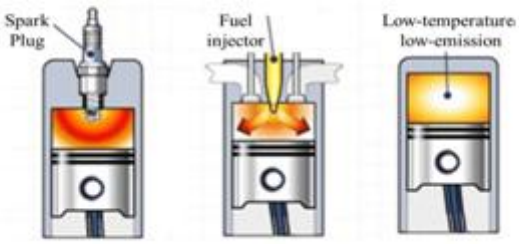


Fig. Schematic diagram of HCCI combustion

Fig. represents the comparison among SI, CI and HCCI operations. In HCCI engines, a lean homogeneous flammable mixture (fuel–air equivalence ratio Φ<1) is formed, before the start of ignition and is auto ignited as a result of increase in temperature in the compression stroke . The HCCI operation is similar to SI engine as both use the homogeneous charge for combustion and similar to CI engine as both depend on the auto ignition of the mixture. Thus, HCCI combustion can be regarded as the hybrid of SI and CI combustion processes. In SI engines, three zones of combustion namely burnt zone, unburned zone and a thin flame reaction zone in-between are generated for turbulent flame propagation through the cylinder. In CI engines, fuel is diffused into the cylinder and a definite diffusion flame traverses within the cylinder. Whereas, in HCCI engine spontaneous auto-ignition of whole cylinder mixture at multiple sites occurs without any diffusion flame or flame front propagation.

Theoretically, diesel fuelled HCCI combustion has the potential to reduce PM and NOx emissions to near-zero level by employing two basic processes: firstly, by forming a homogeneous mixture and secondly, by auto-igniting this mixture due to compression heat.

However, these same features also lead to the main challenges. As diesel fuel possesses high viscosity, a wide range of boiling points, and a high cetane number. It means that the required mixing time scale for forming a homogenous mixture is very long but the chemical ignition time scale is very short . Furthermore, fuel-wetting of diesel-fuelled HCCI combustion is also an issue under consideration.

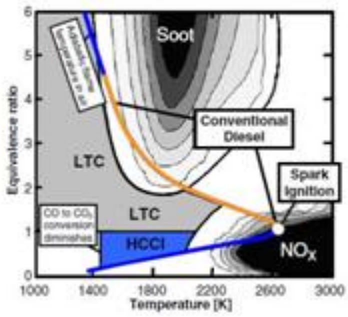


Fig. – Conventional combustion & variants of diesel combustion on T-ϴ space

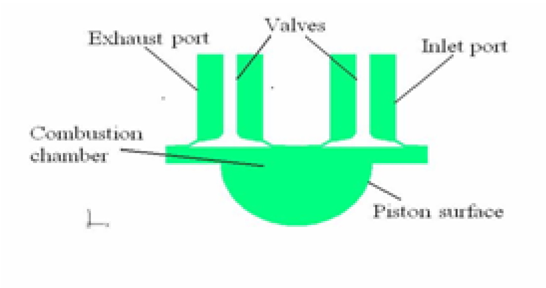
As it is evident from Fig. that for conventional diesel combustion, the adiabatic flame temperature in air stretches through both the soot and NOx generation regions. Conceptually, in a conventional diesel combustion, the fuel and air charge undergoes rich combustion of about ϴ=4 at the end of the adiabatic mixing process during the ignition delay period, and

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

then combustion moves to completion in a stoichiometric (ϴ=1) diffusion flame. This rich combustion may cause soot production depending upon the soot formation tendency of the fuel and the ϴ-T distribution during the pre-mixed combustion period. Once the combustion of the fuel prepared to flammability limits during the ignition delay period is over, the rate of combustion further depends on a mixing controlled basis. In the conventional diesel combustion, thermal NOX is produced when the local in- cylinder temperatures are in excess of 1800-2000 K and there is enough oxygen available. Considering approximately adiabatic combustion, these combustion regions fall in soot and NOx regions respectively, resulting in high levels of emissions. SI combustion also generates significant amount of NOx emissions, but they are removed by modern three-way catalysts. As it is clear from Fig. 2 that the HCCI combustion falls outside the soot and NOx islands. In HCCI combustion as the flame temperatures are considerably lower than the conventional diesel combustion due to lean or diluted mixture, the NOx emissions are low.

Geometry Development and Meshing of Computational Domain

CFD codes are structured around the numerical Algorithms that can transaction with fluid flow problems. Since the computational domain is very complex, poised of four zones with different topologies, each zone has been meshed separately. In present work Geometry has been modeled and meshed in preprocessor Gambit. Figure 1 shows the computational domain of two dimensional combustion chamber geometry counting inlet and exhaust ports. Both intake ports have been meshed with same orientation in the flow direction and they are joined with a cylindrical structured mesh in the zone upstream of the valves. During the compression stroke, once the intake valve is closed, the intake port subdomains are disconnected from the calculation, so that only the combustion chamber is considered. In order to ensure grid independence and improved accuracy of the results, three calculations of the compression stroke, starting at BDC and with no valve movement (intake valve disconnected), were performed and compared to the solution of the absolute intake and compression simulation. The combustion chamber is bowl in piston type, which having a hemispherical groove on piston top. The geometry has been modeled at its zero crank angle position at TDC as shown in figure 2. In ICE it is necessary that for obtaining realistic simulations, computation must include combustion chamber geometry with inlet and exhaust valve. The computations performed on bowlinpiston type combustion chamber reveled that, instead of suction stroke at the end of compression stroke the geometry plays important role to access the combustion.



Geometry of combustion chamber with valves

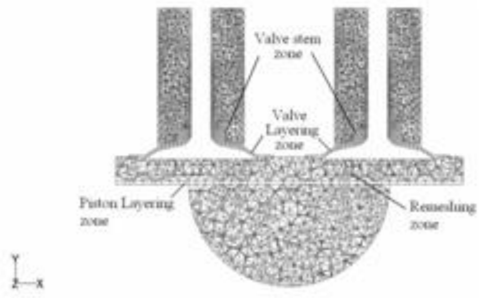


Figure 1 (b): Mesh structure of computational domain for model geometry.

2. 0 Main Approaches/Strategies for Mixture Preparation for Diesel Fuelled HCCI Combustion

The preparation of the homogeneous mixture is the key in reducing the PM emissions and the local fuel-rich regions to minimize the NOx emissions. The local fuel-rich regions can be reduced by an effective mixture preparation. Due to very short span of time available for the preparation of the mixture, it becomes challenging to prepare a homogeneous mixture for cycle-to cycle variation of speed and load. The effective mixture

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

preparation for the HCCI combustion refers to both the fuel–air homogeneity and temperature control during combustion. The various strategies for mixture preparation for diesel fuelled HCCI combustion has been shown in Fig. 3. The external mixture formation suffers with a drawback of low volumetric efficiency, whereas the in-cylinder mixture formation is prone to oil dilution.

Diesel-fuelled HCCI combustion can be achieved with various methods such as high-pressure multiple injections, high density or high boosting, controllable EGR, variable compression ratio, flexible fuel-supplying strategies, variable valve actuation, and fuel-reforming technology.

Diesel-fuelled HCCI combustion can be classified into the following types: 1) external mixture formation or port fuel injection, 2) early direct injection based on the combination of fuel bumping, small- diameter nozzle, narrow spray angle, and multiple injections, 3) late direct injection supplemented by a high level of EGR and high swirl ratio, and 4) compound HCCI combustion with PFI combined with early direct injection. It should be noted hereby that with reference to the typical nomenclature for diesel fuelled HCCI combustion, many researchers use the term Premixed-Charge Compression-Ignition (PCCI) because, theoretically, it is not possible to form a genuinely homogeneous mixture in such a short span of time . External mixture preparation is a straight forward method for homogeneous mixture preparation for diesel fuelled HCCI combustion, whereas early and late direct injection strategies, though very effective, are more suitable for gasoline like fuels .

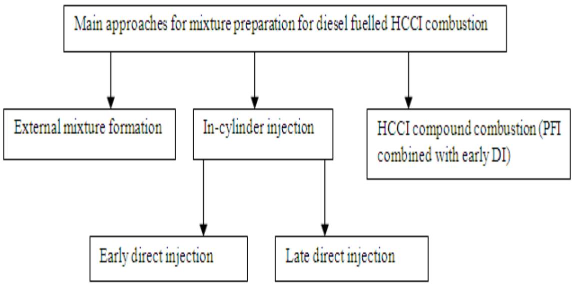


Fig. Main approaches/strategies for mixture preparation for diesel fuelled HCCI combustion

METHODOLOGY AND PROBLEM DEFINITION

3.0 Problem Definition

Experiments were performed on a fully instrumented, single cylinder, four stroke, direct injection, at constant speed of 1500 rpm. The specification of test engine is given in the Table 1. The cylinder pressure data were averaged over 5 consecutive cycles for the same load condition. Pressure was recorded with Crank angle sensor resolution 1degree, speed 5500 rpm has TDC pulse and Piezo sensor Range 5000 PSI. For digital load measurement strain gauge sensor, range 050 Kg with eddy current dynamometer is used. Laboratory view based engine performance analysis software package is provided for on line performance evaluation.

Table 1: Engine specifications

Model and model	Kirloskar, TV1
Rated output (BHP/KW)	7 / 5.2
Bore X Stroke (mm)	87.5 x 110
Displacement volume (cc)	661
Compression Ratio	17.5:1
Fuel Injection Timing	23° BTDC (static)
Injector opening pressure (bar)	200205
Injector inclination	15° to vertical axis

3.1 Model development

The problem is to be solved as unsteady first order implicit with turbulence effects considered to simulate the combustion for CI, DI engine. The numerical methodology is segregated pressure based solution algorithm. For solving species, the discrete phase injection with species transport equation and finite rate chemistry reactions are used. The upwind scheme is employed for the discretization of the model equations. FLUENT uses a control volume based technique to convert the governing equations to algebraic equations that can solve numerically. The governing equations for mass, momentum and energy equations used and appropriate initial boundary conditions were chosen for combustion analysis.

3.2 Turbulence model

Turbulence is distinguished by fluctuation of velocity field. In this work well known RNG k ε model is used for modeling turbulence. The RNG k ε model was derived using a thorough statistical technique. It is analogous in form to the standard k ε model but having an advantage to include effect of swirl, which is important for ICE combustion analysis.

3.3 Spray breakup model

FLUENT offers two spray breakup models, the TAB and the wave model. In the present work TAB model is used. The TAB model is based on the analogy between an oscillating and distorting droplet and a spring mass system. The distorting droplet effect is considered in the present study.

3.4 Droplet collision model

Droplet collision model includes tracking of droplets; for estimating the number of droplet collisions and their outcomes in a computationally efficient manner. The model is based on O'Rourke's method, which assumes stochastic approximation of collisions. When two parcels of droplets collide then algorithm further establish the type of collision. Only coalescence and bouncing outcomes are measured.

3.5 WallFilm Model

Spraywall interaction is an important element of the mixture creation process in diesel engines. In a DI engine, fuel is injected directly into the combustion chamber, where the spray can impinge upon the piston. The modeling of the wallfilm inside a DI engine is compounded by the occurrence of carbon deposits on the surfaces of the combustion chamber. This carbon deposit soak up the liquid layer. It is understood that the carbon deposits adsorb the fuel later in the cycle. The wallfilm model in FLUENT allows a single constituent liquid drop to impinge upon a boundary surface and form a thin film. Interactions during impact with a boundary and the criteria by which the regimes are detached are based on the impact energy and the boiling temperature of the liquid

3.6 Combustion model

The combustion model was combined with species transport and finiterate chemistry with simplified chemistry reactions to simulate the overall combustion process in a diesel engine. This approach is based on the solution of transport equations for species mass fractions. The reaction rates that emerge as source terms in the species transport equations are computed from well known Arrhenius rate expressions. A chemical kinetic mechanism from the FLUENT database is used for modeling diesel combustion.

4.COMPUTATIONAL FLUID DYNAMICS AS A DESIGN TOOL

The real world of fluid dynamic machines-compressors, turbines, flow ducts, airplanes etc-is mainly a three-dimensional world i.e., three dimensional flow solutions are abundant because the storage and speed capacity of digital computers were sufficient to allow to CFD to operate any complex practical problem in three dimensional world indeed some computer programs for calculations of three dimensional flows have become industry standards, resulting in their use as a tool in design process

Example: - In design of modern high-speed aircraft, such as Northrop f-20 for finding complicated transonic aerodynamic flow patterns on the plane CFD is used as design tool.

4.1ENGINEERING APPLICATIONS OF CFD AUTOMOBILE AND ENGINE APPLICATIONS

To improve the performance of modern cars and trucks the automobile industry has accelerated its use of high technology research and design tools. One of these tools is CFD. Whether it is the study of external flow over the body of vehicle, or internal flow through the engine CFD helping automotive engineers to better understand the physical flow process, and in turn to improved vehicles. Today, the massive power of modern CFD Is being applied by automotive engineers to study all aspects of the details of internal combustion engine flow fields, including combustion turbulence, and coupling with the manifold and exhaust pipes. As an example of sophistication of modern CFD applications to a gas turbine engine, a finite volume mesh which is wrapped around both the external region out side the engine and internal passages through the compressor, the combustor, the turbine, etc. From complex mesh the relevant equations are solved after discretization using tools of CFD, and results are analyzed.

INDUSTRIAL MANUFACTURING APPLICATIONS

When a mould was being filled with high temperature liquid state metal .The flow field and temperature is varied as function of time and space. For this a complex mesh is generated for various parts of the mould like runner, riser, gates, mould, etc. From complex mesh the relevant equations are solved after discretization using tools of CFD. Such CFD calculations gives a more detailed understanding of the real flow behavior of the liquid form filling in the mould to end of solidification, and contribute to design of improved of gating system, runner, mould cavity and castings.

CIVIL ENGINEERING APPLICATIONS:

Problems involving the rheology of rivers, lakes, estuaries, etc. are also the subject of investigations using CFD .one such example is pumping of mud water from an underwater mud capture reservoir, by using CFD approach compound velocity field in both the water and mud at a required instant in time are calculated. These results contribute to the design of underwater dredging operations.

4.2 COMPUTATIONAL METHODS

Computation fluid dynamics mainly depends upon three fundamental governing equations of fluid dynamics
Mass conservation (Continuity equation)
Momentum Conservation (Navier-Stokes Equation)
Energy Conservation (Energy equation)

The various complex and interesting flows result form the solution of these equations subjected to different physical boundary conditions.

The numerical solution of these equations subjected to the boundary conditions can be obtained by the use of the following methods:

- Finite Difference Method (FDM)
- Finite Element Method (FEM)
- Finite Volume Method (FVM)

There are a variety of other methods but the three above are mostly used in the fields of CFD research. In this work, we have used the commercial software, "FLUENT" which uses the FVM. In the following paragraphs, the FVM is very briefly described.

Finite Volume method often called the Control Volume Method is formulated from the inner product of the governing differential equations with a unit function. This process results in the spatial integration of the governing equations. The integrated terms are approximated by either finite differences or finite elements discretely summed over the entire domain.

One of the most important features of the FVM is its flexibility for unstructured grids. Designation of the components of a vector normal to boundary surfaces in FVM accommodates the unstructured grid configuration with each boundary surface integral constructed between nodal points.

The principle equation in the CFD models for general viscous flow may be represented by the time averaged Navier-stokes equation.

4.3 EQUATIONS FOR VISCOUS FLOW (The Navier-Stokes Equations)

A viscous flow is one where the transport phenomena of friction, thermal conduction, and/or mass diffusion are included. These transport phenomena are dissipative—they always increase the entropy of the flow. The equations that have been derived and discussed up to this point in the present chapter apply to such a viscous flow, with the exception that mass diffusion is not included. Mass diffusion occurs when there are concentration gradients of different chemical species in the flow. An example is a non-homogeneous mixture of non-reacting

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

gases, such as the flow field associated with the injection of helium through a hole or slot into a primary stream of air. Another example is a chemically reacting gas, such as the dissociation of air that occurs in the high-temperature flow over hypersonic vehicles; in such flows, concentration gradients are induced by different rates of the flow at different pressures and temperatures. Chemically reacting flows as well as non-homogenous flows are discussed at length in Ref.2. These types of flows are not treated in the present book, simply for clarity. Our purpose here to discuss the basic aspects of CFD-we choose not to obscure the computational aspects by carrying along the extra complications and physics associated with chemically reacting flows. For this reason, diffusion is not included in the equations in this book. See Ref.2 for an in-depth discussion of chemically reacting flows and especially for a discussion of the physical and numerical effects of mass diffusion.

With the above restrictions in mind, the governing equations for an unsteady, three-dimensional, compressible, viscous flow is:

4.4 DISCRETIZATION

The discretization procedure approximate the space and time derivatives of the flow variable at each node to algebraic functions of the values of the variables at the given node and a selection of its neighbors is done through the four principal strategies that can be adopted to do this.

These are
1.The Finite Difference Method
2.The Finite Volume Method.
3.The Finite Element Method.
4.The Spectral Method.

FINITE DIFFERENCE METHOD:

This is the most direct and the most obvious of discretizing any differential equation governing a variable φ .The variation of φ is approximated by some function of distance in the coordinate directions and is fitted to the selected nodes. Here a forward differencing approximation has represented the time difference and the spatial derivatives by centered differencing schemes are possible such as upwind differencing approximation. This can be divided into two types.

a) Explicit Method: By definition in an explicit approach each differential equation contains only one unknown and therefore can be solved explicitly for this unknown in straightforward method.

b) Implicit Method:an implicit approach is one where the unknowns must be obtained by means of a simultaneous solution of difference equations applied at all grid points arried at a given time level.

Implicit methods are usually involved with the manipulation of large matrices. The implicit approach involves in more complex set of equation than explicit approach

FINITE VOLUME METHOD:

Finite volume methods are essentially a generalization of the finite difference method but use the integral form of the governing equations of flow, rather than the differential form. This gives greater flexibility in handling complex domains, as the finite volumes need not to be regular. A finite volume method on a non-staggered non-orthogonal grid, applicable to complex geometries. This approach has some problems, the main one being that the staggered grid with non-coinciding pressure and velocity grid points traditionally used by finite difference solver to avoid pressure velocity de-coupling is difficult and computationally expensive to use in complex geometries.

FINITE ELEMENT METHOD:

The finite element method achieves discretization of the governing equation in two approximation stages. First an assumption is made about the spatial variation of φ within each element by means of a ‘trial function’. This is essentially a local interpolation applied over an element, linking the solution at a point within the the element to the nodal values. The second stage uses a weighted residual method

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

SPECTRAL METHOD:

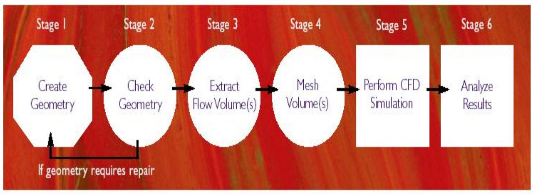
This is a particularly complex technique where the spatial variation of each dependent variable through out the system is represented by an infinite series truncated to a finite number of terms as the series applies over the whole domain the method is said to be global.

4.5 TURBULENT MODELING

The fundamental difference between laminar and turbulent flow lies in the chaotic, random behavior of the various fluid parameters such variations occur in three components of velocity, the variable that has field description. Although it is not yet possible to theoretically predict the random, irregular details of turbulent flows, it would be useful to be able to predict the time averaged flow fields Directly from the basic governing equation .to this end are can time average the governing Navier-stokes equation to obtain equation for the average velocity and pressure, however because these equations are non linear, it is not possible to merely average the basic differential equation and obtain governing equations involving only the desired average quantities ,various attempts have been made to solve this closure problem .In turbulence modeling by utilizing the CFD techniques we can solve those governing equations.

4.6 CFD FLOW:

The fundamental method for creating a CFD model can be represented as a "pipeline" of six steps as illustrated in Figure below. The process begins with the creation of an assembly of one or more parts (geometric shapes) that reflect the design under consideration. Because CFD deals with the flow through or around solid objects, the CFD model is built upon the inverse of the assembly and is referred to as the "Flow Volume Extract". This Flow Volume Extract is meshed to specification, a simulation is performed, and finally, the results of the simulation are analyzed.



Because the CFD model deals with the inverse of the solid assembly, it is important that the original geometry is created using surfaces that are connected, with faces that abut one another perfectly. Geometric models containing "cracks", "holes", or "gaps" will create problems farther down the CFD pipeline, at the meshing stage. Therefore, it is important to insert a geometry check stage into the process. This stage performs an analysis of the initial geometry to ensure that it does not contain features that will cause a failure later on. Geometric features that do not pass these checks are flagged and labeled for repair within a geometric editing environment, such as those found on popular CAD and PLM systems and in software supplied by Fluent. Fluent provides you with a full range of tools to perform successful CFD analyses and meet your individual needs and goals. For example, some user environments utilize standard PLM processes for creating, managing, and manufacturing products. In these cases, geometry creation is performed within the PLM geometry creation modules. For other cases, this stage is performed using Fluent-supplied tools, such as GAMBIT. To help understand how Fluent CFD analysis software fits into your organization, a summary of the various options is provided in this document.

4.7 INTRODUCTION TO FLUENT

FLUENT is a state-of-the-art computer program for modeling the fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility solving your flow problem with un-structured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2-D triangular/ quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also allows you to refine your grid based on the flow solution.

This solution adoptive grid capability is particularly useful for accurately predicting flow field in regions with large gradients, such as free shear layers and boundary layers. In comparison to solution on structured

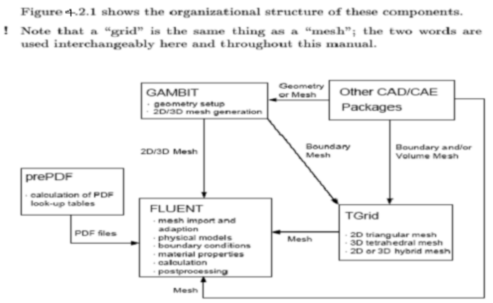
ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

or block-structured grids, this feature significantly reduces the time required to generate a good grid.

FLUENT is written in the C language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous process on client desktop workstations and powerful computer servers for efficient execution, interactive control, and complete flexibility of machine or operating system type. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The user interface is written in a language called Scheme, a dialect of LISP. The advanced user can customize and enhance the interface by writing menu macros and functions. Contact your support engineer for details.

Program Structure

Your FLUENT package includes the following products:
_ FLUENT, the solver.
_ prePDF, the preprocessor for modeling PDF combustion in FLUENT
_ GAMBIT, the preprocessor for geometry modeling and mesh generation.
_ TGrid, an additional preprocessor that can generate volume meshes from existing boundary meshes
_ Filters (translators) for import of surface and volume meshes from CAD/CAE packages such as ANSYS, I-DEAS, NASTRAN, PATRAN, and others.



Basic Program Structure

PROGRAM CAPABILITIES

The FLUENT solver has the following modeling capabilities:
-Flows in 2D or 3D geometries using unstructured solution-adaptive triangular/tetrahedral, quadrilateral/hexahedral, or mixed (hybrid) grids that include prisms (wedges) or pyramids.
_ Incompressible or compressible flows
_ Steady state or transient analysis
_ inviscid, laminar, and turbulent flows
_ Newtonian or non-Newtonian flow
_ Convective heat transfer, including natural or forced convection
_ coupled conduction/convective heat transfer
_ Radiation heat transfer
_ Inertial (stationary) or non-inertial (rotating) reference frame models
_ Multiple moving reference frames, including sliding mesh interfaces and mixing planes for rotor/stator interaction modeling
_ Chemical species mixing and reaction, including combustion sub-models and surface deposition reaction models
_ arbitrary volumetric sources of heat, mass, momentum, turbulence, and chemical species
_ Lagrangian trajectory calculations for a dispersed phase of particles/droplets/bubbles, including coupling with the continuous phase
_ Flow through porous media
_ One-dimensional fan/heat-exchanger performance models
_ Two-phase flows, including cavitations
_ Free-surface flows with complex surface shapes.

These capabilities allow FLUENT to be used for a wide variety of applications, including the following:
_ Process and process equipment applications
_ Power generation and oil/gas and environmental applications
_ Aerospace and turbo machinery applications
_ Automobile applications
_ Heat exchanger applications
_ Electronics/HVAC/appliances
_ Materials processing applications

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

_ Architectural design and _re research

In summary, FLUENT is ideally suited for incompressible and compressible fluid flow simulations in complex geometries. Fluent Inc. also other solvers that address different flow regimes and incorporate alternative physical models.

4.10 OVERVIEW OF USING FLUENT

FLUENT uses unstructured meshes in order to reduce the amount of time you spend generating meshes, simplify the geometry modeling and mesh generation process, model more-complex geometries than you can handle with conventional, multi-block structured meshes, and let you adapt the mesh to resolve the flow-field features. FLUENT can also use body-fitted, block-structured meshes (e.g., those used by FLUENT 4 and many other CFD solvers). FLUENT is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. This exibility allows you to pick mesh topologies that are best suited for your particular application.

You can adapt all types of meshes in FLUENT in order to resolve large gradients in the flow field, but you must always generate the initial mesh (whatever the element types used) outside of the solver, using GAMBIT, TGrid, or one of the CAD systems for which mesh import filters exist.

4.8 PLANNING YOUR CFD ANALYSIS

When you are planning to solve a problem using FLUENT, you should first give consideration to the following issues:
Definition of the Modeling Goals: What specific results are required from the CFD model and how will they be used? What degree of accuracy is required from the model?
Choice of the Computational Model: How will you isolate a piece of the complete physical system to be modeled? Where will the computational domain begin and end? What boundary conditions will be used at the boundaries of the model? Can the problem be modeled in two dimensions or is a three-dimensional model required? What type of grid topology is best suited for this problem?
Choice of Physical Models: Is the flow inviscid, laminar, or turbulent? Is the flow unsteady or steady? Is heat transfer important? Will you treat the fluid as incompressible or compressible? Are there other physical models that should be applied?
Determination of the Solution Procedure: Can the problem be solved simply, using the default solver formulation and solution parameters? Can convergence be accelerated with a more judicious solution procedure? Will the problem fit within the memory constraints of your computer, including the use of multigrid? How long will the problem take to converge on your computer?
Careful consideration of these issues before beginning your CFD analysis will contribute significantly to the success of your modeling effort. When you are planning a CFD project, take advantage of the customer support provided to all FLUENT users.

4.9 PROBLEM SOLVING STEPS

Once you have determined the important features of the problem you want to solve, you will follow the basic procedural steps shown below.
1. Create the model geometry and grid.
2. Start the appropriate solver for 2D or 3D modeling.
3. Import the grid.
4. Check the grid.
5. Select the solver formulation.
6. Choose the basic equations to be solved: laminar or turbulent (or inviscid), chemical species or reaction, heat transfer models, etc.Identify additional models needed: fans, heat exchangers, porous media, etc.
7. Specify material properties.
8. Specify the boundary conditions.
9. Adjust the solution control parameters.
10. Initialize the flow field.
11. Calculate a solution.
12. Examine the results.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

13. Save the results.
14. If necessary, refine the grid or consider revisions to the numerical or physical model.

Step 1 of the solution process requires a geometry modeler and grid generator. You can use GAMBIT or a separate CAD system for geometry modeling and grid generation. You can also use TGrid to generate volume grids from surface grids imported from GAMBIT or a CAD package. Alternatively, you can use supported CAD packages to generate volume grids for import into TGrid or into. For more information on creating geometry and generating grids using each of these programs, please refer to their respective manuals. In Step 2, you will start the 2D or 3D solver.

5. RESULTS AND DISCUSSION

5.1. Cylinder pressure and rate of pressure rise results

Figure 2 shows modeling and experimental incylinder pressure traces operating at full load condition. The modeled cylinder pressure data shows good agreement with experimental results. The maximum pressure rise depends upon the quantity of fuel vaporized during the delay time and occurs in the state of combustion, some degrees after the beginning of combustion. Note that modeling peak pressure is 66.16 bar at 366 degree CA, and experimental peak pressure is 63.55 bar at 366 degree CA. Therefore both scale and timing of occurrence of peak pressure are precisely predicted by the model. Figure 3 shows modeling and experimental rate of cylinder pressure rise which is calculated by first order derivative of cylinder pressure with crank angle. The rate of pressure rise trace reconfirms the peak pressure history obtained from pressure and crank angle diagram.

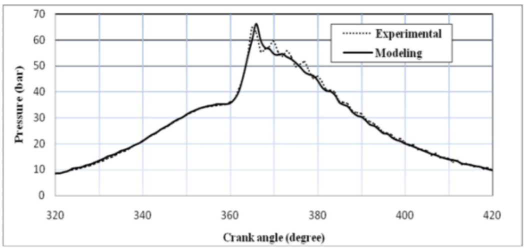


Figure 2: comparisons between modeling and experimental pressure diagram.

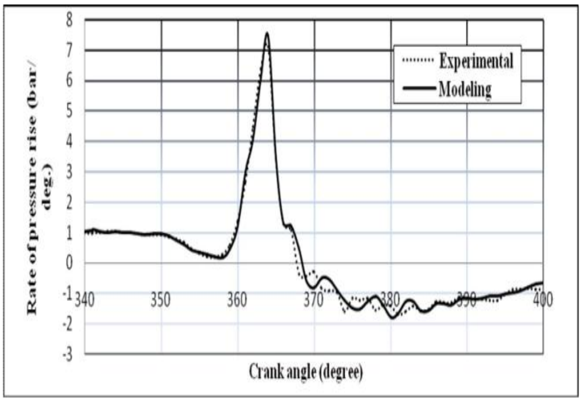


Figure 3: Comparison between modeling and experimental rate of pressure rise diagram.

5.2. Heat release rate results

The observed cylinder pressure profiles reflect the effect of incylinder heat release rate. The heat release rate is determined from pressure data. Figure 4 compares heat release rates computed from modeling and experimental pressure traces. The heat release rate decreases from the start of injection to the start of combustion which is ignition delay period because of the fuel evaporation occurring during this period. The first peak due to premixed combustion strongly depends on the amount of fuel that the prepared for combustion during the ignition delay period. The second peak due to diffusion combustion is controlled by the fuelair mixing rate. Diffusion combustion continues until combustion is completed. Note that, the peak modeling heat release rate is 79.01 J/ degree where as experimental peak heat release rate is 77.34 J/ degree at 364 degree CA.

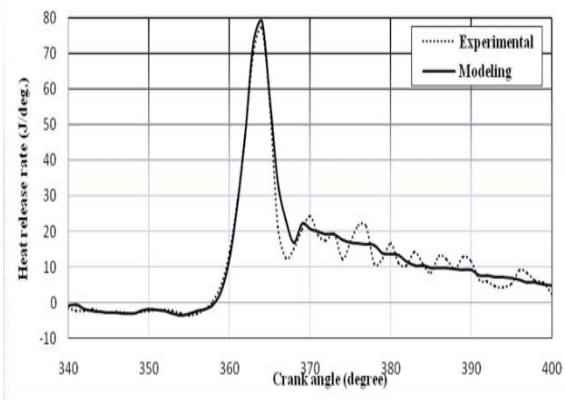


Figure : Comparison between modeling and experimental heat release rate diagram.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

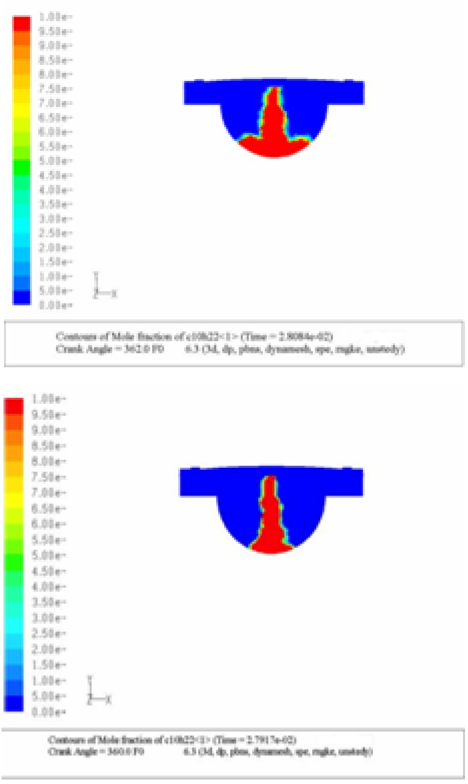


Figure 5(a,b), illustrates the instant for counters of mole fraction diesel fuel at 3600 and 3620 crank angle respectively.

It can be seen that nearly all the liquid fuel has vapourized prior to the onset of combustion. Note that the mixture conditions are same before ignition occours since the same turbulence and spray model pridicts precise results

CONCLUSIONS

The HCCI combustion engines have the potential to reduce the NOx and PM emissions simultaneously, while maintaining the thermal efficiency close to that of conventional diesel engine.

The CFD code FLUENT has been used to simulate the combustion characteristics of direct injection diesel engine. The model also integrated with submodels includes, spray, droplet collision, wall film and combustion model with species transport and finite rate chemistry. The bowlinpiston combustion geometry was used for model construction. In this study RNG k ε model is implemented to confine incylinder turbulence. Simulated results including the incylinder pressure, rate of pressure rise and heat release rate profiles have been analyzed. A good agreement between the modeling and experimental data ensures the accuracy of the numerical predictions collected with this work. Including peak values of in cylinder pressure, rate of pressure rise and heat release rate are shown good agreement between modeling and measured data. The comparison revels that the present model manages to predict the combustion characteristics quite well. The results reported in this paper illustrate that the numerical simulation can be one of the most powerful and beneficial tool to compute the essential features of combustion parameters for ICE development, optimization and performance analysis.

REFERENCES

1. Abani N., Munnanur A., Reitz R. D., 2008, “Reduction of numerical parameter dependencies in diesel spray models”. Journal of engineering for Gas Turbines and Power, ASME, vol.130, 032809-1- 032809-9.
2. Bianchi G.M., Pelloni P., Corcione F.E., Allocca L., Luppino F., 2001,”Modeling Atomization of high pressure diesel sprays”, Journal of engineering for Gas Turbines and Power, ASME, vol.123,pp 419-427.
3. Djavareshkirn M. H. and Ghasemi A., 2009, “ Invistigatation of jet break-up process in diesel engine spray modeling”, Journal of Applied Sciences, Vol. 9, no.11, pp 2078-2087.
4. Hergart C., Barths H. and Peters N., 1999, “Modeling the combustion in a small-bore diesel engine using a method based on Representative Interactive Flamelets ”. SAE, vol.1, pp 45-55.
5. Hountalas D.T., Kouremenos D.A., Mavropoulos G.C., Binder K.B. and Schwarz V., 2004, “Multi-zone combustion modeling as a tool for DI diesel engine development-Application for the effect of injection pressure”. SAE, vol.1, 2004-01-0115.
6. Kadocsa A., Tatschl R. and Kristof G., 2007, “

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Analysis of spray evolution in internal combustion engines using numerical simulation". Journal of Computational and Applied Mechanics, Vol. no. 8, pp 85-100.
7. Kolade B., Thomas M., Kong S.C., 2004, "Coupled 1-D/3-D analysis of fuel injection and diesel engine combustion", SAE International, vol.1, 1-10.

Author Details

Soubhagya Ranjan Das
M.Tech (Thermal Engineering) Student
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali.

B.V.V.Prasada Rao is working as asst Prof In Department Of Mechanical Engineering, Aditya Institute of Technology & Management, Tekkali, AP