International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 14
October2016

The CFD Analysis of CI Engine Combustion Characteristics

1. Mohammad Yousuf Baba*, 2. Dr.M.Janardhan **, 3. Mr.G.Ramesh babu ***

*M.Tech Student ** professor& Principal *** Associate Professor

Department of Mechanical Engg. Abdul Kalam Institute Of Technological Sciences, Kothagudem,Khammam (DT) -507120

ABSTARCT:

Computational fluid dynamics code (CFD) Fluidity used to model the compound The phenomenon of combustion pressure Engine ignition. Temperature profile and nitrogen oxides Produced within the combustion chamber Compared with traditional jatropha Diesel. The simulation results obtained Experimentally verified for both jatropha and test conventional diesel diesel engines. Simulation is fluid in use Ansys- Mixture combustion model is to respond to the real needs The parameters in the combustion cylinder. Two Combustion chamber dimensions network simulation distortion is used.

INTRODUCTION:

Since the beginning of civilization, it has provided the combustion of fuels for most of our energy needs. Tgreater still providing 70% [3] of its current capacity of the planet. Therefore, combustion technology remains the main energy in the foreseeable future. The combustion phenomenon is the conversion of the energy trapped in various types of chemically fuel heat model (and light). This energy can be used to increase the generation of steam and electricity, Tricity, heating, transport and cooking area, and many other applications. It can happen fuel used in domestic and industrial equipment combustion in any of the three phases that occur naturally (solids, liquids and gases). This fuel must react with oxygen, which occur in gaseous form. But also it must be converted into gaseous form before undergoing combustion reactions. This conversion occurs through evaporation of the liquid fuel or solid fuel outgassing at high temperatures.

At the molecular level, which can be of two reagents undergo a change in electronic form to form or break the links leading to a chemical reaction. They have to be thoroughly mixed and Esna implementation cient combustion. Therefore bringing the two reactants, namely. Fuel and oxygen, in proximity to each other at the molecular level, forming part of the challenge for the design of any combustion equipment. Research hill experience provides useful data that can be used in the design of this equipment in trying to theoretical developments to explain the experimental behavior BE- observed. Numerical simulations give predictability needed for the design of combustion systems. And the use of experimental results and theoretical and proposals on the development of numerical simulation successfully. Most fuels used in combustion applications are a mixture of several chemical types. Each of these species reacts with oxygen to release its own heat reemployment. Such reactions do not occur, such as the process from step one, but make up many of the initial steps involving many intermediate species. All these steps intermediate species is necessary to understand the behavior of fuel combustion. Several decades of research have achieved mathematical and numerical method .For treatment of the complex nature of chemical reactions. These include the disarmament of the kinetic model of chemical glue, accounts balance and solving equations of balance of species in the simple flow geometries.

The nature of fluid flow in a variety of combustion systems, ranging from laminar to turbulent some cases in most industrial equipment. The fluid flow may have additional properties, such as compressible, swirling, and is stable with respect to time or any of the groups from above. Ff flow characteristics ect shape and extent of the fuel particles and oxidant contact one another. Therefore, in addition to know the details of the chemistry and behavior of operations problems, it is also necessary to have the descriptive approach to simulate the interactions between the two countries (viz. The chemical interaction agitation). combustion reactions often result in many contaminants such as soot, nitrogen oxides and sulfur oxides. an accurate prediction of these contaminants that all details of the chemical reactions are inserted into the flow simulation is required. This becomes a turbulent flow cases integrals. And it can be used to simulate the response of simplistic give good results with respect to the release of heat patterns approach and flow but not grasp the ignition delay and the formation of pollutants time details. Along similar lines, can be complex shapes (such as Monte Carlo) modeling turbulent flow is also to be exhaustive. There are several methods for modeling the precise details of agitation alone, and chemical reactions, and there alone, and the development of a simplified plan the previous days more complex and intensive studies numerically latter approach. He described some of them in the following chapters. However, its design and following thoughtful and optimization in the industry today, and the balance of accuracy is required for an account. Approach that focuses on a single detail of this phenomenon can be misleading or does not serve the general purpose of simulation studies

LITERATURE REVIEW:

The Process work in a simulation of the combustion process in diesel engines compression ignition (CI) using low ILDM mechanism automatically using technology in the three-dimensional model CFD. While the conditions in the diesel engine online characterized by high pressure and temperature, and a similar approach can be used to simulate the combustion process it is similar to mixing systems.

The compression ignition engines of the diesel engine, in which inspirited only air in the cylinder, unlike spark ignition engines, where air and fuel mixture is inspir-ited. The fuel is injected directly into the combustion chamber in the form of a mist of liquid. combustion chamber of the diesel engine is composed of a cylindrical piston. And inspirited air in the intake stroke (at the beginning of the session) in the case of at-atmospheric (more if the engine is turbocharged) when the piston is at bottom dead center, the intake valve is open. In the compression stroke, the piston compresses the air trapped inside the room, and raise it to a high temperature, and finally to the top dead center (TDC). The compression ratio in spark-ignition engines are in the range of 12 to 24, much higher than in SI engines. Compressed air can be up to 80-100 atm. The temperature can be up to 800 to 1000 K at this point. Shortly before the piston reaches the body, liquid fuel sprayed into the combustion chamber. Due to the high temperature (greater ignition point fuel), fuel ignition engine. Time from the start of injection (SOI) and ignition and the occurrence of so-called time delay ignition. In the expansion stroke, in the light of the flame propagation through the combustion chamber, the heat is released from the fuel. (BDC), and the hot exhaust gases are pressed towards the extended piston BDC. Therefore, the energy is converted into mechanical motion of the piston, which has also resulted in the crankshaft and used to transport in a car or electricity generation etc. integrity arrival combustion piston BDC, reflecting the start address of the exhaust stroke. The release of the exhaust valves open and the flue gases. With this the cycle is completed and start fondling the following entry in the final part of the session.

There are many aspects to be considered for CFD simulation in a diesel engine, which are listed below.

• combustion occurs at high pressures and flow can not be assumed that the tablet.

• simulation requires a base turbulent flow in three dimensions.

• heat transfer due to radiation, convection and conduction to be considered.

• model require liquid spray droplets for mass and heat transfer with the continuous phase, as well as breaking and coalescence.

• suitable for predicting the weather so self-ignition.

• models and polluting chemical and abundant species.

• transmission networks to simulate the movement of the piston.

Each of the above considerations suggest that these simulations are, however, a complicated-com useful exercise.

The presence of the species in the flow interaction requires the inclusion of additional points in the equations change. This is because the terms of the source of all kinds and the energy released due to a reaction. And it is subject to changes in the concentration of species and the heat released due to a reaction rate kinetic equations. Turbulent flows in the reaction, these conditions sources pose a problem because they are not in a closed form. A closure method where the source can be integrated by using a statistical approach, where the formulation of the probability density function (PDF) on the terms of any source. This method is chosen for the job. If a large number of species present in the flow, and this approach can be costly in terms of numbers. Therefore technique, described as ILDM (essentially low dimensional manifold), to reduce the reaction system offers variable representation can be used. A mathematical method for the separation of the reactions in the fast and slow reactive groups on the basis of analysis of making the value of the Jacobi matrix in terms of the source.

METHODOLOGY:

The experiment was performed in a single cylinder four-stroke diesel engine air cooled vertical 5.9 kW nominal power and nominal minute 1500 RPM speed. Precise measurements of the degree of exhaust heat, The rate of fuel flow. emissions biodiesel And measured using diesel gas analyzer. speed continuous pregnancy test has been performed on Motor and previous measurements Taken under different loads. The experiment showed that the above scheme. Establish airbox has a size of about 500 Times the size of the engine cylinder. The idea is To make the free flow of air from any pulse full engine axis AC generator load applications .Electrical Using the engine alternator. Temperature and It is measured by using the exhaust gas emissions Thermal and gas analyzer, respectively. While performing the experiment conditions of the room It remained constant and adequate ventilation He offered to make the surrounding free Exhaust emissions, which may affect emissions Read.

RESULTS:

The results of any simulation is only as good as some input for modeling code. Compared with experimental data can reveal the accuracy and validity of the results. A set of experimental data for the engine 3406 Caterpillar heavy trucks available [159] for this purpose. It is given this engine specifications in the following section. The amounts include pressure measurement average and the average temperature, shutter speed, and the concentrations of nitrogen oxides and soot. Additional qualitative description of the engine can be performed by analyzing propagation of flame, the flame shape, and the numerical rates dissipate sites ignition and ignition delay.

In any CFD simulation, it is very important to ensure that the results do not depend on the network used. This can be achieved by using more consistent, respectively density, and stood on the step and then no change was found in the results. And the use of networks three di ff Erent density in this study, and found suitable for medium dense networks accounts. A brief description in the next section. E ff ect of the many parameters studied is clear from the following addresses sections.

Engine Specifications and network

54 kW simulator diesel engine at 2,100 rpm (revolutions per minute) were classified. And considering the design of the initial conditions in Tables 7.1 and 7.1 for details. Available for download partial cycle with the condition in 1600. minute stays on all the walls under constant temperature data experi mental. Measuring air temperature error of about 10 to 15 K input, which is adjusted to suit the temperature in the injection simulation experience in a state of cold flow. Ho stirring time mogeneous

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 14
October2016

closing the intake valve, which occur in the -137o ATDC (after top dead center) is assumed. the exhaust valve opens 130o ATDC, and not only perform simulations for these events between the two valves. The initial values of the parameters of his turbulence calculated based on the recommended level of turbulence by 10%. And the value of ọ is calculated as the value of freedom of movement and a length of 1.432 cm. The results do not pay attention to these values as previously reported [33].

Use 1/6 the sector to take advantage of the symmetry league, which was established as a result of injector nozzles 6 as it reduces the time required for the account. Numeric

Bore	137.16 mm
Stroke	165.1 mm
Connecting Rod Length	263 mm
Engine Speed	1600 rpm
Number of Nozzle Orifices	6
Injection Timing	-7°, -4°, -1°, 2°, 5° ATDC
Duration of Injection	19.75°
Fuel Injected	0.168 g/cycle
Cylinder Wall Temperature	433 K
Piston Wall Temperature	553 K
Head Temperature	523 K
Initial Gas Temperature	361±15 K
Spray Temperature	341 K

Table 7.1: Caterpillar-3401 engine specifications.

Initial Gas Composition (g/cm³)			
O_2	N_2	CO_2	H_2O
4.6012×10^{-4}	1.5337×10^{-3}	2.8579×10^{-6}	1.2579×10^{-6}

Table 7.2: Engine conditions at the time of inlet valve closure.

CONCLUSION:

Simulating turbulent flows is a difficult problem. Having reactions adds more complexity to this challenge. The ability to achieve such a complex within the limits of computing resources and lifetime challenge are industrially reasonable results it is the main objective of this work. Results and discussion in the previous chapter leads to the conclusion that ILDM provides reasonable to integrate kinetic rate laws for the complex interaction mechanism in the probabilistic description of turbulent flows form. Choice of reaction variables may be slightly ff ect accuracy of the results. model based on the results raised to fully automatic container (including low temperature reactions HC) gives delay times similar to the appointed ignition temperature formations release time measured experimentally. Intro-duction of RNG model as ọ wall and enhance heat transfer equation translates into better heat transfer expectations and consistent with literature disorder. It can also be concluded that the Zeldovich mechanism for predicting nitrogen oxides when used in concentrations of reactive intermediate species which have been obtained from ILDM can improve the accuracy of the accounts for approximately 10 to 20%. Similarly, the expectations of soot, using the model of phenomena including ILDM soot precursors, can be obtained within 20 to 30% of those obtained experimentally. The formation of nitrogen oxides in the inclination equally flame in contrast to soot, tops in emerging areas of the flame. Soot increases rapidly after injection, and I saw a little peak after peak fuel focus seems to indicate that the forms of soot, especially in areas rich flame. In all temperature characteristics, there can be observed a deviation of laboratory values before the start of injection. Initially it was thought that this could be due to the assumption that the ideal gas law. The implementation of the best equation of state (Radash-Kwong or Soave) could not fill this gap, and can not solve the problem. a reassessment of the relationship of pressure and heat is required to address this issue. In short, all important parameters to know. temperature and pressure and heat release rate as well as nitrogen oxides and soot contaminants satisfactorily predicted. Despite his comment on the numerical ILDM remain outside the scope of this Workplaces and public observations about the functions can be done. And to allow simultaneous use of remote zero (balance), in addition to regular programming go a long way to address the question of where is the source where there ILDM points. The need to use the one-step reaction or a point shape and inaccuracies that come with it, then it can be ignored. There are several reasons for this, but the lack of correspondence between the experimental data and attributed the results of the simulation, and. Assuming a uniform ILDM for use with all cases is one pressure. When a one-step model is used when ILDM countries can not find it, it is done NO and soot accounts using only PDF degrees of temperature. Moreover, the concentrations of intermediate species and obtain the cells are not added to each, but only used for accounts NO and soot. This is to avoid errors in the overall balance to calculate the chemical in one step.

Despite being a good Affordable way FF it is more valuable than a simple demonstration purpose, integrating ILDM to simulate the turbulent flow of the reaction is still plagued by many di culties ISNA dispute, and the robustness of the method is LG more doubtful. The results can only be obtained with many of the numerical limitations. One such limitation is the assumption of the probability distributions of statistical independence T, ξ and directed from a distance. Standardization and analysis of the time scale used to address this assumption induces the numerical weakness in the code element Simo-. It may be the future work includes the implementation of a way to remove this assumption common PDF files directly calculated. Remains in the account in terms of ff-mail ORT Compu tational for a procedure of this kind to be seen. However, one should consider the fact that this work is limited to the application in an unstable state of the state where the difference in the mixture with a part-time can be significant. Therefore, at any given time, and is only a certain range of numerical calculation field ILDM.

Systems involving a steady stream of state, for example, process heater, do not face this problem, with the exception of a small area near the areas where the fuel and air mixture. In such a case, and reportedly more accurate and durability can not be achieved. However, in its current form, together with the code KIVA ILDM is a good way to simulate combustion processes in internal combustion engines and forecasting of nitrogen oxides and soot. It can investigate further analysis using the compression ratio and speed of numerical dissipation to link the three PDF files (supposed) independence.

REFERENCES:

[1] I. Mayeres, S. Proost. Why do we still have Diesel cars in Europe? In EAERE 2000 Annual Conference, 30 June - 2 July 2000, pp 1–22, University of Crete, Department of Economics, Rethymnon, Greece,. 2000.

[2] C. Hibbs. Diesel auto sales trending to exceed gasoline in Eu-rope in 2006. PriceWaterhouseCoopers Report, available at http://www.greencarcongress.com/2006/01/diesel_auto_sale.html, pp 14–18, 2004.

[3] N. Blackaby. Once bitten, twice shy. Power Engineering International **8**, pp 20–22, 2005.

[4] N. Peters. Turbulent combustion. Cambridge University Press, New York, NY, 2000.

[5] R. Klein. Numerics in combustion. Von Karman Institute, Belgium, 1999.

[6] R.B. Bird, W.E. Stewart, E.N. Lightfoot. Transport phenomenon. John Wiley & Sons, New York, 1960.

[7] F.A. Williams. Combustion theory. Addison-Wesley, New York, 1985.

[8] M.W. Chase Jr. (Ed.). NIST-JANAF thermochemical tables, 4th ed. National Institute of Standards and Technology, Boulder, CO, USA, 1998.

[9] C. Correa. Combustion simulations in Diesel engine with reduced reaction mechanisms. Phd thesis, University of Heidelberg, 2000.

[10] D. Veynante, L. Vervisch. Turbulent combustion modeling. Von Karman In-stitute, Belgium, 1999.

[11] P.A. Libby, F.A. William. Turbulent reacting flows. Springer Verlag, Berlin-Heidelberg, Germany, 1980.

[12] S.B. Pope. Turbulent flows. Cambridge University Press, Cambridge, UK, 2000.