Geometric Modeling and Fluid Flow Simulation of IC Engine- A Review

Kalpit Patel
(19BME057)
Pandit Deendayal Energy University,
Gandhinagar, Gujarat
kalpit.pmc19@sot.pdpu.ac.in

Abstract

The internal combustion engine (IC engine) is a complex mechanical system that plays a vital role in many industries. In order to design and optimize IC engines, accurate modeling and simulation of the engine's geometry and fluid flow are essential. This review paper provides an overview of the latest research in the field of geometric modeling and fluid flow simulation of IC engines. Firstly, the different techniques used for geometric modeling of IC engines are discussed, including the advantages and disadvantages of each approach. The impact of geometric modeling on the accuracy of fluid flow simulations is also examined. Next, the various approaches to simulating fluid flow in IC engines are reviewed, including the use of different turbulence models and boundary conditions. The challenges involved in accurately modeling and simulating IC engines are highlighted, such as the complex geometry and the presence of multiple phases. Furthermore, this review paper discusses the potential future research directions in this field, including the use of artificial intelligence and machine learning algorithms to improve the accuracy of simulations, and the incorporation of real-world driving conditions in simulations. In conclusion, this review paper provides a comprehensive overview of the latest research in the field of geometric modeling and fluid flow simulation of IC engines, highlighting the challenges and potential future directions in this important area of research.

Keywords
ICE, Fluid flow simulation, Engine, Ansys, CFD.

1. Introduction
Geometric modeling and fluid flow simulation are important aspects of internal combustion engine (IC) design and optimization. Geometric modeling involves creating a digital representation of the engine's internal components, such as the cylinders, pistons, and valves, using Solidworks software. This allows for accurate visualization and analysis of the engine's geometry.

Fluid flow simulation, on the other hand, involves using Ansys software to simulate the flow of fluids, such as air and fuel, through the engine's internal components. This allows for the prediction of important parameters such as fuel consumption, power output, and emissions.

In this review paper, we will explore the latest research in the field of geometric modeling and fluid flow simulation of IC engines. We will discuss the different techniques used for geometric modeling and how they can impact the accuracy of fluid flow simulations. We will also examine the various approaches to simulating fluid flow in IC engines, including the use of different turbulence models and boundary conditions.

Furthermore, we will discuss the challenges involved in accurately modeling and simulating IC engines, such as the complex geometry and the presence of multiple phases (such as liquid fuel and gaseous air). Finally, we will highlight the potential future research directions in this field and their potential impact on IC engine design and optimization.

2. Literature Review
2.1 Combustion in a Spark Ignited Engine

An IC engine is modelled and simulated in three dimensions using computational fluid dynamics. By installing the Producer gas carburetor and switching the fuel injector out for a spark igniter, the diesel (CI) engine is transformed into a SI engine. Usually in diesel engines, the heat produced during compression and the fuel injection is sufficient to start the combustion process without the aid of an external spark (Figure 1 and Figure 2).

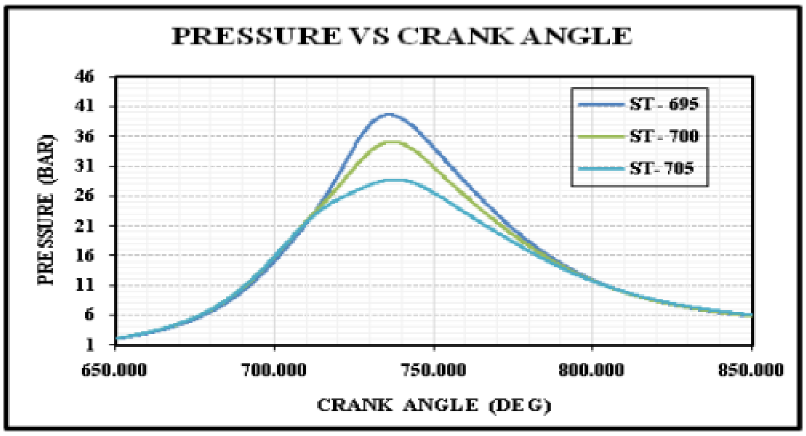

Figure 1. Pressure V/s Crank Angle for different spark time

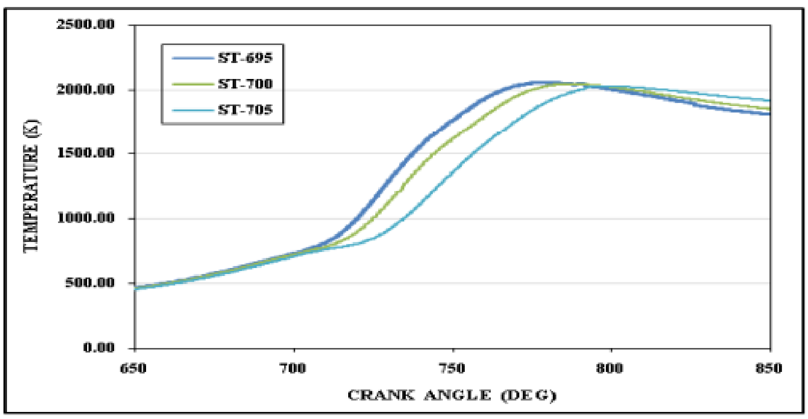

Figure 2. Temperature V/s Crank Angle for different Spark time

The research study models a three-dimensional internal combustion (IC) engine powered by producer gas using the software versions 16.0 of ANSYS Fluent and CATIA V5. To account for the effects of turbulence inside the engine cylinder, the simulation is run using the conventional k-model. Zimont's model is utilised to model the turbulent flame speed, and the premixed C-equation model is used to assess heat in the engine cylinder. A simulation is run using the initial circumstances generated from the experimental investigations after an experimental study is conducted on a 4-cylinder engine under cooled conditions. A variation of as little as 2.23% & 1.52% between the driving peak pressure and temperature from the simulation and that from the theoretical calculation verifies the simulation. The characteristics such as cylinder pressure, temperature, and mean effective pressures rise as the spark time advances; therefore, with a spark time (CA) of 695, the brake power obtained is 5.62 KW.

2.2 Analyze the aerodynamics of the designed manifold geometry

The analysis of the aerodynamics of a designed manifold geometry is an important aspect of engineering design for many applications, such as in internal combustion engines and fluid flow systems. The performance of a manifold is greatly affected by its geometry, as it influences the flow of fluids through the system. A guide vane with the strongest curvature is found to provide a considerable improvement over the baseline situation when the potential for design improvement via guide vanes is studied computationally. The manifold will be manufactured and experimentally investigated in the upcoming work, which will serve as a foundation for validation and additional design advancements.

2.3 Application of CFD (computational fluid dynamics)

CFD is widely used in the analysis and design of internal combustion engines (IC engines). It helps in understanding the complex fluid dynamics within the engine, optimizing the engine performance, and reducing emissions. Some of the common applications of CFD in IC engines are (Figure 3):

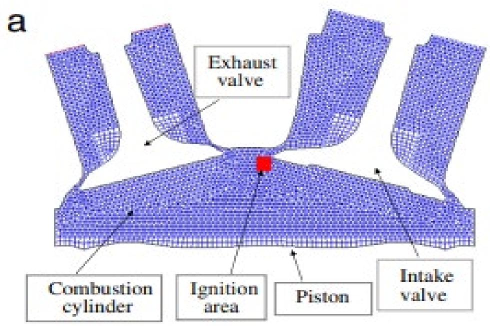

Figure 3. The dynamic mesh of the SI model

- Combustion Analysis: CFD is used to analyze the combustion process within the engine. It helps in predicting the heat release rate, temperature distribution, and pollutant formation. CFD analysis can also be used to optimize the fuel injection strategy and combustion chamber design for improved performance and reduced emissions.
- Intake and Exhaust System Design: CFD is used to examine how air and exhaust gases move through the engine's intake and exhaust systems. For better performance and lower emissions, it aids in improving the design of the intake and exhaust valves, ports, and manifolds.
- Piston and Cylinder Analysis: The air and fuel mixture flow around the piston and cylinder of the engine is examined using CFD. It aids in improving the combustion efficiency and cylinder and piston design for lower emissions.

2.4 LNG Technologies

Liquefied natural gas (LNG) is a useful fuel for internal combustion engines (ICE) due to its several advantages over traditional fossil fuels.

Firstly, LNG has a higher energy density than compressed natural gas (CNG), which is commonly used in natural gas engines. This means that a higher amount of energy can be stored in a smaller volume, allowing for increased vehicle range.

Secondly, LNG burns cleaner than diesel fuel, emitting significantly lower levels of particulate matter, nitrogen oxides, and sulfur oxides. This results in improved air quality and reduced greenhouse gas emissions.

Thirdly, LNG is more cost-effective than diesel fuel, providing cost savings to operators of LNG-fueled vehicles. LNG production and distribution infrastructure is also becoming more widespread, making it easier to access and utilize.
Finally, LNG-fueled engines have been shown to have lower maintenance costs and longer engine life due to the cleaner burning fuel and reduced engine wear and tear.

Overall, the use of LNG as a fuel in internal combustion engines provides a cleaner and more cost-effective alternative to traditional fossil fuels. As more LNG fueling infrastructure becomes available and advancements in engine technology are made, the use of LNG in IC engines is expected to continue to grow.

2.5 Effect of fuel composition on the ignition

Methane and ethane in air at different fuel compositions, flow rates, and initial temperatures. They measured the ignition and extinction limits of the flames and analyzed the data to determine the effect of fuel composition on these parameters.

The addition of ethane to methane increased the laminar flame speed and decreased the ignition delay time. The flames were found to exhibit different modes of extinction depending on the fuel composition, with the ethane-rich flames extinguishing in a more diffuse manner than the methane-rich flames.

The fuel composition has a significant impact on the ignition and extinction characteristics of laminar diffusion flames, and these findings could have important implications for the design and optimization of combustion systems. Specifically, the study suggests that the addition of ethane to methane can improve the combustion efficiency and stability of laminar diffusion flames in practical applications. Overall, the research contributes to a better understanding of the fundamental mechanisms underlying combustion and has potential practical applications in various fields, such as energy production, transportation, and environmental engineering.

2.6 Combustion Engine Flow

A three-dimensional numerical model to simulate the flow inside a motored engine using both RANS and LES approaches. The results showed that LES was able to capture more complex flow structures and turbulence dynamics compared to RANS. LES also provided more accurate predictions of in-cylinder pressure and heat release rates.

The investigation the effect of different subgrid scale models on the accuracy of LES predictions and found that the dynamic Smagorinsky model performed best for capturing the complex flow features of the internal combustion engine.

LES is a more accurate approach for predicting the in-cylinder flow of internal combustion engines compared to RANS. They suggested that future research should focus on improving the computational efficiency of LES and developing more accurate subgrid scale models to make LES more practical for engine design and optimization.

Overall, this research paper presents a comprehensive comparison of RANS and LES approaches for simulating internal combustion engine flows. The paper is well-written, and the results provide valuable insights into the strengths and limitations of these two approaches. The findings have important implications for the design and optimization of internal combustion engines, making the paper a valuable resource for researchers and practitioners in the field.

2.7 Dual Fuel Diesel Engine

According to the research of this study, the features of each proportion of LNG and diesel utilization can be used as parameters to gauge how well dual-fuel diesel engines work. The analysis was made utilizing software simulation and empirical formulae from the literature (Figure 4 and Figure 5).

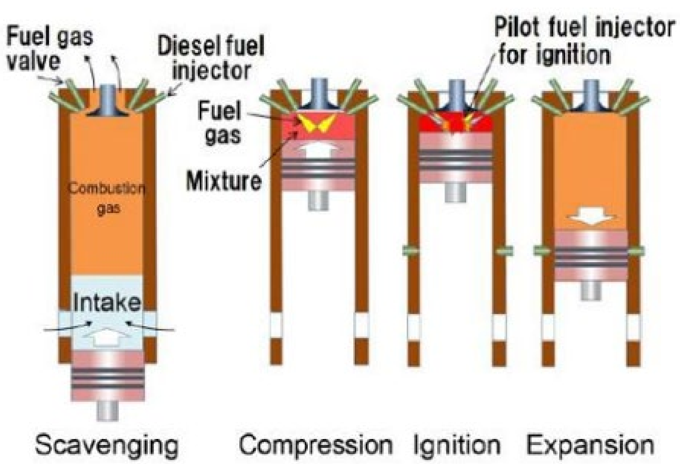

Figure 4. Step work Dual-fuel Diesel Engine

The Following are results drawn from simulation and analysis of this research.
i. The supplied variation value designates 30%, 40%, 50%, 60%, and 70% influence different performance changes in each variation due to the varying fuel grades of LNG and diesel.

ii. The value of power, torque, and mean effective pressure as determined by the performance outcomes from each change in fuel consumption depends on the highest pressure that the LNG-diesel mixture can generate during the compression phase.
iii. Based on the SFC graph and thermal efficiency, the increase of LNG component to this mixture led to lower SFC values and greater thermal efficiency.
iv. While adding LNG to the LNG-diesel mixture greatly improves engine performance, LNG's lower heating value is greater than diesel's lower heating value.

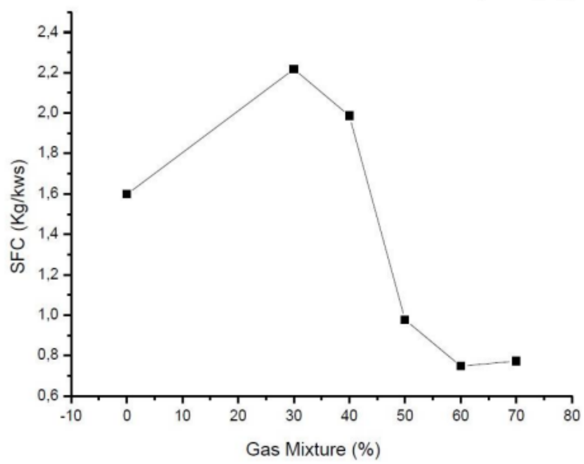

Figure 5. Specific fuel usage under the dual fuel scenario

3.Conclusion

In conclusion, geometric modeling and fluid flow simulation of IC engines have become increasingly important in the design and optimization of IC engine . Advances in computational fluid dynamics (CFD) have allowed for accurate predictions of flow phenomena in engines, leading to improved performance, emissions, and efficiency. The use of CFD has also reduced the need for expensive and time-consuming experimental testing. The studies reviewed in this paper demonstrate the diverse range of applications of geometric modeling and fluid flow simulation in IC engine design, including optimization of intake and exhaust systems, fuel injection systems, and combustion processes. However, while CFD has the potential to provide valuable insights into engine performance, it should be noted that accurate simulation results are dependent on proper modeling and simulation techniques. Further research is necessary to improve the accuracy of CFD simulations and to develop more sophisticated models that take into account complex phenomena such as turbulence, combustion, and emissions. Overall, geometric modeling and fluid flow simulation have emerged as powerful tools for IC engine design and optimization, with significant potential for future advancements in the field.

References

Braga, L.B., Silveira, J.L., da Silva, M.E., Machin, E.B., Pedroso, D.T. and Tuna, C.E., Comparative analysis between a PEM fuel cell and an internal combustion engine driving an electricity generator: Technical, economical and ecological aspects. *Applied Thermal Engineering*, *63*(1), pp.354-361. 2014

Cebula, A., Świątek, P. and Kowalczyk, S., A numerical analysis of the air flow through the IC engine intake manifold. *Progress in Computational Fluid Dynamics, an International Journal*, *19*(1), pp.63-68. 2019.

Dinler, N. and Yucel, N., Numerical simulation of flow and combustion in an axisymmetric internal combustion engine. In *Proceedings of World Academy of Science, Engineering and Technology* (Vol. 22). 2007.

Ra, Y. and Reitz, R.D., A combustion model for IC engine combustion simulations with multi-component fuels. *Combustion and flame*, *158*(1), pp.69-90. 2011

Ravichandra, K.M., Manikanta, D. and Kotresh, M.,. CFD Simulation of an IC Engine by Producer Gas. *International Journal of Civil Engineering and Technology (IJCIET) Volume*, *8*, pp.145-152. 2017

Reitz, R.D., Directions in internal combustion engine research. *Combustion and Flame*, *160*(1), pp.1-8. 2013.

Saffarian, M.R., Bahoosh, R. and Doranehgard, M.H.,. Entropy generation in the intake pipe of an internal combustion engine. *The European Physical Journal Plus*, *134*(9), p.476. 2019

Shafie, N.M. and Said, M.M., Cold flow analysis on internal combustion engine with different piston bowl configurations. *Journal of Engineering Science and Technology*, *12*(4), pp.1048-1066. 2017

Shu, J., Fu, J., Ren, C., Liu, J., Wang, S. and Feng, S., Numerical investigation on flow and heat transfer processes of novel methanol cracking device for internal combustion engine exhaust heat recovery. *Energy*, *195*, p.116954. 2020.

Tian, Z.F. and Abraham, J., Application of computational fluid dynamics (CFD) in teaching internal combustion engines. *International Journal of Mechanical Engineering Education*, *42*(1), pp.73-83. 2014.

Verhelst, S., Turner, J.W., Sileghem, L. and Vancoillie, J., Methanol as a fuel for internal combustion engines. *Progress in Energy and Combustion Science*, *70*, pp.43-88. 2019.