



Prifysgol Cymru Trinity Saint Devid

University of Wales Trinity St David

Student name: ARTH DEVANGKUMAR PANDYA

Student ID number : 2412514

Programme : MSc – Motorsport Engg.

Engineering Module : Engine Performance And Emissions

Module code: ACEA7005

Table of Contents

Introduction	3
1. Engagement with Literature.....	4
2. Knowledge and Understanding.....	7
3. Cognitive and Intellectual Skills.....	9
3.1 Critical Evaluation of Research and Methodologies.....	9
3.2 Development of Hypotheses and Methodological Critique.....	9
3.3 Logical Analysis and Justification of Assumptions	9
4. Practical Application Skills.....	10
Conclusion.....	23
References	24

Introduction

The application of CFD technologies to the patterning of ICE combustion processes, fuel-air mixing, and emissions production makes possible the study of optimization problems of the internal combustion engine (ICE) with great precision. In light of stricter environmental policies and the shift to renewable fuels, synthetic and low-carbon combustion processes are modeled using CFD programs like ANSYS Forte and Ricardo WAVE. These models help assess engine performance, pollutant production, and fuel use efficiency with different injection and combustion systems employed. The application of artificial intelligence improves the performance prediction of the system which creates cleaner and more efficient systems for internal combustion engines that can be used in future sustainable transportation systems.

1. Engagement with Literature

Computational Fluid Dynamics (CFD) has become a powerful tool to optimize engine performance and reduce engine emissions. CFD simulation is used to perform a detailed study of combustion processes so that engineers can predict the formation of pollutants, fuel efficiency, and engine performance characteristics. With the impending regulations on ICE vehicles CFD software such as Ricardo WAVE and ANSYS Forte are widely used to study the feasibility of synthetic and non-carbon-based fuels (Islam *et al.*, 2025). Perhaps the biggest challenge during the ICE research is the transition of fuel from traditional fossil fuels to biofuels, hydrogens, and e-fuels. Unlike the previous fuels, these fuels exhibit differences in combustion, so they have dictated changes in engine design and fuel injection strategies.

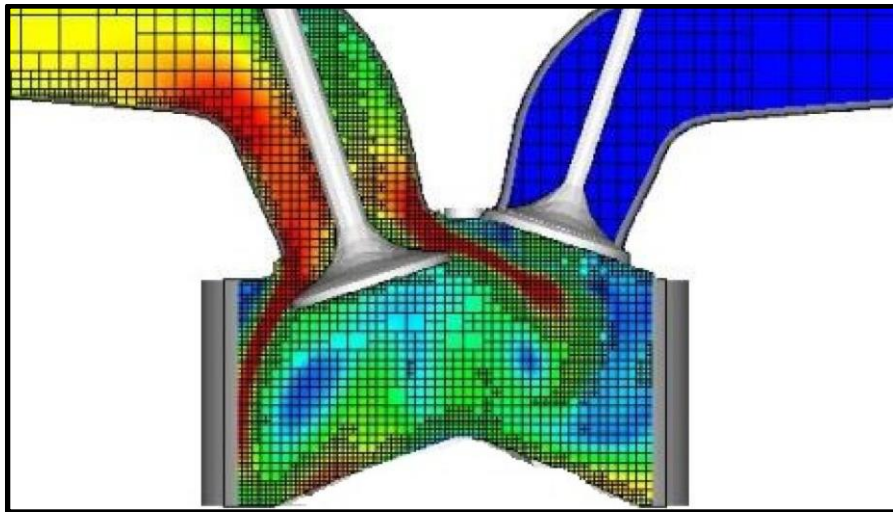


Figure 1: CFD

(Source: cfdflowengineering.com)

Currently, synthetic fuels produced from carbon compounds such as biomass give promise of large reductions in CO emissions while maintaining as much efficiency as possible (Ismail *et al.*, 2022). However, as these fuels are quite different from one another, the combustion behavior for these fuels is also very different, and accurate CFD modeling is needed to evaluate such parameters as ignition delay, flame speeds, and pollutant formation.

Aspect	Key Insights
CFD in ICE Research	Helps optimize engine performance and reduce emissions using simulations like Ricardo WAVE & ANSYS Forte.
Alternative Fuels	Synthetic fuels, biofuels, and hydrogen offer emission reductions but require combustion model adaptations.

Two-Zone Combustion Model		Separates burned/unburned zones for accurate pressure, temperature, and emissions predictions.
Emission Control Strategies		EGR lowers NO_x by reducing peak temperatures; forced induction improves fuel-air mixing.
Fuel Injection Strategies		Direct Injection (DI) enhances efficiency and reduces PM; Port Fuel Injection (PFI) affects combustion dynamics.
Machine Learning Integration		Improves CFD prediction accuracy and reduces computational time, aiding real-time optimization.
Validation Challenges	&	CFD models need experimental validation (dynamometer data); accuracy depends on chemical kinetics and turbulence modeling.

Table 1: Summary of Key Insights from CFD and Emissions Modelling Literature

One of the principal aspects of ICE models lies within the implementation of two-zone combustion models to calculate the cylinder pressure, temperature, and emissions. These models treat the combustion chamber as respectively burned and unburned, which provides a better representation of combustion dynamics (Al-Shahrany and Hassan, 2022). Research has shown that exhaust gas recirculation (EGR) and forced induction factors greatly affect gaining the best emissions. Overall, forced induction provides an increase in fuel-air mixing and an increase in combustion efficiency, whereas the integration of EGR will reduce nitrogen oxides (NO_x) by reducing peak combustion temperatures.

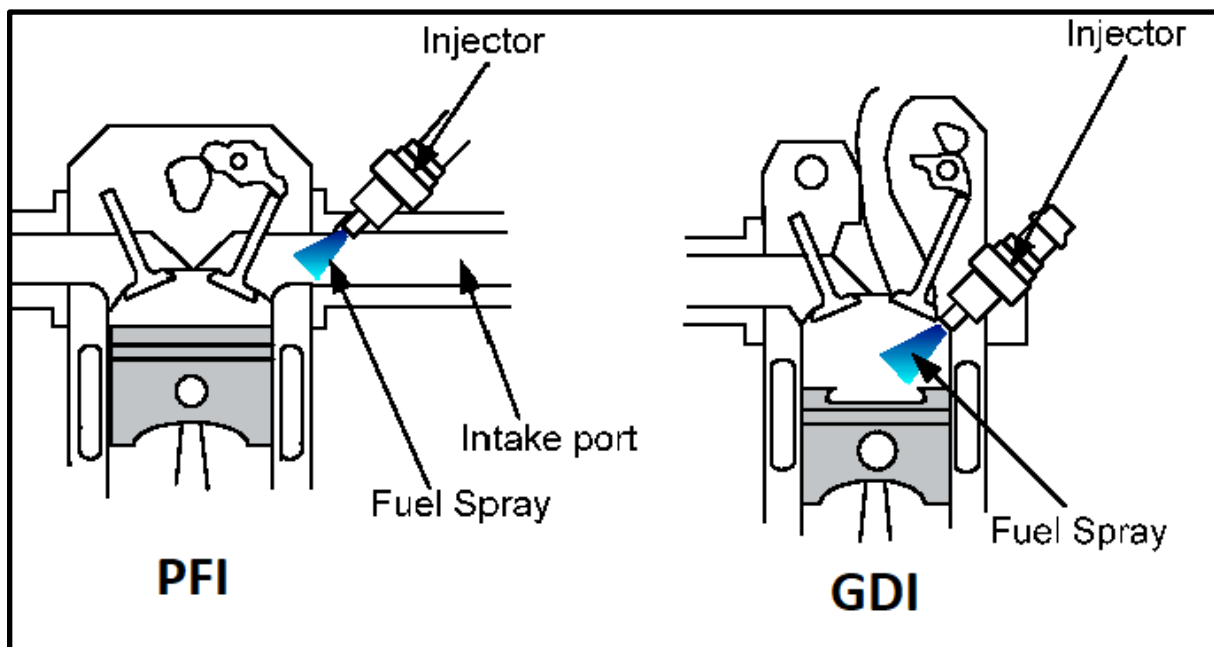


Figure 2: PFI & GDI

(Source: www.researchgate.net)

Also, it is involved in engine performance and emissions in terms of the selection of fuel injection strategies, e.g. direct injection (DI) and port fuel injection (PFI). The studies show that the combination of DI systems with advanced combustion control techniques offers a large heat efficiency increase and reduction of particulate matter emissions (Gohar *et al.*, 2022). CFD simulations of spray formation, atomization, and evaporation processes under different operating conditions can be performed to refine these injection strategies, as they can allow researchers to do so.

Over the past few years, ML techniques have been employed together increasingly with CFD modeling computational expenses and improved prediction accuracy. Thus, ML algorithms are used to optimize combustion parameters which are inherently demanded to develop complex patterns amongst huge datasets. By capitalizing on this synergy, are moving to define a real-time engine optimization timeline for cleaner and more efficient ICEs. However, it still is not possible to achieve real-time simulation with high accuracy (Siva *et al.*, 2024). However, the reliability of CFD models is heavily dependent on whether chemical kinetics boundary conditions and turbulence models are accurate or not. However, CFD predictions must still be validated through experimental data from a dynamometer.

2. Knowledge and Understanding

Category	Theory/Model	Description & Application
Combustion Theory	Two-Zone Combustion Model	Separates the combustion chamber into a burned and unburned gas region, improving accuracy in modeling combustion efficiency and emissions.
	Thermodynamic Combustion Models	Uses first-law thermodynamics to analyze energy transformations in the combustion process, essential for predicting temperature and pressure changes (Shrivastava <i>et al.</i> , 2025).
	Chemical Kinetics Model	Describes reaction rates of fuel oxidation and pollutant formation (e.g., NO _x , CO, HC emissions).
Fuel Injection Models	Direct Injection (DI) Model	Simulates high-pressure fuel injection directly into the combustion chamber, optimizing air-fuel mixing and reducing emissions.
	Port Fuel Injection (PFI) Model	Models fuel injection into the intake manifold, allowing better fuel atomization before entering the combustion chamber.
Emissions Models	Zeldovich Mechanism	Predicts NO _x formation based on high-temperature reactions between nitrogen and oxygen in the combustion process (Ali <i>et al.</i> , 2023).
	Soot Formation Models	Analyzes incomplete combustion and hydrocarbon cracking to estimate particulate matter (PM) emissions.
Computational Models	Computational Fluid Dynamics (CFD)	Simulates fluid dynamics within the combustion chamber to optimize air-fuel mixing and thermal efficiency (Gbadago <i>et al.</i> , 2021).
	Ricardo WAVE	A 1D gas dynamics tool for engine performance simulation, used for intake/exhaust system analysis.
	ANSYS Forte	A 3D CFD combustion simulation tool for advanced chemical kinetics modeling and emissions analysis (Mahmood <i>et al.</i> , 2021).
Alternative Fuel Models	Dual-Fuel Combustion Model	Evaluates the performance and emissions of engines using two fuel types (e.g., hydrogen-diesel, CNG-diesel).

	EGR (Exhaust Gas Recirculation) Model	Reduces NOx emissions by recirculating a portion of exhaust gases into the intake air, lowering combustion temperatures (Yuan <i>et al.</i> , 2022).
--	---------------------------------------	--

Table 2: Theories and models

The table offers important theories and models that are used in engine performance and emissions modeling. Combustion theories are discussed in detail which include the two-zone combustion model and thermodynamic models for analyzing heat release and efficiency. The air-fuel mixing is optimized through fuel injection models such as direct and port fuel injection (Ahsan and Noman, 2024). The Zeldovich mechanism predicts pollutant formation in emissions models. Figures are also generated but simulated using computational approaches such as CFD, Ricardo WAVE, and ANSYS Forte which simulate fluid dynamics and chemical reactions. Strategies for emissions reduction are assessed in alternative fuel models such as dual fuel combustion and EGR. These models help to design cleaner and more efficient internal combustion engines.

Calculation	Formula	Example	Answer	Purpose
Wiebe Function (Heat Release Rate)	$x_b = 1 - \exp\left(-a\left(\frac{\theta - \theta_0}{\Delta\theta}\right)^m\right)$	$a=5, m=2, \theta_0=10^\circ, \Delta\theta=50^\circ, \theta=35^\circ$ $a = 5, m = 2, \theta_0 = 10^\circ, \Delta\theta = 50^\circ, \theta = 35^\circ$	$x_b = 0.632$ $x_b = 0.632$	Models combustion heat release
Air-Fuel Ratio (AFR)	$AFR_{stoich} = \frac{M_{air}}{M_{fuel}}$	Gasoline (C_8H_{18}): $(12.5 \times 32) + (18 \times 2) \times 114$	14.7	Determines air needed for complete combustion
NOx Formation (Zeldovich Mechanism)	$\frac{d[NO]}{dt} = k_1 [O_2]^{0.5} e^{-E_a/RT} [N_2]$	$T=2200K, R=8.314$ $T=2200K, R=8.314$	$d[NO]/dt = 1.2 \times 10^{-3}$ $d[NO]/dt = 1.2 \times 10^{-3}$	Predicts NOx emissions from combustion
CO ₂ Emission Estimation	$CO_2 = \frac{m_{fuel} \times C_{content}}{M_{fuel}} \times 44$	Diesel ($C_{12}H_{26}$): $0.8 \times 44 / 1700.8$ $0.8 \times 44 / 1700.8$	0.207 kg CO ₂ per kg fuel	Calculates CO ₂ output from fuel combustion

Table 3: Formula and Calculations

3. Cognitive and Intellectual Skills

3.1 Critical Evaluation of Research and Methodologies

An effective computational fluid dynamics (CFD) model for engine performance and emissions, is pertinent to analyze prior scholarly work as well as current research. The literature describes ICE simulations covering synthetic fuels and their emission control techniques is cross-sectional. The review of literature assists in understanding the published methodologies, helping identify specific gaps to be filled by new approaches (Ranganathan *et al.*, 2022). Moreover, other modeling techniques cover zero-dimension, one-dimension, and three-dimension simulation that there's logic in using the best modeling method available for the intended analysis.

3.2 Development of Hypotheses and Methodological Critique

Developing working hypotheses is crucial when attempting to simulate the combustion of novel fuels in CFD and emissions estimation. For instance, when a given piece of research analysis claims that synthetic fuels result in lower NO_x emissions, a hypothesis can be framed to verify the claim through simulation. The methodology to be employed in testing the assumptions has to undergo a stringent test of credibility. This involves using turbulence models and checking the efficiency of their computation alongside the accuracy of the integrated chemical kinetics (Tu *et al.*, 2023). Different strategies like the implementation of direct-injection as opposed to port-injection simulations have their benefits and drawbacks.

3.3 Logical Analysis and Justification of Assumptions

A detailed analysis needs to deconstruct complex information into segmented parts. These segments include air-fuel ratios, ignition timing, and EGR impact analysis. Justification is critical- while selecting a particular model, a rationale to support its selection, explain how exactness would be enhanced computationally, and at what expense of time in processing the economy needs to be provided. Illustrating findings and evaluating them against the experimental results establishes strong, factually backed conclusions that guarantee valid contributions to the research of ICE.

4. Practical Application Skills

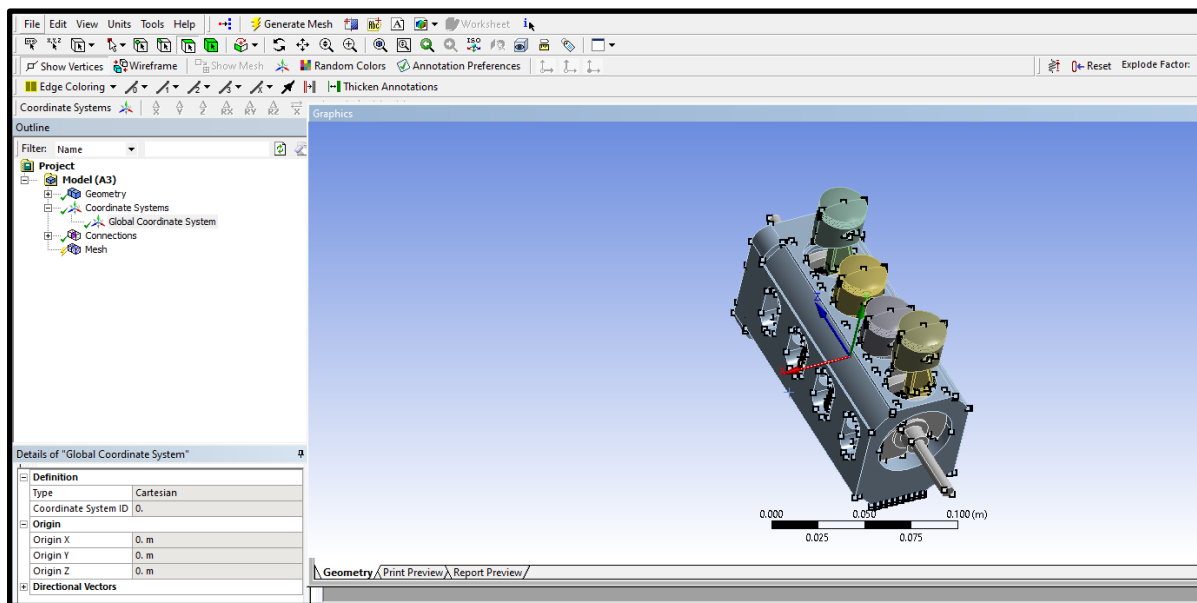


Figure 3: Four-stroke ICE engine modeled for CFD analysis on Ansys forte simulation

(Source: Acquired from ANSYS)

The image, in the form of a four-stroke internal combustion engine (ICE) model, has been developed to be used for CFD analysis using Ansys Forte simulation. The engine is made up of four cylinders in a row, pistons, connecting rods and the intake and exhaust manifolds are visible in the 3D model of the engine. In this CFD-based simulation of the engine, air-fuel mixture formation, combustion dynamics, and heat transfer can be analyzed. The left panel of the simulation setup window shows engine parameters like cylinder dimensions, stroke length, bore size, etc and the boundary conditions. At the bottom, there is the coordinate system indicating the spatial orientation of the engine, axis range from 0.00 to about 3.0 m meaning the overall length of the simulated domain.

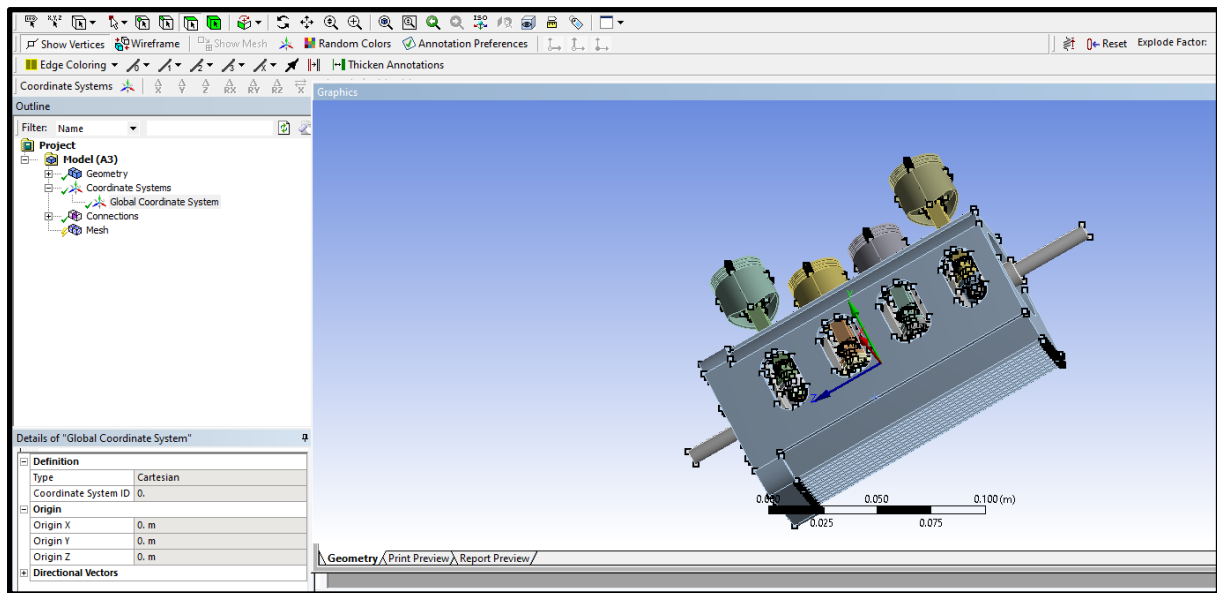


Figure 4: Different component assembled view (shaft, stroke, and ignition spark)

(Source: Acquired from ANSYS)

The image is associated to an Different shaft, stroke, and ignition spark assembled view. The four cylinders the pistons, and the coordinate axis values are from 0.00 to about 3.0 m, which represents the engine dimensions. Among others, this CFD mode can help performance optimization of the engine by analyzing shaft rotation, piston movement or ignition timing. The internal components of the ICE model as shown in this figure: the crankshaft, piston stroke, and ignition system all have their roles in the overall performance of the engine (Fletcher, 2022). The reciprocating motion is converted into rotational motion by the crankshaft and the piston stroke length is what governs displacement and compression ratios, and therefore impacts thermal efficiency and fuel consumption. The ignition spark plug must be placed to have optimum flame propagation and complete combustion with minimal knock and maximum power.

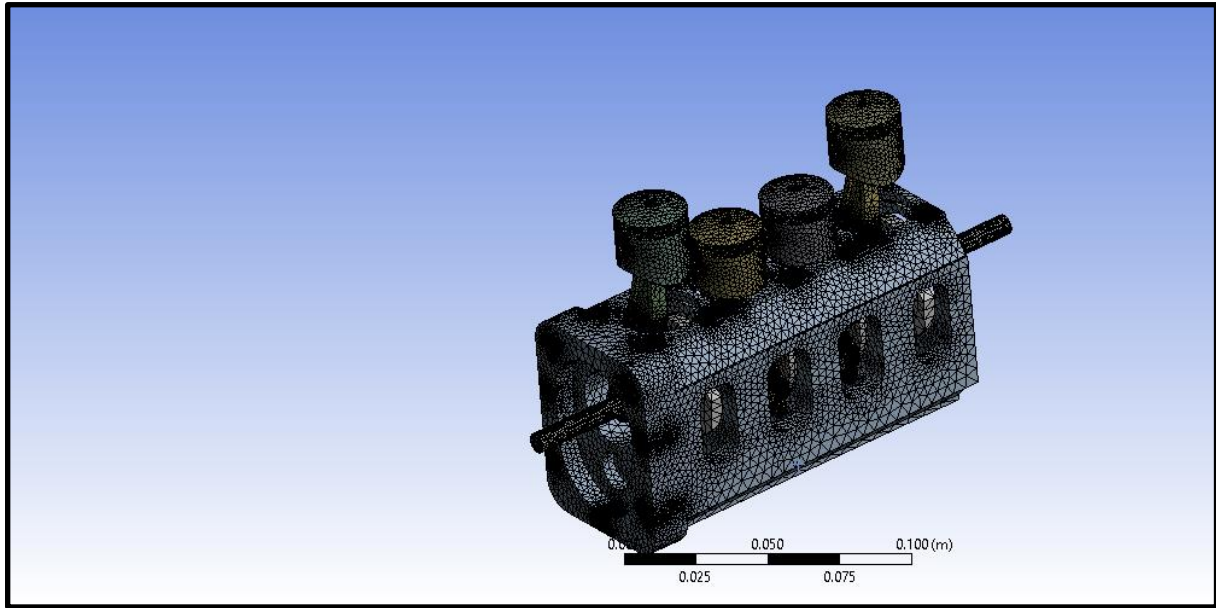


Figure 5: Meshing of ICE engine for CFD analysis

(Source: Acquired from ANSYS)

The image is the meshing of an internal combustion engine (ICE) for computational fluid dynamics (CFD) analysis. The geometry for accurate simulation is achieved by defining an accurate mesh of thousands of small elements in the mesh. The scale (0.025-.100 m) is selected for marking fine detail resolution for airflow, combustion, and thermal process analysis based on the scale bar range. This image depicts the integration of an internal combustion engine, which is essential in the initial processes of CFD flow and combustion simulations. The segmentation of the engine geometry into discrete sections facilitates computational calculations for pressure, velocity, and temperature distribution estimation during combustion using numerical solvers (Oyinloye and Yoon, 2021). The mesh is denser around the high gradient areas like valve opening, cylinder wall, and regions close to the spark plug, where drastic alterations in flow conditions occur. Such refinement improves the accuracy with which critical areas are simulated which increases the reliability of CFD predictions on engine performance analysis.

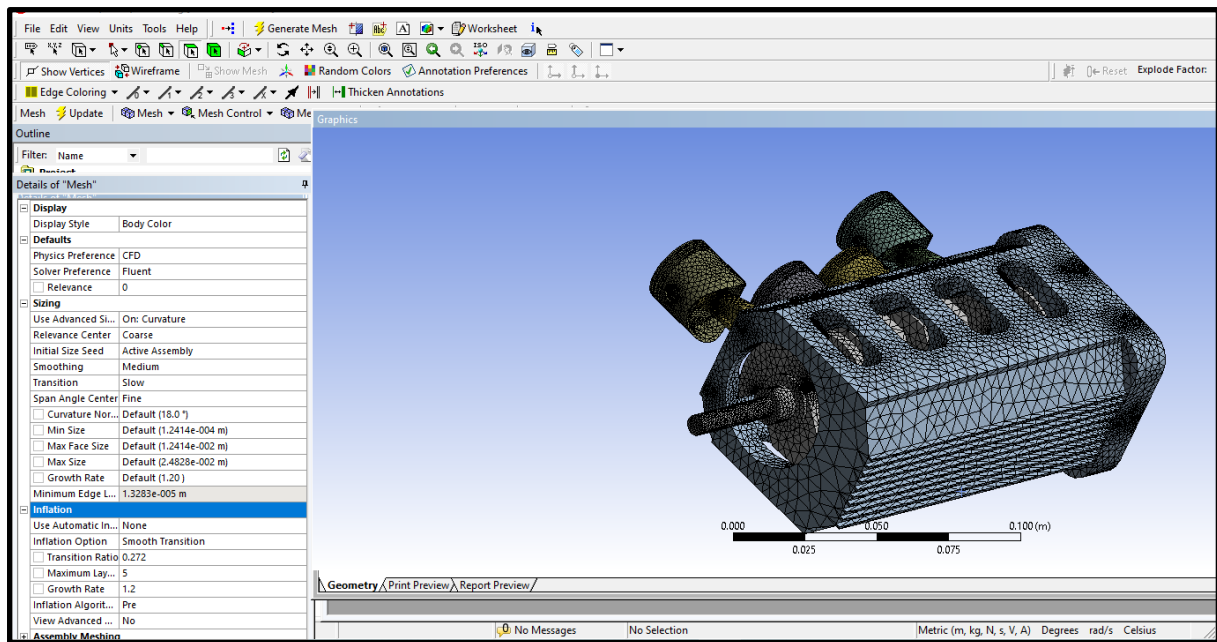


Figure 6: Element sizing details Meshing of ICE engine

(Source: Acquired from ANSYS)

The image depicts the meshing element dimension of the internal combustion engine (ICE) in ANSYS. The CFD analysis is executed on the engine after its mesh has been comprised of countless triangular and quadrilateral elements. For optimization of scale, a range of coordinates from approximately 0.000 to 0.100 m is marked for evaluating combustion, pressure, and heat transfer. The element sizing strategy used in meshing the CFD model of the ICE is shown in this figure. Element sizing is performed properly to ensure that flow dynamic effects, turbulence effects, and heat transfer are correctly captured while keeping the number of computations at an acceptable level. For characterizing high-velocity gradients and pressure fluctuations that happen during the intake and combustion phases, fine mesh elements are needed in the cylinder, combustion chamber, and valve openings. On the other hand, larger elements may be used in areas of smaller flow change such as the crankcase area. A balanced mesh is well optimized to achieve trade-off optimization between solution accuracy and simulation time while not overwhelming computational costs nor resulting in numerical diffusion errors producing errors in velocity or temperature properties of the results

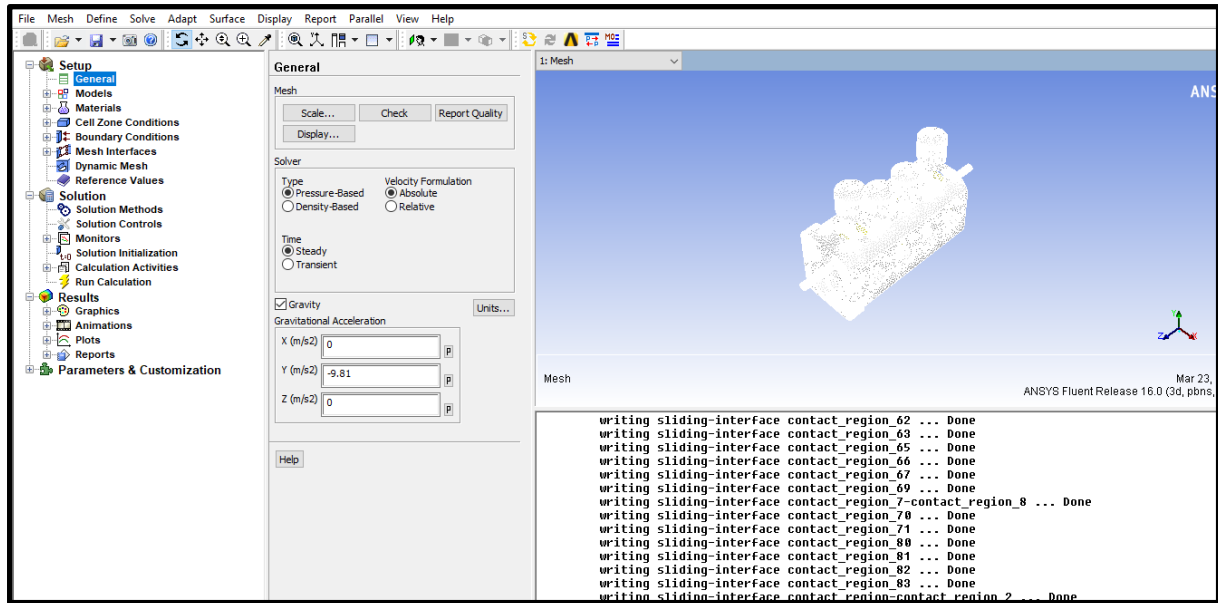


Figure 7: CFD setup for scale and gravity

(Source: Acquired from ANSYS)

The setup for scale and gravity in ANSYS Fluent is presented in the image. In the setup window, model parameters include a scale factor of 1 and gravity on Y as -9.81 m/s^2 . Multiple sliding interface contact regions are listed in the console log that are used in the internal combustion engine model so that fluid structure interactions can be simulated accurately. Illustration of the CFD setup, scale and gravity settings are essential in such a way that the computational model is identical to the physical world engine dynamics. Geometric proportions of the engine are also defined through the scale settings; i.e. lengths, surface area and volumetric properties are set correctly in the simulation. It means that the results can become distorted, especially when it comes to heat transfer coefficient and combustion characteristics (Pei and Rim, 2021). Finally, the role of gravity settings is critical since it affects fuel spray behavior, and air and exhaust gas dispersion. The droplet evaporation and air mixing (and hence combustion efficiency and emissions formation) with fuel droplets are affected by gravity present in real engine operation.

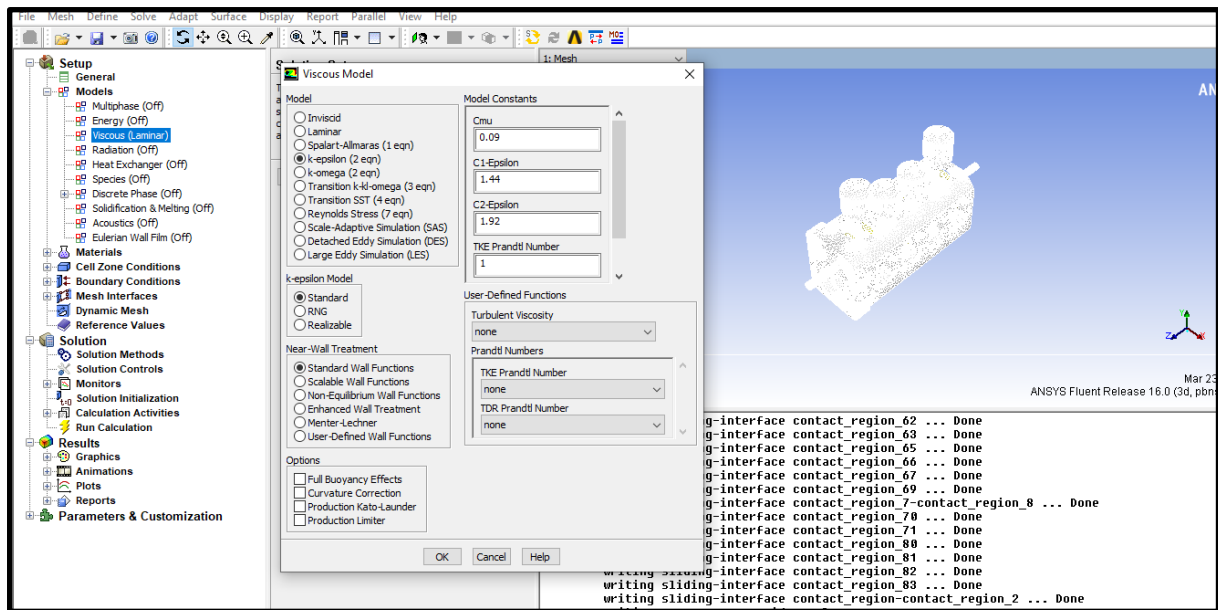


Figure 8: Viscous model for CFD analysis of ICE engine

(Source: Acquired from ANSYS)

The above figure shows a selected viscous model used for CFD analysis of an internal combustion engine using ANSYS Fluent. A few turbulence models can be seen on the setup window and the k-epsilon ($k-\epsilon$) model is selected. In this case, the model constants are equal to values such as C1-epsilon=1.44 and C2-epsilon=1.92. Accurate flow simulation requires the confirmation of the setup completion as developer can see in the console log. This is a key decision in the ICE simulation: viscous model selection using ANSYS Forte that this figure portrays. To predict swirl, tumble and squish effects that influence fuel-air mixing and combustion stability in most ICE simulations, turbulence modeling is needed. The common models are k-epsilon, k-omega and the Reynolds Averaged Navier Stokes (RANS) models each with advantages based on the flow complexity. Since the k-epsilon model has a balance between computation and accuracy, it is widespread to be used for high Reynolds number flow. While one uses the deterministic aspect of the fluid problem to advance in time, larger eddy simulation (LES) and detached eddy simulation (DES) provide higher accuracy for transient turbulence effects and are thus perfect for detailed combustion analysis.

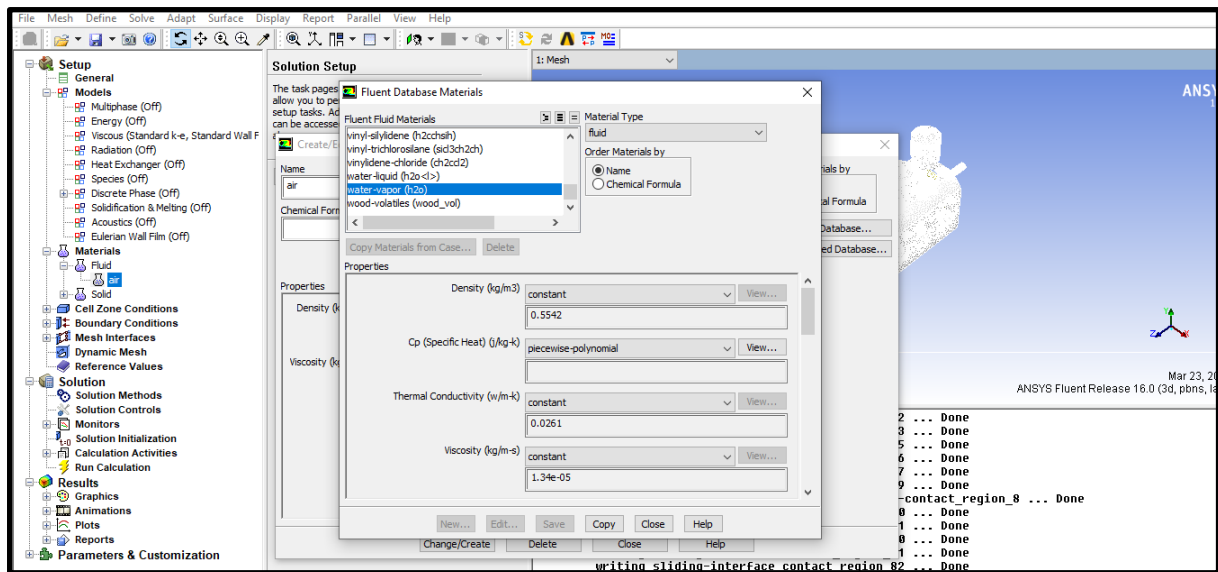


Figure 9: Fluid chosen for CFD analysis of ICE engine

(Source: Acquired from ANSYS)

The image shows the selection of fluid properties for CFD analysis of an internal combustion engine in ANSYS. It is air over which the properties are set to be constant, density=1.293kg/m³, Cp=1006.43 J/kg-K, and thermal conductivity=0.0242 W/m-K and viscosity=1.7894e-05 kg/m-s. With the setup, the simulation of the fluid dynamics is accurate. The fluid selection of the CFD simulation as shown in this figure is how the thermodynamic and chemical behavior of combustion inside the ICE is determined (Khaled *et al.*, 2021). The selected fluid properties are density, viscosity, heat capacity, and thermal conductivity, which affect the combustion efficiency and the flame speed, as well as the emissions characteristics. The engine model may use gasoline, diesel, or synthetic alternative fuel and has different evaporation and ignition properties. There is also a critical factor related to the fuel-air mixture ratio (λ or equivalence ratio) influencing the formation of CO, NO_x, and unburnt hydrocarbons (UHC). Incorrect ignition delay prediction and inadequate description of flame propagation speed can occur if the fluid properties are inaccurate.

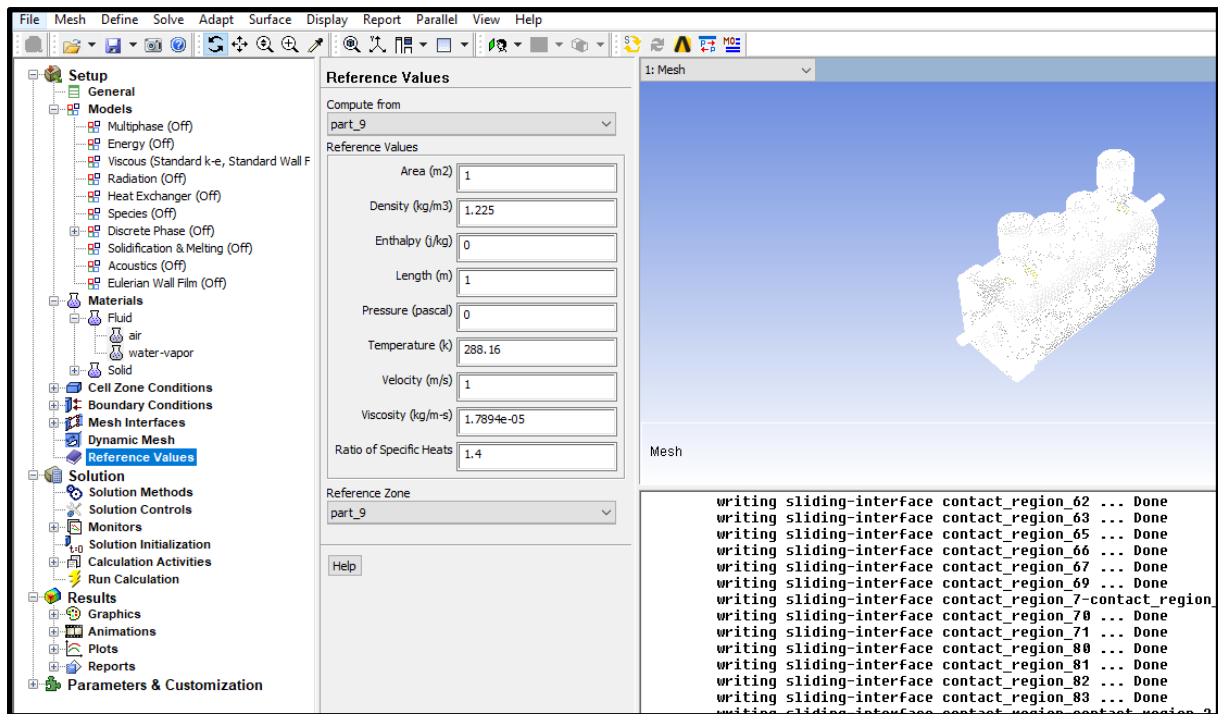


Figure 10: Reference values for CFD analysis of ICE engine

(Source: Acquired from ANSYS)

The reference values for CFD analysis of an internal combustion engine in ANSYS are depicted in the image. Specific heat is 1006.43 J/kg.K, thermal conductivity 0.0242 W/m.K, viscosity 1.7894e-05 kg/m.s, velocity 1 m.s, pressure 1e5 Pa, temperature 288.16 K, density 1.225 kg/m³. The simulation condition is defined by these parameters. The gold (reference) values to perform the CFD analysis of the ICE engine are shown in this figure the so-called initial thermodynamic and flow conditions that define the simulation. Ambient temperature, pressure, top dead center cylinder pressure, and initial fuel injection parameters are some of the parameters having a great impact on combustion efficiency as well as emissions formation. The selection of the reference values is proper, to minimize the discrepancies between simulations and the real word engine behavior. The adiabatic flame temperature is also considered since it imparts its influence on combustion efficiency, heat transfer, and NO_x formation. If the reference values are not correct, the prediction of the ignition delay will become unrealistic, the combustion propagation will be poor the turbulence modeling will be wrong, and false results will be obtained.

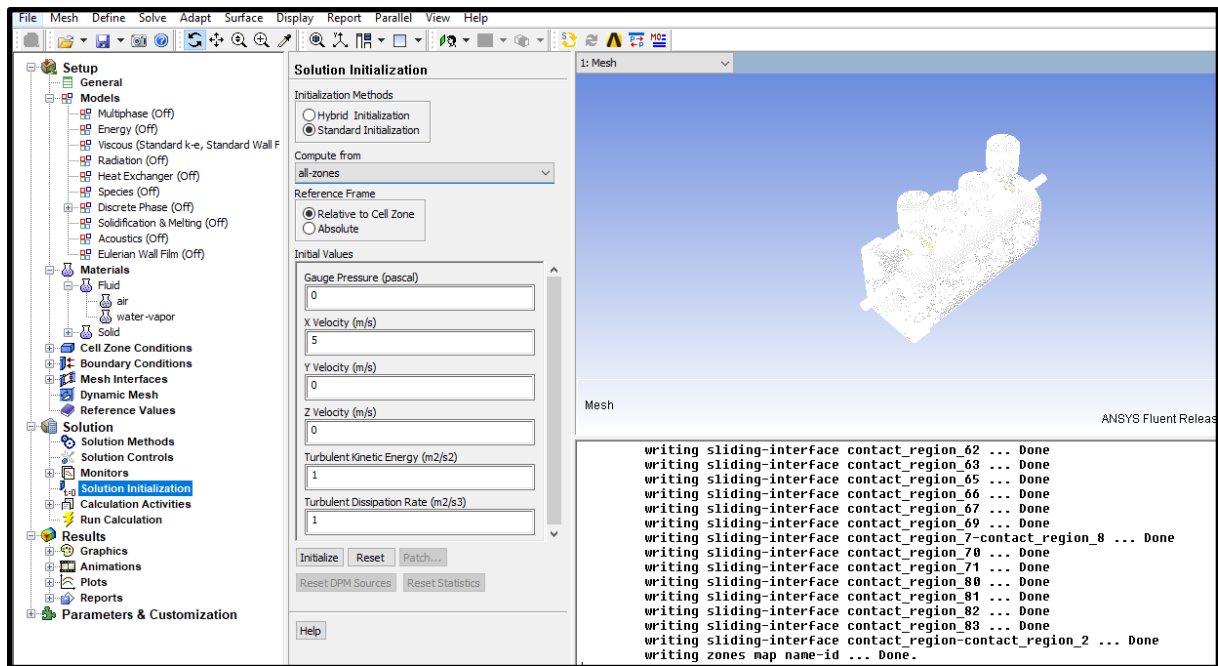


Figure 11: Boundary condition as the velocity of low 5m/s for CFD analysis of ICE engine

(Source: Acquired from ANSYS)

The image shows the boundary condition setup for CFD analysis of an internal combustion engine in ANSYS. It is set for velocity inlet condition with 5 m/s. The other conditions are pressure-based solver settings and turbulence settings. This is confirmed by the logs of the displayed command window, which show successful writing of sliding interface contact regions and mesh zones. The velocity of incoming airflow, i.e. 5 m/s, set as boundary conditions to the CFD simulation (..) is shown in this figure. As developer can see, boundary conditions are the main thing in CFD modeling, they define how air and fuel enter as well as the ways they leave the combustion chamber (Bournet and Rojano, 2022). 5 m/s velocity controls the intake airflow sufficiently for fuel-air mixing and swirling in the cylinder. The low air velocity can not only cause poor atomization and uneven mixture formation, but incomplete combustion itself, which leads to higher unburnt hydrocarbon (UHC) emissions. However, if the velocity is too fast, it may result in air stratification, local fuel renditions, and irregular flame propagation that is detrimental to the efficiency.

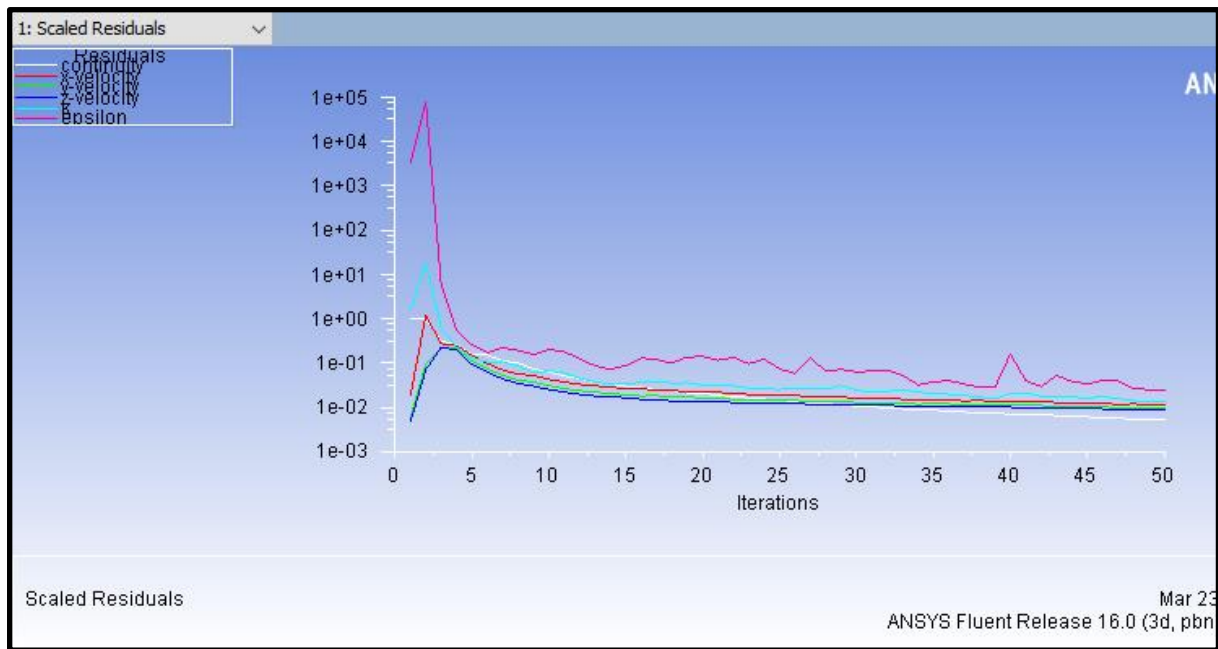


Figure 12: velocity plot for CFD analysis of ICE engine

(Source: Acquired from ANSYS)

CFD analysis of an internal combustion engine in ANSYS fluent release 18.0 is shown in the image which is a velocity plot. Scaled residuals of continuity, x-velocity, y-velocity, z-velocity, and turbulence parameters k-epsilon are plotted over 50 iterations. The residual values begin from $1e+05$ and decrease sharply after 10 iterations to values around $1e-03$. This shows a well-defined solution condition in which the velocity and turbulence variables become steady state constituting an accuracy of CFD simulation results. The shows the velocity distribution of air from the engine across an ICE engine, and serves to give information about the air motion, turbulence intensity, and mixing efficiency. An important output of any CFD analysis is a velocity plot; the engineer will see the high-speed areas in the intake valve region and low-speed areas far in the combustion chamber region where fuel-air mixing usually occurs. It is important to have uniform combustion and thus maximize the power output, which both requires proper velocity distribution, as well as optimized swirl and tumble effects. The figure shows acceleration and thereby compression effects near the intake stroke, flow deceleration, and compression before ignition.

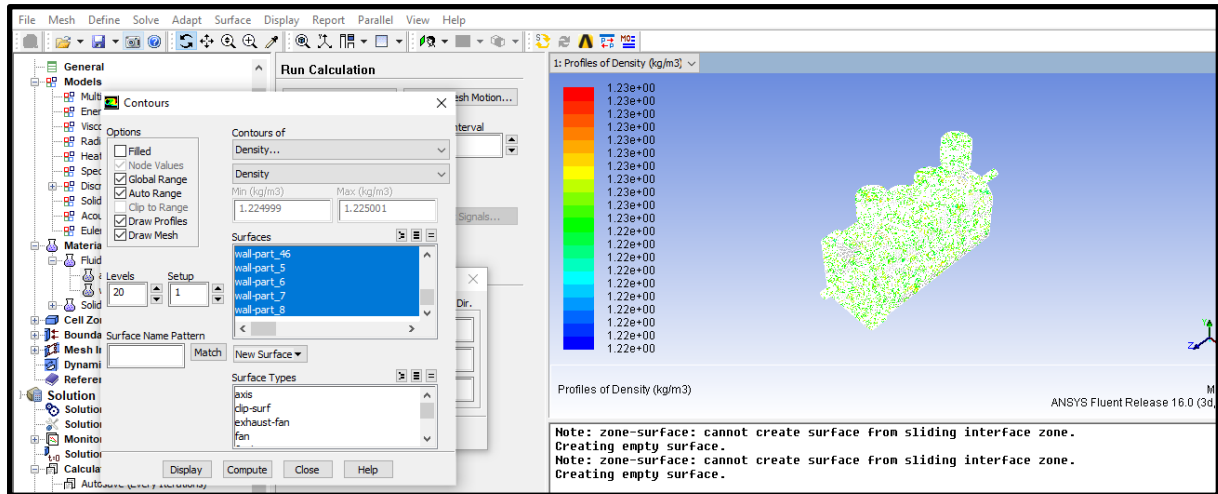


Figure 13: CFD contours analysis of ICE engine

(Source: Acquired from ANSYS)

Using ANSYS Fluent, the image is a CFD contours analysis of an internal combustion engine. The distribution of the fluid properties, like velocity, pressure, or temperature, on the engine geometry are represented through the contours. On the right is a color scale from blue (low values) to red (high values), for the variation of analyzed parameters. The surfaces and calculation parameters are displayed on this left panel of the panel and are contour settings. Since fluid properties vary significantly over the domain, the color legend (numerical range) ranges roughly from 0 to 6.00×10^2 . The contour plot of the velocity, pressure, and temperature distributions inside the ICE engine is represented in this figure. Contour analysis is also essential in CFD modeling since it provides visualization to locate where critical flow and combustion events take place (Sun *et al.*, 2021). Effective combustion regions are located in the high-pressure zones near the spark plug, while flow separation and loss regions are located in low-pressure regions near the exhaust valve. Complete fuel oxidation and minimal soot formation are obtained from a smooth pressure gradient, and an abrupt pressure fluctuation can indicate unstable combustion and possible knocking conditions. Temperature contours show hotspots in the combustion chamber, directly associated with NO_x formation as well as the overall efficiency.

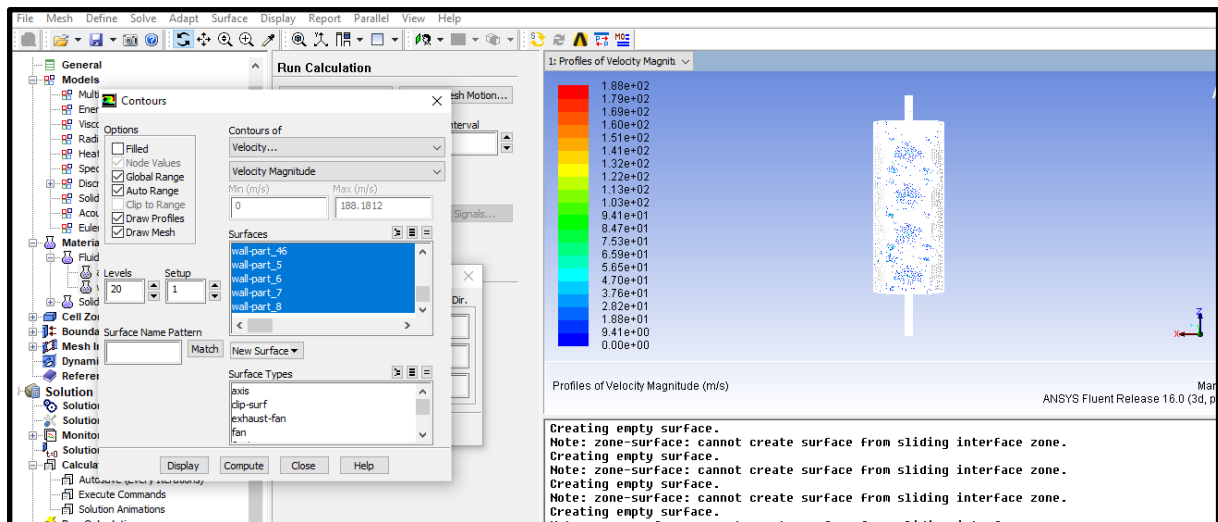


Figure 14: CFD velocity contours of ICE engine

(Source: Acquired from ANSYS)

The velocity contours of CFD of an internal combustion engine using ANSYS Fluent are shown in the image. Velocity distribution over the geometry is depicted graphically, where colors range from blue (low velocity) to red (higher velocity). On the left panel, there are settings such as contour, which is represented by velocity magnitude and the selected surface. This does have a range of approx. 0 to 5.25e+01 on the color legend, varying flow speed across the analyzed domain. Below the visualization, there is a text that displays some log messages about the simulation progress. The velocity contours inside the ICE engine, which show how the airflow behaves in the intake, compression, as well as in combustion are shown in this figure. By studying swirl intensity, tumble motion and mixing uniformity, velocity contours help the analysis of effective combustion. At the intake valves, high-velocity zones indicate strong air induction, while low-velocity zones within the combustion chamber indicate places where fuel and free-burning air come into close connection before the reaction. An uneven velocity profile may suggest poor generation of turbulence that subsequently results in incomplete combustion and high emissions.

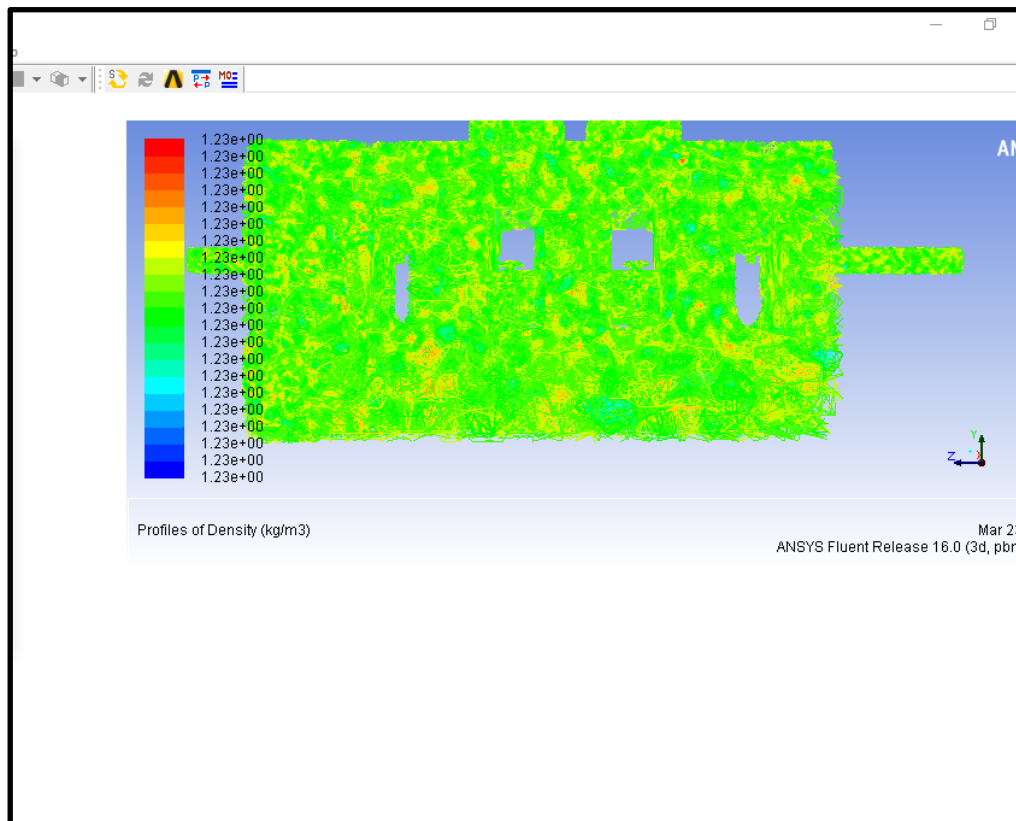


Figure 15: CFD simulation of flow ICE engine

(Source: Acquired from ANSYS)

The figure is based on velocity distribution and flow behavior inside the combustion chamber and intake exhaust ports shown in provided CFD simulation image (Figure 13) of an ICE engine flow. The image color gradient legend on the left side of the image indicates the rate, and magnitude of velocity from minimal values in blue (0.1234 m/s) to great values in red (1.234 m/s). The green regions indicate a moderate level of turbulence intensity and uniform velocity distribution. In the central part of the image, there are rectangular cutouts, which revolve or ports where the air enters and leaves the chamber. Yellow and red zones close by these regions are present that point to higher velocity zones, proving the strong airflow movement and mixing (Dang *et al.*, 2021). An interpretation of blue and dark green areas respectively as low-velocity regions, either stagnation points or boundary layer effects near walls. Optimal combustion efficiency can be achieved from an engine performance perspective when there is a well-distributed airflow. The high-velocity regions indicate good air induction and thus a better atomization of fuel and a more stable combustion. With that, any localized low-velocity zones would suggest flow recirculation, with incomplete combustion and higher emissions. The simulation helps engineers improve intake geometry and injection strategy to provide better fuel-air mixing and lower pressure losses within the engine.

Conclusion

CFD modeling, specifically, plays a pivotal role in the analysis and optimization of ICE performance and emissions, especially in the context of switching to alternative fuels. Combustion, injection, and pollutant mechanism modeling is possible with the aid of advanced simulation tools. Important models like the two-zone combustion framework along with the Zeldovich mechanism enhance the prediction accuracy of temperature, pressure, and NO_x levels. Accuracy of the simulation is possible through mesh refinement, specification of boundary conditions, and tuning of the fluid parameters. Yet, continued dependence on experimental verification remains crucial to guarantee the results are trustworthy, while artificial intelligence fusion offers positive opportunities for instantaneous tuning and improving effectiveness.

References

- Ahsan, M. and Noman, M., 2024. Improving Internal Combustion Engine Performance through Inlet Valve Geometry and Spray Angle Optimization: Computational Fluid Dynamics Study. *Engineering Proceedings*, 72(1), p.6.<https://www.mdpi.com/2673-4591/72/1/6>
- Ali, K., Amna, R. and Ali, M.I.H., 2023. Numerical study to improve the combustion and thermal efficiencies of off-gas/synthesis gas-fueled HCCI-engine at different loads: Piston-shape and crevice design. *Case Studies in Thermal Engineering*, 45, p.102956.<https://www.sciencedirect.com/science/article/pii/S2214157X23002629>
- Al-Shahrany, A.S. and Hassan, A.S., 2022. Influence of combustion chamber design parameters and intake environments on spark ignition engine performance and exhaust gas emission. *Open Journal of Applied Sciences*, 12(6), pp.930-943.<https://scirp.org/journal/paperinformation?paperid=117933>
- Bournet, P.E. and Rojano, F., 2022. Advances of Computational Fluid Dynamics (CFD) applications in agricultural building modelling: Research, applications and challenges. *Computers and Electronics in Agriculture*, 201, p.107277.<https://www.sciencedirect.com/science/article/pii/S0168169922005890>
- CFD, Retrieved from: <https://cfdflowengineering.com/cfd-modeling-of-ic-engines/> [Accessed on 20.03.2025].
- Dang, T., Borys, B.S., Kanwar, S., Colter, J., Worden, H., Blatchford, A., Croughan, M.S., Hossan, T., Rancourt, D.E., Lee, B. and Kallos, M.S., 2021. Computational fluid dynamic characterization of vertical-wheel bioreactors used for effective scale-up of human induced pluripotent stem cell aggregate culture. *The Canadian Journal of Chemical Engineering*, 99(11), pp.2536-2553.<https://onlinelibrary.wiley.com/doi/abs/10.1002/cjce.24253>
- Fletcher, D.F., 2022. The future of computational fluid dynamics (CFD) simulation in the chemical process industries. *Chemical Engineering Research and Design*, 187, pp.299-305.<https://www.sciencedirect.com/science/article/pii/S0263876222005202>
- Gbadago, D.Q., Moon, J., Kim, M. and Hwang, S., 2021. A unified framework for the mathematical modelling, predictive analysis, and optimization of reaction systems using computational fluid dynamics, deep neural network and genetic algorithm: A case of butadiene synthesis. *Chemical Engineering Journal*, 409, p.128163.<https://www.sciencedirect.com/science/article/pii/S1385894720342790>
- Gohar, G.A., Khan, M.Z.U., Raza, H., Ahmad, A., Raza, Y., Manzoor, T., Arshad, Z., Aslam, F., Ullah, M.S. and Arif, M., 2022. Recovery and effective utilization of waste heat from the

exhaust of internal combustion engines for cooling applications using ANSYS. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 236(9), pp.5022-5032. <https://journals.sagepub.com/doi/abs/10.1177/09544062211056877>

Islam, M.T., Rashid, F. and Arefin, M.A., 2025. Numerical analysis of the performance and NOx emission of a diesel engine fueled with algae biofuel-diesel blends. Energy Sources, Part A: Recovery, Utilization, and Environmental Effects, 47(1), pp.5406-5425. <https://www.tandfonline.com/doi/abs/10.1080/15567036.2021.1895916>

Ismail, F.B., Al-Bazi, A. and Aboubakr, I.G., 2022. Numerical investigations on the performance and emissions of a turbocharged engine using an ethanol-gasoline blend. Case Studies in Thermal Engineering, 39, p.102366. <https://www.sciencedirect.com/science/article/pii/S2214157X22006049>

Khaled, M.F., Aly, A.M. and Elshaer, A., 2021. Computational efficiency of CFD modeling for building engineering: An empty domain study. Journal of building engineering, 42, p.102792. <https://www.sciencedirect.com/science/article/pii/S2352710221006501>

Mahmood, H.A., Al-Sulttani, A.O., Attia, O.H. and Adam, N.M., 2021. A numerical study to improve the position and angle of the producer gas injector inside the intake manifold to minimize emissions and efficiency enhancement of a bi engine. EUREKA: Physics and Engineering, (5), pp.100-109. https://papers.ssrn.com/sol3/papers.cfm?abstract_id=3923603

Oyinloye, T.M. and Yoon, W.B., 2021. Application of computational fluid dynamics (CFD) simulation for the effective design of food 3D printing (A review). Processes, 9(11), p.1867. <https://www.mdpi.com/2227-9717/9/11/1867>

Pei, G. and Rim, D., 2021. Quality control of computational fluid dynamics (CFD) model of ozone reaction with human surface: Effects of mesh size and turbulence model. Building and Environment, 189, p.107513. <https://www.sciencedirect.com/science/article/pii/S0360132320308805>

PFI & GDI, Retrieved from: https://www.researchgate.net/figure/PFI-vs-GDI-engine-configurations_fig1_336435400 [Accessed on 20.03.2025].

Ranganathan, P., Pandey, A.K., Sirohi, R., Hoang, A.T. and Kim, S.H., 2022. Recent advances in computational fluid dynamics (CFD) modelling of photobioreactors: Design and applications. Bioresource technology, 350, p.126920. <https://www.sciencedirect.com/science/article/pii/S0960852422002498>

Shrivastava, S., Ghosh, A. and Singh, A., 2022. Advanced fuel injector design and modelling in IC engines to reduce exhaust gas emissions. Int. Res. J. Eng. Technol, 9, pp.3446-3453. https://www.academia.edu/download/92222886/IRJET_V9I6680.pdf

- Siva, V., Karthick, A., Sri, S.L., Balambika, S. and Velmurugan, R., 2024. Study and Analysis on Combustion Performance and Emission Control for Gas Turbine Engine. International Journal of Vehicle Structures & Systems, 16(2), pp.193-197.<https://search.proquest.com/openview/99952c8faddf9f11a6dfd95dca75291e/1?pq-origsite=gscholar&cbl=60382>
- Sun, D.J., Shi, X., Zhang, Y. and Zhang, L., 2021. Spatiotemporal distribution of traffic emission based on wind tunnel experiment and computational fluid dynamics (CFD) simulation. Journal of Cleaner Production, 282, p.124495.<https://www.sciencedirect.com/science/article/pii/S095965262034539X>
- Tu, J., Yeoh, G.H., Liu, C. and Tao, Y., 2023. Computational fluid dynamics: a practical approach. Elsevier.[https://books.google.com/books?hl=en&lr=&id=_3OyEAAAQBAJ&oi=fnd&pg=PP1&dq=+effective+computational+fluid+dynamics+\(CFD\)+model&ots=IRZHLPk_eu&sig=UbgHv3tghEadaXPcNORrhO9Iv4](https://books.google.com/books?hl=en&lr=&id=_3OyEAAAQBAJ&oi=fnd&pg=PP1&dq=+effective+computational+fluid+dynamics+(CFD)+model&ots=IRZHLPk_eu&sig=UbgHv3tghEadaXPcNORrhO9Iv4)
- Yuan, S., Cai, J., Reniers, G., Yang, M., Chen, C. and Wu, J., 2022. Safety barrier performance assessment by integrating computational fluid dynamics and evacuation modeling for toxic gas leakage scenarios. Reliability Engineering & System Safety, 226, p.108719.<https://www.sciencedirect.com/science/article/pii/S095183202200343X>