Computational analysis of mixture (LPG/Air) formation and performance in a dual fuel diesel engine

Thomas Renald C J1, SomasundaramP2, Yuvaraj S3, Karthikeyan K4

1,3Department of Aeronautical Engineering, Sri Ramakrishna Engineering College, Coimbatore, 641022, India.
2Department of Mechanical Engineering, Kongu Engineering College, Perundurai, 638060, India,
4Department of Mechanical Engineering, Sri Ramakrishna Engineering College, Coimbatore, 641022, India.

Abstract. This paper focuses on the use of CFD in advancing the LPG/air blend development of a dual fueled IC engine (Diesel and LPG). LPG is introduced through a secondary gas tube to the intake duct of the engine, keeping up 45° point to the flow direction. The variations in the stream boundaries change the output of the engine. Thus there is a possibility for gas tube design to position it and to get an optimized performance. From the previous literatures, CFD tool can be utilized to streamline the gas flow boundaries to improve execution. Familiar programming has been utilized to tackle this issue and approved with trial results. The outcomes show that the streamlined flow boundaries by FLUENT demonstrate 3% improvement in performance of the engine and 11% decrease in NOx emission.

Keywords: CFD analysis in induction duct, dual fuel diesel engine, Mixture formation.

1. Introduction

Air contamination with fumes is a difficult issue, and a global concern has been raised for its control and limitation. Diesel engine are the significant contributor of particulates and its impact is related with diesel particulate exhaust like carbon monoxide, nitrogen oxides, unburnt hydrocarbon, smoke, residue and different types of dark carbon just as particulate issue, for example, lead. All the gases are unsafe to the climate and mankind. They can cause nursery impact, corrosive downpour, and ozone diminishing and air contamination to the climate. Because of these impacts, human will get all sort of illness, for example, cellular breakdown in the lungs, breathing troubles, toxin and skin disease [2]. In this way, energy protection with high effectiveness and low outflow are significant examination subjects for improvement of engine framework. As of now, the engine which utilizes fills, for example, flammable gas (CNG, LNG), LPG (Liquefied Petroleum Gas), DME (Dimethyl Ether), GTL (Gas to fluids), and hydrogen is effectively evolved to take care of these issues. Particularly, LPG is given to consideration as a helpful fuel which can substitute from the oil refinement as well as the gas refinement to oil [1]. Moreover, as LPG has basic hydrocarbon structure than ordinary fuels, LPG vehicles are by and large quickly created as an affordable and low contamination vehicle. Thus the dual fuel activity of diesel with LPG can radically decrease the particulate outflow level [5]. In dual fuel activity, the essential fuel (LPG) is blended in with air in consumption framework. This combination of air and gas is compacted like in a traditional diesel engine. A limited quantity of diesel normally called the pilot is splashed close to the furthest limit of the pressure stroke to start the burning of LPG gas air combination. The burning of this pilot diesel prompts fire and ignition of the fuel [3]. Then again, for dual fuel diesel with LPG engine, however it can radically decrease the particulate outflow level[4]. The current work utilized Computational Fluid Dynamics (CFD) strategies to look at the progressions that happen in the bay stream field and combination planning.
2. Methodology

2.1 Problem Definition:
The examination was done to accomplish improvement in dual fuel (diesel and LPG) engine. The figure 1 speaks to the current model of channel pipe dual fuel CI engine with LPG stream. Here LPG is enlisted through a gas stream to the bay complex of the engine which is set near the engine, keeping up 45degree point to the wind current bearing.

![Fig.1: Intake manifold with gas jet in existing model of diesel engine. (Model 1)](image)

The variety of flow boundaries and it's upgraded positions will make the quality combination of air and LPG. This quality combination of air and LPG impacts the improvement of engine execution and decreasing the emanation level [7]. Thus there is a degree for planning a gas flow and advancing its situations to improve the combination quality of air and LPG. For existing model, combination arrangement of LPG and air was analysed by using FLUENT procedure as shown in Figure 1. The distinctive situation of gas flow course of action was additionally conveyed in FLUENT to enhance the blend quality.

2.2 Computational Approach:

Computational Fluid Dynamics (CFD) is an innovation that is utilized to examine the elements of anything that can stream in any case in fluid or vaporous state. It is a procedure that can demonstrate or reproduce a stream or marvels of any framework or gadget under investigation. The FLUENT bundle was utilized here, which is a mathematical examination code for transient, receptive, multiphase, three dimensional streams. The CFD approach in FLUENT is utilized in fractional differential conditions of stream factors to ascertain and to mimic various sorts of investigation concerning the liquid stream. Among the stream factors that are generally utilized in investigation are mass, force, energy, species focus, quantum of turbulance and combination parts [6]. Consequently, the overseeing conditions to be utilized in this investigation are the preservation of mass, force and energy conditions.

2.3 Mass Conservation Equation

The progression condition or the mass preservation condition for any liquid stream is communicated as in condition (1) (FLUENT Manual – Mass Conservation Equation 2004):

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = \dot{m}
\]

where

- \(\rho\) = fluid density.
- \(u_j\) = the \(j\)th Cartesian component of the instantaneous velocity.
- \(\dot{m}\) = the rate of mass of the object generated in the system.
2.4 Momentum Conservation Equation

The preservation of energy in an inertial reference casing can be clarified as in condition (2) (FLUENT Manual – Momentum Conservation Equations 2004):

\[
\frac{\partial}{\partial t} (\rho u_j) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_i + F_i
\] (2)

where

- \( \rho \) = Fluid Density
- \( u_i \) & \( u_j \) = The \( i \)th and \( j \)th Cartesian components of the instantaneous velocity
- \( p \) = Static pressure
- \( \tau_{ij} \) = Stress Tensor
- \( \rho g_i \) = Gravitational body force
- \( F_i \) = External body force from interaction with the dispersed phase in “i” direction.

2.5 Energy Conservation Equation

The partial differential equation for energy conservation is expressed as in equation (3) (FLUENT Manual - Energy Equation 2004):

\[
\frac{\partial}{\partial t} \left( \rho e \right) + \frac{\partial}{\partial x_i} \left( u_i \left( \rho e + \rho \right) \right) = \frac{\partial}{\partial x_i} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} - \sum_f h_f J_f + u_j \left( \tau_{ij} \right)_{\text{eff}} \right) + S_h
\] (3)

where

- \( k_{\text{eff}} \) = effective conductivity
- \( k_{\text{eff}} = k + k_t \) (where \( k_t \) = turbulent thermal conductivity)
- \( J_j \) = Diffusion flux of species ‘j’
- \( S_h \) = Additional volumetric heat sources i.e. heat of chemical reaction
- \( h \) = Sensible enthalpy
- \( e \) = Specific total energy

3. Result And Discussion

The experimental and computational analysis are studied for better clarity of the work

3.1 Experimental:

The exploratory arrangement comprises of a Kirloskar make single chamber four stroke CI engine having pressure get with wrench point encoder associated with a PC through pinnacle engine programming. This set up is furnished with air and fuel stream estimating gadget other than temperature estimation in various focuses. AVL make five gas analyser and Smoke meter have likewise been associated with the arrangement to gauge outflow boundaries and ignition boundaries. Acceptance channel has been adjusted for this work. Standard testing system has been followed and results were figured. Fuel flow position and its stream boundaries were streamlined with the assistance of CFD. In light of these outcomes the gas stream was manufactured and testing has been done on similar engine and results were thought about. The outcomes show that the advanced flow boundaries by CFD demonstrate 3% improvement in engine execution and 11% decrease in NOx emanation other than the typical exhibition of acceptance.
3.2 Computational analysis:

The stream reproduction results are introduced and considered under blend readiness measure. The blend delineation is the main factor In-chamber infusion engine. Accordingly, it is important to look at the combination arrangement measure at end of the pipe by changing the area of gas stream and its calculation. In this examination, with and without secondary type gas stream are taken to upgrade the area. Along these lines, the LPG-air blend development measure for every area arrangement was seen regarding stream.

![Figure-4: Distribution of Velocity Vector for Model 2 of Inlet Duct With Nozzle Type Gas Jet.](image)

Combination arrangement measure presents the conveyance of velocity vector and fuel fixation through streamlining the area of gas flow [9]. By enhancing area of gas stream (with and without spout type), rich blend was gotten through the situation of secondary inlet type gas flow kept up inverse to the wind current movement. LPG is splashed against the wind stream movement, dispersion of LPG with air was accomplished to get the quality blend. The Fig 2 and Fig 3 shows the conveyance of speed vector and fuel convergence of air and LPG for the diverse area of gas flow (with and without spout type-Model 1 and Model 2). The appropriation of speed vectors shows us to imagine...
the conduct of stream example and fuel fixation shows to picture the combination development as shown in Fig 4 and Fig 5. This computational technique was useful to complete the test work at Thermal Engineering Laboratory in Kongu Engineering College, Perundurai.

![MODEL 1](image1.png) ![MODEL 2](image2.png)

**FIG.5:** FUEL CONCENTRATION FOR MODEL 1 AND MODEL 2 OF INLET DUCT WITH AND WITHOUT NOZZLE TYPE GAS JET.

4. Conclusion

The current work utilized Computational method to inspect the progressions that happen in the channel pipe of dual fuel diesel engine to improve mixture formation. A mathematical recreation was performed, where diverse situation of gas flow is enhanced. The trial results show improvement in engine execution and exhaust level as shown in Fig 5- model 2.

The consequences of this work can be summed up as follows,

1. Rich combination was achieved in the analysis accomplished using FLUENT.

2. Distribution of velocity vector and fuel fixation guaranteed the blend quality and stream design.

3. Engine performance and exhaust level was improved by usage of plan and creation of enlistment conduit with spout type gas stream.

5. References

[1] Gisoo Hyun and Mitsuharu Oguma, NEDO (New Energy and Industrial Technology Development Organization), Shinichi Goto, National Institute of Advanced Industrial Science and Technology, “3-D CFD Analysis of the Mixture Formation Process in an LPG DI SI Engine for Heavy Duty Vehicles”.

[2] Dr. Ahmad Sharifian, Dr. Fok Sai-Cheong (USQ) and Dr. Talal Yusaf (Malaysia), University of Southern Queensland Faculty of Engineering and Surveying, “The Effect of Compression Ratio on the CNG-Diesel Engine”.

[3] Poonia, M.P., Ramesh, A., and Gaur, R.R. Department of mechanical engineering, Engineering College, Kota, Rajasthan, “Experimental Investigation On The Performance And Combustion Of A Lpg Diesel Dual Fuel Engine”.

[4] Vasu,R., Ramakrishnan,E.C., Ramesh,A., Nagalingam,B and Gopalakrishnan,K,V .Internal Combustion engine laboratory, Department of Mechanical Engineering, Indian Institute of Technology, Madras, Chennai, “Measurement and Control of Particulate Emission in a Four-Stroke Single-Cylinder Diesel Engine using LPG”.

[5] Roberto C .Aguirre, Jennifer C.Nathman, Haris C.Catrakis, Iracletos flow dynamics and turbulence laboratory, University of California, Irvine, CA92697, “Flow geometry effects on the turbulent mixing efficiency”, ASME Journal Engg, of Fluids Vol.128, July 2006.

[6] Kurniawan, W. H., Abdullah,S. Department of Mechanical and Materials Engineering,Faculty of Engineering, University Kebangsaan Malaysia, “CFD prediction and analysis of air motion of flow characteristics for internal combustion engine”.

5
[7] Steve Pierson, Project Engineer, Steve Richardson, Principal Engineer Jaguar Cars Ltd., Whitley, Coventry, United Kingdom, “Computer Simulation of Inlet Port Helps Improve Fuel Economy and Emissions” {JA095}.

[8] Robert S. Laramee and Helwig Hauser, VRV is Research Center, Vienna, Austria, “Geometric Flow Visualization Techniques for CFD Simulation Data”.

[9] White, T.R., Milton, B.E., and Behnia M., School of Mechanical and Manufacturing Engineering, The University of New South Wales, Sydney, NSW, 2052 Australia, “Direct Injection Of Natural Gas/Liquid Diesel Fuel Sprays”.
