

Diffusive Air Jet Combustion Chamber and Its Effect on DI Diesel Engine Combustion and Exhaust Emissions

Prakash Kamat, Yogesh Aghav, Nitin Gokhale, and M N Kumar
Kirloskar Oil Engines Ltd

ABSTRACT

An innovative Diffusive Air Jet (DAJ) Combustion Chamber concept has been introduced in the present work. The DAJ Combustion Chamber design is based on the study of rate of heat release (ROHR) curve and its correlation with emission generation. The objective is to lower the trade-off between NO_x and soot without sacrificing fuel economy of Direct Injection (DI) diesel engine. DAJ Combustion Chamber modifies ROHR curve to the desired one so that it lowers engine out emissions. To study its effect, a large bore, six cylinder engine with mechanical fuel injection system has been used. Three dimensional simulation software is used for the model calibration of basic reentrant cavity. Local emissions and ROHR curve have been studied using reentrant cavity shape. It has been modified to DAJ Combustion Chamber using Air Jet Chambers (AJCs). AJCs are positioned in the three dimensional model in such a way that they affect local in-cylinder emissions. Combustion in the DAJ combustion chamber has been studied at two different load conditions at the same engine speed. The results obtained with DAJ Combustion Chamber have been validated experimentally for the selected multi cylinder engine. In this paper simulation and experimental results with basic reentrant cavity and DAJ Combustion Chamber have been discussed in detail. Effectiveness of DAJ combustion chamber is further explored during experimentation which has been covered in this paper.

CITATION: Kamat, P., Aghav, Y., Gokhale, N., and Kumar, M., "Diffusive Air Jet Combustion Chamber and Its Effect on DI Diesel Engine Combustion and Exhaust Emissions," *SAE Int. J. Engines* 8(1):2015, doi:10.4271/2015-26-0105.

INTRODUCTION

To reduce harmful emissions of diesel engines i.e. NO_x and Soot, extensive research is happening in the engine field. Parametric studies have been carried out by different researchers to find out the best combination of bowl shape, fuel line pressure and aftertreatment device which gives the optimum result. Fuel system pressures are continuously rising to reduce the particulate emissions. Using high pressure fuel injection system engine soot can be reduced but it increases in-cylinder NO_x due to uncontrolled heat release in the premixed and diffusion phases. To reduce NO_x, Exhaust Gas Recirculation (EGR) is generally adopted but driving more EGR increases soot emissions hence trade-off of NO_x and soot needs to be lowered. To meet stringent emission norms people are using aftertreatment devices. It can reduce the engine out emissions but doesn't break the trade-off curve. Engine emissions can be reduced by working on combustion chamber shape. It can be achieved by designing innovative cavity shape which improves the charge movement in the combustion space. As EGR is used for reducing in-cylinder NO_x, effect of EGR on emissions, combustion delay and engine performance were studied in [1]. Wide parametric study using

genetic algorithm on different bowl shapes have been discussed in [2, 3]. Similarly Bowl geometries, Spray targeting and swirl ratio have been studied using different algorithms for optimization in [4]. Cavity shapes, local emissions and their correlation with ROHR have been discussed in [5]. From this study, it can be concluded that design of engine combustion parameters can improve engine emissions and performance over which benefits from other emissions reduction techniques can be ripped. In-cylinder combustion and emissions generation can be effectively studied using Computational Fluid Dynamics (CFD) tool. AVL Fire three dimensional simulation tool is used in this work. This tool is used for the engine closed cycle simulation that is from inlet valve closure (IVC) to exhaust valve open (EVO). Combustion process involves number of processes and complex physics. To capture the physics, various validated models are available in the three dimensional simulation tool. Local emissions can be effectively studied using this tool. Inputs required to define the initial boundary conditions are obtained from one dimensional tool called AVL Boost. This tool is used for engine open cycle simulation. EGR composition and amount can be found out

using the same. In this work, advanced simulation techniques and engine experimentation are used collectively to find out an innovative combustion chamber shape.

OBJECTIVE

To meet the future stringent emission targets trade-off between NO_x and Soot must be lowered. Aim of this work is to design and validate an innovative cavity shape based on engine in-cylinder combustion study which targets local emissions and reduce both NO_x and soot emissions simultaneously without sacrificing fuel economy for six cylinder heavy duty application engine.

Engine Details

Research engine used for this work is of heavy duty application. Mechanical fuel injection system with pressure capability of 1000 bar is used in this work. Details of the Research Engine are tabulated in the following table

Table 1. Research Engine Details

Sr. No.	Parameter	Unit	Engine Details
1	Number of Cylinder	--	6
2	Engine Displacement	Liter	9
3	Aspiration	--	Turbocharged inter cooled
4	Number of holes	--	7
5	Nozzle hole size	mm	0.193
6	Spray Cone Angle	Deg	146
7	Swirl Number of engine	--	1.75
8	Fuel injection system type	--	Mechanical
9	FIP Max. Pressure ability	Bar	1000 bar
10	Exhaust gas recirculation	--	Internal

EXPERIMENTAL SETUP

To build simulation model of the engine, boundary conditions need to be defined. These conditions are obtained from the experimental data. Engine experimentation was done on the test bed compliant to standards ISO 8178. Engine swirl study was done on the test rig comprising of paddle wheel. Engine performance and emissions data are recorded on transient dynamometer. Along with performance parameters, different temperatures and pressures have been recorded. Engine specific fuel consumption, exhaust temperature, pressure, air flow are measured on engine test bed. Engine testing was done as per standard ISO 8178. Engine emissions were recorded at different load conditions at the same engine speed. In engine emissions, NO_x, HC, CO and Smoke are measured. NO_x is measured from the chemical

luminescence of excited NO₂* molecule. A heated Flame Ionization Detector (FID) is used for measuring HC. [When sample gas entered into a hydrogen flame inside the FID. Hydrocarbons from the exhaust sample produce ions when they are burnt. These ions are collected by the metal collector with high voltage Direct Current (DC)]. CO measurement is based on the Non dispersive infrared radiations (NDIR) principle, [CO emission absorbs typical wavelength of infrared radiations. The device measures amount of light absorbed by the CO]. Opacimeter was used for smoke measurement as it measures opacity of exhaust continuously flowing through the sample tube. Engine dynamic characteristic data was measured using different sensors. Dynamic data are recorded using oscilloscope which are used for the derived calculation. Cylinder pressure measurement was done using piezoelectric pressure transducer. It was used to define cylinder pressure at inlet valve closure and to calculate instantaneous rate of heat release. Piezoresistive pressure transducer was used for fuel line pressure measurement which measures the mechanical stress on the doped silicon resistor area. Fuel line pressure data is used for the rate of injection calculations. Needle lift is measured using instrumented injector working on Hall Effect principle. It measures the strength of magnetic field produced because of the current flowing perpendicular to it, which gives the start of injection, end of injection and injection duration. All the above data are recorded at different load conditions for the same engine speed. Engine experimental data is used for one dimensional simulation. For the selected research engine, during experimentation, emissions were found critical at 100% and 25% loads. Hence model calibration is done at these two load conditions. While matching experimental results with simulation Power, BSFC (Break Specific Fuel Consumption), Pressure curve and NO_x ppm value are matched.

ONE DIMENSIONAL MODEL

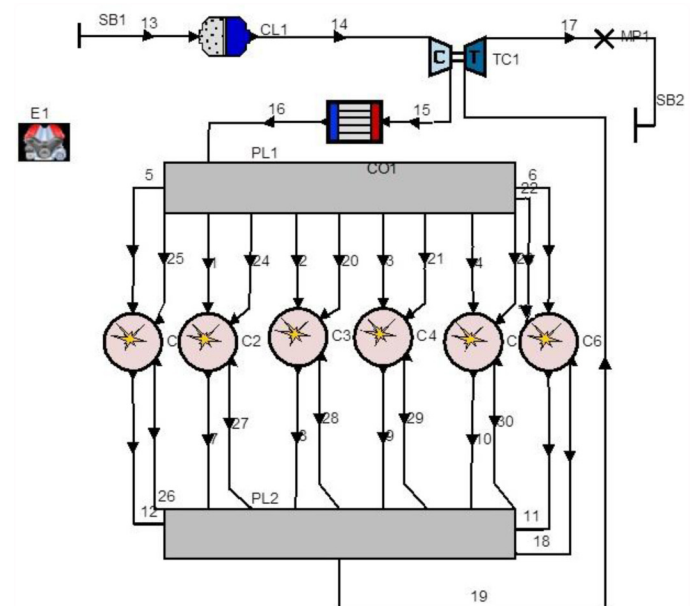


Figure 1. One Dimensional Engine Model

To reduce NO_x emissions, internal EGR has been employed in the research engine. To find out its amount and composition at initial boundary conditions of three dimensional model, one dimensional engine open cycle simulation software AVL BOOST is used. Boost model of research engine is as shown in the [figure 1](#). In this software NO_x is modeled using Patta and Hafner model in which reactions from Zeldovich Mechanism are considered. Engine model calibration results have been covered in the earlier published work [5]. Output obtained is EGR (%) at 100% load condition from boost model is 12 % and composition is 0.92. Similarly EGR (%) at 25% load condition is 16% and composition is 0.87.

COMBUSTION MODELING IN 3D SOFTWARE MESHING

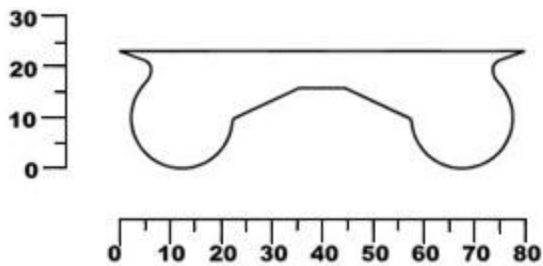


Figure 2. Cavity A

Cavity A which is shown in [figure 2](#) is a reentrant cavity used for the simulation and has been meshed in the three dimensional tool. Meshing parameters have been selected based on the mesh dependence study. Moving mesh of the Cavity A has been generated in the three dimensional simulation tool AVL FIRE. After meshing, boundaries and boundary conditions are defined.

MODEL SELECTION FOR COMBUSTION MODELING

For modeling complex combustion process, different models have been selected. Details of each of these models are described in the following section.

Flow Modeling (K- Zeta- F)

In-cylinder flows are highly turbulent due to the squish and swirl interaction. So four equation k-zeta-f model is used for achieving better numerical stability. It is more accurate than two equation eddy viscosity model. Eddy viscosity model is better for the near wall anisotropy and stress strain effect. In the k-zeta-f model, damping function which is proportional to $1/y^4$ is used. Hence this model increases the numerical stability. For the cells next to the wall, very simplified approach has been adopted called as hybrid wall treatment. It further increases the robustness of the model keeping the same accuracy.

Combustion Process Modeling

Combustion process is modeled using Three Zones Extended Coherent Flame Model (ECFM-3Z). This is a combustion model based on a flame surface density transport equation and a mixing model that can describe inhomogeneous turbulent premixed and diffusion combustion. According to the ECFM-3Z zone splitting, each computational cell is divided into three mixing zones: the unmixed fuel zone (F), the mixed zone containing fuel, air and EGR (M), and the unmixed air + EGR zone (A) [6]. As shown in [figure 3](#), unburnt fuel and unburnt air are mixed in the mixed zone. Arrow indicates the direction of reaction progress. Reaction takes place between the very thin layer entrapped between pure fuel and pure air zone. This combustion model is valid for both premixed and diffusion phase of combustion. This model is popular for diesel engines as it considers EGR and its effect on NO_x formation during combustion.

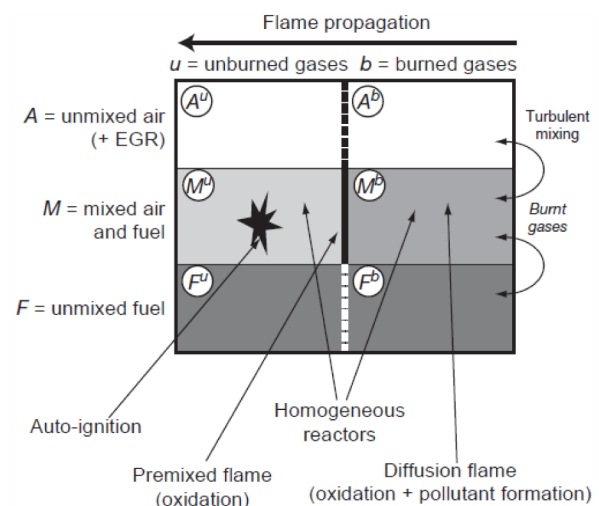


Figure 3. Extended 3 zone coherent flame model [6]

Spray Modeling

In spray modeling, wave model with blob injection which is shown in [figure 4](#) is used. It is based on the principle that initial change occurs on the liquid surface of injected fuel that is proportional to its wavelength and other physical and dynamic parameters. In high pressure fuel injection, break up takes place due to high velocities. Radius reduction of parent drop is given by the following equation



Figure 4. Wave model with blob injection [7]

$$\frac{dr}{dt} = - \frac{(r - r_{stable})}{\tau_a}$$

(1)

Where break up time τ_a is given by

$$\tau_a = - \frac{3.726 \cdot C_2 \cdot r}{\Omega \cdot \Lambda} \quad (2)$$

$$r_{stable} = C_1 \cdot \Lambda \quad (3)$$

Where C_1 is constant, default value of C_1 is 0.61. In the above equations, Ω is wave growth rate. The constant C_2 corrects the characteristic break-up time and varies from one injector to another. r_{stable} is the droplet radius of the product droplet, which is proportional to the wavelength Λ of the fastest growing wave on the liquid surface [7]. For spray wall interaction, wall jet model is used. This model is based on the spray/wall impingement model of Naber and Reitz. The concept is that under engine conditions a vapor cushion is formed under the droplets and that they rebound or slide along the walls shown in Figure 5. This allows spray/wall impingement calculations to be performed without using the wall film module. In the selected model, it is assumed that a droplet which hits the wall, suffers one of the two consequences, namely rebound or reflection in the manner of a liquid jet, depending on the Weber number.

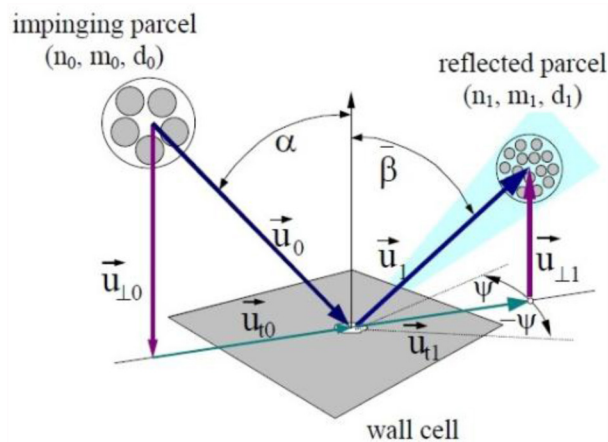


Figure 5. Spray wall interaction [7]

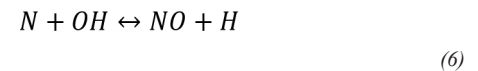
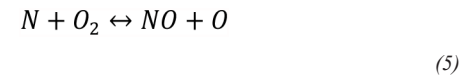
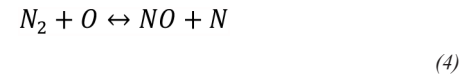
Pollutant Modeling

For NOx generation Extended Zeldovich and soot generation Frolov Kinetic model have been chosen.

NOx Extended Zeldovich Model

This model considers NO formed in both pre flame and post flame gases. There are three sources of NOx generation during combustion i) Thermal NOx - Dissociation of molecular Nitrogen into NO at high

temperatures. ii) Fuel NOx - NO formed from nitrogen containing components in the fuel iii) Prompt NOx - Reaction between hydrogen fragments and air nitrogen. Thermal NOx contributes maximum out of three. Main source of the NOx formation is in the flame front reactions which are effective at temperature (>1800 K). Following reactions are considered in the NOx formation



Soot Model

Under high temperature and fuel rich conditions hydrocarbon fuels exhibit a strong tendency to form carbonaceous particles called as soot. Increased temperature of air fuel mixture causes thermal pyrolysis, so more soot gets generated. Frolov Kinetic Model has been chosen for the soot modeling as it considers soot production and soot oxidation. Hence final soot value is difference between the rate at which soot gets generated and rates at which soot is oxidized. It considers detail chemical reactions for the calculation of soot formation and oxidation.

MODEL CALIBRATION

In three dimensional simulation software, engine closed cycle simulation is done that is from IVC to EVO. For this engine, IVC is at 610 Crank angle degree (CA deg) and EVO is at 799 CA deg. (Here Compression stroke is considered from 540 CA deg - 720 CA deg and Power stroke is from 720 CA deg - 900 CA deg). Results obtained from the boost model are used for defining initial boundary condition. Model calibration results are tabulated in the following table

Table 2. Model Calibration Results with Cavity A

Parameter	Unit	100% Load Condition		25% Load Condition	
		Simulated	Experimental	Simulated	Experimental
Engine Speed	RPM	1500	1500	1500	1500
Air Fuel Ratio	--	24.52	23.4	35.46	33.5
Break Power	kW	28.6	30.2	7.48	7.7
Brake Specific Fuel Consumption	gm/kW.hr	236	225	257.15	255
Peak Cylinder Pressure	bar	110	110	51	51
NOx	gm/kW.hr	4.3	4.1	2.85	2.72