Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 2, Number 9; April-June, 2015 pp. 689-692

http://www.krishisanskriti.org/jbaer.html

Large Eddy Simulation of Fuel Injector using Methane Gas Passes inside the Combution Chamber with Filtered Mass Density Function

Munesh Kumar Paswan[1] and K.M. Pandey[2]

[1]M.Tech Scholar, Department of Mechanical Engineering National Institute of technology, Silchar, Assam, 788010, India
[2]Department of Mechanical Engineering National Institute of technology, Silchar, Assam, 788010, India
E-mail: [1]muneshpaswan@gmail.com, [2]kmpandey2001@yahoo.com, [3]muneshpaswan@gmail.com

Abstract—*The analysis of this paper is contribute to design of the combustors chamber that field currently under rapid development in particular for propulsion. This study is focused on a cylindrical combustor of 30 cm3, fuelled by methane gas and air entered into the combustion chamber to calculate maximum temperature inside the combustion chamber, the exhaust gas temperature and the combustion efficiency are discussed. Most engine system is designed with lean non-premixed turbulent flames, for ecological reasons and pollutant emission restrictions. Unfortunately, these combustion systems are instabilities, since the heat liberate is very sensitive to air-fuel ratio variations for lean mixture. In the present work, k-ε models for lean in homogeneously non-premixed turbulent combustion are compared, implemented and validated using an unstructured commercial solver against experimental results. Filtered mass density function (FMDF) model is employed for large eddy simulation (LES} of high speed non-premixed methane gas have to be considered.*

1. INTRODUCTION

Forced ignition in premixed flames has been the focus on many previous works, whereas forced ignition in non-premixed flames is established and understood .Ignition in non-premixed flames is important in many practical applications such as high-altitude regenerate of variation gas turbines or rocket engines. Whereas recent developments of sub grid scale (SGS) closures for large eddy simulation (LES) of turbulent reacting flows. Several recent reviews are available [5-7] one of the closure filtered density function (FDF) methodology, first introduced by Pope [9] This is the counterpart of the probability density function

Forced ignition in premixed flames has been the focus on many previous works, whereas forced ignition in non-premixed flames is established and understood .Ignition in non-premixed flames is important in many practical applications such as high-altitude regenerate of variation gas turbines or rocket engines. Whereas recent developments of sub grid scale (SGS) closures for large eddy simulation (LES) of turbulent reacting flows. Several recent reviews are available[5-7]one of the closure filtered density function (FDF) methodology, first introduced by Pope [9]This is the counterpart of the probability density function(PDF) method that has confirmed quite effective in Reynolds averaged simulations (RAS). This success is due to the inherent property of the PDF to provide complete statistical information about the variables. In comparison to RAS, the LES/FDF methodology provides a more detailed and reliable prediction of turbulent reacting flows and is better suited to account for the large scale unsteady phenomena, which are prevalent in combustion devices. The main part of this presentation is devoted to RANS simulations of two variants of the GE10 gas turbine combustor, which include simulations of the premixed combustion in the chamber and the preliminary partial mixing of gas fuel and air. Simulations of the non-premixed pilot flame (new version premixed pilot flame (old version air jets cooling system of the chamber. In conditions of industrial gas turbines, on the spot combustion takes place in non-laminar and strongly wrinkled sheets with small-scale structure, which fundamentally cannot be resolved by model RANS and LES equations. The stochastic field method in conjunction with LES has been previously used for a variety of flame configurations such as non-premixed flames [17], auto-ignition [18–20] in methane and oxygen flames and partially premixed flames with extinction, [21]. A main problem in forced ignition arises from the energy deposition of the spark, as plasma thermodynamics are not easily incorporated into conventional CFD codes. The detailed simulation of turbulent flows may remain time-consuming. The description of turbulent eddies requires dividing the computational domain into very small volumes in which equations of flow must be solved: The smallest scales of turbulence are investigated, that more cells are required. In order to resolve all these equations, the influence of turbulence has been modeled using averaged equations (Reynolds Averaged Navier-Stokes or RANS equations). The main advantage of the combustion chamber mesh is designed according to the geometrical configuration, rather than on the turbulence properties. This enables the reduction of required cells. Due to the increasing performance of computers; new models for turbulence have appeared. With Large Eddy Simulation (LES), rather than averaging the effect of turbulence, the equations are filtered. A part of the turbulent eddies (the largest) are explicitly resolved and computed the smallest ones are modeled as with RANS modeling. LES costs more CPU-time and memory capacities than RANS, because of finer resolved scales and meshes.

2. COMBUTION CHAMBER AND OPERATING CONDITION

The diameter of the chamber is 0.001 m, its height 0.001 m (aspect ratio = 0.312), for a total volume of 30 cm3. The methane feeding duct has a diameter of 0.0018 m, the air duct 0.006 m. exhaust duct on the lower side of the chamber diameter has been given by0.016 m. To improve mixing the gaseous methane is injected in the radial direction at 90° with respect to the air flow which is tangential to the combustion chamber. The dimensions of the chamber have one of the strong influence of the mixing dynamics and consequently on the combustion efficiency due to the effects of the strong aspect ratio (S/V) on the mean streamline curvature and on the swirling motions. Operating and boundary conditions are reported in mass flows rate at inlets are imposed r with the pressure of 2 atm at the outlet pressure imposed with the pressure 2.6 atm. while the walls are at constant temperature and pressure of the combustion chamber. Two different values are specified depending on the device zones, the chamber and air duct walls are at 700 K while the methane and exhaust ducts are at 300 K. The fuel inlet Renlond'sturbulent (Re_{air} = 14,900) this feature is characteristic and peculiar of combustors; it is well known that the possibility of relaminarization is enhanced by combustion that causes an increase of the effective viscosity of the burning mixture on the other hand the slight expansion inside of the combustion chamber introduces a contrasting effect, so that the final regime is determined by the dynamic competition between these two effects. From here, FMDF show the most accurate method in CFD but highly cost and need very fine grid. So, LES is overtake by taking large eddies into account since large eddies carries massive energy. Even so LES is cheaper than k-ϵ but when compared to RANS reliability, LES is quite cost and demanding processor. So, LES modeling has problems with boundaries and is less computationally efficient than RANS techniques. RANS generally, k-ϵ especially is the most efficient in term of computational cost, time processing and processor demand. Even the result that obtained is not exactly same as k-ϵ but still acceptable and well known in engineering problems.

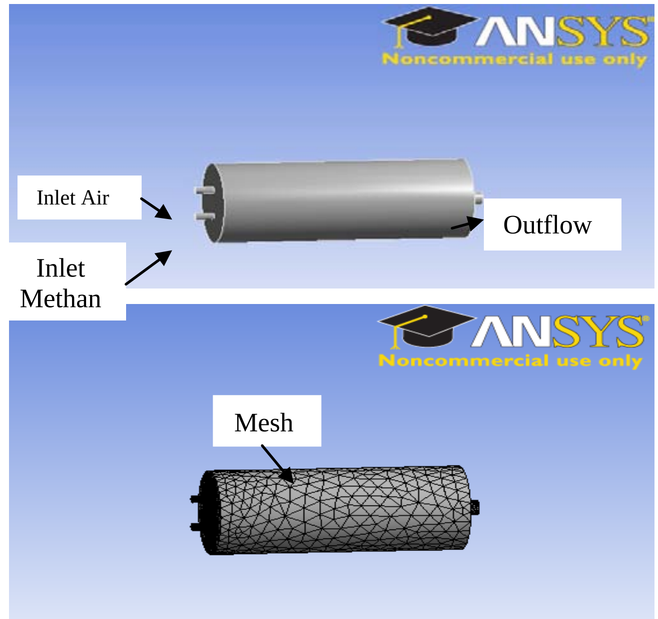
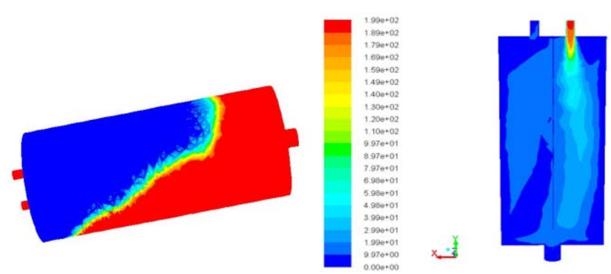


Fig. 1: Representative geometry and mesh of the combustion chamber.

3. TURBULENCE AND COMBUSTION THEORY

Turbulence combustion described before focusing on the specific subject. LES simulation of in homogeneous premixed turbulent combustion. The conservation and Navier Stokes equations are described in the first section. In the second section of the combustion chamber, the concept of turbulence is presented. Combustion theory with different flame configurations and relevant variables is then described, neglecting for a while the interaction with a moving flow. The last section gives an overview on premixed turbulent combustion.

4. RESULT AND DISCUSSION



(a) **(B)**

Fig. 2: Completely conversing the geometry of combustion chamber.

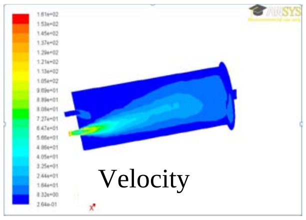


Fig. (C)

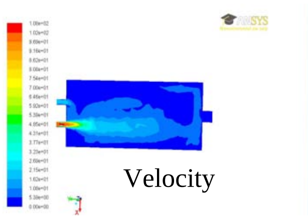


Fig. (D)

Ignition of methane-oxygen scramjet engines is characterized by high speed injection jets and very fast chemistry, making the ignition time a key parameter for the success or failure of ignition. if it is too short, reactants are not sufficiently mixed to react strongly enough compared to the short convective time and to sustain combustion. If it is too long, the mixed reactants ignite too strongly and generate high and dangerous pressure levels. Simplified sketch of an injection plate of a cryogenic scramjet engine supporting fuel injectors feeding a chamber connected to the exit nozzle. The ignition of such an engine follows a specific sequence controlled by the timing of the valves opening. A usual ignition sequence may be described in two phases.

Table 1

INLET	AIR	METHANE
Mass Flow Rate(Kg/S)	1.9e-3	3.93e-5
Velocity Inlet(M/S)	100	18
Inlet Temperature(K)	700	300
Outlet Pressure(atm)	2.67	2.67
Chamber Temperature(K) Methane Duct	700	700
Wall Temperature(K) Air Duct	300	300
Wall Temperature(K) Exhaust Duct	700	700
Wall Temperature(K)	300	300
Kinetic Energy Ratio	0.038	0.038
Global Equivalence	0.0355	0.0355

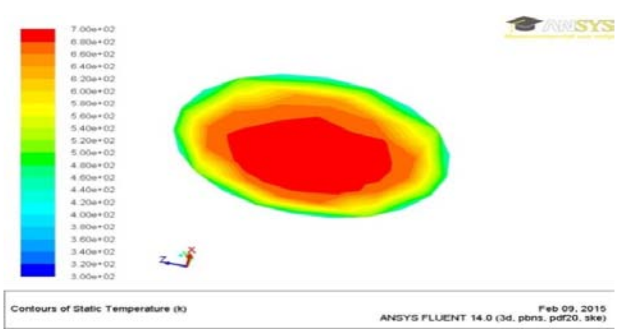


Fig. 3: Combustion takes place at the end of the chamber

First the system is purged with a methane gas to reach a nominal state and to cool down injection lines. Then the fuel injection (usually oxygen) starts and after a few milliseconds the igniter is triggered. In real engines, the igniter is either a pyrotechnic system or a spark torch usually located at the center of the injection plate (producing a strongly under expanded jet in the chamber) that blows a stream of hot gases. Fig. A represented completely conversion takes place inside the combustion chamber and B represented internal temperature 1990(k) and contour of static temperature 700(k) represented at the exit of the outflow.

Fig. C and D representing the plane flow velocity of the internal combustion chamber.

5. MASS CONSERVATION EQUATION

In a control volume, the mass conservation law is valid for any fluid. It describes the local change of the density *b*ecause of density fluxes through the surfaces of the volume control:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho ui}{\partial xi} = 0$$

This conservative form of the equation corresponds to the so-called Euler description of the flow, based on a control volume observation. An integral result can be deduced employing Ostrogradsky’s theorem. This theorem expresses the rate of change of a variable in a volume by evaluating its flux through the surrounding surface. A second description, the so-called *Lagrange* form, consists in tracking particles or volumes of the fluid along their movement (*convective form*) Using the *partials deviation.*

$$\frac{d}{dt} = \frac{\partial}{\partial t} + ui\frac{\partial}{\partial xi}$$

5.1 Conservation of Equation can be written as

$$\frac{d\rho}{dt} + \rho\frac{\partial ui}{\partial xi} = 0$$

In this thesis the Euler description is mostly used, since it corresponds to the CFD solver description. This is also the description which is implicitly obtained from experiments: measurements are always carried out along fixed planes of the space, as stressed by Mathieu A property concerning the compressibility of the fluid or of the flow is derived from mass conservation equation. A fluid is incompressible if its density is constant. Generally, gases are considered as compressible fluids, whereas liquids can mostly in case of usual pressure and temperature conditions considered as incompressible fluids. This definition is generalized for a flow. A flow is said to be incompressible if each particle of fluid has a constant density.

$$\frac{d\rho}{dt} = 0, \frac{\partial ui}{\partial xi} = 0$$

. Energy and Enthalpy Conservation Equation

Newton’s second law of motion can be applied to any volume of fluid taken on its movement i.e. using the Lagrangian description as demonstrated by Piquet. Momentum change can be due to volume forces **f** (typically the gravity effect), or surfaces forces **B** (typically pressure and viscosity effects)

$$\rho\frac{\partial ui}{\partial xi} = \rho\left(\frac{\partial ui}{\partial xi} + \frac{\partial ui}{\partial xj}\right) = \frac{\partial B_{ij}}{\partial x_j} + f_{¡g}$$

The tensor **B** groups the effect of the pressure *p*, which acts perpendicularly to the surface of the fluid volume, and the effect of the viscosity tensor.

$$B_{ij} = -\,p f_{ij} + t_{ij}$$

This equation can be written in its equivalent conservative form. Only modified and expanded with the conservation equation.

$$\rho\frac{dui}{dt} = \frac{\partial \rho ui}{\partial t} + \frac{\partial \rho uiuj}{\partial xj}$$

The momentum transport equation under the conservative form yields

$$\frac{\partial \rho ui}{\partial t} + \frac{\partial \rho uiuj}{\partial xj} = -\frac{\partial \rho}{\partial xi} + \frac{\partial f_{ij}}{\partial x_j} + \rho f_{¡}$$

Compared to the convective form the only but important difference is the presence of the density in the derivation operators. Practical consequences for the averaging procedures in CFD solvers are explained in section.

6. ACKNOWLEDGEMENTS

The filtered density mass function (FDMF) methodology [7, 8] is now at a stage that it can be used for prediction of complex turbulent reacting flows. This is demonstrated in this work by utilizing the simplest form of the FDMF for k epsilon method of a piloted, none premixed turbulent and methane jet flame. This equation is solved by hybrid finite difference Monte Carlo method. After stabilizing the consistency and accuracy of the hybrid solver the predictive capability of the overall scheme is assessed comparisons with experimental data. The deposition of the spark was molded through a Gaussian distributed source term added to the enthalpy equation. The result overall showed that the method is capable of quantitavely reproducing the different stage of an ignition sequence and underline that the presented K-ε and FMDF method approach is capable is describing a wide range of flame configuration with usually the same modeling parameter. The sensitivity of the calculation to the sub grid mixing model is also investigated by performing simulation with different model coefficients and with dynamics model. Even through the result with different mixing coefficient and models are not those much different for fuel injected into the combustion chamber. The calculation conducted most accurate overall agreements for both flames and hence further improvement in the prediction might be possible with better mixing.

REFERENCE

[1] J.A. van Oijen, F.A. Lammers, L.P.H. de Goey, Combust. Flame 127 (3) (2001) 2124–2134.

[2] C. Schneider, A. Dreizler, J. Janicka, Flow Turbulence Combust. 74 (1) (2005) 103–127.

[3] J.A.s van Oijen, F.A. Lammers, L.P.H. de Goey, Combust. Flame 127 (3) (2001) 2124–2134

[4] F. Biagioli, "Stabilization Mechanism of Turbulent Pre- Mixed Flames in Strongly Swirled Flows," *Combustion Theory and Modelling*, Vol. 10, No. 3, 2006, pp. 389-412.

[5] V. Zimont, "Kolmogorov's Legacy and Turbulent Pre- Mixed Combustion Modelling," In: W. J. Carey, Ed., *New Developmentsin Combustion Research*, Nova Science Publishers, New York, 2006, pp. 1-93.

[6] R. Cabra, J.Y. Chen, R.W. Dibble, A.N. Karpetiss, R.S.Barlow, Combust. Flame 143 (4) (2005) 491–506.

[7] C.J. Sung, C.K. Law, J.-Y. Chen, an augmented reduced mechanism for methane oxidation with comprehensive global parametric validation, in: Proceedings of 27th Symp. (Int.) on Combustion, the Combustion Institute, Pittsburgh, PA, 1998, pp. 295–303.

[8] H. Pitsch, Improved pollutant predictions in large-eddy simulations of turbulent non-premixed combustion by considering scalar dissipation rate fluctuations, Proc. Combust. Inst. 29 (2002) 1971–1978.

[9] Q. Tang, J. Xu, S. Pope, Probability density function calculations of local extinction and no production in piloted-jet turbulent methane/air flames, in:28th Symposium (Int.) on Combustion, the Combustion Institute, Pittsburgh, PA, 2000, pp. 133–139.

[10] S. Subramanian, S. Pope, A mixing model for turbulent reactive flows based on Euclidean minimum spanning trees, Combust. Flame 115 (1998) 487–514.

[11] C.J. Sung, C.K. Law, J.-Y. Chen, an augmented reduced mechanism for methane oxidation with comprehensive global parametric validation, in: Proceedings of 27th Symp. (Int.) on Combustion, the Combustion Institute, Pittsburgh, PA, 1998, pp. 295–303

[12] M. Sheikhi, T. Drozda, P. Givi, F. Jaberi, S. Pope, Large eddy simulation of a turbulent non premixed piloted methane jet flame (Sandia Flame D), Proc.Combust. Inst. 30 (2005) 549–556.

[13] S. James, F. Jaberi, Large scale simulations of two dimensional non-premixed methane jet flames, Combust. Flame 123 (4) (2000) 465–487.

[14] R. Cao, S. Pope, The influence of chemical mechanisms on PDF calculations of nonpremixed piloted jet flames, Combust. Flame 143 (2005) 450–470.

[15] M. Sheikhi, T. Drozda, P. Givi, F. Jaberi, S. Pope, Large eddy simulation of a turbulent non premixed piloted methane jet flame (Sandia Flame D), Proc.Combust. Inst. 30 (2005) 549–556.

[16] H.W. Wu, S.W. Perng, LES analysis of turbulent flow and heat transfer in motored engines with various SGS models, Int. J. Heat Mass Transfer 45 (11)(2002) 2315–2328.

[17] J. Wäsle, A. Winkler, and T. Sattelmayer. Spatial Coherence of the Heat Release Fluctuations in Turbulent Jet and Swirl Flames. *Flow, Turbulence and Combustion*, 75(4):29–50, 2005.

[18] C.M. Cha, P. Trouillet, A subgrid-scale mixing model for large-eddy simulations of turbulent reacting flows using the filtered density function, Phys. Fluids 1(6) (2003) 1496–1504.

[19] P.J. Colucci, F.A. Jaberi, P. Givi, S.B. Pope, Filtered density function for large Eddy simulation of turbulent reacting flows, Phys. Fluids 10 (2) (1998) 499–515.

[20] H. Yamashita, M. Shimada, and T. Takeno. A numerical study on flame stability at the transition point of jet diffusion flames. In *Symposium on Combustion, the Combustion Institute*, volume 26, pages 27–34. Symposium on Combustion, the Combustion Institute, 1996