Effect of Adding Water Content in the Fuel on Soot Formation and Pollutant Emission in Methane Combustion by CFD Simulation

Md. Samiullah
M.Tech. Student
Department of Mechanical Engineering
Narsimha Reddy Engineering College Hyderabad, India

Abstract— Owing to the major source of energy production, combustion technologies are commercially available throughout the world. However, combustions are responsible for too much atmospheric pollution which is detrimental to human health and environment. Thus concerted efforts are being made to reduce the particle emissions due to soot formation caused by incomplete combustion and acid precipitation originating mainly from Nitrogen Oxide (NO_x) during combustion process at high temperature. However, with the emergence of new fuels and combustion modes, it is necessary to improve the computational models. Hence turbulent combustion needs to be optimized through better understanding to meet demands for lower emissions without sacrificing power and fuel consumption. In fact, emission reduction measures can possibly be developed by understanding the mechanism of pollutants formation during combustion process, which requires knowledge of trace species. These species are produced by finite rate chemical kinetic mechanisms that involve large numbers of elementary steps. For this purpose, the application of a 3D Computational Fluid Dynamics (CFD) model simulations are proven to be very effective to support the experimental investigations of turbulent combustion. In the present paper, CFD simulation of combustion has been reported in order to reduce the pollutant emission and soot formation. The Pollutant reduction is performed on well-validated comprehensive mechanism that was designed to simulate the combustion of natural gas (methane-air) constituents and production of NO_x and sooth, and model is able to predict the NO_x emission levels with all possible gas phase reaction mechanism such as thermal, prompt, and nitrous oxide taken into account. The reaction rates of pollutant formation in combustors are determined by both chemical kinetics and turbulence and modeled using eddy-dissipation mode. The combustion is simulated by steady-state governing equations solved using the SIMPLE algorithm and the effect of turbulence on the mean flow field was accounted for using the RNG k-$\in$ turbulence model, this non-premixed type combustion is molded using ANSYS Fluent-species transport model, followed by single-step reaction mechanism and NOx model assuming conversion of the fuel to CO_2, H_2O, NO and NO_2 etc. The reaction mechanism is validated for using constant-volume combustion chamber experiments. Further, the model is also validated for inlet diffusion and diffusion energy source. The flow parameters like velocity, temperature, and mass fraction are defined prior to the simulation. In the work presented in this thesis, the first part focused on the effect of numerical variation in mass fraction of different species during the process using line probes at different location of combustor has been studied and plotted CO_2, H_2O, CH_4, N_2, O_2 and NO_x emission and Soot formation accordingly. In the second part of the research, an attempt has been made by adding water content in the fuel by 5% to 30% by mole in order to predict its effects on the result for sooth and emission reduction using methane-air two steps as a mixture material followed by parametric study approach. Consequently, the optimal values were evaluated and model is able to predict the reduction in NO_x emissions and sooth formations when natural gas (methane-air) is used.

Keywords: CFD Simulation, Turbulence combustion, Soot formation, Emission reduction, NO_x Production, Mass fraction, Chemical Kinetics

I. INTRODUCTION

A tremendous amount of heat released through a chemical reaction by oxidation of combustible element (fuel) in combustion phenomenon which globally recognized as a major source of energy production. Combustion technologies are commercially available throughout the world. However, combustions are responsible for too much atmospheric pollution which is detrimental to human health and environment. Greenhouse gas emission and acid precipitation resulting from combustion of fossil fuels such as coal, gasoline, and natural gas has been a great concern. Thus concerted efforts are being made to reduce the particle emissions due to soot formation caused by incomplete combustion and acid precipitation originating mainly from Nitrogen Oxide (NO_x) during combustion process at high temperature. However, with the emergence of new fuels and combustion modes, it is necessary to improve the computational models. Hence turbulent combustion needs to be optimized through better understanding to meet demands for lower emissions without sacrificing power and fuel consumption.

Being the most challenging limits to be met, current research interests target on reduction of NO_x and PM (particulate matter-mostly Soot) emissions because these two are the major pollutants resulting from combustion. NO_x is a chemical compound of oxygen and nitrogen and is a mixture of nitrogen oxides, i.e. NO and NO_2 that is formed by reacting each other during fuel rich combustion at high temperature. NO_x cause serious health effects and can result in acid rain and formation of ozone which leads to damage the ecosystem plants and animals. NO_x is also responsible for smog and respiratory disease. It easily react with ammonia and water vapors to form nitric acid (HNO_3) Soot on other hand is a deep black powdery or flaky substance which largely consists of amorphous carbon formed due to incomplete burning of organic matter results in the formation of dioxins and other toxic compounds. It is more properly restricted to the product of the gas-phase combustion process. Formation of soot can take place in all kinds of practical combustion systems, especially in systems

which operate on a diffusion flame concept. The main constituents are carbon atoms. It contains up to 10 mole percent hydrogen and even more if it is young, as well as traces of other elements. The soot particles are a penalty for the entire world where they contribute to climate warming, ice melting and health hazard which causes various types of cancer and lungs disease. Therefore, research on clean and efficient combustions has been an active area of research. Under this context, emission reduction measures can possibly be developed by understanding the mechanism of pollutants formation during combustion process, which requires knowledge of trace species. These species are produced by finite rate chemical kinetic mechanisms that involve large numbers of elementary steps. For this purpose, the application of a 3D Computational Fluid Dynamics (CFD) model simulations are proven to be very effective to support the experimental investigations of turbulent combustion.

Computational Fluid Dynamics (CFD) is a tool that can mathematically model the fluid flow in a combustion device and can give detailed insights into the pollutant formation process. It is crucial to use accurate CFD models to predict the emissions accurately. With the application of supercomputing, detailed chemistry can be utilized to predict the combustion process and resolve all the major species that are involved in combustion. Accurate computational models that consider the complex fluid flow and detailed chemistry are critical to help design and optimize clean and efficient combustion system.

In the open literature, many works has been made to study the combustion behavior when Methane-air used. However, this unique investigation that is, introduction of water content in the fuel to reduce the NO_x emission and soot formation together during combustion process followed by study the numerical variation of mass fraction of each species were carried out very certain. Thus this paper attempts to numerically investigate the combustion and emission in methane combustor. Emphasis is made to the prediction of pollutant emission, in particular NO_x and Soot.

II. COMBUSTION CFD SIMULATIONS

The first stage of the present study is to perform numerical simulations using a computational model (ANSYS Fluent), and reduced chemical mechanisms to model the chemical reactions. In present study numerical simulations were performed for non-premixed combustion. The combustor geometry and simulation details are given in the following sections.

A. Primary Classification of possible Combustion Simulation

1) Based on Mixing
- Non Premixed Combustion (Direct Injection)
- Premixed Combustion(Carburetor, Burner)
- Partially pre-mixed (Example: IC engine where ignition took place early but not very early)

2) Based on Phase
- Fluid Phase (Volumetric Reactions)
- Wall (Surface Reactions)
- Particles (Surface Reaction-Coal combustion)
- Porous Region (Example: After treatment System)

For example laminar flames, where fick's law is not valid and we need to adopt multi-component diffusion properties, which then cost us high too.

B. Modeling Species Transport and Reaction:

Fluent provides several modes for chemical species transport and chemical reactions. Fluent can model species transport with or without chemical reactions.

C. Species Transport Equation:

In CFD generally, we need to solve for
- Mass
- Momentum
- Energy Equations
- Turbulence Equation(K, K-epsilon etc)
- Additionally in combustion we have Species Transport Equation(i.e.)

In order to solve the conservation equation for chemical species, the local mass fraction of each species, Y_i, through the solution of a convection-diffusion equation for the i^{th} species shall be in under consideration. The conservation equation is:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho \vec{v} Y_i) = -\nabla.\vec{J_i} + R_i + S_i$$

Time derivative

Convection Motion of species because of the velocity field

Diffusion (Motion due to concentration gradient)

Sources (Reactions + user defined)

Where; Y_i = Mass fraction, and 'i' is No. of species; R_i is the net rate of production of species I by chemical reaction and S_i is the rate of creation by addition from the dispersed phase.

D. Approaches to Combustion Reaction Calculation:

ANSYS Fluent provides basically five approaches to the modeling gas phase reaction flows:

1) Finite Rate Chemistry Model:

It takes into account of chemical kinetics and calculate rate of production of each and every species to solve complete OD system which can cost high and the process is slow as well. This model is basically suitable for a wide range of applications including premixed, partially premixed, non-premixed turbulent combustion, and ignition delay in diesel engines.

2) Mixing based Model:

There are many classification of this model, which uses turbulence of the air and fuel mixer to ignite with the help of K, Epsilon models etc, that calculate time scale factor (for example: X sec) after which combustion starts during mixing. This type of phenomena is called as Turbulence Chemistry Interaction Model, in other words it can be called as mixed is burnt. It means that these model doesn't require ignition source and also faster, but require user experience (when to use these models).

This model is Eddy Dissipation Model which uses K and epsilon to calculate time for each reaction. This model is used when have inlet and assume a steady state

temperature distribution at outlet. Turbulence-chemistry interaction model, based on the Magnuseen and Hjertager called the eddy-dissipation model. The net rate of production of species i due to reaction r, R_{ir}, is given by the smaller (i.e., limiting value) of the two expressions below:

$$R_{i,r} = \upsilon'_{i,r} M_{w,i} A\rho \frac{\epsilon}{k} \min_R \left(\frac{Y_R}{\upsilon'_{i,r} M_{w,R}} \right)$$

$$R_{i,r} = \upsilon'_{i,r} M_{w,i} AB \rho \frac{\epsilon}{k} \frac{\sum PY_P}{\sum_j^N \upsilon''_{i,r} M_{w,j}}$$

Where, Y_P is the mass fraction of any product species, P
Y_R is the mass fraction of a particular reactant R
A is an empirical constant equal to 4.0
B is an empirical constant equal to 0.5

3) Finite Rate-Eddy Dissipation Model:

This is a combination of both finite rate chemistry and eddy dissipation model. It is used to prevent instantaneous burning. It is helpful when you need ignition delay, because in eddy dissipation model sometimes in a cell when Air-fuel mix combination immediately starts and if air-fuel mix ratio is good then even before mixing combustion takes place.

The rate of consumption or production of the species is given by following expression.

$$q_i = \left(\sum_{k=1}^{K} (a_{ki})[X_k] \right) \left(k_{fi} \prod_{k=1}^{K} [X_k]^{\upsilon'_{ki}} - k_{ri} \prod_{k=1}^{K} [X_k]^{\upsilon''_{ki}} \right)$$

Where, $\left(\sum_{k=1}^{K} (a_{ki})[X_k]\right)$ takes the value 2.40 or 15.40 or 2.00 which is show in above mechanism file, which acts a multiplier that enhance the reaction.

Calculation of k_{fi} of k_{ri}

$$k_{fi} = A_i \, T^{\beta_i} \exp\left(\frac{-E_i}{R_c T}\right)$$

Where, A = Pre Exponential factor; T = Operating Temperature; R_c = Characteristic Gas constant and E = Activation Energy

$$k_{ri} = \frac{k_{fi}}{K_{ci}} \qquad K_{ci} = K_{pi} \left(\frac{P_{atm}}{RT}\right)^{\sum_{k=1}^{K} \upsilon_{ki}}$$

$$K_{pi} = \exp\left(\frac{\Delta S_i^o}{R} - \frac{\Delta H_i^o}{RT}\right)$$

Where, S and H are the entropy and Enthalpy, which can be calculated from thermal data file

III. MATHEMATICAL MODELING

The reacting flow process that occur in the combustor can be described using the following steady state governing equations viz. Conservation of mass, a scalar transport equation for each one of the chemical species, conservation of momentum, and energy conservation. The steady conservation forms of the governing equations are written as follow:

A. Balance equation for Mass:

The total mass conservation equation (2.2.1) is unchanged compare to non reacting flow (combustion does not generate mass)

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \mu_j)}{\partial x_j} = 0$$

Where ρ denotes mixture density and μ_j the components of hydrodynamic velocity.

B. Balance equation for Momentum:

The balance equation for momentum is identical to the Navier-Stokes equation for non-reacting fluids:

$$\frac{\partial (\rho \mu_j)}{\partial t} + \frac{\partial (\rho \mu_j \mu_i)}{\partial z} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho f_j$$

Where i=1, 2, 3…; p is the pressure; τ_{ij} is the viscous stress tensor component and f_i the i^{th} component of the external force.

C. Balance equation for Species:

If the chemical reactions are not described by a simpler, approximate model, a balance equation has to be written and solved for each chemical species considered in the mixture.

$$\frac{\partial (\rho Y_k)}{\partial t} + \frac{\partial (\rho \mu_j Y_k)}{\partial x_i} = -\frac{\partial (\rho Y_k V_{kj})}{\partial x_i} + \omega'_k$$

With k=1……N_s, where N_s is the total number of species, V_{kj} is the component of the diffusion velocity of species k in the direction j and ω'_k the mass reaction rate of the species k per unit time and volume.

D. Balance equation for Energy:

One can derive multiple forms for the energy balance equation, depending on which variable is solved for (enthalpy, internal energy, total energy, etc.). If we chose to consider here the specific total energy e; the conservation equation can be written as:

$$\frac{\partial (\rho e)}{\partial t} + \frac{\partial (\rho \mu_j e)}{\partial x_j} = -\frac{\partial q_i}{\partial x_j} + \frac{\partial (\sigma_{ij} \mu_i)}{\partial x_j} + q_s + \rho \sum_{k=1}^{N_s} \left[Y_k f_{k,j} \left(u_j + \mathbf{V_{kj}} \right) \right]$$

Where q_j is the j^{th} component of the heat flux; vector q_s is a heat source term corresponding to an external energy source (introduced for example to model spark ignition), $\boldsymbol{\sigma_{ij}}$ the stress tensor and f_i the j^{th} component of the body force. The last term for this equation correspond to the power produced by (possibly different) volume forces f_x acting on species k.

E. Turbulence Model

It is a fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of the problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for one application, one needs to understand the capabilities and limitations of the various options.

In this study two closure approaches were used; the k - ε turbulence model and k - ε RNG (Re-Normalization Group) turbulence model.

F. Transport Equation for the RNG k-ε Model:

The RNG k - ε model has a similar form to the standard k - ε model.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k \mu_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff} \frac{\partial \epsilon}{\partial x_j}\right) + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} - R_\epsilon + S_\epsilon$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as described above. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_m represents the contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ϵ are the inverse effective prandtle numbers for k and ϵ, respectively. S_k and S_ϵ are user defined source terms.

G. Species Transport Modeling

The evolution of the flow composition in the system is modeled by the species transport equation. FLUENT software solves N-1 partial differential equations, where N is the total number of chemical species included in the simulation. The steady state species transport equation is shown as:

$$\nabla.(\rho \vec{v} Y_i) = -\nabla.J_i + R_i$$

Where; Y_i = mass fraction, and 'i' is No. of species; R_i is the net rate of production of species i by chemical reaction and J_i is the Mass Diffusion Flux.

Mass Diffusion Flux is modeled using the Fick's law including turbulent mass fluxes

$$\vec{J_i} = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right)\nabla Y_i$$

Where Sc_t is the turbulent Schmidt number $\frac{\mu_t}{Sc_t}$ =0.7, μ_t is the turbulent viscosity and D_t is the turbulent diffusivity.

$$\nabla.\sum_{i=1}^{n}[h_i \vec{J_i}] \qquad Le_i = \frac{k}{\rho c_p D_{i,m}}$$

H. Approaches to Combustion Reaction Calculation:

Above rate calculation depends on the turbulence-chemistry approach used. The selection of the model also depends on the kind of chemical mechanism used, for example one step global reaction mechanism or multi-step reaction and if the combustion is premixed or non-premixed. Premixed combustion refers to the case where fuel and oxidizer are mixed before entering the combustor whereas the non-premixed refer to the opposite case. The current study presents two turbulence chemistry models were selected.

1) Finite Rate-Eddy Dissipation Model:

This is a combination of both finite rate chemistry and eddy dissipation model. It is used to prevent instantaneous burning. It is helpful when you need ignition delay, because in eddy dissipation model sometimes in a cell when Air-fuel mix combination immediately starts and if air-fuel mix ratio is good then even before mixing combustion takes place. The rate of consumption or production of the species is given by following expression.

$$q_i = \left(\sum_{k=1}^{K}(a_{ki})[X_k]\right)\left(k_{fi}\prod_{k=1}^{K}[X_k]^{\upsilon'_{ki}} - k_{ri}\prod_{k=1}^{K}[X_k]^{\upsilon''_{ki}}\right)$$

Where, $\left(\sum_{k=1}^{K}(a_{ki})[X_k]\right)$ takes the value 2.40 or 15.40 or 2.00 which is show in above mechanism file, which acts a multiplier that enhance the reaction. The reaction rate constant k_{fi} and k_{ri} are calculated using the Arrhenius equation.

$$k_{fi} = A_i\, T^{\beta_i}\, exp\left(\frac{-E_i}{R_c T}\right)$$

Where, A = Pre Exponential factor; T = Operating Temperature; R_c = Characteristic Gas constant and E = Activation Energy

These above variables are obtained from experimental measurements or from theoretical values from ab-initio calculations. The equation of rate of consumption, the rate is composed of forward and reverse reaction rates, and the reverse rate is usually obtained from the following expression.

$$k_{ri} = \frac{k_{fi}}{K_{ci}} \qquad K_{ci} = K_{pi}\left(\frac{P_{atm}}{RT}\right)^{\sum_{k=1}^{K}\upsilon_{ki}}$$

$$K_{pi} = exp\left(\frac{\Delta S_i^o}{R} - \frac{\Delta H_i^o}{RT}\right)$$

Where, S and H are the entropy and Enthalpy, which can be calculated from thermal data file.

2) Eddy Dissipation Model:

This model uses K and epsilon to calculate time for each reaction. This model is used when have inlet and assume a steady state temperature distribution at outlet. Turbulence-chemistry interaction model, based on the Magnuseen and Hjertager called the eddy-dissipation model. The net rate of production of species i due to reaction r, R_{ir}, is given by the smaller (i.e., limiting value) of the two expressions below:

$$R_{i,r} = \upsilon'_{i,r}\, M_{w,i}\, A\rho\frac{\epsilon}{k} min_R\left(\frac{Y_R}{\upsilon'_{i,r}\, M_{w,R}}\right)$$

$$R_{i,r} = \upsilon'_{i,r}\, M_{w,i}\, AB\, \rho\frac{\epsilon}{k}\frac{\sum PY_P}{\sum_j^N \upsilon''_{i,r}\, M_{w,j}}$$

Where, Y_P is the mass fraction of any product species, P

Y_R is the mass fraction of a particular reactant R

A is an empirical constant equal to 4.0

B is an empirical constant equal to 0.5

IV. CFD MODEL ASSUMPTION AND BOUNDARY CONDITION

A CFD solver package, ANSYS Fluent, was used to perform all the CFD computations; it allows evaluating the thermal and flowing fields based on continuity, momentum, and heat transfer equations. The calculation of flow and combustion uses pure methane of different composition. In reality however, the fuel is made up of several other species and this will be consider into simulation in an increasing complexity manner. The combustion model was based on eddy dissipation model which compute the rate of reaction under the assumption that the chemical kinetics are fast compared to the rate at which the reactant are mixed by turbulent fluctuations. The mixture fraction model involves

the solution for one or two conserved scalars (the Mixture fractions). In this approach, the transport equation for individual species is not solved. Instead, the individual component concentrations for the species of interest are divided from the predicted mixture fraction distribution. Physical properties of chemical species and equilibrium data are obtained from the chemical database. The concentration of turbulence and chemistry is derived with K- ∈ models of non-premixed type combustion. This model is computationally efficient because these model doesn't require ignition source and also faster, but require user experience (when to use these models). The flow is also assumed to follow the ideal gas behavior and the specific heat values are assumed to obey the mixing law. These assumptions are general for most gaseous combustion found the open literature.

A. Geometry

The combustion chamber used in this simulation is a cylindrical shape model with a small inlet for fuel (methane). The 3-D geometry of the model is shown below:

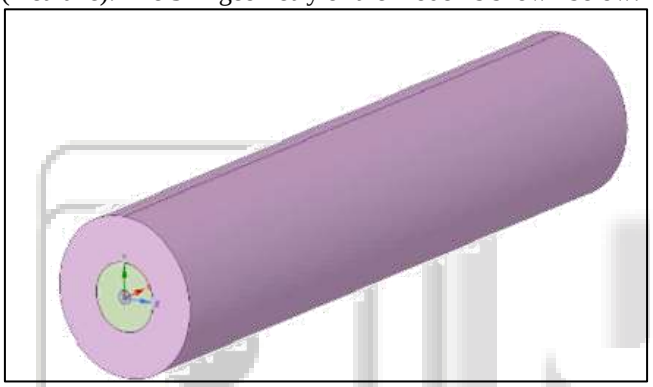


Fig. 1.:3D-Geometry Combustor Model

Fig. 2: Cut Section of Combustor Model

Ideally the entire cylindrical model should be analyzed. However, due to the limitations of the student version of ANSYS Fluent, the model is reduced to a 2-D axisymmetric model. The axisymmetric model assumes that the solution does not vary in the radial direction, which does not happen in practical scenarios but is a close approximation of the real phenomenon.

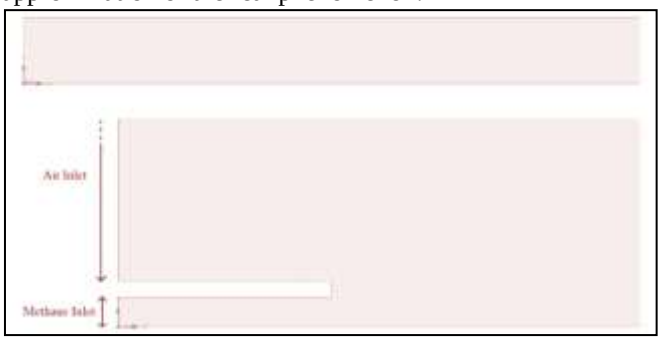


Fig. 3: 2-D Axisymmetric Model

B. Meshing

Element Type : Triangular
Capture Proximity : enabled
Element Size : 5e-3 m
No of Elements : 39141
No of Nodes : 20021

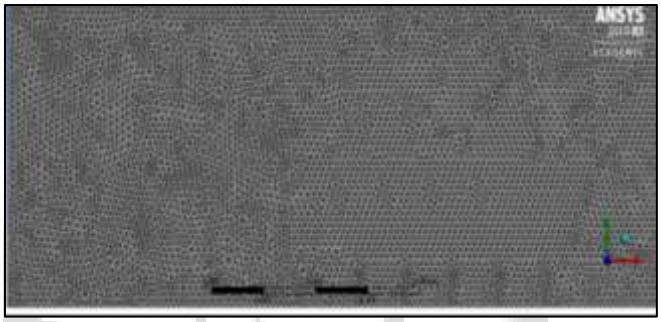


Fig. 4: Meshing of Combustor Model

C. SOLVING

1) Solver:

Type : Pressure-based
Time : Steady
2D Space : Axi-symmetric

2) Model:

Viscous : K-epsilon (standard)
Species : Species Transport
Reactions : Volumetric
Mixture Material : Case I- Methane+ Air
Case II - Methane+Air+2Step
Turbulence-Chemistry: Eddy Dissipation

3) Boundary Conditions:

	Vel. m/s^2	Temp. (K)	Mass Fraction (CH4)	Mass Fraction (O2)	Pressure (Pa.)
Air- inlet	0.5	300	Case 1: 0 Case 2: 0	Case 1: 0.23 Case 2: Parametric	-
Fuel Inlet	80	300	Case 1: 1 Case 2 : Parametric	Case 1: 0 Case 2 : Parametric	-
Outlet	-	300	-	-	0

Table 1

4) Chemical Equation:

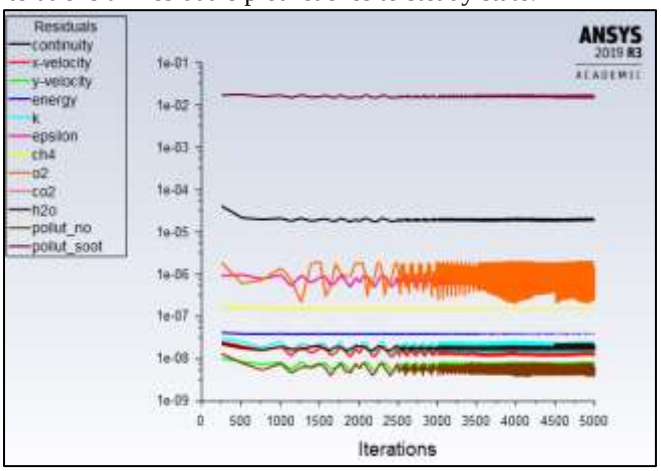
- $CH_4 + 2\ (O_2 + 3.76\ N_2) \rightarrow 2CO_2 + 2H_2$
- $O_2 + 3.76\ N_2$, Constitutes the air mixture.
- By stoichiometric calculations, the mass fractions of Oxygen and Di-Nitrogen gas are 0.23 and 0.77 respectively.
- Di-Nitrogen is inert in the above reaction and hence it is excluded from the calculation.

V. RESULT AND DISCUSSION

The developed combustion CFD simulation is not validated until it has been demonstrated that it predicts experimental

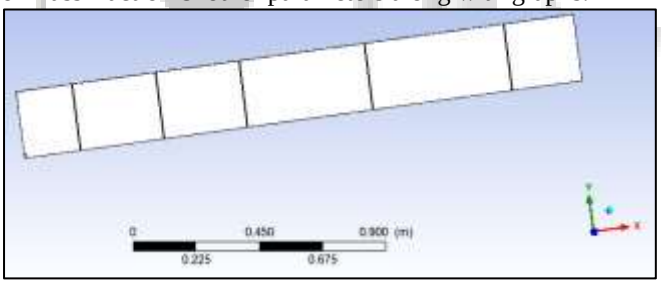
data fairly well. As a result, the purpose of this section is to show that the simulation is capable of predicting pollutant emission reduction when subsequent amount of water content added to the fuel without sacrificing power and fuel consumption.

After set up of all the conditions, as shown mentioned above, the solution is initialized and run for some iterations till residuals plot reaches to steady state.



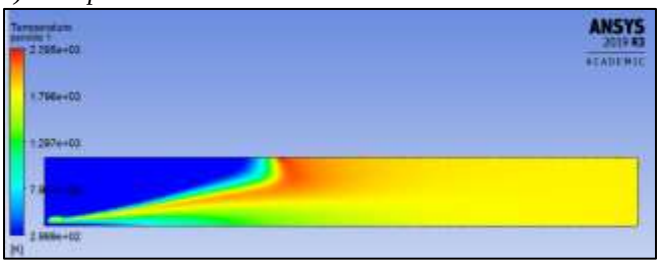
A. Results:

The results are observed on a plane with variation in mass fraction of species and other parameters, by plotting line probes across the plane along with graphs with respect to the line probes where Y distance is taken on the X-axis and the mass fraction, temperature or other parameters variation on Y-axis. The below figures representing the plane with line probes on it at varies points in X direction and the counters of mass fraction & other parameters along with graphs.

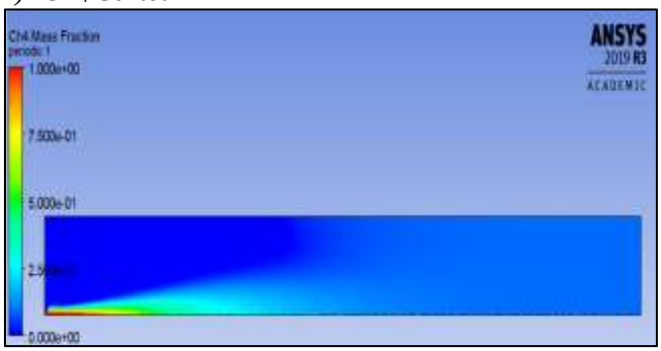


B. Case-I

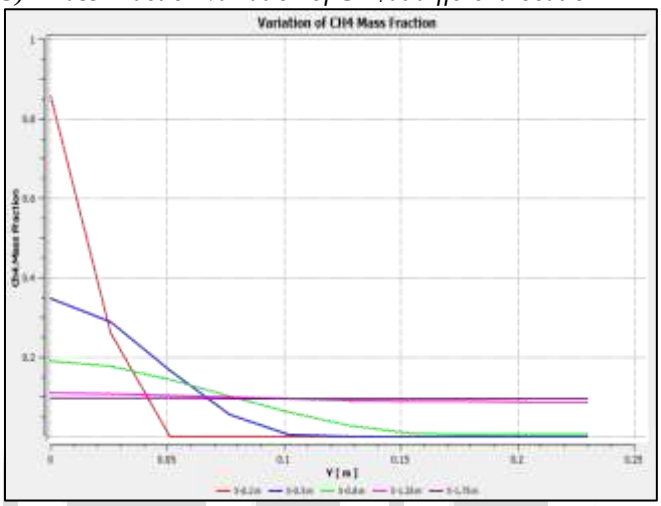
1) Temperature Contour



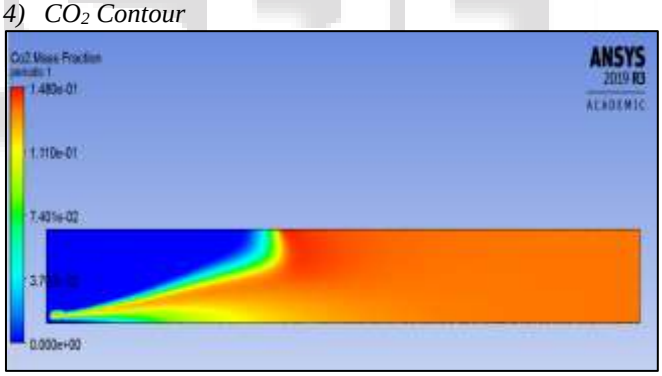
2) CH_4 Contour



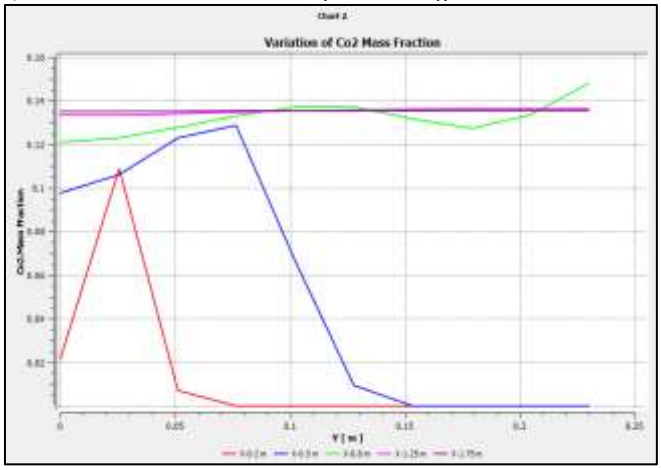
3) Mass Fraction variation of CH_4 at different location



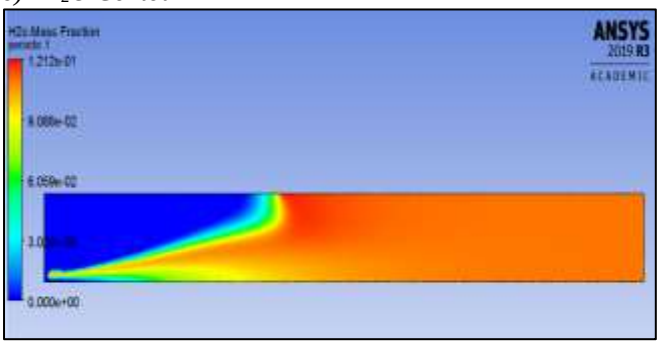
4) CO_2 Contour



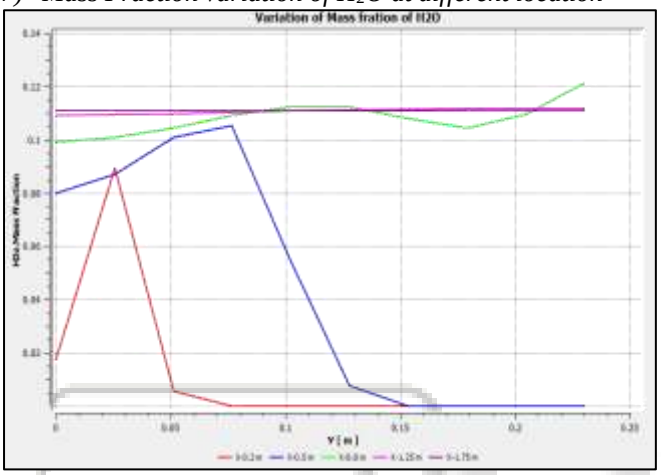
5) Mass Fraction variation of CO_2 at different location



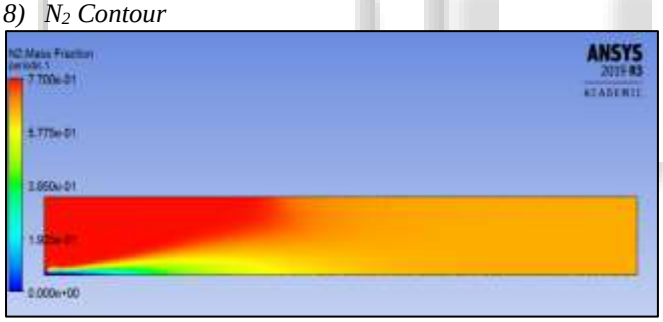
6) H_2O Contour



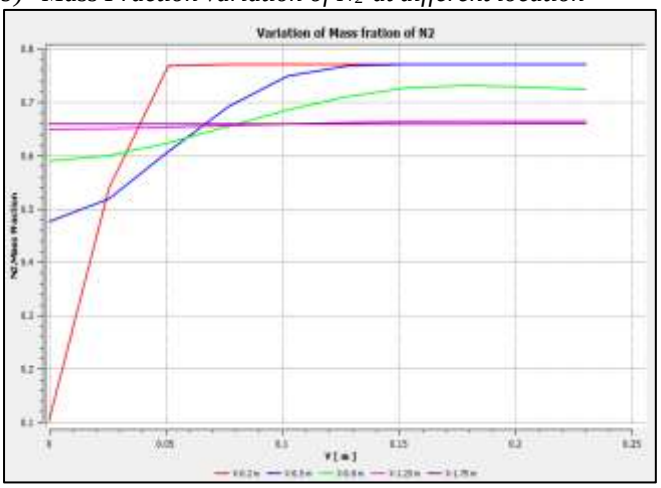
7) Mass Fraction variation of H_2O at different location



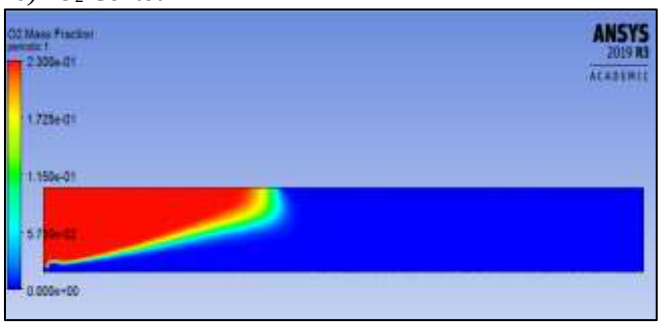
8) N_2 Contour



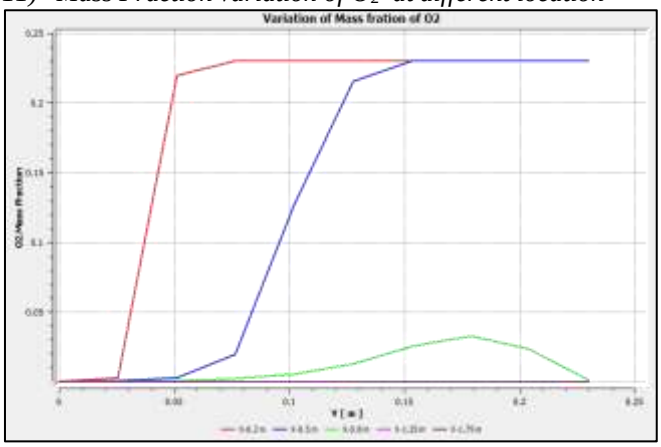
9) Mass Fraction variation of N_2 at different location



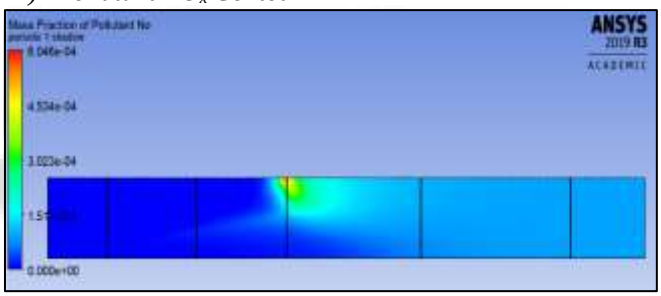
10) O_2 Contour



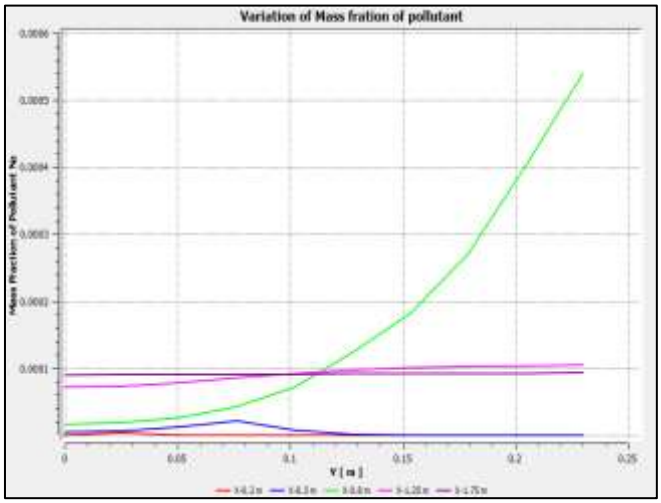
11) Mass Fraction variation of O_2 at different location



12) Pollutant NO_x Contour

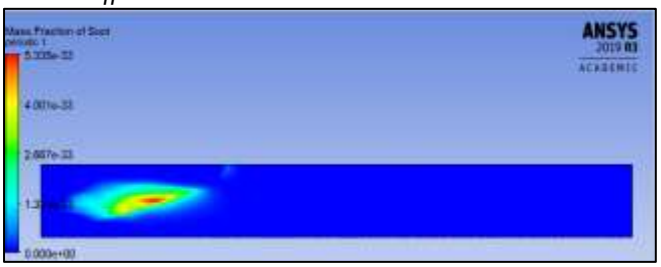


13) Mass Fraction variation of Pollutant NO_x at different location



C. Case-II Parametric Study

1) Soot Formation Contour and Mass Fraction variation of Soot at different location



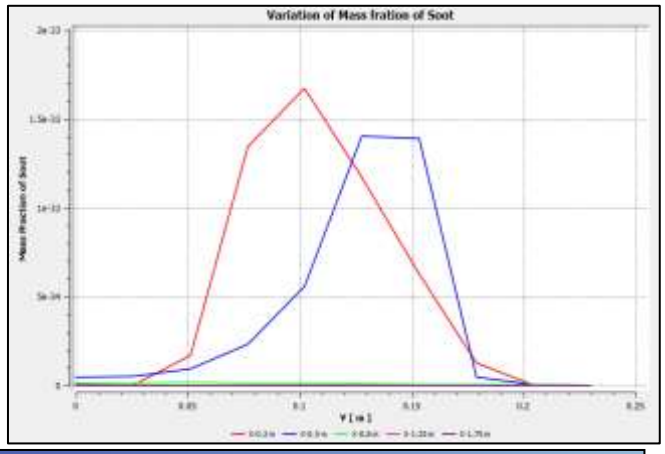
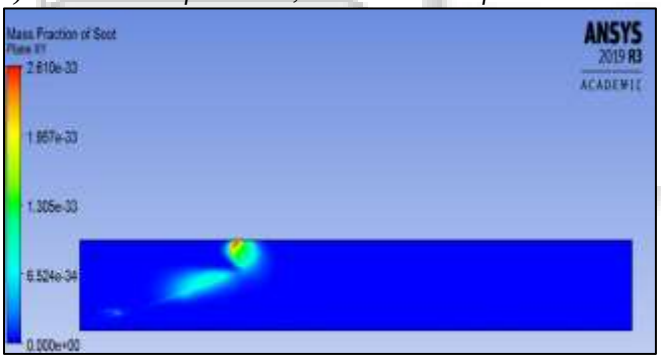


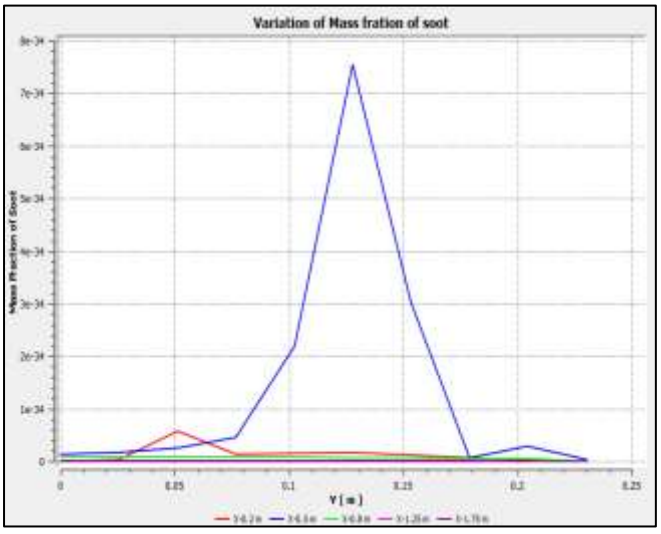
Table of Design Points

	A	B	C	D	E	F	G	H
1	Name	Update Order	P1 - parameter-1	P2 - parameter-2	P3 - report-temp-outlet-op	P4 - report-velocity-op	Retain	Retained Data
2	Units				K	m s^-1		
3	DP 0	1	1	0	1832.3	3.9935	☑	✓
4	DP 4	5	0.95	0.5	1903.6	4.1681	☑	✓
5	DP 5	6	0.8	0.2	1871.3	4.0612	☑	✓
6	DP 6 (Current)	7	0.7	0.3	1893.5	4.1296	☑	✓

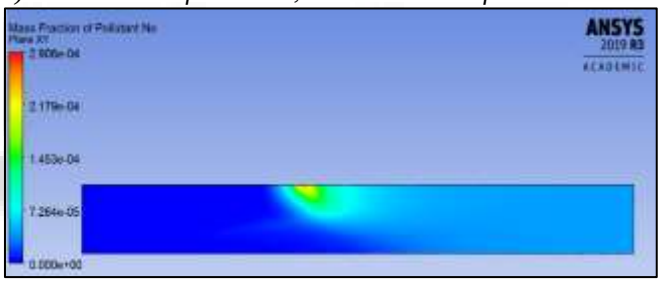
Note: 0% H2O addition results can be taken from case-I

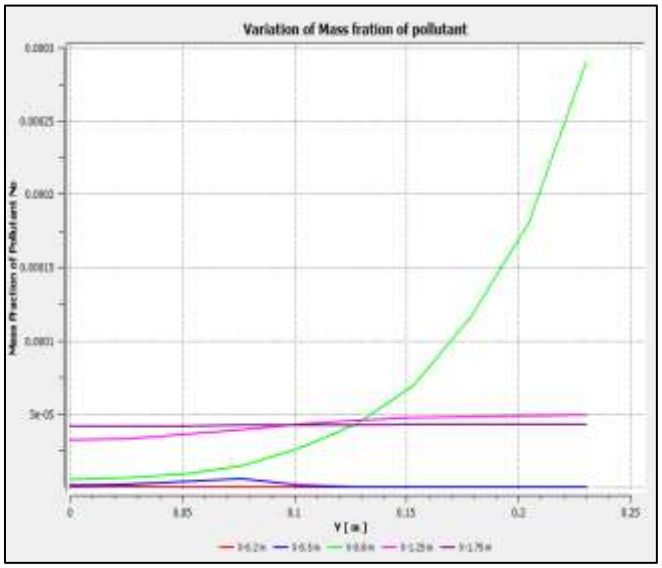
2) At Addition of 5% H2O, Mass Fraction of Soot.



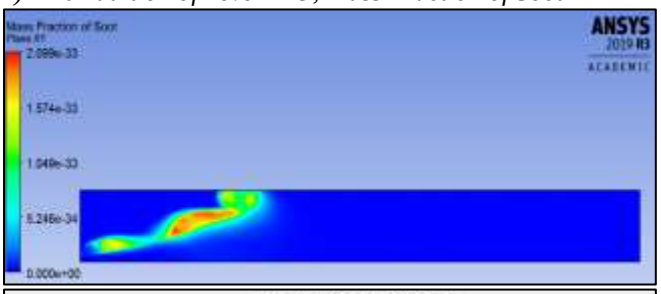


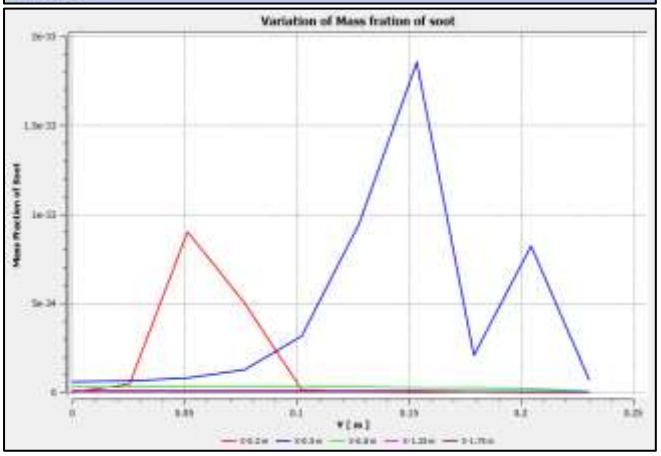
3) At Addition of 5% H2O, Mass Fraction of Pollutant



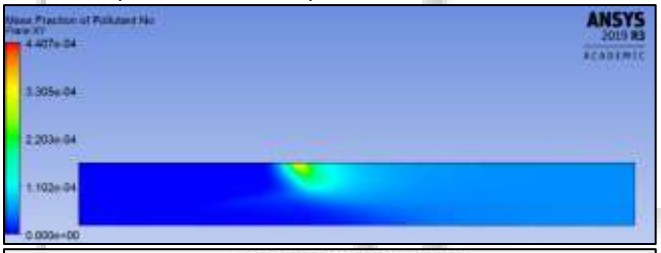


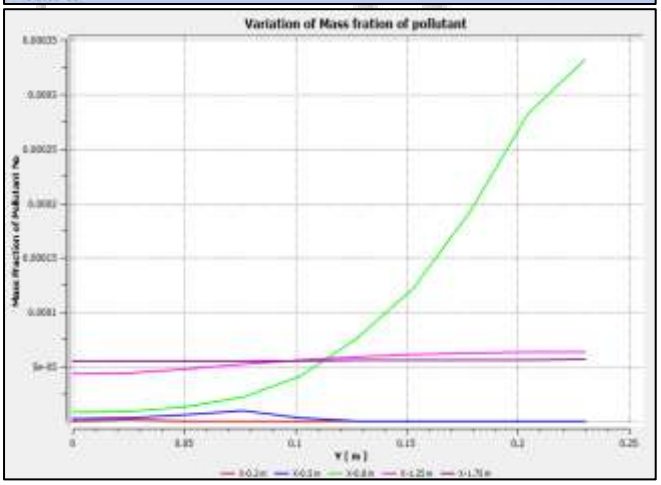
4) At Addition of 20% H2O, Mass Fraction of Soot



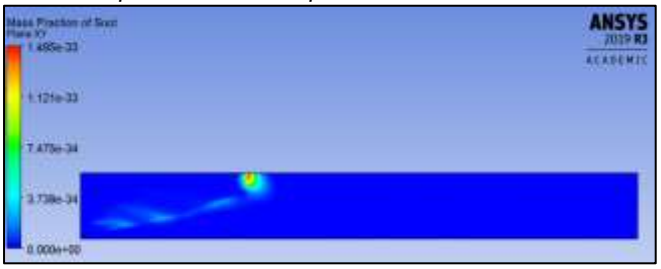


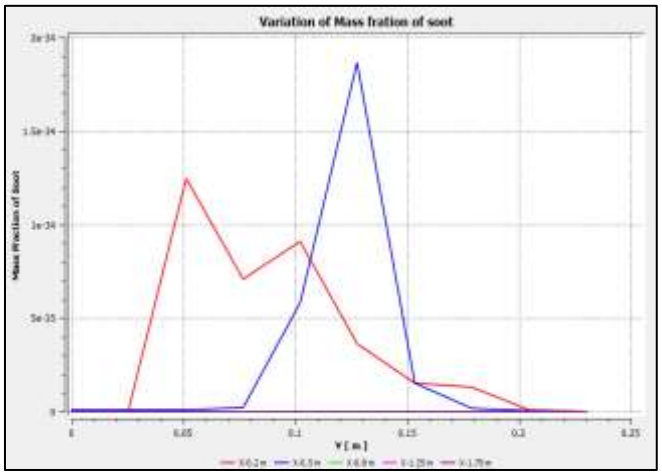
5) At Addition of 20% H_2O, Temperature Contour and variation of Mass Fraction of Pollutant NO_x



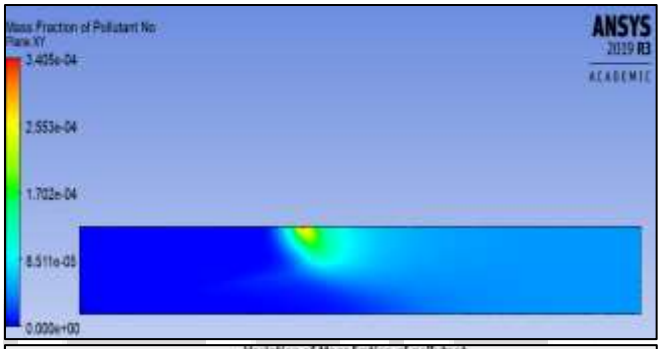


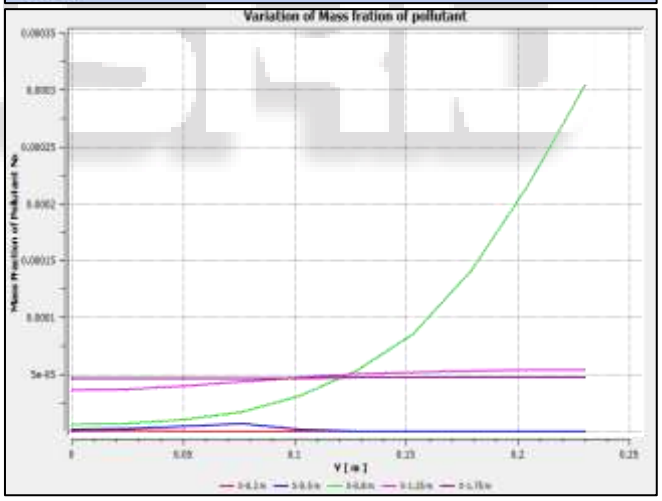
6) At Addition of 30% H_2O, Temperature Contour and variation of Mass Fraction of Soot



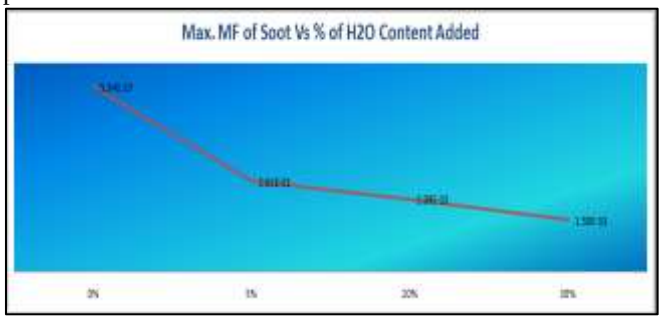


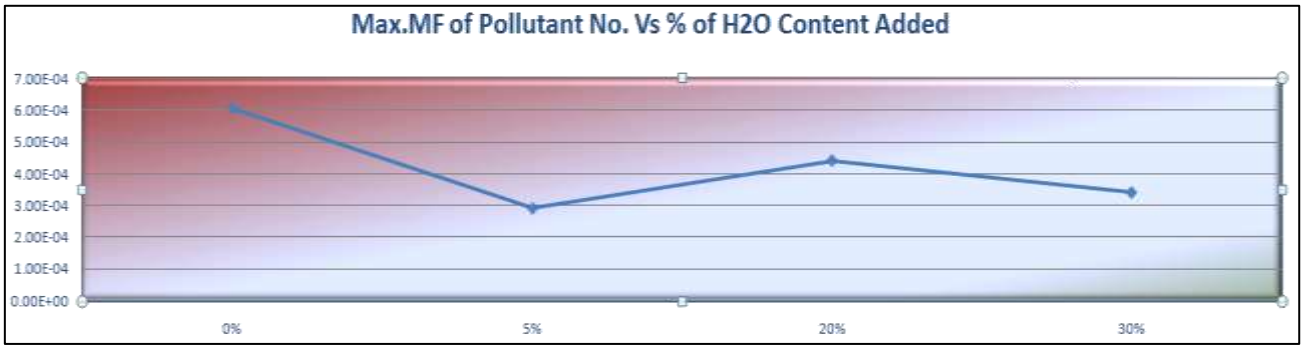
7) At Addition of 30% H_2O, Temperature Contour and variation of Mass Fraction of Pollutant NO_x at different location





Upon adding water to methane at fuel inlet, it has been seen a decrease in soot and pollutant No. in terms of mass fraction which can be noticed by plotting the graphs shown below by taking maximum value of MF soot and MF pollutant No into account.





VI. CONCLUSIONS

This study successfully demonstrated, Combustion simulation on combustor model is performed in ANSYS Fluent. The temperature reaches during simulation is 2131K. While the mass fraction of CO_2, H_2O, NO_x, N_2 O_2 and CH_4 are successfully calculated. Finally it can conclude that addition of water in the fuel can reduce the soot formation and pollutant NO_x emission to some extent and on further addition it may affect the perforation of combustion too, from previous research data.

REFERENCES

[1] Coupling of chemical kinetics with CFD in a 3D engine model. By Hassan A. Mazi. University of Illinois at Urbana, Champaign, 2009
[2] CFD Analysis of combustion phenomena at high speed fuel inlet of Mach 0.9 for methane with and without bluff body. September 2010 http://www.resarchgate.net/publication/283568682
[3] CFD investigation of Pollutant Emission in Can-Type Combustor Firing Natural Gas, LNG and Syngas.2016. ser: Earth Environ. Sci.32 012073.
[4] Numerical modeling of pollutant emission in practical combustion systems using detailed chemical kinetics. By sujit Sukumaran, Iowa State university.http://lib.dr.iastate.edu/etd/13693
[5] Soot formation-an overview, by H. Gg. Wagner, institute fur Physikalische Chmie der University, west Germany.
[6] How to reduce emission of nitrogen oxides from marine diesel engine in relation to Anex VI of MARPOL73/78, by Dong Nam, world Maritime university dissertation 2000 http://commons.wmu.se/all_dissertations/298

