Numerical Simulation of Methane/Air Combustion with CO₂ Dilution in A Gas Turbine Combustor

S. R. Razavi*,§, S. Tabejamaat**
***Amirkabir University of Technology, Tehran, Iran
(§Seyed.reza.razavi@aut.ac.ir)

Abstract: In this paper, a numerical simulation of Methane/Air combustion within a model of gas turbine combustor is presented. The effect of CO₂ as diluent is investigated in primary and secondary zone. Eddy dissipation model for the turbulence and flamelet model for chemistry simulation used. Temperature distribution and NO fraction in flow affected due to decreased mount of O₂ in air flow. Results showed as the fraction of CO₂ increased in air fraction especially in primary zone, the amount of NO reduced significantly and flame length and maximum temperature decreased in combustor.

Keywords: Combustion, Dilution, Gas Turbines, NOₓ
1 Introduction

Nowadays, the optimization of combustion performance and the reduction of pollutant emissions require considerable research efforts from the gas turbine industry. The basic objectives in combustor design are to achieve easy ignition, high combustion efficiency and minimum pollutant emissions. Under such conditions two phenomena are to be controlled or avoided. First, the increase of NOx emissions due to the high levels of pressure and temperature needs to be restrained. Second, the feedback mechanism caused by the coupling between the flow and the flame may result in instabilities which originate in the periodic formation of inhomogeneous fuel pockets, the periodic shedding of large structures and the amplification of the acoustic waves.

Numerical methods are in that context very attractive alternatives to the expensive experimental set-ups required in such areas of research. Reynolds Averaged Navier Stocks (RANS) combustion simulation is widely used in academic researches. In other hand the Large Eddy Simulation which are offer numerous potential advantages and by solving for large flow structures while small scale effects are modeled. Eighty percent of eddies scale solved without any modeling and just 20% close to kolmogrov scale modeled. The implications on the numerical predictions are of importance but still need to be illustrated and validated in the context of industrial applications. For information and illustrations of the potential of each method, the reader is referred to results obtained with state-of-the-art RANS[8]- [12]and LES[13]- [18].

This study constitutes a step further toward the full demonstration of RANS for gas turbine engines with flamelet model.multiphase flows. It addresses the problem of dilution jets in an industry-like configuration.

The design of gas turbine combustion chamber is based on combined theoretical and empiricalapproach and the design of combustion chamber is a less than the exact science. A technicaldiscussion on combustion technology status and needs will show that the classic impediments that have hampered progress towards near stoichiometric combustion still exist. The process of combustor design has taken a new meaning over the past several years as three dimensionalcodes and other advanced design and validation tools have finally changed the approach from a“cut and burn” technique to a much more analytical process.

Mixing processes are of paramount importance in the combustion and dilution zones. In theprimary zone, good mixing is essential for high burning rates and to minimize soot and nitricoxide formation, whereas the attainment of a satisfactory
temperature distribution (pattern factor) in the exhaust gases is very dependent on
the degree of mixing between air and combustion products in the dilution zone. A
primary objective of combustor design is to achieve satisfactory mixing within the
liner and a stable flow pattern throughout the entire combustor, with no parasitic
losses and with minimal length and pressure loss. Successful aerodynamic
design demands knowledge of flow recirculation, jet penetration and mixing, and
discharge coefficients for all types of air admission holes, including cooling slots.
Good amount of literature is available on modeling of the process of combustion
for and hydrocarbon fuels ([11]-[27]). Charles K. Westbrook et al. [28] have
reviewed the progress in the field of computational combustion over last 50 years
encompassing 3D DNS and LES approaches. They have observed that many
commercial CFD codes use unstructured grid which offer the advantage of being
more suitable to massively parallel computing environment, as well as an ability to
deal with complex geometries.
The paper presents the design of tubular and annular combustion chamber followed
by three dimensional simulations in tubular and annular combustor with full film
cooling to investigate the velocity profiles, species concentration and temperature
distribution within the liner. The fuel under consideration is methane.
The computational approach attempts to strike a reasonable balance to handle the
competing aspects of the complicated physical and chemical interactions of the
flow and the requirements in resolving the three-dimensional geometrical
constraints of the combustor contours, cooling slots, and circular dilution holes.
The modeling employs non-orthogonal curvilinear coordinates, second-order
accurate discretization, multi-grid iterative solution procedure, the k-ε turbulence
model, and a combustion model comprising of an assumed probability density
function flamelet concept. The complicated mixing process can be better
understood with more detailed information supplied by the numerical simulation.
Accordingly, in present study an attempt has been made through CFD approach
using CFX 12 to analyze the flow patterns within the combustion liner and through
different air admission holes, namely, primary zone and dilution zone and wall
cooling, and from these the temperature distribution in the liner and at walls as well
as the temperature quality at the exit of the combustion chamber is obtained for
tubular and annular combustion chambers designed for gasturbine engine.
NOx emissions were calculated by the use of thermal and prompt models.
Computations of NOx formation rates and NOx concentrations were carried out
using a post-processor based on previously calculated velocities, turbulence,
Numerical Simulation of Methane/Air Combustion with CO₂ Dilution in A Gas Turbine Combustor

temperature, and chemical composition fields. To take into account turbulence fluctuations, a joint two-variable PDF was used in the calculation of the mean NOₓ formation rate.

2 Geometry

The can combustor is a feature of the gas turbine engine. Arranged around a central annulus, can combustors are designed to minimize emissions, burn very efficiently and keep wall temperatures as low as possible. The basic geometry is shown below with a section of the outer wall cut away.

Due to the fact that the fuel (methane) and oxidizer (air) undergo “fast” combustion (whereby the combustion rate is dominated by the rate of mixing of the
materials), the Finite Rate Chemistry model is not a suitable combustion model for the combustor in this paper. The Combined EDM/FRC model capability is a superset of the Eddy Dissipation model capability, and has no benefit over the Eddy Dissipation model in this case. In fact, the convergence behavior of the Combined EDM/FRC model may be worse than that of the Eddy Dissipation model.

The Eddy Dissipation model, the Laminar Flamelet model, and the Burning Velocity model are suitable for modeling “fast” combustion. The Burning Velocity model capability is a superset of the Flamelet model capability, with the extra capability of being able to handle premixed fuel/oxidizer. Because the combustor in this tutorial does not use premixed fuel/oxidizer, the extra capability of the Burning Velocity model is not required and therefore it is sufficient to use the Flamelet model.

The Eddy Dissipation model tracks each individual chemical species (except for the constraint material) with its own transport equation. This model is flexible in that you can readily add new materials, such as additional fuels, to the simulation without complications. A limitation of this model is that radical or intermediate species, such as CO, cannot be calculated with adequate accuracy. This may lead to over-prediction of flame temperature, in particular in fuel-rich regions.

The Laminar Flamelet model can simulate the products of incomplete combustion; for this reason, it generally provides a more accurate solution than the Eddy Dissipation model. One drawback of the Flamelet model is that it requires the availability of a flamelet library suited for the required fuel/oxidizer combination over the pressure and temperature ranges of interest.

The NO models used in the two parts of this tutorial are very similar. The only difference is in how the O-radical concentration is obtained for the 'Thermal NO' formation step:

- Eddy Dissipation model: The O-radical is not a component of the mixture; instead, its concentration is estimated using the O\textsubscript{2} concentration and temperature.
- Flamelet model: The O-radical concentration is calculated from the Flamelet library, where its concentration information is directly available.

### 3 Mesh Generation

The grid is fully unstructured and composed of tetrahedral. To control the total number of cells and reduce computational cost, mesh refinement is enforced where necessary. More specifically, injection areas, wall regions and jet trajectories must
be sufficiently resolved to capture the appropriate range of length-scales. To facilitate refinement along the jet, the grid is fully unstructured and composed of tetrahedral. To control the total number of cells and reduce computational cost, mesh refinement is enforced where necessary. More specifically, injection areas, wall regions and jet trajectories must be sufficiently resolved to capture the appropriate range of length-scales. To facilitate refinement along the jet, the tetrahedral grid is used to discretize the geometry and 106651 is the number of nodes.

Figure 2- generated grid for combustor model

To study the independence of the results with respect to the grid size, several grids are used in reference case. It is seen the temperature distribution and the species fraction remain constant by increasing the grid over 106651 nodes. The near wall grids conformed and mesh density over whole geometry are the parameters which improved in mesh study.
Table 1- Grid independency study

<table>
<thead>
<tr>
<th>Case</th>
<th>Grid #1</th>
<th>Grid #2</th>
<th>Grid #3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>1330.05 [K]</td>
<td>1331.05 [K]</td>
<td>1331.14 [K]</td>
</tr>
<tr>
<td>HO2 Mass Fraction</td>
<td>0.0601247</td>
<td>0.0602047</td>
<td>0.0602053</td>
</tr>
<tr>
<td>CO2 Mass Fraction</td>
<td>0.0732185</td>
<td>0.0735185</td>
<td>0.0735195</td>
</tr>
<tr>
<td>NO Mass Fraction</td>
<td>1.68052e-006</td>
<td>1.58052e-006</td>
<td>1.50052e-006</td>
</tr>
</tbody>
</table>

In Table 1, three grids result presented, 50231, 106651 and 210467 are nodes of grids respectively. It is concluded the changes in parameters listed above is under 1% so the grid #2 is the basic mesh in the numerical cases.

4 Governing Equations

The governing equations including the continuity, momentum, energy and species conservation are as followed:

Continuity equation:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m
\]  

Momentum equation (with respect to i direction):

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \frac{\partial}{\partial x_i} \delta_{ij}
\]  

where

\[
\tau_{ij} = \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{2}{3} \frac{\partial u_i}{\partial x_i} \delta_{ij}
\]  

The energy equation is (with respect to i direction):

\[
\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} (u_i (\rho E + p)) = \frac{\partial}{\partial x_i} \left( k_{eff} \frac{\partial T}{\partial x_i} \right) - \sum_j h_j J_j + u_j (\tau_{ij})_{eff} + S_h
\]  

where
Numerical Simulation of Methane/Air Combustion with CO₂ Dilution in A Gas Turbine Combustor

\[ h_{j'} = \int_{T_{ref}}^{T} c_{p,j'} dT \quad h = \sum_{j'} m_{j'} h_{j'} \quad E = h - \frac{p}{\rho} + \frac{u_i^2}{2} \]  \hspace{1cm} (5)

The species conservation equation is:

\[ \frac{\partial \rho (\rho m_i)}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i m_i) = - \frac{\partial}{\partial x_i} J_{i,i} + R_{i'} + S_{i'} \]  \hspace{1cm} (6)

where

\[ J_{i',i} = -\left( \rho D_{i',m} + \frac{\mu_t}{S_{c_i}} \right) \frac{\partial m_i}{\partial x_i} \]  \hspace{1cm} (7)

And:

\[ R_{i'} = M_{i'} \sum_{k=1}^{N_p} v_{i',k} M_{i} AB \frac{\varepsilon}{k} \frac{\sum_{p} m_p}{\sum_{j} v_{j',k} M_{j'}} \]  \hspace{1cm} (8)

In the above equations, \( S_m \) is the mass added to the continuous phase, \( S_h \) includes heat of chemical reaction and \( S_i \) is the rate of creation by addition from the dispersed phase.

The turbulence model was chosen to be \( k-\varepsilon \). The constants of the \( k-\varepsilon \)model are taken from [1]. The interaction between turbulence and chemical reactions has been considered by using the eddy dissipation method [2] and [3].

The simple reaction (stoichiometric) between \( \text{H}_2 \) and \( \text{O}_2 \) is considered as the chemical reaction. The governing equations are solved based on a finite volume approach.

5 Numerical scheme

An algebraic multi-grid method is applied to the coupled linearized discrete equations arising from an implicit pressure based finite volume discretization system of equations. The whole methodology is modeled by using CFX solver. Each simulation normally takes about 9 CPU hours.
The solution convergence is judged according to the residuals of governing equations. The results reported in this paper are based on the criteria that the residual of each equation should be smaller than $1.0 \times 10^{-6}$.

### Table 2- Fluid model characteristics

<table>
<thead>
<tr>
<th>Fluid Models</th>
<th>Material</th>
<th>Methane Air Mixture</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Pressure</td>
<td>1 [atm]</td>
<td></td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>Thermal Energy</td>
<td></td>
</tr>
<tr>
<td>Combustion</td>
<td>Eddy Dissipation</td>
<td></td>
</tr>
<tr>
<td>EDM Coeff. B</td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>Thermal Radiation</td>
<td>P 1</td>
<td></td>
</tr>
<tr>
<td>Component Models &gt; N2</td>
<td>Constraint</td>
<td></td>
</tr>
<tr>
<td>Option</td>
<td>Reacting Mixture</td>
<td></td>
</tr>
<tr>
<td>Material Group</td>
<td>Gas Phase Combustion</td>
<td></td>
</tr>
<tr>
<td>Reactions</td>
<td>Methane Air Single Step Reaction</td>
<td></td>
</tr>
</tbody>
</table>

### 5.1 Boundary conditions

Three inlet conditions and one outlet shown in Figure 3. Boundary and initial conditions dealing with a fully compressible flow solver, For clarity, the set of BC’s used in the RANS is listed in Table 3, Table 4 and Table 5.

### Table 3- Fuel inlet boundary

<table>
<thead>
<tr>
<th>Boundary Details</th>
<th>Normal Speed</th>
<th>40 [m s⁻¹]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static Temperature</td>
<td>300 [K]</td>
<td></td>
</tr>
<tr>
<td>Component Details</td>
<td>CH4</td>
<td></td>
</tr>
<tr>
<td>Mole Fraction</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>
The mass fraction of CO2 in air inlet (main and dilution oxidizer injectors) varies in cases. Six different cases studied. CO\textsubscript{2} as diluent added to oxidizer injector and
mole fraction 0.10, 0.15 and 0.20 varies to investigate the effects in main and dilution injectors.

6 Results

![Velocity profiles in six different cases](image)

The critical study of flame structure reveals that the present design offers narrow flame with maximum temperature in primary zone being of the order of 1300 K. H. The intense mixing and formation of re-circulation zone at the core of primary zone are responsible for short and narrow flame structure as is observed through flow visualization (Figure 8) in present study.

Similar trends for velocity It is important to note that the pressure drop across the combustion chamber is around 10% of the inlet pressure which is quite tune with the published results for tubular combustion chamber.

In Figure 5, Figure 6 and Figure 7, 10 cases are demonstrated. Case 1 corresponds to no dilution case, case number 2 - 4, 5-7, 8-10 related to dilution at primary, secondary (dilution) and both zones respectively.
The fluid flow in the tube can be mainly divided into back flow zone, mixing zone and trailing fluid zone. This fluid flow texture ensures the proper flow field in the combustor and affects the air distribution through different air admission holes. High levels of radial and tangential turbulence were observed in the central core region throughout the combustor. High losses are encountered in the dump region and where the jets bend towards the outer and inner annuli. The design of these zones needs to be taken utmost care to reduce the losses in the combustor. The flow behavior in the combustor, including the flow back zones, eddies, and their size and intensity, will apparently control the combustion behavior and then the performance of the engine. High velocity from primary and dilution air admission holes of the order of 10 m/s is witnessed.

The formation of counter rotating vortices from the second row of primary zone ensures cutoff of flame.
It is shown in Figure 6 that the NO mass fraction in case number are minimum, which the dilution is 0.15 mole fraction for CO\textsubscript{2}. However temperature profile (Figure 8) and averaged temperature in outlet are similar to NO formation. Apparently CO\textsubscript{2} dilution decreases the amount of pollutant but the reduction in temperature in case #4 and #10 in which the primary zone has 0.01 mole fraction (N\textsubscript{2} mole fraction in all cases is constant to 0.79) is lower than other cases.
Turbine inlet temperature (TIT) is the most important parameter in gas turbine engine performance. Except material limitation as the TIT increased the efficiency of whole cycle falls up so the basic goal in dilution aspect should be keeping TIT (combustor outlet temperature) constant to has the constant efficiency. The case # has the best pollutant in outlet but as the dilution is 0.15 CO₂ the temperature reduction is not order of other cases such as .it is very worthy while the NO fraction decreased by diluting the temperature (important parameter in gas turbine engine) remains approximately constant.

The flow from the pre-diffuser inlet to the combustor exit can be mainly divided into three zones, namely back flow zone, mixing zone and trailing flow zone. In the first zone or back flow zone the air leaves the swirl cone to the dome at high swirling velocity. After that, the swirling mixture of the air and fuel flows forward and entraps the air in the center area of the liner, and at the same time, the downstream air refills the region. As a result, a counter-rotating vortex pair (CRVP) forms in the central region of combustor liner. Corresponding to a pair of large eddies in this zone, there should be two symmetrical small eddies in the corners of the liner. In the downstream area of the back flow zone, the outside mixture further mixes with the fresh air injected from the dilution holes distributed on the liner wall. The injection flows retard the revolution of the mixed gas and restrain the anterior reverse flow zone. As a result, the back flow zone is cut off by the first pair of the dilution holes and the combustion should mainly occur in the backflow zone under very hot condition, i.e., primary zone. Although a small amount of the mixed
gas is inhaled into the primary zone, most gas flows downstream and mixes with the cold air from the second pair of the dilution holes. This makes the flow in the combustor more uniform.

![Figure 9](image9.png)

**Figure 9** - Temperature profile in combustion chamber with 0.10 mole fraction CO\(_2\) in First air zone

Decreasing dilution in the combustion chamber (comparison between Figure 9 and Figure 10) reduces the flame width and maximum temperature. The effect of secondary air inlet is more significant in higher amount of CO\(_2\) dilution.

![Figure 10](image10.png)

**Figure 10** - Temperature profile in combustion chamber with 0.15 mole fraction CO\(_2\) in First air zone
Numerical Simulation of Methane/Air Combustion with CO₂ Dilution in A Gas Turbine Combustor

Figure 11- Temperature profile in combustion chamber with 0.2 mole CO₂ dilution in first air zone

Figure 12- Temperature profile in combustion chamber 0.1 mole fraction of CO₂ in second air zone

Temperature profile in cases without dilution and dilution (Figure 8 with Figure 12 through Figure 14) in secondary zone has similar contour. It is due to low mass flow rate in secondary zone.
Figure 13 - Temperature profile in combustion chamber with 0.15 mole fraction CO$_2$ in second air zone

Figure 14 - Temperature profile in combustion chamber with 0.2 mole fraction CO$_2$ in second air zone

7 Conclusion
In the present study, a numerical approach to investigate CO$_2$ dilution effect in a gas turbine combustor is presented. A model combustor with two section and 6 dilution holes are the basis of the study. 10 different cases which diluent fraction varies are perspective. The temperature profile across the combustion chamber and
species fraction are the parameters which described in detailed. Several wealthy observations and conclusions are obtained:

- Dilution in secondary zone has no significant effect on flame length in the combustor but the amount of NO as pollutant is lower than other cases.
- Temperature profile decreases higher as the diluent increases. In the case #5 (CO₂ mole fraction at 0.15 in secondary zone) the maximum temperature in outlet is equal to the case without dilution.
- Flame width in the primary dilution has bigger size rather than the dilution in secondary zone and this effect higher averaged temperature in whole combustor.

**Nomenclature**

- **P** Pressure
- **EDM** Eddy Viscosity Method
- **Z** Axial length
- **T** Temperature
- **TIT** Turbine Inlet Temperature
- **φ** Equivalence ratio
References


Numerical Simulation of Methane/Air Combustion with CO₂ Dilution in A Gas Turbine Combustor


