

# Numerical Simulation of Non-Premixed Turbulent Methane-Air Combustion.

O.A Erinfolami, O.S Ismail, O. Oyewola

**Abstract:** This paper examined the heat-transfer coefficients and the Nusselt number for turbulent flow for a methane-air flame occurring in cross-flow streams in an open duct. The governing equation of momentum, energy, mass fraction and radical nuclei concentration were solved in a fully coupled fashion at each control volume. The model encompasses conservation equations of gas components participating in the process and energy equation. A FORTRAN code was written with a Tridiagonal matrix solver (TDMA) used to obtain the temperature, velocity and particulates profiles on the flat plate. The Nusselt number for the flow was calculated and plotted as a function of the calculated Rayleigh number. It was observed that a curve obtained with temperature and pressure far from the critical region approaches the line obtained with a classic correlation.

**Index Terms**— Computation Fluid Dynamics, Eddy dissipation model, Large eddy simulations, Non-Premixed combustion, Probability Density Functions, Reynolds stress model, Reynolds Averaged Navier-Stokes, Turbulence,.

## 1 INTRODUCTION

Nearly all methods of engineering power production involve fluid flow and heat transfer as essential processes.

The same processes governed the heating and air-conditioning of buildings. Major segments of the chemical and metallurgical industries use components such as furnaces, heat exchangers, condensers, and reactors (nuclear), where thermo-fluid processes are at work. Aircrafts and rockets movements owe much too to fluid flow, heat transfer, and chemical reactions. In the design of electrical machinery and electric circuits, heat transfer is also often the limiting factor. The pollution of the natural environment is largely as a result of heat and mass transfer activities. Similarly are storms, floods, fires and some other natural events. Due to change in weather conditions, the human body resorts to heat and mass transfer for its temperature control.

Combustion is one of the oldest technical processes, and is an everyday phenomenon. It is the dominant mode of energy conversion. Today, combustion related researches are driven mainly by three objectives: Obtaining maximum efficiency, with an eye on the limited supply of fossil fuels; minimizing pollution to preserve the environment and achieving safe and stable operation of machinery and equipments.

The main challenge for combustion researchers, however, is to accomplish all these goals simultaneously. Turbulent non-premixed flames exhibit phenomenon found in other turbulent fluid flows. In some circumstances a thin flame sheet forms a connected but highly wrinkled surface that separates the reactants from the products. This flame surface is convected, bent, and strained by the turbulence and propagates (relative to the fluid) at a speed that can depend on the local conditions (surface curvature, strain rate, etc.). Typically, the specific volume of the products of combustion is seven times that of the reactants, the flame surface being a volume source. Because of this large volume source there is a pressure field associated with the flame surface that affects the velocity field and hence indirectly affects the evolution of the surface itself.

Looking at the detailed structure of a turbulent non-premixed flame, we can examine mean quantities. Within the

flame there is a mean flux of reactants due to the fluctuating components of the velocity field. Contrary to normal expectations and observations in other flows, it is found that this flux transports reactants up the mean-reactants gradient, away from the products (hence counter gradient diffusion).

A second notable phenomenon is the large production of turbulent energy within the flame: Behind the flame the velocity variance can be 20 times its upstream value. Both of these phenomena result from the large density difference between reactants and products and from the pressure field due to volume expansion. There is a wide variety of theories and models for premixed and non-premixed turbulent flames. More ambitious are the probabilistic field theories that attempt to calculate statistical properties of a flame as functions of position and time. In non-premixed flames, also called diffusion flames, the fuel and the oxidizer are mixed during the combustion process itself. Both premixed and non premixed flames can further be classified as either laminar or turbulent depending on the regime of gas flow.

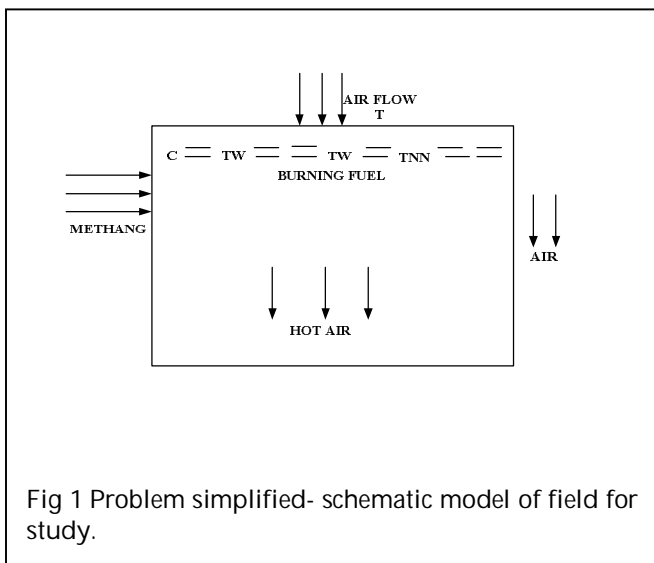
There is an extensive literature describing the development of modeling techniques suitable for large-scale engineering simulations. Early studies of this type using single step chemistry were performed by [1], [2]. More recently, [3] have performed direct numerical simulations of turbulent, premixed hydrogen flames in three dimensions with detailed hydrogen chemistry. Bell et al. [4] have performed similar Numerical Simulation of Premixed Turbulent Methane Combustion. Three different statistical techniques use in turbulent modeling are the Large Eddy Simulations (LES), Reynold Average Navier-Stokes Simulations (RANS) and Probability Density Function (PDF) transport equations model [5]. In LES, the volume-averaging of the governing equations is considered, but will require large computational resources for routine simulations. It is however considered to be a promising tool. RANS technique solves the governing equations by modeling both the large and the small turbulent eddies, taking a time – averaged of the variables being considered. The PDF transport equation model consist of a procedure considering the proba-

bility distribution of the relevant stochastic quantities directly by means of probability density functions.

## 2.0 MODEL.

### 2.1 Field Model.

The schematic model of turbulent diffusion flame is as presented in Fig 1. The overall effects of flame in the flow field has to be reduced to generation of heat and the dispersal of heat and burnt particules by either natural air or draughted air towards an object for heating. To carry out detail analysis of actions reflecting occurrences in the flow field a model of the flow field employing time averaged Navier–Stokes equations.



### 2.2 Flame configuration and Field Definition.

For a two-dimensional parabolic flow, when a steady two-dimensional flow has one-way space coordinate, it is called a two dimensional flow. Most flame flow has a predominant velocity in one way coordinate and hence the convection always dominates the diffusion in that coordinate. It is this feature that imparts the one way character to the stream wise direction. No reverse flow in this direction is acceptable. Examples of two-dimensional parabolic flow are plane or axisymmetric cases of boundary layers on walls, duct flows, jets, wakes, and mixing layers. The solution for such situation is obtained by starting with known distribution of the variable,  $\Phi$ , at upstream station and marching in the streamwise direction. For every forward step, the distribution of  $\Phi$  in the cross stream coordinate is calculated at on streamwise station. Thus computationally only one-dimensional problem needs to be handled, for which TDMA can be employed as solvers to the discretized equations of these variables  $\Phi$ .

### 2.3 Governing Equations.

Employing a conservative finite – volume methodology requires all the governing equations to be expressed in a conservative form in which tensor notation is generally employed. The basic governing equations are for the conservation of mass, momentum and energy.

#### 2.3.1 Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \quad (1)$$

Where  $u_j$  is the  $j^{\text{th}}$  Cartesian component of velocity and  $\rho$  is the fluid density. ( $j = 1,2,3$ )

#### 2.3.2 Momentum equation

$$\frac{\partial}{\partial t} (\rho u) + \frac{\partial}{\partial x_i} (\rho u_i u_i) = \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho f_j \quad (2)$$

Where  $p$  is the static pressure,  $\tau_{ij}$  is the viscous stress tensor and  $f_j$  is the body force. For Newtonian fluids  $\tau_{ij}$  can be expressed as:

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

Where  $\mu$  is the fluid dynamic viscosity and  $\delta_{ij}$  is the Kronecker delta

#### 2.3.3 Energy Equation

The equation for the conservation of energy can take several forms. The static enthalpy form of the energy equation can be expressed as:

$$\frac{\partial}{\partial t} (\rho h) + \frac{\partial}{\partial x_j} (\rho u_j h) = -\frac{\partial q_j}{\partial x_j} + \frac{\partial p}{\partial t} + u_j \frac{\partial p}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \frac{\partial}{\partial x_j} (J_{mj} h_m) \quad (4)$$

Where  $J_{mj}$  is the total (concentration – driven + temperature – driven) diffusive mass flux for species  $m$ ,  $h_m$  represents the enthalpy for species  $m$ , and  $q_j$  is the  $j$  – component of the heat flux.  $J_{mj}$ ,  $h_m$  and  $h$  are given as:

$$J_{ij} = -\rho D \frac{\partial Y_m}{\partial X_j} \quad (5)$$

$$h_m = \int_{T_o}^T C_p m(T) dT + h^o f_m \quad (6)$$

$$h = \sum_{m=1}^n Y_m h_m \quad (7)$$

$D$  is the diffusion coefficient,  $C_p$  is the constant – pressure specific heat, and  $h_f^o$  is the enthalpy of formation at standard conditions ( $P_o = 1.atm., T_o = 298K$ ).

The Fourier's law is employed for the heat flux:

$$q_j^n = -K \frac{\partial T}{\partial x_j} \quad (8)$$

#### 2.3.4 Mixture Fraction

$$\frac{\partial}{\partial t} (\rho f_k) + \frac{\partial}{\partial x_j} (\rho u_j f_k) = \frac{\partial}{\partial x_j} \left( D \frac{\partial f_k}{\partial x_j} \right) \quad (9)$$

Where D is the diffusion coefficient,  $f_k$  is the mixture fraction for the  $k^{\text{th}}$  mixture.

### 2.3.5 Standard K - $\varepsilon$ model expressed

$$\frac{\partial}{\partial t}(\ell k) + \frac{\partial}{\partial x_j}(\ell^{u_j k}) = \frac{\partial}{\partial x_j} \left( \Gamma_\phi \frac{\partial k}{\partial x_j} \right) + P_k - \ell \varepsilon \quad (10)$$

$$\frac{\partial}{\partial t}(\ell \varepsilon) + \frac{\partial}{\partial x_j}(\ell^{u_j \varepsilon}) = \frac{\partial}{\partial x_j} \left( \Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_j} \right) + C_1 \frac{\varepsilon}{k} P_k - C_2 \ell \frac{\varepsilon^2}{k} \quad (11)$$

Here  $P_k = T_{ij} \left( \frac{\partial u_i}{\partial x_j} \right) = \mu_T S^2 \quad (12)$

$S = \text{mean strain rate} = 2S_{ij} S_{ij} \quad (13)$

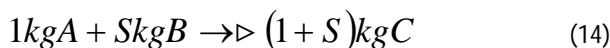
$P_k$  – generation of turbulent due to buoyancy

### 2.3.6 Chemistry /Reaction Model

The reaction model employed here for CFD development is the instantaneous chemistry model in which the reactants are assumed to react completely upon contact. The reaction rate is infinitely rapid and one reaction step is assumed. Two reactants, which are commonly referred to as “fuel” and “oxidizer”, will be involved. A surface “flame sheet” separates the two reactants (this assumption can be made only for non-premixed flame).

The eddy dissipation model for the chemistry of reactions, Standard EDC model

For combustion reactions, a single step irreversible reaction with finite reaction rates



S is the stoichiometric mass requirement of species B to consume 1kg of A.

Mixture composition is determined by solving for only two variables, mass fraction of specie A,  $Y_A$  and mixture fraction  $\varepsilon_j$ . Using (13), above

$$\frac{\partial}{\partial t}(\ell Y_A) + \frac{\partial}{\partial x_j}(\ell^{u_j Y_A}) = \frac{\partial}{\partial x_j} \left( \frac{\mu_T}{Sc_T} \cdot \frac{\partial Y_A}{\partial x_j} \right) + R_A \quad (15)$$

and

$$\frac{\partial}{\partial t}(\ell Y_A) + \frac{\partial}{\partial x_j}(\ell^{u_j \varepsilon_j}) = \frac{\partial}{\partial x_j} \left( \frac{\mu_T}{Sc_T} \cdot \frac{\partial \varepsilon}{\partial x_j} \right) \quad (16)$$

Where  $R_A$  – tone averaged reaction rate.  $Sc_T$ - turbulent Schmidt number, validity requires  $Sc_T$  to be same for all species.

$\beta$  - conserved combined variable

$$\varepsilon = \frac{\beta - \beta_\infty}{\beta_0 - \beta_\infty} \quad (17)$$

For components A and B

$$\beta = Y_A - \frac{Y_B}{S} \quad (18)$$

$\beta_\infty \equiv$  value of  $\beta$  at a B rich inlet,  $\beta_0 \equiv$  value of  $\beta$  at a A rich inlet

For species A, reaction rate is expressed as

$$R_A = A \cdot \ell \cdot \frac{\varepsilon}{k} \cdot \min \left( Y_A, \frac{Y_B}{S}, B \frac{Y_C}{1+S} \right) \quad (19)$$

$S \equiv$  mixture wt of components B, The constants  $A = 4.0$ ,  $B = 0.5$ , for micro mixing requirements

$$\Gamma_{m12} = \frac{3}{2} \cdot \frac{k}{\varepsilon} + \frac{1}{2} \ln(Sc) \left( \frac{Y}{\varepsilon} \right)^{\frac{1}{2}} - 5.53 \quad (20)$$

Further work (comprehensive energy calculations) Num flame

Total Energy in a chemical reaction is given by

$$E = h - \frac{P}{\ell} + \frac{V^2}{2} - h \equiv \text{sensible enthalpy} \quad (21)$$

$$h = \sum Y_i h_i + \frac{P}{\ell} \quad \text{Species } i, \quad h_i = \int_T^T c_{p_i} dT$$

one step reaction model



Methane activation energy  $E = 48.0$  cal/mole

$A =$  the pre exponential factor =  $8.3 \times 10^{16}$ ,  $h_j$  – enthalpy of component j in a mixture,  $C_p$  of mixture with temp variations,  $\ell m$  - density of mixture

### 2.3.7 Heat and Mass Transfer Model

Transport Equation for energy can be written as

$$\frac{\partial}{\partial t}(\ell E) + \frac{\partial}{\partial x_j}[\mu_i(\ell E + P)] = \frac{\partial}{\partial x_j} \left( k_{eff} \frac{\partial T}{\partial x_j} \right) + S_h \quad (22)$$

Where E = total energy,  $K_{eff}$  = effective thermal conductivity

$$K_{eff} = K + \frac{C_p \mu_t}{Pr_t} \quad (23)$$

$K$  – thermal conductivity of component, Turbulent diffusion coefficient,  $\Gamma_t = \frac{\mu_t}{Sc_t}$

Effective diffusion coefficient,  $\Gamma_{eff} = \Gamma_\phi + \Gamma_t$

## 3.0 THE SOLUTION PROCEDURE.

### 3.1 Development Of The Simpler Algorithm.

Once Navier – Stokes partial differential equations governing a flow field are formulated it is incumbent to develop the means of solving the equation. One of the popular methods in use is the method of procuring numerical solution via the SIMPLE algorithm. The procedure is as follows.

If  $\phi$  is a variable and represented by a polynomial of the form, in x-direction

$$\phi = a_0 + a_1 x + a_2 x^2 + \dots + a_m x^m \quad (24)$$

The unknowns  $a_0, a_1, a_2, \dots, a_m$  are calculated by treating the dependent variable  $x$  at a finite number of locations (the boundaries) on a grid of a chosen domain. The methods involves the generation of a set of algebraic equations for these unknowns and the prescription of an algorithm for solving the equation.

### 3.2 Control Volume Formulation.

First we replace a given continuum with a grid formation of continuous discrete units over the given space, the flow field. The calculation domain is divided into a number of non – overlapping control volumes such that there is one control volume surrounding each grid point. The differential equation is integrated over each control volume. Piecewise profiles expressing the variation of  $\phi$  between the grid points are used to evaluate the required integrals. The result is the discretization equation containing the values of  $\phi$  for a group of grid points. It is not compulsory that distances between the grids are equal. The use of non – uniform grid spacing is often desirable, to deploy computer power effectively, we should note that close to – accurate experimental results are obtained when grid spacing are fine. The number of grid points that should be distributed and numbers depend on the nature of problems to be solved.

#### 3.2.1 Boundary Condition.

The boundary conditions for the two dimensional flame modeling described below:

- (1) Symmetric centerline
- (2) Outlet: The outlet is modeled by means of a pressure outlet; this pressure is set to be atmospheric.
- (3) Outlet or burner wall: Depending on the choice, this boundary is an outlet or a wall. When a free flame is modeled, this is a pressure outlet. When a flame in a box is modeled a burner wall is chosen.
- (4) Burner wall: The burner wall is set to a constant temperature, and a non-slip condition is used.
- (5) Inlet: The inlet will be given by a velocity profile, using the determined profiles from the Fig 1.

#### 3.2.2 Discretization for Two Dimensions.

We shall assume the heat flux  $q$  obtained prevails over face area  $\Delta y \times 1$ . Using similar assumptions for other faces, a differential equation of the form

$$\rho c \frac{\delta T}{\delta t} = \frac{\delta}{\delta x} \left( k \frac{\delta T}{\delta x} \right) + \frac{\delta}{\delta y} \left( k \frac{\delta T}{\delta y} \right) + S \quad (25)$$

can be derived. This can be discretized, as

$$a_p T_p = a_E T_E + a_w T_w + a_N T_N + a_s T_s + b \quad (26)$$

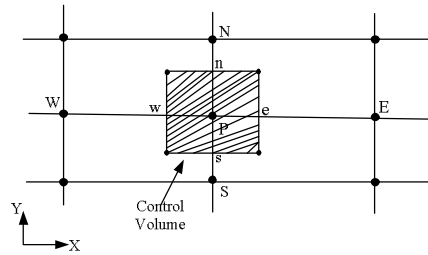


Fig. 2 – Two Dimensional Control Volume

Where

$$\begin{aligned} a_E &= \frac{ke\Delta y}{(\delta x)_e}, a_w = \frac{k_w \Delta y}{(\delta x)_w} \\ a_N &= \frac{k_n \Delta x}{(\delta y)_N}, a_s = \frac{k_s \Delta x}{(\delta y)_s} \\ a_p^o &= \rho \frac{c\Delta x\Delta y}{\Delta t}, b = Sc\Delta x\Delta y + a_p^o T_p^o \\ a_p &= Q_E + a_w + a_N + a_s + a_p^o - S_p \Delta x\Delta y \end{aligned} \quad (27)$$

### 3.2.3 Method of Solution of the Linear Algebraic Equations

The solution of the discretization equation can be obtained by the Gaussian – Siedel iteration method. In a system with grids 1,2,3,....., N with 1 and N being the boundary points, the discretization equation can be written as, for a variable T

$$a_i T_i = b_i T_{i+i} + C_i T_{i-i} + d_i \quad (28)$$

Showing Temp  $T_i$  is related to temperature  $T_{i+i}$  and  $T_{i-i}$ . Effort is made to produce a matrix form of the derived equations of any of the given variable and the matrix is subsequently solved using a special form of Gauss-siedel iterative procedures called Tridiagonal algorithm. The method gives better results with one and two dimensional problems. Though convergence of the method is slow for large numbers of grid points, the accuracy could be very high. In order to speed up or slow down the changes from iteration to iteration in order to obtain quick convergence to solution, we employ a overrelaxation as speed up, underrelaxation as slow down. Use of underrelaxation is more common to avoid divergence. Overrelaxation is more employed in Gauss – Siedel procedure.

## 4.0 NUMERICAL SOLUTION

### 4.1 Inlet Data.

Methane is delivered through a horizontal orifice that did not protrude into the flame control volume. Composition of gas is as tabulated below.

Table 1 Particulates Data.

No	Property Description	Methane	Ethane	Oxygen	Carbon-dioxide	Water Vapour	Nitrogen
1.	Molecular Wt (Kg)	16.04	30.07	32.00	44.01	36.04	28.02
2.	Mass Fraction (%)	0.86	0.1389	0.2315	0.25	0.45	0.7556

It is assumed that a fuel of 86 % methane content delivered at about 15 m/s at 25° C /1bar via a 24'' pipe (alternatively use mass flow rate for fuel estimates). Air current is in the range 3 – 10m/s at normal 1 bar and 27°C chosen for the study. One step reaction is assumed outward boundary condition applies to flows

#### 4.2. Combustion model

The 2-D flow and mixing fields were resolved by the solution of the 2-D, axisymmetric forms of the density-weighted fluid flow equations, supplemented with the k-ε model. Transport equations of several species are solved with equations for the mixture fraction and its variance. Solution of the transport equations is achieved using backward difference for first derivatives centered differencing for second derivatives with the upwind scheme for flow constants as developed by [6]. The solver employed is the one-way adaptive TDMA procedure as stated above.

The conservation equations of species mass fraction are transformed into the flamelet space. The flamelet equations are solved in pre-processing. The stationary solution is stored in tables containing the profiles of temperature and mass fractions for all chemical species as function of the mixture fraction, its variance and the scalar dissipation rate. The coupling of chemistry and flow field is performed via the mixture fraction, its variance and the scalar dissipation which were provided from the flow field calculations. These values at each computational cell are used to extract mean scalar properties from the chemistry look up tables. The flow field properties are updated and iterations continue until convergence criteria are met. The governing equations are discretized using the finite volume method in an axisymmetric Cartesian coordinates. The Revised Semi-Implicit Method for Pressure Linked Equation, SIMPLER numerical scheme is used to handle the pressure and velocity coupling. The computational domain covers an area from 0 to 1.0 m in the axial direction and 0 to 0.40m in the radial direction. Totally 62(x) × 24(y) uniform grids are used in the simulations. It has been checked that the further increase of grid number does not significantly influence the simulation results. The mass flow rate, total tem-

perature, turbulence intensity and hydraulic diameter are specified for the inlet boundary. At the outlet region, outflow condition is assumed and the symmetry condition on the side boundary. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give all unknown quantities.

#### 4.3 Grid Size and Mesh generation

A uniform size adaptive grid system is employed. Grid testing was carried out on a limited scale for different plate lengths and a grid size, the results was satisfactory as changes in results conform to expectations. It was observed that as grid size numbers increases the number of iterations required for convergence also increases considerably and beyond certain limit convergence to a solution was not achieved.

#### 4.4 Boundary Conditions

Boundary conditions at the inlet are given separately for the fuel stream at the central jet and the air stream cross-flow. The streams are considered to enter the computational domain as plug flow, with velocities calculated from their respective flow rates. The temperatures of fuel and air are specified. In conformation with the conditions used the fuel flow rate is taken as 0.9517 kg/s and the air flow rate is calculated to be 3.7638 kg/s. The temperatures for both the streams at inlet are 300 K. The temperature for air is taken to be 300 K. The inlet velocities were calculated from the respective mass flow rates and densities of air and methane fuel using mass transport equations. No soot is considered to enter with the flow through the inlet plane. Considering the length of the computational domain to be 1.0m x 0.4 m, the fully developed boundary conditions for the variables are considered at the outlet. In case of reverse flow at the outlet plane, which occurs in the case of buoyant flame, the stream coming in from the outside is considered to be atmospheric air. Axi-symmetric condition was considered at the central axis, while at the wall a no-slip, adiabatic and impermeable boundary condition is adopted.

#### 4.5 Stability Criteria and Convergence Conditions

The choice of the incremental time-step was done very carefully as it is directly related to the stability of the explicit scheme. The first criteria demands that pure advection should not convey a fluid element past a cell in one time increment as the difference equations consider fluxes only between adjacent cells. Hence the time increment should satisfy the following inequality. Solution use an under-relaxation factor of 0.2 - 0.5 to achieve quick convergence as suggested by previous scholars. The corrected velocity is employed to determine convergence of derived solution.

### 5.0 RESULTS AND CONCLUSIONS.

#### 5.1 Results, Observations and Discussions.

The results presented below revealed the events occurring in the visible forms of flame that we see. Flames are known to exhibit wavy pattern of flow due to recirculation currents found within its flow, this is shown by the negative values of particulates as seen in fig 2a and fig. 2b for the concentrates

under consideration.

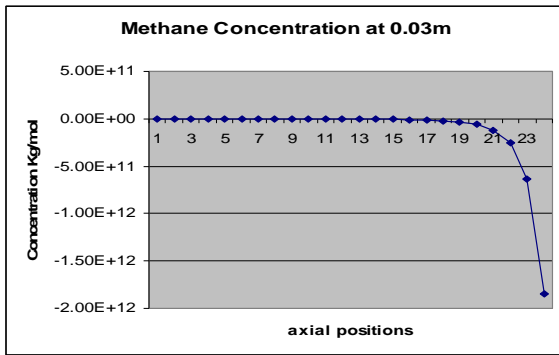


Fig 2a CH<sub>4</sub> particulate concentrations at 0.03m

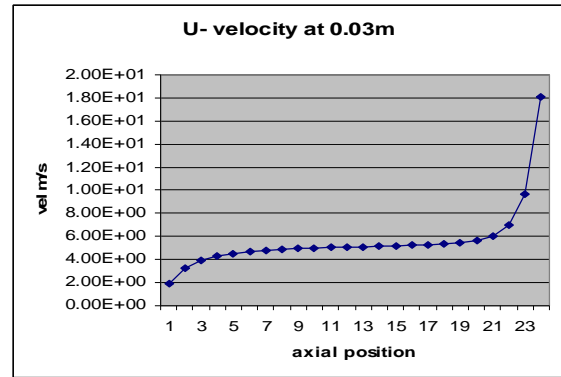


Fig 3b U-velocity distribution at 0.03m

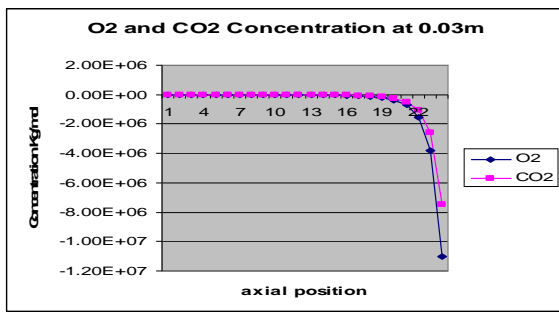


Fig 2b O<sub>2</sub> and CO<sub>2</sub> particulate concentrations at 0.03m

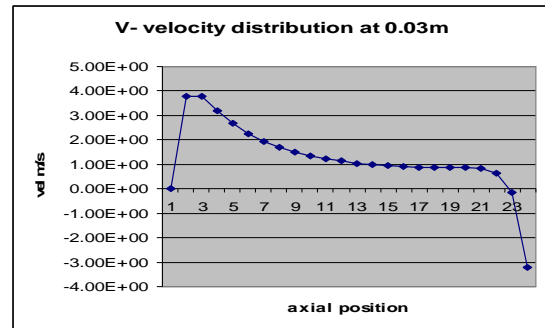


Fig 3c Velocity distribution at 0.03m.

In some circumstances a thin flame sheet forms a connected but highly wrinkled surface that separates the reactants from the products. The flame surface is convected, bent, and strained by the turbulence and propagates (relative to the fluid) at a speed that can depend on the local conditions (surface curvature, strain rate, etc.) Further, the temperature (Fig 3a) shows definite increases as well as the u-velocity (Fig 3b). At this point the v-velocity (Fig 3c) of flow of methane gas/vapour begins to fluctuate rapidly this evidently supportive of buoyancy effects accompanying temperature rise till the methane ignition temperature is reached.

This rapidly changing v-velocity of methane as a result of rapid temperature changes and resultant buoyancy continued beyond 0.1m till about 0.3m where stable positive values were obtained till the end of the control volume (Fig 4a and Fig 4b).

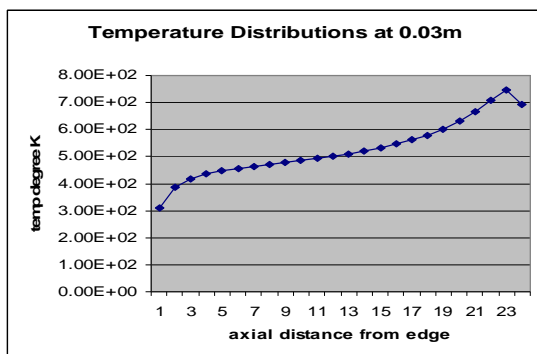


Fig 3c Temperature distribution at 0.03m

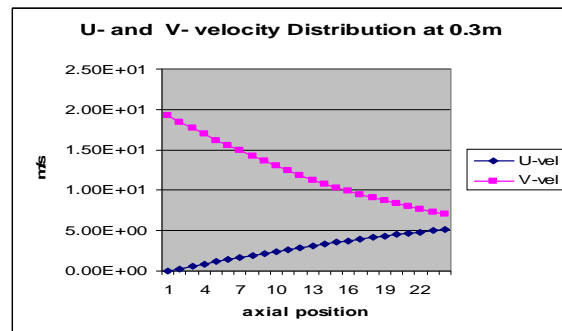


Fig 4a Velocity distribution at 0.3m

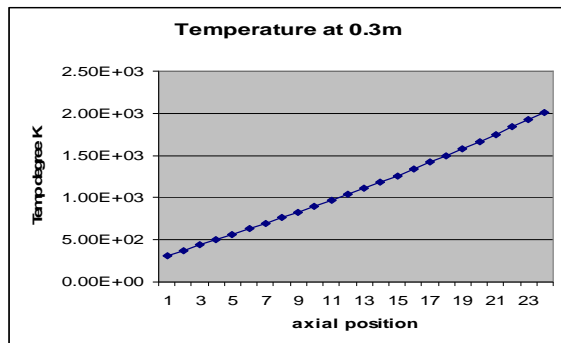


Fig 4b Temperature distribution at 0.3m

One finds that the pollution of the solution with oscillations is dependent on the domination of the convection terms over other terms of the differential equation, in practice most often diffusion terms in the governing transport equations. The higher the Peclet and flow Reynolds numbers are, the more dominant is the convection term and the stronger is the pollution with oscillations.

As the cross wind velocity increases, a transition from buoyancy dominated flow to cross wind dominated flow can be noted along with a reduction in oscillation amplitudes. The changes of stability of flow begins to emerge in the diffusion flame leading to positive values showing that the v-velocity component profile presents in the flow is moving up. This effect is not only attributed to the buoyancy forces, but also due to the presence of the wall normal to the stream-wise direction. It is evident that the buoyancy forces have their impact in this region of the flame.

## 5.2 Conclusions.

A finite difference scheme for solving the fluid dynamic equations of two dimensional elliptic flows has been used to predict turbulent diffusion jet discharging perpendicularly into an unconfined cross-flow air current. The mathematical model employed a standard k-e model to calculate the distribution of Reynolds stresses. The turbulent gas phase, non-premixed combustion process was model using probability density function approach assuming one step fast chemical reactions. Soot calculation was omitted in this study as its formation was considered negligible. The purpose of the numerical modeling/simulation on the turbulent diffusion flame is to determine the profiles of variables and flow patterns for velocity, species concentration profiles, and temperature in the combustor area.

## REFERENCES

- [1] Trouve, A. and Poinot, T., J. Fluid Mech., 278:1–31 (1994).
- [2] Zhang, S. and Rutland, C. J., Combust. Flame, 102:447–461
- [3] Tanahasi, M., Fujimura, M., and Miyauchi, T., Proc. Combust. Inst., 28:529–535 .2000.

[4] Bell, J. B., , Day, M. S., and Grcar, J. F., Proc. Combust. Inst., 29 (2002).

[5] Zegers R.P.C., Flame stabilization of oxy-fuel flames, MSc. Thesis, Eindhoven University of Technology. Report number WVT 2007.13

[6] Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Taylor & Francis, Philadelphia / New York.. 1980.