# MODELLING OF FLAMELESS OXIDATION BURNER

(Paper # 6)

Achmad Hattary

Department of Mechanical Engineering, The University Of Adelaide, Adelaide SA 5005, Australia

## Abstract

A special laboratory burner that operates in flameless oxidation mode has been modelled by finite volume based CFD package CFX-4.3 with mixed is burned combustion model. Numerical solution of model is assessed by comparison with experimental data. It is seen that predictions to some state of engineering accuracy can be obtained by mixed is burned model.

## Keywords

Flameless Oxidation, diluted combustion, NOx, CFD, air preheating, mixed is burned.

## Introduction

Environmental concerns and limited resources of fuels have been the major constraints in designing combustion systems. These constraints have triggered researchers and manufacturers of combustion system to developing low polluting and fuel-efficient combustion systems. The major pollutants produced by combustion are unburned and partially burned hydrocarbons, nitrogen oxides or $NO_x$ (NO, $NO_2$), carbon monoxide (CO) and sulfur oxides ($SO_2$ and $SO_3$). Nitrogen oxide is one of the most significant pollutants, There are three sources of nitrogen oxides related with combustion process: Prompt $NO_{x,}$ fuel $NO_{x,}$ and thermal $NO_{x.}$ Thermal $NO_{x.}$ formation, is the most relevant source for the combustion of cleans fuels like natural gas, even it can be emitted from combustion of clean gas such as hydrogen and natural gas, because it can be formed from air nitrogen and oxygen at elevated temperature [1].

In many high temperature combustion processes, higher fuel efficiency is achieved by air preheating, which involves withdrawal of energy either from recycled exhaust or flue gas. The most significant drawback of implementing air preheating is increased in maximal temperature in the flame, which results increased thermal NOx emission [1,2]. Therefore the conflict of interest between fuel saving and reduction of nitric oxides emissions must be taken into account in burners design.

Great deal of efforts to overcome the dilemma of achieving fuel saving and reduction of nitric oxide have been made during last decade [1,3]. Recently, a regime of combustion has been found at high exhaust gas recirculation rates with the temperature of the recycled gas exceeds the auto-ignition temperature of the air-fuel mixture, such that combustion is sustained after ignition of the mixture, At the same time the recirculated exhaust gas ensures that the adiabatic flame temperature is lowered, resulting in low NOx emissions [4]. The recirculation of exhaust gases increases the concentration of inert gasses in the combustion air, so the oxygen concentration in the combustion air is reduced below the normal level in air. At optimal condition, this combustion regime allows silent, colourless, and low peak temperature combustion process and has been termed as flameless oxidation. Milani and Saporano [2] have summarised the advantages of flameless oxidation or so called diluted combustion include: abatement of NOx emission, fuel savings, productivity increase, and increase in thermal uniformity.

Idealised process of flameless oxidation is shown schematically in figure 1. Firstly combustion air is mixed with recirculated exhaust gas (region I), after mixing process, fuel is added (Region II), if the recirculation rate is sufficiently high, temperature rise is only few hundred Kelvin. In region III the energy is withdrawn from the combustion product to ensure suitable temperature of reaction in region II.

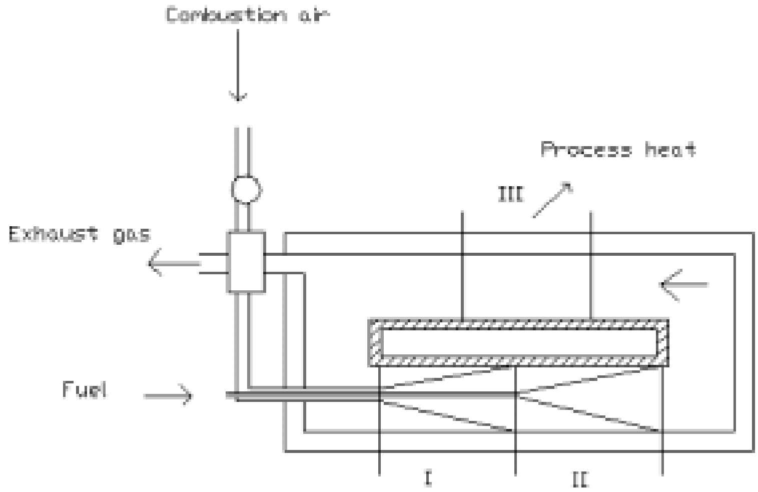



**Figure 1:** Idealised process of flameless oxidation [1]

High experimental costs and increasing power of computers have made viable the use of Computational Fluid Dynamics (CFD) for engineering applications. CFD has become a useful tool for combustion modelling and for modern burners design such as flameless oxidation burners. In addition, as modern CFD tools rely on assumptions, experimental validation of the simulations are required [5]. In this project a commercial finite volume based CFD package called CFX-4.3 is utilised to model a special flameless oxidation burner that is designed by Dr. Bassam Dally.

## 1. CFD Modelling

The model of the burner is a 2-Dimensional computational domain as shown schematically in figure 2. The computer domain only covers the top half of axial slice of the burner for the sake of computational efficiency. Boundary conditions of this domain are summarised in table1.

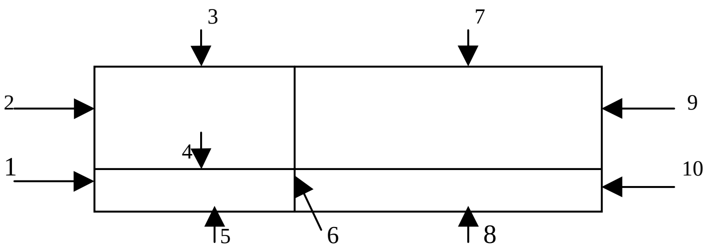



**figure 2:** Computational Domain

| No. | Boundary | Condition(s) |
|---|---|---|
| 1 | Jet Inlet | Fuel (mixture of 50% Hydrogen and 50% Methane by volume), u= 66.3 m/s, T=300K |
| 2 | Hot air inlet | Oxidant stream consists of : 6% oxygen, 5.5% water, 88.5% nitrogen; u = 3.09 m/s ; T = 1300K |
| 3 | Second annulus wall | Non-adiabatic ( Heat transfer allowed) |
| 4 | Jet wall | Non-adiabatic ( Heat transfer allowed) |
| 5,8 | Symmetry | (not pre-specified) |
| 6 | Jet outlet | (not pre-specified) |
| 7 | Pressure | Pressure = 0 pa |
| 9,10 | Outlet | (not pre-specified) |

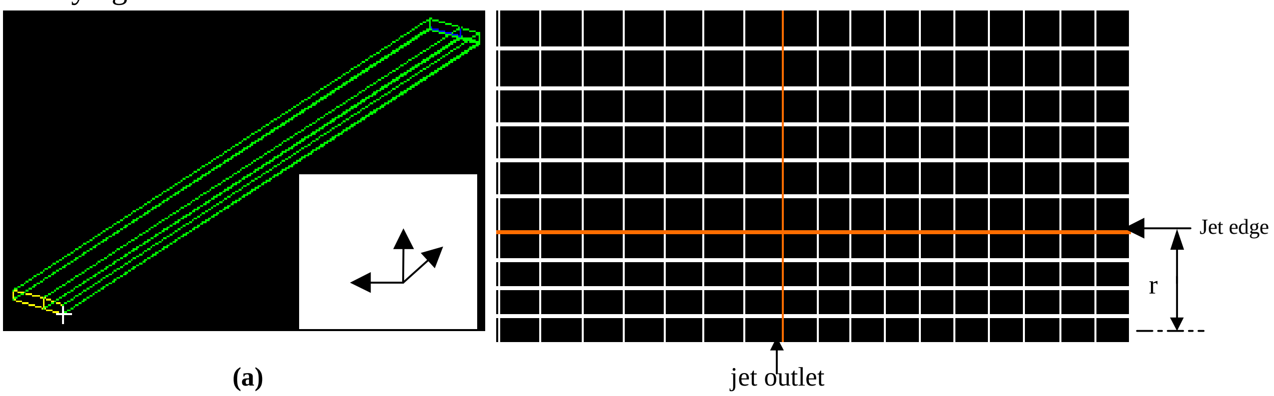
**Table 1:** Summary of boundary conditions

## 1.1 Methodology

### 1.1.1 Build geometry and generate a grid

The geometry of the burner was constructed using CFX-Build, firstly the geometry of the burner is plotted in 2-D, then extrude the whole geometry in z-direction. The geometry is made as rectangular block instead of cylindrical object. The intention of rectangular block geometry is to avoid the use of non-orthogonal grid, and thus improving the accuracy of solution [6]. Next a grid is generated by splitting a region into small volumes. This grid was made by quad 4 elements that made up an axisymmetric grid of 150 x 84 x 1 computational cells. Although the model geometry is a block, the property of the model is 2-dimensional because there is only one cell in the Z-direction.

Part of the grid in the vicinity jet outlet is shown schematically in figure 4(b). In this region the volume of cells are smallest whereas solution variables such as velocity, and temperature are varying most.




(a) (b)

**Figure 4:** wireframe of geometry **(a)**; detail of grid around jet outlet **(b)**

### 1.1.2 Write a command file

Command file is the input specification of the problem, where model options, differencing scheme, boundary conditions, fluid properties, and output options are specified. Below is the summary of the command file.

- The combustion model is 'mixed is burnt', which assumes that fuel and oxidant cannot instantaneously coexist. This model is suitable for systems that have separate fuel and oxidant supply. The following relationship holds for mixed is burned model: if the equivalence ratio greater than 1, the mixture consist of fuel and products; if the equivalence ratio is less than 1, the mixture consist of oxidant and products [7]. Flow is weakly compressible which is an approximation which makes compressible flow behave similarly to incompressible flow. In weakly compressible flow, the density is found from ideal gas equation [7].

- Number of mass fraction equation is three (for fuel, oxidant, and product). The mass fraction must be specified at inlet boundaries.
- Flow is steady state, the conditions of mass and energy within the burner and its inlets and outlets do not vary with time.
- K-Epsilon turbulence model.
- Double precision, this will improve the numerical precision of the solution and also improve convergence, but it requires more memory and computational time.
- Due to model limitation the, CO2 is not included in the hot stream and is replaced with nitrogen and water, such that the mixture in the model has same constant pressure specific as in the experiment.

### 1.1.3 Running the programs and monitoring convergence

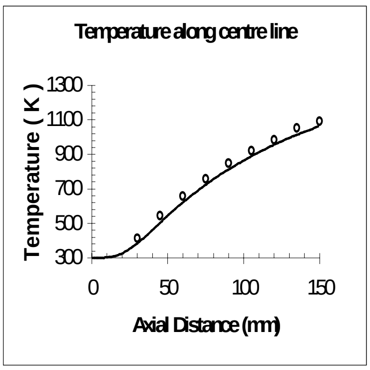
After the command file was completed, the program called CFX-Solver executed command file, and solved the problem by using iteration technique. It is worth mentioning that in order to obtain accurate results solution must converge. The convergence can be monitored by observing mass source residual. Mass source residual must be sufficiently small (in order of $10^{-6}$)

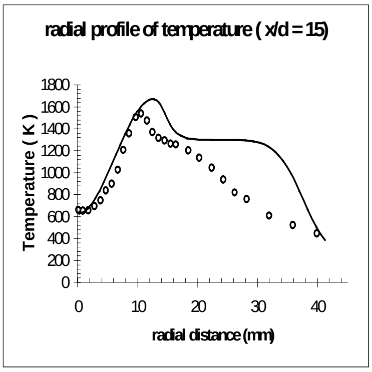
## 2. Results and Discussion

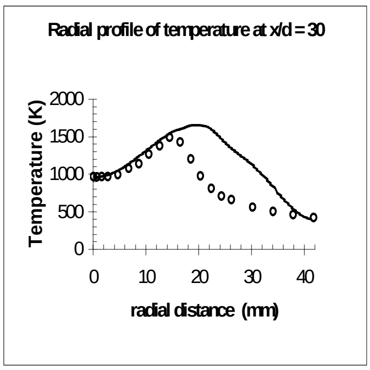
The experiment on similar burner has been performed by Dr.Dally. In this experiment, temperatures, fuel mixture fractions, and rms fuel mixture fractions have been measured along the center line and three radial traverses at stations up to 120mm (x/d = 30) downstream of jet outlet. The experimental data are then plotted together with the corresponding predicted data from CFD model.

### 2.1 Temperature profile

It can be seen from figure 5, that along the center line, the difference between model data and experimental data is reasonably small. The measured and model temperature profile along two radial traverses agree up to about 10mm above center line, then agreement recovered at radial distance of 40mm. The limited ability of the model to predict the radial profile of temperature might be attributed to: finite rate chemistry effects, which are not accounted for mixed is burnt model; the computer domain not include cold air stream (mixed is burnt model can only solve two stream problems) this results the predicted temperatures tend to be higher than experimental temperatures.



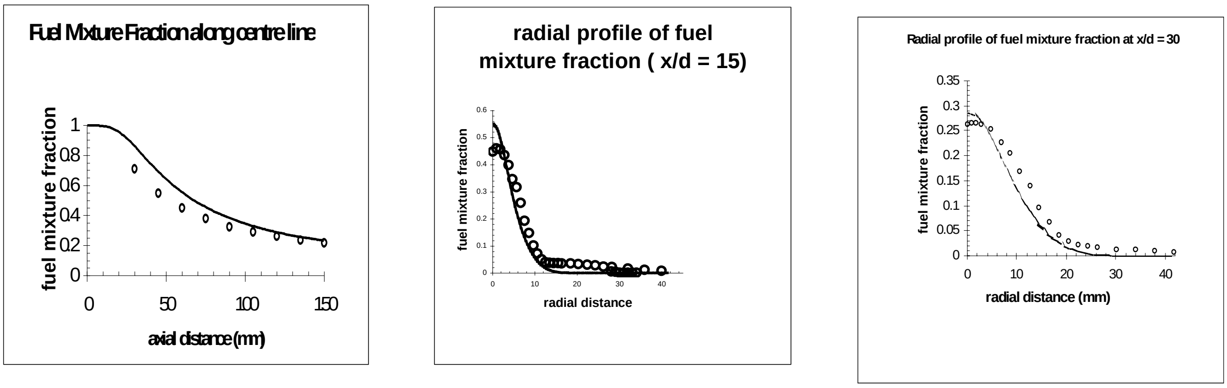



o experimental ——— model

**Figure 5**: Model and experimental temperature profiles

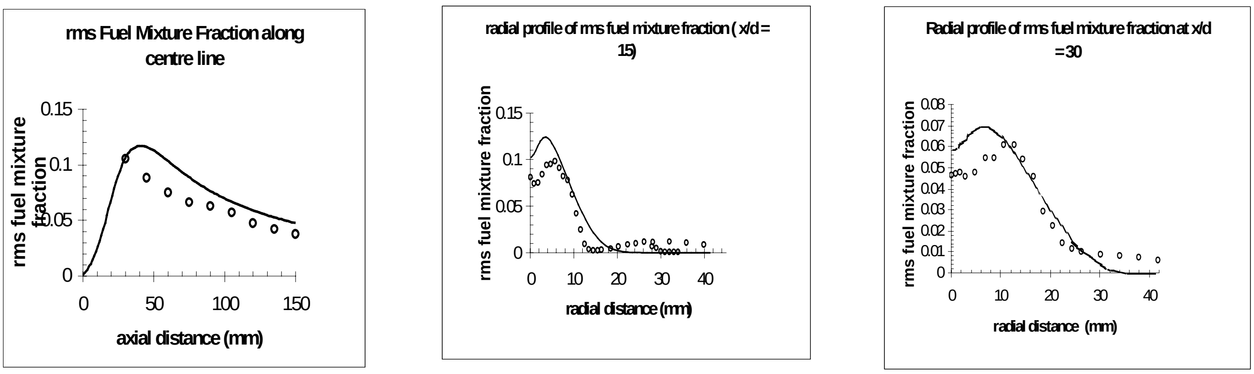
## 2.2 Fuel mixture fraction and RMS fuel mixture fraction profile

Fuel mixture fractions are satisfactorily predicted by the model as shown in figure 6. The axial and radial profile of rms fuel mixture fractions are presented in figure 7, despite there are some differences in values of rms mixture fraction, the trend of the model rms mixture fraction agree with experimental data. The difference might be caused by errors in turbulent parameters specification.




0 experimental ——— model

**Figure 6 Model and experimental fuel mixture fraction profiles**




0 experimental ——— model

**Figure 7 Model and experimental rms fuel mixture fraction profiles**

## Conclusions

The 'mixed is burnt' combustion model has showed some promising agreement with experimental results. However, mixed is burnt combustion model can only accommodate two streams flow. Realistically the burner that is modelled in this project, has three streams. In order to obtain more accurate results the cold air stream should be included in model. Also finite rate chemistry effects may contribute to the discrepancy. In near future the Eddy Brake-up model which can solve three streams problem will be utilised to model the burner.


## Acknowledgements

I would like to thanks to my supervisors Dr. Bassam Dally and Ms. Ellizabeth Smith for their supervision and valuable advices.


## References


1. W  nning J.A, W  nning J.G, "Flameless Oxidation to Reduce Thermal NO-Formation", *Prog. Energy Combust Sci*, Vol.23 (1997), pp.81-94.

2. Milani, A., Saporano, A.,"Diluted Combustion Technologies", *IFRF Combustion Journal*. Article No 200101 (2001).

3. Katsuki, M., Haswegawa, T., "The Science and Technology of Combustion in Highly Preheated Air", *The 27th Symposium (International) on Combustion*, (1988), paper #803

4. Levy, Y., Nagel, A.,. Nekhamkin, Y., *"Low-NOx FLOX Combustor for High Efficiency GasTurbines"*, European Community Project of Fifth Framework Programe, (1999).

5. Anderson, J.D., *"Computational Fluid Dynamics : The basic with applications"*, Mc grawhill, (1995), New York.

6. Fletcher, D.F., *Lecture Notes on Fluid Engineering ( U4.421)"*, Department of Mechanical and Mechatronic Engineering , The University Of Sydney, (1994).

7. AEA Technology, CFX4.3 manual