Computational Analysis of Mixing and Transport of Air and Fuel in Co-Fired Combustor

JAVAID IQBAL*

RECEIVED ON 30.10.2013 ACCEPTED ON 17.10.2014

ABSTRACT

Computational analysis for air fuel mixing and transport in a combustor used for co fired burner has been done by RANS (Reynolds-Averaged Navier–Stokes) model comparing with 3D (Three Dimensional) LES (Large Eddy Simulation). To investigate the better turbulence level and mixing within co fired combustor using the solid fuel biomass with coal is main purpose of this research work. The results show the difference in flow predicted by the two models, LES give better results than the RANS. For compressible flow the LES results show more swirling effect, The velocity decays along axial and radial distance for both swirling and non-swirling jet. Because of no slip condition near boundary the near the wall velocity is about zero.

Key Words: **Swirling Flows Large Eddy Simulation, Co-Fired Combustor Reynolds-Averaged Navier-Stokes.**

1. INTRODUCTION

The biomass co firing may be used as a NO_x reduction method [1]. This method is being used in most of power plants in UK due to depleting natural resources and is a better use of renewable energy [1]. Power plants burn fuels like gas, oil and coal and release gases as flue gases, the aim of the project is to understand physics of air flow inside biomass co-fired burner assembly and to find out CFD (Computational Fluid Dynamics) model showing better results. In future the rapid increase in demand will cause limitations in availability of fossil fuels [1]. The biomass co firing is one of the major solutions for the above problems. Biomass is a fuel source used by men for centuries; the biomass is used for domestic as well as industrial purposes. If the wood, crop waste, etc. in pulverized form is mixed with coal in burners burners the more clean energy can be obtained from coal fired power plants [2]. This research has investigated better swirling and turbulence level for mixing before combustion using the commercial code FLUENT. Here the two CFD models LES and RANS have been compared. The LES model gives better results to investigate the swirling and mixing before combustion as compared with The RAMS model [3].

2. THEORETICAL BACKGROUND

2.1 CFD Modeling

The expertise needed for evaluation of flow problems may be got and analyzed using CFD numerical methods and algorithms. CFD is a tool used in the range renewable energy applications like wind flow, wave and tidal flow. It can also be used for modeling

* Department of Mechanical Engineering, Balochistan University of Engineering & Technology, Khuzdar

complex flow especially turbulence, heat and mass transfer, material transport chemical reactions etc. The Navier-Stokes equation is solved to resolve fully developed 3d or 2d complicated problems [4].

For FLUENT solvers based on the finite volume method Control volume is divided into several small cells. The finite volume method solves the partial differential equations; the values can be evaluated at discrete points on mesh [4].

In reality the most pre combustion processes are turbulent; for turbulence modelling various computational approaches can be used [5]. Here two methods RANS, and LES has been used and compared.

2.2 Turbulence Modelling

Research has proved that the dependence over transport equations or chemical species. Variables like pressure and velocity have fluctuating and mean value.

$$U = U^{*} + u \qquad (1)$$

$$P = P^{*} + p \qquad (2)$$

Where U* and P* are mean value for velocity and pressure in Equations (1-2), respectively

The continuity equation can be written as:

$$\partial\rho / \partial t + (\rho U_i) = 0 \qquad (3)$$

The conservation of linear momentum can be written as:

$$= g \frac{1}{\rho} VP + v\mu^2 .V \qquad (4)$$

Here ρ, $\boldsymbol{v}$, µ and V represent density, kinematic

viscosity, viscosity and volume respectively here

The energy Equation (5) is:

Q=1/2ρg.u.u+h (5)

Where Q shows total energy, 1/2ρg.u.u is kinetic energy in flow and h is enthalpy. In all equations g represents the acceleration due to gravity

2.2 Simulation Methods

- Using RANS to compute mean flow
- After solution convergence switch down to LES

The turbulence is instability of Laminar flow depending on turbulence model, the numerical details required, Reynolds number and Courant number.

2.2.1 RANS Models

RANS model has been used to investigate the complexity of flow and wall shear and results has compared with the LES model

2.2.2 Large Eddy Simulations

It is the numerical technique for solving turbulent flow. The LES provides more detailed and efficient results than other methods. The LES is a better tool for compressible flow, wall jet impingement and main jet shear. The LES results can easily be compared with published and experimental results for validation. Following are the main properties for large eddy simulations.

- Mass, momentum, energy are mostly transported by large eddies.
- Large eddies depend upon flow, geometries and boundary conditions of flow involved.
- Fluent has several eddy viscosity sub grid scale models.

In LES the governing equation is filtered Navier-Stokes equation, which is gained from the filtration of original Navier-Stokes equation.

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i\, u_j}{x_j} = \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} v \frac{\partial u_i}{[\partial x_j]} \qquad (6)$$

Here $\frac{\partial u_i}{\partial t}$ is rate of x velocity changing with time, $\frac{\partial u_i\, u_j}{x_j}$ ρ is density, ν is kinematic viscosity here.

The results from the simulation for this case are of two-fold:

- Results from k-ε (RANS)

- LES

It can clearly be seen that LES provide more detailed results than k-ε (RANS).

$$N_{LES}\ \frac{0.4}{[\mathrm{Re}^{1/4}]\ N_{RANS}} \qquad (7)$$

Equation (7) is the general equation for comparison of LES model with all RANS model like (a) K-Ω Standard (b) k-ε Realizable (c) k-ε RNG (d) RSM (e) k-ε RNG Transient.

2.3 FLUENT Simulation Code

Finite volume commercial code FLUENT 6.3 has been used for CFD simulation. FLUENT can be used for two and 3D turbulent flows in both steady and transient conditions.

3 TURBULENCE SIMULATION

3.1 RANS Modelling Results

Objective of this work to apply k-ε RANS modelling to the turbulent mixing flow simulation RANS models achieve quick results and need little computation but it has a limit where it may fail. Before LES velocity profiles of mean and fluctuating flow has been got using RANS model and compared with LES results. As it is known that nonlinear terms in the Navier-Stokes equation give rise to the Reynolds stress term [7]. Here the k-ε RANS model has been chosen because it is turbulent flow RANS model results shown in Fig. 1(a-b) are the contours of x, y and z velocity profiles; it can be observed that free stream velocity lowers in downstream direction. The RANS model results show better flow and turbulence but the large turbulent eddies are affected by boundary conditions and to resolve large eddies is too difficult.

3.2 LES Modelling Results

The LES modelling results have been compared with RANS model simulation results. The aim is to run LES on same geometrical model and comparison with RANS model. The x velocity contours have been shown using RANS models like K-Ω standard, k-ε realizable, k-ε RNG transient etc. in Fig. 2(a-b), and Fig. 3(a-b), similarly x velocity contours have also been shown using LES at different flow time. Basically the LES is more advanced numerical technique than the others as stated above and can be get more detailed results for the specially for swirling or turbulent flows. The x velocity contours using LES at different flow time have been shown in Fig. 3(b), and Fig. 4(a-b).

The y-velocity contours using RANS models can be shown in Figs. 5(a-b), Fig. 6(a-b), and Fig. 7(a-b) similarly the y-velocity contours using LES technique are shown in Fig. 8(a-b). z-velocity contours have been shown in using RANS models can be shown in Fig. 9(a-b), Fig. 10(a-b) and Fig. 11(a) similarly the z-velocity contours using LES at different flow time are shown in Fig. 11(b), Fig. 12(a-b). Fig. 13 is the graphical representation of the x velocity variation with flow time. If the LES results are compared with the RANS results the LES results in all direction (x,y, or z) gives better results for turbulence modeling in near wall swirling and velocity variation.

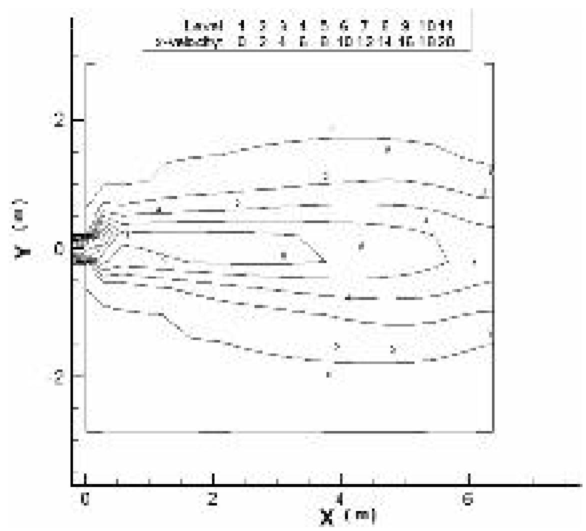

FIG. 1(a). X-VELOCITY CONTOURS k-ε RNG

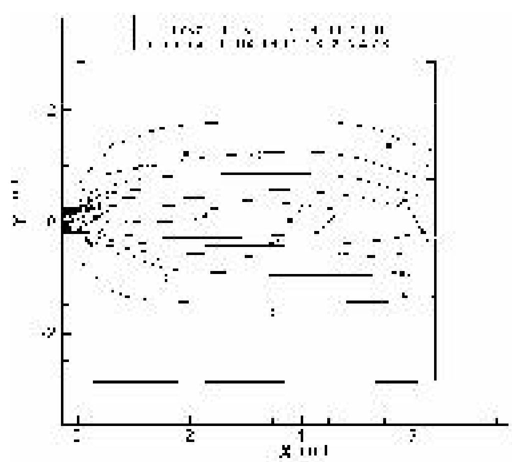
FIG. 1(b). X-VELOCITY CONTOURS RSM

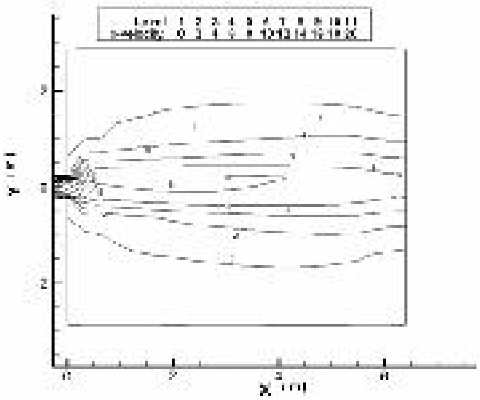

FIG. 2(a). X-VELOCITY CONTOURS K-Ω STANDARD

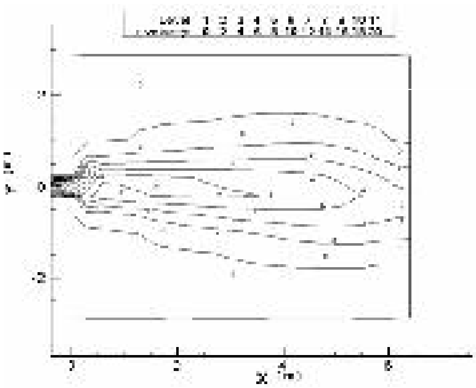

FIG. 2(b). X-VELOCITY CONTOURS K-ε REALIZABLE

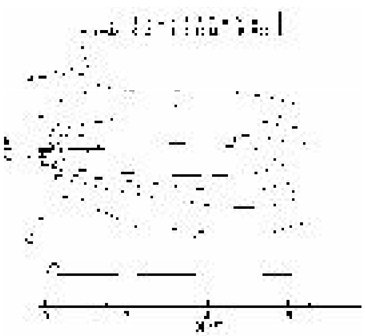

FIG. 3(a). X-VELOCITY CONTOURS K-ε RNG TRANSIENT

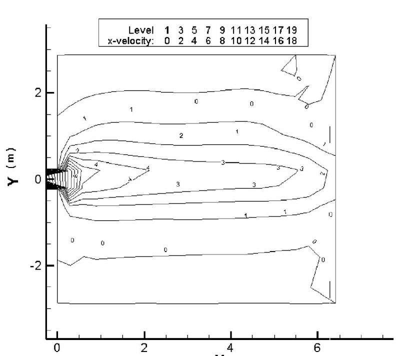

FIG. 3(b). X-VELOCITY CONTOURS LES T=10813SEC

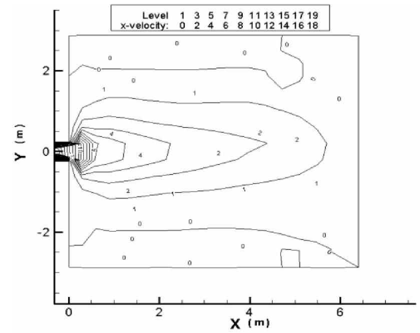

FIG. 4(a). X-VELOCITY CONTOURS LES T=5600 SEC

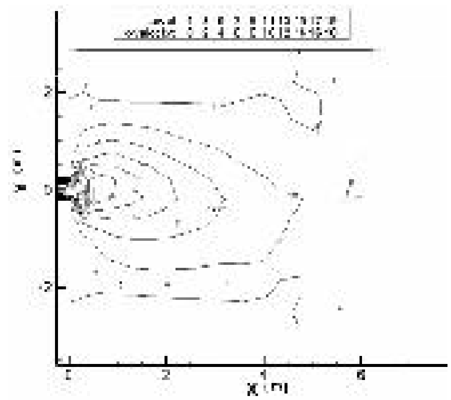

FIG. 4(b). X-VELOCITY CONTOURS LES T=5926SEC (H)

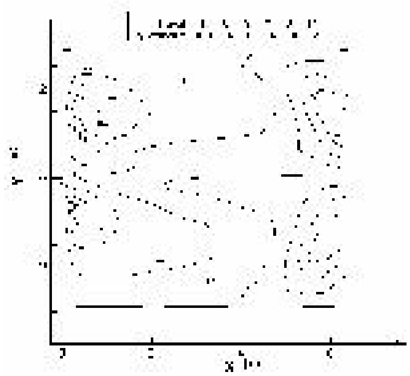

FIG. 5(a). Y-VELOCITY CONTOURS K-Ω STANDARD

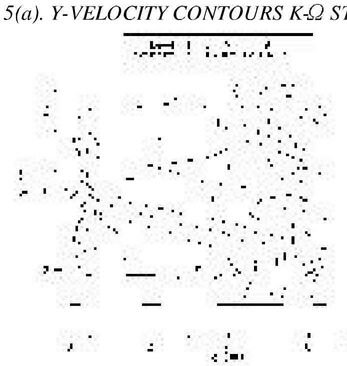

FIG. 5(b). K-Ω REALIZABLE

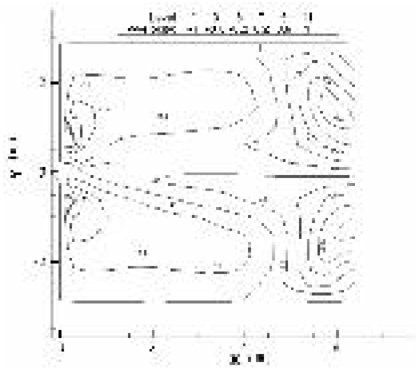

FIG. 6(a). Y-VELOCITY CONTOURS K-ε RNG

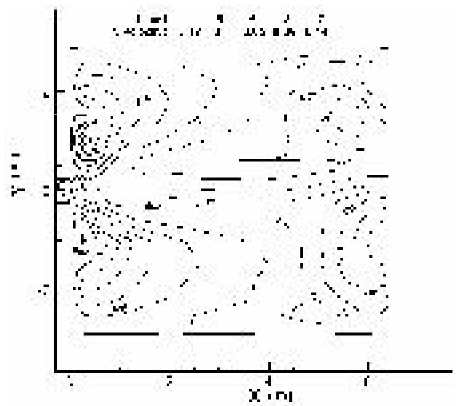

FIG. 6(b). Y-VELOCITY CONTOURS RSM

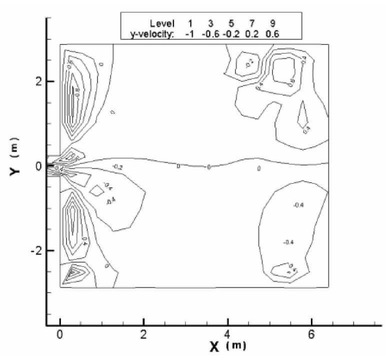

FIG. 7(a). Y-VELOCITY CONTOURS RSM

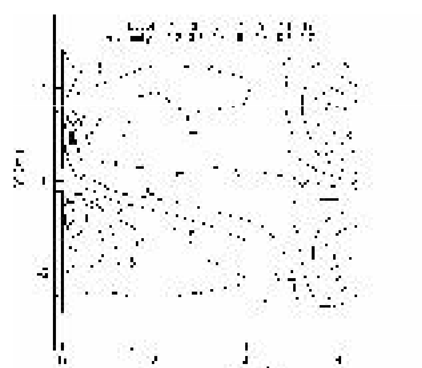

FIG. 7(b). Y-VELOCITY CONTOURS K-ε RNG TRANSIENT

FIG. 8(a). Y-VELOCITY CONTOURS LES T=5600SEC

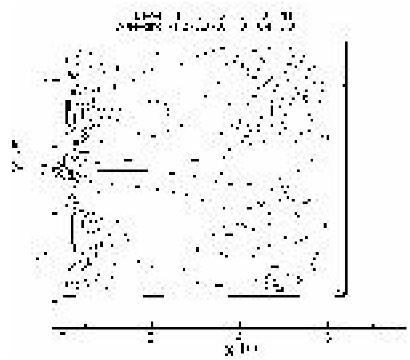

FIG. 8(b). Y-VELOCITY CONTOURS LES T=5926SEC

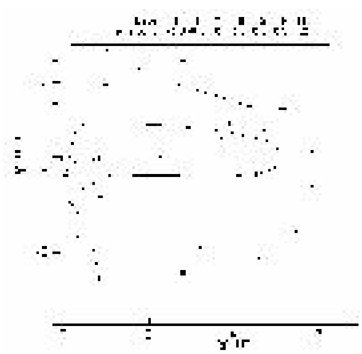

FIG. 9(a). Z-VELOCITY CONTOURS K-Ω STANDARD

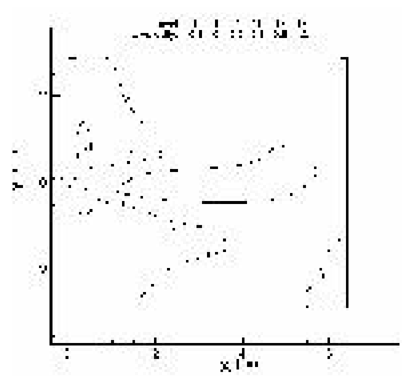

FIG. 9(b) Z-VELOCITY CONTOURS K-ε REALIZABLE

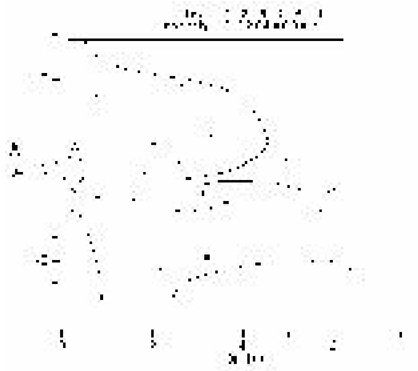

FIG. 10(a). Z-VELOCITY CONTOURS K-ε

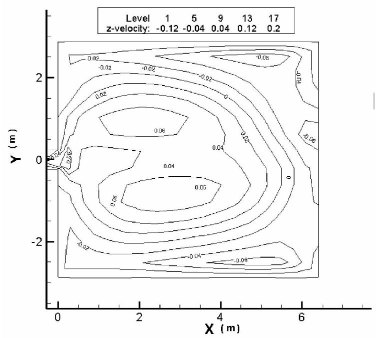

FIG. 10(b). Z-VELOCITY CONTOURS RNG RSM

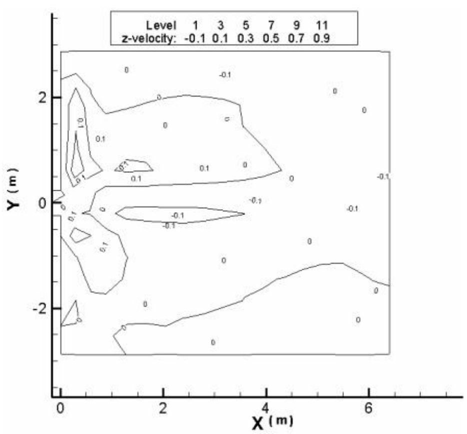

FIG. 11(a). Z-VELOCITY CONTOURS K-ε RNG TRANSIENT

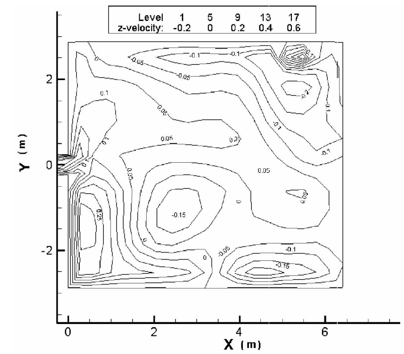

FIG. 11(b). Z-VELOCITY CONTOURS LES T=5600SEC

FIG. 12(a). Z-VELOCITY CONTOURS LES T=5926SEC

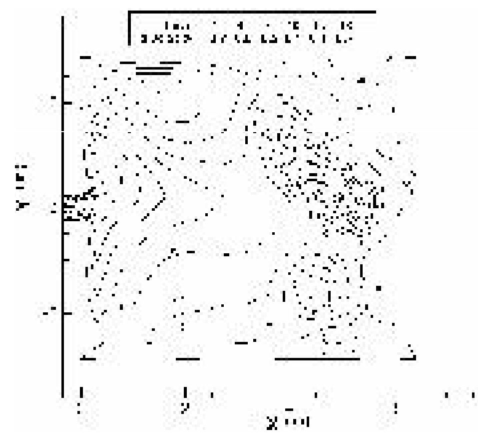

FIG. 12(b). Z VELOCITY CONTOURS LES T=10813SEC

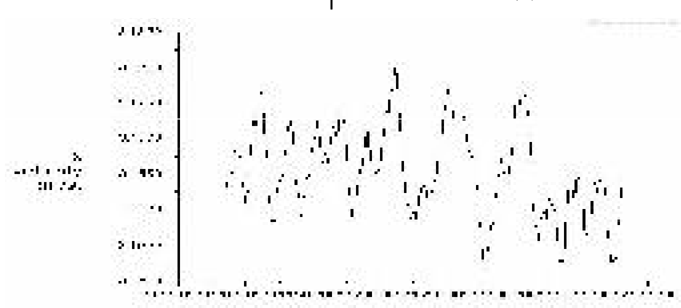

FIG. 13. X-VELOCITY VARIATIONS RNG TRANSIENT SIMULATION (T=400SEC)

4. CONCLUSIONS

The simulation has been done using RANS and LES model; the results include the turbulence and velocity fluctuations. LES have been taken at (2,2,0), (2,0.5,0) and (2,1.5,0). Velocity in centerline at (2,2,0) fluctuation is normal, at (2,0.5,0) which is near the center line fluctuations are maximum and at (2,1.5,0) near wall velocity fluctuation is minimum.

Second observation is that the large eddies can be seen in LES contours, LES gives better results than RANS model during the solution of turbulent flow. The contours results agree better with the results.

ACKNOWLEDGEMENTS

Authors efforts in this research have been taken. However, it would not have been possible without the kind support and help of Blochistan University of Engineering & Technology, Khuzdar, Pakistan, and University of Leicester, UK. Authors would like to extend our sincere thanks to all of them. Our thanks and appreciations also go to our colleague in developing the project and people who have willingly helped us out with their abilities.

REFERENCES

[1] Aroussi, A., and Iqbal, J., "Solid Fuel Biomass Cofiring with Coal", 4th International Symposium on Hydrocarbon and Chemistry, pp. 2-10, Ghardaia, March 24-26, 2008.

[2] Dubuisson, "Energy and CO_2 Balances in Different Power Generation Routes Using Wood Fuel from Short Rotation Coppice", Biomass and Bioenergy, Volume 15, November, 1998.

[3] Charlea, G., and Speziale., "A Combined Large-Eddy Simulation and Time-Dependent RANS Charlea, G.S., "A Combined Large-Eddy Simulation and Time-Dependent RANS Capability for High-Speed Compressible Flows", Journal of Scientific Computing, Volume 13, No. 3, 1998.

[4] CFD-Solutions, "Why Should You Use CFD Airflow Model", 2002 (Retrieved July 22, 2009 www.cfd-solutions.com).

[5] Davidson, L., "An Introduction to Turbulence Modes", 2003 (Retrieved January 08, 2013, http;//www.chamfers.se).

[6] Black, J.S., "Effects of Firing Coal and Biomass under Oxy-Fuel Conditions in a Power Plant Boiler Using CFD Modelling", Fuel, Volume 113, pp. 780-786, November, 2013.

[7] Knudsen, E.D., and Veenema, D., "Large Eddy Simulation of Premixed and Non-Premixed Gas Combustion", 1996. (Retrieved 2013 www.stanford.edu)

[8] Ferziger, J.H., "Simulation and Modelling of Turbulent Flows", Oxford University Press, New York, 1996.

[9] McMullan, "Large Eddy Simulation of Turbulent Shear Layers", Combustion, 2005.

[10] Rudman, M., and M, Blackburn, M., "Large Eddy Simulation of Turbulent Pipe Flow", 2nd International Conference on CFD in the Minerals, Melbourne, Australia, 6-8 December, 1999.