RMIT UNIVERSITY

Thank you for downloading this document from the RMIT Research Repository.

The RMIT Research Repository is an open access database showcasing the research outputs of RMIT University researchers.

RMIT Research Repository: http://researchbank.rmit.edu.au/

See this record in the RMIT Research Repository at:

https://researchbank.rmit.edu.au/view/rmit:20313

Version: Published Version

Link to Published Version:

http://assetedu.org/viewjc.php?id=j1&page_id=18&volume_id=5&content_id=19

PLEASE DO NOT REMOVE THIS PAGE

Evaluation of brick kiln performances using computational fluid dynamics (CFD)

A H Tehzeeb[a,*], Muhammed Bhuiyan[a] and Nira Jayasuriya[a]

[a] RMIT University, Melbourne, Australia.

[*] Corresponding Author
E-mail: s3313683@student.rmit.edu.au
Tel: +61-422430604

Abstract

Modern history of civilization is concurrent to the use of brick and its manufacturing. Nowadays tunnel kiln is the most widely used technology for brick production. This paper tries to provide an idea of the brick making process in tunnel kiln. The computational fluid dynamics (CFD) software, ANSYS CFX is being used to evaluate kiln performances using gas as its fuel. Simplified geometry was drawn and meshed using appropriate tools of ANSYS CFX. Several pertinent assumptions were made to reduce the complication of the simulation. Turbulence, combustion, radiation and NO models were adopted for simulation of a realistic tunnel kiln environment. Simulated temperature profile almost replicates industrial kiln situation as found in existing literature. CFD analysis helps to simulate the temperature profile of the brick kilns, the mass flow fractions of CO_2 and NO emissions at outlet, and also the air velocity profile inside the kiln. The simulated temperature generated in a tunnel kiln is found to be between 1300K and 300K. CO_2 and NO_x volume generated inside the kiln is estimated as 1.01 m^3/s and 0.108 m^3/s respectively.

Keywords: *brick burner; computational fluid dynamics (CFD); tunnel kiln; CO_x, NO_x emission*

1   Introduction

Tunnel kilns are widely used in ceramic and brick manufacturing industries. In tunnel kiln, green bricks are exposed to a sequence of heat treatment cycles, moving slowly through various temperature zones over the kiln cars. A tunnel kiln consists of a series of connected counter current heat exchangers with the solids on the kiln car moving continuously in the opposite direction of the air flow. This method of heat transfer is popularly known as solid-gas recuperation as some heat from the brick body is recovered by incoming cold air. Tunnel kiln has typical length between 35m and 250m and its width varies from 1m to 6m [1]. The layout of a typical tunnel kiln is shown in Fig. 1, where the three temperature zones as shown are: preheating, firing and cooling.

An in-depth review of literature reveals that not many researches applied Computational Fluid Dynamics (CFD) to evaluate the performance of fuel combustion inside a brick kiln. Most of the researches found on the application of CFD in tunnel kiln are related to understand the combustion situation and how to improve the burner efficiency [2]. Majority of these simulations are modeled in two dimensions; three-dimensional modeling was largely overlooked due to complexity. Research is also conducted to find out the optimum placement of the burner [3]. There are quite a few number of researches where mathematical model of the cooling, preheating and firing zones of a tunnel kiln was developed to increase its efficiency [4][5][6]. Some researches considered the emission performances of

kilns/burners used for other purposes [7].   CFD application in similar other sectors include optimizing the design of different types of burners and improving its emission performances, drying processes, etc [7][8]. Heat transfer efficiency of counter travelling tunnel kiln was analyzed where the brick stakes move in opposite directions in two side by side tunnels [1]. This method of heat transfer in counter travelling tunnel kiln is popularly known as solid-solid recuperation. In this method, heat is extracted from brick body by air and is released to the other counter travelling tunnel kiln. Simulation is also conducted for analyzing the flow of other type of brick furnace (e.g. Hoffman kiln) [9]. Though there are few researches that evaluated the performance of a tunnel kiln using CFD analysis, but none of them considered emission performance of those kilns.

So the purpose of this research is to use this popular tool to evaluate the performance of the brick kiln in three dimensions and identify ways to improve its emission performances. Incomplete combustion and emission are related to improper kiln/burner design. Emissions to air from natural gas fired kilns usually consist of carbon-dioxide, carbon monoxide (CO_x), sulphur dioxide (SO_2), sulphur trioxide (SO_3), oxides of nitrogen (NO_x), gaseous fluorine/chlorine compounds, particulate matters, etc. In this research, only NO_x and CO_x emissions from the brick kiln are modeled. Brick kiln/burner environment is drawn in 3D using CFX Design Modeller. Appropriate turbulence, NO emission, radiation, and combustion
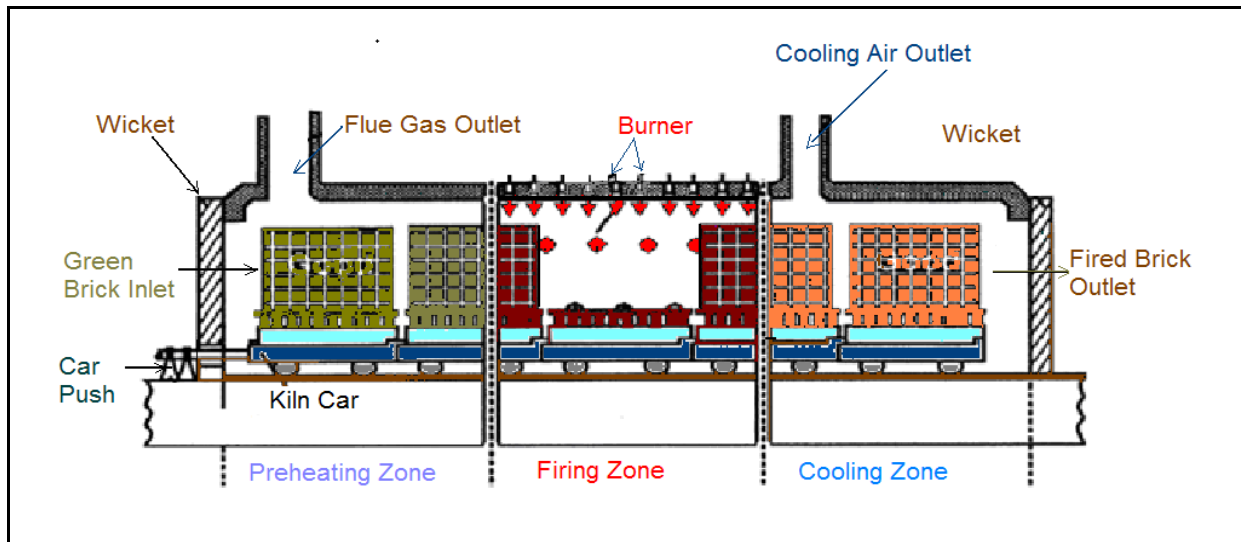
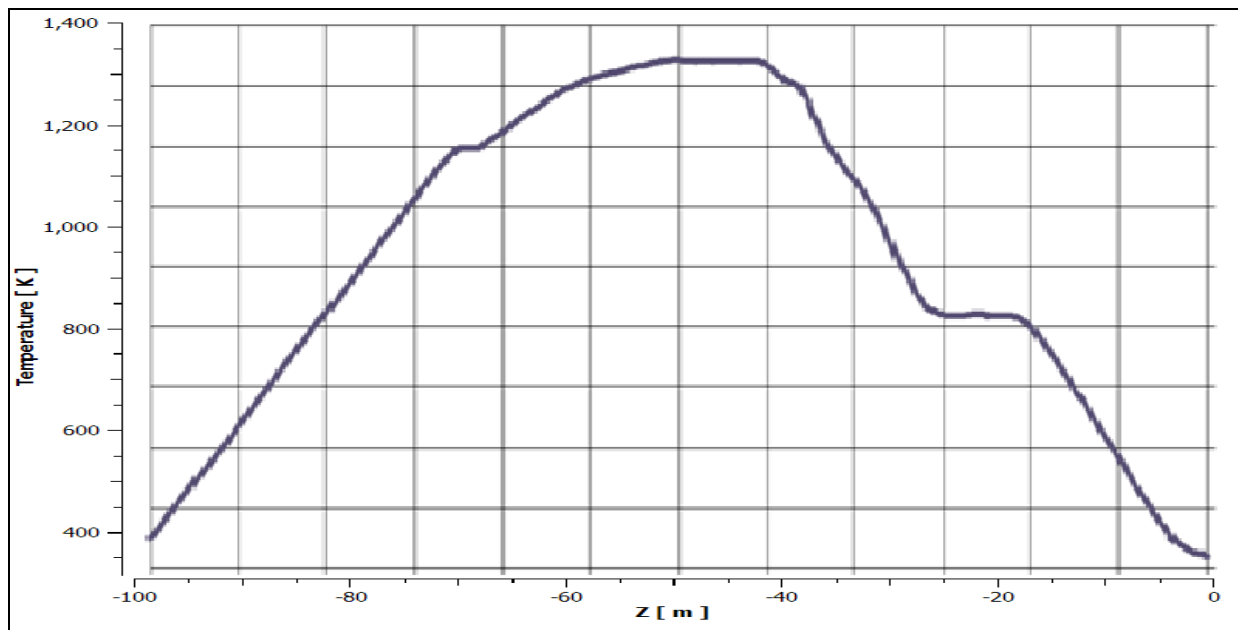**Figure 1** Typical cross-section of a tunnel kiln [1]

Figure 2 Tunnel kiln curve [10]

models are identified from previous researches [8][7]. Inlet and outlet air temperatures and other properties along with fuel are provided as input. Temperature and CO_x, NO_x emissions are simulated using ANSYS CFX software. Air and gas flow velocities in preheating, firing and cooling zones of the kiln are also determined.

2   Kiln Operation Procedure

Green bricks have to go through the procedure of preheating, firing, and cooling processes. Green bricks that will be fired are placed on rail-mounted kiln cars either manually or mechanically. Usually there are around 40 cars or more in a tunnel kiln move opposite to the air flow direction [11]. The cars carrying the green bricks are

pushed slowly through the entire length of the tunnel. There is a fixed period of time for the bricks to stay inside the tunnel kiln. During the firing stage, green bricks undergo through some physical-chemical changes as the temperature rises from room temperature to about 1300-2000 K [11]. In the firing zone, gases are mixed with incoming air flow to burn around gas supply nozzles. As the hot air moves forward, loses its temperature near to the preheating zone. After preheating the green bricks, exhaust gases are forced to leave the kiln with the help of exhaust fans. Concurrently the finished fired bricks on the kiln cars that entered the tunnel earlier are cooled down in the cooling zone by incoming cold air flow. The temperature/time relationships required for kiln operation are principally determined by clay composition. Typically

a tunnel kiln curve shows the characteristics as in Fig. 2. Temperature is risen fast to 900 K. The highest temperature is around 1300 K for typical bricks. In the cooling zone when the temperature is decreasing, it is kept stable at around 800-900 K as at this temperature vitrification occurs. After the completion of vitrification, temperature inside the kiln is steadily decreased to room temperature at the outlet. At a particular interval, entry of one car carrying green bricks through the input end pushes out another car carrying finished bricks at the exit end of the kiln.

3   Firing Process

Firing is the most important stage of brick production, as many important properties of finished bricks are controlled by this process. Strength, abrasion resistance, dimensional stability, resistance to water and chemicals as well as fire resistance are some of these important properties.

When the green bricks are fired in a kiln, most of the soil moistures are driven off at temperatures between 300 and 400 K [1]. Water remaining between the structures of the clay minerals is usually released at temperatures between 500 K and 650 K [1]. Carbonates break apart and releases carbon-dioxide at the temperature range of 1000 to 1200 K.

$$CaMg(CO_3)_2 \rightarrow CaO + MgO + 2CO_2$$
$$CaCO_3 \rightarrow CaO + CO_2$$
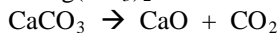
The transformation of the lattice structure of the original clay compound is the most important change takes place to the development of ceramic properties. The next stage is the formation of new crystalline compounds at different glassy phases. In different types of clays, vitrification takes place at different temperatures. Vitrification starts at about 1100 K and is completed at about 1300 K [1]. However, in most cases much higher temperatures are required to accomplish the desired properties in bricks. Heat is conveyed between the air and bricks by convection due to temperature difference.

4   Application of CFD to Evaluate the Performance of the Brick Kiln

Computational fluid dynamics (CFD) is becoming an important research tool in combustion simulation. In recent years, computational fluid dynamics has been used increasingly to improve the efficiency in many industrial applications including combustion processes. With recent advancement of mathematical techniques and simulation performance of the computer due to its high performance hardware, CFD is found to be successful in simulating combustion. The CFD solutions are being used to increase efficiency, optimize and reduce emission from combustion, replacing expensive and time consuming experimentations. Working with CFD involves six

fundamental stages: i) define model goal, ii) create model geometry and grid, iii) set up the solver and physical models, iv) compute and monitor the solution, v) examine and save result, and vi) consider revisions to the numerical or physical models if necessary. In order to initiate a CFD simulation to brick kiln, the limitations made for the model are: the process simulated is under steady state condition, the temperature at any cross section remained temporarily constant, and there is no temperature variation along the brick height inside the kiln.

Building a CFD Model:
To build up a model of a typical burner some simplification was done from the geometric point of view. Simplification was necessary to reduce the complexity of the geometry to effectively simulate what is happening inside the burner. The geometry was successfully created by using pre-processor code ANSYS Design Modeller. The length of the kiln was taken as 100m while the width was 3.2 m. Brick kiln height is taken as 1.5 m. After a brick stack of 440 mm a standard gap of 10 mm is given as shown in Fig 2. Brick height on a kiln car is provided as 1.4 m. Each of the kiln cars has a length of 1.32 m and width of 3.24 m [1]. There is 10 mm clearance between brick stack and the ceiling of brick kiln.

A longitudinal-half of the kiln as shown in Fig. 3, is taken for simulation as because the geometry of the kiln is symmetric. So, one-sixth of the kiln width (540 mm) is taken for simulation as shown in Fig. 4. Only top burners are considered, no side burners are placed in the kiln. Assumption has been made that air enters near the fired brick outlet of the kiln and leaves through the green brick inlet of the kiln. Number of kiln cars inside the kiln is taken as 75, however in real life it may vary according to the actual length. Hot flue gas which is used for preheating green bricks in either solid-solid recuperation or solid-gas recuperation is not considered in this study.

After drawing the geometry, the volume was discretized into finite elements called "cells" using appropriate meshing tool, ANSYS Mesh. These cells are the fundamental units of calculation, as all the derived algebraic equations will be solved at each node of these cells. After meshing, the generated nodes and elements are 104,184 and 101,526, respectively. Most of the elements are hexahedral type as shown in Fig 5. Local body sizing is done to improve the number of elements. Fig. 5 shows the quality of meshing generated by ANSYS Mesh.

Boundary conditions:
As boundary condition, initial velocities for the fuel and air were chosen from [1]. Gas and air inlet temperatures are also defined along with the mass fraction of the components of fuel and air [1]. As initial condition, it is assumed that the domain is consisted of air, where the
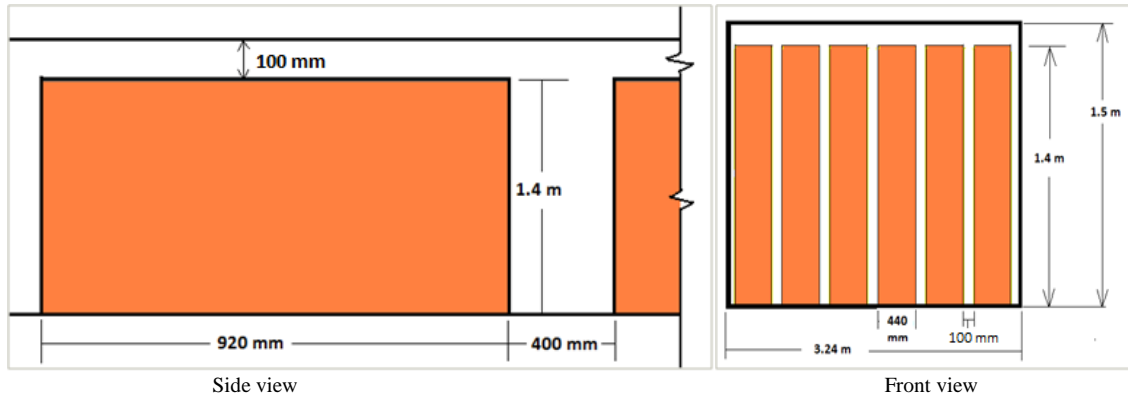
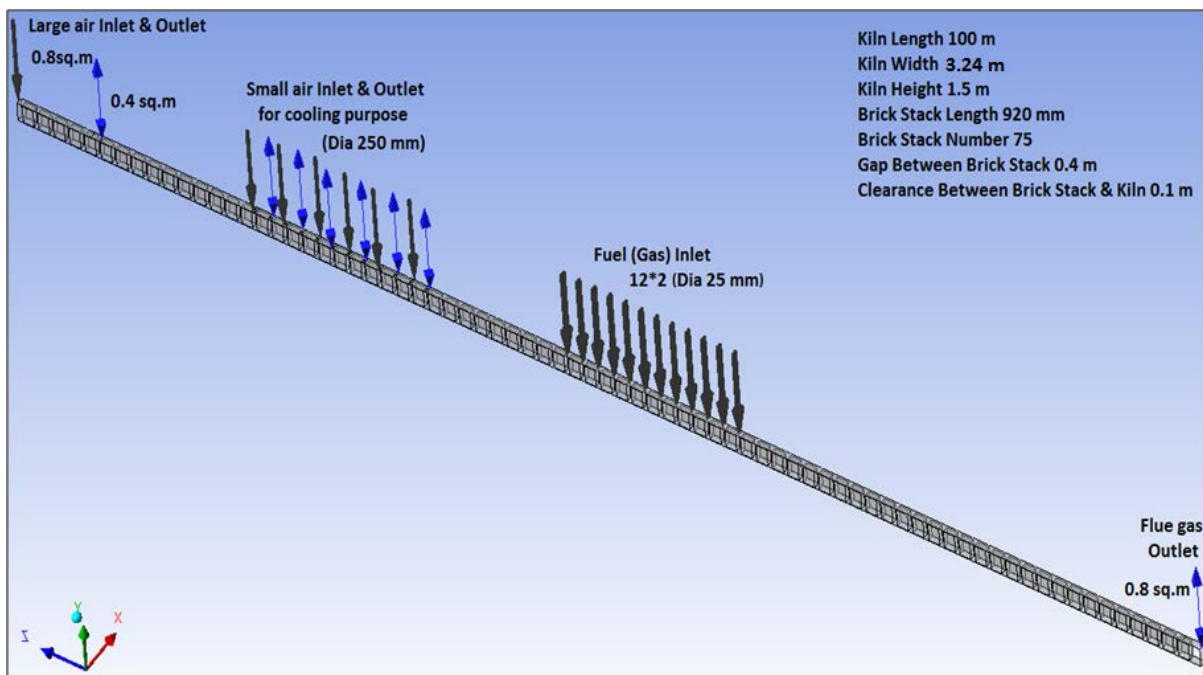**Figure 3** Side view and front view of brick stack in the modelled tunnel kiln

Figure 4 Geometry of a tunnel kiln modelled by ANSYS Design Modeller

fraction of oxygen is only 23.2%. The remaining mass fraction at the inlet will be made up of nitrogen. A small fraction of reaction products (CO_2 and H_2O) has to be given at inlet for the combustion model to initiate simulation.

The default boundary for any undefined surface in CFX-Pre is a no-slip, smooth, adiabatic wall. The external walls of the kiln were considered to be adiabatic, considering the gas nozzle is inside the refractory fireplace. For radiation purposes, the wall is assumed to be perfectly absorbing and emitting surface (emissivity = 1). The kiln wall is non-catalytic, which means, it is not taking part in the chemical reaction. Temperature of the different brick stacks are also given initially, brick stack near the inlet has lower temperature. As it moves near the firing zone the temperature increases. Opposite trend is defined as the brick stack moves from firing zone to the outlet. Heat transfer coefficient for the brick wall is given as 10 W/m²K [1]. The gas volumetric flow is taken as 460 m3/h and gas inlet temperature is assigned as 400 K as found from [1]. The velocity of incoming gas is calculated as 6 m/s. Air inlet velocity is taken as 1 m/s. The models that have been applied for the simulation of the brick kiln are: turbulence model standard, radiation model P-1, and combustion model Eddy Dissipation. As combustion occurs inside the tunnel kiln so there are some turbulence generated. To simulate this turbulence properly turbulence model is necessary. Combustion model is presented to properly simulate the generation of CO2. Separate NO model has to be added to simulate the generation of NO. As high temperature is generated inside the burner so radiation heat transfer also becomes significant, so radiation model is also incorporated. These models are briefly described in the following paragraphs.
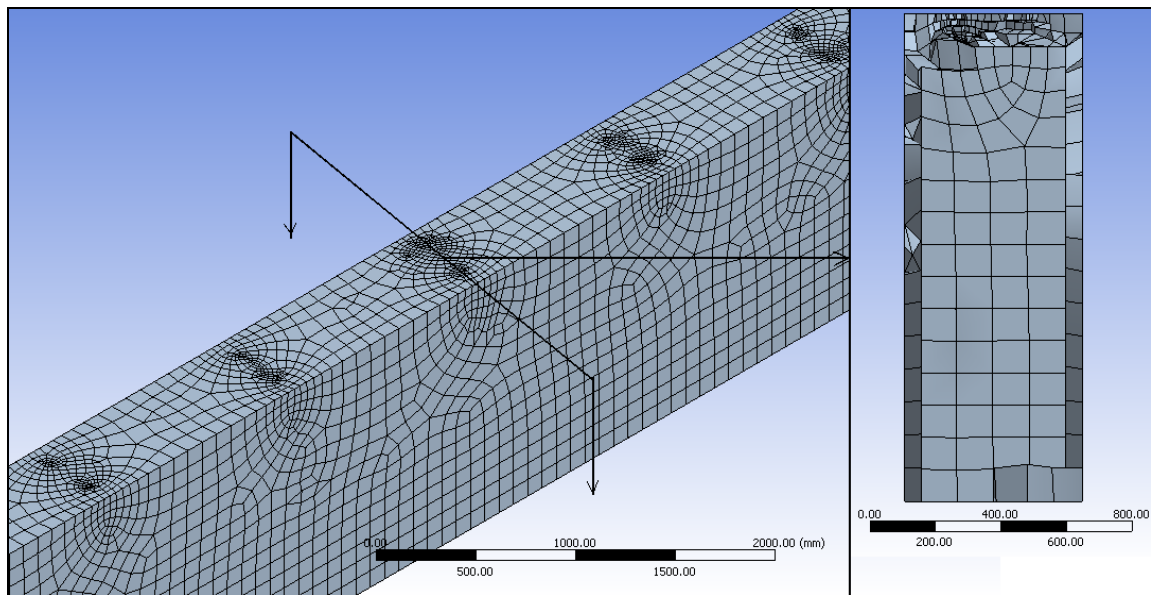
Figure 5 Meshing done by ANSYS Mesh

Turbulence model:

Turbulence models are used to predict the effects of turbulence in fluid flow. There is no turbulence model that has universal acceptability of dealing with all problem areas. The choice of turbulence model will depend upon various considerations such as flow characteristics, established practice for specific types of problems, level of accuracy, available computational resources, and time availability for simulation. To make the most appropriate choice of a model for the brick burner, it was necessary to understand the capabilities and limitations of those various options.

Based on the Reynolds Averaged Navier-Stokes (RANS) equations, a number of models have been developed that can be used to approximate turbulence. Some of these have very specific applications, while others can be applied to a wider class of flows with a reasonable degree of confidence. The following turbulence models available in CFX are: Laminar Model, Zero Equation Model, $k - \varepsilon$ Model, RNG $k - \varepsilon$ Model, $k - \omega$ and SST Models, Curvature Correction for Two-Equation Models, Reynolds Stress Model, Omega-Based Reynolds Stress Models, Explicit Algebraic Reynolds Stress Model, CFX Transition Model, Large Eddy Simulation Model (LES), Detached Eddy Simulation Model (DES), Scale-Adaptive Simulation (SAS) Model, Buoyancy Turbulence Model, etc. These models can be classified as either eddy-viscosity or Reynolds stress models [12].

In this paper, standard $k - \varepsilon$ Model has been chosen for application. The standard $k - \varepsilon$ Model in CFX is one of the simplest complete models of turbulence, in which the solution of two separate transport equations of turbulence velocity and length scales are independently determined [7].

Thermal radiation model:

Radiative heat transfer was included because the radiant heat flux was high compared to the heat transfer rate as a result of convection or conduction. Typically, this will occur at high temperatures, where the fourth-order dependence of the radiative heat flux on temperature implies that radiation will dominate.

Heating or cooling of surfaces because of radiation and/or heat sources or sinks within the fluid phase can be included in our model using a radiation model. CFX includes several radiation modelling options: The Rosseland Model (or Diffusion Approximation Model), the P-1 Model (also known as the Gibb's Model or Spherical Harmonics Model), the Discrete Transfer Model and the Monte Carlo Model [12].

The Differential Approximation or P-1 Model adds an additional transport equation to the simulation. The model has proved adequate for the study of pulverized fuel (PF) flames, in regions away from the immediate vicinity of the flame. However, it has been used for lower temperature values with varying success. This model is only allowed for opaque diffuse walls. [7]

Combustion model:

CFX includes combustion models to allow the simulation of flows in which combustion reactions occur. The models that currently available are: Eddy Dissipation Model (EDM), Finite Rate Chemistry (FRC) Model, Combined FRC/EDM Model, Laminar Flamelet Model for diffusion flames, model for premixed or partially combustion using the Flamelet model for the burned mixture [12]. For this paper, Eddy Dissipation Model (EDM) was used. The EDM was developed for use in a wide range of turbulent reacting flows covering premix
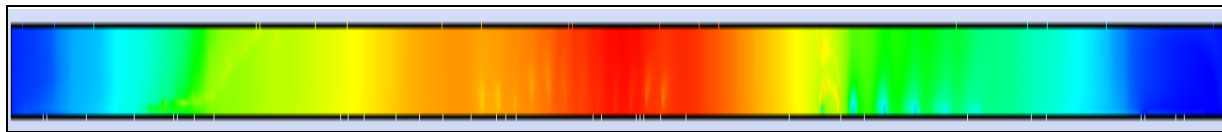
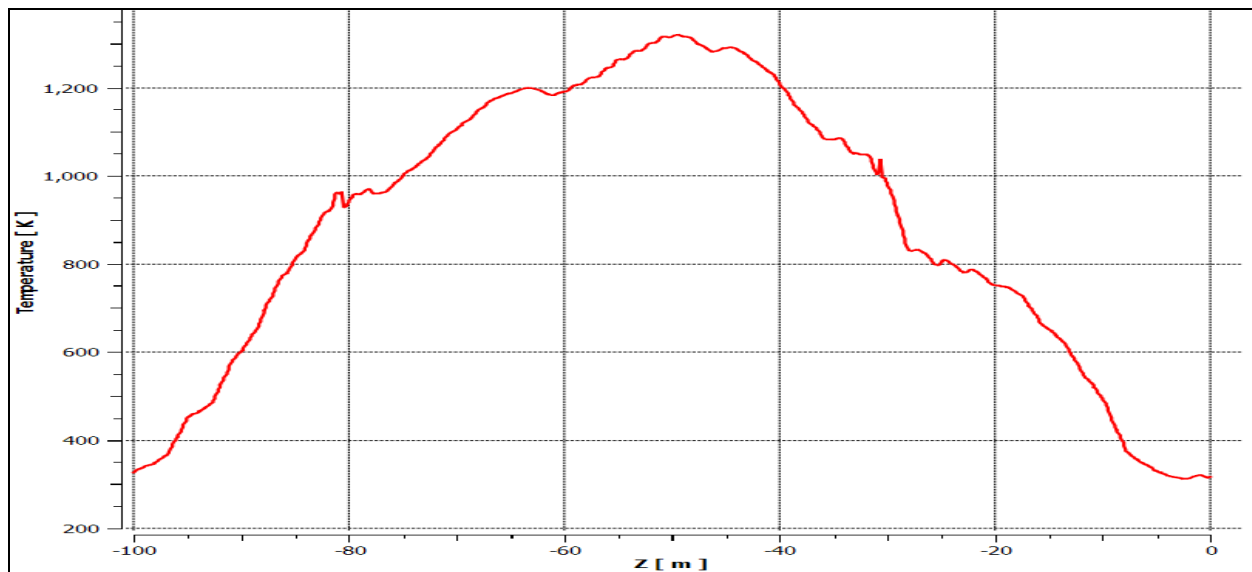**Figure 6 (a)** Temperature profile inside the brick kiln

Figure 6 (b) Temperature profile graph inside the brick kiln

and diffusion flames. Because of its simplicity and robust performance in predicting turbulent reacting flows, this model has been widely applied in the prediction of industrial flames. To simulate the generation of CO_2, combustion model is essential [7].

NO model:

The NO model calculates mass fractions of NO formed in the combustion process. It solves additional transport equations for these variables but does not affect the main combustion calculation. NO concentrations are typically very low and combustion is negligible. The NO is created or destroyed through four mechanisms: Thermal NO, Prompt NO, Fuel Nitrogen, and NO Re-burn. The fuel nitrogen mechanism only affects coal and oil combustion [7]. In CFX, the temperature variance and NO are solved together with the other equations. To model NO, the reaction scheme Methane Air WD1 NO PDF is selected. This introduces NO as an additional component and adds reactions for Thermal and Prompt NO models [12].

5   Simulation using ANSYS CFX

The combustion inside the burner was simulated after defining all the turbulence, combustion, radiation and NO models along with the boundary conditions. The convergence criterion with RMS value is given as 1E-4. After 600 iterations, the solution converged. However it is imperative that, if more iteration is given the result would have improved further. From CFX-Post, temperature profile as well as CO_2 and NO emission profiles are obtained. Temperature profile matches closely with situation given in existing literature [11] which indicates that simulation has progressed in the right direction. However some modification is necessary to match it with the real life situation. Mesh quality has to be improved further and some of the boundary conditions have to be changed. Also the geometry needs to be modified slightly to make it more realistic.

Fig. 6 (a) shows the temperature profile obtained from the CFD analysis. It clearly distinguishes the firing zone, the cooling zone and the pre-heating zone. Also the way heat is transferred between bricks and air is identified. The highest temperature generated as found from Fig. 6(b) is to be 1350 K from the simulation and the lowest temperature is 300 K as initially provided. Fig. 7 shows high CO_2 concentration generated right after the firing zone. CO_2 volume generated inside the kiln is estimated as 1.01 m^3/s. Simulated concentration of NO is shown in Fig. 8, generated mainly in the firing zone. NO_x volume generated inside the kiln is estimated as 0.108 m^3/s. The air and gas flow direction and velocity inside the tunnel are also identified as shown in Fig. 9 (a), (b) and (c). It shows that, at the inlet the air velocity is not that high, however after the combustion zone, the air velocity rapidly increases due to the increase of temperature. Due to the existence of the barrier in the lower position of the kilns, velocity is much higher in upper position of the kiln in comparison to the lower one.
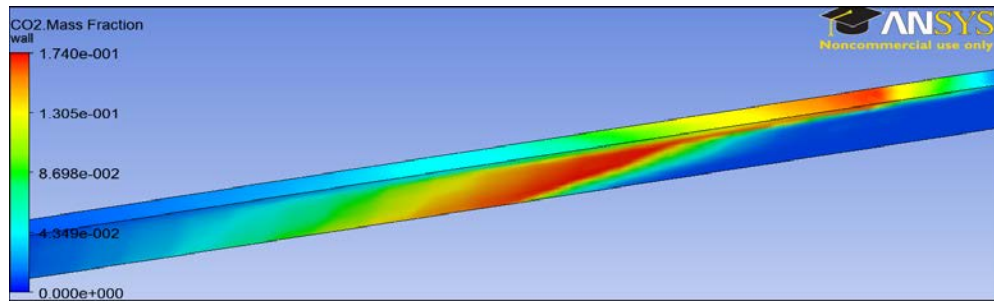
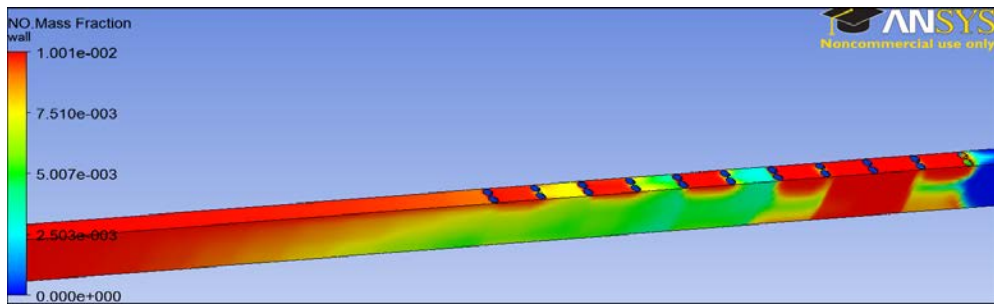**Figure 7** CO$_2$ concentration profile inside the brick kiln

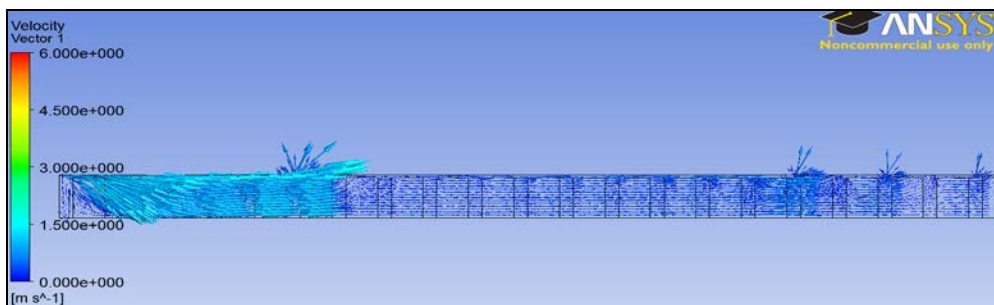**Figure 8** NO concentration profile inside the brick kiln
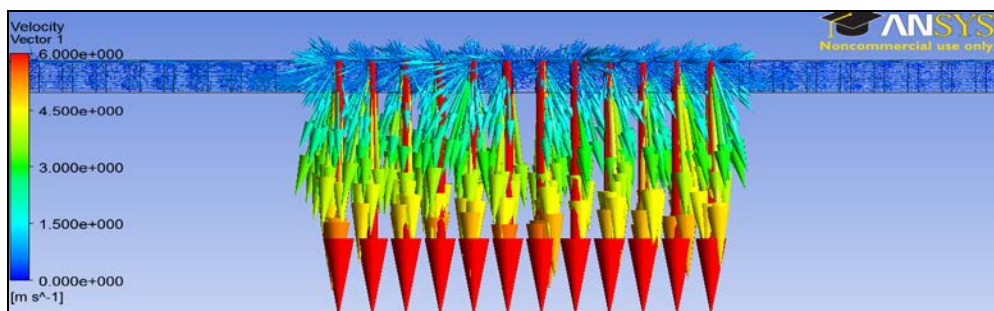

Figure 9 (a) Velocity vector at kiln inlet

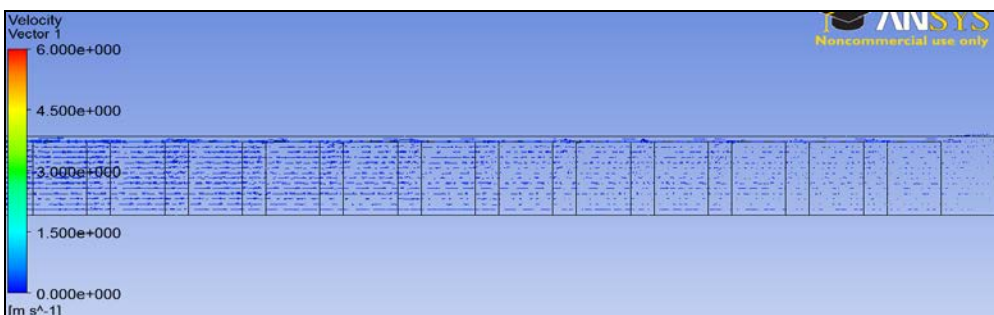**Figure 9 (b)** Velocity vector at kiln burner

Figure 9 (c) Velocity vector near flue gas outlet

The air velocity is below 4 m/s in the preheating zone. Air velocity near the firing zone is around 6 m/s and then the velocity gradually decreases to almost 0 m/s at air outlet. However these values are yet to be validated. The velocity profile shows that the flow of hot air is not uniform through the bricks as some energy is lost by the high speed hot air on the top. Design modification should be done to extract energy from this high speed hot air.

6   Conclusion

A sequence of models on turbulence, NO emission, radiation, and combustion, have been applied for the CFD simulation of brick kiln. Commercially available software ANSYS CFX is used to evaluate temperature profile and CO_x and NO_x emission in the system. From the simulation the highest temperature obtained is around 1350 K and the lowest temperature is around 300 K. So, the simulated values closely match the values provided in previous research and thus justify the simulation. CO_2 and NO_x volume generated inside the kiln is estimated as 1.01 m^3/s and 0.108 m^3/s respectively. The highest velocity inside the tunnel is found to be 6 m/s. Velocity increased gradually from the firing zone to the air outlet. However, in the preheating zone, the velocity was quite low.

At this stage of the research, the result can be compared with real emission data obtained from local industry. However, adopted turbulence, combustion, radiation and NO models can be modified to match it with practical situation. Some modifications can be done to make the geometry more realistic. Once the model is verified against practical data then design optimization can be done to improve the performance of the brick kiln. For design optimization several simulations can be run by changing the design of the kiln and altering the boundary conditions. Then these simulation results can be compared to find out the optimum design of the brick kiln. Also at the same time, CO_x, NO_x emitted from this various designs can be compared and a low emitting design can be suggested.

Clay minerals in Australia have a considerably higher source of fluoride compounds, so fluoride emissions simulations can also be incorporated in future. To perform this simulation, existing reaction formulas of ANSYS can not be utilized. So, a user defined function (UDF) can be included to model the fluoride emission. Velocity profile is also quite important as from this profile, air and gas flow characteristics could be understood and uniform velocity throughout the geometry can be suggested. So CFD simulation will help to improve the performance of a typical tunnel kiln. This study demonstrated that CFD can be used as an effective tool for design and research for brick burning application.

REFERENCES

1. Meng, P., *Solid-solid Recuperation to improve the energy of tunnel kiln*. PhD thesis, Otto von Guericke University, (2011).
2. Sheng, C., Moghtaderi, B., Gupta, R., Wall, T. F., *A computational fluid dynamics based study of the combustion characteristics of coal blends in pulverized coal-fired furnace*, Fuel, (2004). 83: 1543-1552.
3. Hauck, *Optimizing burner placement,* © Elster Group, (2005)
4. Mancuhan, E., *Analysis and optimization of drying of green bricks in a tunnel dryer,* Drying Technology: An International Journal, Taylor and Francis, (2009). 27: 707-713.
5. Kaya, S., Kurtul, K., Mancuhan, E., *Modelling and optimization of heat recovery in the cooling zone of a tunnel kiln,* Applied Thermal Engineering, Taylor and Francis, (2007). 28: 633-641.
6. Kaya, S., Mancuhan, E., Kurtul, K., *Modelling and optimization of firing zone of a tunnel kiln to predict the optimal feed locations and mass fluxes of the fuel and secondary air,* Applied Energy, (2009). 86: 325-332.
7. Chacon , J., Sala, J. M., Blanco, J. M., *Investigation on the design and optimization of a low NO_x-CO emission burner both experimentally and through computational fluid dynamics (CFD) simulation.* Energy & Fuels, (2007). 21: 42-58.
8. Jamaleddine, T. J., Ray, M. B., *Application of computational fluid dynamics for simulation of drying process: A Review*, Drying Technology: An International Journal, Taylor and Francis, (2010). 28: 120-154.
9. Garcia, C., A., Sanchez, F., Pirajan, C., M., *Simulation and flow analysis for a brick furnace*, Electronic Journal of Environmental, Agricultural and Food Chemistry, (2006). 5: 1500-1508.
10. Durakovic, J., Delalic, S., *Temperature field analysis of tunnel kiln for brick production,* Temperature field analysis of tunnel kiln for brick production, (2006). 53:3 403-408.
11. Yu, B., *Dynamic modeling of a tunnel kiln,* Heat Transfer Engineering, Taylor and Francis, (2007). 15:2 39-53.
12. ANSYS Inc, *ANSYS CFX help file and tutorial*, version 13, © SAS IP, (2009).