**Proceedings of the
International Conference on Mechanical Engineering 2003
(ICME2003) 26-28 December 2003,Dhaka, Bangladesh**

ICME03-FL-04

CFD APPROACH FOR MODELLING OF COMBUSTION OF A SEMI ENCLOSED COOKING STOVE

W.M.S.R.Weerasinghe,U.D.L.Kumara

Department of Mechanical Engineering, University of Moratuwa,
Moratuwa 10400,
Sri Lanka.
e-mail: rohitha@mech.mrt.ac.lk

ABSTRACT

This paper presents a CFD approach for modelling of combustion in a cooking stove. The combustion inside a semi-enclosed cook stove has been modelled. The stove is made of fire-brick clay and the cooking pan is an Aluminum square plate. The combustion phenomena inside the stove is modeled using the Eddy-break-up combustion model and the k-ε turbulence model. The flow considered, is due to the buoyancy effect and the spatial variation of species concentrations. Empirical data of a typical cook stove has been used as boundary conditions. Only the flaming mode of combustion has been modelled. Temperature, Velocity and combustion product concentration predictions are presented. The predicted temperature fields have been compared with the measurements.

Keywords: Eddy break up, k-ε turbulence, Combustion efficiency

1.INTRODUCTION

Energy sector will be the driving force of the modern system of social development and the global economy. It is clear that the energy sector should be properly optimized for the better subsistence of human being. It should be balanced ecologically, economically and socially. Present attention is paid towards the non-conventional energy sources due to the fast depletion of fossil fuels and rate of growth of population. Among various alternatives biomass holds a special pledge due to its intrinsic ability to store solar energy and it being the only renewable source of carbon and a pool of other chemicals.

The energy sector of Sri Lanka is dominated by bio energy. Generally 50% of energy demand is supplied by bio energy and 90% of that is consumed for cooking purposes hence, one of the most energy-consuming devices in Sri Lanka is the cook stove [1]. Cook stoves are associated with drawbacks that heavily affect the performance of stove. Low overall efficiencies and high levels of emissions are the main drawbacks [4]. The existing designing process of stoves does not properly address these problems. This is mainly due to that the designing process is mostly relied on empirical information and trial and error methods. An appropriate analysis of the fluid flow inside the stove, which is caused by buoyancy effect due to the temperature variation existing in the combustion region and the spatial variation of species concentrations, has hardly been addressed yet. This paper presents a CFD approach, which has addressed the above drawbacks.

Computational Fluid Dynamics can be used to analyze the problems that are associated with fluid flows, reacting and non-reacting systems. During this effort CFD has been applied to simulate the semi enclosed fuel wood stove which is made of clay. **Figure 1,** shows the stove and the computational domain of the problem.

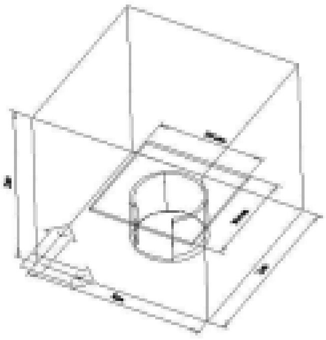

Figure 1. The configuration of semi-enclosed cook stove

The governing equations of fluid flow and heat transfer, which are known as conservation laws of physics can be applied to the control volume. For analysis, fluid will be regarded as a continuum. By applying conservation laws it is possible to derive a set of equations, that describe the process of momentum, heat and mass transfer of the control volume. These are partial differential equations. They have no known general analytical solutions but can be discretised and solved numerically. Equations describing the combustion process are solved in conjunction with above set of equations. Approximate models were used to derive these additional equations. Eddy Breakup combustion model and the k-ε turbulence model were selected as the combustion and flow models respectively. Finite volume technique was used to solve equations iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained.

CFX-5.6 CFD code was used to model the above conditions on a computer-based environment. By using these codes the predictions of temperature profiles of the stove and cooking pan were taken and compared with experimental values. It is expected to use these temperature profiles to optimize the heat transfer efficiency by changing the governing parameters of the stove.

2.EXPERIMENTAL ARRANGEMENT

The cooking stove measurements were taken in a typical field environment. Figure1.1 shows the experimental arrangement and the control firebox, the semi-enclosed stove has a diameter of 23cm and height of 16cm. It is made out of brick fire clay having a density of 1600kg/m^3 and an opening angle of 70^0. The mild steel-cooking pan of 36cmx36cmx5mm was placed symmetrically on the cooking stove. The cooking stove was filled with firewood. It is assumed that the firewood produces wood-gas only, which has a composition of 50 % Methane, 30% Carbon Dioxide and 20% Hydrogen.

Type K thermocouples of diameter 1.5mm were mounted along the two symmetrical axes of the cooking pan. Temperature readings of the plate surface were logged through a data logger until the readings became steady.

3.MATHEMATICAL MODEL

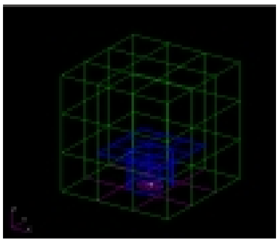

Figure 2. Domain for the mathematical model

It can be considered that the above cubical computational domain (1x1x1m^3), that includes the stove and the cooking plate (0.36x0.36x0.005m^2) as the geometrical space for the governing equations. The fluid flow due to combustion within this region is being modelled by the governing equations. Considering a small element of the domain with sides δ x, δ y, δ z, we can derive mathematical equations for the conservations laws of physics; Conservation forms of the governing equations of the computational domain can be expressed as follows. Generally this set of equations is called as the Navier –Stokes equations in their conservative form.

Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho U) = 0 \tag{1}$$

The Momentum equation

$$\frac{\partial \rho U}{\partial t} + \nabla \bullet (\rho U \otimes U) = \nabla \bullet (-P\delta + \mu(\nabla U + (\nabla U)^T)) + S_M \tag{2}$$

The full buoyancy model is implemented by adding a source term to above momentum equation. The buoyancy force source term is a function of the local density variation

$$S_{M,buoy} = (\rho - \rho_{ref})g \tag{3}$$

The Energy Equation

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial P}{\partial t} + \nabla \bullet (\rho U h_{tot}) = \nabla \bullet (\lambda \nabla T) + S_E \tag{4}$$

Where h_{tot} –Specific Total Enthalpy
It is defined as $h_{tot} = h + 1/2U^2$
Where h = h(P,T)
The change in enthalpy dh is calculated in two steps first at constant pressure then at constant temperature. The first step is equivalent to the change in enthalpy for an ideal gas, while second step is a correction required to for real fluids. The total chage in enthalpy is given by

$$h_2 - h_1 = \int_{T_1}^{T_2} c_p \, dT + \int_{P_1}^{P_2} [\upsilon - T_2(\frac{\partial \upsilon}{\partial T})] \, dp \tag{5}$$

The k-ε Flow Model

Industry standard two-equation turbulence model was used. k is the turbulent kinetic energy and is defined as variance of the fluctuation in velocity. ε is the rate of dissipation of turbulence kinetic energy. This model introduces new variables in to the system of equations. The momentum equation becomes

$$\frac{\partial \rho U}{\partial t} + \nabla \bullet (\rho U \otimes U) - \nabla \bullet (\mu_{eff} \nabla U) = \nabla P' + \nabla \bullet (\mu_{eff} \nabla U)^T)) + B \tag{6}$$

Where B is the sum of body forces, μ_{eff} is the effective viscosity accounting for turbulence, P' is modified pressure given by P' =P +2/3ρk

This is based on eddy viscosity concept, so that
$\mu_{eff} = \mu + \mu_t$
The values for k and ε can be calculated from differential transport equations.

$$\frac{\partial \rho k}{\partial t} + \nabla \bullet (\rho U k) - \nabla \bullet (\frac{\mu_{eff}}{\sigma_k} \nabla k) = P_k - \rho \varepsilon \tag{7}$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \nabla \bullet (\rho U \varepsilon) - \nabla \bullet (\frac{\mu_{eff}}{\sigma_\varepsilon} \nabla \varepsilon) = \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon) \tag{8}$$

Where $C_{\varepsilon 1} = 1.45$, $C_{\varepsilon 2} = 1.9$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$ are constants.

The Eddy break-up Combustion Model

The energy release as heat due to combustion process can be approximated. The rate of consumption of fuel is specified as a function of local flow properties [2]. The model considers the dissipation rates of fuel, oxygen and products, and takes the slowest rate as the reaction rate of fuel. This value is used as the source term of the transport equation for mass fraction of fuel.

$$R_{fu} = -\rho\ (\varepsilon /k) \min\ [C_R m_{fu},\ C_R\ (m_{ox}/s),\ C'_R\ (m_{pr}/1+s)] \tag{9}$$

The transport equation for mass fraction of fuel

$$\frac{\partial(\rho m_{fu})}{\partial t} + \nabla \bullet (\rho m_{fu} U) = \nabla \bullet (\Gamma_{fu} \nabla m_{fu}) + R_{fu} \tag{10}$$

Assumptions

The combustion power out put of the stove is mainly dependant upon the flaming combustion of firewood. It is assumed that 75 % of the total power of the stove comes from flaming combustion, therefore, only the flaming combustion is modelled. The flaming power out put of the stove is assumed to be 3kw.

4.COMPUTATIONAL MODEL

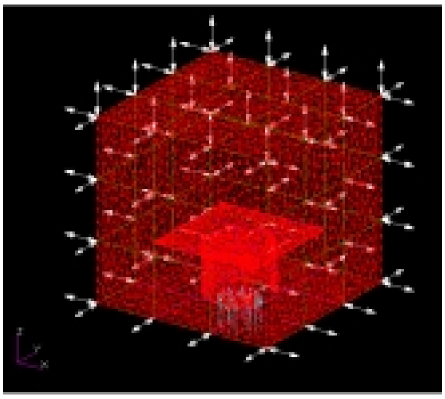

Figure 3. The Computational domain with 200,000 grid nodes

4.1 Boundary Conditions

Conservation equations are solved by the numerical methods in which the boundary conditions will be the preparatory points of the equations. It is important to take the right boundary conditions that are most appropriate to the practical problem. There are three types of boundary conditions have been used for the simulations while getting a practical in- touch with the cook stove problem.

4.1.1 Inlet

Inlet boundary conditions were used where it is known that the flow is directed in to the domain. The fuel inlet for the domain belongs to this category. Details of the boundary conditions are given in T**able 1.**

Table 1. Inlet boundary conditions

Condition	Value
Static temperature	300 K
Flow regime	Subsonic
Black body temp.	300 K
Turbulence	5 % (Intensity)

4.1.2 Opening

This boundary type is used where we have some boundary information but don't know whether the flow is into to or out to the domain. Six sides of the cubical domain belong to this category. Details are as follows;

Table 2. Opening boundary conditions

Condition	Value
Relative pressure	0
Static Temp.	300K
Turbulence	3.7 %(Intensity)
Black Body Temp.	300 K

4.1.3 Wall

These are solid boundaries to fluid flow. Walls allow the permeation of heat and additional variables into and out of the domain. Details of wall boundaries are as follows.

Table 3. Wall boundary conditions

Condition	Value
Wall influence on flow	No slip
Wall roughness	Smooth
Heat transfer	Domain interface
Emissivity	Grey color

4.2 Computational Procedure

Computational domain and the physics of the simulation were built on the builder of the CFX-5.6. The conservation equations were discretized using the finite-volume method in the solver. The SIMPLEC algorithm achieved velocity and the pressure relation. The total time duration for the simulation was 16 hrs & 33 minutes.

5.Results & Analysis

5.1 Simulation Results

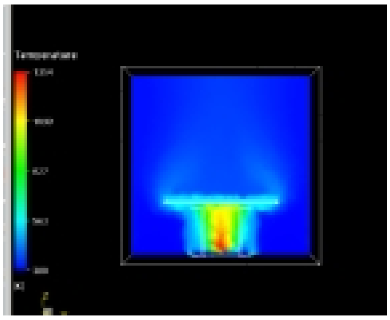

Figure 4. Temperature profile on symmetric plane

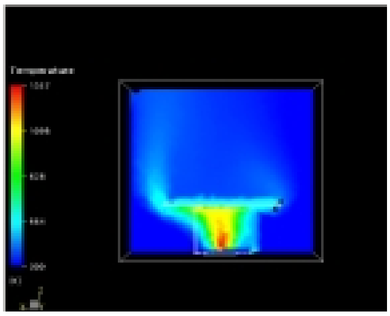

Figure 5. Temperature profile on asymmetric plane

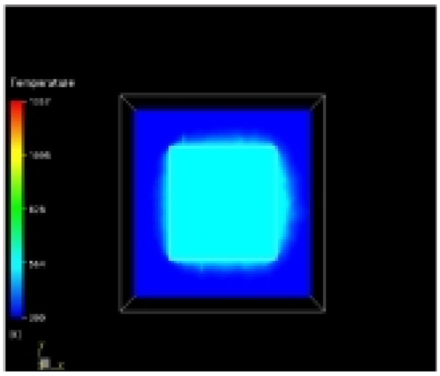

Figure 6. Temperature distribution on the plate

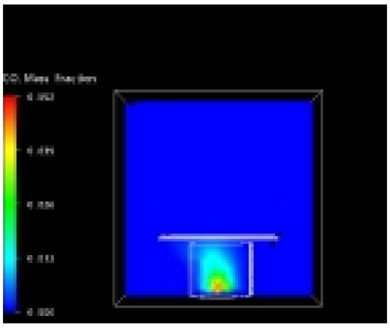

Figure 7. Carbon monoxide distribution

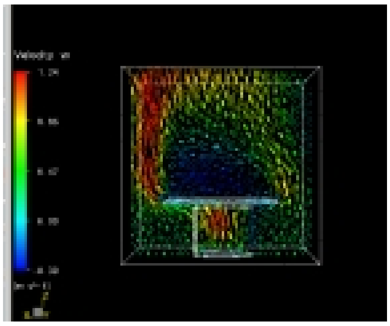

Figure 8. Velocity profile on asymmetric plane

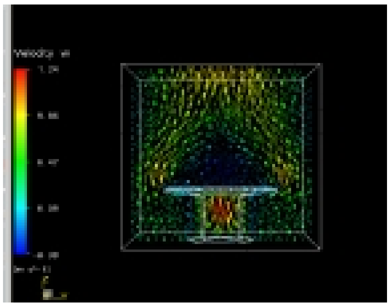

Figure 9. Velocity profile on symmetric plane

Table 4.Measurements

Distance from centre(cm)	Avg.Temp.(K)
-9	603.6
-6	664.8
-3	639.1
3	663.1
6	720.4
9	657.5

Table 5. Simulation Results

Distance from centre(cm)	Temp.(K)
-9	527.4
-6	538.6
-3	574.3
3	554.7
6	541.4
9	532.1

5.2 Analysis

The overall efficiency of the stove will be dependant upon on combustion and heat transfer efficiencies.It is important to note that the steady state efficiencies are have been cosidered for the caculations.

Overall efficiency = Combustion EfficiencyxHeat tranfer efficiency

Combustion Efficiency [2]

$$\frac{(Q_{net,p})_{woodgas} - \sum m_i(-\Delta H_{25})_i}{(Q_{net,p})_{woodgas}} \qquad (11)$$

Heat Transfer Efficiency

$$\frac{ms(T_1 - T_2)}{(Q_{net,p})_{woodgas} - \sum m_i(-\Delta H_{25})_i} \qquad (12)$$

Numerical calculations have been done and it is expected to optimize the heat transfer efficiency by varying geometrical goverving parametrs of the stove.

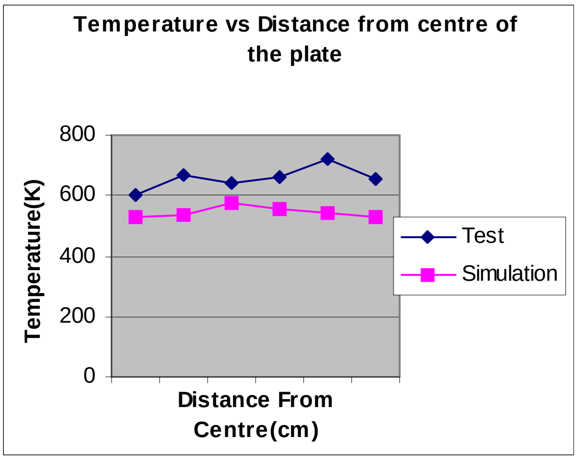

Figure 10. Comparison of Measurements & Simulation results

6.Conclusion

Comparisons are made between the measurements and simulated temperature profiles of the cooking plate. It shows that the discrepancies between two methods are not significant hence, the simulated results can be used to calculate the performance parameters of the stove but it should be noted that burning rates at small radiation levels are usually difficult to simulate and the results may some what differ from the actual values. Simulation is being considered only the flaming mode of combustion, so that also may be a reason for differences between the experimental and simulation results. The calculations of heat transfer and combustion efficiencies have been done. It is expect to vary the stove height and find the most appropriate height for the optimum heat transfer efficiency of the stove.

7.REFERENCES

[1]. K.K.C.K.Perera & A.G.T. Sugathapala,Fuelwood-fired cookstoves in Sri Lanka and related issues,Energy for Sustainable Development,the journal of the international Energy Initiative,Volume VI, No 1,March 2002

[2]. H.K.Versteeg & W.Malasekara, An introduction to Computational Fluid Dynamics, The Finite Volume Method, Longman Group Ltd., 1995

[3] .W.M.S.R.Weerasinghe & R.M.P.S.Bandara, Computational Modelling of Cook Stoves, ERU Symposium, 2002, University of Moratuwa, Sri Lanka.

[4]. W.M.S.R.Weerasinghe & R.M.P.S.Bandara, Computational Modelling of a Wood fired Semi Enclosed Cooking stove, HEFAT 2003,2nd International Conference on Heat Tranfer,Fliud Mechanics and Themodynamics,June 2003,Victoria Falls,Zambia

7.NOMENCLATURE

Symbol	Meaning	Unit
u	Velocity X direction	$\mathrm{ms^{-1}}$
v	Velocity Y direction	$\mathrm{ms^{-1}}$
w	Velocity Z direction	$\mathrm{ms^{-1}}$
P	Static Pressure	$\mathrm{Nm^{-2}}$
μ	Dynamic Viscosity	$\mathrm{kgm^{-1}s^{-1}}$
c_p	Specific heat capacity at constant pressure	kJ
λ	Thermal conductivity	$\mathrm{kgms^{-3}K^{-1}}$
ρ	Density	$\mathrm{kgm^{-3}}$
g	Acceleration of gravity	$\mathrm{ms^{-2}}$
k	Turbulence Kinetic Energy	$\mathrm{m^{2}s^{-2}}$
T	Static Temperature	K
h	Static enthalpy	kJ
m	Conservative mass fraction	_
I	Radiation intensity	$\mathrm{kJm^{-2}}$
Γ	Diffusivity	$\mathrm{Kgm^{-1}s^{-1}}$
S	Source term	_
t	Time	s
U	Velocity vector	$\mathrm{ms^{-1}}$
M	Momentum	$\mathrm{kgms^{-1}}$
E	Energy	kJ
x,y,z	Directional components	_
t (subscript)	Turbulent	_
ε	Rate of dissipation of turbulent	$\mathrm{m^{2}s^{-3}}$