University of Strathclyde
Engineering

CFD DESIGN OF IMPROVED COOKSTOVES

John James Brewster

4th Year Mechanical Engineering Student
2005-2006

Supervised by Dr. Tom Scanlon

ABSTRACT

This study focuses on the development of smokeless stoves for application in Nepal. The aim of the study is to identify feasible ways of utilising computational fluid dynamic software to assist in the development of cookstove designs. In light of this, a holistic understanding of the present situation in Nepal has been sought after.

The Nepali Insert Stove design was selected for CFD simulation and analysis. Once the effects of natural convection and turbulence throughout the model were accounted for, the process of optimising the design with a goal of reducing the amount of smoke back drafted from the stove's front door, was investigated. An optimum baffle angle of 40° was found to induce the smallest amount of smoke back drafted into the room where the stove would be located.

It is concluded that the study brings light of a feasible and concurrent method of cookstove development. It is however recommended that experimental data is gathered and further research into other designs and geometry variations is conducted prior to making modifications to the current manufacturing of improved cookstoves in the developing world.

TABLE OF CONTENTS

1.0 INTRODUCTION

1.1 Definition of the Problem

It was reported by the World Heath Organisation in 2002 that more than two billion people worldwide continue to depend on solid fuels, including, wood, animal waste, agricultural residues and coal, for their energy needs. Utilising such sources to meet cooking and space heating requirements results in high levels of indoor air pollution being produced. Indoor air pollution as the name suggests is caused by the burning of fuel indoors, and can even be polluted from kerosene lamps. The majority of the problem however is the result of smoke back drafted from stoves or in the most extreme case produced from an open fire where no attempts have been made to extract the smoke produced.

It can be seen that indoor air pollution is a global problem, however its most significant effects can be witnessed in developing world countries where open fires and cookstoves are still widely used and furthermore heavily depended upon for cooking and space heating needs.

The hazardous effects of indoor air pollution are a result of incomplete combustion, and the resulting wide range of pollutants, such as small particles and carbon monoxide, produced. The fact that it has been reported by the World Health Organisation that 2.7% of the burden of global disease is a result of indoor air pollution, is an illustration of the severity of the problem.

1.2 Focus of this Study

This study focuses on the use of Computational Fluid Dynamics to assist in the optimisation of stove designs, with a hope to obtain a more concurrent strategy to obtaining possible solutions to the problem of indoor air pollution. Furthermore the study homes in on the CFD development of stoves in Nepal, and the Nepali Insert Stove in particular. The stove is constructed from pottery and mud, and does not rely on the expensive materials for manufacture. It would be all very well designing a superbly efficient stove, with minimal smoke emissions, however, if it were not possible to manufacture it due to limitations of resources or if it were not possible to distribute it, then any recommendations made as to development in the field would already be obsolete.

First hand experience of the situation present in the country was established after a trekking expedition around the Sagarmatha National Park for three weeks in May of 2005. It is hoped that the exposure to the country and broadened understanding of local requirements, enhances the study.

Figure 1 – An example of the type of stoves in use in the Sagarmatha region of Nepal. Note the carbon build-up on front of the stove.

Since no experimental data with regards to the amount of smoke emitted from the Nepali Insert Stove exists, approximations and assumptions needed to be made. In light of this, a general study exploring the effects of varying the stove's geometry on the amount of smoke back drafted in to the room was conducted, once a respectable model of the stove was generated.

1.3 Objectives

The objective of this study is to ascertain *feasible* methods of applying technology that exists in the modern developed world to assist in solving the problem of indoor air pollution. Furthermore to produce conclusive evidence that CFD can be used in a viable manner to assist in the development of smokeless cookstoves.

2.0 TWO DIMENTIONAL CFD MODEL

2.1 Computational Fluid Dynamic Software

This study utilises the use of Gambit and Fluent codes for Computational Fluid Dynamic simulations and analysis. Gambit is a pre-processor used to define the geometry of the model and to create a computational mesh. Boundary conditions for the problem (e.g. walls, velocity inlets etc) and continuum types (e.g. fluid, solid) are also created using Gambit. Fluent solves the CFD model generated after various initial parameters are set. Fluent may also be used as a post-processor to visualise and analyse the fluid flow. Colour vector and contour plots may be created for parameters such as velocity, pressure and temperature, to assist in giving a visual representation of the fluid.

2.2 The Nepali Insert Cookstove

A decision was made to base the CFD study on the Nepali Insert Cookstove. The two-pot stove is manufactured from four separate pottery sections, the details of which can be seen in *figure 2.*

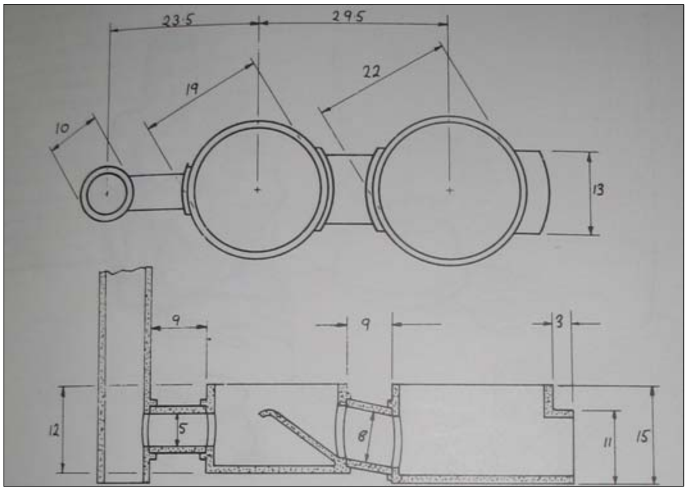


Figure 2– Nepali Insert Stove's Geometry

2.3 Generation of Cookstove Geometry and Rationale of Analysis

The process of defining a model with the use of Gambit is fairly elementary once a suitable stove geometry has been determined. Various assumptions with regards to the stoves geometry were made to simplify it in order to ease the process of simulation. Firstly, a two dimensional representation of the cross-section of the stove was utilised, this is considered an appropriate approximation to make for the purposes of this investigation. Experimenting with a three dimensional model was carried out, however, it was deemed too time consuming and unnecessary at this stage of investigation. *Figure 3* displays the geometry defined.

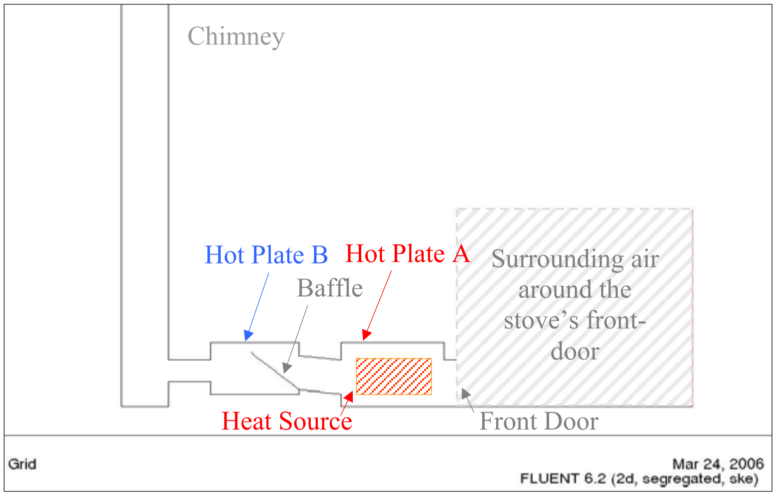


Figure 3 – Defining the Stove Geometry

Due to the nature of the Nepali Insert Stoves geometry, it was considered appropriate and interesting to monitor the effects of the stoves performance when the baffle angle, beneath the second hot plate, was varied. Eleven angles were defined, and the resulting varying performance data was collected.

2.4 General Assumptions for the Model

The majority of previous work in the field concerns with methods of improving the combustion efficiency of stoves in general. This study however essentially investigates ways reducing smoke emissions from the front door of a stove. It is therefore possible to ignore many of the variables of the combustion process. This can be considered acceptable as fundamentally the primary factors effecting the transferral of smoke through the stove's body are;

- The pressure gradient that exists between the entrance and exhaust of the stove, dependant only on the height of the chimney and the effects of gravitational pull.
- The temperature gradient inflicted by the magnitude of the heat source.
- The resulting effects of turbulence in the stove, equated for by the turbulence model and parameters used.

Summary of additional assumptions for the model;

- A two dimensional study with approximations being made as to the size of heat source.
- Composition of smoke is assumed the same as air.
- Concerned only with natural convection occurring in the stove. The effects of radiation and conduction are considered negligible.
- Effects of turbulence accounted for with k-ε turbulence model, with the effects of boundary layers being considered negligible.

2.5 The Mathematics of CFD

Fluent utilises an integrated discretized Navier-Stokes solver. This discretizion process generates a set of linear algebraic equations relating the flow field variables (pressure, velocity, turbulence etc) at numerous node points of the mesh. Once reasonable boundary and initial conditions have been defined, it is possible to solve these equations iteratively for the flow field variables at each computational point. It is important to stress due to the iterative nature of the solution process, it will only ever provide approximate numerical solutions to the governing equations. The accuracy of which is determined by a number of factors. The investment of time and computing power is required to produce valid results, however the exact amount of the investment of both can be reduced with good discretizion and careful monitoring of the solution.

2.6 The Governing Equations of Natural Convection

When heat is added to a gas, specifically air in the case of a stove, it expands, and thus changes density. The presence of gravity and change of density induces a change in the body forces, and the forces cause the fluid to move "by itself" without any externally imposed flow velocity. This is the phenomenon of natural convection. This phenomenon would occur throughout the stove, and the main driving force for the fluids flow would be buoyancy, inflicted by high a temperature gradient.

The governing equations for natural convection flow are of a considerable complexity. Hence several approximations have therefore to be made, with the effects of the varying density of the air in the model being accounted for by the Boussinesq approximation. The governing equations of Natural Convection are therefore defined by e*quation set 1.*

- Mass

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

- Energy

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\frac{\partial^2 T}{\partial y^2}$$

Equation set 1

- Momentum

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=g\beta(T-T_\infty)+v\frac{\partial^2 u}{\partial y^2}$$

where, the coefficient of volumetric thermal expansion,

$$\beta=-\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p=\frac{1}{\rho}\left(\frac{p}{RT^2}\right)=\frac{1}{T}$$

2.7 Turbulence Modelling

Due to the buoyant nature of the air flowing throughout the stove, turbulence will undoubtedly exist. To provide an accurate representation of what would occur in reality and a good platform for analysis accounting for its effects is a very important part of the CFD simulation.

The turbulence modelling capabilities of Fluent are very varied, and since the extent of previous CFD work carried out in the field of study is limited, it is hard to determine the best model to utilise. However after taking some advice and reviewing the turbulence models available, it was decided that the use of a Navier-Stokes solver with a two-equation k-ε model would be appropriate. The model is widely used within the CFD community for simulating turbulence and is considered to be the best of the eddy viscosity models due to its good numerical stability.

The two-equation, k-ε eddy viscosity turbulence model considers the convective and diffusive transportation of the turbulence itself. The model is named after its two transported variables k and ε, where k is the turbulent kinetic energy of the flow and ε is the viscous dissipation rate.

In common with all other eddy viscosity models, in the k-ε model, the Reynolds stresses are obtained from the Boussinesq approximation and the turbulent kinematic viscosity takes the Prandtl – Kolmogorov form. These are combined to form the two transport equations which are solved numerically by the computer; these are given in *equation set 2*.

- k- Transport equation

$$\frac{\partial\left(\overline{U}_j k\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left\{\left(\frac{v_l}{\sigma_l}+\frac{v_t}{\sigma_k}\right)\left(\frac{\partial k}{\partial x_j}\right)\right\}+P_k-\varepsilon$$

Equation set 2

- ε- Transport equation

$$\frac{\partial\left(\overline{U}_j \varepsilon\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left\{\left(\frac{v_l}{\sigma_l}+\frac{v_t}{\sigma_\varepsilon}\right)\left(\frac{\partial \varepsilon}{\partial x_j}\right)\right\}+C_{\varepsilon 1}\frac{\varepsilon}{k}P_k-C_{\varepsilon 2}\varepsilon$$

Where, $P_k = v_t \frac{\partial \overline{U}_i}{\partial x_j}\left(\frac{\partial \overline{U}_i}{\partial x_j}+\frac{\partial \overline{U}_j}{\partial x_i}\right)$ is the production rate of turbulent kinetic energy.

Rigorous derivations of the above equations are available from some of the various references utilised in this study but for the purposes investigating stove design no further understanding is necessary other than to note the limitations of the model.

It is normally considered a limitation that the k-ε model is a high Reynolds number version turbulence model, i.e. is only valid in fully turbulent cases. However for the purposes of this study deemed appropriate, since the nature of the air travelling through a stove since is considered highly turbulent.

3.0 INITIALISING AND SOLVING THE CFD MODEL

3.1 Inputting of the Various Parameters

The first step in inputting of the various parameters into Fluent is the activate energy equation. Furthermore the definition of a heat source in the heart of the first pottery section, beneath hot plate A. The size of the heat source determines many characteristics of the fluid’s flow; hence it is important that a good approximation for the stoves heat source is made. In addition it is important to review the resulting hot plate and flue gas temperatures after the model is solved as the temperatures give an indication of whether the magnitude of the heat source is sufficient. The k-ε turbulence model is then initialised with the various default coefficients and parameters for air being utilised.

The user defined scalar function is utilised to simulate how smoke would be transmitted through the model. It works in the same way as introducing an ink tracer into a water flow does. Its effects on the composition of the fluid can be defined as negligible, and hence for the purposes of this investigation are considered an appropriate method of monitoring exactly how much smoke is transmitted or back-drafted into the room the stove is located in. Although the production of smoke would alter the composition of the fluid flowing through the stoves body, it is possible for the sake of this study to simplify the model and consider its effects minimal due to the lack of experimental data, wide array of variables and the sheer unnecessary added complexity.

3.2 Definition of Operating Conditions

The gravitational acceleration is set to the standard -9.81m/s^2 in the vertical plane. The Boussinesq operating temperature is also defined as an operating condition and for the purposes of this study it is considered as being ambient, i.e. 22ºC. However if a specific surrounding air temperature for a specific stove were determined this parameter would be varied to accommodate. Finally the Variable-Density parameters are also defined, with a Specified Operating Density classified as 1.125 Kg/m^3.

3.3 Definition of Materials

As discussed before the effects of the addition of smoke to the model on the composition of the resulting mixture produced is considered negligible, and the nature of this study; to investigate the effects of varying the geometry on the fluid flow through the Nepali Insert Stove, it is essentially possible to only define two materials other than the user defined scalar (acting as a tracer). The properties of which are defined in *Table 1*.

Material type	Fluid	Material Name	Air
Density (Boussinesq) (kg/m^3)	1.225	Cp (cons.) (j/kg-k)	1006.43
Thermal Conductivity (cons.) (w/m-k)	0.0242	Viscosity (cons.)(kg/m-s)	1.789e-5
Thermal Expansion Coefficient (1/k)	0.0025	UDS Diffusivity (cons.) (kg/m-s)	1e-20
Material type	Solid	Material Name	Wall
Density (cons) (kg/m^3)	2719	Cp (cons.) (j/kg-k)	871
Thermal Conductivity (cons.) (w/m-k)	202.4		

Table 1

The various parameters are all now inputted and the model is ready to be solved.

3.4 The Solution Process

The model is initialised from the exhaust conditions, at the top of the chimney. This provides the initial guess values for the flow domain parameters to be computed from. The values of which are listed in *table 2*. Finally the iteration process then occurs obtaining the steady state conditions for the model.

Gauge Pressure (Pascal)	-21.582	Turbulence Kinetic Rate (m^2/s^3)	0.005
X Velocity (m/s)	0	Temperature (ºC)	200
Y Velocity (m/s)	0.05	User Defined Scalar( )	0
Turbulence Kinetic Energy (m^2/s^2)	0.005		

Table 2

3.5 Mesh Refinement

Reviewing the solutions produced prior to the iteration process occurring is very important. The definition of this mesh plays a large part in their accuracy and hence many need to be refined. Emphasis on adequate mesh refinement in areas of interest, i.e. where highly turbulent flows occur, must therefore be adopted.

After experimenting with various meshes for the Nepali Insert Stove model, it was found that a fairly uniform and fine mesh throughout the stove gave the most consistency in results. A reduction in the grading of the mesh up the length of chimney and out to the surrounding air space was also defined, to save on recourses. This is shown in *figure 4*.

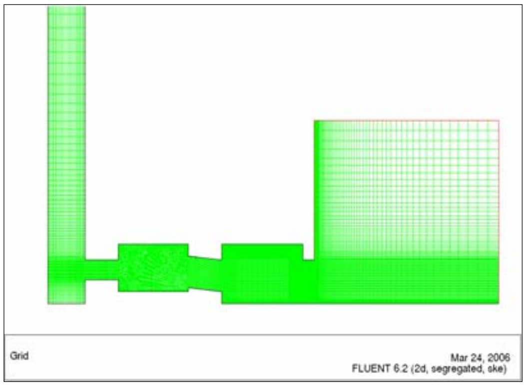


Figure 4 – Definition of Mesh

4.0 RESULTS AND ANALYIS

4.1 Effects of Varying the Baffle Angle

It can be seen from *figure 5* that when the baffle angle is varied between 0° and 50° the temperatures throughout the stove increase. This can be explained by the fact that increasing the angle of the baffle, beneath the second hot plate reduces the amount of cold air being sucked through the stove. In addition by closing the cross-sectional area of air being driven through the stove there is a reduction in the mass flow of the air, hence it has slightly more time to heat up as it passes through the two main pottery sections and out through the exhaust. Moreover as the baffle angle increases the effects of turbulence within the stove are increased and the combustion gases have more chance to diffuse their heat to the two hotplates and the rest of the stove body.

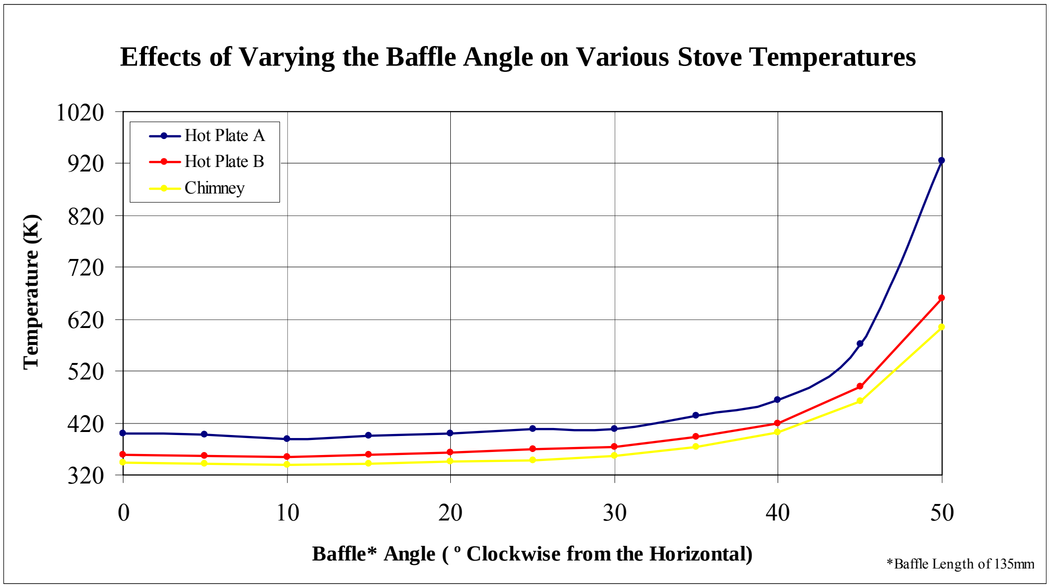


Figure 5 – Effects of Varying the Baffle Angle on the Stove Temperatures

It is interesting to note that after the baffle angle increases past 40° the gradient of the curve of temperature on hotplate A increases at a higher rate than the curves of hotplate B and the exhaust. This can be accounted for by the fact that there is effectively less air being able to pass out to the exhaust and the baffle is beginning to cause the stove to choke. This effect is illustrated in *figure 6*, where a plot of the amount of user defined scalars, an indication of how smoke is dispersed through the stove, concentration across both the front door and exhaust is logged.

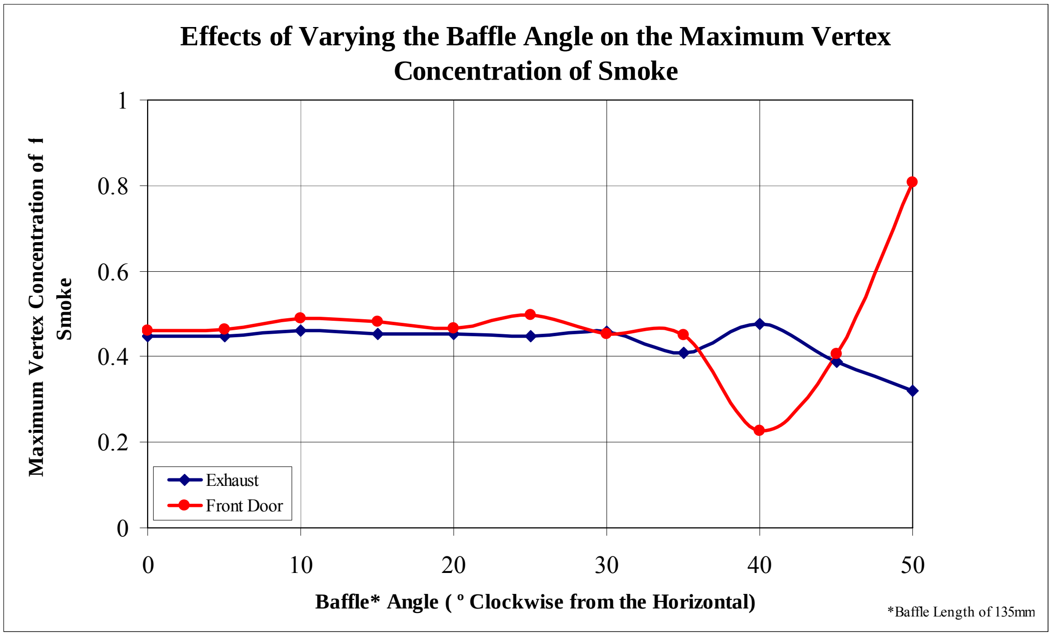


Figure 6 – Effects of Varying the Baffle Angle on the Maximum Vertex Concentration of Smoke

It can be observed that at a baffle angle of 40° there is a minimum amount of smoke at the front door, and a maximum amount of smoke exiting through the exhaust. In addition there is a rapid increase of smoke at the front door of the stove when the baffle angle increases past 40°, supporting the theory that the stove begins to choke after this optimum angle.

4.2 User Defined Scalar Contour Plots

Plots of the concentration of user defined scalars varying with a change in baffle angle are displayed in *figures* 7*, 8, and 9*. As already discussed, it can be clearly seen that at an angle of 40° the user defined scalar concentration is at a minimum.

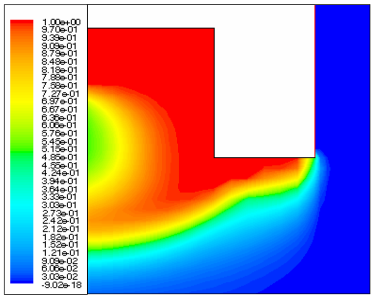


Figure 7 - Smoke Concentration with a 25° Baffle Angle

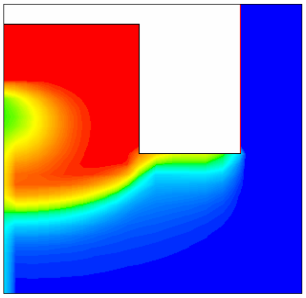


Figure 8 - Smoke Concentration with a 40° Baffle Angle

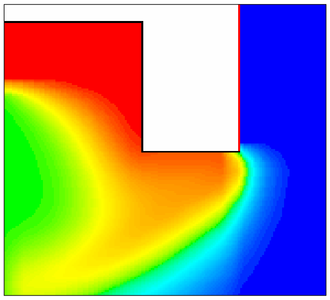


Figure 9 - Smoke Concentration with a 50° Baffle Angle

The variation in the concentration of user defined scalar or smoke, can be accounted for by the change in the magnitude of the recirculation region at the stoves front door. Controlling the size of this turbulence region is key to reducing the amount of indoor air pollution produced by the Nepali Insert Stove.

4.3 Verification of the Optimised Baffle Angle Temperatures

With an optimised baffle angle of 40º, it can be seen from *figure 10* that the various temperatures throughout the stove are approximately what would be expected for a stove utilised for cooking and space heating needs.

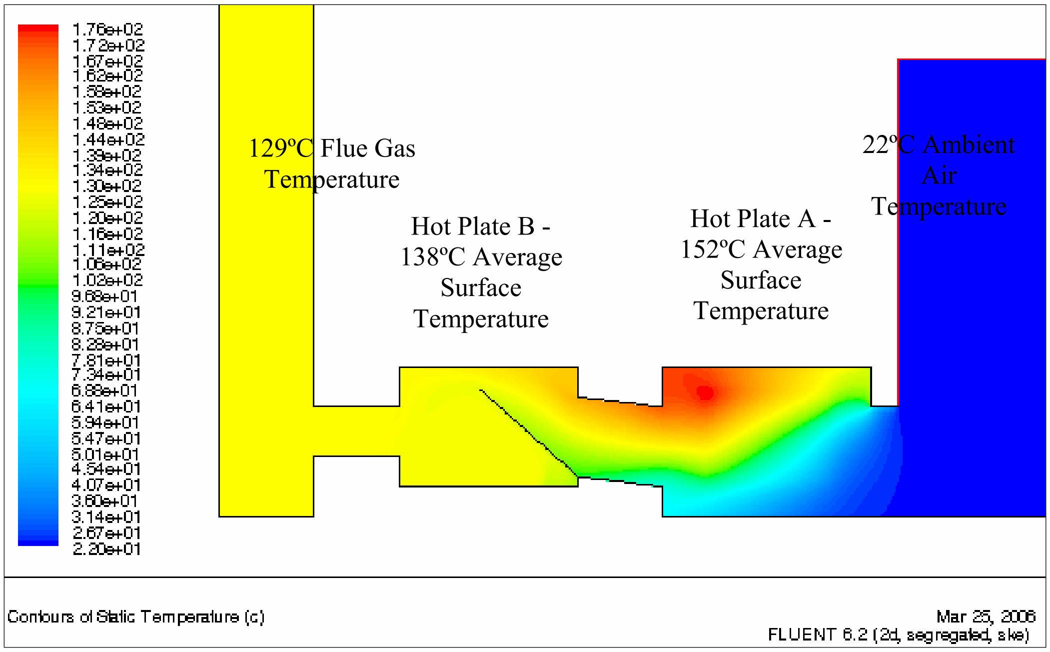
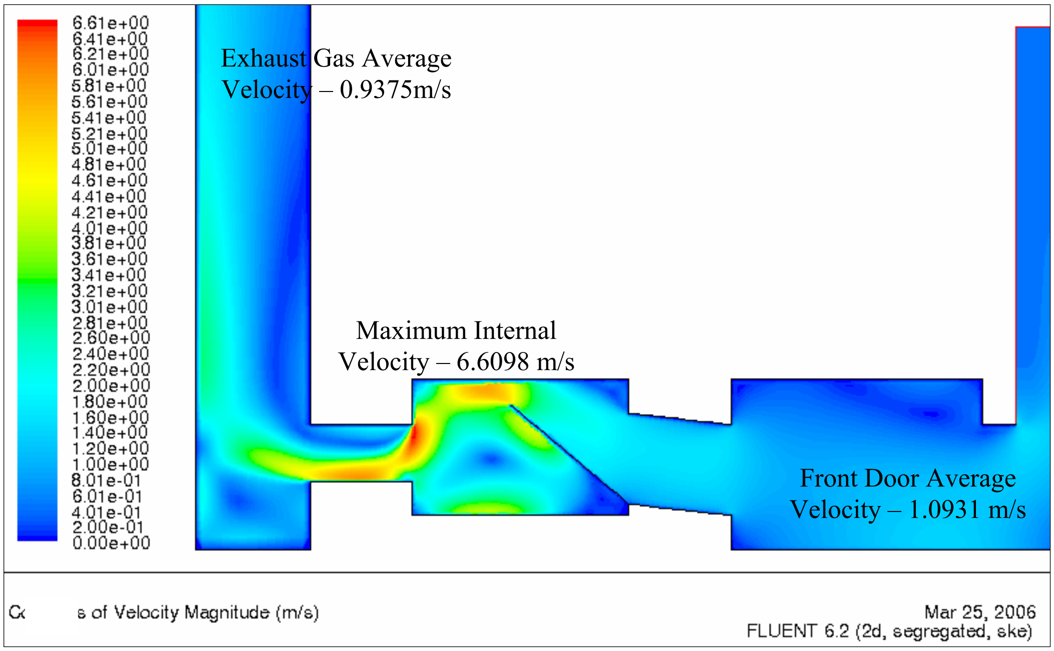


Figure 10 – Optimised Baffle Angle Temperatures

Both the hotplates A and B are required to be used for boiling and simmering water, and it is believed that with an average surface temperature of 152 ºC and 138 ºC respectively, it would be more than capable of doing so.

4.4 Verification of the Optimised Baffle Angle Velocities

The resulting velocity profile of the cookstove with 40º baffle angle can be seen in *figure 11*.



Exhaust Gas Average Velocity – 0.9375m/s

Maximum Internal Velocity – 6.6098 m/s

Front Door Average Velocity – 1.0931 m/s

Figure 11 – Optimised Baffle Angle Temperatures

It can be seen that sensible front door, internal and flue gas exit velocities are present. Without experimental data for the specific stove design it is hard to verify if the model is an exact representation of the stove. However it is considered a rational that reasonable approximation hence it is believed there is credibility in the method of investigation.

5.0 FUTURE DEVELOPMENTS

As stated any further CFD design developments with sight of implementation would need to be supported with accurate experimental data. Apparatus to monitor smoke emissions, inlet and exhaust velocities, hotplate temperature etc, would need to be utilised to allow for the simulation of specific stove designs to be conducted accurately.

This study utilises a two dimensional model to allow an ease of simulation. It is a good representation of the stove, however a three dimensional study would assist further in the understanding of how smoke is transmitted through a stove. The three dimensional model shown in *figure 12* was generated to develop such a simulation, however varying the geometry resulted was an extremely time and resource consuming process. Although there was not enough time to full develop the model and complete a full three dimensional study for the stove, a better awareness of the full potential of CFD was established.

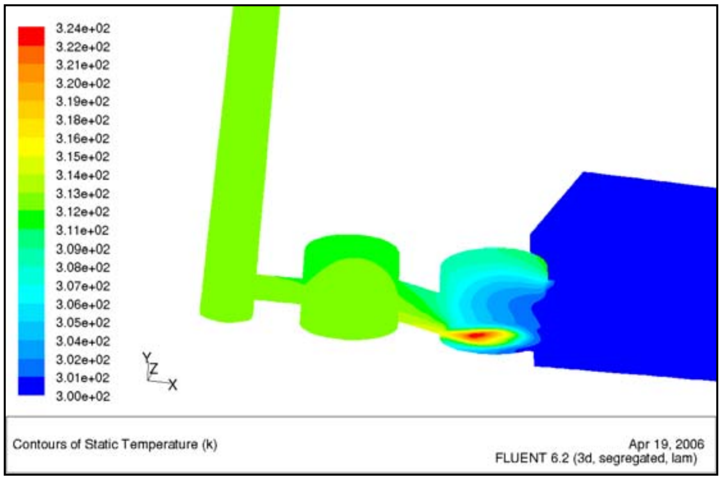


Figure 12 – Extended Three Dimensional Study

6.0 CONCLUSION

The lives of millions in the developing world are affected by the problem of Indoor Air Pollution. Efforts have been made to develop improved cookstoves to reduce the associated problems however developments have essentially been a process of trial and error, with stoves being manufactured from designs that are not necessarily optimised. This study assists in researching possible methods of design optimisation by utilising the concurrent technology of Computational Fluid Dynamics, which is commercially available in the developed world.

Focusing in on cookstove developments in Nepal and specifically on a CFD optimisation study of the Nepali Insert Stove it has been proven possible to define a method of optimisation to reduce the amount of smoke back drafted from the front door of the stove. It is however important to note that without accurate experimental data it is hard to prove the significance of the recommended optimum baffle angle of 40°. However it is believed that having adopted sensible assumptions and a logical rationale of analysis, the study has proved just how useful CFD can be in the development of smokeless stoves. In addition it is important to realise that there are many aspects to producing a feasible solution. An awareness of the fact that the implementation and the effective communication of a solution could in effect be the biggest obstacle to overcome has been established. It is hoped that the studies holistic approach to researching feasible methods of assisting in the development of cookstoves has increased its significance.

ACKNOWLEDGEMENTS

Composing a thesis for first time required the assistance of a number of people, all of who I wish to thank as without their help it would have been a great struggle. Firstly I would like to show my appreciation to Dr. Tom Scanlon, my supervisor for his technical assistance and guidance, specifically with the computational fluid dynamics simulation and analysis work. Furthermore I am grateful for his patience and enthusiasm.

Thanks have also to go to the various people at Engineering Without Borders, Practical Action UK and CRT Nepal for their general assistance throughout the study.

Last but not least I would like to thank my friends and family, and in particularly Mr. Julius Abensur and my parents, Jane and John Brewster, for their assistance and encouragement during my fourth year at the University of Strathclyde. Their much needed moral support and guidance proved of great aid.

REFERENCES

1. World Health Report 2002, World Health Organisation, 2002.
2. Bill Stewart and others, *Improved Wood, Waster and Charcoal Burning Stoves,* Intermediate Technology Publications, 1987.
3. Dr T. Scanlon, *Heat Transfer and Fluid Dynamics*, ME402 Course Notes, The University of Strathclyde.
4. Dr T. Scanlon, *CFD at Strathclyde Course Notes*, The University of Strathclyde.
5. Versteeg H.K. and Malalasekera W., *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*, Prentice Hall, 1995.
6. Dr. Mark Bryden, Dean Still, Peter Scott, Geoff Hoffa, Damon Ogle, Rob Bailis, Ken Goyer, *Design Principles for Wood Burning Cook Stoves*, Aprovacho Research Centre, 2003.
7. R. K. Aggarwal, S. S. Chandel, *Review of Improved Cookstove Programme in Western Himalayan State of India*, Elsevier, 2002.
8. Sjoerd Nienhuys, *Cooking Stove Improvements*, SREA, 2005.
9. G. Ballard-Tremeer and H. H. Jawurek, *Comparison of Five Rural, Wood-Burning Cooking Devices: Efficiencies and Emissions*, 1996.
10. Dana Charron, *The Ecostove – getting rid of nearly 90% of kitchen wood smoke*, Boiling Point Magazine, Issue 50, 2005.
11. W.K. Chow, R. Yin, *A new model on simulating smoke transport with computational fluid dynamics*, Hong Kong Polytechnic University, 2003.

12. Y. Jaluria, *Natural Convection, Heat and Mass Transfer*, Pergamon Press, 1980.
13. R. Peyret, T. Taylor, *Computational Methods of Fluid Flow*, Springer-Verlag, 1982.
14. C. T. Shaw, *Using Computational Fluid Dynamics*, Prentice Hall, 1992.
15. K. Hoffmann, S. T. Chiang, *Computational Fluid Dynamics for Engineers – Volume II*, Engineering Education System, 1995.
16. D. J. Acheson, *Elementary Fluid Dynamics*, Clarendon Press, 1990.