ISSN 2395-1621

Computational Modelling of Cook-Stove for Parametrical Changes

#1S SuriyaNarayanan, #2Gandigude A U

1suriya.swamy@gmail.com

2ashish.gandigude@zealeducation.com

#12Department of Mechanical Engineering, Zeal DCOER
Narhe, Pune-41, Maharashtra, India

ABSTRACT

This paper presents on the development of smokeless stove for application in worldwide. Many researchers have been followed for only some parametric changes on cook stove with defined particular optimized angle. But here the proposed design was found with major parametrical change which supports the various baffle angles and different baffle length to induce the smallest amount of smoke back drafted into the room with desired value of source terms where the stove would be located. This paper brings light of a feasible and concurrent method of cook stove development, 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 cook stoves in the developing world. Here the recommended optimum design has been obtained by the using the existing cook design which has been tested with proper experimental data was gathered.

Keywords— **Cook Stove, Baffle Angle, Baffle Length, Indoor air pollution, Computational Fluid Dynamics**.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

It was reported by the World Health Organization in March 2014 [1], 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 cook stoves 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. The method of ignition did not affect the efficiency of the stove and it showed significant influence on design of cook stove. In general this paper, exploring the effects of varying the cook stove geometry on the amount of smoke back drafted into the room was conducted.

II. COMPUTATIONAL MODELLING

This paper utilizes the use of ANSYS Design Modeller, Meshing & Fluent software for Computational Fluid Dynamic simulations and analysis. ANSYS Design Modeller used as a pre-processor to create the model with parametric approach which has been constructive for changing the baffle angle and length; ANSYS Meshing used to create the computational mesh with defining the name selection which usually taken as an boundary condition on the model in Fluent & ANSYS Fluent used to define the material properties and all boundary conditions for the problem (e.g. walls, velocity inlets, pressure outlet etc) & solves the CFD model generated after various initial parameters are set. ANSYS Fluent may also be used as a post-processor to visualize and analyse the fluid flow.

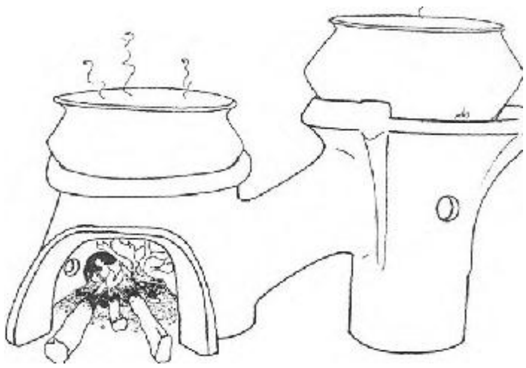

Fig. 1 Traditional two pot cook stove

During this effort, the CFD has been applied to simulate the above real cook stove model which has been shown above as Figure 1

III. Cook Stove Model

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

A. Design of Existing Cook Stove

The Figure 2 shown below displays the dimension of the cook stove which has been suggested with minimal constraints.

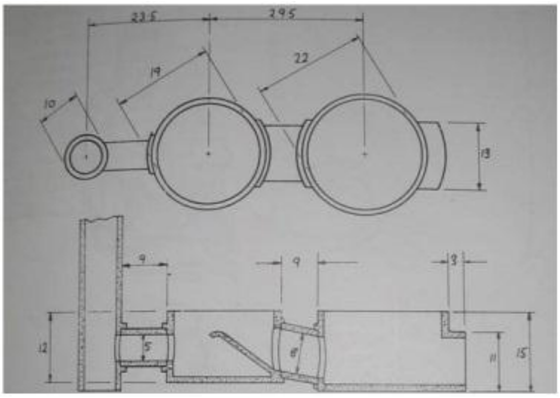

Fig. 2 Geometrical design of existing cook stove

B. Defining the Cook Stove Geometry

The process of defining a model with the use of Design Modeller is fairly elementary once 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 [3]. 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 [4]. Figure 3 displays the geometry defined.

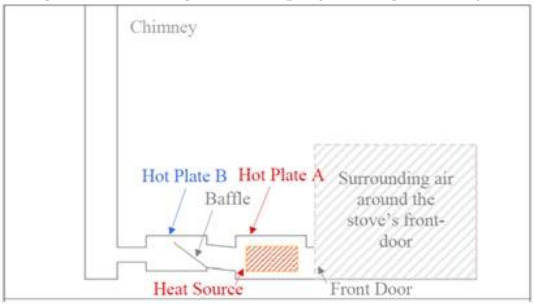

Fig. 3 Cross-Section of the Cook Stove

C. Name Selection with respect to Boundary Conditions

The figure illustrates the name selection which would involve as boundary conditions in the solver.

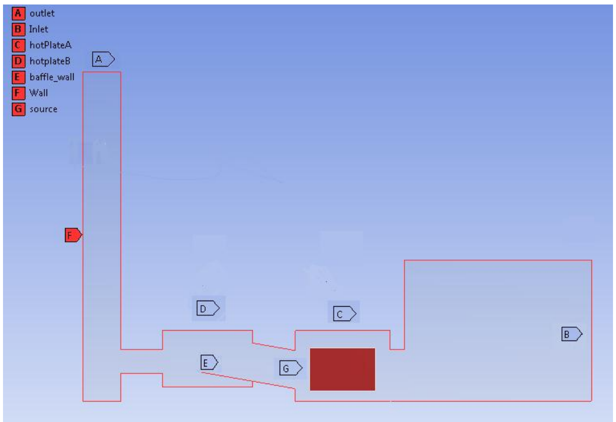

Fig. 4 Define Name Selection in the Geometry

D. General Hypothesis for the Model

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 affecting the transferral of smoke through the stoves 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 hypothesis 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.

IV. Computational Mesh

Certain mesh settings has been place to target the quad mesh only with respect to particular needs of changing for each variation of baffle angle and length for an every analysis, the CFD calculations are obtained on meshes where the solution attains grid independence [5].

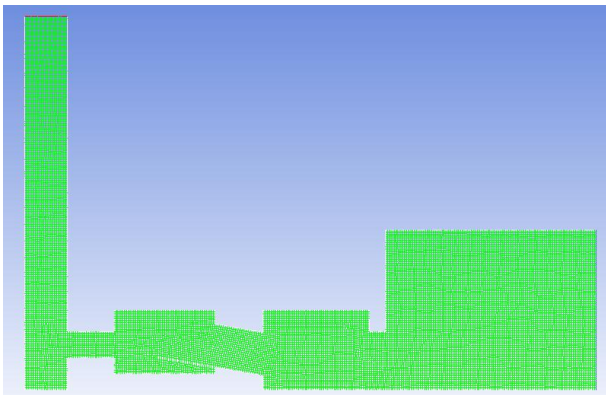

Fig. 5 Computational Model of Cook Stove

V. Initialising and Solving the CFD Model

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 discretization and careful monitoring of the solution.

A. General Setup

Pressure-Based enables the Navier-Stokes solution algorithm with Velocity Formulation specifies the velocity formulation to be used in the calculation. Thus will results in most of the flow domain having the smallest velocities in that frame, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution. Here the steady flow is being considered with 2D Space Planer options available only two-dimensional problems as the problem defined as 2D. Gravity enables the specification of gravity of $9.81 m/s^2$.

B. 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 modeling 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 utilize. 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. 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.

C. 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 equation set 1 gradient.

D. Materials

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 most of the properties has been from the referred report [2] as it was tested with experimental data.

TABLE I
MATERIAL PROPERTIES

Material: Air	
Density (kg/m3)	1.225
Cp (Specific Heat) (j/kg-k)	1006.43
Thermal Conductivity (w/m-k)	0.0242
Viscosity (kg/m-s)	1.7894e-04
Molecular Weight (kg/kgmol)	28.966
Material: Smoke	
Density (kg/m3)	1.225
Cp (Specific Heat) (j/kg-k)	1006.43
Thermal Conductivity (w/m-k)	0.0242
Viscosity (kg/m-s)	1.7894e-04
Molecular Weight (kg/kgmol)	28.966
Thermal Expansion Coefficient (1/k)	0.0025

E. Cell Zone conditions

As maintain the heat source as a separate surface entity in the geometry itself. Consequently the source terms an option has to define the heat and smoke associated to provide as input on the heat source, which would easily solve the problem in species transport model. Consider the remaining surfaces as also fluid.

F. Boundary Conditions

As the initial conditions, the fluid in the cook stove has been set with uniform ambient temperature (22^oC) and zero smoke concentration. 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 this simulation while getting a practical in- touch with the cook stove problem. Hence the inlet as velocity-inlet where it is known that the flow is directed in to the domain; hotplateA, hotplateB & baffle as wall - it allows the permeation of heat and additional variables into and out of the domain and outlet as pressure-outlet.

For pressure outlets, the specified static pressure is independent of frame. However, when there is backflow at a pressure outlet, the specified static pressure is used as the total pressure. For calculations using the absolute velocity formulation, the specified static pressure is used as the total pressure in the absolute frame; for the relative velocity formulation, the specified static pressure is assumed to be the total pressure in the relative frame. As for the flow direction, ANSYS Fluent assumes the absolute velocity to be normal to the pressure outlet for the absolute velocity formulation; for the relative velocity formulation, it is the relative velocity that is assumed to be normal to the pressure outlet.

VI. Solution

This problem has been solved by Pressure Velocity solution method achieved and to derive an additional condition for pressure by reformatting the continuity equation involves with simple scheme as it was considered as Steady-state calculations.

This problem solved Laplace's equation by using Hybrid Initialization which determines the velocity and pressure fields and other variables such as temperature, turbulence, species fractions, volume fractions, and so on.

G. Monitoring Solution Convergence

This problem is solved by monitoring the Vertex Maximum output for Mass fraction of smoke as around 0.4, since the proved existing cook stove has this value for all baffle angle variations. Hence the proposed design should maintain it for all changing parametric cases.

H. Run Calculations

Here the case as considered as steady-state calculations, the start of the solution process can be limited to 100 as the problem was simplified.

VII. Results and Analysis

This study investigated and analysed on the static temperature, velocity magnitude, and mass fraction of smoke and molar concentration of smoke for the model according to changing the various baffle angle and length by using the computational modelling.

I. Effects of Baffle length of 30 mm with varying the Baffle Angle 45^0

Baffle is a vital element for any this type of stove design. Here the role of the baffle on this experiment is forced the burned gases to move up toward the Hot Plate B, before turning right in the direction of the chimney. So that it defines the combustion volume better. The effect of baffle angle and length are adequate area of contact between the Hot Plate B and the hot gases in the cook stove.

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.

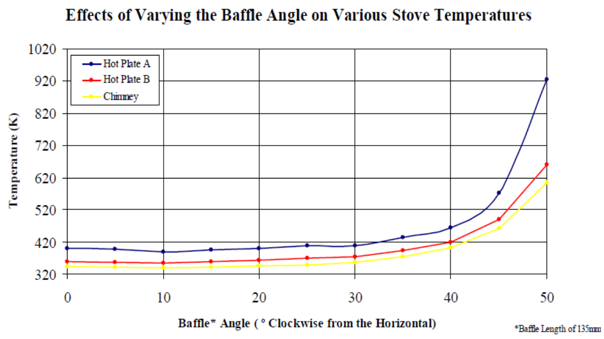

Fig. 5 Variation of Maximum Vertex Concentration of Smoke in Outlet Vs Baffle Angle with Length 300 mm

Overall the solution has been obtained with maintain the front door average velocity of 1 m/s^2 [6] for all variation of baffle angle changes.

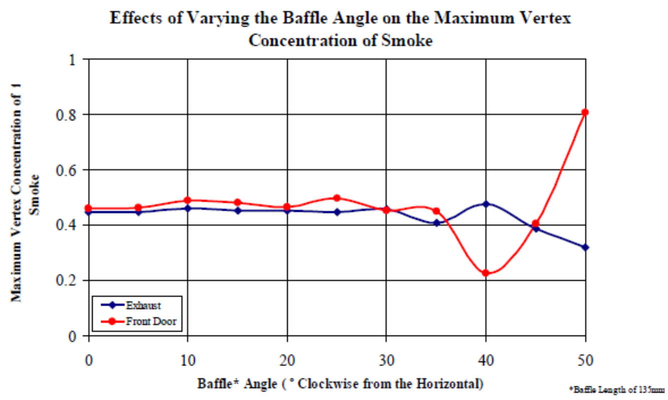

Fig. 7 Variation of Velocity Vs Baffle Angle with Length 300 mm.

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 7, 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.

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.

J. Verification of the Optimised Baffle Angle Temperatures

With an optimised baffle angle of 40º, that the various temperatures throughout the stove are approximately what would be expected for a stove utilised for cooking and space heating needs. 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 Insert Stove. 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.

VIII. Conclusions

The aim of this study not to modify the existing cook stove system i.e. Here we are trying get optimized design for cook stove with n-numbers of preferred baffle angle. Because of this aspect the model/parameter defined would be recommended for new installation of smoke free room within the desired amount of source terms. Parameters involved in this project are practically tested with modified of exiting systems. Since the baffle is installed by layman so called users don’t have anticipate engineering knowledge and with effect of lacking of knowledge, the accuracy of baffle angle won’t be consistent in all the time during the installation. Therefore we are not trying to suggest any preferred particular baffle angles to the final design of the system. However, as this was already suggested by earlier with few researchers. With all above specified constraints, this project try to obtain proposed cook stove design with smoke free at room with various angle with minimal constraints of desired source terms. This experiment plots the different baffle angles and lengths with desired amount of heat source and smoke to obtain the maximum temperature (^{o}C) at Hotplate A & Hotplate B. Incidentally, we are always considering the front door velocity in and around ~1.0 m/s, since this value is ambient air flow velocity for ordinary room.

Acknowledgment

The Authors would like to thank the HOD of Mechanical Department of Zeal DCOER, Narhe, Pune-41, Maharashtra, for giving an opportunity to publish this work. First Author would also like to thank Gandigude A U for rigorous review, continuous support and encouragement.

References

[1] "World Health Organization," [Online]. Available: http://www.who.int/mediacentre/factsheets/fs292/en/. [Accessed 27 December 2014].

[2] J. J. Brewster, "CFD DESIGN OF IMPROVED COOKSTOVES," http://www.ewb-uk.org/, 2005-2006.

[3] I. Y. Maevski, Design Fires in Road Tunnels, NATIONAL COOPERATIVE HIGHWAY RESEARCH PROGRAM, 2011.

[4] D. Ogle, "ROCKET STOVES – CONTROLLING DRAFT - Variations in chimney height and air flow openings in a modified “rocket stove"," August 17, 2002.

[5] A. T. M. Y. Robin Elder, Advances of CFD in Fluid Machinery Design, Professional Engineering Publishing Limited, 2003.

[6] John Kaiser Calautit & Ben Richard Hughes, "Measurement and prediction of the indoor airflow in a room ventilated with a commercial wind tower," ScienceDirect,, 15 August 2014.

[7] T. K. Fannelop: Fluid Mechanics for Industrial Safety and Environmental Protection, Industrial Safety Series Vol. 3, Elsevier, 1994

[8] Lee, CK, Chaiken, RF, and Singer, JM, Interaction between Duct Fires and Ventilation: An Experimental Study Combustion Science and Technology vol. 20, Page No .59-72

[9] McGrattan, KB, Forney, GP, Prasad, KFloyd, , JE, and Hostikka, S, Fire Dynamics Simulator (Version3)-Users Guide, U.S. Dept. of Commerce, National Institute of Standards and Technology. 2002.

[10] Baldwin, S. F, Biomass Stoves: Engineering Design Development and Dissemination, Vita Publications, Arlington, VA, 1988.

