# Use of CFD simulation as a design tool for biomass stoves

***M.R. Ravi, Sangeeta Kohli and Anjan Ray***
*Department of Mechanical Engineering, Indian Institute of Technology, Hauz Khas*
*New Delhi 110 016, India*

***Design of biomass stoves has relied mostly on empirical information and trial-and-error experiments backed up by simple thermodynamic and heat transfer calculations. Quite frequently, the details of fluid flow inside the biomass stove, which is caused by buoyancy due to the high temperature prevailing in the combustion region, have been ignored and correlations of heat transfer based on forced flow have been used in the heat transfer analysis of stoves. This paper presents an approach in which detailed CFD simulations of the flow, heat transfer, pyrolysis and combustion in the configuration of a simple sawdust stove are used to evolve simple algebraic equations that describe individual phenomena. Such equations are needed in the field for performance analysis and prediction, and could also be used for the optimization of stove geometry for performance. The paper describes the development of the building-blocks of the detailed simulation model and its use in the derivation of simple model equations relating design and performance parameters.***

## 1. Introduction

Design of biomass stoves for high thermal efficiency and low emissions of toxic gases and particulate matter requires detailed understanding of various fundamental physical phenomena, namely, heat transfer, fluid flow, pyrolysis and combustion, and the complex interaction among them. All these phenomena are so intimately coupled with one another that a realistic prediction of performance of a biomass stove needs detailed modelling of each of the above phenomena and their coupling. Design of a biomass stove basically involves the choice of the various geometric dimensions of the stove for achieving a desired range of operating conditions of the stove with the best possible thermal efficiency and the minimum possible harmful emissions. The complete statement of the design problem needs the mathematical expression of the functional relationship between the geometric parameters and the performance parameters such as the thermal efficiency, power and emission factors. These functions need to be obtained for different stove configurations for a given fuel. It should also be noted that these functions would be different for different fuels used in the same stove configuration.

Thus, the design of stoves requires extensive experimentation under controlled conditions to obtain the functional relationships between the design parameters and operating parameters of the stove. For obtaining reliable data, these experiments are often difficult and require sophisticated instrumentation and careful control over the experimental conditions. Besides, experimentation by varying geometric design parameters involves the construction of several different stoves, and this can be a time- and resource-consuming exercise. Instead, if a computational model of the stove can be built, so that the necessary experimentation can instead be carried out on a computer, then the optimization problem can be formulated and solved in considerably less time and at considerably less cost. The theme of the work of the team at the Indian Institute of Technology (IIT), Delhi, is the development of such a computational model which uses detailed simulation using computational fluid dynamics (CFD) to predict the performance of the actual stove, and obtain the model equations that relate design and performance parameters. These equations would help in the analysis and prediction of performance of the stove under various operating conditions, and would also be useful in carrying out design optimization of a given type of biomass stove for a given fuel.

The development of the computational model involves the development and validation of sub-models of buoyancy-induced fluid flow, heat transfer, pyrolysis and combustion. The equations governing these phenomena are the same, irrespective of the geometric shape and construction of the stove: the basic equations of conservation of mass, momentum and energy, and the equations describing the rates of various chemical reactions. The differences would lie in the shapes of the domain of computation and the corresponding boundary conditions for the various equations. With this in mind, the model development was begun for a sawdust stove. Figure 1a shows the cross-section of a sawdust stove. In order to keep the geometry axisymmetric, the horizontal segment of the central passage has been removed for the purpose of model development, as shown in the cross-section in Figure 1b.

Kohli [1992] first proposed the theme of development of a simple model correlating the design parameters with performance parameters using CFD simulation. Kohli [1992] and Kohli et al. [1993] developed the sub-model

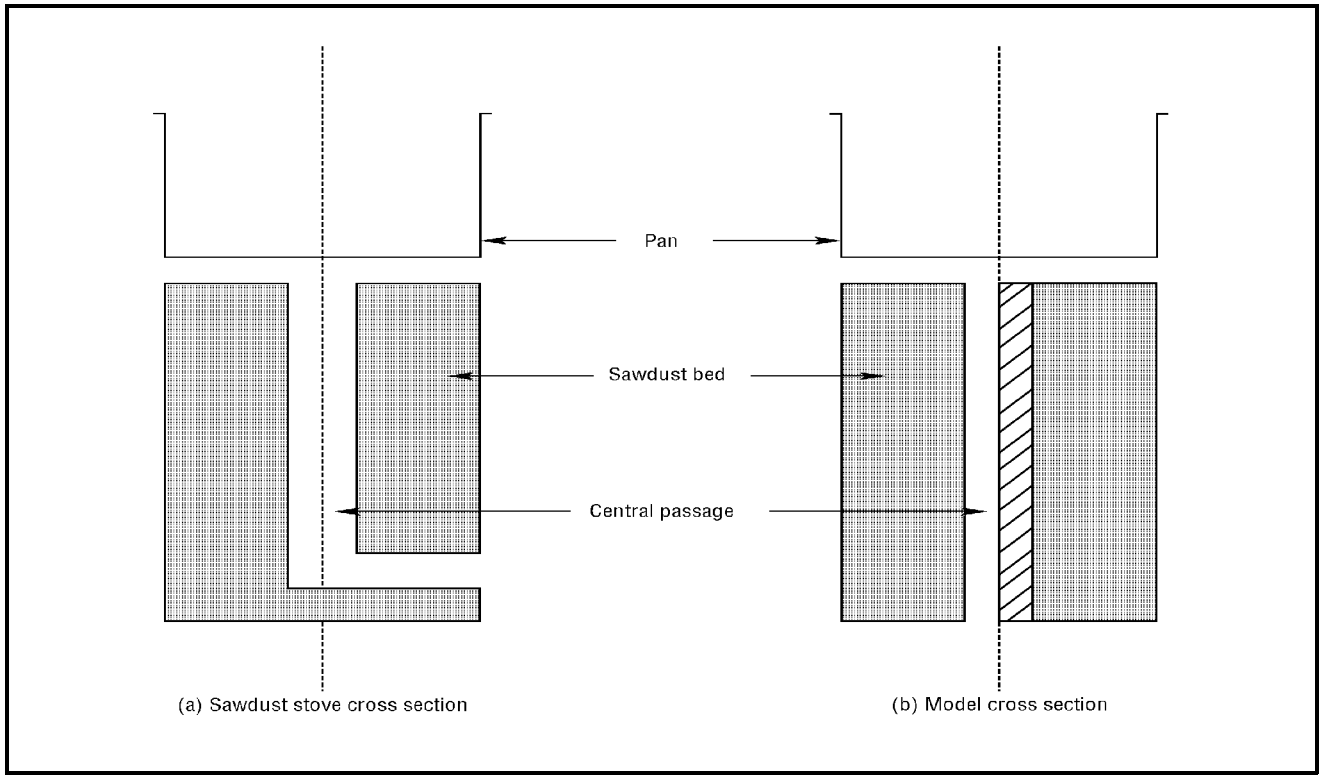


Figure 1. Schematic of a typical sawdust stove: (a) stove cross-section (b) model cross-section

for buoyancy-induced flow in the configuration shown in Figure 1b. An experimental rig for measuring the mass flow rate through the configuration was designed, fabricated and calibrated to validate the predictions of the sub-model [Kohli et al., 2002]. Jhalani and Sinha [1999] and Ravi et al. [2002] developed the sub-model for pyrolysis, and validated it against their own experiments. Agrawal and Saini [2000] made improvements in the experimental set-up of Jhalani and Sinha [1999] and developed a more realistic model for pyrolysis, accounting for the flow of volatiles in the sawdust bed. A combustion sub-model has been developed by Ray et al. [2002] and Jain and Upadhyay [2001] included the effect of radiative heat transfer on the temperature field in the combustion region. Gupta and Abhishek [2001] developed a simple thermodynamic model for the sawdust stove without the pan, using the information available from the detailed simulations cited above. Patra [2001] improved this model and included the pan in the geometry of the stove.

This paper presents an overview of the work [Kohli et al., 2002; Jhalani and Sinha, 1999; Ravi et al., 2002; Agarwal and Saini, 2000; Ray et al., 2002; Jain and Upadhyay, 2001; Gupta and Abhishek, 2001; Patra, 2001] carried out by the group at IIT, Delhi. The following sections describe the development and validation of the various sub-models for buoyancy-induced fluid flow and heat transfer, pyrolysis and combustion. During the course of model development, an attempt has been made to isolate one phenomenon at a time, and experiments have been designed to provide necessary inputs for each sub-model and to validate the sub-model. Subsequently, the coupling of these models to predict the model-stove performance is described. Then, the theme of development of a simple algebraic model using these simulations is described, paving the way for design optimization. The paper concludes with the directions for diversifying the application of this approach to other stove configurations, and the promise of the approach in improving stove design.

## 2. Modelling natural convection flow and heat transfer

Flow in biomass stoves is entirely due to natural convection, with rare exceptions of stoves using a mechanical draught of air. Kohli [1992] points out that it has been an inherent shortcoming of most stove models in the literature that they tend to use correlations which are strictly valid only for fully developed forced flows to estimate the mass flow rate of air through the stove. By accounting for natural convection, it was shown that the predictions of heat transfer efficiency of a stove were in better agreement with the experimentally measured values. To simulate buoyancy-induced flow in a stove, one needs to solve the Navier-Stokes equations, which are the equations of conservation of momentum, simultaneously with the equations of conservation of mass and energy, with the appropriate boundary conditions. Such a simulation was carried out by Kohli [1992]. These equations were solved in the axisymmetric (r-z) domain shown hatched in Figure 1b using the finite difference method [Kohli et al., 1993].

To validate the model for natural convection flow and heat transfer in the absence of combustion, an electrical analogue of the sawdust stove model geometry shown in Figure 1b was considered. The sawdust bed was replaced by an electrically heated ceramic heater, and this heater served as the source of heat for the central tubular passage. The configuration shown in Figure 1b consists of a

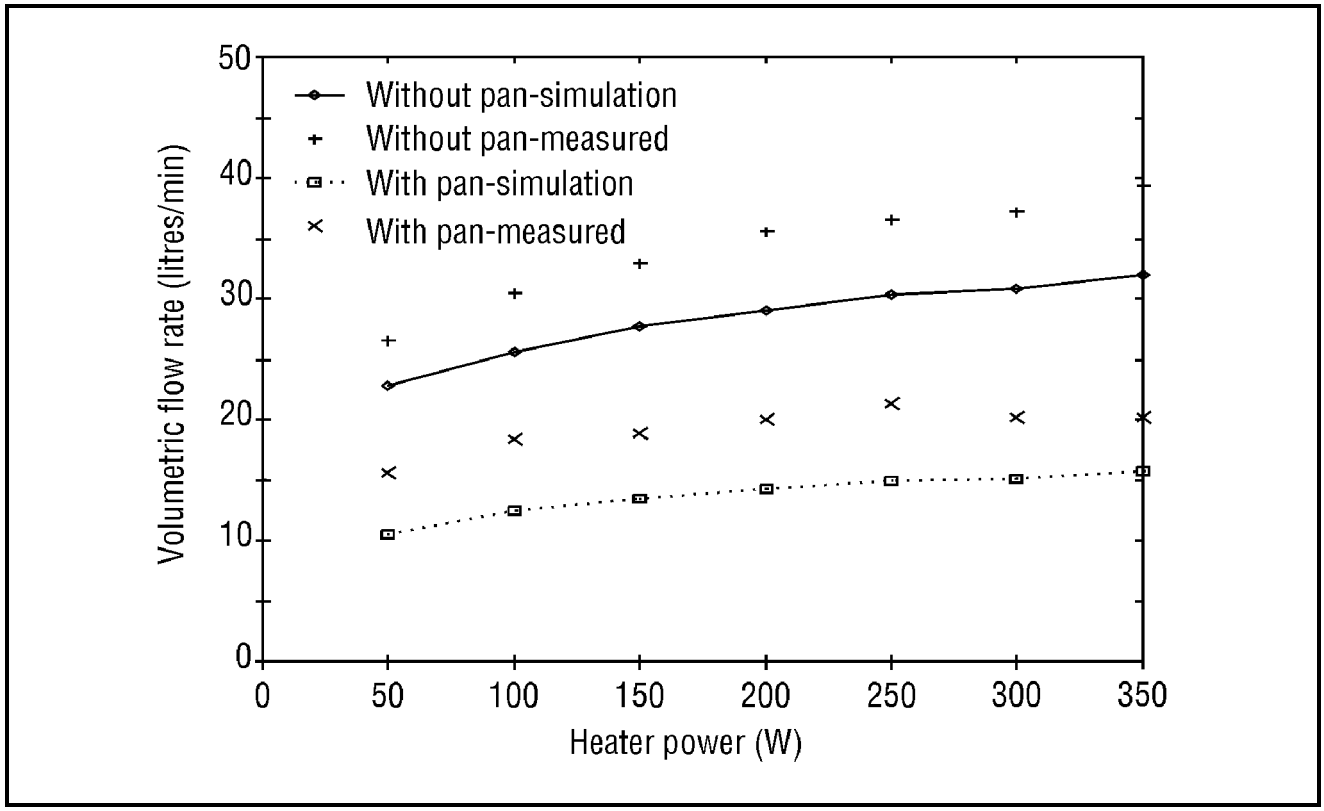


Figure 2. Comparison of predicted mass flow rates with measured values without pan, and with pan-to-stove spacing of 4mm

tubular region followed by the space between the pan and the support plate. The experimental rig and methodology used for the measurement of mass flow rate of air induced by buoyancy through the electrically heated stove analogue have been reported in Kohli et al. [2002]. Figure 2 presents a comparison of the measured mass flow rates with the predicted values. It compares the predictions with measurements for the electrical analogue of the stove without the pan, for different power inputs, and for the case with the pan at a pan-to-stove spacing of 4 mm. The predictions show the same trend as the measured values, although they are consistently lower than the measured data. The reasons for the discrepancies are currently being analysed.

The CFD model of the electrical analogue, where heat was supplied through the wall of the tubular region, helped in understanding the dependence of flow induced due to buoyancy effects in such a configuration on the various geometric parameters. The flow field, mass flow rate of air through the stove and heat transfer, and hence the heat transfer efficiency of the stove were found to be most sensitive to the spacing between the support plate and the pan. The understanding gained through detailed flow and temperature fields was helpful in isolating the factors affecting the heat transfer and induced mass flow rate. The flow field in the region between the pan and the stove had the distinct feature of a recirculation zone where the flow turned from the vertical direction in the combustion chamber to a horizontal flow. The size of this recirculation zone was found to be a strong function of the spacing between the stove and the pan. The presence of this zone, in turn, affected the heat transfer to the pan. Moreover, at very small spacings, the frictional pressure drop was the main cause of the resistance to the buoyancy-induced flow. Thus, this model provided a good insight into the features of the buoyancy-induced flow through such a configuration.

The CFD model of natural convection flow in an actual stove was developed by introducing uniform heat generation in the bulk of the gas present in the combustion region in the central tube of the stove [Kohli, 1992]. This was to simulate heat release due to combustion without involving the complexities associated with modelling of combustion. The detailed analysis in this case threw light on the balance of terms in the momentum equations. Contrary to the traditional belief in the stove literature, it was found that in such a configuration, the buoyancy force is balanced by the flow acceleration terms rather than the frictional forces. In a later section, we shall discuss the combustion model which superseded the bulk heat generation, to make the predictions even more realistic.

### 3. Modelling pyrolysis

Pyrolysis of wood is a phenomenon in which the complex molecules that constitute wood, viz., cellulose (50 %), hemi-cellulose (25 %) and lignin (25 %), are thermally degraded into simpler molecules, thereby producing volatile combustible gases, tar and char. Modelling of pyrolysis therefore involves the modelling of heat transfer from the hot flame zone into the bulk of the solid fuel and chemical kinetics of the thermal decomposition reactions. Sinha et al. [2000] present a detailed review of the literature available on modelling of pyrolysis of wood and the complexities involved therein.

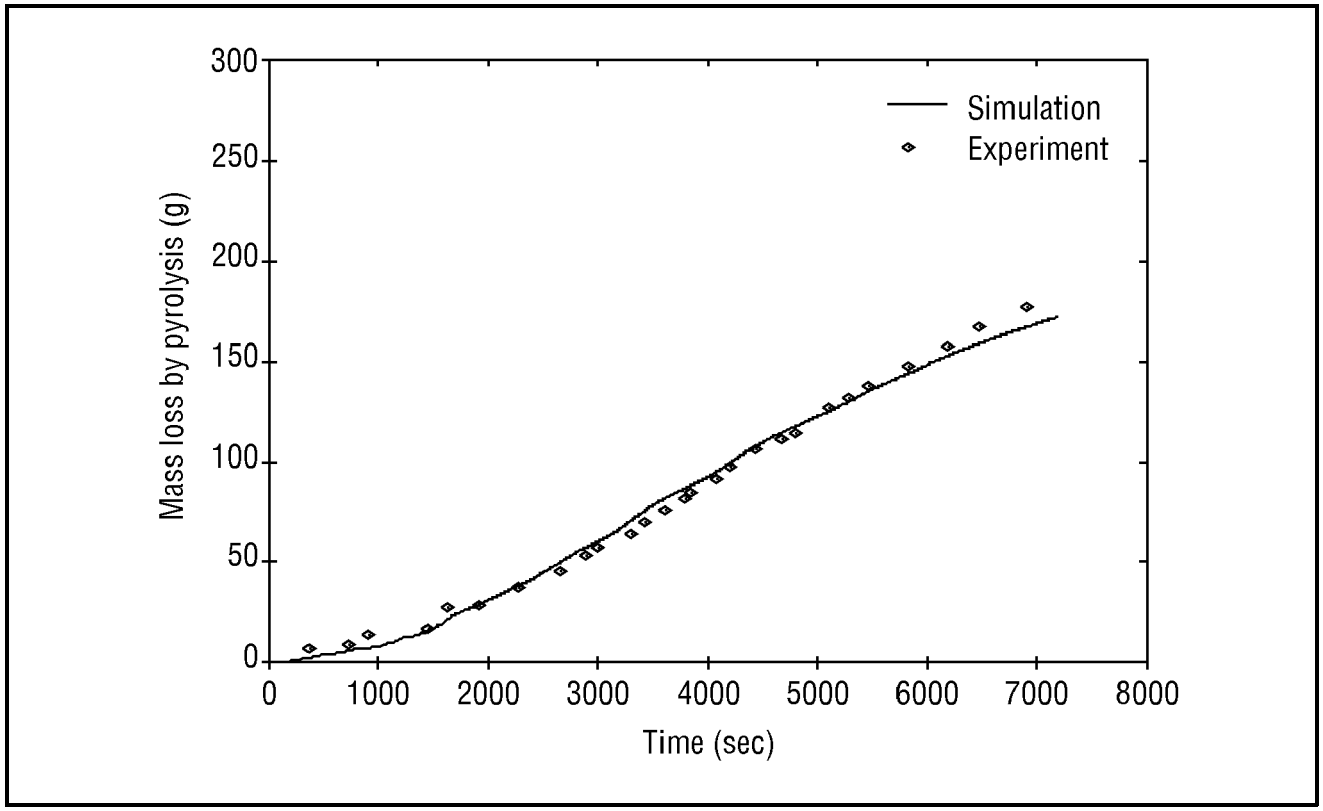


Figure 3. Comparison of predicted and measured mass loss histories [Ravi et al., 2002]

Jhalani and Sinha [1999] and Ravi et al. [2002] developed a model for pyrolysis in the annular sawdust bed in the geometry of the sawdust stove shown in Figure 1b. They assumed heat transfer to be purely due to conduction, and represented the pyrolysis reaction by an equivalent single-step first order reaction. They obtained the thermal conductivity of the sawdust bed as well as the reaction rate constant for the equivalent first order reaction by experiment. They also designed an experiment to pyrolyse an annular bed of sawdust using a cylindrical electric heater embedded in the central region of the sawdust, to simulate the heat input from the combustion zone. They measured temperatures in the sawdust bed and mass loss from the bed, to corroborate heat transfer model predictions and pyrolysis predictions respectively. Figure 3 presents the comparison of the predicted and measured mass loss histories [Ravi et al, 2002]. The agreement obtained was fairly good.

The experimental set-up presented in Jhalani and Sinha [1999] and Ravi et al. [2002] allowed no volatiles generated by pyrolysis to escape through the central region, which is quite unlike the actual sawdust stove. Agrawal and Saini [2000] redesigned the experiment with a porous stainless steel cylinder surrounding the electric heater, through which the volatiles could escape into the central cylindrical region of the bed. In order to be able to predict the mass flow of volatiles arriving in the central region, which is a necessary input for coupling the pyrolysis model with a combustion model, they introduced the capability of simulating the flow of volatiles through the porous sawdust bed. Their experimental results showed a more smooth variation of mass loss with time compared with the results of Jhalani and Sinha [1999]. The mass loss rates predicted by their model were in good agreement with the experimental measurements, but there was some time lag between the cumulative mass loss predicted by the model and that measured. The latter was attributed to the observed fact that the pyrolysis gases start escaping from the sawdust bed only after a small time lag from the time the heating starts, while in the model this aspect has not been taken into account.

**4. Modelling combustion**

Combustion of wood or sawdust can take place in two modes: flaming combustion and glowing combustion. During the flaming combustion phase, a flame, i.e., a high temperature homogeneous reaction zone, is established in the gas phase. Glowing combustion is the heterogeneous oxidation reaction of char that manifests itself in the luminosity of the burning char. Flaming combustion is the primary mode of combustion in a sawdust stove where the power output is high. The glowing phase gives a much lower power output, though it can continue for many hours in a typical sawdust stove and hence is generally used for water-heating or cooking activity requiring very slow heating. For most cooking activity, the flaming combustion phase is used.

In the present work, only the flaming combustion phenomenon was modelled. Since it is known from experimental observations that a sawdust stove operates quite steadily at constant power after about 10 minutes of initial transients, it was decided to model the combustion of volatiles in the central tube of the configuration in Figure 1b as a steady state phenomenon. In the combustion

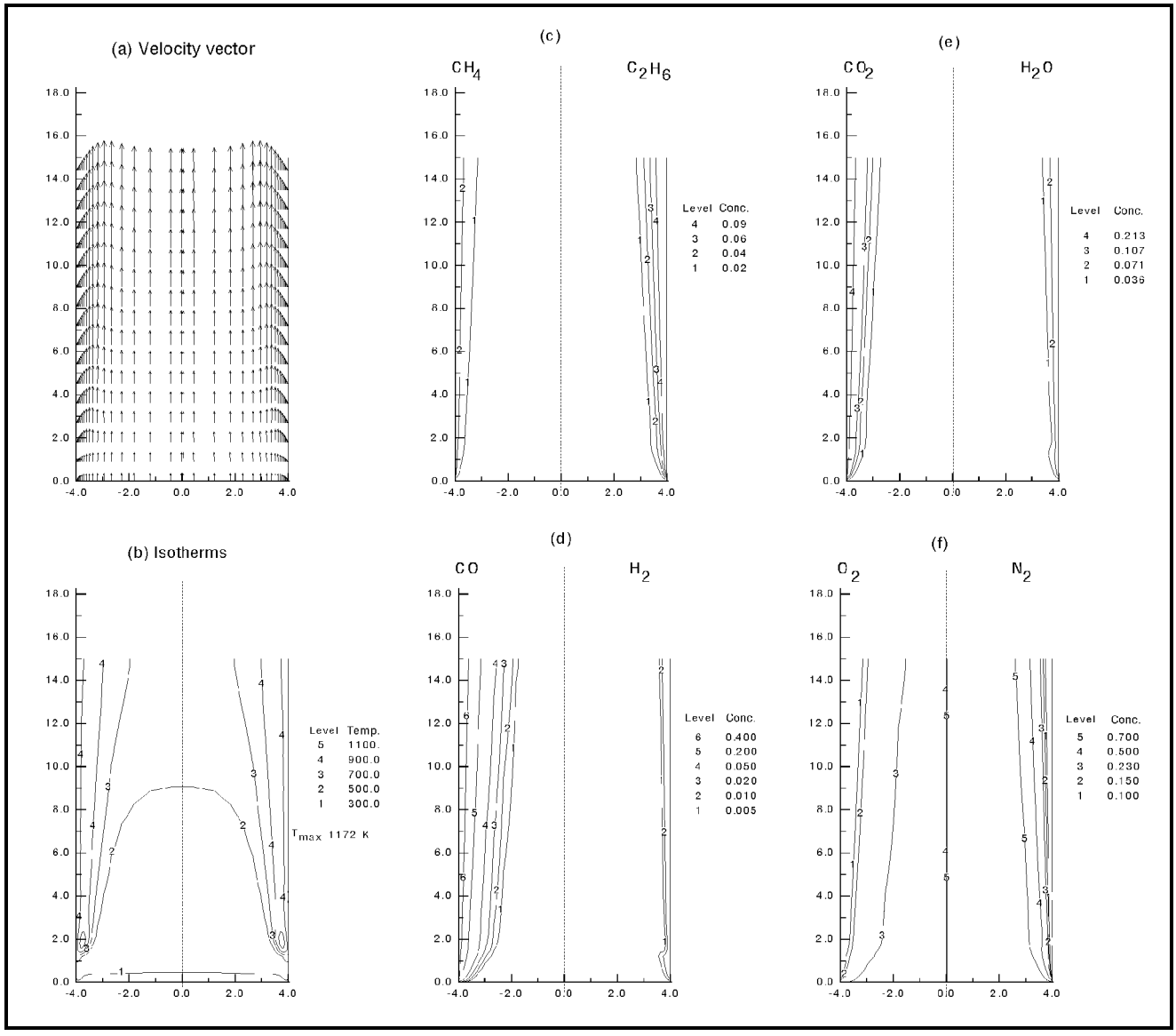


Figure 4. Flow field predictions using the finite rate combustion model with radiation heat transfer [Ray et al., 2002; Jain and Upadhyay, 2001]

process, fuel issuing out of the sawdust bed into the central tubular region reacts with the oxidizer drawn into the stove by virtue of the buoyancy generated by heat release due to combustion. While developing the combustion model, the fuel flow rate from the sawdust bed into the central passage was taken to be known and was determined by the power of the stove and the calorific value of the volatile matter released by pyrolysis of the sawdust bed.

For modelling the chemical conversion process, two approaches were used. First, the chemical reaction was assumed to take place at an infinite rate. This results in an infinitesimally thin reaction zone where the fuel and oxidizer get completely depleted. The results of this model indicate that the assumption of infinite reaction rate leads to unrealistically high temperatures in the combustion domain. Subsequently, the chemical reaction model with a finite, single-step, irreversible chemical reaction was incorporated. This model resulted in more realistic prediction of combustion temperatures [Ray et al., 2002] and realistic species concentration profiles. Jain and Upadhyay [2001] included a model for radiative heat transfer, which accounts for the radiative exchange of energy between the hot gases in the combustion zone and the solid boundaries of this region. This made the temperature fields even better, by lowering the peak temperature and raising the temperatures predicted in the central region of the tubular combustion zone to more realistic values, as seen in Figure 4. Results of the combustion model also indicated that the flame is located close to the wall for small fuel inflow rates and moves towards the centre as the fuel flow rate is increased [Ray et al., 2002]. A rigorous experimental validation of the combustion model is, however, yet to be carried out.

## 5. Interfacing the various models

The combustion model has been implemented on the flow and heat transfer simulation program, and therefore, these two models are being solved for simultaneously. The pyrolysis model is a transient one, while the flow, heat transfer and combustion model is a steady state model. However, the results of the pyrolysis model show a steady

rate of volatile matter generation after a few minutes of initial transients (Figure 3). Since a transient combustion simulation is computationally very demanding, it was decided to interface the combustion model with the pyrolysis model during the period of steady operation of the stove. The combustion model requires as input the rate at which volatile gases enter the combustion zone from the sawdust bed. Once this input is available, it can predict the temperature profile along the inner surface of the sawdust bed. The pyrolysis model, on the other hand, requires the inner surface temperature profile of the sawdust bed as input, and predicts the mass flow rate of volatiles corresponding to this input, as a function of time. Strictly speaking, these models need simultaneous solution in the transient mode. As a simplification, it was decided to interface them at the level of inputs.

The iterative procedure is as follows. Make a guess of the power of the stove, and calculate the corresponding mass flow rate of volatiles, from the knowledge of the heating value of the volatiles. With this guess as input, run the combustion simulation to obtain the corresponding temperature profile on the inner surface of the sawdust bed. Now, with this temperature profile as input, run the transient pyrolysis simulation to obtain the mass flow rate of volatiles as a function of time. This predicts the mass flow rate of volatiles in the phase of pyrolysis when this rate is steady. This could be different from the guessed mass flow rate. On the basis of the trends, make a new guess, which is likely to be closer to the correct solution, and repeat the iteration. This iteration needs to be repeated a few times before the guessed volatile flow rate matches the predicted rate. At this point, the power of the stove corresponds to the actual power that can be produced by the stove with the chosen geometry.

## 6. Development of simplified model equations

As mentioned before, the purpose of the detailed simulations is to finally assist in the development of a simple model which can be used to predict the performance of a given stove. This model could also be used to carry out a design optimization that gives the optimal dimensions of a given stove configuration for best efficiency and/or minimum emissions. A simple model has been developed for a sawdust stove of configuration shown in Figure 1b [Gupta and Abhishek, 2001; Patra, 2001]. Gupta and Abhishek [2001] considered the sawdust stove with no pan on it, while Patra [2001] included the pan in the model. For modelling with the pan, the sawdust stove was divided into three main control volumes: the tube region, the stagnation region and the horizontal space between the stove and the pan. Mass, momentum and energy equations were written for each control volume. These equations are only algebraic in nature for finite control volumes, unlike the differential equations used in the detailed models presented in the earlier sections. Certain parameters used in developing the simplified equations were to be provided as inputs to the model in addition to the geometrical dimensions of the stove. These inputs need to be obtained from the results of detailed simulation of the various phenomena.

The CFD model for fluid flow and heat transfer indicated that the heat transfer and pressure drop in the tubular region were primarily dependent on the temperature and velocity profiles in a thin annular region near the inner surface of the sawdust bed, as seen from Figure 4. Similarly, the results presented by Kohli [1992] show that the heat transfer in the stove-pan gap is better predicted when the presence of the recirculation zone is taken into account. From the results of the detailed model of pyrolysis, it was observed that the volume of sawdust getting pyrolysed was nearly constant at any given time during the phase when the mass loss rate was steady. The fraction of volatiles that burn in each zone of the combustion region and the average temperature of each zone is also obtainable only from the results of detailed simulation of combustion. Some of the parameters that represent the above information have been obtained from the detailed simulation results and incorporated in the simple model, while some others are yet to be included.

For the simple model, the radius and height of the combustion chamber, the diffuser spacing and the pan diameter were supplied as inputs. Then, the equations were solved by first guessing a mass flow rate, and then iteratively converging to the correct value of the mass flow rate for a given stove geometry. The converged final results included the mass flow rate of air and the average temperatures in the three control volumes. With this model, it was then possible to change the dimensions of the stove and arrive at not only mass flow rates and temperatures, but also quantities such as thermal efficiency and heat transfer to the pan. A lattice search method was used with the data generated by this model to determine the optimum stove geometry in a limited range of variation of geometrical dimensions of the stove. The results obtained so far are encouraging and in conformity with expected trends. Figure 5a shows the predictions of power output of the stove without pan [Gupta and Abhishek, 2001] compared with experimental data of Mukunda et al. [1993]. The agreement is seen to be quite good. Figure 5b presents the predicted thermal efficiency [Patra, 2001] as a function of pan-to-stove spacing, for two heights of combustion region, viz., 20 cm and 30 cm. Although these results are yet to be validated against experiments, the trends shown and the values of efficiency are realistic. Further work is now being carried on to include more sophistications in the model, e.g., more accurate determination of heat transfer coefficients and accounting for the conical shape of the support plate for the vessel.

## 7. Current status and future directions

The detailed simulation models for flow, heat transfer, pyrolysis and combustion as described in the earlier sections have been developed and all models other than combustion have been validated against experiments. The results of the detailed simulation are useful in developing a comprehensive understanding of the various phenomena and their coupling, and their effect on the performance of the sawdust stove. The insight and information obtained from

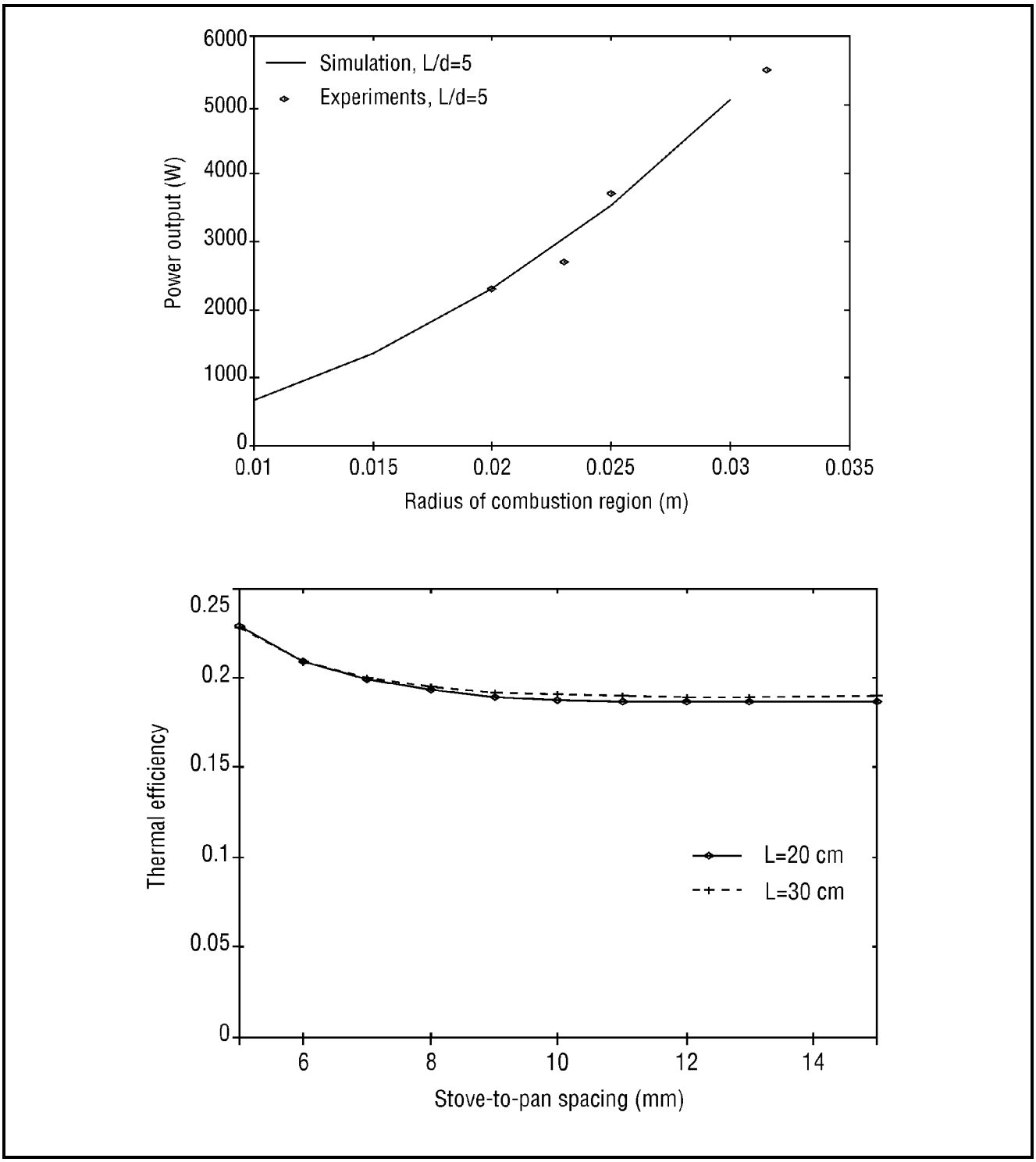


Figure 5. Predictions using the simple model derived from detailed simulation results: (a, above) comparison of predicted power output without pan [Gupta and Abhishek, 2001] with experimental data [Mukunda et al., 1993]; (b, below) predicted variation of thermal efficiency with stove-to-pan spacing [Patra, 2001] for different heights (L) of combustion region

these simulations have been used to develop a simple model. This model is found to produce qualitatively correct predictions, while a quantitative validation of this model is pending. An optimization of sawdust stove dimensions has been carried out with this tentative model, and the results are encouraging. Prediction of carbon monoxide emissions was attempted, in the form of a post-processor for the combustion model, with limited success. Work is currently under way to account for the conical shape of the support plate to study its effects on the stove performance. Besides these, research is also under way on the generalization of the detailed model to a wood-stove, accounting for primary and secondary air inlets, and combustion in a fuel-bed. The culmination of all these projects will, it is hoped, result in a set of model equations and a design procedure that would help stove designers take real effects into account in the design of improved woodstoves. ■

**References**

Agrawal, N., and Saini, M., 2000. *Experimental and Numerical Study of Pyrolysis in an Annular Sawdust Bed,* B.Tech. dissertation, Mechanical Engineering Department, Indian Institute of Technology, Delhi.

Gupta, G., and Abhishek, 2001. *Development of a Semi-empirical Model for Design Optimization of a Sawdust Stove,* B.Tech. dissertation, Mechanical Engineering Department, Indian Institute of Technology, Delhi.

Jain, A., and Upadhyay, R.R., 2001. *Computation and Validation of Coupled Models of Radiation, Combustion and Pyrolysis in a Sawdust Stove,* B.Tech. dissertation, Mechanical Engineering Department, Indian Institute of Technology, Delhi.

Jhalani, A., and Sinha, S., 1999. *Experimental and Theoretical Study of Pyrolysis of Sawdust,* B.Tech. dissertation, Mechanical Engineering Department, Indian Institute of Technology, Delhi, April.

Kohli, S., 1992. *Buoyancy-induced Flow and Heat Transfer in Biomass Stoves,* Ph.D. thesis, Indian Institute of Science, Bangalore, India.

Kohli, S., Srinivasan, J., and Mukunda, H.S., 1993. "Heat transfer to a horizontal disc using a buoyancy-induced jet", *International Journal of Heat and Mass Transfer,* 36(16), pp. 4046-4066.

Kohli, S., Ravi, M.R., and Subbarao, R., 2002. "Mass flow measurement in buoyancy-induced flow through a vertical tube", *Heat and Mass Transfer 2002,* Proc. of the Fifth ISHMT-ASME Heat and Mass Transfer Conference, Saha, S.K., Venkateshan, S.P., Prasad, B.V.S.S.S., and Sadhal, S.S., (eds.), pp. 796-801.

Mukunda, H.S., Dasappa, S., Bhogle, S., and Shrinivasa, U., 1993. "Studies on stoves for powdery biomass", *Int. J. Energy Research,* 17, pp. 281-291.

Patra, S., 2001. *Modelling and optimization of sawdust stove,* M.Tech dissertation, Mechanical Engineering, Indian Institute of Technology, Delhi.

Ravi, M.R., Sinha, S., Jhalani, A., and Ray, A., 2002. "Development of a semi-empirical model for pyrolysis of an annular sawdust bed", *J. Anal. Appl. Pyrolysis,* (in review).

Ray, A., Karthikeyan, S., Kohli, S., and Ravi, M.R., 2002. "Numerical simulation of combustion in buoyancy-induced airflow in a vertical tube with radial fuel inflow", *Heat and Mass Transfer 2002,* Proc. of the Fifth ISHMT-ASME Heat and Mass Transfer Conference, Saha, S.K., Venkateshan, S.P., Prasad, B.V.S.S.S., and Sadhal, S.S., (eds.), pp. 940-945.

Sinha, S., Jhalani, A., Ravi, M.R., and Ray, A., 2000. "Modelling of pyrolysis of wood: a review", *SESI Journal,* 10(1), pp. 41-62.

## Readers' reactions welcomed

Energy for Sustainable Development would appreciate correspondence from readers regarding the contents of the journal and related subjects.

The aim of the journal is to achieve the widest possible dissemination of information pertaining to renewable sources of energy, energy efficiency, the shaping of energy policy, and energy devices, with a focus on solving the energy problems of developing countries.

Readers who would like to contribute to this dissemination effort by making brief comments on material that appears in the journal, or by drawing other readers' and the editors' attention to interesting work and developments in the field of energy, would be aiding the work of dissemination.

Please send your commentative or informative letters to:

***"Correspondence"***
c/o the Associate Editor
Energy for Sustainable Development
25/5, Borebank Road
Benson Town
Bangalore-560 046
India
Fax: +91 80 353 8426
e-mail: ieiblr@vsnl.com