Computer Modelling of Compartment Fires

Hadjisophocleous, G. V.; Yakan, A. C.

Publisher's version / Version de l'éditeur:
https://doi.org/10.4224/20375596
Internal Report (National Research Council of Canada. Institute for Research in Construction), 1992

National Research Council Canada
Conseil national de recherches Canada

Institute for Research in Construction
Institut de recherche en construction

NRC·CNRC

Computer Modelling of Compartment Fires

G.V. Hadjisophocleous and A.C. Yakan

Internal Report No. 613

Date of issue: August 1991

This is an internal report of the Institute for Research in Construction. Although not intended for general distribution, it may be cited as a reference in other publications

Canada

COMPUTER MODELLING OF COMPARTMENT FIRES

by

G.V. Hadjisophocleous and A.C. Yakan

ABSTRACT

This report briefly describes a two-dimensional transient computational fluid dynamics model for compartment fire simulations. The model is based on the numerical control volume solution of the governing partial differential equations for mass, momentum and energy conservation and it models fire as a heat and mass source. The governing equations are solved over a boundary-fitted coordinate system which allows the application of the model to rooms with complex geometries.

This report also presents numerical predictions obtained by this model for a variety of fire scenarios, as well as comparisons with experimental data. These comparisons show that the model can predict compartment fires with reasonable accuracy. Results for other cases are presented to demonstrate the effect of fire intensity, fire location and compartment openings on the flow characteristics in the room.

1. INTRODUCTION

The air flow characteristics, resulting from a fire in a compartment, are influenced by factors such as the fire intensity and location, the compartment openings and the surrounding environmental conditions. An understanding of the conditions in a room during a fire is important as it allows for better designs for fire safety in addressing such issues as optimum locations for sprinklers and smoke detectors and smoke control vents. These conditions are especially critical when dealing with structures with complex geometries.

In recent years, considerable research, using both physical experiments and mathematical models, has been dedicated to understanding the dynamics of fire in a compartment. Full-scale compartment fire experiments have been conducted by several research establishments, such as the Fire Research Station (FRS) [1], the National Institute of Standards and Technology (NIST) [2, 3], Factory Mutual Research Corporation (FMRC) [4] and the Technical Research Centre of Finland [5, 6]. These experiments investigated the flow characteristics resulting from a fire in a compartment under a variety of fire intensities, fire locations, room dimensions and openings and made significant contributions to the enhancement of our understanding of how fires begin and propagate. Full-scale experiments, however, are costly and require considerable time and effort for their preparation. An alternative to some full-scale experiments is computer simulation of full-scale fire scenarios using numerical models.

Computer models require extensive effort for their development and validation but, once validated, they can be used with minor modifications to simulate a variety of fire scenarios. Computer models are of particular importance when dealing with large buildings with complex geometries for which full-scale experiments are not feasible. Several computer models for fire simulations have been developed and used to solve a variety of fire problems. A discussion of these models can be found in Ref. 7.

In this project, a two-dimensional computer model has been modified so that it can be used to simulate compartment fires for different conditions. The numerical predictions obtained by this model are compared with available experimental data to examine the ability of the model to predict events during a fire. Several simulations have been performed to study the effect of fire intensity, fire location and compartment openings on the airflow characteristics in the room. This report gives a brief description of the model and presents the results obtained.

2. PROGRAM DESCRIPTION

The model used to simulate compartment fires consists of two programs. The first program (RMFIRE) solves the equations governing free convection over a boundary-fitted coordinate system and the second (GRIDGN) generates this coordinate system. This section briefly describes these two programs. Additional details can be found in Ref. 8.

2.1 Program RMFIRE

The free convective flows resulting from a fire in a compartment are governed by the conservation equations for mass, momentum and energy. These equations were simplified using the Boussinesq approximation and were normalized using the initial air properties and compartment height as reference parameters. The normalized equations are solved on a boundary-fitted curvilinear coordinate system using numerical methods. The boundary-fitted coordinate system is employed to allow the use of this model for problems with complex geometries without major modifications.

To solve the governing equations on the boundary-fitted coordinate system, it is necessary to transform them to this system. This transformation was made using the chain rule of mathematics and a method described by Maliska and Raithby [9]. In this method, the Cartesian velocity components are maintained as the dependent variables. The equations resulting from this method can be solved using a solution procedure similar to the SIMPLE algorithm.

The equations were discretized following the control volume formulation, in which the solution domain is divided into a number of small volumes. The transformed equations were then integrated over each control volume. A typical control volume, centred at node P and surrounded by the four nodes E,W,N, and S, is shown in Figure 1. The staggered grid approach was used for the locations at which the variables were computed. The Cartesian and Contravariant velocity components were computed at the centre of the control volume faces (e, w, n, s), while pressure and temperature were computed at the centre of the control volume (P).

The solution algorithm follows the SIMPLE Consistent method (SIMPLEC). In this method, a pressure correction equation is derived from the continuity and momentum equations. The solution to this equation yields the pressure corrections which are used to correct the pressure field and to correct the velocities so that they satisfy continuity.

2.1.1 Modelling the Fire

The fire in the compartment was modelled as a heat source and a mass source at the control volumes where the fire was located. The values given for the heat generated were obtained from experimental heat release data. The mass source was determined by the combustion equation and the quantity of fuel burned.

2.2 Program GRIDGN

This program generates the boundary-fitted coordinate system. The method employed for grid generation is the method developed by Thompson et al [10]. This method generates boundary-fitted coordinate systems in any arbitrarily shaped domain by solving the following system of Poisson equations;

$$\xi_{xx} + \xi_{yy} = P(\xi,\eta)$$

$$\eta_{xx} + \eta_{yy} = Q(\xi,\eta)$$

where P and Q are functions which provide control of the mesh concentration.

The boundary conditions required for the solution of these equations are the values of ξ and η at the boundaries.

Since it is desired to perform all numerical computations in the transformed plane, the above equations were transformed from the Cartesian system to the (ξ,η) system using the chain rule. The transformed equations were then solved using numerical methods. The boundary conditions for the transformed equations are the values of the Cartesian coordinates x and y at the boundaries. The solution of these equations yields the values of x and y at discrete ξ and η locations. The x and y values at the grid points as well as their derivatives in both directions are computed and stored as they are needed by the program RMFIRE.

2.3 Solution Procedure

The main computational steps of the method can be summarized as follows:

1. Generate the boundary-fitted coordinate system;
2. Guess values for all dependent variables;
3. Calculate heat release rate and mass source;
4. Solve the energy equation;
5. Solve the momentum equations to obtain the cartesian velocities u and v;
6. Calculate the contravariant velocities U and V from the cartesian velocities u and v;
7. Solve the pressure correction equation;
8. Update U and V using the computed pressure corrections;
9. Compute new u and v from the corrected U and V velocities;
10. Treat all computed values as guessed values, return to 3 and continue until the specified convergence is obtained;
11. Proceed to the next time step, go to 3 and continue until maximum time is reached.

3. DISCUSSION OF RESULTS

Model RMFIRE is still in its early stages of development and requires extensive validation using experimental data before it can be used with confidence to model compartment fires. Two comparisons are given in this study using the experimental data reported in [2].

Some preliminary runs were carried out to demonstrate how this model can be used. These simulations were directed at investigating the effect of fire intensity, fire location and openings on the flow characteristics in the compartment. All simulations performed for this study are summarized in Table 1. The room dimensions and fire locations are shown in Figures 2 and 3.

3.1 Comparisons with experimental data

In this section, the results predicted by the numerical model are compared to experimental data obtained by Steckler et al [2], who carried out a series of compartment fires using various configurations. Two sets of data are used: one for a room with a door on one wall and the second with a window. The fire room for these experiments, shown in Figure 3, was square with walls 2.8 m long and a height of 2.18 m. The burner was located near the rear wall and the fire intensity was 62.9 kW. The ambient temperature during the experiments was 27°C. Velocities and temperatures were measured at steady state along two vertical lines; one at the opening and the other inside the room 0.3 m from the door (Figure 4).

The experimental geometric, fire and ambient data were used as input into the model. Simulations were carried out until steady state conditions were reached. For these simulations, the walls, floor and ceiling of the room were assumed to be adiabatic and no radiation heat transfer is considered. To be able to observe the flow of gases from the room, the computational domain was extended outside the room as shown in Figure 4.

3.1.1 Room with a Door

The predicted velocity and temperature distributions for this run are discussed in Section 3.2. This section presents only the comparisons between predictions and experiments. Figure 5 depicts a comparison between the predicted and experimental temperature profiles at the door. The temperature near the floor is that of the incoming air, hence, is near ambient temperature. The temperature is constant up to about 1.0 m. At that height, the predicted temperature rises sharply while the experimental change is more gradual. The maximum predicted temperature at the door is 125°C while the maximum experimental temperature was 138°C.

The velocity profiles near the door are shown in Figure 6. The maximum velocity measured during the experiment is greater than the predicted one and its location is higher than the location of the predicted maximum velocity. The neutral plane, however, is at almost the same height for both. The predicted height of the neutral plane is 1.1 m while the measured height in the experiment was 1.05 m.

A comparison of the temperature profiles 0.3 m inside the room is shown in Figure 7. The predicted thickness of the hot layer is slightly greater than the experimental one. The maximum predicted hot layer temperature, however, is lower than the experimental. An interesting point is that the predicted temperature near the floor is the ambient one while the experimental is at about 10 degrees higher. This increase of the temperature near the floor might be due to mixing of the two layers and radiation from the hot gases to the lower layer. Radiation heat transfer is not considered by the model at this stage.

3.1.2 Room with Window

The comparisons between the predicted and experimental temperatures and velocities when the opening is a window are shown in Figures 8, 9 and 10. The results for this case are similar to the ones discussed for the door case. Despite the difference in the profiles, it is interesting to see that the height of the neutral plane is almost the same for both data. This interface height is an important factor in compartment fires. The lower the interface, the higher the risk to the occupants trying to evacuate the building.

3.2 Effect of room opening on flow characteristics.

In this section, the predicted results of the two simulations used for the comparisons with experiments are analyzed further to examine the effect of openings on the flow characteristics. Figure 11 depicts the velocity vectors in the room and the adjoining space outside the room for the door case. The hot plume generated by the fire moves upward along the wall, then follows the ceiling and moves towards the door. At the door, it exits the room creating a hot plume along the exterior wall. This motion of the hot gases creates a negative pressure inside the room causing cold air to enter the room. The temperature contours in °C for this case are shown in Figure 12.

Figures 13 and 14 depict the velocity vectors and isotherms for the window case. The flow pattern in the room is slightly different from the door case as now there are, at the centre of the room, secondary recirculating vortices causing an increase in the thickness of the hot layer (Figure 14). This increase in the hot layer thickness is also affected by the decrease of the quantity of hot gases leaving the room, a result of the smaller opening. This can be seen in Table 2, which shows a summary of the results for these two simulations. The mass flow rate through the door is 0.76 kg/s, and through the window, 0.68 kg/s. The maximum inflow velocity, also shown in Table 2, is 0.75 m/s for the door case and 1.14 m/s for the window case. The maximum outflow velocity, however is about the same for the two cases at 1.38 m/s. As shown in Table 2, the maximum and mean temperatures in the room with the window are higher than for the room with the door. This result is caused by the difference in the mass flowrates out of the compartment.

3.3 Effect of fire location on flow characteristics

Three simulations were performed to study the effect of fire location on the flow characteristics, the first was with the burner at the inner corner, the second with the burner in the middle of the room and the third with the burner near the opening, which was a door for all cases. Table 3, which summarizes the results of these simulations, indicates that the mass flow rate decreases as the burner moves towards the door. The decrease of the incoming air causes an increase of the mean temperature in the room. The maximum temperature at the door increases as the burner moves from the back corner to the door. The velocity profiles at the door for the three cases are shown in Figure 15. The profiles are similar for the cases with the burner located near the door and in the middle of the room, with a maximum incoming velocity of about 0.8 m/s. When the burner is located at the corner, the maximum incoming velocity increases to about 1.4 m/s and the height of the neutral plane decreases.

3.4 Effect of fire size on flow characteristics.

Four simulations were performed to study the effect of fire size on the flow behaviour in the fire compartment. The heat release rates used were 19, 39, 63 and 97 kW. For all cases, the burner was located near the back wall (Location C) and the room opening was a door.

As expected, the mean temperature in the room and the maximum temperature at the door increase as the fire size increases (Figure 16). Figure 17 indicates that the mass flow rate in and out of the compartment increases as the fire size increases. This is due to the stronger buoyancy forces generated.

The velocity profiles at the door for the four heat release rates used are shown in Figure 18. The velocities increase with the increase in the heat release rate, however, it is interesting to note that the elevation of the neutral plane does not change.

4. CONCLUSIONS

The air flow characteristics resulting from a fire in a compartment are modelled using a two dimensional field model. The results predicted by the model are compared with experimental data. The comparisons show reasonable agreement for velocities, temperatures and mass flow rates. The model, however, needs further improvements and validation before it can be used to accurately predict compartment fires.

To demonstrate some potential applications of such a field model, a number of simulations have been carried out to investigate the effect of openings, fire location and fire size on the flow characteristics and temperature distribution in the room. The predicted results show that:

1. The size of the openings affects the mass flow rates in and out of the room which, in turn, affect the room temperature and the height of the hot layer. As the combustion process is not modelled, the effect of ventilation on the fire itself cannot be seen.
2. Fire location has a significant effect on the flow characteristics in the room. The closer the fire is to the door, the lower the mass flow rates in and out of the room, causing higher room temperatures.
3. The fire size simulations indicate that a larger fire will create larger flow rates in and out of the room and higher room temperatures.

These simulations were done to illustrate that the model is capable of predicting compartment fires, and to determine the modifications necessary to improve the model. As a first step towards improving the model the following are recommended:

1. modify the model to account for three dimensional effects;
2. incorporate radiation heat transfer from the hot gases to the walls, floor and ceiling;
3. incorporate convection heat transfer from the hot gases to the room walls; and
4. incorporate the k-ε turbulence model.

REFERENCES

1. Heselden, A.J.M., "Fire Problems of Pedestrian Precints. Part 1. The Smoke Production of Various Materials", Joint Fire Research Organization, Fire Research Station, Fire Research Note 856, 1971.
2. Steckler, K. D., Quintiere, J. G. and Rinkinen W. J., "Flow Induced by Fire in a Compartment", NBSIR 82-2520, National Bureau of Standards, Washington, D.C., 1982.
3. Quintiere, J. G. and McCaffrey, B. J., "The Burning of Wood and Plastic Cribs in an Enclosure: Volume I", NBSIR 80-2054, National Bureau of Standards, Washington, D.C., 1980.
4. Heskestad, G. and Hill, J. P., "Experimental Fires in Multiroom/Corridor Enclosures", NBS-GCR-86-502, National Bureau of Standards, Gaithersburg, MD, (January 1986).
5. Ahomen, A., Holmlund, C. and Kokkala, M., "On the Influence of Ignition Power and Burner Size on the Fire Growth in Room Fire Experiments", Technical Research Centre of Finland, Research Report 383, 1986.
6. Ahomen, A., Kokkala, M. and Weckman, H., "Burning Characteristics of Potential Ignition Sources of Room Fires", Technical Research Centre of Finland, Research Report 285, 1984.
7. Galea, E. "On the Field Modelling Approach to the Simulation of Enclosure Fires", J. of Fire Prot. Engr., Vol. 1, pp 11-22, 1989.
8. Hadjisophocleous, G.V., "Numerical Modelling of LPG Horizontal Cylindrical Tanks Subjected to Heating Environments", Ph.D. Thesis, University of New Brunswick, 1989.
9. Maliska, C.P. and Raithby, G.D., "A Method for Computing Three Dimensional Flows Using Non-orthogonal Boundary Fitted Coordinates", Int. J. Numerical Methods in Fluids, Vol. 4, pp. 519-537, 1984.
10. Thompson, J.F., Thames, F.G. and Mastin, C.W., "Automatic Numerical Generation of Body Fitted Curvilinear Coordinate System for Field Containing any Number of Arbitrary Two Dimensional Bodies", J. Comput. Physics, Vol. 15, pp. 299-319, 1974.

LIST OF FIGURES

Figure 1. Typical control volume
Figure 2. Three dimensional view of the fire compartment
Figure 3. Burner locations
Figure 4. Solution domain and locations where temperature and velocities are computed for model validation
Figure 5. Comparison of predicted and experimental temperatures at the door
Figure 6. Comparison of predicted and experimental velocity profiles at the door
Figure 7. Comparison of predicted and experimental temperatures inside room near door
Figure 8. Comparison of predicted and experimental temperature profiles at the window
Figure 9. Comparison of predicted and experimental velocity profiles at the window
Figure 10. Comparison of predicted and experimental temperature profiles inside the room (with window opening)
Figure 11. Velocity vectors for the door case and burner located at the inside corner
Figure 12. Temperature contours for the door case and burner located at the inside corner
Figure 13. Velocity vectors for the window case and burner located at the inside corner
Figure 14. Temperature contours for the window case and burner located at the inside corner
Figure 15. Predicted velocity profiles at the door for various fire locations
Figure 16. Predicted mean room temperature and maximum temperature at opening as a function of fire size
Figure 17. Predicted mass flow rate through door as a function of fire size
Figure 18. Predicted velocity profiles at the door for various fire sizes

LIST OF TABLES

Table 1. Summary of cases simulated
Table 2. Comparison of results between window and door opening; fire intensity 62.9 kW, location inner corner of room.
Table 3. Comparison of results for different burner locations; fire intensity 19 kW, opening door.

Table 1 Summary of cases simulated

Cases Simulated	Opening	Fire Intensity kW	Fire Location
1	Door	62.9	Corner
2	Window	62.9	Corner
3	Door	19.0	Corner
4	Door	19.0	Entrance
5	Door	19.0	Center
6	Door	19.0	Entrance
7	Door	19.0	Entrance
8	Door	19.0	Entrance
9	Door	19.0	Entrance

Table 2 Comparison of results between window and door opening; fire intensity 62.9 kW, fire location inner corner of room.

Opening	Maximum Inflow Velocity m/s	Maximum Outflow Velocity m/s	Mass Flow Rate Out kg/s	Mean Room Temperature °C	Maximum Temperature at Opening °C
Door	0.75	1.38	0.76	55.6	123.4
Window	1.14	1.37	0.68	61.4	132.8

Table 3 Comparison of results for different burner locations; fire intensity 19 kW, opening door.

Fire Location	Maximum Inflow Velocity m/s	Maximum Outflow Velocity m/s	Mass Flow Rate Out kg/s	Mean Room Temperature °C	Maximum Temperature at Opening °C
Entrance	0.52	0.95	0.51	40.5	71.7
Center	0.74	1.07	0.66	40.3	63.5
Corner	1.41	1.18	0.82	37.2	58.9

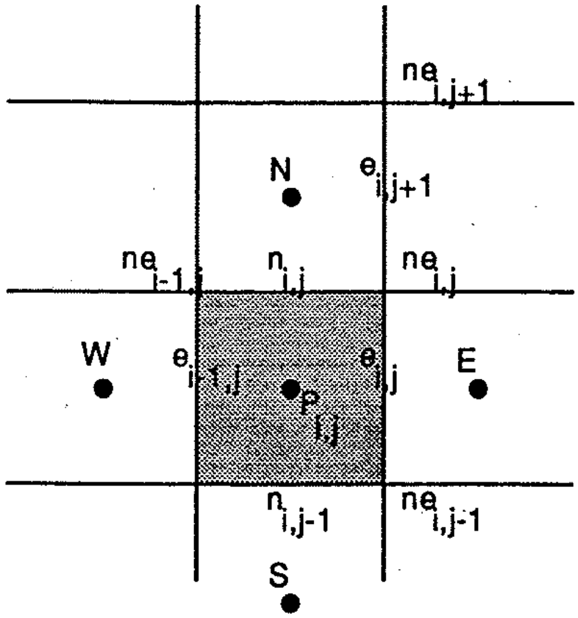

Figure 1: Typical Control Volume

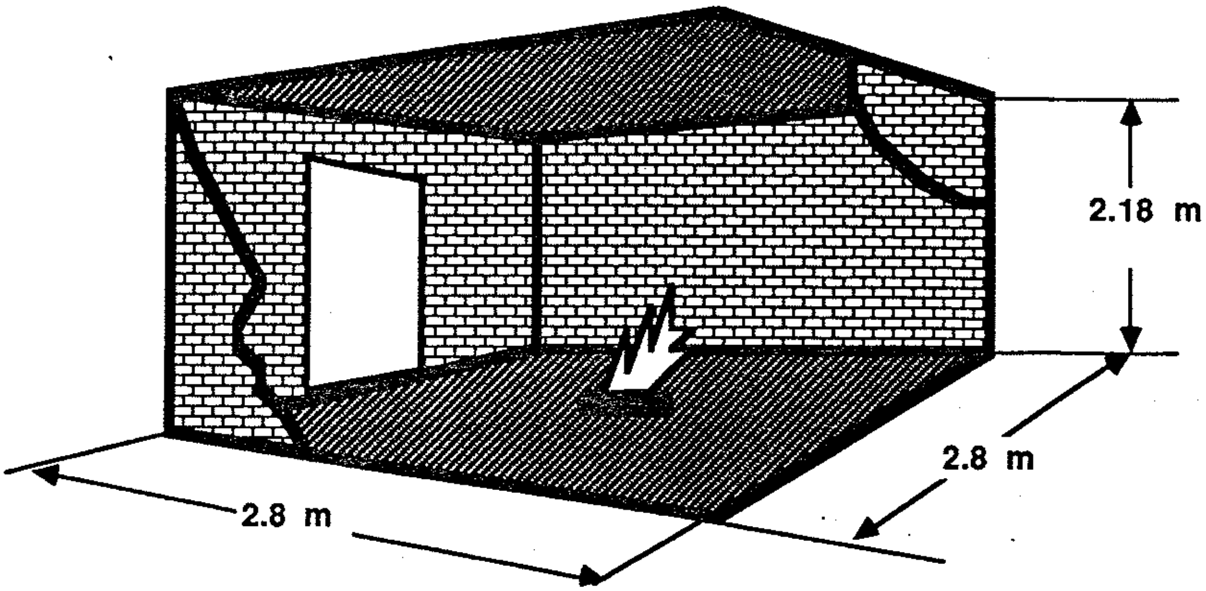

Figure 2 : Three Dimensional View of the Fire Compartment

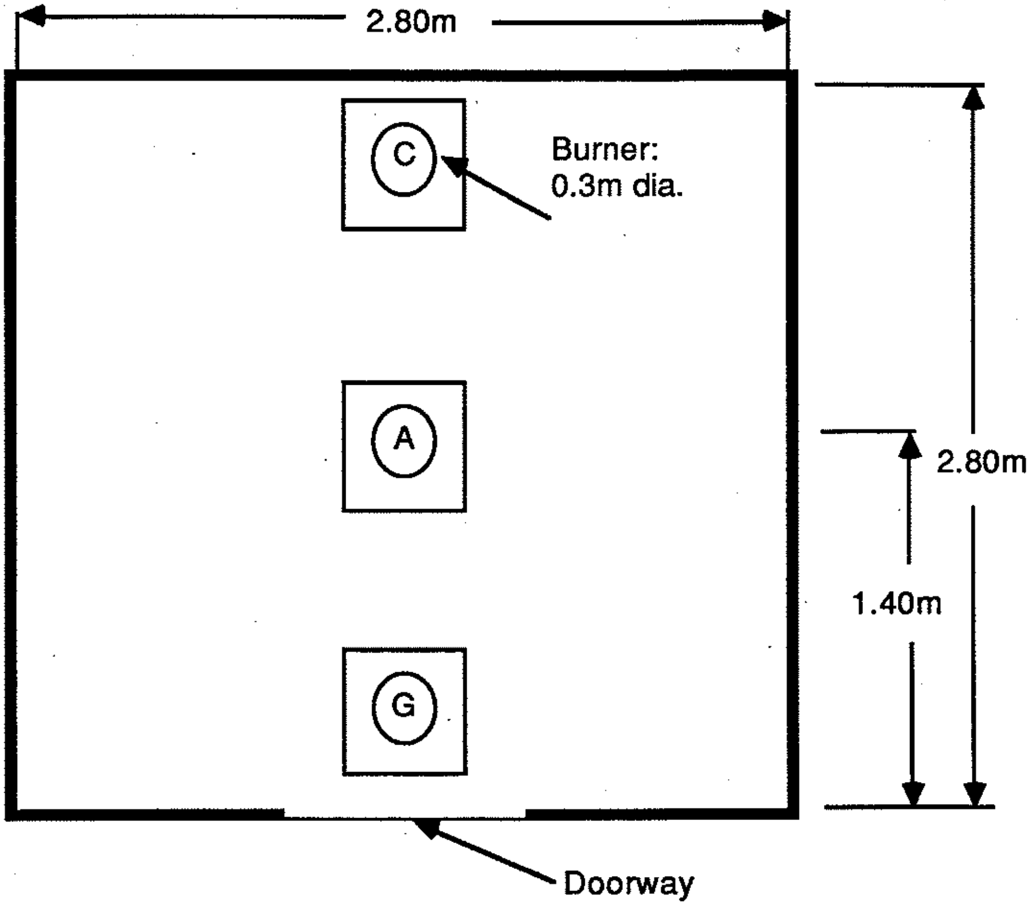

Figure 3: Burner Locations

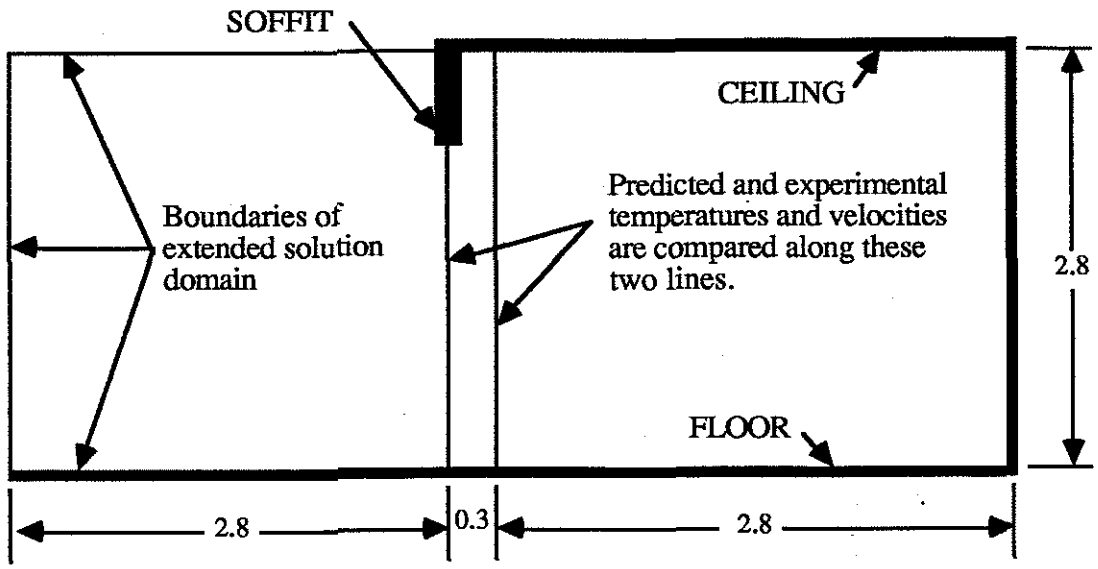

Figure 4 : Solution domain and locations where temperature and velocities are computed for model validation.

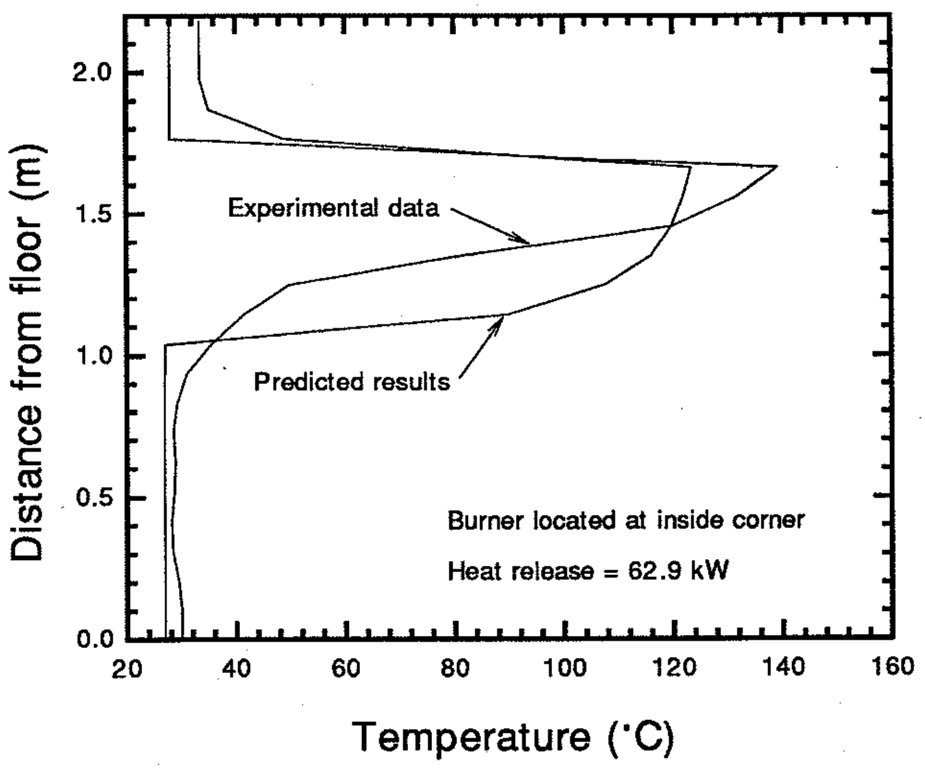

Figure 5 Comparison of predicted and experimental temperatures at the door.

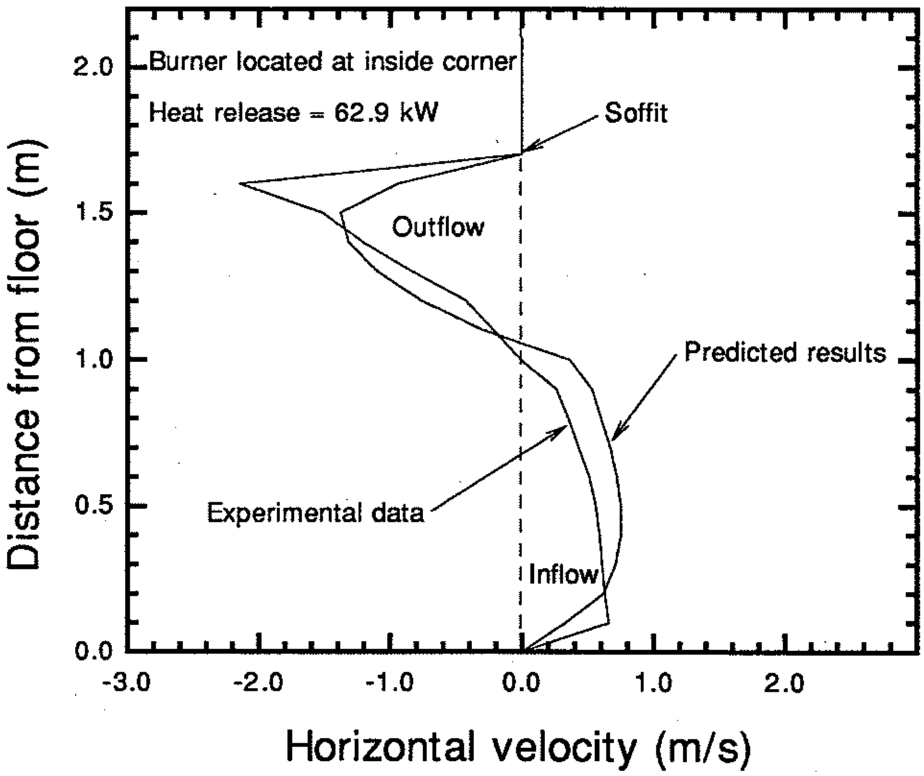

Figure 6 Comparison of predicted and experimental velocity profiles at the door.

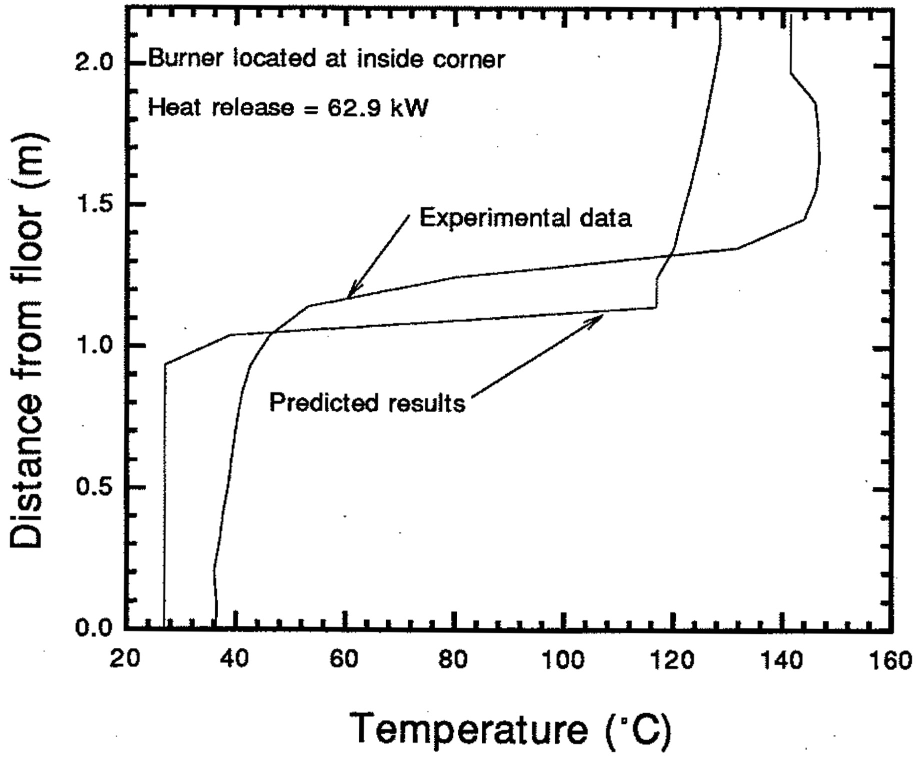

Figure 7 Comparison of predicted and experimental temperatures inside room near door.

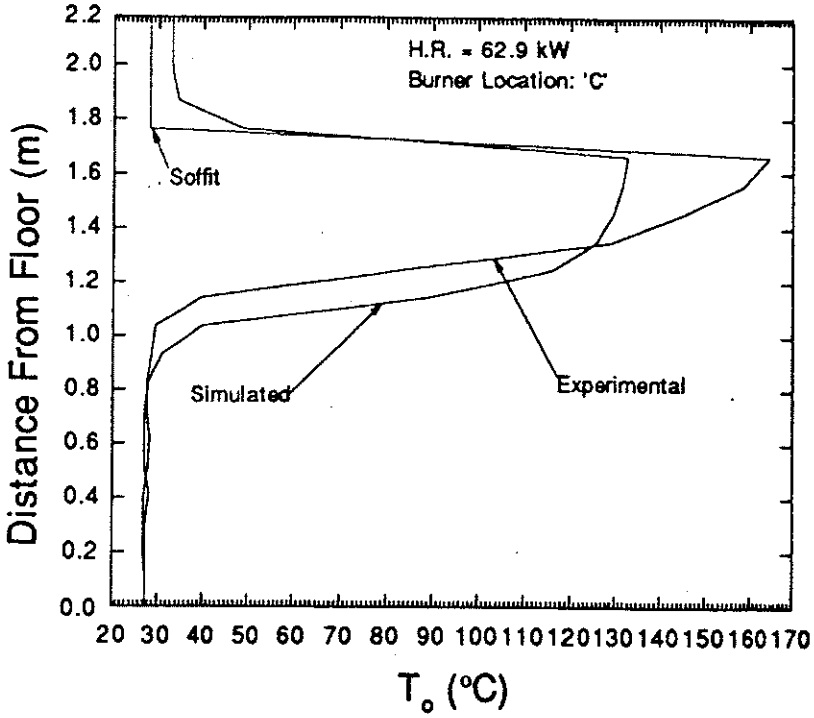

Figure 8 Comparison of predicted and experimental temperature profiles at the window.

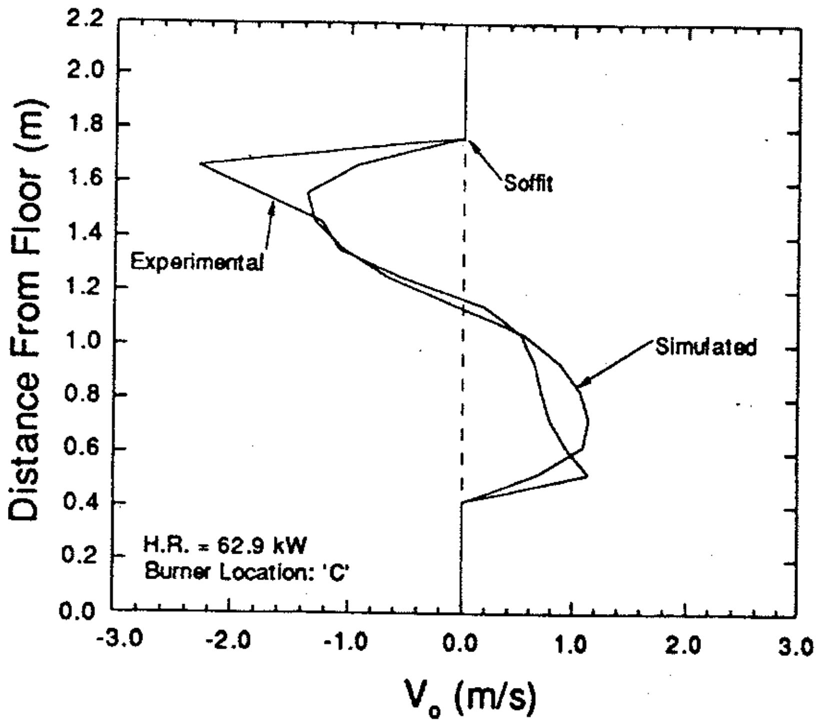

Figure 9 Comparison of predicted and experimental velocity profiles at the window.

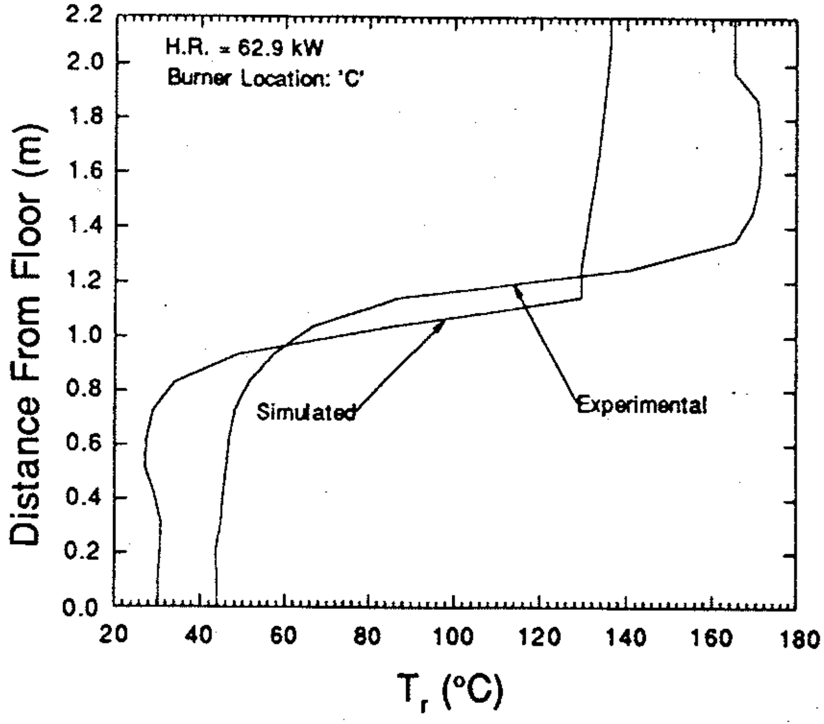

Figure10 Comparison of predicted and experimental temperature profiles inside the room (with window opening).

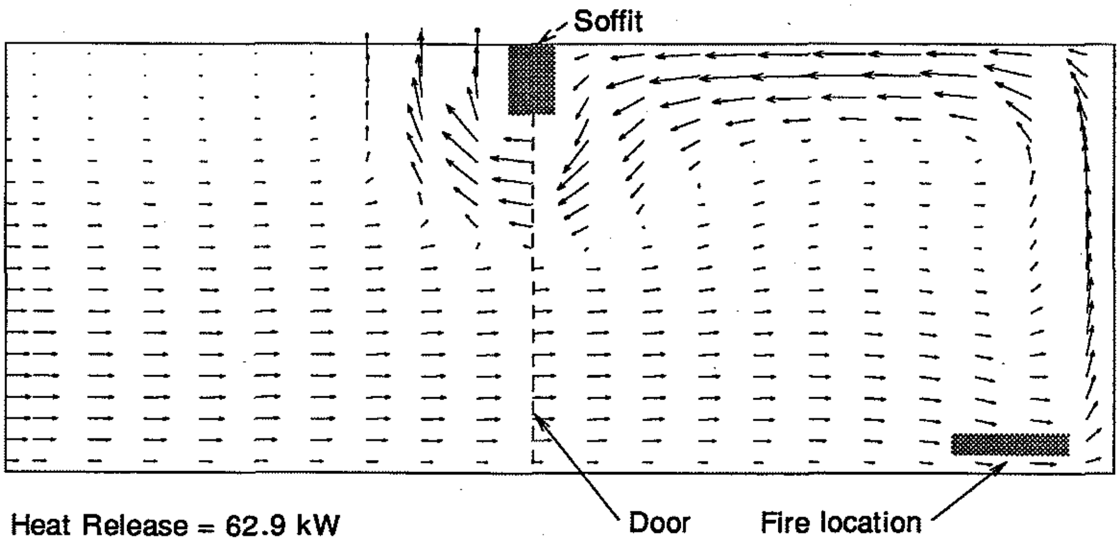

Figure 11 Velocity vectors for the door case and burner located at the inside corner.

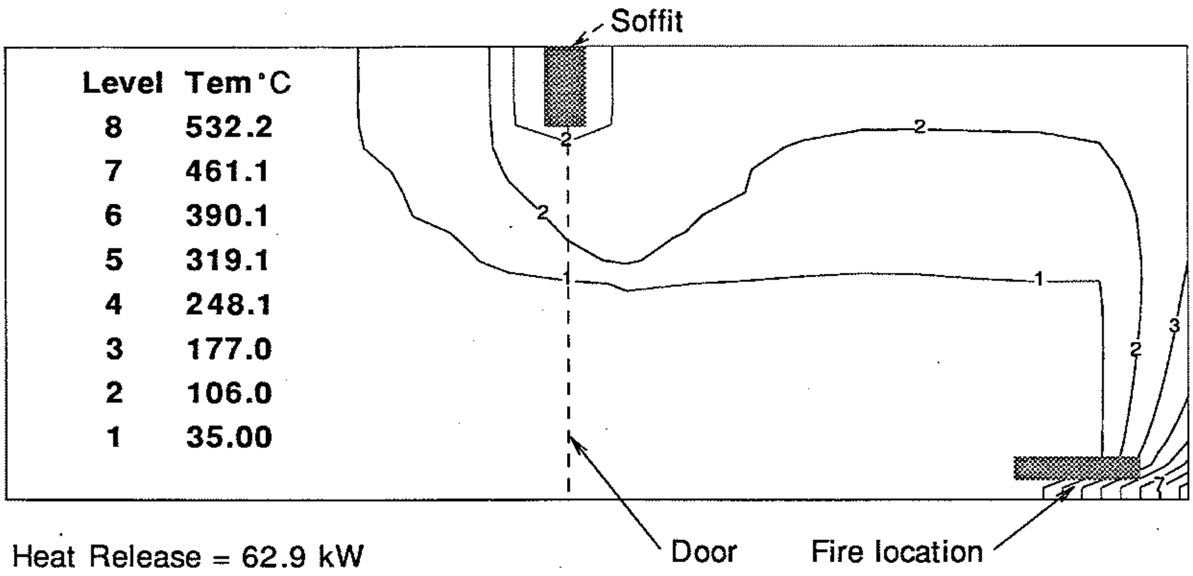

Figure 12 Temperature contours for the door case and burner located at the inside corner.

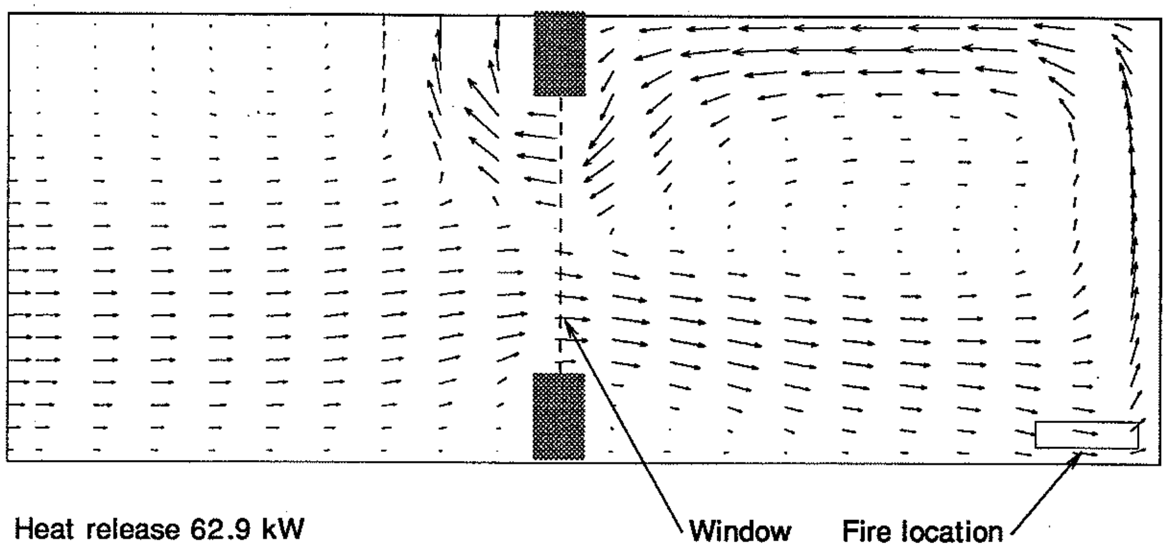

Figure 13 Velocity vectors for the window case and burner located at the inside corner.

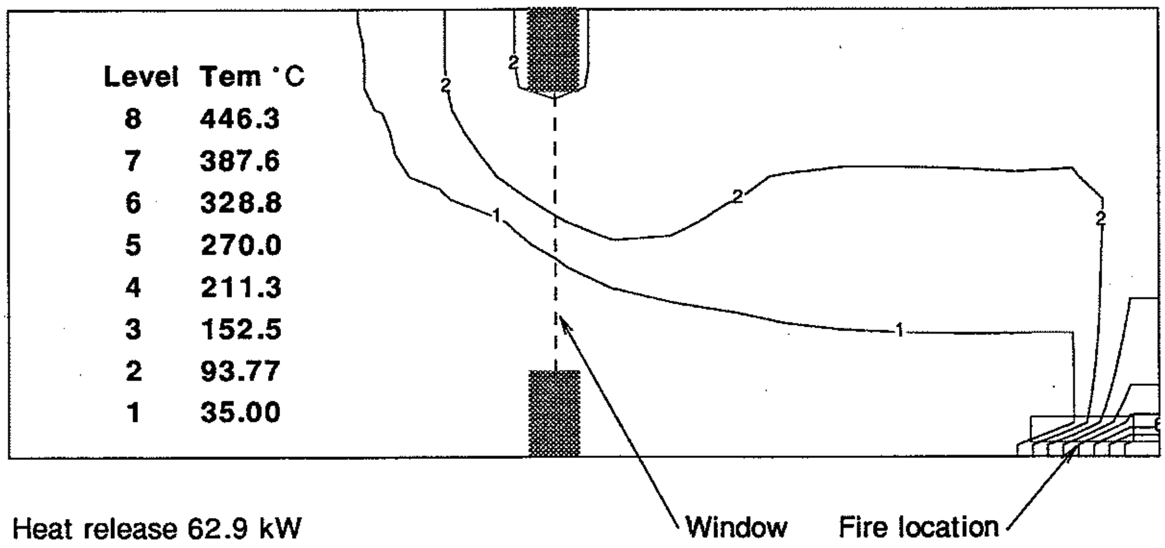

Figure 14 Temperature contours for the window case and burner located at the inside corner.

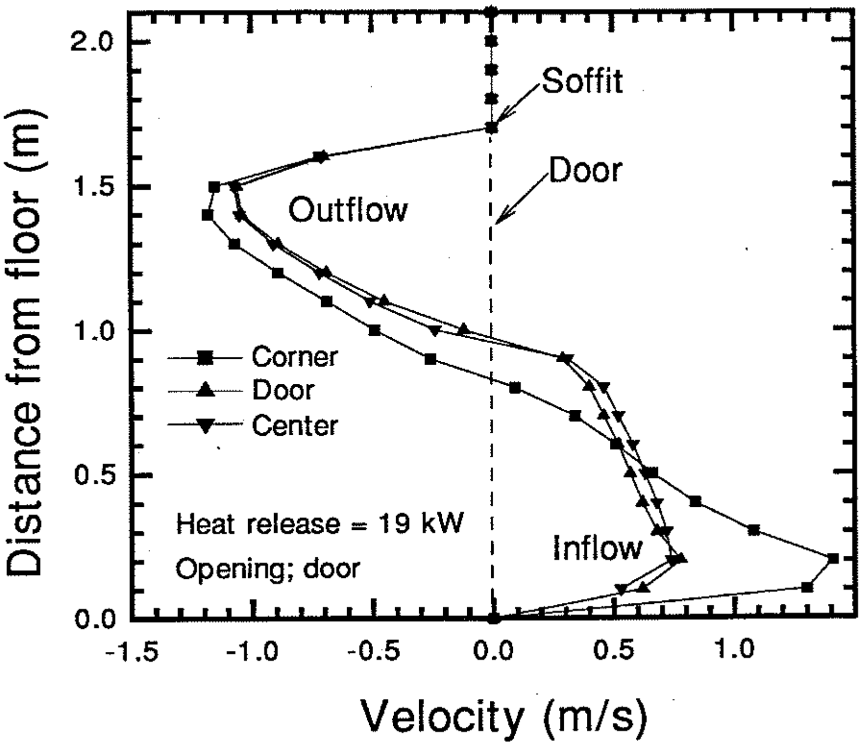

Figure 15 Predicted velocity profiles at door for various fire locations.

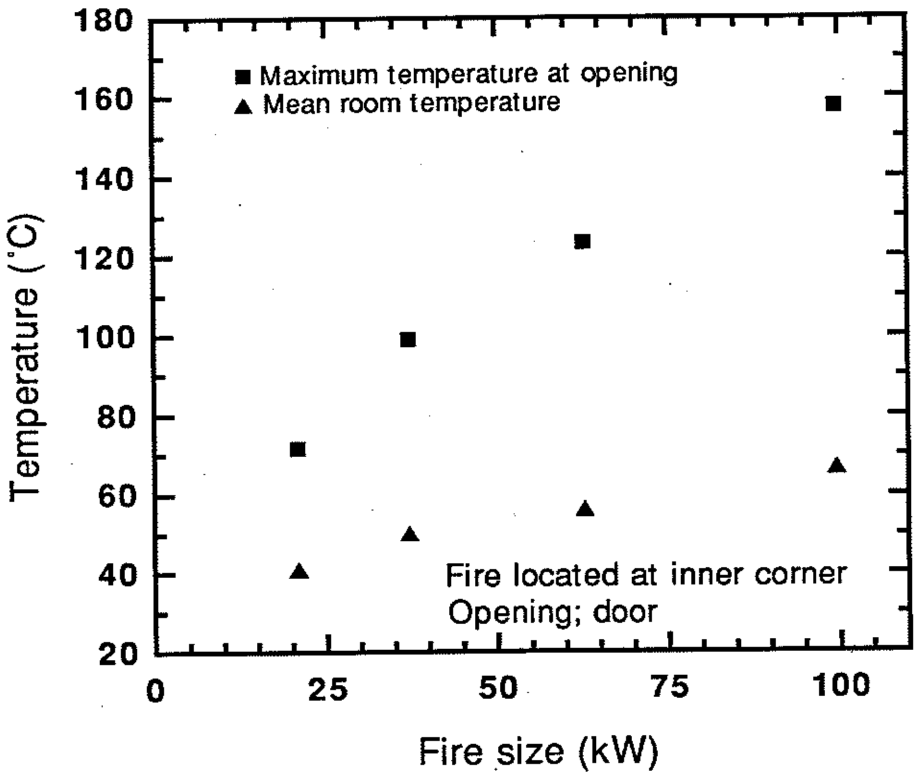

Figure 16 Predicted mean room temperature and maximum temperature at opening as a function of fire size.

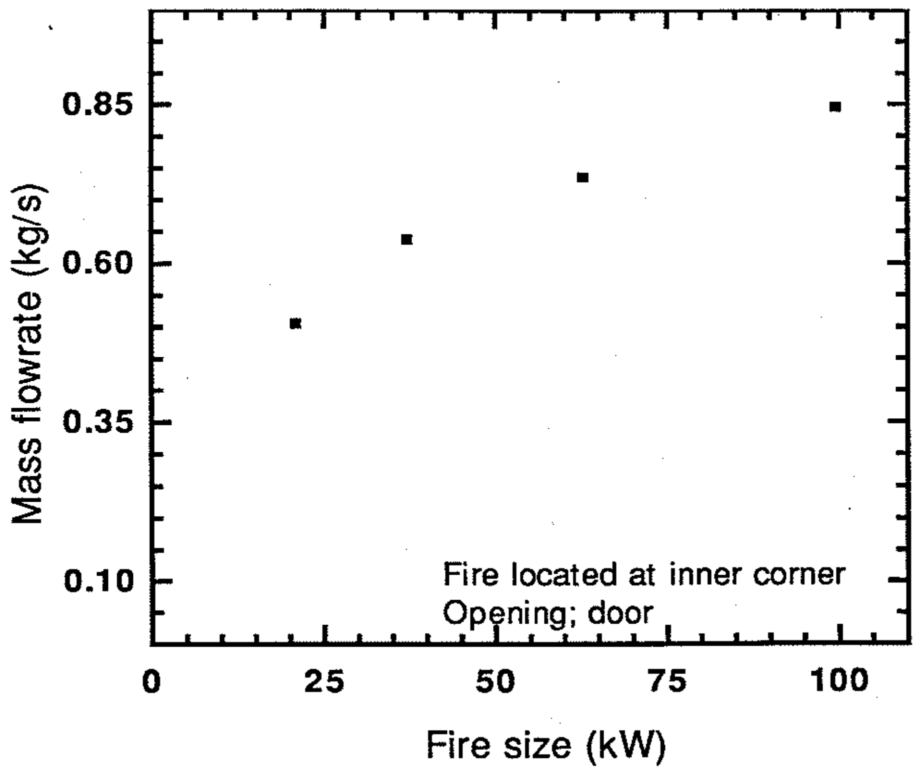

Figure 17 Predicted mass flowrate through door as a function of fire size.

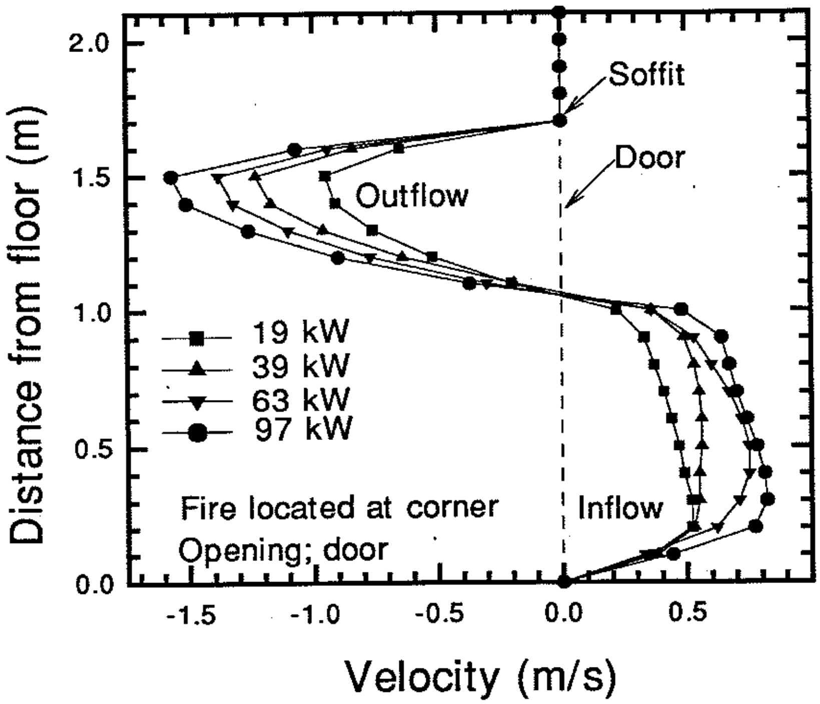

Figure 18 Predicted velocity profiles at door for various fire sizes.