EVALUATION OF PHOENICS CFD FIRE MODEL AGAINST ROOM CORNER FIRE EXPERIMENTS

Yunlong Liu and Vivek Apte

CSIRO Fire Science and Technology Laboratory
PO Box 310 North Ryde, NSW 1670, Australia
TEL:+61 2 9490 5444, FAX: +61 2 9490 5777
Email: Yunlong.Liu@csiro.au

ABSTRACT

This paper evaluates the performance of a general-purpose CFD package, PHOENICS, by comparing the temperature field predicted by PHOENICS with the measurements obtained in room corner fire tests conducted at CSIRO. A critical input to the model is a design fire, which, in the present case was a heat release rate (HRR) profile of a gas burner, used in the fire tests. Two types of burner programs - ISO and ASTM, with different heat release rate profiles, were chosen as examples of enclosure fires to validate PHOENICS. These tests were conducted in a room, which was 3.6m long x 2.4m wide x 2.4m high with a 0.8m wide x 2.0m high door. The only fire source in the test was a gas burner in the corner of the room. These tests involved measurements of heat release rate, temperature field and heat fluxes on the floor, induced by the gas burner. There was no fire spread in this case, as the walls and the ceilings were covered with non-combustible plasterboard.

The k-ε turbulence model adapted for describing buoyancy-generated turbulence was employed for turbulence modelling. Radiation was approximated by the Radiosity model, and the absorption coefficient of the fluid media was set at 0.25m^{-1}, as little soot was generated during the tests. The influence of the wall boundary condition treatment has also been investigated in this study, including the applicability of adiabatic boundary condition and the conjugate heat transfer (CHT) boundary condition. It is concluded that the temperature profile predicted using the conjugate heat transfer boundary condition agrees better with the experimental results. It has also been concluded that PHOENICS can serve as a tool for modelling fire development in an enclosure.

KEY WORDS
CFD Fire modelling, validation, PHOENICS, field model

INTRODUCTION

Enclosure fire modelling can fall into two categories, zone model and field model, i.e. the CFD model. The zone model assumes the room as two zones and analyses the hot layer and cold layers' fire related parameters. The field model utilising CFD divides the computational domain into many small elements, in which the fluid flow and heat transfer parameters are resolved. If the mesh size is small enough, the field data of gas temperature and flow velocity approach the exact value within the computational domain.

With the fast development of digital computer systems, computational fluid dynamics (CFD) is gaining popularity in the modelling of a fire scenario. CFD modelling techniques can be used to model the fire development and extract a comprehensive transient picture of a real fire scenario[1]. The objective of the fire modelling is to quantitatively predict the fire behaviour in relevant situations and thereby provide information to fire protection engineers for fire safety design purposes.

G. Cox et al[2] successfully simulated a steady burner generated fire plume inside an enclosure, the heat transfer inside the solid wall being assumed as one-dimensional. Lockwood et al[3] simulated two test cases and found that the numerically simulated gas temperature in the enclosure agrees with the test data very well.

Several special-purpose and general-purpose software packages have been developed in recent years. Fire Dynamics Simulator (FDS)[4], SmartFire[5] are popular special-purpose softwares, and CFX[6], FLUENT[7] and PHOENICS[8] are among the popular general-purpose CFD softwares. This study took one of the general-purpose CFD software packages, PHOENICS[8], as an example, to investigate the feasibility of using this program for the modelling of a fire in an enclosure.
In this study, the k- ε turbulence model is employed to resolve the buoyancy-generated turbulence, and the thermal radiation is approximated by the Radiosity model[9].

CSIRO WALL LINING TEST EXPRIMENT

This investigation is based on two fire experimental tests that were conducted by the CSIRO fire research group[10]. The test room was 3.6m long x 2.4m wide x 2.4m high with a 0.8m wide x 2.0m high door. Figure 1 gives a cut-away view of the ISO test room[10]. In this study, two experiments with plasterboard wall lining materials are considered, where there was no fire spread and the heat release was contributed only by a methane burner in a corner of the room. The burner was located in the corner opposite the door opening, and the burner dimensions were 0.3mx0.3mx0.3m.

Two burner heat release rate profiles were used in the experiments. In the ISO test method[10], named as case A hereafter, the supplied methane generated a heat release rate (HRR) of 100kW in the first ten minutes, which was then increased to 300kW and maintained at this rate during the following ten minutes. In the ASTM method[10], named as case B hereafter, the supplied methane generated a heat release rate of 40kW during the first five minutes, which was then increased to 160kW, and was maintained at this level for 10 minutes. The gas temperature development history at several locations below the ceiling was recorded with type K thermocouples at 5-second intervals. These monitor points were located 0.05m below the ceiling centre, 0.1m below the top of the doorway centre, and 0.05m below the ceiling directly above the burner. The recorded time-dependent temperature data formed the basis for the validation of PHOENICS software package.

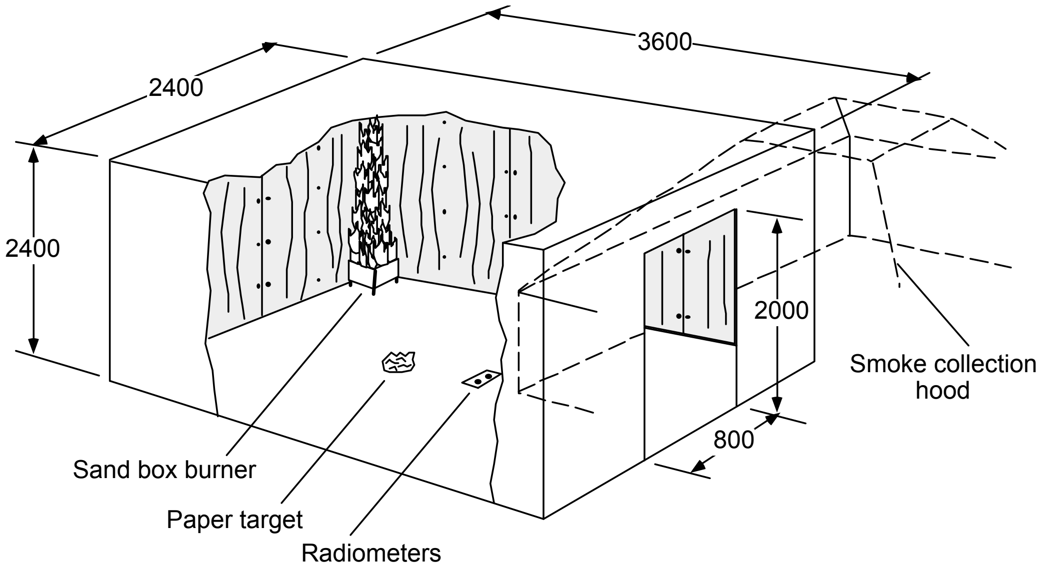

Figure 1. Cut-away view of the ISO and ASTM room fire test.

NUMERICAL DETAILS

The present study uses PHOENICS v3.4 to calculate the temperature field generated by the burner inside the room. The model used in this study does not incorporate fire spread, and the experiments also did not result in flame spread on the linings, which were non-combustible.

Two types of boundary conditions have been tested, adiabatic boundary condition and the conjugate heat transfer. For the case with adiabatic boundary condition, the whole computational domain was fluid with the computational domain ending at the inner wall surface of the linings, where the fluid-wall interface was assumed to be adiabatic. For the case with a conjugate heat transfer boundary condition, the computational domain was made up of the indoor gas domain, 0.1m-thick ceiling, 0.1m-thick walls and 0.1m-deep solid floor. In this case, the computational domain and the boundary were extended to the exterior wall surface to take into account the heat transfer into the wall. To eliminate the influence of the boundary conditions imposed on the doorway plume region, the computation domain was extended 1.8m beyond the door, where pressure boundary conditions were applied. Figure 2 shows the computation domain and the CFD simulated plume flow through the door.

In this study, the Radiosity radiation model[9] has been tested. As the fire-generated buoyancy driven flow is turbulent, and results into natural convection, the RANS turbulence model was employed to resolve the subscale turbulence. The k-ε RANS model adapted to account for the buoyancy effects was used is in this investigation.

The fire source is taken as the input parameter, which is a stable heat release rate that was designed to represent the experimental measured HRR by the calorimeter, as shown in Table-1a and Table-1b. A typical t^2 curve HRR was employed for the first 100 seconds from the ignition and for the HRR jump period during the test, as shown in Figure 3 and Figure 4.

Table-1a: Heat release rate from the experimental fire source: case A

Time (s)	0-10	10-20
Fire intensity (kW)	100	300

Table-1b: Heat release rate from the experimental fire source: case B

Time (s)	0-10	10-15
Fire intensity (kW)	40	160

As a fire scenario is transient, a smaller time step is taken in the initial stage of the fire when the temperature and flow field development is fast, and a larger time step is taken for the steady developing stage. Detailed time step lengths are shown in Table 2a and Table 2b.

Table-2a: Time step length for transient calculation of case A

time	0-30s	30-200s	200-600s	600-630s	630-800s	800-1200s
Step number	60	200	150	60	200	150
Time step	0.5s	0.85s	2.67s	0.5s	0.85s	2.67s

Table-2b: Time step length for transient calculation of case B

time	0-30s	30-100s	100-300s	300-330s	330-400s	400-900s
Step number	60	100	100	60	100	200
Time step	0.5s	0.7s	2.0s	0.5s	0.7s	2.5s

Non-uniform mesh distribution was employed to reduce the CPU cost. In the region near the fire source, a fine mesh was used, for example, a mesh size was as small as 0.02m in the fire source region. In the region where temperature gradient is not large and large eddies exists, a coarse mesh is used, for example, in the region far from the walls, mesh size can be as large as 0.1m.

Two different mesh sizes, coarse mesh and fine mesh, were tested. The coarse mesh was 37 x 32 x 34 in three dimensions, which gave a near wall mesh resolution of 0.045m x 0.070m x 0.070m in three directions. The fine mesh was 73 x 61 x 61 in three dimensions, which gave a near wall mesh resolution of 0.025m x 0.022m x 0.030m.

For the coarse mesh case, about 2hours CPU time was needed to finish a case on an Intel Pentium-4 PC with a CPU processor speed of 2GHz. For the fine mesh case, about 12 hours CPU time was needed.

The temperature and the gas flow development history at different locations below the ceiling was recorded in the results of the CFD modelling. The CFD modelling accuracy is evaluated on the basis of the predicted gas temperature development history at these locations.

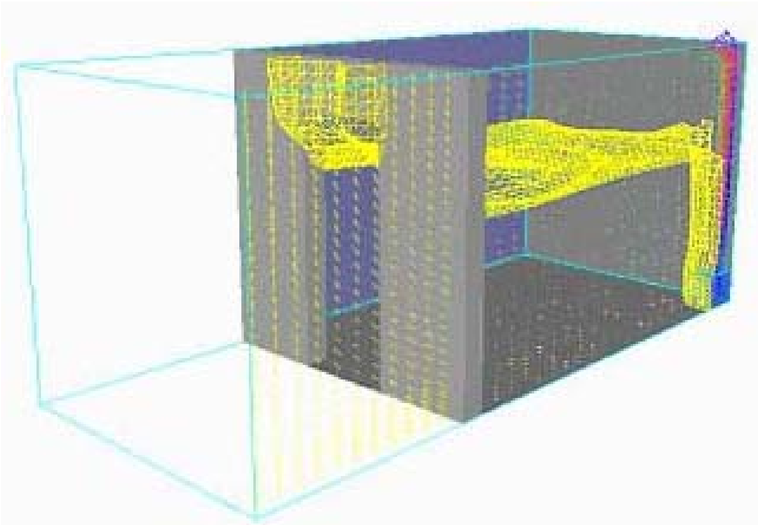

Figure 2: Temperature Iso-surface of 373 degree C

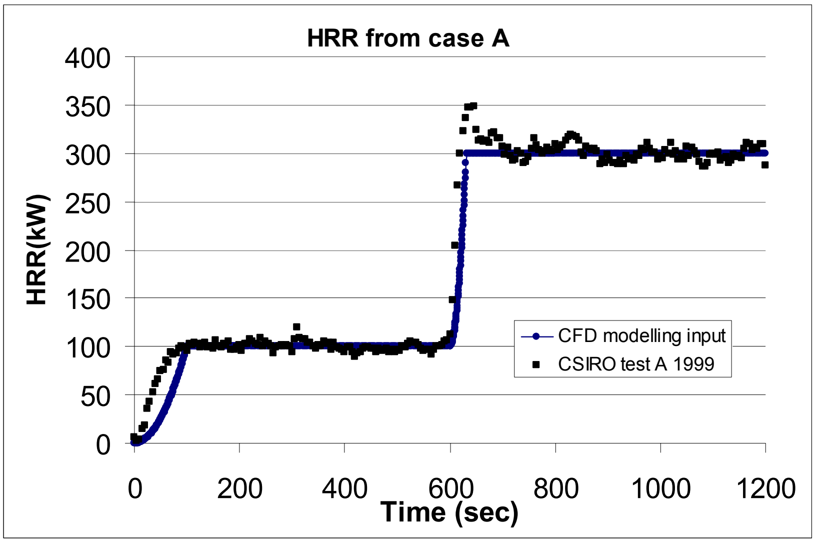

Figure 3: Heat release rate (HRR) from the fire source in case A

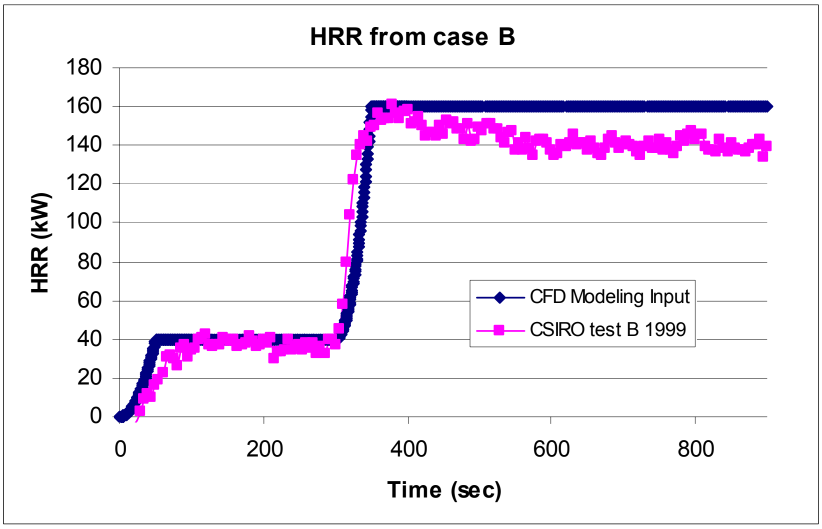

Figure 4: Heat release rate (HRR) from the fire source in case B

DISCUSSION OF RESULTS

Based on the CSIRO experiment[10], the temperature development history at several key locations below the ceiling were taken as the criteria for comparison. As the flow field and the temperature field interact with each other, the accuracy resolution of the transient temperature field can serve as an indicator to evaluate the applicability of the CFD software package.
Figure 2 presents a CFD predicted temperature iso-surface of 373 ºC 20 minutes after the ignition of the gas. This shows that the air in the test room was divided into two layers – a hot layer and a cold layer below the iso-layer. According to Figure 2, the hot gas plume rose to the higher region of the room and exited through the doorway by natural ventilation. In Figure 2, the hot layer gas temperature is above 373 °C.

All the CFD predicted curves presented in Figure 5-8 for case A using a fine mesh, and the CFD results in Figure 5-7 were for the conjugate heat transfer boundary conditions.
Figure 5 gives a comparison of the CFD predicted and measured gas temperature development history at a location 0.05m below the ceiling centre. Both CFD predicted and the measured temperature history at this location shows that gas temperature increased very fast in the first 100 seconds after the ignition. After that, when the heat release from the burner is stable, little temperature increase was detected. This is because the heat released from the burner is dissipated through natural ventilation and radiative heat transfer into the wall surfaces, so the gas temperature in the room achieved a balance. Relatively larger error between the CFD predicted and the measured gas temperature can be found in the first 100 seconds after the ignition and during the burner HRR jump period at 10 minutes after the ignition. This is because the simulated design fire HRR has some errors compared to the actual heat release of the CSIRO experiment, as shown in Figure 3 and Figure 4. According to Figure 5, during the stable burning period, the gas temperature development had been correctly predicted by PHOENICS, with an error of less than 10 percent.

Figure 6 presents the comparison of the CFD predicted and the measured gas temperature development history at a location 0.1m below the top of the doorway centre. This figure also shows the fast gas temperature development during the first 100 seconds after ignition; the gas temperature remained stable with little temperature change during the following ten minutes, this trend agrees with the CSIRO measured results, the relative error being about 20 percent.

Figure 7 gives a comparison of the CFD predicted and the measured gas temperature development history at a location above the burner and 0.05m below the ceiling. This figure shows that the CFD computation under-predicted the temperature by about 15 percent when compared to the CSIRO test results. However, it should be noted that the general trend of the CFD predicted gas temperature is very good when compared to the measured result.

Figure 8 shows the influence of the boundary conditions on the prediction of the gas temperature. The gas temperature at a location 0.1m below the top of the doorway centre was taken as an example. The dashed line in figure 8 is the gas temperature obtained using a conjugate heat transfer (CHT) boundary condition, and the black line is the gas temperature obtained using an adiabatic wall boundary condition, which excluded the heat conduction into the solid wall. The third line is the CSIRO experiment measured temperature. According to Figure 8, the CFD predicted gas temperature obtained using CHT boundary condition closely followed the measured gas temperature development curve. However, the CFD predicted gas temperature obtained using the adiabatic boundary condition deviated significantly from the measured gas temperature curve. This is because a large portion of the heat generated from the fire source was absorbed by the wall, and the assumption of an adiabatic wall was not applicable in this case. This revealed that the heat deposited into the walls must be taken into account in the CFD modelling, and the adiabatic boundary condition is not applicable in this case.

All the CFD results presented in Figure 9-11 are for case B using a fine mesh with the conjugate heat transfer boundary conditions, and the fine mesh was used.
Figure 9 gives a comparison of the CFD predicted and the measured gas temperatures development history at a location 0.05m below the ceiling centre. During the first 5 minutes, the CFD predicted gas temperature is about 10 percent lower than the measured result, however, during the period 5-15 minutes after ignition, the CFD predicted gas temperature agrees well with the CSIRO measured gas temperature.

Figure 10 presents the CFD predicted and the measured gas temperature at a location 0.1m below the top of the doorway centre. The gas temperature development trend was correctly simulated by CFD with an error of about 10 percent, when compared to the CSIRO test data.

Figure 11 is the comparison of the CFD predicted and the measured gas temperature at a location above the burner and 0.05m below the ceiling. Figure 11 shows that the CFD model under-predicted the gas temperature by about 80 °C during the 5-15 minutes period, the relative error is about 15 percent when compared to the measured result. This under-prediction is similar to case A (Figure 7). If the CSIRO test data is assumed to have an error of less than 10 percent, it can be concluded that in both cases, CFD predicted a relatively more uniform gas temperature. Several parameters could have caused this inconsistency, including the turbulence model or the accuracy of the heat transfer computation, as well as the experimental measurement accuracy. As thermal radiation, heat conduction and convective heat transfer co-exist in these cases, all the sub-models needs to be validated separately for the modelling of the thermal radiation, convective heat transfer and heat conduction, which is the research topic for further investigation.

From the above discussion, it can be concluded that PHOENICS can predict the gas temperature development trend with an error of 10 to 20 percent. As fire scenario is always related to high temperature and strong thermal radiation, both measurement and CFD modelling can have errors.

To investigate and validate a fire model and a software package, more comprehensive research work on more test cases is necessary. Based on the above comparison of temperature data, it can be concluded that PHOENICS can serve as a reliable CFD fire modelling tool, and care is needed to ensure that the fire model is correctly devised and correctly implemented.

CONCLUSION

From the above discussion, following conclusions can be drawn:

1. Reasonable temperature field can be obtained for the modelling of a fire in a test room using the PHOENICS software package.

2. The solid wall should be included into the computation domain as the heat conduction into the wall accounted for a large portion of the total heat transfer, and this can influence the CFD modelling accuracy of the indoor gas temperature development.

3. The k-ε turbulence model is suitable for the modelling of buoyancy-generated turbulence, if the meshing size is sufficient to resolve the subscale turbulence.

ACKNOWLEDGMENTS

The authors thank Alex Webb, Vince Dowling and Neville McArthur at CSIRO Fire Science and Technology Laboratory for providing the test data, without which this modelling would not be possible. We also acknowledge the help from Dr. Dong Chen and other researchers at Fire science group, CSIRO in providing guidance regarding running of PHOENICS.

REFERENCES

1. V. Novozhilov, Computational fluid dynamics modeling of compartment fires, Progress in Energy and Combustion Science V27 (2001), P661-666

2. G. Cox, S. Kumar, Field modeling of fire in forced ventilated enclosures, Combustion Science and Technology, V52, PP7-23, 1987.

3. F.C. Lockwood, W.M. Malalasekera, Fire computation: the "flashover" phenomenon, 22nd International Symposium on combustion/The combustion institute, PP1319-1328, 1988.

4. Kevin B, McGrattan, et al, Fire Dynamics Simulator (FDS) Manual, Technical Reference Guide, 2002Ed.

5. J Ewer, E R Galea, et al, An Intelligent CFD Based Fire Model, Journal of Fire Protection Engineering, V10(1), 1999, pp12-27.

6. Simcox S, Wilkes et al, "Computer Simulation of the Flows of Hot Gases from the Fires at King's Cross Underground Station, Fire Safety Journal, V18, P49-73, 1992

7. D. Barrero, B. Ozell and M. Reggio, On CFD and graphic animation for fire simulation, The Eleventh Annual Conference of the CFD Society of Canada, Vancouver, BC, May 2003

8. Glynn DR, Eckford DC & Pope CW, Smoke concentrations and air temperatures generated by a fire on a train in a tunnel, The PHOENICS Journal of Computational Fluid Dynamics and its Applications, Vol. 9 No. 1, pp 157-168, 1996

9. Robert Siegel, John R. Howell. Thermal radiation heat transfer, 4th ed., New York, 2002.

10. A.K.Webb, Dowling, V.P. and N. A. McArthur "Fire performances of materials", internal CSIRO report by Fire Science & Technology, North Ryde, CSIRO, November 1999.

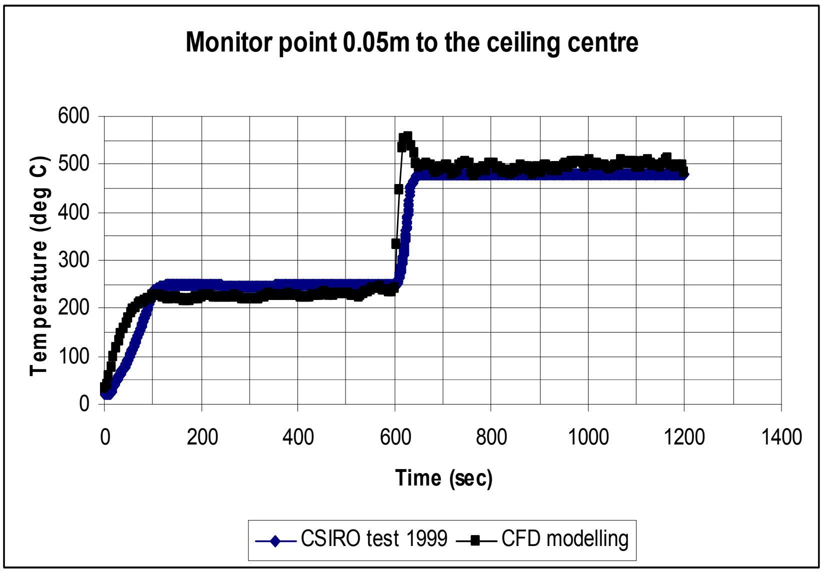

Figure 5: Gas temperature at 0.05m below the ceiling centre for case A

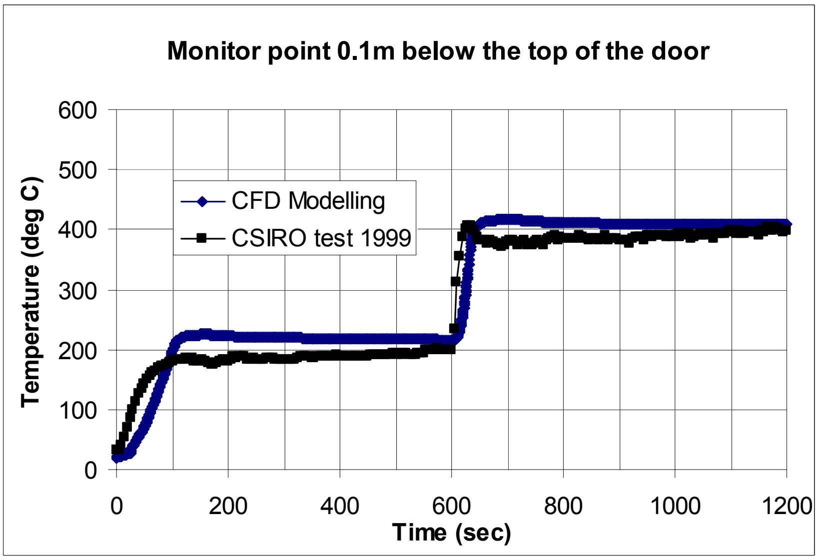

Figure 6: Gas temperature at 0.1m below the top of the door centre for case A

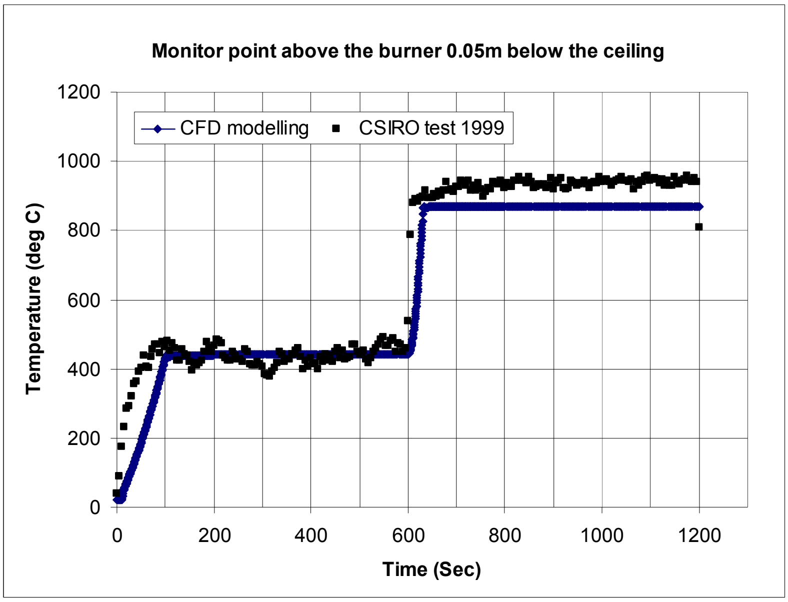

Figure 7: Gas temperature above the burner and 0.05m below the ceiling for case A

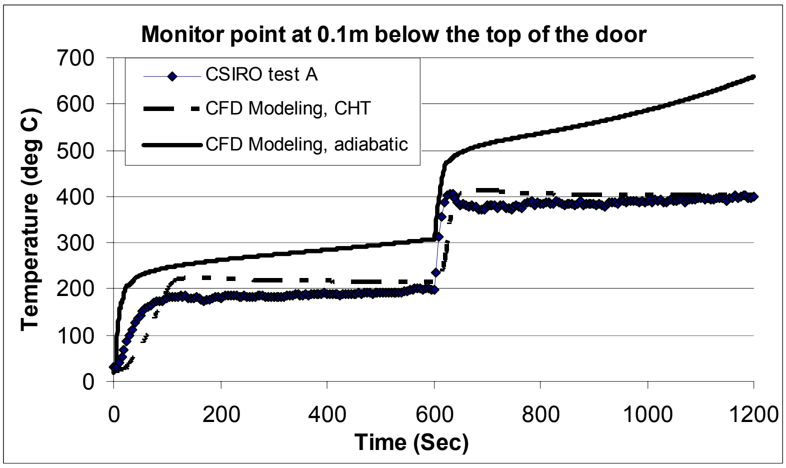

Figure 8: Gas temperature obtained with different boundary condition for case A

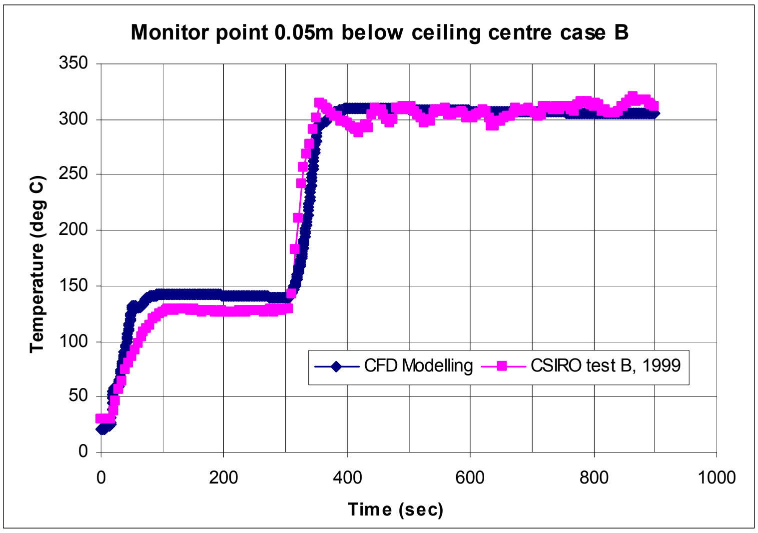

Figure 9: Gas temperature at 0.05m below the ceiling centre for case B

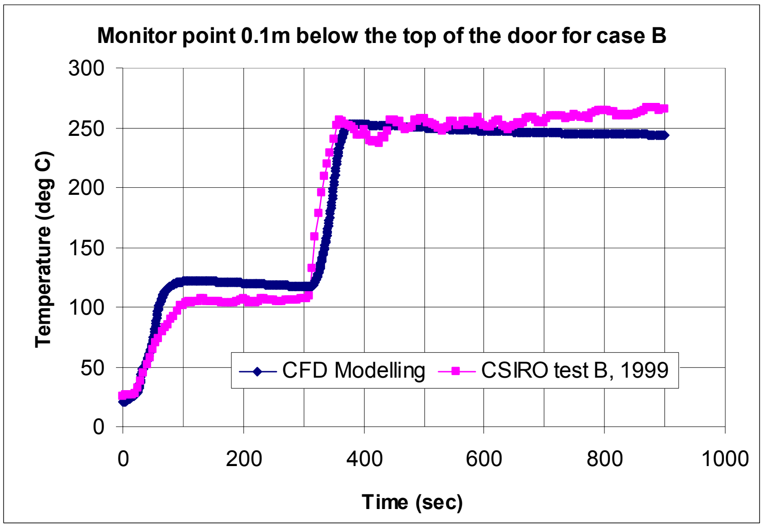

Fig 10: Gas temperature at 0.1m below the top of the door centre for case B

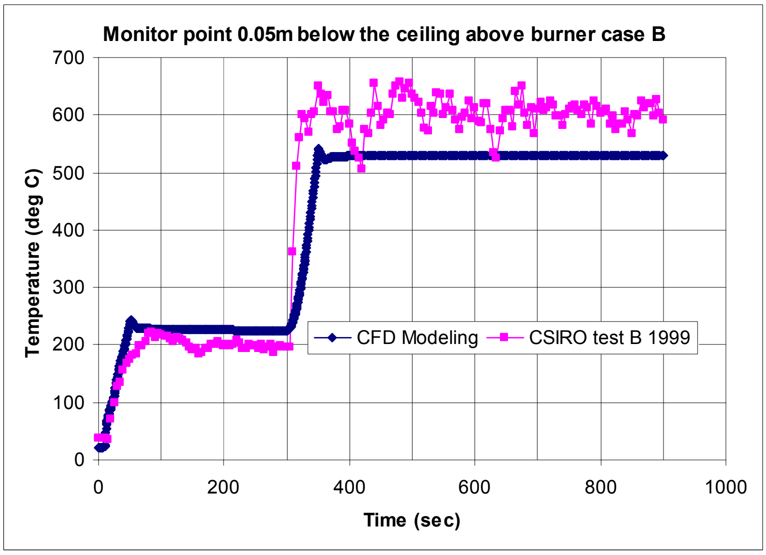

Figure 11: Gas temperature above the burner 0.05m below the ceiling for case B