■ Research Paper ■

Application of Computational Fluid Dynamics for Different Fire Strengths in a Compartment Using Combustion Modelling

Aravind Kumar. A[1], Rajiv Kumar[1] and Shorab Jain[1]
[1] *CSIR-Central Building Research Institute, India*

ABSTRACT

Fire field modelling (Computation Fluid Dynamics) has become more and more attractive as a critical design tool to meet Performance-based fire design on advanced modern buildings. This paper describes the application of Computational Fluid Dynamics (CFD) to predict velocities and temperature distributions induced by a fire in a Steckler's experimental data [1]. The experimental data of different fire loads is taken as case study for present investigation. The experiments of Steckler's compartment fire were conducted to investigate fire-induced flows through the opening in a compartment of size 2.8 m × 2.8 m × 2.18 m (height). The compartment has a doorway opening of 0.74 m × 1.83 m to account the ventilation condition. A porous gas burner is flushed at the floor in the centre of the room with the diameter of 0.3m in the compartment. With the above experimental data, simulation studies were performed with combustion modelling using commercial code of ANSYS CFX-5. The comparison of simulation results of fire field models with experimental domain for different strengths of fire 31.6, 62.9, 105.3 and 158.0 kW is reported. The boundary conditions of the simulation are kept constant, only fire strength is changed to see the performance of the CFD tool. The door centreline temperature, velocities and room corner temperatures are predicted and compared with experimental data as well as with FDS. The results are in good agreement with the experimental data.

Keywords: Fire Modeling, Fire Strength, Model Validation, Computational Fluid Dynamics.

1 INTRODUCTION

Compartment fires have the potential to cause major loss to life and property. The importance of understanding the fire behaviour involved movement of heat and smoke within a burning structure is evident. Various methods of modelling these fires using zone and parametric models have been used over the past few decades. Recently Computational Fluid Dynamics (CFD) is being utilized to model fire scenarios.

Computational Fluid Dynamics (CFD) is now extensively used in building design to help devise suitable mitigation and evacuation strategies. CFD fire simulations can

Coresspondence to: Aravind Kumar. A, Email: aravind991@gmail.com

Received September 16, 2013
Accepted September 5, 2014

provide insight and understanding for hazardous scenarios that may not be practical to investigate experimentally. For applications in the field of safety case analyses, one of the barriers to the practical use of CFD to make important decisions is validation of the model. There is argument on whether CFD can be used to demonstrate a safe design and operation basis, by making intelligent decisions on the known strengths and weaknesses of the mathematical model. Therefore, a procedure to validate the models to predict complex flow is very important, that is, a systematic comparison of model predictions with reliable experimental data. Popular field models [e.g. 2-6] such as PHOENICS, JASMINE and CFX/FLOW3D have been evaluated for simulating building fires.

The mathematical field model JASMINE [2] used for validation of Six-bed hospital ward fire. The transient temperature predictions agree reasonably well with the measurements, especially in the far field. Some discrepancies are, however, evident in the gas concentration profiles. PHOENCIS CFD [3] software used to simulate fire induced flows in domestic-sized rooms. Several scenarios are examined consisting of various fire sizes, fire locations, and door sizes. However, significant smearing is observed in predicted temperature profiles in the vicinity of the confining walls. The CFD studies performed with PHOENICS, CFX and SMARTFIRE [5] to provide the standards/benchmarks is to aid fire safety. The cases had been simulated with identical physics and computational meshes with similar convergence criteria applied. All of the software products tested is capable of generating similar results. These three codes have a similar basic capability and are capable of achieving a similar basic standard. While there are minor differences between the results generated by each of the software products. Only one of the software producers chose to participate in vigorous study, namely SMARTFIRE [6]. In studying the outcome it is clear that by activating sophisticated physical models, the software product tested was capable of generating improved predictions against theoretical and experimental data in all of the cases examined. The detailed guideline has been discussed to choose the fire models [4] in the construction using computer-based models related to fire risk.

The flow rate of air into a compartment fire is a key factor in controlling the burning rate of the fuel and the products of combustion that emerge. To study the fire-induced flow a number of 55 experiments have been performed by Steckler, et al. [1]. The Steckler's experimental data had been extensively used as a benchmark to validate various mathematical/computer fire models [14-16]. To account the door way flow behaviour due to fire in a room, an approximate mass flow formula [14] has been developed with the help of two existing zone models on effect of fire entrainment and vent mixing, a model based on an ideal point source plume fire. A new formula for fire-induced wall vent flow rate is developed with the extension of the above study to include the window mass flows [15]. A thorough examination concerning the difference between the window and doorway flow modes is conducted. Both sill height and width of the windows pose key influence on the formula. An Investigation had been carried out on the accuracy of predictions of fire-induced flow into a compartment by Fire

Dynamics Simulator (FDS) [16]. The predictions of the study are reported with different grid sizes, an inclusion of radiative heat in the combustion model, and an increased computational domain. The input and setup changes (shift of fire location and door way width) made to the FDS simulation to account the prediction of mass flow rates, lower layer temperature, smoke layer height, and neutral plane height. The study allowed significant improvements in the prediction of mass flow rates for all three positions of the fire source. However, there is not much improvement for the remaining three parameters being compared: lower layer temperature, smoke layer height, and neutral plane height [16].

The commercial CFD code ANSYS CFX-5 is used as a simulation tool in this paper. CFD results would be evaluated by comparing with the physical data on room fire by Steckler et al. That set of experiments was commonly used for validating CFD fire models. The computational studies are reported by several authors for the validation of compartment fire using volumetric heat source modelling. Very few studies were reported using combustion modelling using CFD tools of FDS, JASMINE and SOFFIE. The validations study for the different fire strengths is still is a research gap, which has been addressed in this paper.

2 GOVERNING EQUATIONS OF COMPARTMENT FIRE MODELLING

The fire-induced in an enclosure can be described by the equations expressing the conservation of mass, momentum, enthalpy, and species concentrations in literature [7, 8]. A turbulence model, typically a two-equation model [9] or a large-eddy simulation model [10] is required for the closure of the above governing equations for turbulent flow situations. It is important to emphasize that the turbulence model must include the contribution of buoyancy force to the turbulent kinetic energy generation and dissipation. An illustration of the consequence of omitting these terms is given [11]. In the current study, the standard two-equation $k–\varepsilon$ model of turbulence with buoyancy modifications was used because of its wide application in engineering problems and its validation pedigree for fire applications [9, 12]. Standard wall functions were used to estimate the turbulence kinetic energy and dissipation at the cell near the wall. A modified version of the eddy-break-up (EBU) model [13] was used here to account for the effect of the turbulence mixing on the combustion process.

3 EXPERIMENTAL DETAIL

Compartment fire experiments [1] performed in a room size of 2.8 m × 2.8 m × 2.18 m incorporating ceramic fibre board insulation and a circular gas burner, fuelled by commercial grade methane and having a diameter of 0.3 m. The layout of the room is illustrated in *Figure 1*, of the burner position and ventilation configuration reported, the present paper concentrates on the burner is positioned centrally in the room and ventilation is provided by a room opening (doorway) 1.83 m high by 0.74 m wide. The fuel flow rate selected corresponds to a heat output of 31.6, 62.9, 105.3 and 158.0 kW for different set of experiments. Conditions in the opening attracted particular attention and detailed measurements of temperature, using aspirated thermocouples, and velocity by bi-directional probe are reported there.

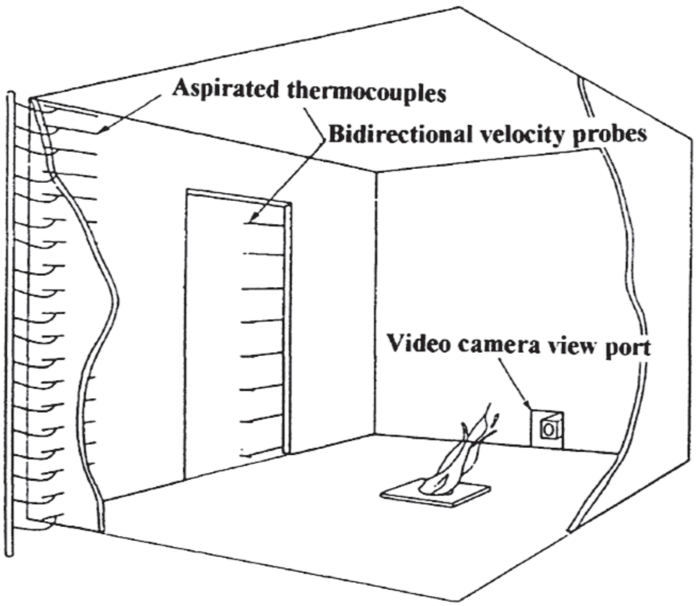


Figure 1 Schematic Room fire Compartment [1]

4 GEOMETRY AND BOUNDARY CONDITIONS

Figure 2 shows the CFD fire model for Steckler's room, which includes the internal room and the external surrounding environment outside the opening. The free pressure boundary conditions are applied on the extended domain boundary, where conditions were assumed to be ambient. For the combustion model, the fuel is released from the fire source at a constant mass flow rate to determine the heat release rate of the fire (or the fire strength). CFD simulations are performed using ANSYS CFX-5. The heated zone is axis symmetrical from the centreline. ANSYS workbench Design Modeler is used to create the 3D geometry of the domain.

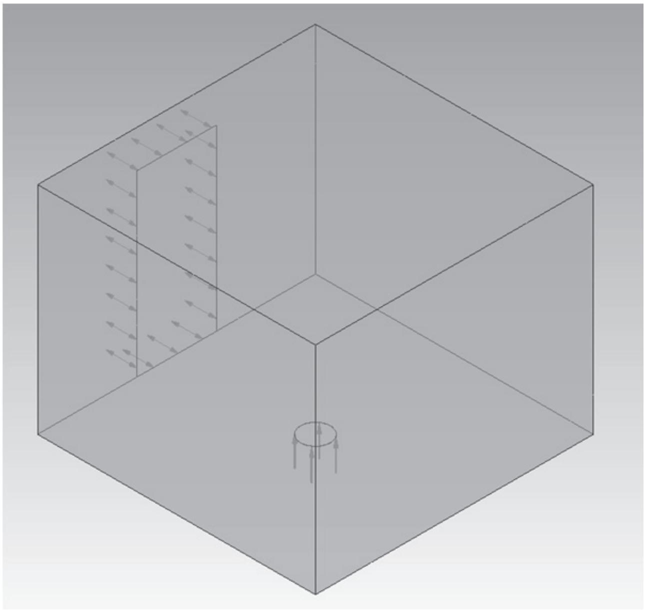


Figure 2 Room domain with burner

The CFD model represents the room geometry with a numerical grid used to discretize the governing equations within the solution domain. Generally speaking, the simulation results are sensitive to the grid when the cell size is relatively coarse but should become independent of the grid when the cell size is fine enough. The meshes were created using the mesh generator provided as part of the ANSYS CFX-5 module package. The details of the meshing used for the study are reported in the *Table 1*. The tetrahedral elements with medium size mesh mapping are selected for the domain. A non-uniform grid, with cell size increasing with distance away from the fire source, was used in order to optimize the total number of grid nodes for the whole computation. *Figure 3* gives view of the mesh domain. The mesh has a maximum face size of 0.11 m. These fining of mesh is done at near the fire source to account for turbulence, velocity effects in order to get more accurate result at the nodes. The details of the geometry and boundary conditions are given in *Table 2*. The differencing scheme utilizes High Resolution with Turbulence equations are solved in first order. The runs were performed on a windows server with Intel(R) Xenon(R) CPU with 12 GB RAM.

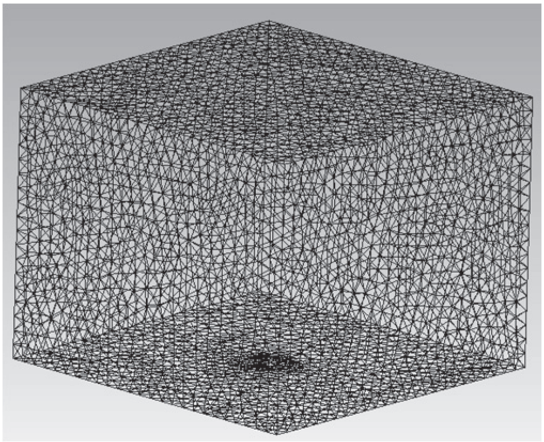


Figure 3 Mesh mapping of the domain

Table 1 Total number of nodes, elements, and faces

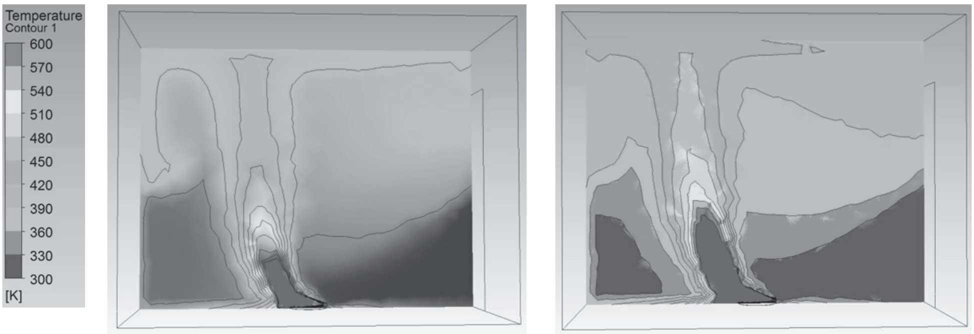
Total number of nodes	12871
Total number of tetrahedrons	66270
Total number of faces	6938

Table 2 Geometry and Boundary Conditions

Details of the Geometry		**Domain Boundary Conditions**	
Domain size	2.8 m × 2.8 m × 2.18 m	Fluid	Air as Ideal Gas
Door way	0.74 m × 1.83 m	Pressure	Atmospheric
Fire size	0.3 m diameter is flushed at floor level	Walls	No slip wall and Adiabatic
Fuel used	Methane	**Opening/Doorway Boundary Conditions**	
Fire Location	Centrally located	Pressure	Relative Zero
Simulation Condition		Temperature	Same as domain
Simulation type	Steady state	Mass Fraction of air	0.23
Turbulence model	k-ε model	Pressure	Relative Zero
Combustion model	Eddy dissipation	**Initialization**	
Buoyancy turbulence	Production and Dissipation	Velocity	Zero
Radiation	p1 grey body	Pressure	Atmospheric
Air density	1.145 kg/m^3	Temperature	Experimental data at 29, 31, 35 and 36 C for 31.6, 62.9, 105.3 and 158.0 kW respectively

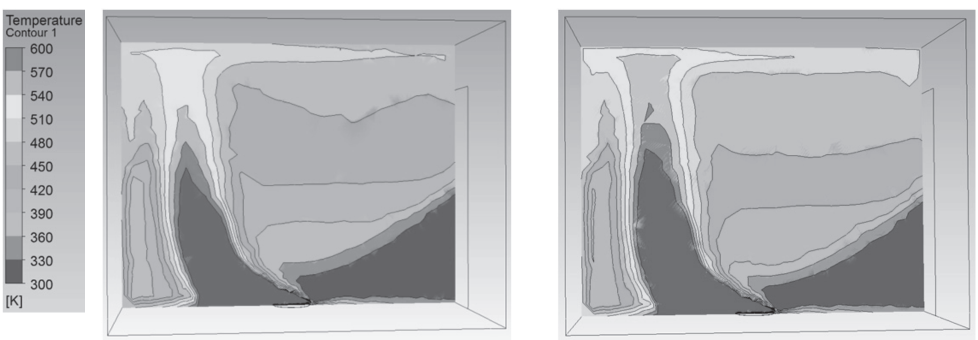
5 RESULTS AND DISCUSSION

The contour section view from the centreline of the room for the predicted fire temperature distribution inside the compartment was shown in *Figure 4 (a-d)*.



a) Fire strength of 31.6 kW

b) Fire strength of 62.9 kW



c) Fire strength of 105.3 kW

d) Fire strength of 158 kW

Figure 4 Temperature contour inside the room at centerline on a plane

Temperature contour measurements are taken at the central plane of the room. The flame height is mostly visible for all fire strengths scenario and it was increasing proportionately. The blue colour zone in the contour will gives the significance of induced flow of the air from the ambient conditions. The temperature contours clearly indicates there are two zone formations taking place in the compartment. The hot zone and cold zone interaction layer at 1 m height from the ground level. These two layers are interacting at approximately 1 m above from the ground. The interaction height is influenced by the ventilation condition of the room. The interface layer is decreasing with respect to fire strength as well as distance from the opening. This is proven fact of the experimental measurements of fire.

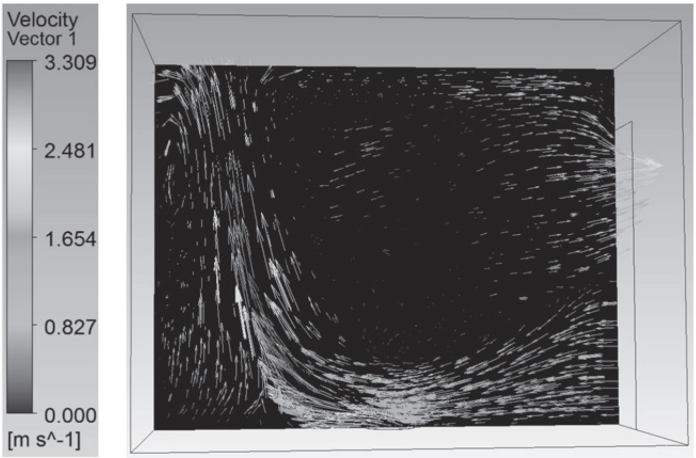
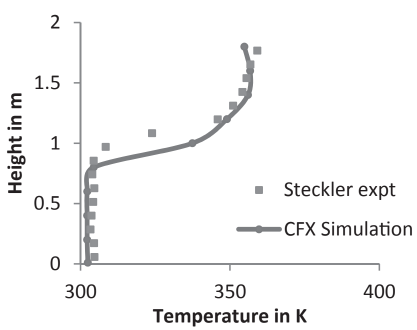
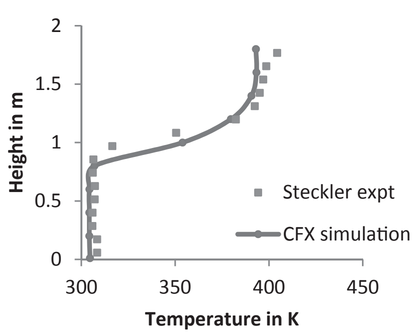


Figure 5 Velocity vector profiles of the compartment fire

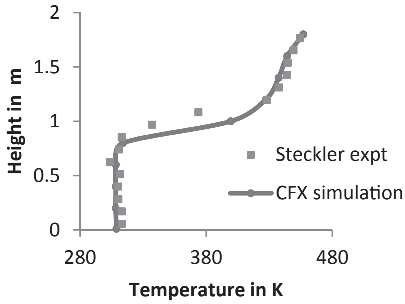
The temperature inside the room varies maximum in the range of 300 to 600 K. This is due to the continuous movement of hot and cold gasses moving through the door way. In the hot zone the gases are moving continuously upward and after striking the wall there is formation of wake near to the wall (opposite to the door). The other side there is ventilation provided through the door way these hot buoyancy gasses are leaving. The boundary conditions of the door is defined as opening where the movement of the air and gases will takes place from low pressure to high pressure. The induced flow of air will takes place to compensate the pressure difference. The velocity vector at room centreline of the induced air and hot gases are shown in *Figure 5* for 158.0 kW fire. Comparisons of the modelling results and experimental results for the natural ventilation tests are shown in *Figure 6* to *Figure 8*.



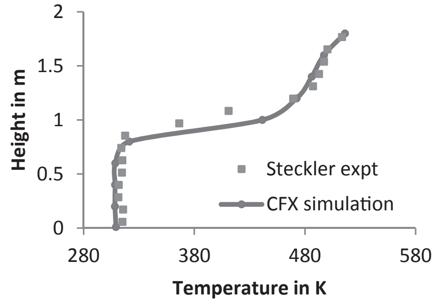
a) 31.6 kW fire



b) 62.9 kW fire



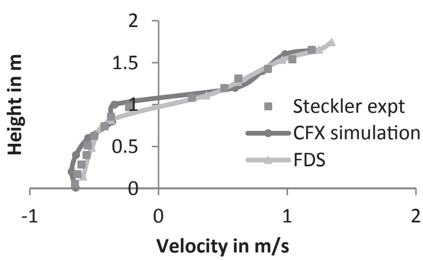
c) 105.3 kW fire



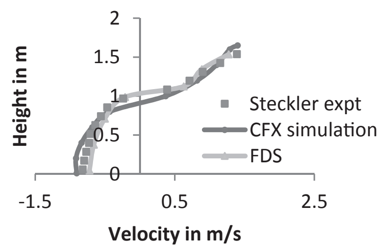
d) 158.0 kW fire

Figure 6 Experimental and Predicted temperature profile at the door centreline

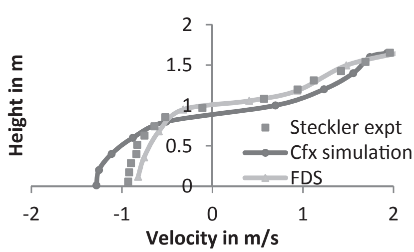
The above curves (*Figure 6: a-d*) indicates that predicted temperature is matching with the experimental data. However there is a slight deviation near to vertically 1m door height at the door centreline. This is likely due to the accuracy of measurement or computation in this sharp transition region. During the experiment, the thermal interface layer is a stably stratified region, where it has periodic stable waves. So the flow is not entirely steady.



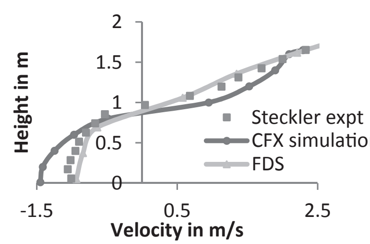
a) 31.6 kW fire



b) 62.9 kW fire



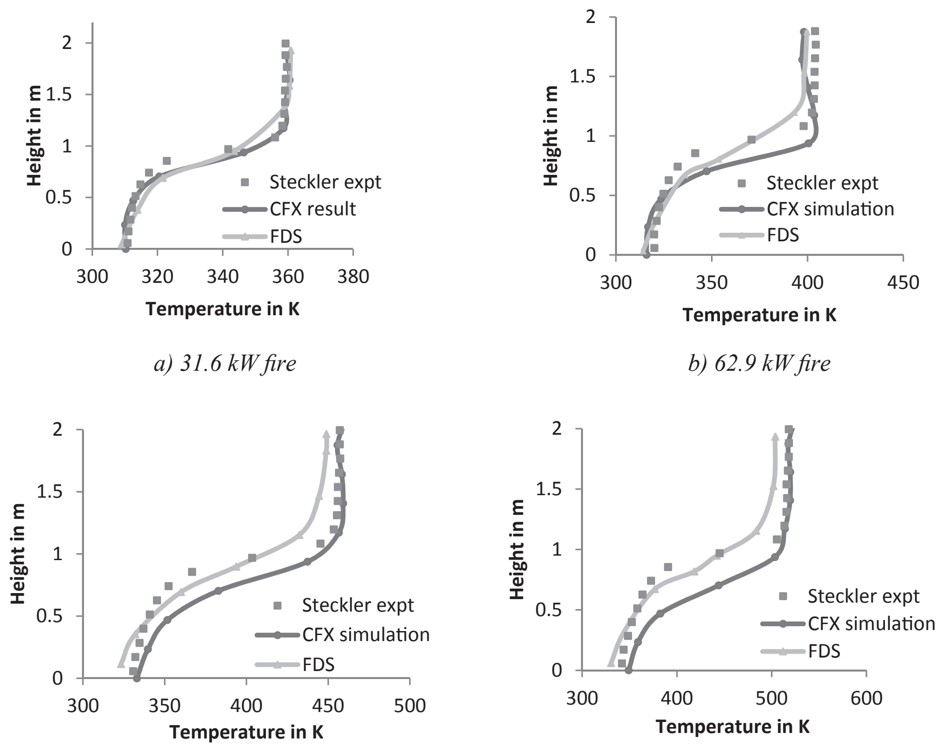
c) 105.3 kW fire



d) 158.0 kW fire

Figure 7 Experimental and Predicted velocity at the door centreline

The predicted velocity profiles at door centreline are compared with the experimental data and also with the FDS predicted results [17] reported in the literature are shown in *Figure 7*. The predicted velocity profile is matching exactly with the experimental data with slight deviation. The sign of the velocity is negative if the direction of air flow is into the compartment and positive it is leaving the compartment. From the figure it indicates nearly at 1 m height at the door centreline the velocity is almost zero due to thermal interface layer movement of hot and cold gases. From the *Figure 7 (c)* and *(d)* it can be seen that the velocity deviation is more at the bottom of the compartment, caused because of induced velocity mass flow predicted by CFX is high compared to the experimental and FDS results. Temperature at the room corners are also measured and compared with the experimental data to account the behaviour of fire inside the room as shown in *Figure 8 (a-d)*.



a) 31.6 kW fire

b) 62.9 kW fire

c) 105.3 kW fire

d) 158.0 kW fire

Figure 8 Experimental and Predicted temperature profile at the room corner

The computational time for the prediction of different fire strengths behaviour is compared with FDS by incorporating identical physics for the model input. The results of computational time are given in *Table 3*. The computational time is more consumed for the 158.0 kW fire compared to 31.6 kW, 62.9 kW and 105.3 kW. The time consumed for the prediction of results by ANSYS CFX-5 relatively less compared to FDS. The computational time is increasing with fire strengths in both the cases is observed.

Table 3 Computational time for the prediction of different fire strengths

Fire strength size	Computational time	
	ANSYS CFX-5	FDS
31.6 kW fire	15 hour 41 min	18 hours 21 min
62.9 kW fire	22 hour 11 min	23 hours 46 min
105.3 kW fire	24 hour 06 min	27 hours 52 min
158.0 kW fire	31 hour 25 min	39 hours 42 min

6 CONCLUSIONS

ANSYS CFX-5 CFD tool results agreed reasonably well with the measured fire data in the room fire validation. The study has shown that the CFD fire model can provide reasonable predictions of the thermal flow fields under fully ventilated conditions. Further studies are needed to be performed for under-ventilated, Air changes per hour (ACPH) conditions. The improved understanding of the fire dynamics and advances made in the numerical methods and computer hardware would have significant impact on the performance of CFD fire models in the future.

7 ACKNOWLEDGMENT

The author is grateful to the Director, CSIR-Central Building Research Institute for his encouragement and kind support.

8 REFERENCES

1. K.D. Steckler, J.G. Quintiere, W.J. Rinkinen, Flow Induced by Fire in a Compartment, 19th Symp. Int. Combustion, 1982, pp. 913-920.
2. S. Kumar, N. Hoffmann, G. Cox, Some validation of JASMINE for fires in hospital wards, Lecture notes in engineering 18, Numerical simulation of fluid flow and heat/mass transfer processes, Springer-Verlag, New York,1986, pp. 159-169.
3. R.N. Mawhinney, E.R. Galea, N. Hoffmann, M.K. Patel, A critical comparison of a PHOENICS based fire field model with experimental compartment fire data, Journal of Fire protection Engineering, 1994, Vol. 6(4), pp. 137-152.
4. A.N. Beard, Fire models and design, Fire Safety Journal,1997, Vol. 28(2), pp. 117-138.
5. A.J. Grandison, E.R. Galea, M.K. Patel, Fire modelling standards/benchmark report on phase 1 simulations, Paper no. 01/IM/72, CMS Press, 2001.
6. A.J. Grandison, E.R. Galea, M.K. Patel, Fire modelling standards/benchmark report on phase 2 simulations, Paper no. 01/IM/81, CMS Press, 2001.
7. S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemishpere Publishing Co., New York, 1980.

8. B.E. Launder, D.B. Spalding, The Numerical Computation of Turbulent Flows, Computer Methods in Applied Mechanics and Engineering, 1974, Vol. 3(1), pp. 269-289.
9. G. Cox, Turbulent Closure and the Modelling of Fire using CFD, Phil. Trans. Roy. Soc. London, 1998, pp. 2835-2854.
10. K.B. McGrattan, H.R. Baum, R. G. Rehm, Large Eddy Simulation of Smoke Movement, Fire Safety J., 1998, Vol. 30(2), pp. 161-178.
11. N.C. Markatos, M.R. Malin, G. Cox, Mathematical Modelling of Buoyancy Induced Smoke Flow in Enclosures, Int. J. Heat Mass Transfer, 1982, Vol. 25(1), pp. 63-75.
12. S. Kumar, (1983). Mathematical Modelling of Natural Convection in Fire - A State of the Art Review of the Field Modelling of Variable Density Turbulent Flow, Fire Materials,1983, Vol. 7, pp. 1-14.
13. B.F. Magnussen, B.H. Hjertager, On Mathematical Models of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, In: Proceedings of Sixteenth Int. Symp. Combustion, Cambridge, MA, 1976, pp. 719-729.
14. J. G. Quintiere, Lei Wang, A General Formula for the Prediction of Vent Flows, Fire Safety Journal, 2009, Vol. 44(5), pp. 789-792.
15. Lei Wang, James G. Quintiere, An Analysis of Compartment Fire Doorway Flows, Fire Safety Journal, 2009, Vol. 44(5), pp. 718-731.
16. L. Wang, J. Lim, J.G. Quintiere, On the Prediction of Fire-Induced Vent Flows Using FDS, Journal of Fire Sciences, 2012, Vol. 30 (2), pp. 110-121.
17. K. McGrattan, S. Hostikka, J. Floyd, R. McDermott, Fire Dynamics Simulator (Version 5) Technical Reference Guide, Volume 3: Validation, NIST Special Publication 1018-5, 2010.