Proceedings of the 12th International Fire Science & Engineering Conference, Interflam 2010, 5-7th July 2010, University of Nottingham, UK, **Volume 1, pp. 913-924. ISBN 978 0 9541216-5-5, 2010.**

SIMULATION OF THE FLOW INDUCED BY POSITIVE PRESSURE VENTILATION FAN UNDER WIND DRIVEN CONDITIONS

Arun Mahalingam, Mayur K. Patel, and Edwin R. Galea
Fire Safety Engineering Group, University of Greenwich, London SE10 9LS

ABSTRACT

Positive Pressure Ventilation (PPV) is a tactical forced ventilation technique used by many fire departments to remove smoke, heat and contaminants from a burning building. PPV usage as a post fire strategy is generally proven and widely accepted as an effective ventilation tool. However, its success as a pre-attack strategy in controlling the spread of fire and smoke is only ensured when it is used correctly and with caution. The efficiency of PPV depends mainly upon net air flow rate through the fire structure. The amount of fresh air blown into the building during a PPV attack is affected by various factors such as fan capacity, distance between the fan and inlet door, inlet dimensions, exhaust opening area, wind and fire conditions etc. Computational Fluid Dynamics (CFD) is a useful and cost-effective tool in improving our understanding on various factors affecting the effectiveness of PPV and could be used to improve both the fire fighter and fire victim's safety. In the present investigation, the SMARTFIRE CFD fire field model is validated using two full-scale experiments characterising a PPV fan. This work is extended by investigating the relationship between the exhaust/inlet area ratio and the net air flow rate into the room geometry under wind and no wind conditions. Finally, results from the simulations of a complex multi-storey structure involving wind driven fire and PPV fan are also presented. Suggestions have been made on the choice of vent locations for better fan performance.

INTRODUCTION

Positive Pressure Ventilation (PPV) is accomplished by using high powered fans outside the structure, typically close to a door opening, to force smoke and heat out of the building through an existing or a newly created vent. PPV fans can aid fire fighters to locate the seat of fire and create a more tenable environment for the fire fighters to tackle the fire. Svensson[1] through various tests demonstrated that PPV increases the temperature in rooms on the leeward side of the fire (region between fire and exhaust vent) and reduces temperatures on the windward side of the fire (between the fire and fan). PPV tactics can also be used to prevent smoke entering into a specific structural area. In a high-rise building, it is possible to prevent smoke infiltration into the stairwell if the fans are properly selected and positioned[2]. As a standard operating procedure during ground operations, an existing vent is used or an exhaust vent is created prior to starting the fan. Experimental investigations on the effect of exhaust area on the efficiency of PPV fans by Ezekoye et al.[3] suggests that the exhaust flow rate increases with exit vent area.

Effectiveness of PPV tactics depend upon various factors such as building geometry, vents size, vent location, fan size, fan offset distance, wind and fire conditions etc. Hence the flow induced during a PPV attack is a complex flow phenomenon. Using Computational Fluid Dynamics (CFD) modelling tools, it possible to model fire phenomena under PPV attack from first principles, via solution of the basic conservation equations. In the present investigation, the SMARTFIRE[4-6] CFD field modelling tool is validated using two full-scale experiments characterising a PPV fan. Further the effect of exhaust vent area on the effectiveness of PPV fans under wind driven conditions has also been investigated for a single room geometry. Under external wind conditions, the failure of a window in the fire apartment combined with open doors to public corridors and stairs can create extremely hazardous fire conditions in the corridor and stairwell that are untenable even for a fire fighter in full protective gear[7]. The ability of a PPV fan in preventing smoke spread outside the fire compartment and into the stairwell under external wind conditions is demonstrated numerically in this paper using a three-storey building.

VALIDATION OF FULL-SCALE PPV FAN FLOW SIMULATIONS

Full-scale experiments were conducted at NIST's BFRL Large Fire Facility to characterise a PPV fan, in terms of velocity[8]. The experiments were performed in an open atmosphere and in a simple room scenario. The SMARTFIRE[4-6] fire field modelling tool is used in the present study to simulate the detailed jet flow and fire environment induced by PPV fans. In the CFD modelling methodology, three-dimensional transport equations based on conservation of mass, momentum, species and enthalpy are solved iteratively. The governing transport equations for all fluid variables can be expressed in a general form

$$\frac{\partial}{\partial t}(\rho\phi_i)+div(\rho u\phi_i)=div(\Gamma_i\, grad(\phi_i))+S_{\phi_i} \qquad (1)$$

Where S_{ϕ_i} is the source term. The dependent variable ϕ_i may represent any of the following: three velocity components (u, v, w), pressure (p), enthalpy (h), turbulent kinetic energy (k), dissipation rate of turbulent kinetic energy (ε) or mass fraction of a species. The code uses the SIMPLE[9] algorithm and models turbulence using the two-equation k-ε model with buoyancy modifications[10,11]. Eddy dissipation model proposed by Magnussen and Hjertager[12] is used to model gaseous phase combustion. Finally, radiation is modelled using an enhanced six-flux radiation model[13].

Flow field induced by a stand-alone PPV fan

The first series of experiments were conducted inside the BFRL's Large Fire Facility to determine the flow field created by PPV fans. The facility has the interior dimensions of 36.6 m x 18.3 m x 7.6 m. An 18-inch, electric PPV fan mounted on a stand setting the centre of the fan to a height of 1.28 m from the ground was used. The fan had a depth of 0.48 m, width of 0.62 m and height of 0.62 m. It had a power rating of 1 hp, a speed rating of 2200 rpm and a flow rating of 6.64 m^3/s. The velocity field was mapped on a vertical plane using a wooden frame grid of dimension 2.44 m x 2.44 m. The velocities were measured using a digital anemometer. These measurements were recorded with the grid frame located at 1.83 m, 2.44 m, and 3.05 m away from the fan inlet. A detailed description of the experimental procedure can be found in NISTIR 7065[8].

The computational domain used to simulate this test case is depicted in Figure 1. A domain size was chosen such that the computational boundary walls do not impact on the jet profile. A uniform velocity of 17.89 m/s was imposed at the fan outlet based on the maximum speed of the PPV fan[8]. The depth of the fan shroud was taken to be 0.25 m. The jet flow produced by the fan is a swirling flow with high turbulent intensity. To model this type of flow correctly requires the k-ε turbulence model to be modified. The standard k-ε turbulence model contains five adjustable constants[14] . A 15% change to the default values of the constants $C_{1\varepsilon}$ and $C_{2\varepsilon}$ resulted in the best agreement with the measured velocity field. In Figures 2 and 3 the horizontal velocities are compared between experiment and computational data on vertical planes at 2.44 and 3.05 m from the fan outlet respectively. It should be noted that the experimental flow field displays quite marked asymmetric behaviour, especially in the outer ring of measured values (radius of 0.948 m). It is not clear if these asymmetric experimental values are a result of measurement errors, physical asymmetries in the experimental or equipment setup which have not been reported or represent genuine asymmetric behaviours. The numerical simulations did not generate such an asymmetric flow field. With the exception of these asymmetries, the computed velocities of the jet flow using the standard k-ε turbulence model, with the modified constants, are in reasonable agreement with the experimental data except at several locations. At 3.05 m from the fan, predicted centre line velocities are within 1.0% of the measured velocities, predicted velocities within a radius of 0.424 m of the centre line are within 6.5% and within a radius of 0.848 m predicted values are within 25.5% of the measured values. At greater radii, the asymmetries in the measured flow speed result in quite large percentage differences in places.

Figure 1 Computational domain for the simulation of PPV fan 'free jet'

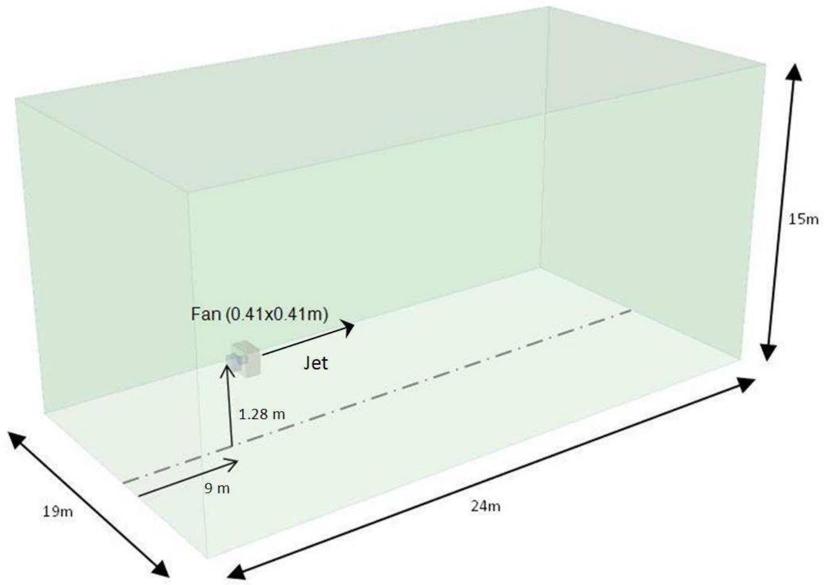
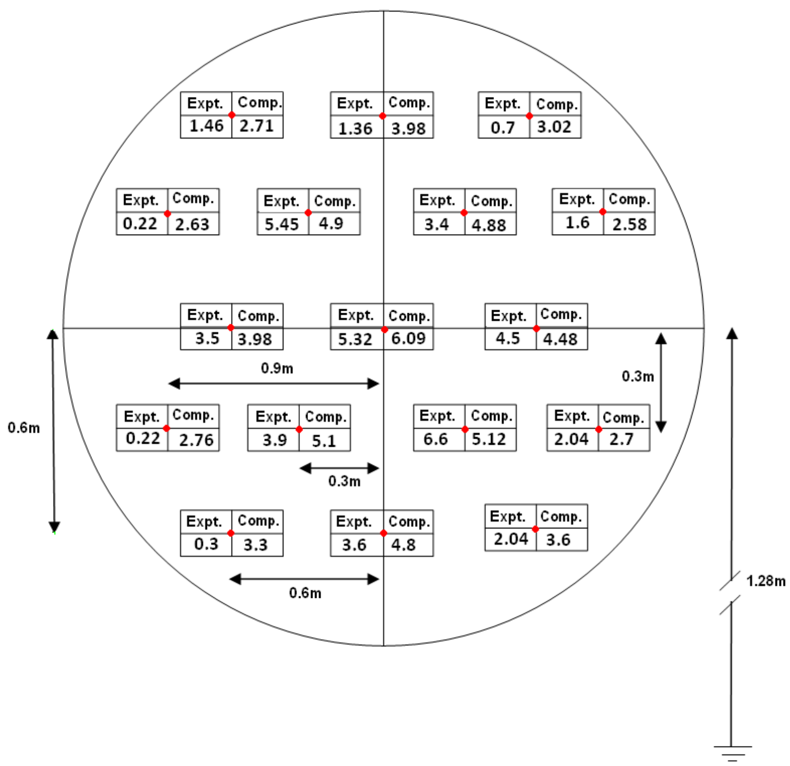
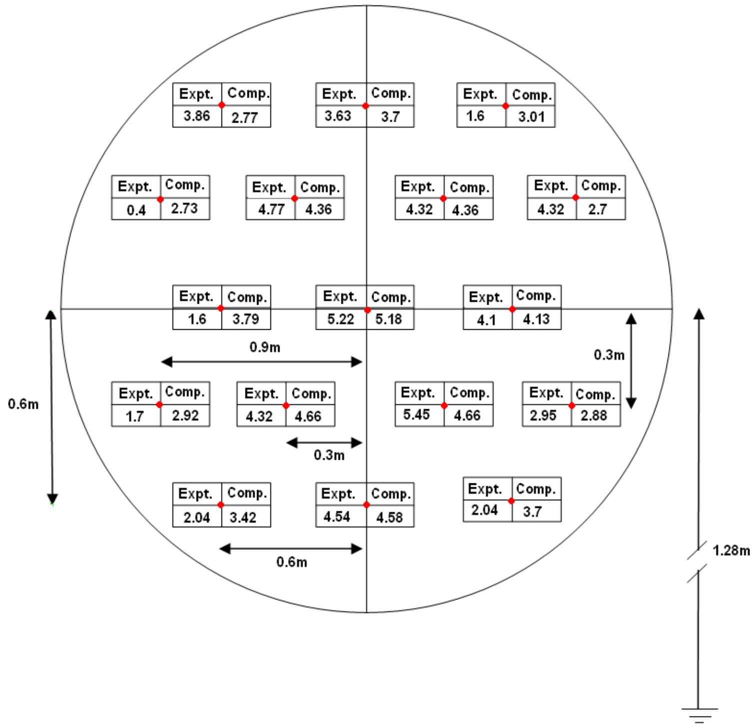


Figure 2 Horizontal velocity comparisons on a vertical plane at 2.44 m from fan outlet



Proceedings of the 12th International Fire Science & Engineering Conference, Interflam 2010, 5-7th July 2010, University of Nottingham, UK, **Volume 1, pp. 913-924. ISBN 978 0 9541216-5-5, 2010.**

Figure 3 Horizontal velocity comparisons on a vertical plane at 3.05 m from fan outlet



Single Room Experiment

This experiment[8] was also conducted at the NIST BFRL Large Fire Facility. The objective was to determine the impact of PPV fans in a simple room geometry. The geometric details of the single room experiment are shown in Figure 4. The room was constructed on a 0.2 m high plywood decking base and had a ceiling at a height of 2.6 m measured from the top of the base. The window on the left hand side of the room was 0.91 m x 1.37 m in dimension with the sill at 0.46 above the ground. The door located at the centre of the front wall was 0.91 m wide and 2 m tall. The ceiling was extended 1.8 m from the front of the room. The fan and anemometer used in this experiment was the same as the previous experiment. The fan was positioned in the centre of the doorway and at a distance 3 m from the front wall. The fan was running at a rated maximum speed of 2200 rpm with the maximum velocity of 17.89 m/s. Figure 5 presents the computational horizontal velocities on the door and window plane along with the experimental velocities. Similar to the stand-alone fan experiments, the single room experimental flow field also displays some marked asymmetric behaviour, especially at the corner locations of the doorway and window. The numerical simulations did not generate such an asymmetric flow field. With the exception of these asymmetries, the computed velocities of the jet flow using the standard k-ε turbulence model, with the modified constants, are in reasonable agreement with the experimental data except at the corner locations. At the doorway plane, the computed velocities are within 47% of the experimental values whereas the predicted velocities are within 16% of the measured values at the window plane.

Effect of vent area on the flow rate through the doorway

Using the standard k-ε turbulence model with the modified constants, SMARTFIRE was used to study the effect of exit vent area on the air flow rate through the inlet doorway. Ezekoye et al[3] performed several PPV experiments on a three-storey brick building to investigate the effects of fan offset distance, size and number of exhaust vents, and volume of building on vent flow rates. We find, similar to the observations of Ezekoye et al[3], that the air inflow increases with exhaust vent area under no external wind, but only up to a critical area ratio (ratio of exhaust area (A2) to the inlet area (A1)) and then reaches a maximum flow limit. Figure 6 shows the SMARTFIRE simulated doorway flow rates for various area ratios. As the exhaust vent area increases relative to the doorway size, the pressure (flow) loss of the system (room) decreases and hence the mass flow rate into the doorway increases.

Figure 4 Geometric details for the single room experiment

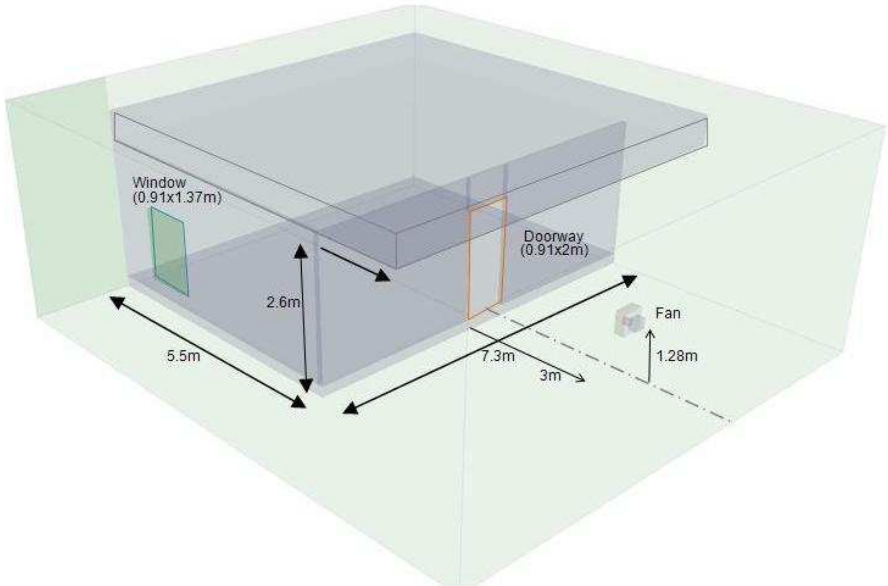


Figure 5 Horizontal velocities (m/s) compared at locations (✚) along door and window planes

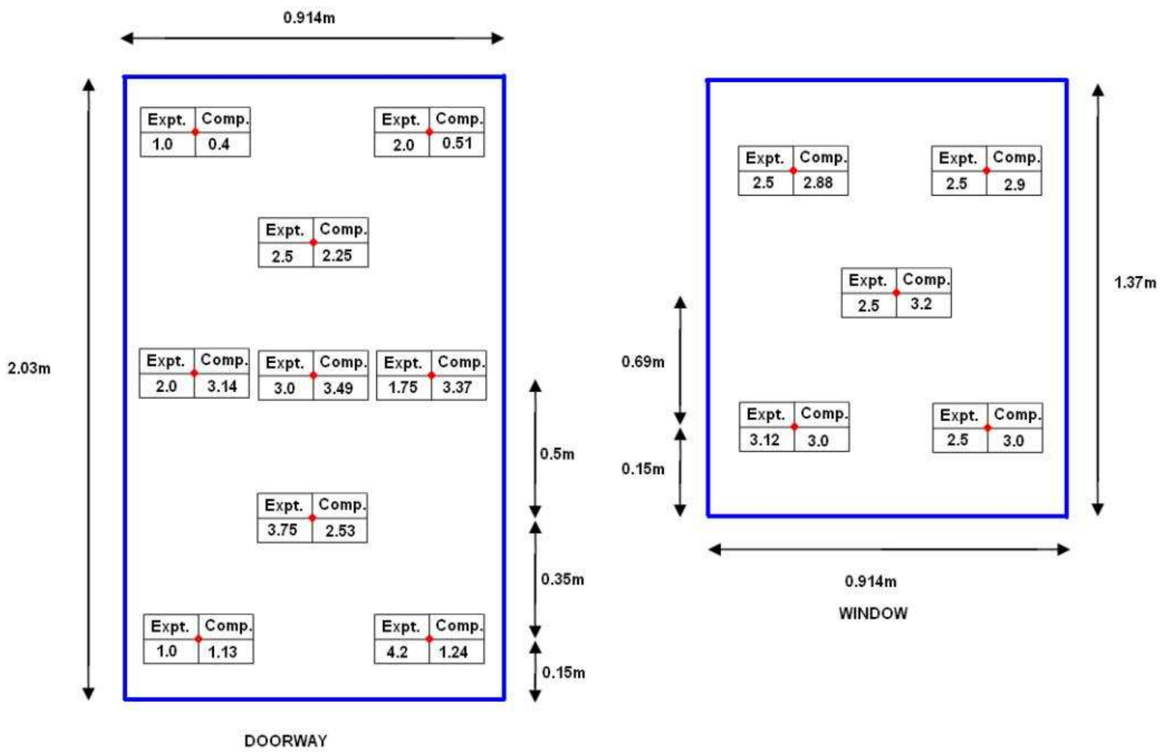
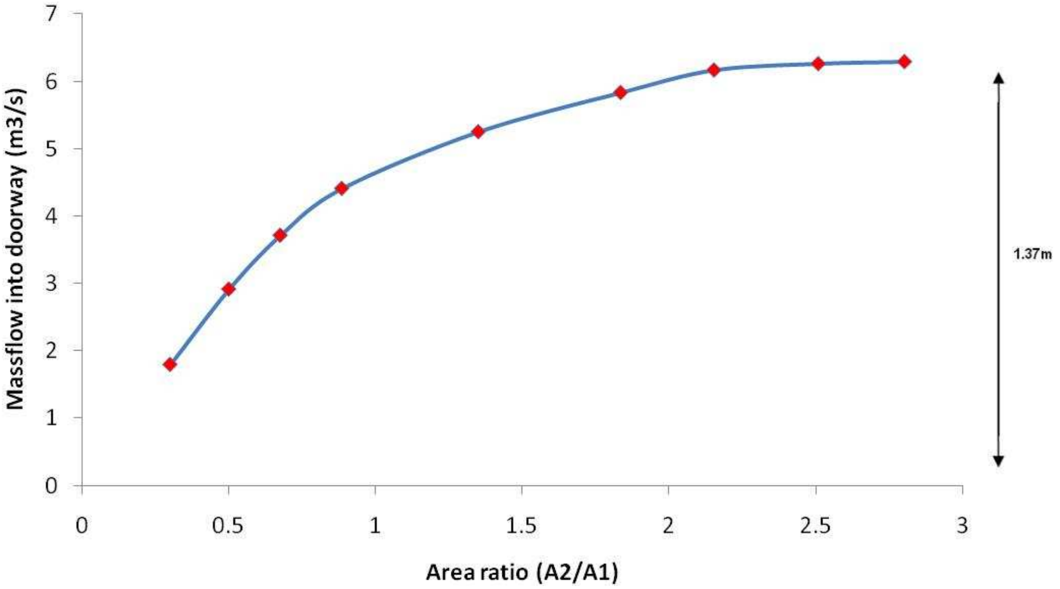


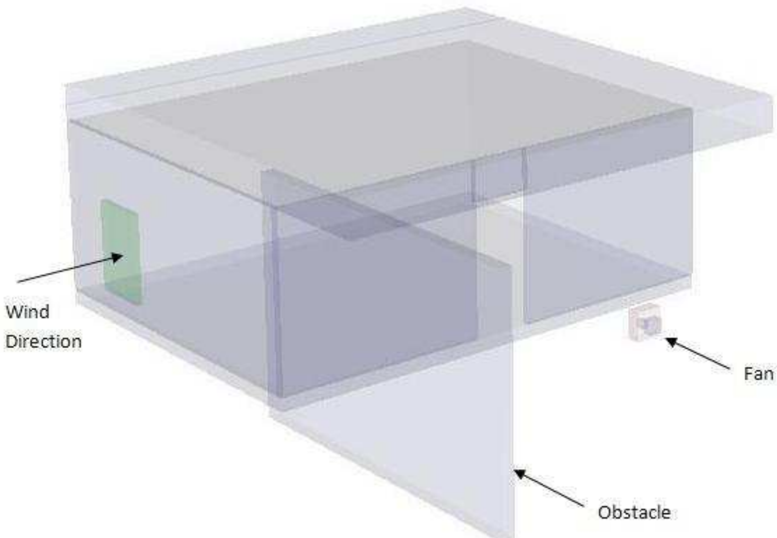
Figure 6 Exhaust flow rate for various window area (A1= Doorway area, 1.86m^2)



Effectiveness of PPV fans under wind driven conditions

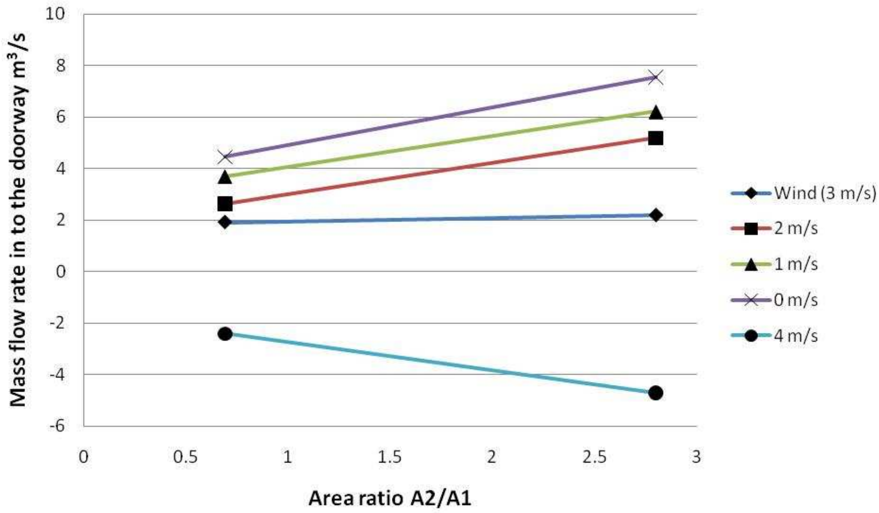
External wind can significantly affect the effectiveness of PPV fans. After validating the SMARTFIRE model for the simulation of flow due to a PPV fan on a simple room geometry in the previous section, simulations were carried out with the same geometry at various external wind speeds. The wind was assumed to be blowing perpendicular to the window plane. Since the wind might affect the PPV fan jet externally, a thin partition (obstacle) extending from the left wall of the room was placed in order to block the fan from interacting with the imposed external wind. The modified single room geometry is shown in Figure 7.

Figure 7 Modified single room geometry for wind driven simulations



Presented in Figure 8 is the net mass flow rate through the doorway at various wind speeds with the wind direction being perpendicular to the exhaust vent plane. The wind driven simulations were carried out for wind speeds ranging from 1 m/s to 4 m/s. For any particular wind speed, the flow rate through the room increased by approximately 100 % as the area ratio (A2/A1) increased to 2.7 with the exception of the 3 m/s wind case. This may be due to the fact that 3 m/s case was closer to the 'critical wind speed' of 3.3 m/s. The 'critical wind speed' is the computed average velocity at the doorway under no wind condition. Above the critical wind speed, the flow through the room reverses, as the external wind overpowers the PPV fan pressure created at the inlet doorway. The effect of area ratio (A2/A1) reverses above the critical wind speed as the exhaust vent becomes the inlet for wind speed above 3.3m/s. For the 4 m/s case, bigger window area reduces the pressure loss for the external wind and hence the reverse flow through the room is increased.

Figure 8 Mass flow rate in to the doorway at various external wind speeds



INVESTIGATION OF PPV TACTICS IN A REAL-SCALE THREE-STORY FIRE FIGHTER TRAINING BUILDING

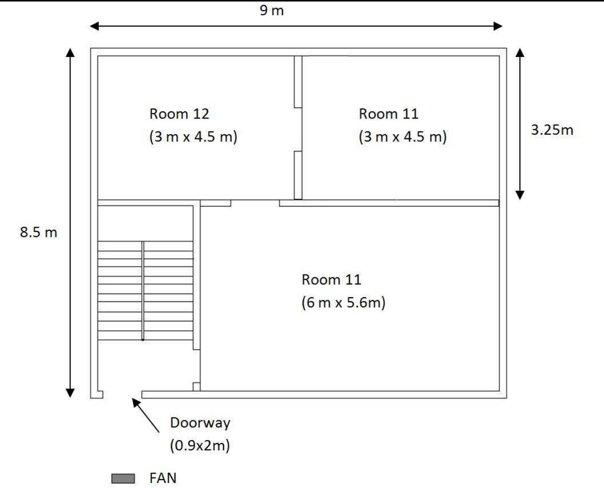
Real-scale fire simulations were carried out in a structure similar to the NIST three-story fire fighter training building[15] to investigate the effect of Positive Pressure Ventilation (PPV) on the fire environment. The building was constructed using concrete blocks and floors. Kerber and Walton (2006)[15] conducted a series of full-scale experiments in the Fire Fighter Training Building to compare the natural ventilation with the Positive Pressure Ventilation. After evaluating various ventilation scenarios, Kerber and Walton[15] suggested that PPV fans can assist in making the building environment more conducive for firefighting operations.

In the present numerical investigation a fire was considered in a corner room of the first floor as depicted in Figure 9. The simulations focused on the first floor and the three-storey stairwell. The ground and the second floor were isolated by closed doors. The layout plans of the ground, first and second floors are shown in Figure 9a-c. The ceiling height of the rooms were 3.4 m. The ceiling and floor slabs were 0.2 m thick each. The fire source had a constant heat release rate of 1.5 kW/s and a smoke generation rate of 0.4 kg/s. The PPV fan used in the simulations had a specification and rating similar to that used in the previous sections. For the PPV simulations cases, the fan was located at 2.4 m from the front door of the ground floor. The stairwell was 2.4 m in width, 5.6 m in breadth, and 10.8 m in height.

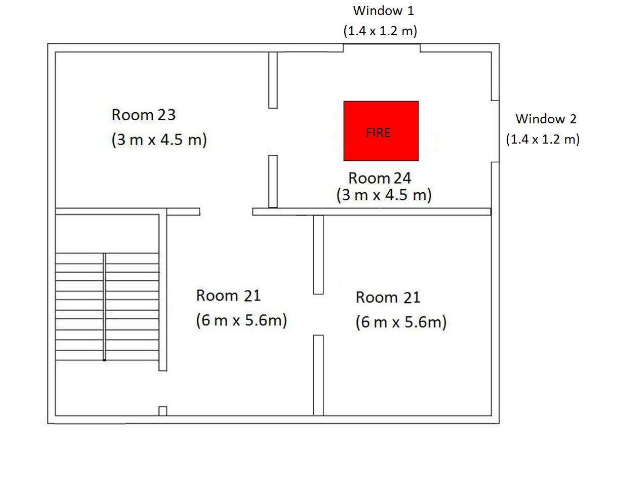
Comparison between Natural Ventilation and Positive Pressure Ventilation

Simulations were caried out for the building configuration shown in figure 9 with and without PPV fans. The external wind speed was 2 m/s in both cases and blowing in the direction perpendicular to Window 1. Both windows 1 and 2 were kept open. Figures 10-13 shows the computed temperatures and smoke distribution in the stairwell and inside the first floor apartment at steady-state. The simulations were run in transient mode and reached a steady-state solution at 350 seconds. During the natural ventilation scenario, the smoke filled all the rooms on the first floor apartment and entered the stairwell through the main entrance of the first floor apartment. The stairwell area from the floor level of the first floor to the top of stairwell was filled with smoke. This situation impedes fire service rescue and fire fighting operations. Furthermore, the fire environment in the natural ventilation case is untenable as the temperature reaches 160^0 C in the stairwell. Conversly, under PPV attack the smoke was contained within the room of fire origin, keeping the stairwell free of smoke. In the given PPV scenario, the fan was able to overcome wind and pressurise the stairwell and first floor apartment and eventually, clear them of heat and smoke.

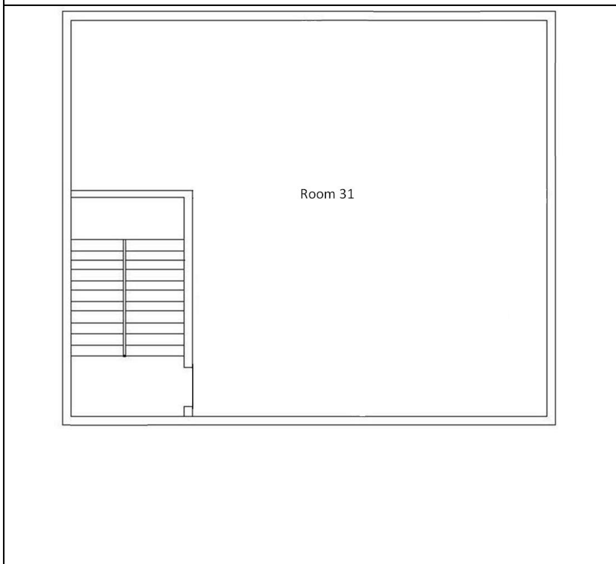
Figure 9 Three-story Fire Fighter Training Building



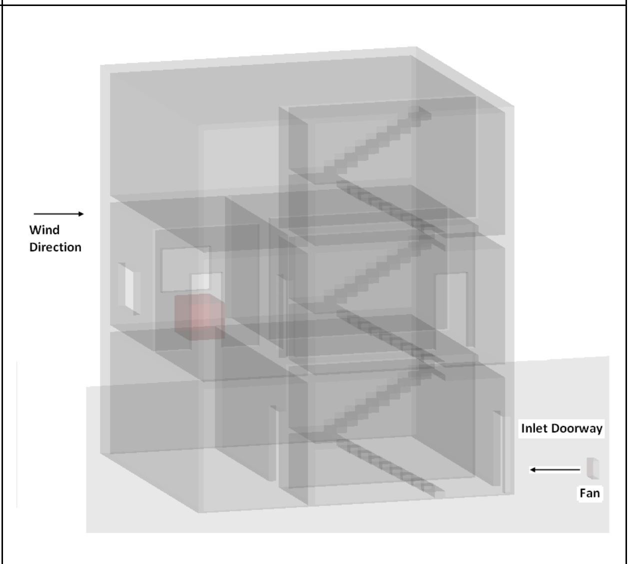
(a) Ground floor layout



(b) First floor layout

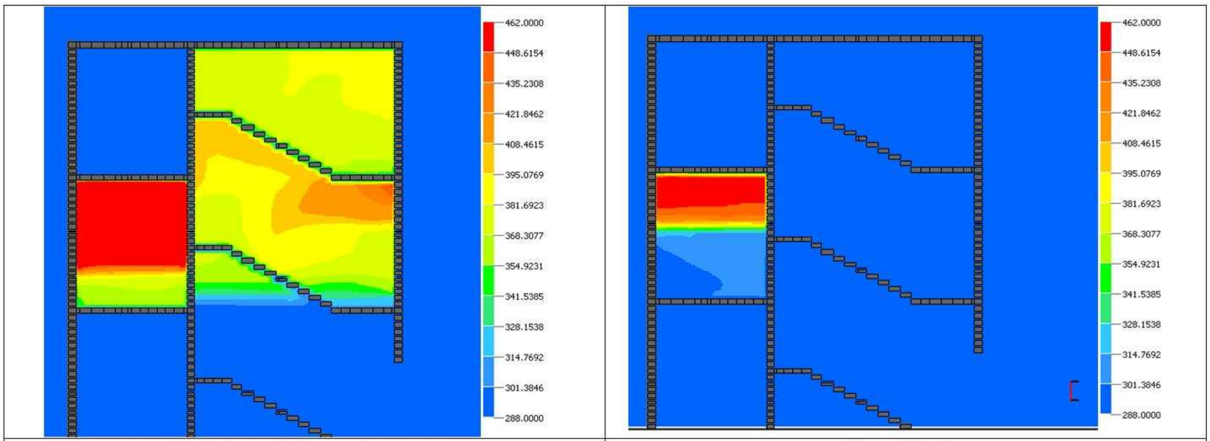


(c) Second floor layout



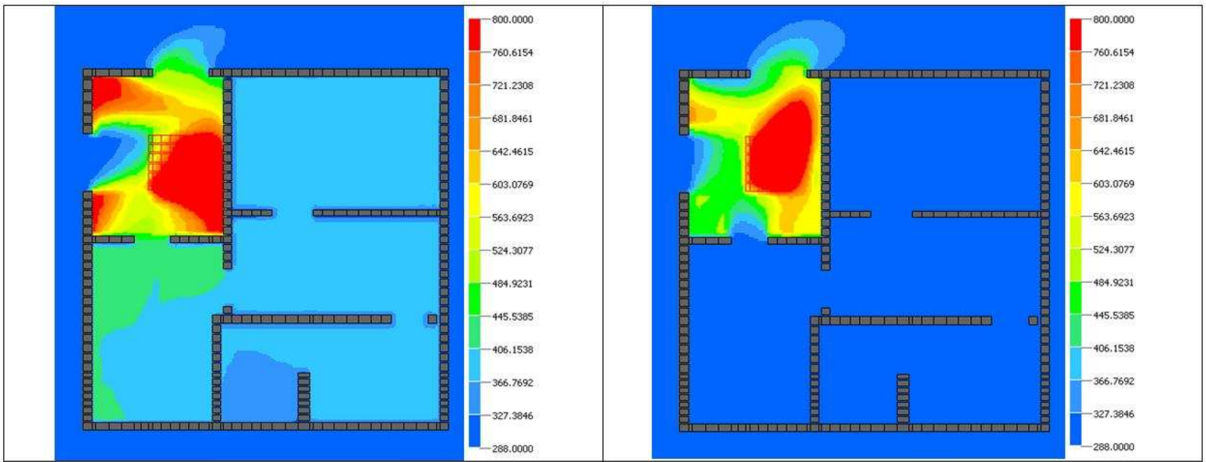
(d) 3-D view of the building

Figure 10 Temperatures (K) along stairwell vertical plane (350 sec)



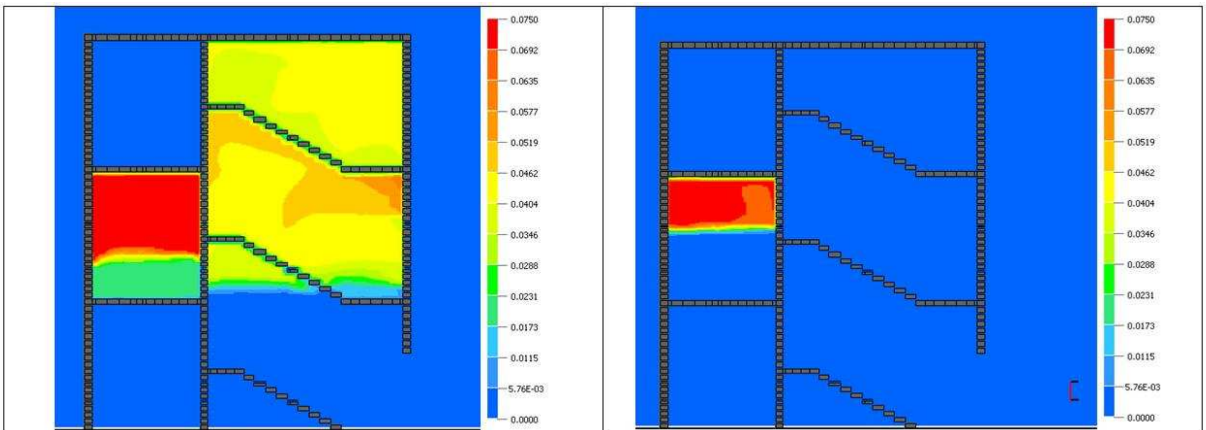
(a) without fan | (b) with fan

Figure 11 Temperatures (K) across the first floor layout at 1.22 m height (350 sec)



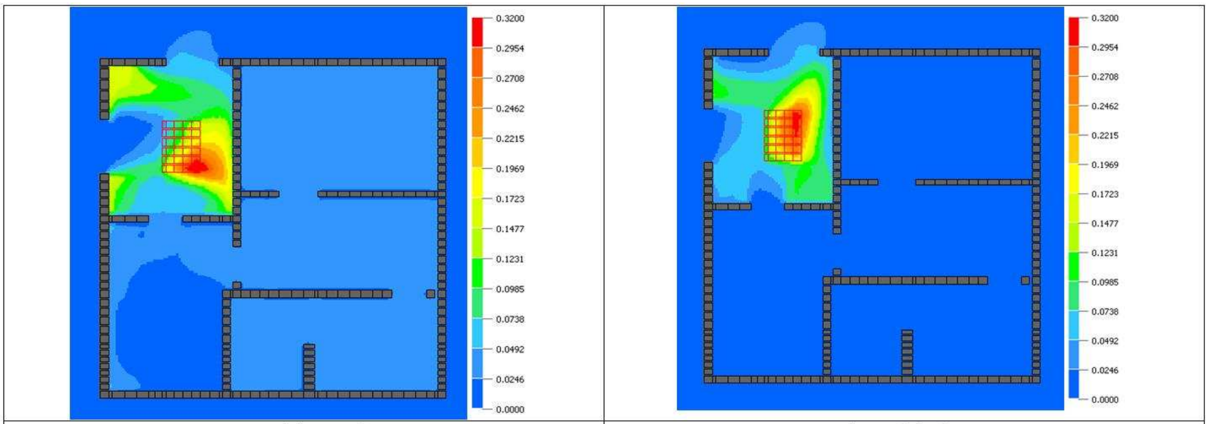
(a) without fan | (b) with fan

Figure 12 Smoke mass fraction along stairwell vertical plane (350 sec)



(a) without fan | (b) with fan

Figure 13 Smoke mass fraction across the first floor layout at 1.22 m height (350 sec)

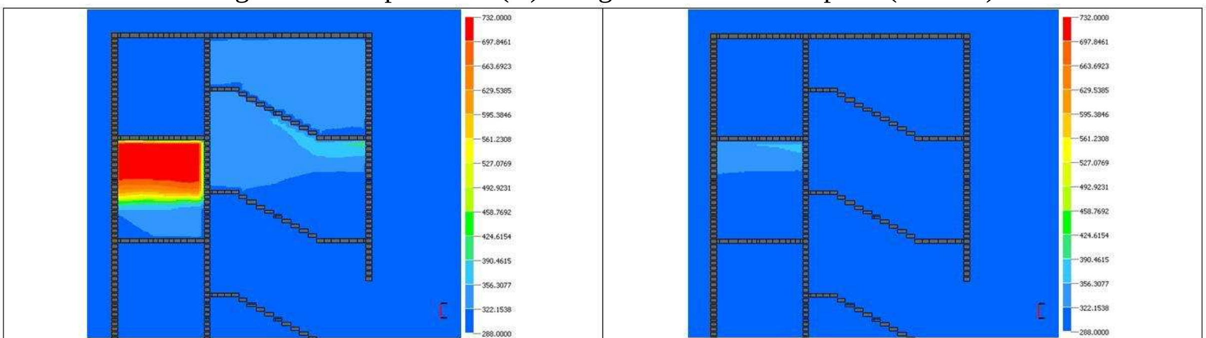


(a) without fan | (b) with fan

Simulation of three-storey building with one window open

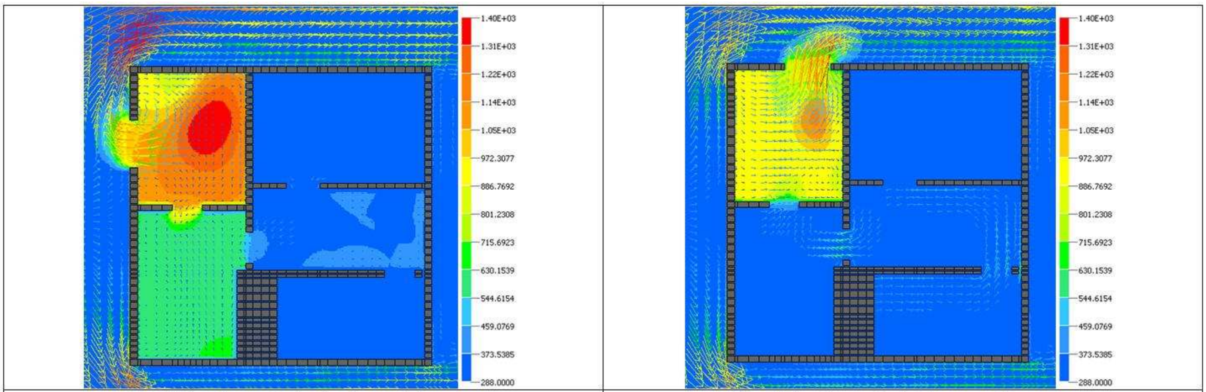
Two further scenarios were numerically investigated in which either window 1 or window 2 was open to investigate the effect of vent location relative to the wind direction on the effectiveness of PPV fans. Figures 14-17 depicts the steady state temperature and smoke distributions in the stairwell and the first floor fire apartment. Figures 14a and 15a show that the temperature reaches 100^0 C outside the fire room and in the stairwell region above the first floor which would put the occupants of the fire apartment and the floors above at risk (Window 1 open). Figures 14b and 15b demonstrates that the PPV fan is able to contain the fire to within the room of fire origin when the sideward window (Window 2) is open instead of windward side window (Window 1).

Figure 14 Temperatures (K) along stairwell vertical plane(350 sec)



(a) Window 1 (windward)open | (b) Window 2 (sideward) open

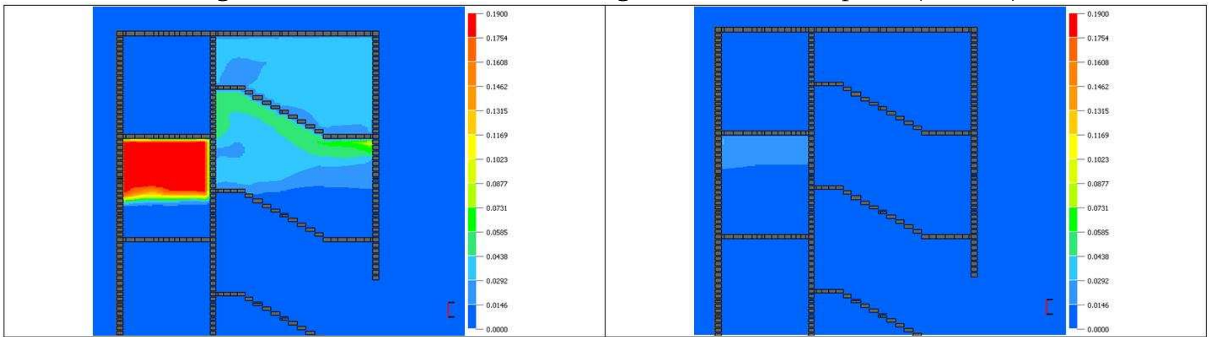
Figure 15 Temperatures (K) across the first floor layout at 1.22 m height (350 sec)



(a) Window 1 (windward)open | (b) Window 2 (sideward) open

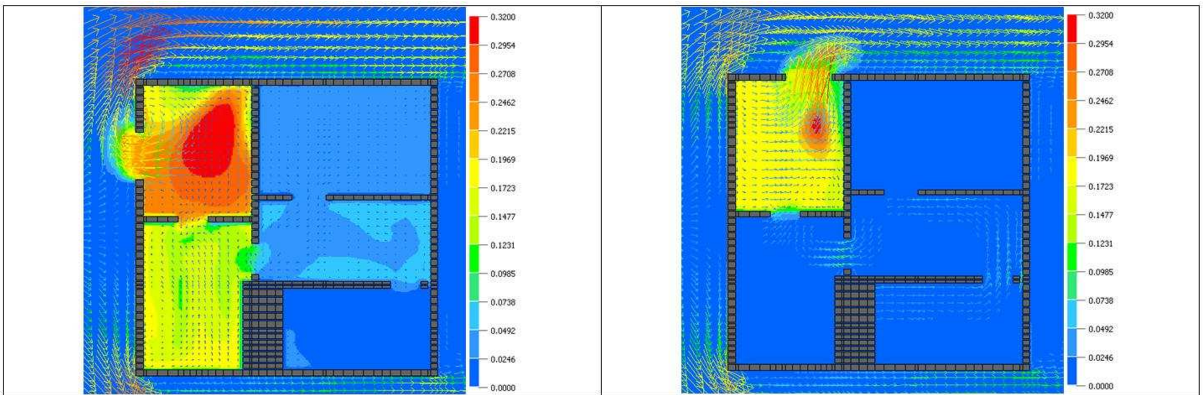
Figures 16a and 17a show that the smoke fills all the rooms of the fire apartment and also the stairwell region above the first floor which might put the occupants of fire apartment and the floors above at risk. It will also make the search and rescue operations difficult for the fire fighters. Figures 16b and 17b demonstrate that the PPV fan is able to contain the smoke within the room of fire origin and maintain better visibility in the stairwell area.

Figure 16 Smoke mass fraction along stairwell vertical plane (350 sec)



(a) Window 1 (windward)open | (b) Window 2 (sideward) open

Figure 17 Smoke mass fraction across the first floor layout at 1.22 m height (350 sec)



(a) Window 1 (windward)open | (b) Window 2 (sideward) open

CONCLUSIONS

The objective of this investigation is to improve our understanding of PPV effectiveness and fire fighting ventilation tactics under wind driven conditions. The SMARTFIRE, CFD fire field modelling tool was used for the study and was validated using two full-scale experiments characterising a PPV fan. The experiments were performed in an open atmosphere and in a simple room scenario. The computed fan velocities were able to capture the centre line velocity decay of the fan jet flow. Furthermore, the computed velocities at the doorway and window plane of the single room geometry compared reasonably well with the experiments.

Under no external wind conditions, for a fan of given size, the flow rate into the doorway increases non-linearly with the area ratio (A2/A1) and reaches a maximum flow rate for a window vent area of 2.5-3.0 times the inlet area. As the window area increases the pressure loss decreases and hence increases the flow rate. The effectiveness of PPV fan under various external wind speeds was studied using the single room geometry. The results suggest that there exists a 'critical wind speed' which is roughly equal to the average velocity at the doorway under no external wind. For wind speeds above the 'critical wind speed' the fan pressure is overcome by the external wind. Furthermore, for a particular wind speed, the flow rate through the room could increase up to a maximum of 100% as the area ratio (A2/A1) reaches 2.7 where A1 is the exhaust area and A2 is the inlet area.

Finally, PPV tactics applied to a real-scale three-storey apartment building were studied for a wind driven room fire at the first floor and presented. The simulation results demonstrate that PPV fans can assist in containing fire and smoke within the room of fire origin when appropriate vent(s) and vent location are chosen. The study demonstrates that careful consideration must be given to the location of exhaust vents relative to prevailing winds. The numerical studies show that creating vents in the windward facing wall can overcome the PPV fan flow resulting in poor conditions developing within the structure. For the PPV fan considered in the present study, wind speeds even as low as 2 m/s could easily overrun the fan causing difficulties to both the occupants and the fire fighters. Location near the side wall corners close to the windward facing wall seems to be one of the best location for venting as the pressure outside was negative. This study demonstrates that CFD based fire modelling can be a useful tool in developing operational fire fighting tactics for the optimal use of PPV fans.

REFERENCES

[1] Svensson, S., Experimental study of fire ventilation actions during fire fighting operation, Fire Technology, vol. 37 (1), 2001.

[2] Kerber, S., Evaluation of fire service Positive Pressure Ventilation tactics on high-rise buildings, Interflam2007, p1289 - p1300, 2007.

[3] Ezekoye, O.A, Svensson, S., Nicks, R., Investigating Positive Pressure Ventilation, Interflam'07, vol. 2, p1277-p1288, 2007.

[4] Galea E. R., Wang Z., Veeraswamy A., Jia F., Lawrence P. J. and Ewer J., Coupled fire/evacuation analysis of station nightclub fire, Proc of 9th IAFSS Symp, Sep. 21-26, 2008, Karlsruhe, Germany, pp 465-476.

[5] Wang, Z., Jia, F., Galea, E.R., Patel, M.K., Ewer, J., Simulating one of the CIB W14 round robin test cases using the SMARTFIRE fire field model, Fire Safety Journal, 36, p661-p677, 2001.

[6] Wang, Z., Jia, F., Galea, E.R., Predicting toxic gas concentrations resulting from enclosure fires using local equivalence ratio concept linked to fire field models, Fire and Materials, Vol. 31(1), p27-p51, 2007.

[7] Kerber, S., and Madrzykowski, D., Fire fighting tactics under wind driven conditions: 7 story building experiments, NIST Technical Note 1629, 2009.

[8] Kerber, S.I and Walton, W.D., Characterizing Positive Pressure Ventilation using computational fluid dynamics, NIST Report, NISTIR 7065, 2003.

[9] Patankar, S.V., Numerical heat transfer and fluid flow, McGraw Hill: New York, 1980.

[10] Rodi, W., Calculation of stably stratified shear layer flows with buoyancy extended k-ε turbulence model, Turbulence and Diffusion in Stable Environments, Ed. Hunt J.C.R., Oxford UK, Clarendon Press, p111-p140, 1985.

[11] Galea, E.R., On the field modelling approach to the simulation of enclosure fires, J. Fire Protection Eng., vol. 1(1), p11-p22, 1981.

[12] Magnussen, B.F. and Hjertager, B.H., On mathematical modelling of turbulent combustion with special emphasis on soot formation and combustion, 16th Int. symp. on combustion, The Combustion Institute, 1977.

[13] Jia, F., Galea, E. R., and Patel, M. K., The prediction of fire propagation in enclosure fires, Fire Safety Science – Proc. of 5th Int. Symp., p345-p354, 1997.

[14] Launder, B.E., and Spalding, D.B., The numerical computation of turbulent flow, Computational methods in Applied Mechanics and Engineering, 3, p. 269, 1974.

[15] Kerber, S., and Walton, W.D., Full-scale evaluation of Positive Pressure Ventilation in a Fire Fighter Training Building, NIST, NISTIR 7342, July 2006.