Numerical Study of Two Air Intake Strategies for a New Fire Laboratory

Z. DONG CHEN* AND DAVID YUNG

Fire Science and Technology Laboratory, CSIRO Manufacturing and Infrastructure Technology, PO Box 56 (Graham Rd) Highett, Victoria 3190, Australia

ABSTRACT: Computational fluid dynamic (CFD) simulations are carried out to select a better air intake system for a new fire laboratory, designed for experiments with a maximum fire heat release rate of 7 MW. Two different air intake systems are considered: a louver system around the lower perimeter of the building or a chimney system that employs vertical ducts to bring fresh air down from the roof to the lower part of the building. The two systems are investigated with different fire sizes and wind speeds. The results show that the chimney system gives a more stable smoke and temperature stratification inside the building and a higher hot smoke layer above the floor. The louver system, on the other hand, gives rise to unsymmetrical air movements inside the building when the wind speed outside is high. The louver system also yields a lower hot smoke layer that can spill smoke through the louvers to the outside of the building, especially for small fires with low exhaust flow rates. The results, therefore, suggest that an evenly distributed chimney system would provide a more stable ventilation environment for conducting fire experiments and would prevent smoke leakage to the outside of the building.

KEY WORDS: fire laboratory, CFD modeling, smoke, air intake, louver, chimney.

INTRODUCTION

THE NEW CSIRO fire laboratory, to be located at North Ryde just outside Sydney, is designed for experiments with a maximum fire heat release rate of 7 MW. The column free building is 20 m long × 20 m wide × 20 m high. Combustion gases from fire experiments will be exhausted from the top of the building to a pollution control facility before they are discharged to the atmosphere. Environmental pollution control also requires that no

*Author to whom correspondence should be addressed. E-mail: Dong.Chen@csiro.au

1042-3915/07/01 0027–14 $10.00/0 DOI: 10.1177/1042391507063859

combustion gases will leak out from the building envelope. The exhaust system and the air intake system are required to provide a stable air environment for the proper conduct of fire experiments. The indoor environment should not be easily disturbed by any wind effect from outside of the building. The hot smoke layer, created by fire experiments, should stay sufficiently high to provide a safe environment for the personnel conducting fire experiments. Consequently, a careful design of the fresh air intake and exhaust system is required, which is the subject of this study.

A common fresh air intake for fire laboratories is the use of louvers along the lower perimeter of the building. The louvers have small opening percentages that allow fresh air to come in and cause a pressure drop. The pressure drop acts as a damper to the wind disturbance outside. Another option is to use a chimney system that employs vertical ducts to bring fresh air down from the roof to the lower part of the building. The ducts cause a pressure drop and therefore dampen the wind effect. A chimney system can employ a number of ducts and distribute them evenly along the perimeter of the building to create an evenly distributed air movement inside the building.

Both air intake designs can dampen the wind effect on the air movement inside the building and may prevent smoke leakage to the outside. However, the relative effectiveness of these two designs needs to be investigated. The objective of this study is to use computational fluid dynamic (CFD) simulations to compare the performance of these two air intake designs for various fire sizes and wind speeds.

NUMERICAL PROCEDURES

Building Representation and Smoke Exhaust/Air Intake Systems

PHOENICS with FLAIR interface [1] is chosen for the simulations in this study that is specifically tailored for HVAC and fire engineers to simulate smoke spread with turbulence models that take into account buoyancy effects. The thermal expansion caused by heat release from large fires is included in the software by using the ideal gas properties for air. Figure 1 shows a schematic diagram of the building geometry. For the simulations in this work, the fire is assumed to be located at the center of the building, occupying an area of $2\,m \times 2\,m$ and at 0.1 m above the floor. Figure 1 also shows the two air intake systems that are being studied. One is the louver system which employs 1 m high panels along the lower perimeter of the building. These panels allow air to come in through the small openings, assumed to be at 20% porosity for this study. The other is the chimney system that employs vertical ducts to bring the fresh air down from the roof to the

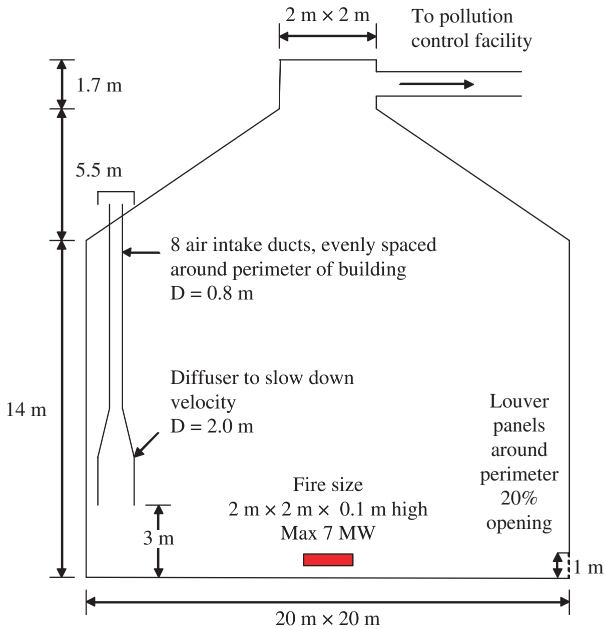

Figure 1. *Schematic diagram of the new fire laboratory with the two air intake systems that were studied: (1) a louver system around the lower perimeter, and (2) a chimney system using ducts to bring fresh air down from the roof (not to scale). (The color version of this figure is available online.)*

lower part of the building. Eight evenly-spaced air ducts are assumed for this study. The air ducts have a smaller intake diameter of 0.8 m, to provide the pressure drop that is necessary to dampen the wind effect and then a larger diameter of 2 m, to slow down the velocity. Although air ducts are used in this study, other means to bring air down, including the use of the wall cavity, can be considered in actual designs expecting similar performance.

The geometry of the CFD model was constructed as closely as possible to represent the fire test building design. Figures 2(a) and (b) show the representations of the fire test building with the louver air intake system and the chimney air intake system, respectively. For both air intake systems, a 1 m diameter circular exhaust fan was assumed to be at the roof exhaust.

Non-uniform distributions of the calculation grid are used in the present study. Finer grids are used near the fire, inside the chimney ducts, near the louver panels and close to the walls. Simulation results employing 323,000 ($85 \times 76 \times 50$) grid points and 623,700 ($105 \times 99 \times 60$) grid points showed no significant changes in temperature and smoke concentration results. To be on the safe side, the larger number (623,700) of grid points is used for both the louver and the chimney designs. A total of 500 non-uniform

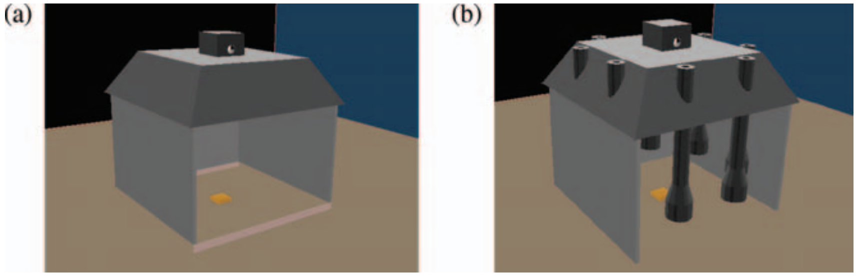

Figure 2. *CFD representations of the new fire building with: (a) the louver air intake system and (b) the chimney air intake system. (The color version of this figure is available online.)*

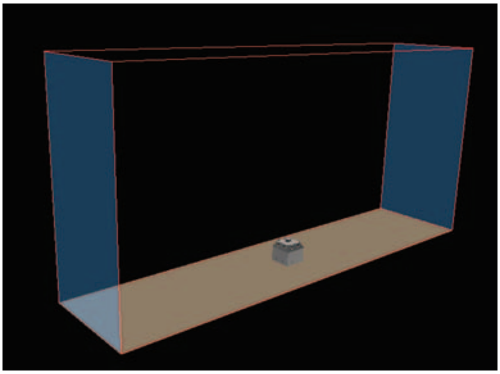

Figure 3. *The computational domain used for modeling the wind effect. (The color version of this figure is available online.)*

time steps, ranging from 1–5 s, are used for a real-time simulation of a 25 min fire duration. Each simulation took approximately 180 CPU h using a 3.06 GHz Pentium IV computer.

Modeling Wind Effect

The wind effect was modeled using a large computational domain surrounding the building as shown in Figure 3. The dimensions of the computational domain, based on the general recommendation [2] that the appropriate cross sectional area should be around 50 times that of the building, are as follows:

- windward direction: 10 times the building height
- leeward direction: 10 times the building height
- lateral wind direction: 5 times the building width
- vertical direction: 10 times the building height.

The wind speed was modeled using a 1/4 power-law profile for a suburban terrain [3].

Required Exhaust Flow Rate

During a fire experiment, a certain minimum smoke exhaust flow rate is required in order to clear the smoke that is generated by the fire in the building. The exhaust flow rate, however, cannot be too large. Otherwise, it would cause a forced ventilation flow that affects the fire. In the present simulation, the smoke exhaust flow rate is estimated using the larger of the following two calculations. The first is based on the energy balance, the equation of which is:

$$m = \frac{q}{C_p(T_\mathrm{e} - T_\mathrm{a})}. \tag{1}$$

Using Equation (1), the required exhaust mass flow rate, m, can be calculated depending on the heat release rate, q, and the hot smoke layer temperature, T_e, that one wants to maintain. The second is based on plume entrainment, the equation of which is [4]:

$$m = 0.21\left(\frac{\rho_\mathrm{a}^2 g q}{C_p T_\mathrm{a}}\right)^{1/3} H^{5/3}. \tag{2}$$

Using Equation (2), the required exhaust mass flow rate, m, can be calculated depending on the heat release rate, q, and the hot smoke layer height, H, that one wants to maintain.

In fire experiments, the hot smoke layer height and temperature are usually maintained to provide a safe environment for the building and for the staff conducting the experiments. In the present simulations, the smoke layer is assumed to be maintained at higher than 6 m with a temperature less than 190°C. In addition, at 2.1 m, the optical density should be less than $0.1\,\mathrm{m}^{-1}$ (visibility more than 10 m) and the temperature rise above ambient should be less than 15 K. Based on the smoke layer temperature of 190°C and a height of 6 m, the required exhaust flow rate can be calculated using the larger of the two values that can be obtained from Equations (1) and (2).

Modeling Assumptions

Fires were assumed, as shown in Figures 2(a) and (b), to have dimensions 2 m long × 2 m wide × 0.5 m high and to be located at the center of the building and at 0.1 m above the floor. All fires modeled were assumed to be t-squared, ultra-fast fires, represented by $q = 0.1876t^2$. For an ultrafast fire, the time required to reach the maximum of 7 MW size is 193 s. Simulations were carried out for a 1500 s duration. The growth period of the fire occupies only a small fraction of the simulation duration and has, therefore, a negligible effect on the relatively long time, steady state results.

The following values were also assumed:

- Ambient pressure at 1×10^5 Pa.
- The pressure drop across the louver with 20% porosity is 37.7 times the dynamic pressure of the approaching air [5].
- Initial temperatures inside and outside of the building are the same at 20°C.
- Thermal radiation is not included.
- Turbulence is modeled using a standard k-ε turbulence model with buoyancy terms.
- Thermodynamic properties of the smoke are similar to air.
- Heat of combustion, $\Delta H = 20$ MJ/kg, assuming a mixture of synthetic and cellulosic polymer fuels.
- Smoke conversion factor (mass of soot produced per mass of fuel burned) $\varepsilon_s = 0.1$.
- Specific extinction coefficient (light extinction coefficient divided by smoke density) $k_m = 7.6 \times 10^3\ m^2/kg$.

The neglect of thermal radiation is to reduce the computational effort. While this may affect the accuracy of the simulation of the smoke extraction system performance, a comparison of the relative performance of the louver and the chimney systems is believed to be valid. Although turbulence models using large eddy simulation (LES) and direct numerical simulation (DNS) are generally more accurate, the standard k-ε turbulence model is still a widely accepted and most economical modeling technique for fire smoke spread simulations, especially for cases with complex building geometry [6].

RESULTS AND DISCUSSIONS

A total of seven simulation runs are carried out in this study. Table 1 shows the various combinations of heat release rate, wind speed, and exhaust flow rate for the two air intake systems. In the simulations, the volumetric flow rate at the exhaust exit at the top of the building was used as the input parameter (converted from the design mass flow rate) and was kept the same for both the louver and chimney system. Based on a heat balance for the whole system and assuming no smoke leakage to the outside,

Table 1. Seven cases were investigated in this study.

Run no.	Heat release rate (MW)	Wind speed (m/s)	Exhaust rate (m^3/s)	Estimated exhaust temp (°C)	Intake system	Grid numbers
1	1	3	15	89	Louver	$105 \times 99 \times 60$
2	1	6	15	89	Louver	$105 \times 99 \times 60$
3	5	3	39	190	Louver	$105 \times 99 \times 60$
4	5	6	39	190	Louver	$105 \times 99 \times 60$
5	1	3	15	89	Chimney	$105 \times 99 \times 60$
6	5	3	39	190	Chimney	$105 \times 99 \times 60$
7	5	6	39	190	Chimney	$105 \times 99 \times 60$

the air temperature at the exhaust exit should be nearly the same for both systems. If the exhaust temperatures are similar and the volumetric flow rates are kept the same, then the mass flow rates are similar for both systems. Results from the simulations, presented in this section, show the air temperatures at the exhaust exit to be similar for both systems. Therefore, the comparisons of the two systems are based on similar mass flow rates even though in the simulations the volumetric flow rate was used as the input parameter.

Two heat release rates were used, 1 and 5 MW, representing a small and a medium fire. Two wind speeds were used, 3 and 6 m/s, representing a range of moderate wind speed that is possible during fire experiments. The required exhaust flow rate is the larger of the two values that are obtained from Equations (1) and (2). The predicted smoke exhaust temperature, T_e, from Equation (1) is also shown in Table 1. All the results presented in this article are the instantaneous results at 25 min from the start of a fire.

Results for 5 MW Fire and 3 m/s Wind

Figures 4(a) and (b) show the temperature distributions for the 5 MW fire, wind speed at 3 m/s and exhaust flow rate at 39 m^3/s for the louver and the chimney systems (Run Nos. 3 and 6). In these figures, the wind is from left to right. The figures show that both air intake systems result in hot smoke layers much higher than 2.1 m above the floor and the air temperatures are less than 35°C within 2.1 m above the floor. Careful examination of these figures show that the hot smoke layer height is around 5.5 m for the louver system and 6.4 m for the chimney system, which are close to the designed 6 m height. The temperature of the hot exhaust smoke for both cases is around 180°C, which is close to the estimated temperature shown in Table 1. For the louver system, wind appears to affect the fire

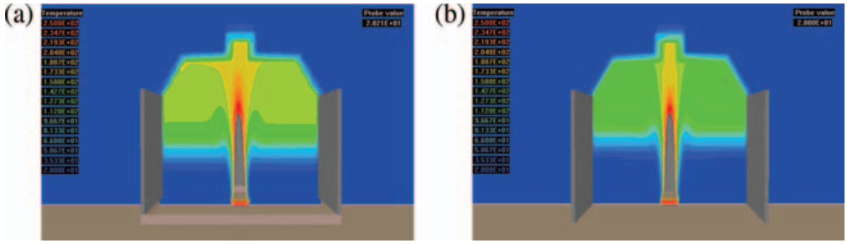

Figure 4. Temperature distribution in the vertical section across the center of the fire for the 5 MW fire, wind speed at 3 m/s, and exhaust flow rate at 39 m^3/s: (a) louver system (Run No. 3) and (b) chimney system (Run No. 6). (The color version of this figure is available online.)

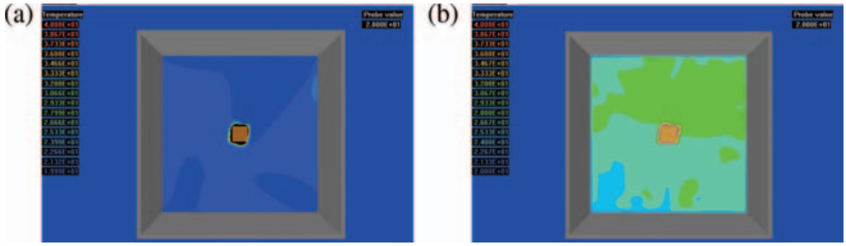

Figure 5. Temperature distribution at 2.1 m above the floor for the 5 MW fire, wind speed at 3 m/s, and exhaust flow rate at 39 m^3/s: (a) louver system (Run No. 3) and (b) chimney system (Run No. 6). (The color version of this figure is available online.)

plume and drive the plume slightly towards the left side, as can be seen in Figure 4(a).

Figures 5(a) and (b) show that the air temperatures at 2.1 m above the floor for the chimney case are much higher than those for the louver case. This is due to the heating up of the intake air by the hot smoke layer through the chimney ducts. Consequently, the results suggested that it is necessary to insulate the chimney ducts or run the chimney ducts outside the building.

Figure 6 shows the smoke mass fraction at 0.9 m above the floor (near the top of the louver height) for the louver system. This figure shows that the smoke is generally controlled within the building envelope with a slight leakage to the outside near the top left corner. This result is expected for large fires with high exhaust flow rates.

Results for 5 MW Fire and 6 m/s Wind

Figures 7(a) and (b) show the temperature distributions for the 5 MW fire with wind speed at 6 m/s, for the louver and the chimney systems

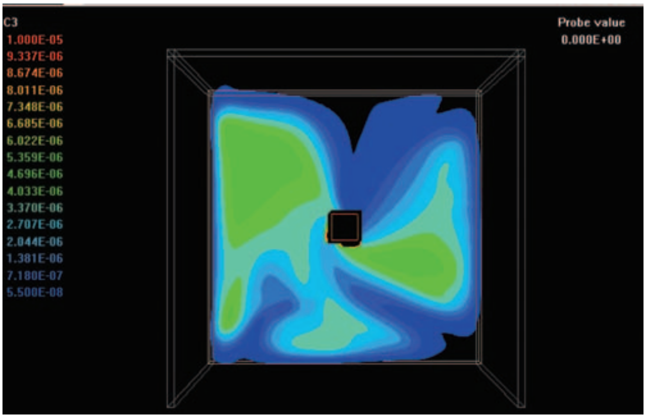

Figure 6. *Smoke mass fraction at 0.9 m above the floor, for the louver system, 5 MW fire, wind speed at 3 m/s, and exhaust flow rate at 39 m^3/s (Run No. 3). (The color version of this figure is available online.)*

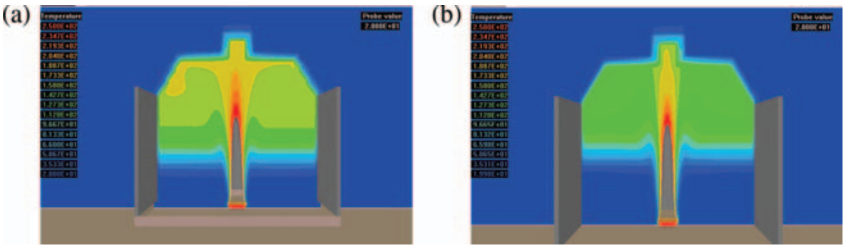

Figure 7. *Temperature distribution in the vertical section across the center of the fire for the 5 MW fire, wind speed at 6 m/s, and exhaust flow rate at 39 m^3/s: (a) louver system (Run No. 4) and (b) chimney system (Run No. 7). (The color version of this figure is available online.)*

(Run Nos. 4 and 7). The figure shows, for the louver system, the hot smoke layer height is now at 4.8 m above the floor, slightly lower than the 5.5 m that was predicted for the same fire size but with a lower wind speed at 3 m/s. For the chimney system, the hot smoke layer is relatively unchanged from the 6.4 m that was predicted for the same fire size but with a wind speed of 3 m/s. This suggests that the wind has more of an effect on the louver system than on the chimney system.

Figures 8(a) and (b) show the visibility levels for the two systems. In both cases, the 10 m visibility line is higher than 2.1 m above the floor, with the louver case at 4.2 m and the chimney case much higher at about 6.5 m. The 10 m visibility lines are close to the hot smoke layer heights.

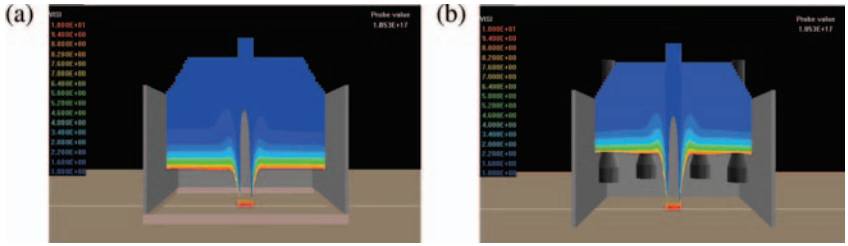

Figure 8. Visibility distribution in the vertical section across the center of the fire for the 5 MW fire, wind speed at 6 m/s, and exhaust flow rate at 39 m^3/s: (a) louver system (Run No. 4) and (b) chimney system (Run No. 7). (The color version of this figure is available online.)

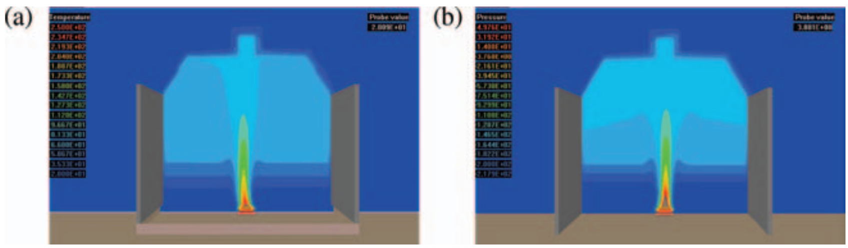

Figure 9. Temperature distribution in the vertical section across the center of the fire for the 1 MW fire, wind speed at 3 m/s, and exhaust flow rate at 15 m^3/s: (a) louver system (Run No. 1) and (b) chimney system (Run No. 5). (The color version of this figure is available online.)

Results for 1 MW Fire and 3 m/s Wind

Figures 9(a) and (b) show the temperature distributions for the 1 MW fire, wind speed at 3 m/s, and exhaust flow rate at 15 m^3/s, for the louver and the chimney system (Run Nos. 1 and 5). Figures 10(a) and (b) show the corresponding temperature distributions at 2.1 m above the floor. Again, for both air intake systems, the hot smoke layers are much higher than 2.1 m above the floor, and the air temperatures are much less than 35°C at 2.1 m above the floor. The hot exhaust smoke temperatures for both cases are around 80°C, close to the estimated temperature shown in Table 1. With the louver system, the wind affects the fire plume and drives the plume slightly towards the left side, as in the case for the 5 MW fire which was shown in Figure 4(a). The chimney ducts are assumed to be insulated in this case and this results in a significant reduction in air temperature at 2.1 m above the floor when compared to that for the 5 MW fire (Run No. 6).

The hot smoke layer is around 3.9 m above the floor for the louver system and 4.2 m for the chimney system. These hot smoke layer heights are lower

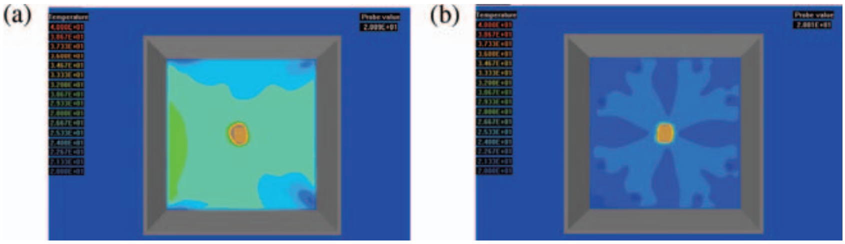

Figure 10. *Temperature distribution at 2.1 m above the floor for the 1 MW fire, wind speed at 3 m/s, and exhaust flow rate at 15 m^3/s: (a) louver system (Run No. 1) and (b) chimney system (Run No. 5). (The color version of this figure is available online.)*

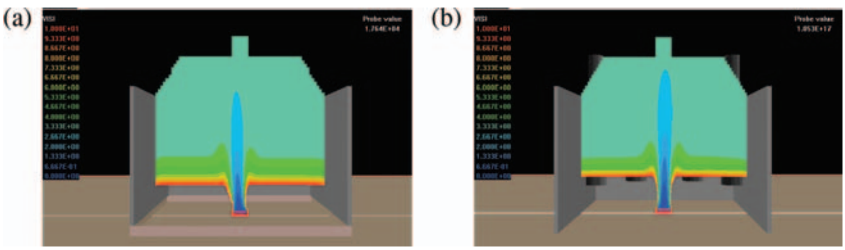

Figure 11. *Visibility distribution in the vertical section across the center of the fire for the 1 MW fire, wind speed at 3 m/s, and exhaust flow rate at 15 m^3/s: (a) louver system (Run No. 1) and (b) chimney system (Run No. 5). (The color version of this figure is available online.)*

than the 6 m height that was designed using the hot smoke exhaust flow rate shown in Table 1. One possible explanation for this is that the 1 MW fire provides a relatively weak stratification. For the 1 MW fire, the temperature difference (which is proportional to the density difference) between the hot layer and the fresh air, as shown in Table 1, is around 60 K, which is approximately 1/3 of the temperature difference for the 5 MW fire.

Figures 11(a) and (b) show the visibility levels for the louver and the chimney systems. In both cases, the 10 m visibility line is higher than 2.1 m above the floor with the louver case around 3.2 m and the chimney case at about 4.2 m.

Results for 1 MW Fire and 6 m/s Wind

Based on the previous results for the 5 MW fire and 6 m/s wind, the wind mainly affects the louver system. In this section, the results for the louver system are discussed. From the results in the previous section, the hot smoke layer is getting low for the 1 MW fire. A stronger wind effect may cause the smoke to leak out. Figures 12(a) and (b) show the smoke mass fraction

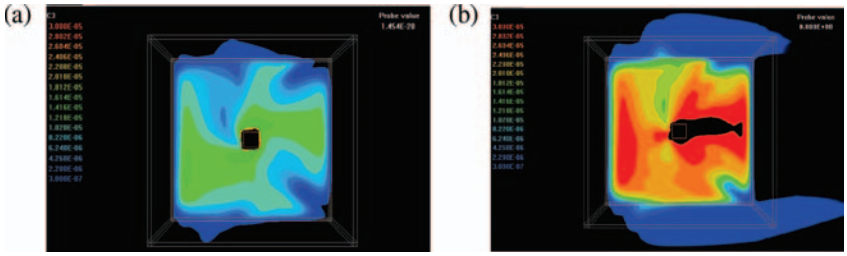

Figure 12. Smoke mass fraction at 0.9 m above the floor for the louver system, 1 MW fire, exhaust flow rate at 15 m^3/s: (a) wind speed at 3 m/s (Run No. 1) and (b) wind speed at 6 m/s (Run No. 2). (The color version of this figure is available online.)

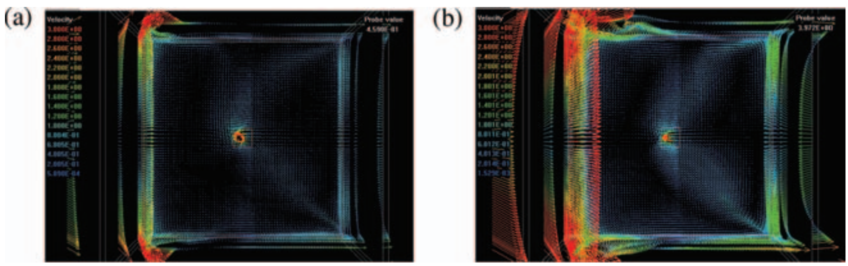

Figure 13. Velocity distribution at 0.9 m above the floor for the louver system, 1 MW fire, exhaust flow rate at 15 m^3/s: (a) wind speed at 3 m/s (Run No. 1) and (b) wind speed at 6 m/s (Run No. 2). (The color version of this figure is available online.)

at 0.9 m above the floor for the louver system with the wind speed at 3 and 6 m/s, respectively (Run Nos. 1 and 2). These figures show that the smoke leaks out through the louver to the outside of the building. The smoke leakage is caused by the wind effect, which results in a negative pressure around the building and draws the smoke to the outside. The leakage of the smoke is worse with a higher wind speed at 6 m/s, as shown in Figure 12(b). To prevent smoke leakage to the outside, a higher exhaust flow rate is required. This, however, may affect the experimental conditions.

Figures 13(a) and (b) show the velocity distributions at 0.9 m above the floor for the louver system with a 1 MW fire and wind speeds at 3 and 6 m/s, respectively (Run Nos. 1 and 2). The figures show that the wind blows in from the left wall of the building and drives smoke to the outside of the building along the sidewalls. It can also be seen from these figures that there are two relatively strong fresh air streams from the bottom right and top right corners. It is believed that these two strong fresh air streams may have pushed the fire plume towards the left side of the building, as seen in Figures 4(a) and 9(a).

SUMMARY

The CFD simulations were carried out to study the air intake system for the new CSIRO fire laboratory, to be located at the North Ryde site just outside Sydney. Two different air intake systems, a louver system around the lower perimeter of the building and a chimney system that uses vertical ducts to bring fresh air down from the roof, were investigated. Simulations were carried out to determine which air intake system would provide a more stable environment for fire experiments and also prevent smoke leakage to the outside of the building. Two different fire sizes, 1 and 5 MW, and two wind speeds, 3 and 6 m/s, were used.

The results show that the chimney system provides more stable smoke and temperature stratifications inside the building and a higher hot smoke layer above the floor for all the conditions investigated. The louver system, on the other hand, gives rise to a slightly unsymmetrical air movement inside the building when the wind speed is high. The louver system also results in hot smoke layers lower than those with the chimney system. The lower height allows smoke to leak through the louvers to the outside of the building, especially for small fires with low exhaust flow rates. The results show that a chimney system employing vertical ducts evenly distributed along the perimeter of the building would provide a more stable environment for fire experiments and would prevent smoke leakage to the outside of the building.

NOMENCLATURE

A = roof surface area (m^2)
C_p = specific heat of air (J/kgK)
H = smoke layer height (m)
ΔH = heat of combustion (J/kg)
k = thermal conductivity (W/mK)
k_m = specific extinction coefficient (m^2/kg)
m = air mass flow rate (kg/s)
q = heat release rate (kW)
t = time (s)
T = temperature (K)
T_a = ambient temperature (K)
T_e = exhaust temperature (K)

GREEK

γ = emissivity
ε_s = smoke conversion factor
ρ = air density (kg/m^3)

SUBSCRIPTS

a = ambient
e = exhaust

REFERENCES

1. PHOENICS On-Line Documentation, Version 3.4 with FLAIR Interface, Concentration, Heat and Momentum Limited (CHAM), Wimbledon, London, 2001.
2. Wright, N.G. and Easom, G.J., "Non-linear k-ε Turbulence Model Results for Flow Over a Building at Full-scale," Applied Mathematical Modelling, Vol. 27, 2003, pp. 1013–1033.
3. Awbi, H.B., "Ventilation, Renewable and Sustainable Energy Reviews," Vol. 2, Chapter 7, 1998, pp. 157–188.
4. Karlsson, B. and Quintiere, J.G., "Enclosure Fire Dynamics," CRC Press LLC, Boca Raton, 2000, p. 205.
5. Miller, D.S., "Internal Flow Systems," Unwin Brothers Ltd, 1990, p. 367.
6. Lo, S.M., Chen, D.H., Yuen, K.K. and Lu, W.Z., "A Numerical Study of External Smoke Spread in Designated Refuge Floors," Building and Environment, Vol. 37, No. 3, 2002, pp. 257–268.