PERFORMANCE BASED EVALUATION OF FIRE SCENARIOS USING FLUENT

Shyam Dayanandan, Dan Chong
Computational Engineering Solutions LLP
E-mail: shyam@ce-sol.com
Website: www.ce-sol.com

ABSTRACT

The Fire and Engineering Codes prescribe many requirements to ensure the safety of occupants and facilitate fire fighting. Often times cost and structural limitations cause civil and M&E designs to deviate from these prescriptions. This paper presents a brief account of 2 case studies where CFD (Fluent) is used to assure Fire Safety and Shelters Department that although there is deviation from the prescription in these specific instances, the alternative solutions adequately meets safety requirements.

Keywords: Performance Based Evaluation, Smoke Spread, Visibility, Radiation Heat Flux

1. INTRODUCTION

In Singapore, the code of practice provides general guidance in facility and mechanical ventilation system designs. Often times cost and structural or space limitations cause civil and M&E designs to deviate from these prescriptions. This paper presents 2 instances where the desire for efficiency and space limitations bring about a deviation from the guidelines prescribed in the codes of practice. In these instances, the designs are submitted to authorities as a Performance Based Design. In these instances, the design has to be accompanied by rigorous Computational Fluid Dynamics (CFD), to substantiate that the level of safety and function that the prescriptive designs hopes to achieve, are also met by the performance based designs. The 2 instances are as follows:

Ductless Jet Fan Mechanical Ventilation System

The main purpose of mechanical ventilation systems for an enclosure in a fire situation is to achieve acceptable level of fire safety requirements within the enclosure. Most importantly, the high temperatures and lack of visibility (due to smoke spread) within the enclosure does not hinder the fire fighting operations by the fire brigade. Before the introduction of Ductless Jet Fan mechanical ventilation system, the conventional mechanical ventilation system for carparks consist of the main exhaust fans and/or main supply fans with its dedicated network of exhaust air duct work and supply air duct work. The high static losses in the duct work require high power consumption by the main exhaust fans and/or main supply fans. Furthermore, the duct work takes up a lot of valuable space and makes the task of coordinating the building services (e.g. fire protection system, air-conditioning and mechanical ventilation system, electrical system, sanitary & plumbing system, etc) more difficult.

The use of Ductless Jet Fan mechanical ventilation system as an alternative solution to the conventional mechanical ventilation system for carparks has grown in popularity as it has addressed the system constraints posed by the conventional mechanical ventilation systems. The Ductless Jet Fan mechanical system for carparks consist of main exhaust fans and/or main supply fans with a series of jet fans strategically located to transport air from one location to another, with the strategy of moving the air from the main supply air fans or supply air permanent openings to the main exhaust air fans.

The use of Ductless Jet Fan MV system must be accompanied by rigorous CFD to substantiate that the level of safety and function provided by prescriptive designs are met. The criteria to ensure acceptability is as follows; after 20mins of the onset of the fire, the region in the carpark where the visibility is below 5m must be kept to an area within $1000m^2$ and the temperature in the carpark (except at the immediate vicinity of the fire) must be kept below 250^oC.

Petrol Station Boundary Line

The boundary line / fence of the petrol station must be a prescribed distance away from the nearest petrol dispenser. This offset distance is calculated based on the natural opening area of the petrol station. For a typical petrol station the offset distance is about 15m. However, for a large majority of petrol stations in Singapore, the lack of space forces the placement of petrol dispensers at a distance much less than 15m from the boundary fence. This is of great concern because in the event of a fire in the petrol station the extreme radiation flux from the fire might ignite items in the adjacent property and cause the fire to spread. So the shorter the offset distance the greater the concern for authorities. From research done by V Babrauskas [1], the Radiation flux required to ignite an item is about 20kW/m^2. Therefore, rigorous CFD studies must be preformed to ensure that although the minimum required offset distance is not met, the Radiation Flux at the offset boundary is well below the amount required for ignition.

2. CASE STUDIES

2.1 Ductless Jet Fan Mechanical Ventilation System for Residential Carpark

2.1.1 Problem Description

The floor area of the car park is about 9550m^2 and has 55 ductless Jet fans (see Fig. 1). About 386820cmh of Fresh Air is supplied and 386820cmh of air is extracted for emergency mode (in the event of a Fire). The objective of the CFD Fire Simulation study is to predict the flow field, visibility and temperature distribution in the carpark. The flow field, visibility and temperature will be analyzed at average human head level (1.7m from the floor). In this study, fire size is 4MW and the fire was modelled by the volumetric heat and smoke release source. Combustion modelling is not used to simulate the fire due to associated numerical instabilities and computational demands. However, the use of a volumetric source model only marginally compromises accuracy, as presented by Saha et. al. [2]. Upon the onset of the fire, it is assumed that the sensors will take 2mins to detect the fire. So after 2mins, the mains fans will switch from normal operation to emergency mode operation. Therefore, the CFD fire simulation strategy is as follows; the transient simulation will be marched in time for 2mins with the fans at low flow and then for a further 18mins with the fans at high flow. The results are analyzed at 20mins after the start of the fire.

2.1.2 Results and Discussion

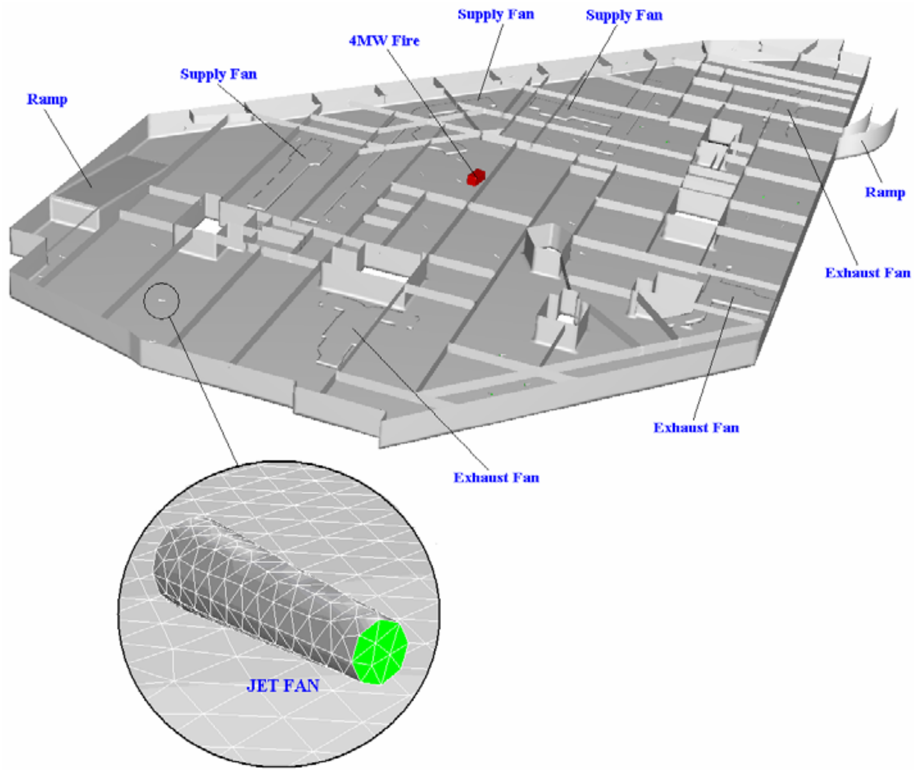

Fig. 1: Computational Grid of Carpark

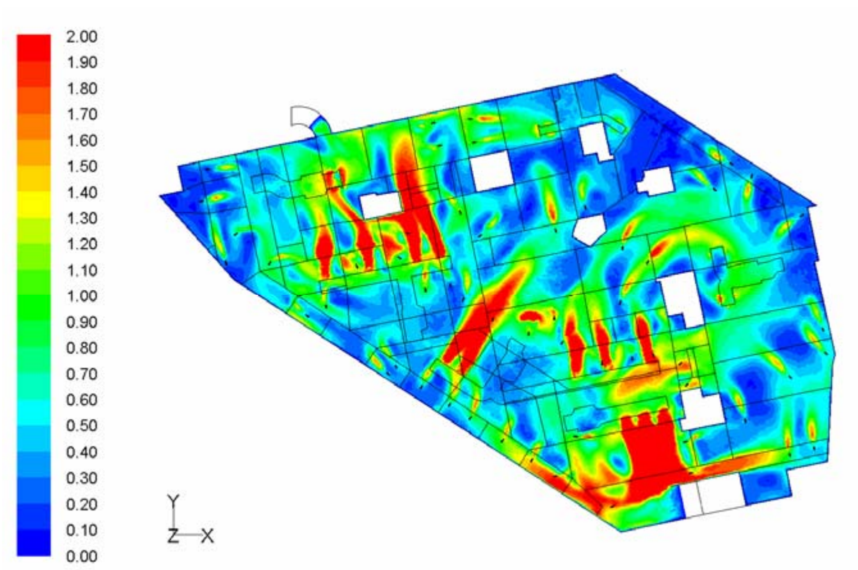

Fig. 2: Velocity Contour at 1.7 Height 20mins after Start of Fire

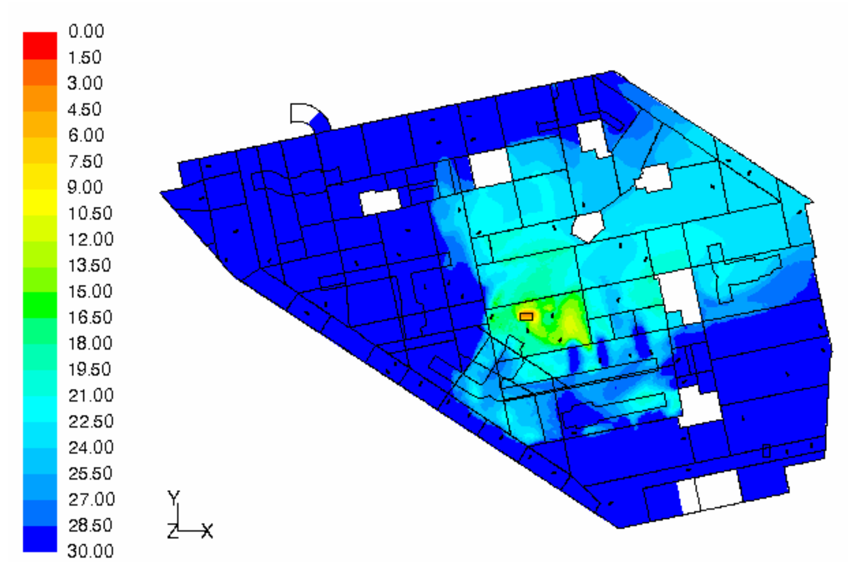

Fig. 3: Visibility Contour at 1.7 Height 20mins after Start of Fire

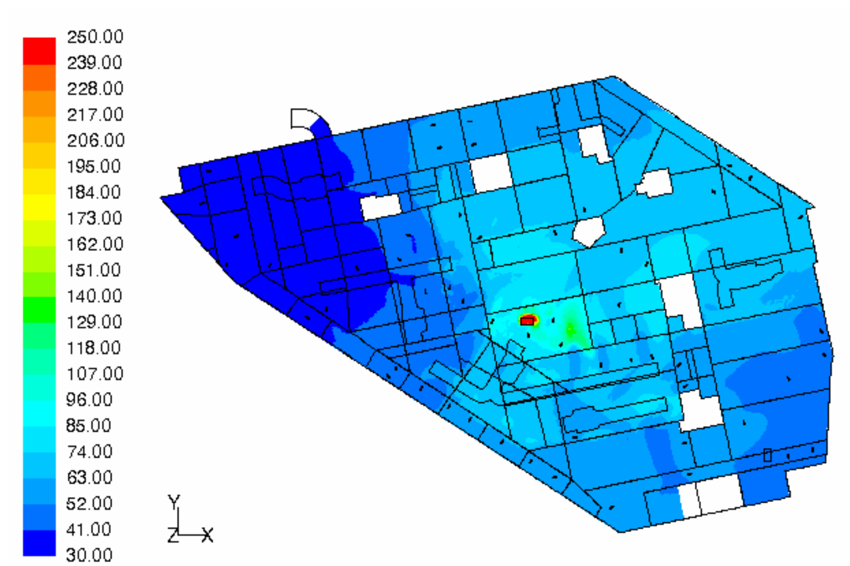

Fig. 4: Temperature Contour at 1.7 Height 20mins after Start of Fire

Velocity Profile

The velocity profile shows velocities more than 0.4m/s. There seems to be hardly any regions of stagnation or regions where the Mean Age of Air is high.

Visibility Profile

For fire fighting purposes, the critical limit for visibility is 5m (i.e. the visibility should be more than 5m). For the 4MW Fire, the region where the visibility is more than the critical limit is only a small region around the fire (indicated in red in the contour plots). The visibility in the large majority of carpark ranges from perfectly visible to about 13m in visibility. For immediate regions surrounding the fire (see Fig.3) the visibility is about 9m. So the ventilation system seems to be able to keep the visibility within acceptable limits in the carpark.

Temperature Profile

For fire fighting purposes, the critical limit for temperature is 250^{o}C (i.e. the temperature should be less than 250^{o}C). For the 4MW Fire, the region where the temperature is more than the critical limit is only a small region around the fire (indicated in red in the contour plots). The temperature in the large majority of Zone 1 is from 30^{o}C to about 85^{o}C. For immediate regions surrounding the fire (see Fig.4) the temperature is about 160^{o}C. So the ventilation system seems to be able to keep the temperature in the carpark within acceptable limits.

The results indicate that the strategic placement of the jet fans is adequate to maintain conditions necessary for the fire fighters. In the event that the jet fan placement is not adequate, the quantity and/or location and orientation of the jet fans must be modified and subsequent CFD simulations must be performed to establish performance.

2.2 Petrol Station with Offset Distance of Only 6m

2.2.1 Problem Description

The petrol station considered has an offset distance of 6m from the boundary fence to the nearest petrol dispenser. This is way below the prescribed minimum of 15m for this petrol station. From Fig. 5, it can be seen that there are two dispensers nearest to the boundary line. Therefore, the worst possible case of two 5MW truck fires (one at each dispenser) is considered for the simulation. Of the two 5MW fires simulated for this study, 30% of the heat energy is assumed to be transmitted by radiation and 70% by conduction and convection. For this CFD simulation, the P-1 radiation model is used to model the Radiation flux. To add conservativeness to the results, all surfaces are prescribed to provide a reflection of 10% of the incident radiation and the absorption and scattering within the medium (air and smoke) is not considered. The Radiation Flux is monitored at 9 points just after the boundary fence, as shown is Fig. 6.

2.2.2 Results and Discussion

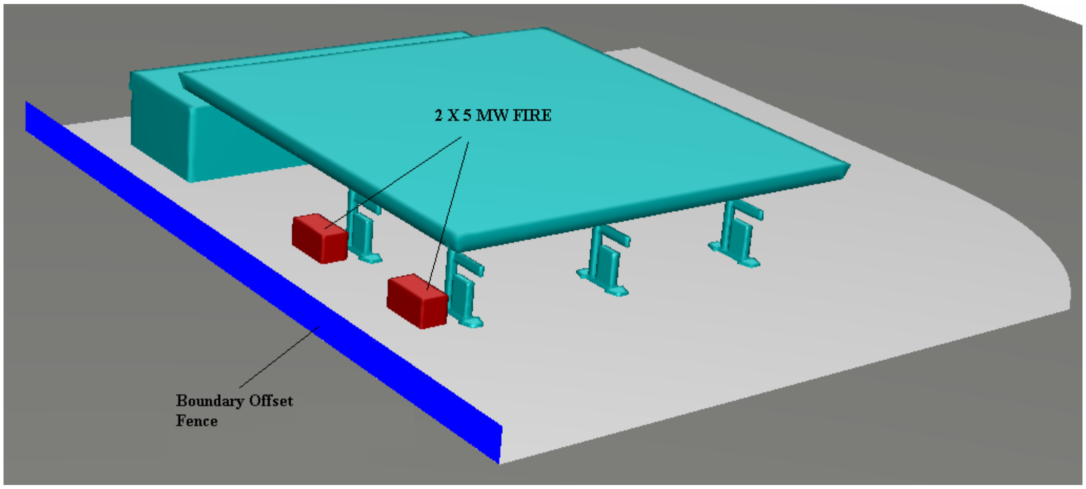

Fig. 5: Computational Grid of Petrol Station

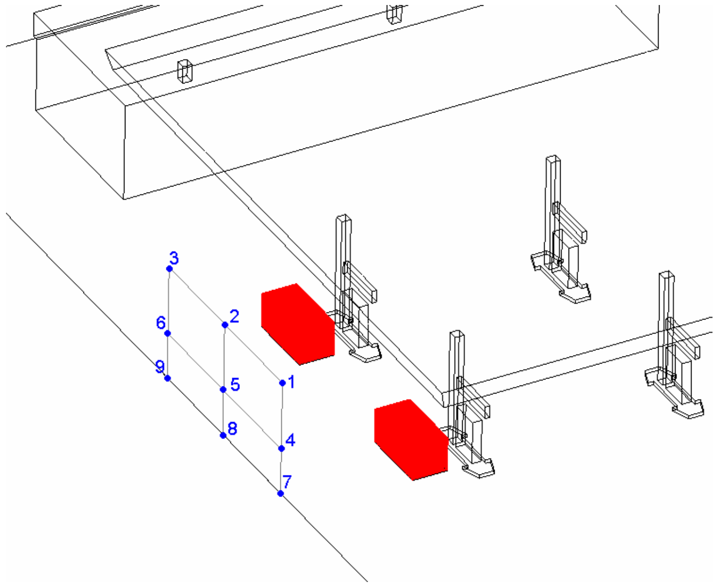

Fig. 6: 9 Sampling Data Points just after Boundary Fence

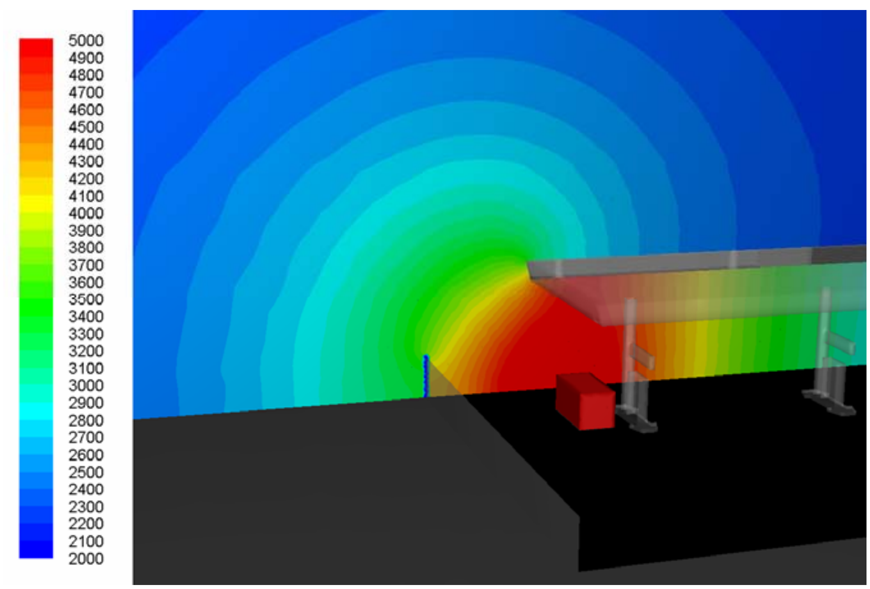

Fig. 7: Radiation Flux Contour Plot on Vertical Plane between the 2 Fire Points

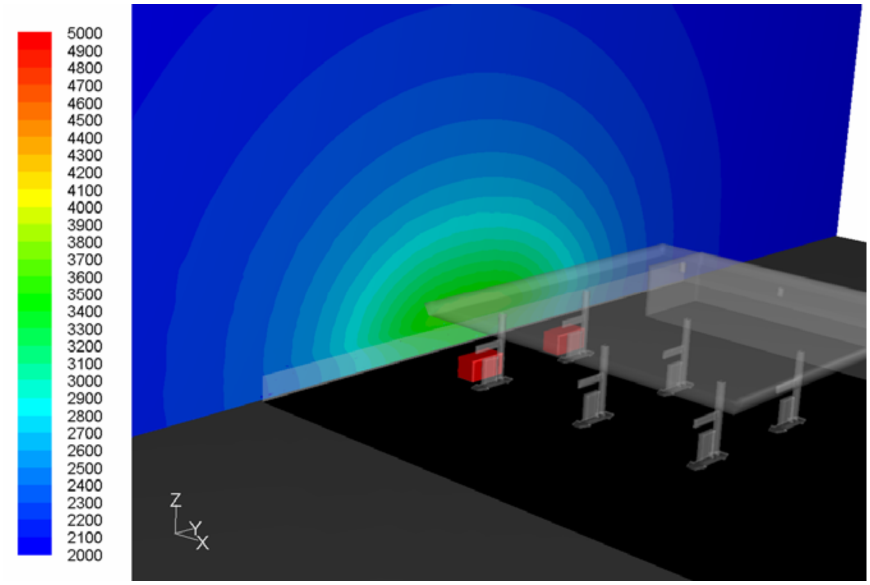

Fig. 8: Radiation Flux Contour Plot on Vertical Plane Along the Outside of Boundary Fence

Points	1	2	3	4	5	6	7	8	9
Radiation Flux (W/m^2)	3236	3326	3267	3460	3556	3500	3210	3311	3243

Table 1: Radiation Flux at 9 Points along the Boundary Plane

Radiation Flux

The Radiation flux at the offset boundary is predicted to reach a maximum of about $3600W/m^2$. The above predictions are probably over-predicted since the absorption coefficient and scattering coefficient of air and smoke (from the fires) are not taken into account. The Radiation flux of $3.6kW/m^2$ is way below $20kW/m^2$ required to cause ignition, as presented by V Babrauskas [1]. Thus, although the offset distance of 6m is way below the prescribed minimum of 15m for this particular petrol station, the CFD analysis shows that the Radiation flux at the boundary is well within safe limits.

3. CONCLUSION

The CFD studies conducted assures the Fire Safety and Shelters Department that although there is deviation from the prescription in these specific instances, the alternative solutions adequately meets safety requirements. Therefore CFD allow engineers to design and implement efficient solutions, while ensuring that safety is not compromised.

4. REFERENCES

1. S. Saha, S. Bandyopadhyay and S. Sarkar, Fluent CFD Conference, *Numerical Study of Flow Induced by Fire in a Compartment*. Fluent India, Pune 2003.

2. V. Babrauskas, *Will the Second Item Ignite?*, NBSIR 81-2271, Center for Fire Research, National Engineering Laboratory, US Department of Commerce, National Bureau of Standards, May 1981