Mesh techniques and uncertainty for modelling impulse jetfans

O. A. (Sam) Alshroof

CFD manager – Olsson Fire and Risk

Email: Sam.Alshroof@olssonfire.com

Abstract

This study presents the numerical modelling of a jetfan stream using an open source Computational Fluid Dynamics (CFD) software named Fire Dynamics Simulator (FDS)6.5.2. The software is based on an accurate turbulence model named Large Eddy Simulation (LES) along with a structure mesh to discretize the numerical domain.

This research is focussed on the feasibility and accuracy of modelling jetfans using FDS, where a simplified three-dimensional model is built to represent a single jetfan in an open-deck car park, to compare the air jet downstream of the jetfan using various mesh techniques.

This study included a verification of the model by solving a single jetfan using a relatively fine mesh and comparing it to a coarser mesh using various mesh techniques. The results obtained show that the behaviour of a jetfan can be resolved using FDS but it requires a significant number of grid points which makes it unfeasible to model a medium to a large size car park.

Introduction

The use of Jetfans to ventilate car parks in buildings has become an acceptable alternative solution in more countries compared to traditional ducted systems. The efficiency and the energy reduction [2] are the main reason for that significant switch.

The design of the jetfan location, direction and minimum fan capacity is quantified using Computational Fluid Dynamic (CFD) models, purely based on CO concentration during the peak hour during the day [3]. Once the design of the jetfans is finalized, a smoke and fire model will be conducted to ensure early detection of smoke, so the jetfans are shut down until the fire-brigade arrive to have a manual over ride to clear the smoke if needed.

Jetfans have built-in smoke detectors at the side of the jetfan body, which will help for early detection of the smoke to eliminate any delay caused by sprinkler head activation.

Limited research been done on impulse jetfans car park with a narrow nozzle as is shown in Figure 1, which includes mesh sensitivity analysis or even qualitative comparison to a smoke test of a jetfan.

Figure 1 impulse fan (JISU-CPC-100N JetVent Fan)

Tony carried out a study on impulse jetfans, where he modelled the impact of a jetfan air stream on sprinkler activation, using FDS. A jetfan was modelled with a flow rate of 1.35 m^3/s through a nozzle of 900 x 100 mm. As claimed the centreline velocity at the outlet was measured approximately 15 m/s at maximum. The mesh resolution used was between 100 and 400 mm, where the fine mesh of 100 mm where used to resolve the aerodynamic plume. The plume inlet boundary condition is resolved using one cell size across the height of the nozzle, which might lead to a numerical error. The study didn't include a mesh refinement study to verify the use of the single cell across the height of the nozzle.

Most numerical studies been carried out using a cylindrical type jetfan with a bigger jetfan nozzle opening. This makes resolving the inlet boundary condition of the jetfan easier.

Research was done by Lu et. al. [5] where they modelled cylindrical type jetfans in a car park sized 80 m long, 40 m wide and 3.2 m in height using a uniform mesh sized 200 mm, with a fire heat release rate of 4 MW. The mesh was considered to be a fine mesh based on studies done by Viegas [6, 7]. The researchers found that the jetfans will limit the fire smoke spreading but it may cause temperature rise on the downstream side of fire source. They also analysed the effect of the jetfan capacity and the number of jetfans.

This study focuses on the mesh resolution and its impact on the jet stream.

Jetfan modelling

Commercial software Pyrosim was used to build up the numerical model and define the boundary conditions of the single jetfan in a computational domain sized 25000, 7500 and 3000 mm. The performance of the jetfan modelled was set at 1.62 m^3/s. The nozzle of the jetfan was 1000 mm in width and 125 mm in height and the flow was directed 7° from the horizontal level.
The jetfan was assumed to be located in an infinitely wide carpark with a ceiling height of 3000 mm. The top and the bottom surfaces of the computational domain were set as a wall boundary condition to represent the ceiling and the floor of the car park, and the four sides of the domain were assumed to be an opening boundary condition.

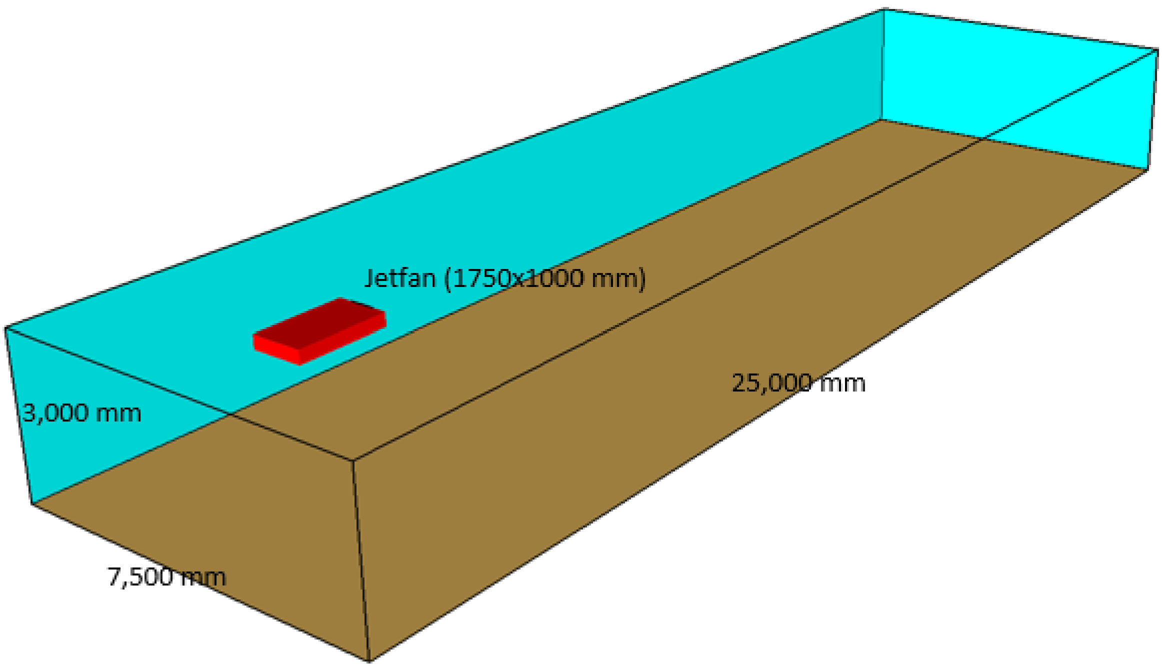


Figure 2: Iso-view of the computational domain generated for a single jetfan

The computational domain was discretized using various techniques to verify a suitable mesh size to solve the jet stream up to a level of accuracy.
In order to accurately resolve the air stream of the jetfan it required a three cell size cross the height of the nozzle, which makes it numerically expensive to fit a uniform equal mesh size across the computational domain.
This study will demonstrate the effect of the stream velocity using the following mesh reduction techniques

1- Reducing the number of grid cells across the nozzle height.
2- Increasing the nozzle height to fit one coarse cell size.
3- Modifying the nozzle aspect ratio by maintaining the area, to fit more cells across the nozzle height.
4- Modifying the aspect ratio of the mesh.
5- Refining the mesh at the jetfan level.
6- Using refined embedded mesh in the vicinity of the jetfan nozzle.

The following table summarizes the mesh arrangement used for all modelled scenarios:

Table 1: list of modelled scenarios

Case #	Base mesh size [mm]	Refined mesh size [mm]	Depth of refinement [mm]	Number of cells across inlet	Nozzle size [mm]
1	41.66	-	-	3	8
2	62.5	-	-	2	8
3	125	-	-	2	2
4	250	62.5, 125	500, 750	2	8

5	250	62.5 (imbedded) , 125	500(imbedded), 1250	2	8
6	250	125	1250	1	8
7	250	125	1250	2	4
8	250	-	-	1	4
9	250x62.5	-	-	2	8
10	250x125	-	-	1	8
11	250x125	125x62.5	1000	2	8
12	125	-	-	1	8

Jetfan Results

The production of high air volume at high jet velocity is the key factor to entrain the flow downstream of a jetfan. This entrainment behaviour makes the use of jetfans in carparks preferable compared to traditional duct work.

CFD is the tool used to design the jetfans location and capacity within a car park; therefore, resolving the air jet downstream of a jetfan correctly is important. FDS uses LES as a default turbulence model which needs a level of accuracy in choosing a suitable mesh size to resolve even the basic behaviour of the air jet.

Table 2 below shows a zoomed side view the mesh used for each scenario modelled, and Table 3 summarizes the velocity slice across the centreline of the jetfan for all scenarios listed in Table 1.

The results shown in Table 3 indicate that the mesh has a significant impact on the results, where the steam jet on scenario number 1 and 2 are different to the rest of the models, where the air jet is clearly oriented downward away from the ceiling for approximately 2.5 m from the jetfan nozzle, then the air jet starts to dissipate and spread across most of the floor height at about 6 m from the nozzle.

This type of behaviour was not observed in any other scenarios, which indicate clearly that reducing the mesh quality will impact directly on the velocity field downstream of the jetfan. The u-velocity for all 12 cases modelled was measured for three points located at 1.5 m, 2.0 m and 2.5 m AFL and is presented in Figure 3, Figure 4 and Figure 5 respectively. The figures also show that the u-velocity is comparable for Case 1 and Case 2, but for the rest of the cases is not. Some of the cases are different by 300 % compared to Case 1, although in some of the cases, the jetfan was moved one cell below the ceiling level to eliminate the wall effect on the air jet, but the velocity measured was still comparable.

It was also noted that reducing the Case 5 (embedded mesh) shows the least accurate results. Likewise, changing the aspect ratio of the mesh (Case 9-11) also creates an error which impacts on resolving the air jet accurately.

Changing the aspect ratio of the nozzle from 8 to 2 as modelled in Case 3 leads to overestimating the jet air stream compared to Case 1 and Case 2, as is shown in Table 3. The horizontal slices of velocity presented in Table 4 at 2.85 m and 1.5 m above floor level (AFL) demonstrate that the jet at the jetfan level is less turbulent and at a higher speed compared to Case 2, however the jet speed at lower level is reduced.

Table 2: Zoomed side view for the mesh used for various scenarios

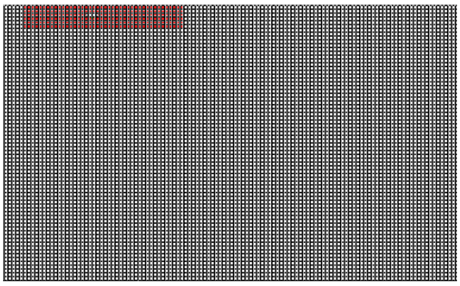
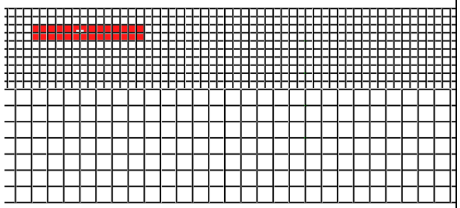
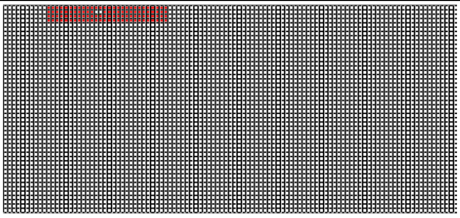
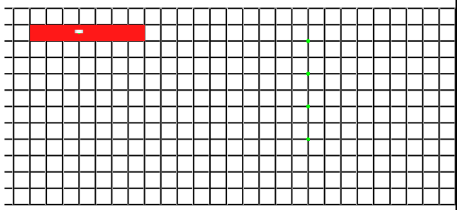
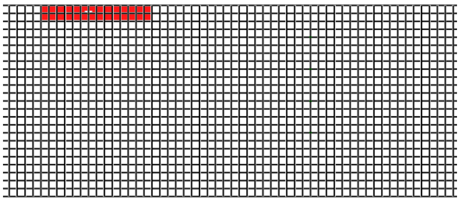
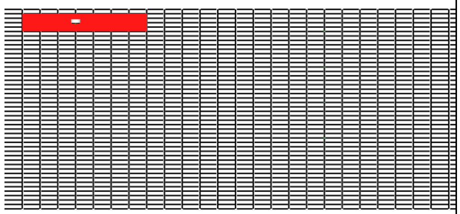
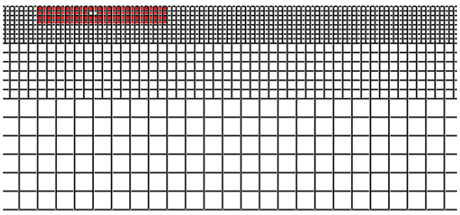
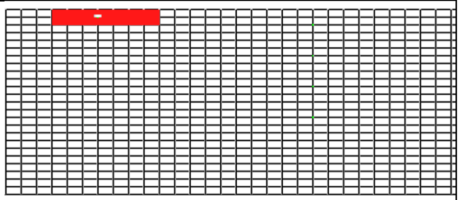
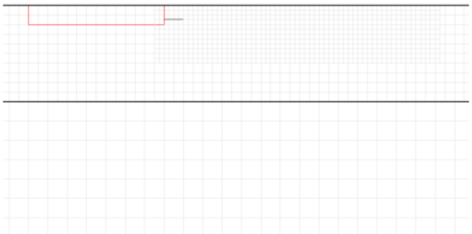
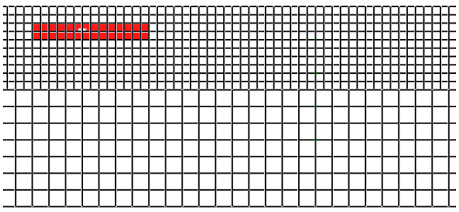
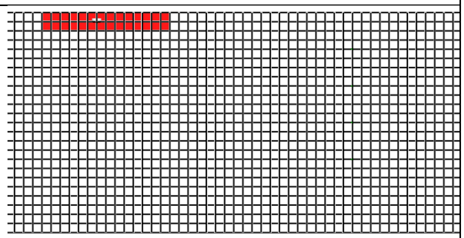
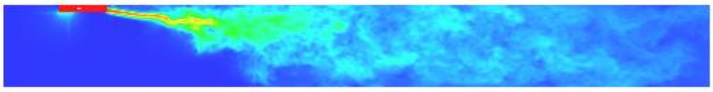
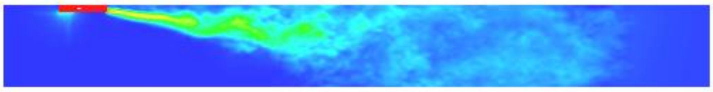
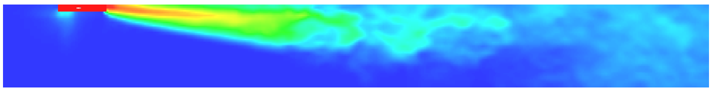
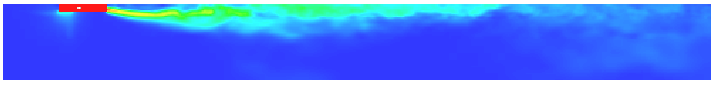
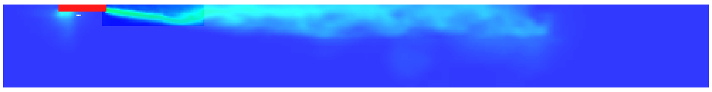
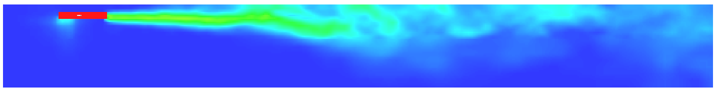
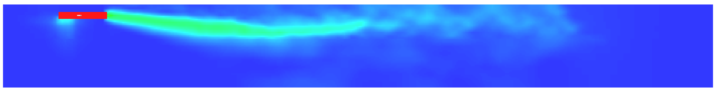
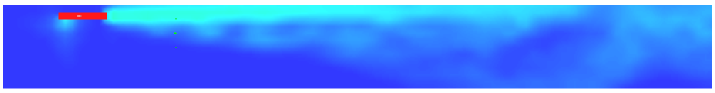
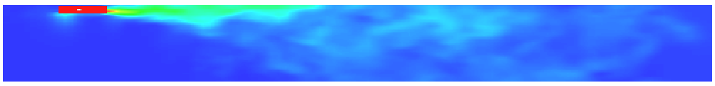
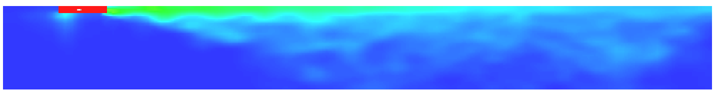
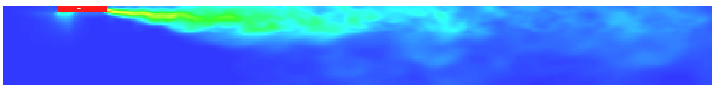
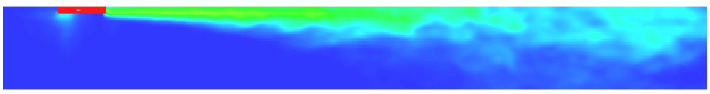
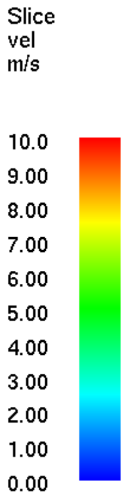
Case #	Mesh	Case #	
1		7	
2		8	
3		9	
4		10	
5		11	
6		12	

Table 3: Vertical slice of velocity located along the centreline for all cases

Scenario number	Velocity
1	
2	
3	
4	
5	
6	
7	
8	
9	
10	
11	
12	



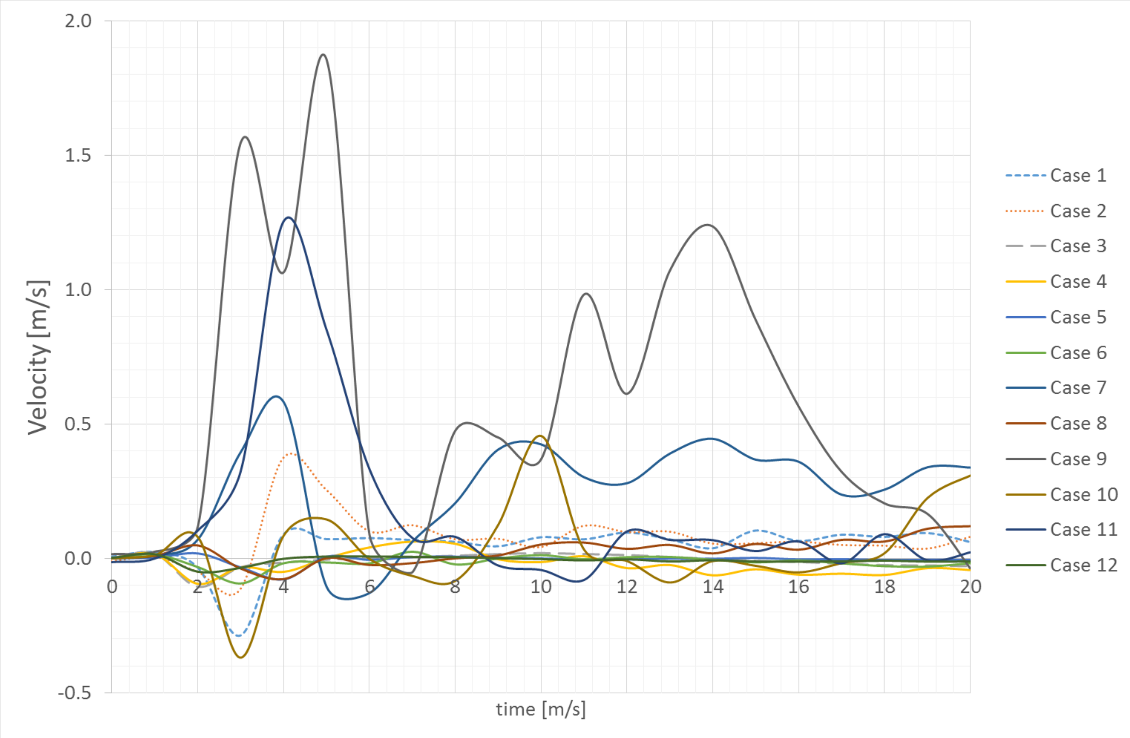


Figure 3: u-velocity for a point located 1.5 m AFL

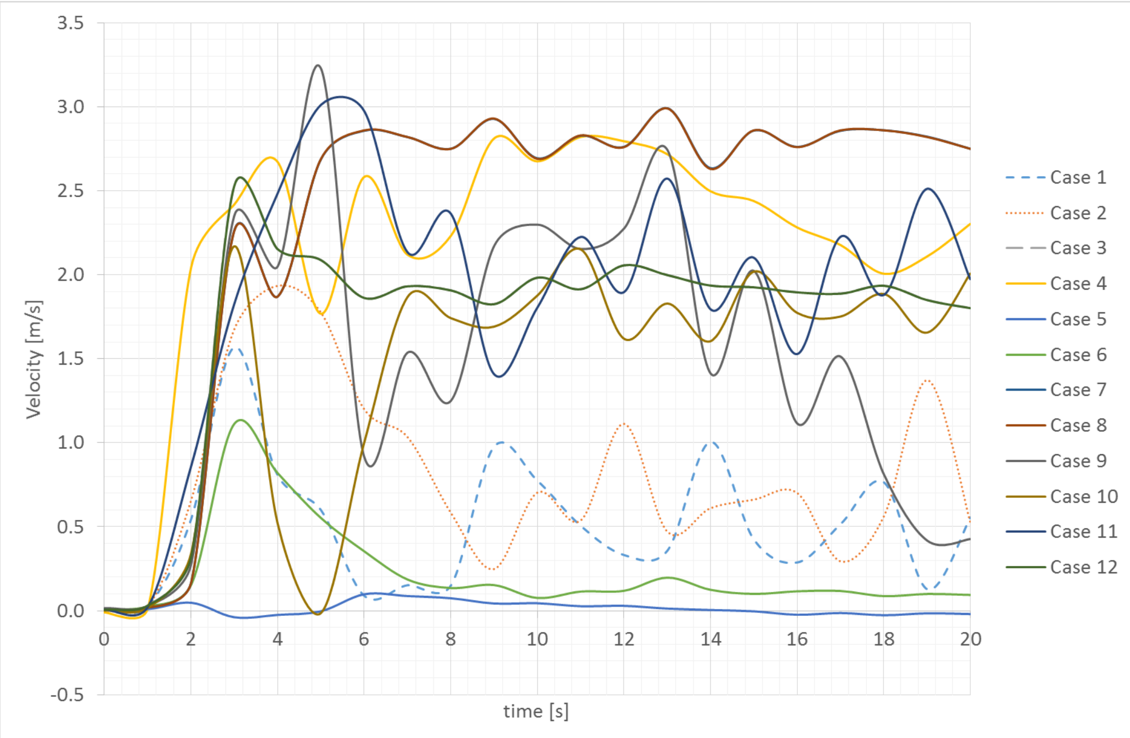


Figure 4: u-velocity for a point located 2.0 m AFL

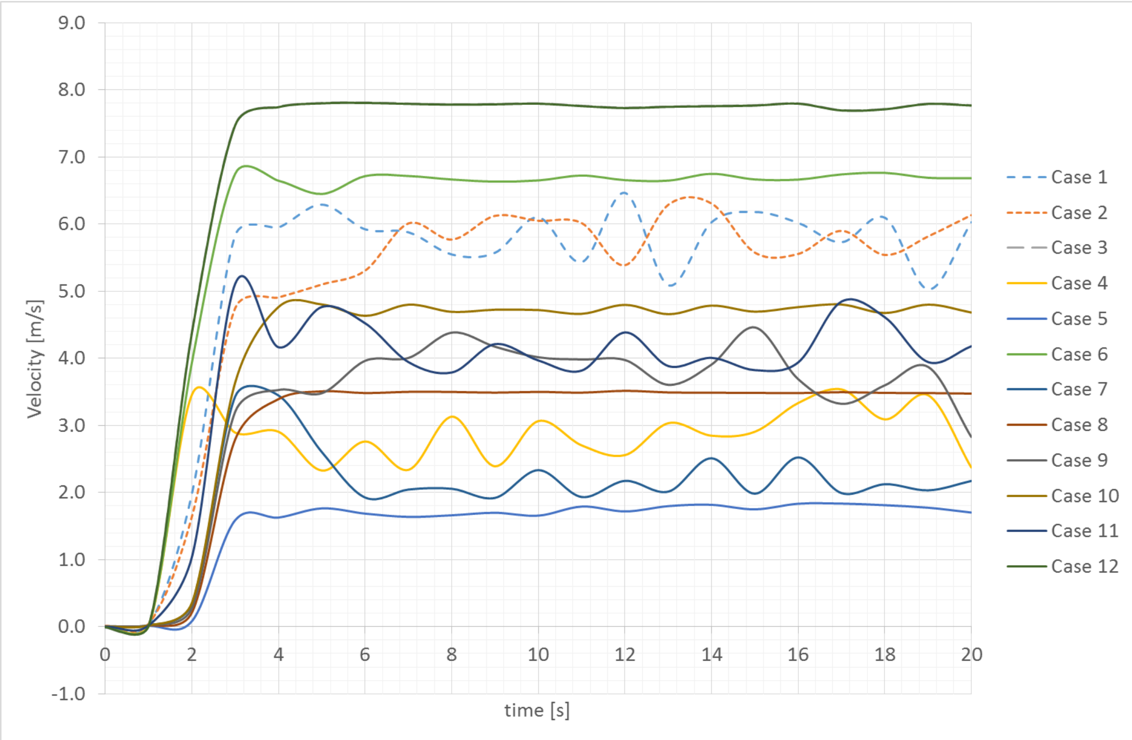
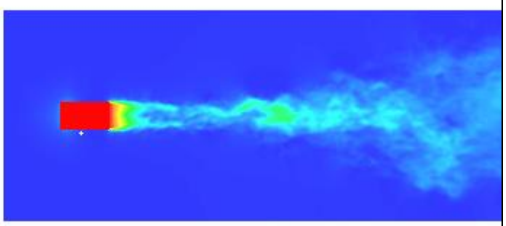
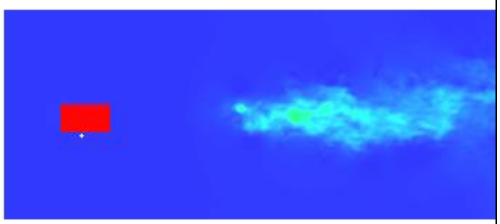
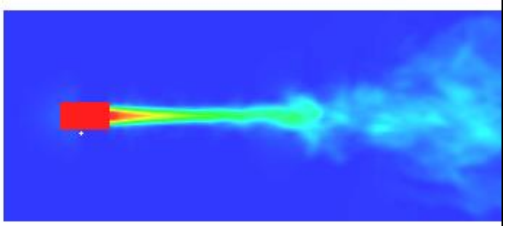
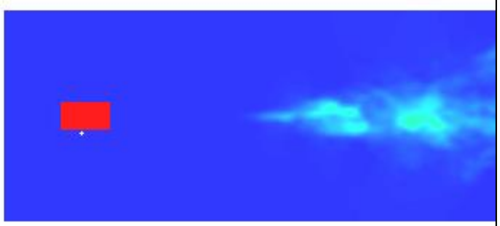


Figure 5: u-velocity for a point located 2.5 m AFL

Table 4: Horizontal slice of velocity at various slices

Case #	Slice at 2.85 m AFL	Slice at 1.5 m AFL
2		
3		

To get an idea of the numerical time required to complete the iterative solution for a period of 20 s, times are presented in Table 5. The table summarizes the grid cell count for each of the scenarios modelled and the time required to complete the run using a Linux cluster machine, Intel(R) Xeon(R) CPU E3-1240 v3 @ 3.40GHz. To resolve the jet fan in accurately it requires 2501 min (~ 1.74 days) to solve only 20s, without considering any extra complexity of adding fire to the model. Reducing the mesh as presented for Case 2 will reduce the time to 328 min (~5.5 hours), which is significantly less than Case 1 but still putting a limitation on how much larger the model can be and for how long it can be run for.

In Australia, the jetfans need to be turned off on detection and turned back on when the fire brigade intervene. This can lead to a simulation period of 1200 s – 1800 s which clearly indicates that the time required to resolve the model within a reasonable time is central.

Table 5: list of cell number and time required to solve up to 20 s

Case #	Cell Number	Time [min]
1	8,009,280	2501
2	1,749,600	328
3	296,640	119
4	488,220	440
5	182,046	93
6	145,230	107
7	145,230	37
8	37,080	8
9	139,050	46
10	74,160	22
11	244,110	141
12	296,640	50

Conclusion

Based on the verification method used in this research, the jetfan behaviour can be resolved by using FDS software. In order to obtain a relatively accurate solution a significant number of cells is needed which requires significant time to resolve the model.

Due to the lack of experimental results, the results obtained in this research were not validated, and further research is recommended.

Reference:

[1] J.W. Deardorff. Numerical Investigation of Neutral and Unstable Planetary Boundary Layers. Journal of Atmospheric Sciences, 29:91–115, 1972.27

[2] Senveli, A., Dizman, T., Celen, A., Bilge, D., Dalkılıç, A. S., & Wongwises, S. (2015). CFD Analysis of Smoke and Temperature Control System of an Indoor Parking Lot with Jet Fans.*Journal of Thermal Engineering*,*1*(2), 116-130.

[3] Australian Standard AS 1668.2 – 2002, The use of mechanical ventilation and air-conditioning in buildings, Part 2: Mechanical ventilation for acceptable indoor-air quality.

[4] P.A. (Tony) Enright, Impact of jet fan ventilation systems on sprinkler activation, Case Studies in Fire Safety 1 (2014) 1–7.

[5] Lu S, Wang YH, Zhang RF, Zhang HP (2011). Numerical study on impluse ventilation for smoke control in an underground car park. Procedia Engineering, 11: 369–378.

[6] Viegas JC. The use of impulse ventilation to control pollution in underground car parks. International Journal of Ventilation 2009;8(1):57- 74.

[7] Viegas JC. The use of impulse ventilation to control pollution in underground car parks 2010;25(1):42-53.