

Building Generated Wind Shear and Turbulence Prediction utilising Computational Fluid Dynamics

Neihad Hussen Al-Khalidy
 SLR Consulting
 CFD, Wind and Energy Technical Discipline
 2 Lincoln Street, Lane Cove 2066
 AUSTRALIA

Nal-khalidy@slrconsulting.com <http://www.linkedin.com/pub/neihad-al-khalidy/19/a5/341>

Abstract: Wind shear and turbulence caused by buildings in the vicinity of airports have been recognised as a cause of aircraft instability during landing or take-off for some time. The study of the pre and post-development winds has been undertaken using a quantitative Computational Fluid Dynamics (CFD) analysis. Turbulent (gust) wind flow in an urban environment is better modelled with wind tunnel when compared with CFD models employing low end turbulence models (e.g. the standard k-epsilon model). On the other hand small building features are better modelled in CFD versus wind tunnel testing, where model buildings are typically built to a scale of around 1:400 or 1:300. This is important when modelling small structures in the vicinity of airports or assessing the benefit of rounding the corners, adding small canopies, trees, porous screens and blades provided to mitigate adverse wind conditions. Wind tunnel measurements are also performed at a few selected points within the tested model while CFD can provide comprehensive output of the entire flow field. With advances in computational method Scale Resolving Simulation (SRS) models are used for high Reynolds number flows with strong separation. SRS models such as Large Eddy Simulation (LES) or hybrid LES/Reynolds Averaged Navier Stokes (RANS) approaches are used to overcome the limitation of statistical RANS models and resolve significant parts of the turbulence spectrum in CFD simulations. LES is still prohibitively computationally expensive when used for solving external high Re numbers flows in complex built environment even with today's computational power. In this study hybrid Detached Eddy Simulation (DES) is used to assess the risks of buildings near the threshold of runways and predict wind shear and turbulence for critical flight paths. DES is proposed due to ability to combine the benefits of RANS and LES while minimising their disadvantages. The RANS (Realizable k-epsilon in this study) can achieve good prediction for attached boundary layers while LES can capture unsteady motions of large eddies in separated regions. The paper provides a procedure to reliably assess the risk of building generated wind shear and discusses some of the parameters that may have influence on the results accuracy.

Key-Words: - CFD, Airport Wind Shear, Turbulence, Detached Eddy Simulation, Built Environment

1 Introduction

Wind shear and turbulence caused by buildings in the vicinity of airports have been recognised as a cause of aircraft instability during landing or take-off for some time. Examples are shown in [1] and [2].

The instability which building-induced wake effects can cause to an aircraft is significantly reduced once the airplane has touched down (upon landing) or is at reasonable height (200ft off the ground prior to landing). After touch-down, the aircraft has increased stability/support from contact with the runway pavement and above 200 feet, both the consequences of a drop in altitude or a change in wind bank are considerably less and the pilot has increased latitude and so time to correct an induced effect on the aircraft prior to touch-down.

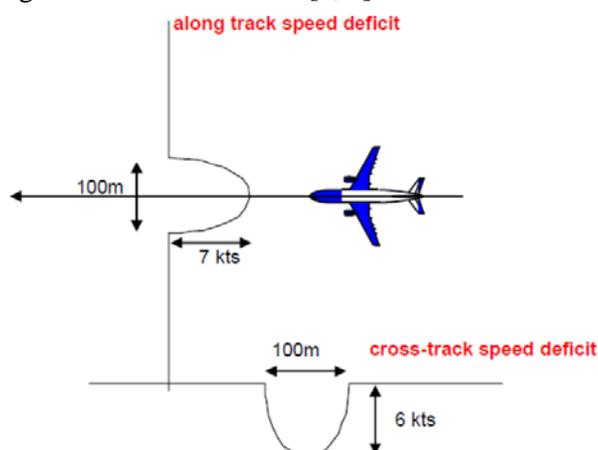
Wind shear and turbulence guidance criteria due to obstacles at and around airports were developed by the National Aerospace Laboratory of The Netherlands (NRL) [3]. The NLR report contains the following criteria and conclusions:

- The "1:35 height rule"
 - Any building lies within the 1:35 height regime is deemed acceptable.
- The "7-knot along wind criterion"
 - The variation in mean wind speed due to wind disturbing structures must remain below 7kt (3.6 m/s) along the aircraft trajectory at heights below 200 ft. The speed deficit change of 7kt must take place over a distance of at least 100 m.
- The "6-knot cross-wind criterion"

- The variation in mean wind speed due to wind disturbing structures must remain below 6 knots across the aircraft trajectory at heights below 200ft. The speed deficit change of 6 knots must take place over a distance of at least 100m.
- The “4-knot turbulence criterion”
 - The standard deviation of wind speed must remain below 4 knots at heights below 200ft.

The wind shear criteria are depicted schematically in Figure 1.

Fig.1 Wind Shear Criteria [3, 4]



The above criteria were also adopted by the National Airports Safeguarding Framework (NASF) in Australia [4]. The NASF guideline sets out a summary of steps to follow when assessing the risks from proposed buildings near the threshold of runways. In general, wind tunnel testing or Computational Fluid Dynamics (CFD) is required to assess the proposed structure when a proposed development penetrates the 1:35 rules.

Turbulent (gust) wind flow environment in an urban environment is better modelled with wind tunnel when compared with CFD models employing low end turbulence models (e.g. the standard k -epsilon model). On the other hand small building features are better modelled in CFD versus wind tunnel testing, where model buildings are typically built to a scale of around 1:400 or similar. This is important when modelling small structures in the vicinity of airports or assessing the benefit of small canopies, porous screens and blades provided to mitigate adverse wind conditions. Wind tunnel measurements are performed at a few selected points within the tested model while CFD can provide comprehensive output of the entire flow field.

Turbulence is predicted using one or a combination of the following methods:

- Direct Numerical Simulation (DNS)
- Reynolds-Averaged Navier-Stokes (RANS) Equations
- Large Eddy Simulation (LES)

DNS refers to solving the Navier-Stokes equations numerically without any turbulence model by resolving all scales down to the scale of viscous dissipation. At this stage DNS is possible for low Reynolds number flows and usually performed on simple geometries.

For most real world building problems turbulence is, in principle, described by the Reynolds Averaged Navier-Stokes (RANS) equations [6]. The quality of CFD simulation depends on the selected turbulence model. In practical problems the turbulence model should be as simple as the relevant physics will permit.

A considerable number of CFD publications have been published in the International Conferences and Journal of Wind Engineering & Industrial Aerodynamics in the past two decades. CFD predictions of wind flow around bluff bodies have been compared and validated against wind tunnel and full scale measurements in the open literature [i.e. 5, 6, 7, 8 and 9]. In general good agreement is obtained when best practice guidelines are used for the modelling.

Certain classes of high Reynolds number flows with massive separation are better covered by models in which all or part of the spectrum is resolved in all or part of the computational domain using Scale-Resolving Simulation (SRS) models (e.g. Large Eddy Simulation (LES) and hybrid LES/RANS approaches). Those models will overcome the limitation of statistical RANS models that are calibrated on the basis of mean parameters of thin shear flows. A comprehensive review of SRS models are summarised in [10 to 13]. The following comments are made with regards to using SRS models for built environment studies:

- LES has been the most widely used model over the last decades. The model is still prohibitively computationally expensive when used for solving external high Re numbers flows in complex built environment even with today's computational power.
- Hybrid models such as Detached Eddy Simulation (DES) [11, 12 and 13] or Scale-Adaptive Simulation (SAS) [14], etc. are used to reduce computational cost for making SRS models affordable in industrial applications.
- In hybrid models the wall boundary layers are covered by a RANS model with the LES region or large eddies are solved only away from walls.

It is important to understand that hybrid models are still substantially more computationally expensive than RANS simulations but less than LES computational cost.

- SRS models are still requiring extensive experience and attention to various details including model selection, grid generation, numerical setting and solution interpretation [15].

The main objectives of this study are to:

- Analyse the local weather data and calculate wind shear and natural turbulence exceedance probabilities data.
- Develop 3D CFD models for complex built environment including topography, trees and detailed design features at areas of interest.
- Utilise hybrid LES/RANS approaches, namely DES for wind shear and turbulence studies for pre and post development scenarios.
- Develop procedures to predict wind shear and turbulence along possible aircraft trajectories.
- Discuss some of the challenges facing CFD to reliably predict turbulence or modelling built environment.
- Provide recommendations to mitigate adverse wind conditions where required and possible.

2 Problem Formulation

The averaged Navier Stocks RANS/LES equations with eddy viscosity can be written as follows:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j}$$

Where ρ is the density, u is the velocity, p is the static pressure, ν is the kinematic viscosity, τ_{ij} is the stress tensor including both normal and shear components of the stress. The over bar represents time averaging in RANS models and spatial filtering in LES.

Hybrid models are possible due to similarity in filtered momentum equation to the RANS equation.

DES is proposed in this study due to ability to combine the benefits of RANS and LES while minimising their disadvantages while the RANS (Realizable k-epsilon in this study) can achieve good prediction for attached boundary layers, LES can capture unsteady motions of large eddies in separated regions. The model was first proposed by Spalart et al [11, 12].

The DES model switches between RANS and LES by a comparison of the turbulent length scale

L_t with the computational grid Δ_{max} spacing (maximum $(\Delta x, \Delta y, \Delta z)$). The model selects the minimum of both and thereby switches between RANS and LES mode by replacing ϵ in the k-equation by:

$$\epsilon = \frac{k^{3/2}}{L_t} \rightarrow \epsilon = \frac{k^{3/2}}{\min(L_t, c_{DES} \Delta_{max})}$$

A validation study [10] for a single high rise building and for a flow field in an actual urban environment against measurement data and wind tunnel testing concluded that the DES results show significantly better prediction of wind flows in comparison to other approaches.

In this study DES was used to predict wind shear and turbulence in complex urban environment in the vicinity of airport.

3 Case Study

3.1 Project Description

The project site is located adjacent to a local airport with three parallel operating runways. The airport operates 24 hours a day, with limitations placed on night circuit training.

A 3D model of one of the proposed design schemes and surrounding built environment is shown in Figure 2. The proposed precinct comprises a number of low rise warehouses.

The average roof height for the proposed warehouses is approximately 10 m and some of the warehouses penetrate the 1:35 surface rule then wind shear and turbulence effect must be considered.

An initial assessment of the most intensive proposed building layout identified a number of areas of potential adverse wind conditions leading to the following wind amelioration recommendations which were subsequently incorporated into the current CFD model:

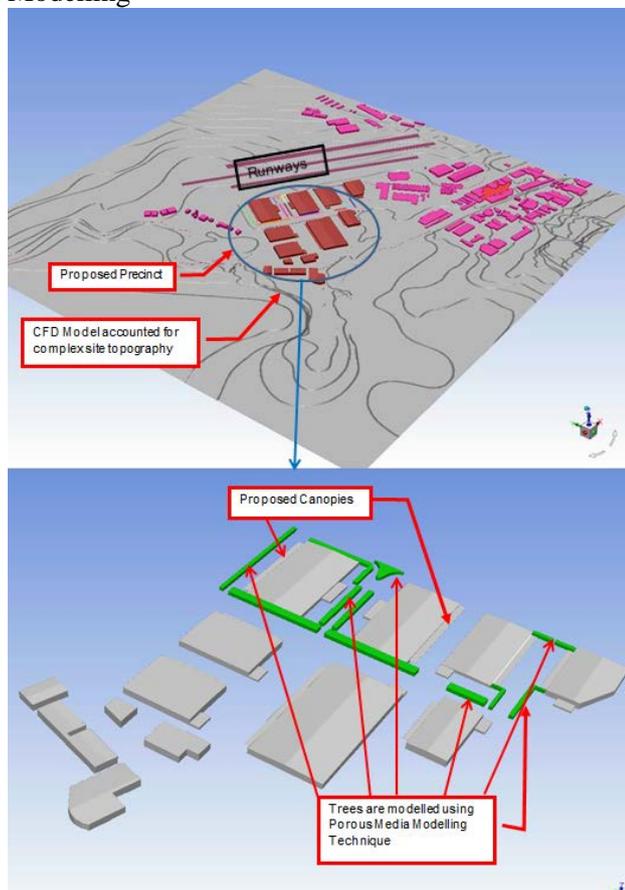
- Staggered building configurations are proposed to avoid wind tunneling between buildings.
- Roof Pitch = 2 degrees.
- Ridge to run along the longest axis of the warehouse.
- Landscaping and wind screen buffers as noted on Figure 2:
 - A maximum tree height to top of buildings
 - A tree porosity of 30% is assumed.

The wind directions deemed to have the greatest impact on the runways are the winds originating from:

- South East (SE)

- South Southeast (SSE)
- South (S)
- South Southwest (SSW)
- South West (SW)

Fig.2 Development Site and Geometry for CFD Modelling



3.2 Local Weather data

Wind shear and natural turbulence may already be exceeded at many airports. Statistical assessment of the weather data will therefore be an important part of the wind turbulence study.

The data recorded during 5 years period at the airport weather station comprise the mean wind speed (hourly average), the peak gust speed (occurring at any time within the relevant hour) and the mean wind direction. Occurrence statistics of the mean wind speeds is shown in Table 1. The following major conclusions can be reached from Table 1:

- A 25 knots or higher mean wind speed is exceeded 0.062% (27 hours total) in a five year period taking in to account all wind directions.
- From the southwest (i.e. between 202.5° and WSW 247.5°) there were no hours where the mean wind speed exceeded 25 knots.

- From the southeast to southwest (i.e. between SE 168.75° and SW 247.5°) there were 21 hours per five years where the mean wind speed exceeded 25 knots.

Table 1 Mean Wind Annual Exceedance Probability in 5 Year at the Airport Weather Station

10m ht MEAN Wind Speed (knots)	Wind Direction								
	N 0°	NE 45°	E 90°	SE 135°	S 180°	SW 225°	W 270°	NW 315°	ALL
5	4.36%	7.65%	6.41%	10.11%	7.24%	7.62%	6.38%	6.90%	56.6%
10	0.71%	2.88%	3.09%	6.50%	3.38%	2.06%	2.57%	1.34%	22.5%
15	0.08%	0.14%	0.24%	1.97%	1.01%	0.39%	0.89%	0.39%	5.13%
20	0.000%	0.000%	0.016%	0.262%	0.203%	0.023%	0.169%	0.059%	0.732%
25	0.000%	0.000%	0.000%	0.027%	0.016%	0.000%	0.016%	0.002%	0.062%

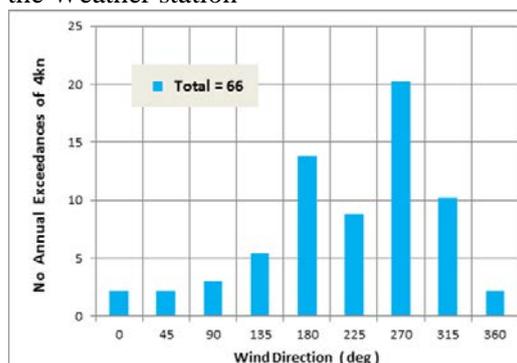
The turbulence and corresponding turbulence intensity are not directly recorded but need to be inferred from the mean wind and peak gusts. The relationship between mean wind speed and gust wind is shown in below equation:

$$V_{gust} = \bar{V} + g \cdot \sigma_V$$

Where V_{gust} = peak gust speed (occurring at any time within the one hour measurement period); \bar{V} = mean wind speed (averaged over the one hour measurement period), σ_V = root mean square (RMS) wind speed or “turbulence”, and g = peak factor. The natural and existing built environment turbulence exceedance is shown in Figure 3. The following major conclusions can be reached from Figure 3:

- There were 66 times in 1 year where the 4-knots exceeded taking into account ALL wind directions.
- The wind directions which relate to the location to the proposed development are the winds originating from the SE to SW. There were approximately 28 occasions in 1 year where the 4-knots exceeded between SSE 168.75° and SSW 258.75°).

Fig. 3 4 knot Turbulence Exceedance Probability at the Weather station



The turbulence criterion is therefore exceeded without the addition of the proposed development. Turbulence impact assessment is therefore of particular interest for decision maker when approving off-airport buildings.

3.3 Wind Shear and Turbulence CFD Modelling

3.3.1 Modelling Configuration

A 3D model of the project site and surrounding buildings and structure blocks was created from the supplied architectural drawings and a CAD Model supplied by the client (Refer Figure 2).

The geometry for CFD Modelling is shown in Figure 4. A calculation domain of 2,448 m length, 2,448 m wide and 200 m high was used for the CFD analysis. The developed model accounts for all small features of the proposed development (e.g. canopies, gaps, etc.).

3.3.2 Discretization

The software package utilised in the current CFD analysis is the commercially available code ANSYS-Fluent [16]. The CFD model solves continuity and momentum in the computational domain to predict the unsteady state airflow at and around the project site.

- The quality of the mesh is a critical aspect of the overall numerical simulation and it has a significant impact on the accuracy of the results and solver run time.
- For the current analysis, polyhedral elements with a total number of 18,852,516 cells were used to cover the computational domain. Polyhedral cells are especially beneficial for complex geometries including site topography and can handle recirculating flows and may provide more accurate results than even hexahedra mesh. For a hexahedral cell, there are three optimal flow directions which lead to the maximum accuracy while for a polyhedron with 12 faces there are six optimal directions which, together with the larger number of neighbours lead to a more accurate solution with a lower cell count. It is also worth mentioning that the development of hexahedral element for real and complex built environment is challenging.
- The following techniques were used for discretization:
 - A second order numerical scheme for discretization of pressure and momentum to obtain more accurate results.

- A Detached Eddy Simulation (DES) turbulence model was used due to ability to handle separation, recirculation, high gradients and computational time advantages. Realizable k-epsilon was used for the RANS region.
- Proposed trees are modelled using porous media modelling technique.
- The solution is also combined with a wall function to avoid using very fine elements near the wall.
- An iterative procedure was used to estimate the air velocity in terms of three directions, pressure profile and turbulence parameters.

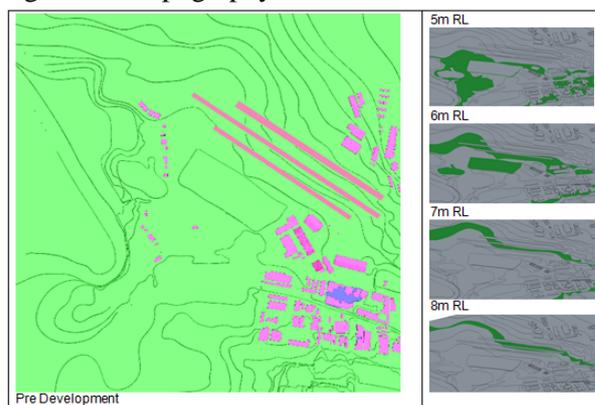
3.3.3 Sensitivity Analysis

3.3.3.1 Site Topography

It is strongly recommended to review detailed survey data (if available) when conducting wind turbulence studies. Topographic features less than 2 m in the areas of interest may have a minor impact on the results accuracy. Features less than 1 m are also ignored when developing scale models for wind tunnel testing.

The available survey data for the proposed development site has shown elevated ground at the areas of interest. All topographic features were therefore included in the pre- and post-development scenarios (Refer Figure 4).

Fig. 4 Site Topography



3.3.3.2 Extent of Built Environment

The size of the computational domain should be selected according to the best practice guideline. A building with height H may have a minimal influence if its distance from the region of interest is greater than $6-10H$ [17].

All buildings within more than 600 m diameter were included in the developed CFD model (Refer Figure 2).

3.3.3.3 Top of the Computational Domain

The top of the computational domain should be at least 5H away from the tallest building with height H [17].

In order to avoid artificial flow over the building, 20H is used in this study where H is the building height.

3.3.3.4 Mesh Sensitivity Analysis

As mentioned above the wall boundary layers are entirely covered by the RANS model and the LES model is applied to the separated flow. Careful grid generation is required to avoid having low grid resolution or grey areas in areas of interest.

Ideally, a wall resolution $y^+ = 1$ is recommended at the region of interest [18]. Grid development in DES is more complicated than RANS and benefits may reduce if $y^+ \gg 1$. Grid resolution in complex urban environment could be challenging especially if area of interest includes mid or high rise building. A mesh sensitivity analysis is strongly recommended to optimize the mesh based on the given flow conditions and modelled urban complexity.

Based on a mesh sensitivity assessment, 18,852,516 polyhedral cells were used to cover the computational domain. A minor accuracy benefit is gained when number of cells is increased to +20 million. On the other hand a reasonable computational time is saved for the 18,852,516 cells scenario.

In general, the grid resolution should be as high as the computing powers permits.

3.3.3.5 Time Steps

DES requires significant computational resources to obtain a valid numerical solution. Challenges include very small time steps (0.01 to 0.05 s) to meet the recommendations in best practice guidelines. The best way to reduce computational time is to

- Start with a steady state RANS solution and then switch to DES model;
- Initially use variable time steps if possible to speed up the simulation;
- Reduce time step as per the best practice guidelines; and then
- Reduce number of iterations per time step if possible.

3.3.4 CFD Results and Discussion

3.3.4.1 Wind Shear

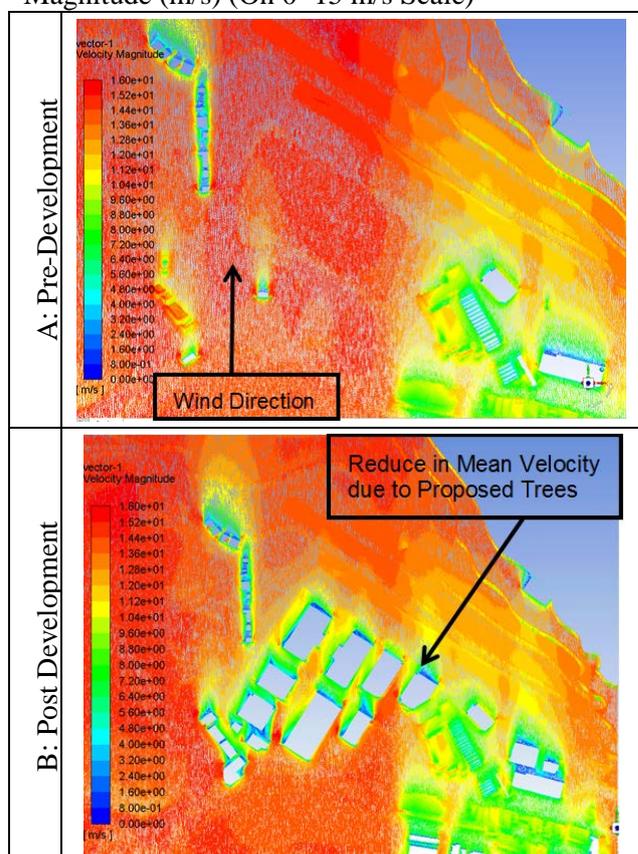
Sample results for the following wind conditions are shown in Figure 5 to Figure 11:

- Wind Speed = 25 knots (12.78 m/s) at 10 m above ground.
- Wind Angle = 180°

Figure 5 shows the wind speeds at 1 m above the ground. Dark blue represents still conditions at 0 m/s and red representing the strongest wind speed. The following conclusions can be reached from the figure:

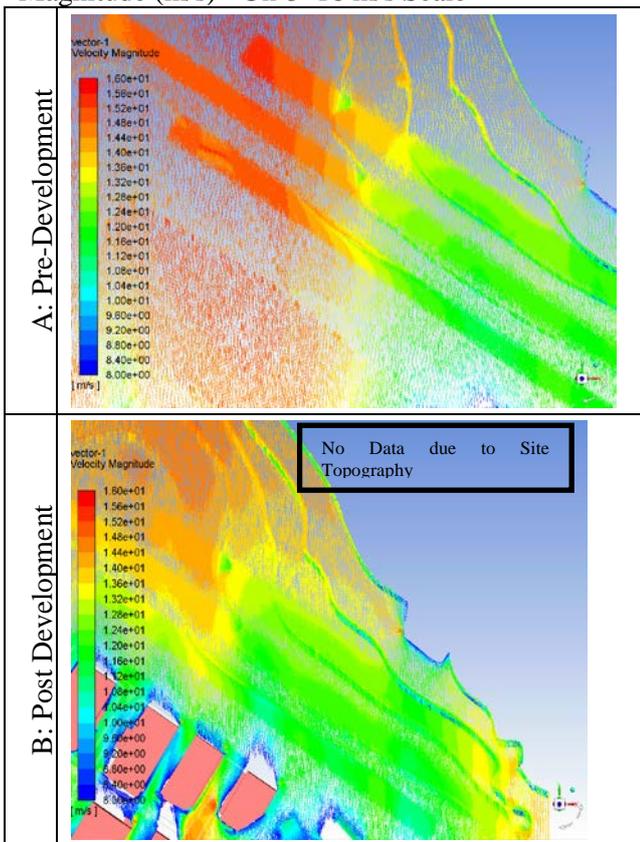
- The CFD model captures the fluid flow characteristics in significant detail.
- Wind is approaching the site from the south at 180° as per the given boundary condition. Wind is then accelerated near the edges and stagnated and recirculated behind the buildings.
- Wind is tunneling through the gap between the proposed warehouses as anticipated
- Means wind speeds are reduced at a number of locations due to proposed trees.
- There is a minor variation in wind speeds along the width of the runways.
- A comparison for the wind shear close to the ground shows that the proposed buildings slightly expand the existing shears upstream the nearest runway at most locations.

Fig. 5 Velocity Vector Coloured by Velocity Magnitude (m/s) (On 0 -15 m/s Scale)



Velocity vectors at the area of interest (Three runways) are shown in Figure 6. Wind speeds below 8 m/s are clipped in Figure 6 so that the wake effect of the post development scenarios can be easily compared with the pre-development scenario. One can see that the airflow pattern at the runways is similar for the pre and post development scenarios. The proposed buildings appear to extend the existing shear at the nearby runway.

Fig. 6 Velocity Vector Coloured by Velocity Magnitude (m/s) - On 8 -16 m/s Scale



In the wind shear and turbulence modelling study, wind speed and turbulence must be reported at all possible critical aircraft trajectories. The approaching angle for landing will be between 2.7 degrees and 4 degrees, with 3 degrees considered as the average. Six possible landing scenarios per runway with an approaching angle of 3 degrees are analysed in this study. Landing paths are shown in Figure 7.

The variation in wind speeds for a selected flight path is shown in Figure 8. The following comments are made with regards to the above graph:

- The graph presents the results at variable height of the aircraft trajectory (3° approaching angle).

- The approaching mean wind speed at 10 m above ground is 25 knots.
- Wind speed increases with height. Higher wind speed is therefore predicted at 30 m above ground (e.g. at position ~1,100 m).
- Highest wind deficit is obtained at the wake of nearby building.
- The wind shear and wind deficit results are proportional.
- The mean wind speed at the ground = 0 (not shown in the graph).

The following major conclusions can be achieved from Figure 8:

- The variation in the mean wind speed is less than 6 knots along the analysed flight path at a height below 60 m.
- The NASF wind shear criterion is therefore not exceeded for the analysed flight path.

In order to assess wind shear criteria exceedance, all critical flight paths and wind directions must be assessed.

Fig.7 Possible Landing Scenarios at 3 Degrees Approaching Angle

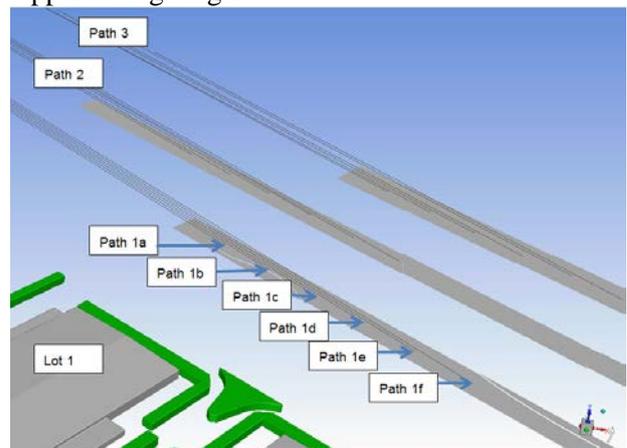
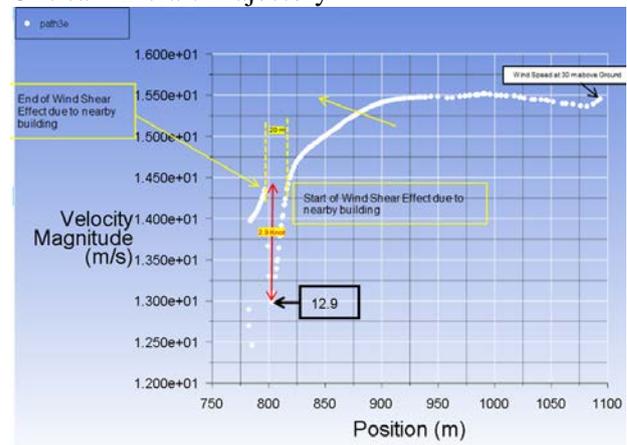


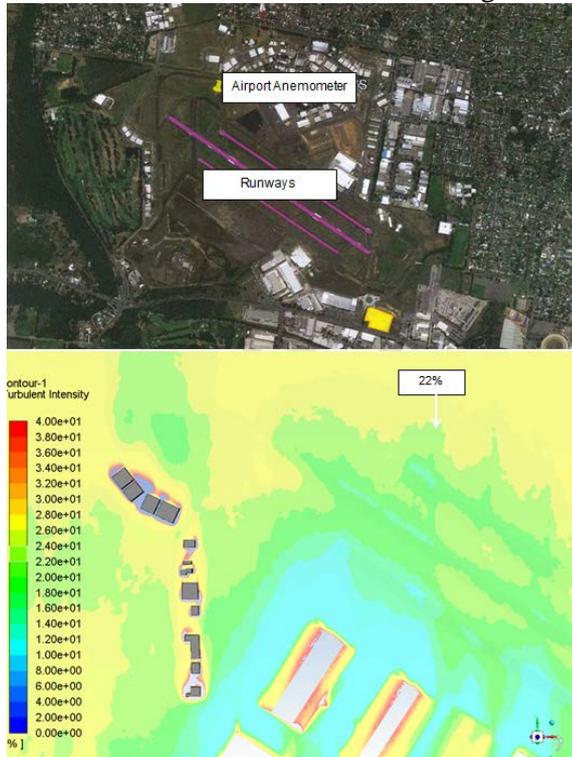
Fig.8 Velocity Magnitude (m/s) along a possible Critical Aircraft Trajectory



3.3.4.2 Turbulence

The turbulence intensity is initially predicted at the site weather station (Refer Figure 9). Figure 9 included an extended computational domain and detailed site topography in the vicinity of the weather station, runways and areas of interest. The predicted turbulence intensity (~22%) using the advanced DES correlates well with the measured intensity (19%). The airport terrain and built environment at the Airport are such that the anemometer siting does not experience localised “interference” caused by the presence of nearby buildings which would significantly distort the wind characteristics recorded by the anemometer. The exposure of the anemometer to southeast winds in particular is very open, with a large stretch of runway area upstream in the southeast direction. This is borne out by the turbulence intensity recorded at the anemometer site, which is fairly constant, at around 0.19, from the south clockwise around to the northeast, and then drops to around 0.14 from the southeast.

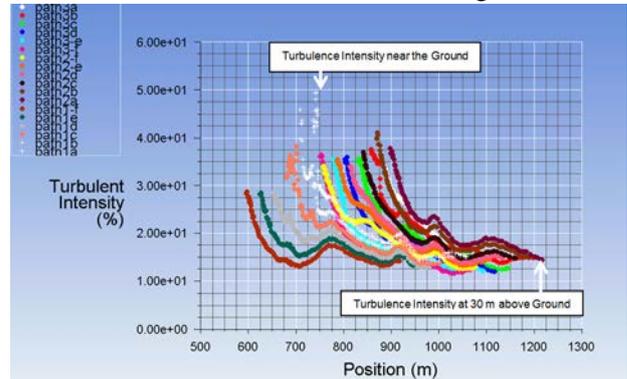
Fig. 9 Predicted Turbulence Intensity at the Airport Weather Station at the Anemometer Height



The predicted turbulence intensity at a number of critical flight paths is shown in Figure 10. The following major conclusion can be reached from Figure 10:

- Highest turbulence intensity occurs near the ground and in the wake of upstream buildings as expected. The mean wind speeds at these locations are generally low.

Fig. 10 Turbulence Intensity along the Aircraft Trajectory at All Runways (Approaching Wind = 25 Knots at 10 m above Ground, Wind Angle = 180°)



The turbulence or root-mean-square (RMS) value along the aircraft trajectory for the flight paths in Figure 11 is calculated based on the wind shear and turbulence intensity results. The following conclusions can be achieved from Figure 11:

- The RMS (standard deviation) is above the 4 kt for an approaching wind of 25 kt at 10 m above ground for the pre-development and post development scenarios.
- In general, the turbulence due to the proposed developed is increased by ~1.2 kt at a number of flight paths.
- Turbulence levels for some flight paths are reduced due to reduced mean wind speed while the increase in turbulent intensity is moderate.

The RMS reduces with reducing the crosswind speeds. Approaching mean wind speed which can generate a 4 kt turbulence can therefore be easily calculated for the pre and post-development scenarios.

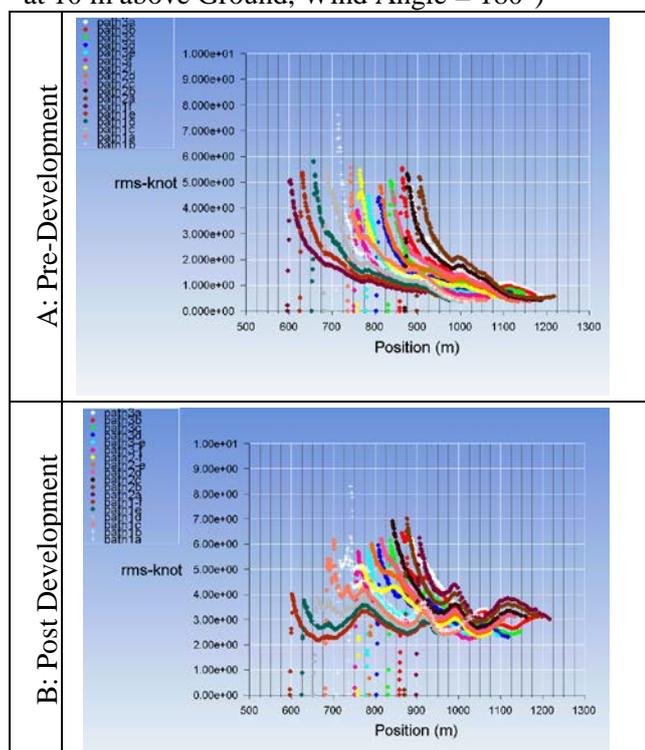
3.3.5 Mitigation Options

Section 3.3.4 provided guidance as to the areas where the wind shear and turbulence acceptability criterion had the potential to be exceeded on nearby runways. Two mitigation strategies are recommended:

- **Building Shape Augmentation:** For the area of adverse wind conditions the following amelioration recommendations are considered:

- Reduce building height (RLs) where possible.
 - Revise aspect ratio (e.g. increase depth and/or reduce width) where possible.
 - Round edges if necessary.
 - Provide or amend landscape and wind screen buffers to reduce wind intensity and variation in mean wind speeds. Trees act as a momentum sink for the mean flow.
 - Provide additional structural on the roof (e.g. PV solar) if required.
- Operational Plan Management: Risk can also be mitigated through operational procedures. However, this involves the approval of various stakeholders including airport operators.

Fig. 11 RMS (Standard Deviation) Value in Knot along the Aircraft Trajectory at All Runways – DES Turbulence Model (Approaching Wind = 25 Knots at 10 m above Ground, Wind Angle = 180°)



4 Conclusion

Hybrid Detached Eddy Simulation (DES) is used to predict wind shear and turbulence caused by buildings in the vicinity of airports. DES is proposed due to ability to combine the benefits of RANS and LES while minimising their disadvantages. The RANS (Realizable k-epsilon in this study) can achieve good prediction for attached boundary layers while LES can capture unsteady motions of large eddies in separated regions.

Complex built environment including site topography, trees and small building features at areas of interest are presented. The paper also discusses some of the parameters that may have influence on the results accuracy.

References:

- [1] H. Krüs, J. Haanstrab, R. van der Hama and B. Wichers, Numerical Simulations of Wind Measurements at Amsterdam Airport Schiphol, *Journal of Wind Engineering and Industrial Aerodynamics*, No.91, 2003, pp.1215-1223.
- [2] P. Chan, W. Lo and D. Leung, Low Level Wind Effects of the Hangars at the Hong Kong International Airport. *The 5th International Symposium on Computational Wind Engineering*, 23 - 27 May 2010, Chapel Hill, NC, USA.
- [3] G., Nieuvoort, de Pins, Wind criteria due to obstacles at and around airports, *Report NLR-CR-2006-261*, National Aerospace Laboratory of The Netherlands, May 2008.
- [4] Managing the Risk of Building Generated Wind shear and Turbulence at Airports, *Guideline B, National Airports Safeguarding Framework, Department of Infrastructure and Regional Development, Australia*, 2018.
- [5] J. Fröhlich and D. von Terzi, Hybrid LES/RANS methods for simulation of turbulent flows, *Progress in Aerospace Sciences*, Vol. 44, Issue 5, 2008, pp. 349-377.
- [6] N. Al-Khalidy, The Role of Computational Fluid Dynamics in Solving Wind Engineering Problems, *3rd International Conference on Mathematics and Computers in Sciences and in Industry*, Crete, Greece, 2016.
- [7] M. Glanville and N. Al-Khalidy, Comparison of CFD Prediction with Wind Tunnel and Full-Scale Measurements for Bounded and Unbounded Wind Flow Scenarios, *5th International Colloquium on Bluff Body Aerodynamics and Applications*, Ottawa, Canada, 2004.
- [8] W. Janssen, B. Blocken and T. van Hoof, Computational Evaluation of Pedestrian Wind Comfort and Wind Safety around a High-rise Building in an Urban Area, *7th Int. Congress on Environmental Modelling and Software*, San Diego CA, USA, 2014.
- [9] B. Blocken; T. Stathopoulos, F. ASCE and J. Carmeliet, Wind Environmental Conditions in Passages between Two Long Narrow Perpendicular Buildings, *Journal of Aerospace Engineering*, Issue 4, Vol. 21, 2008.

- [10] R. Dadioti and S. Rees, Validation of a Detached Eddy Simulation Model in Complex Urban Environment, Athens, *Journal of Sciences*, June 2016.
- [11] P. Spalart, W. Jou, M. Strelets and S. Allmaras: Comments on the Feasibility of LES for wings, and on a hybrid RANS/LES approach, *1st AFOSR Int. Conf. on DNS/LES*, Ruston, LA, 1997.
- [12] P. Spalart, *Young person's guide to Detached-Eddy Simulation grids*, NASA CR-2001-211032, 2001.
- [13] K. Squires, Detached Eddy Simulation: Current Status and Perspectives, *Part of the ErRCO Direct and Large Eddy Simulation*, Vo. 9, 2004, pp. 465-480.
- [14] F. Menter and Y. Egorov, Scale Adaptive Simulation Method for Unsteady Flow Predictions, Part 1: Theory and Model Description, *J. Flow Turbulence and Combustion*, No. 1, Vol 85, 2010, pp.113-138.
- [15] F., Menter, Best Practice: L Scale –Resolving Simulation in ANSYS CFD, Version 2, 2015, ANSYS USA.
- [16] Ansys Fluent Theory Manual, Ansys Inc., 2017.
- [17] J. Frank, A. Hellsten, H. Schlunzen and B. Carissimo, Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment, *Cost Action 732*, 2007
- [18] Best Practices for Large Eddy Simulations (LES), *Ansys Fluent, KR527*, 2010.