Validation of plume dispersion modelling for approval/acceptance against CASA requirements

M. Beyer & C. Stacey
Stacey Agnew Pty Ltd, Darra, Australia

A. Gálffy
Turbulence Solutions GmbH, Vienna, Austria

ABSTRACT: The impact of road tunnel portals and stacks on aircraft safety has not been well treated by the tunnel building industry but an example from the power industry provides support for an effective approach. The Tallawarra B power station was constructed with a plume dispersion device (PDD) on top of the exhaust stack to mitigate updraft velocities that could affect aircraft safety. The resulting plume velocity profile is a non-Gaussian characteristics and therefore a 3D CFD (computational fluid dynamics) analysis was seen to be appropriate to assess the detailed plume behaviour and updraft velocity. The CFD modelling approach and results were accepted by the planning department following review by the Civil Aviation Safety Authority (CASA) and the stack operation with the PDD was approved, with the proviso that the updrafts be measured in operation. The CFD results are compared to on-site measurements of the exhaust plume after project completion. Within the slight variation of meteorological conditions between the model and the measurements, the shape of the plume and the magnitude of the updraft velocities resulting from the model compare reasonably well with the measured values, justifying our approach to assessing the simpler case of road tunnel stack impacts on aircraft safety.

1 INTRODUCTION

In unidirectional urban road tunnels with high traffic, the vehicle piston effect can induce high portal outflows with temperatures considerably above the ambient temperature. That buoyant plume, either released via the portal or partly via stacks, could lead to updraft velocities affecting aircraft at relevant flight protection surfaces in the proximity of the portals or stacks. Therefore, appropriate modelling should be done to inform project decisions at early stages.

The impact of road tunnel portals and stacks on aircraft safety has not been well treated by the tunnel building industry. All recent Australian road tunnel stacks (up to 2023) where aircraft upset is relevant have been designed and sized by project managers, who relieved engineers of the task. That is; the designs were based on grossly conservative rules of thumb, that could get procedural approval without further attention. No cost comparisons have been done, but the cost for such portal plants is tens of millions of dollars, and in many cases, the majority of that could have been saved by engineers applying their professional skills, with no greater approval risk.

Stack plumes causing flight interference are not unique to road tunnels. More detailed work has been conducted by the power industry where plume buoyancy and vertical velocities can be significantly greater. Recent work associated with the Tallawarra B power station provides a useful and instructive case study of a CFD modelling approach and plume measurement that is relevant to road tunnels and portals.

Tallawarra B power plant near Shellharbour City in New South Wales operates an open-cycle gas turbine (OCGT). The gas turbine is designed for fast-start and can attain full load within 30 minutes, to supply power to the grid once there is high demand in the National Electricity

Market (NEM). The OCGT unit produces a large flow of hot exhaust gas, that rises quickly into the atmosphere, generating updraughts with potentially high vertical velocities and turbulence up to elevations that are likely to adversely affect aircraft flying in the vicinity of the exhaust stack. To reduce the vertical velocity and buoyancy of the plume, a plume dispersion device (PDD) was mounted on top of the stack, splitting the exhaust into twelve equal parts, each discharging nearly horizontally and in different directions from a central point. Since the PDD emits the ‘plumelets’ nearly horizontally, the initial vertical plume momentum is eliminated. In addition, the entrainment of ambient air into the twelve parts of the plume is enhanced. This greatly reduces overall plume acceleration, resulting in reduced vertical plume velocity.

The plume behaviour initiated by the PDD is complex and cannot be characterised accurately by a Gaussian plume model or other screening model. Instead, a CFD analysis was seen to be appropriate to assess the detailed plume behaviour and height-dependent updraft velocities during the design and approval stages of the power station’s development.

After completion and commissioning of the Tallawarra B power station, on-site measurements of the detailed plume characteristics and updraft velocities were conducted with a high wing Cessna 172 equipped with inertial sensors and a tailored differential pressure sensor probe mounted on the strut. The on-site measurements were compared with the CFD model results to evaluate the appropriateness of the model assumptions and to confirm that the power station’s approval condition, as defined by the Civil Aviation Safety Authority (CASA), regarding updraft velocities, is met.

2 CASA’S PLUME RISE ASSESSMENT AND CRITERION

2.1 *Overview of assessment process*

CASA’s Advisory Circular (Civil Aviation Safety Authority, 2023) provides information about assessing the potential impacts of exhaust plumes on aircraft safety. If the stack exit velocity exceeds 4.3 m/s, several exhaust plume rise assessment steps are required. According to (Civil Aviation Safety Authority, 2023), that velocity criterion relates to a level of turbulence in the plume that may be capable of affecting normal flight.

CASA via (Civil Aviation Safety Authority, 2023) describes the plume rise assessment process, which firstly requires a proponent of a new activity that may generate an exhaust plume to submit details of the exhaust plume to CASA if the exit vertical velocity exceeds 4.3 m/s. If the stack exit velocity is higher than 4.3 m/s, CASA uses these details to conduct a screening assessment. If that screening assessment confirms that the plume velocity will exceed 4.3 m/s at or above a flight protection surface, the impact of the exhaust plume is to be further examined by CASA using the Exhaust Plume Analyzer (EPA). The EPA is a model developed by MITRE in the US (Gouldey, et al., 2012).

If the exhaust plume violates certain assumptions related to the EPA model (e.g., non-identical stacks with non-aligned and non-uniform spacing, stacks with engineered modifications such as the use of a PDD), CASA via (Civil Aviation Safety Authority, 2023) suggests using CFD modelling or scale modelling to assess the plume behaviour. While this all sounds like risky/complicated work, and therefore may frighten project managers, seeking to avoid it by limiting efflux velocity to 4.3 m/s not only gives bad project outcomes, but also does not guarantee that the updraft velocity stays below 4.3 m/s due to the usual expected buoyancy of the plume once the exhaust left the stack (see next Section).

2.2 *Critique of vertical velocity criterion*

The application of the vertical velocity criterion of 4.3 m/s at stack exit does not necessarily ensure that the vertical velocity induced by the plume remains below 4.3 m/s at higher elevations. A plume with stack exit velocity at or below 4.3 m/s, can accelerate if the initial buoyancy of the plume, together with the meteorological conditions (ambient temperature, wind speed and atmospheric stability) causes a density deficit and consequently results in a buoyancy head that is higher than the dynamic head of the stack exit velocity. Thus, looking only at a low stack exit velocity is not appropriate, as it does not provide sufficient information on whether the updraft velocities might affect normal flight.

As an example, vehicle 'piston' effects in a three-lane unidirectional road tunnel can generate a mass flow rate of up to 1000 kg/s with a portal exit temperature of up to 20°C above ambient. The flow coming out of that tunnel portal would have a thermal power, relative to ambient, of approximately 20 MW. Such tunnel portal flows with almost zero initial vertical velocity would bend upwards and generate updraft velocities with peak values that have the potential to exceed CASA criteria. Releasing part of the tunnel flow via portal exhaust stacks with an average exit velocity of 4.3 m/s and an exit temperature of 20°C above ambient has also been found to generate averaged updraft velocities in the plume that may affect aircraft safety.

A plume rise velocity criterion in general is also not sufficient, as the shape of the vertical velocity profile (velocity gradients across the plume) and the aircraft characteristics (wingspan, weight, speed, etc.) are the essential parameters to understand potential impacts on aircraft. The problem is not as simple as any vertical velocity measure. It is recommended that the plume characteristics be analysed and then used to simulate/calculate the aircraft response (for site-typical aircraft types) based on the plume data at relevant elevations.

2.3 *MITRE Exhaust Plume Analyzer (EPA)*

For typical tunnel stack applications, once the initial assessment and the screening tool indicates that the plume velocity will exceed 4.3 m/s at or above a flight protection surface, CASA will conduct a detailed assessment using MITRE's EPA (Civil Aviation Safety Authority, 2023). However, there are several limitations and problems with the EPA by MITRE (Gouldey, et al., 2012). Firstly, the mean flow behaviour of the plume is estimated by the Spillane model. The Spillane model has no ability to calculate the Zone of Flow Establishment (ZOFE). The ZOFE is from stack outlet up to 6.25 times the stack diameter if there is no cross wind. Flow calculations in the Spillane model start above that zone. Flow within that zone is relevant if the flight protection surface is close to the stack outlet (e.g. for a stack diameter of 8 m the ZOFE is 50 m high).

In addition to predicting potential aircraft upset conditions, the MITRE EPA outputs the probability of G-loads occurring for different aircraft types depending on the stack flow parameters and a set of meteorological conditions. The probability calculation is mainly driven by a gust probability function but besides that, it depends only on the meteorological data (probabilities of stack operation, of aircraft flying directly over the plume etc. are not considered). The gust probability function applies an additional updraft velocity component on the mean updraft centreline velocity obtained from the Spillane model, which should account for turbulence related velocity fluctuations.

This is an improper assumption as the velocity fluctuations assumed in the MITRE EPA relate to small scale turbulent fluctuations obtained from measurements (Papanicolaou & List, 1988) and not to large scale atmospheric turbulence that may affect aircraft. The small-scale turbulent gusts considered in MITRE's EPA have negligible / no impact on flight dynamics, as they are small in scale, and get averaged along the wingspan. As the applied gust probability function in MITRE's EPA significantly increases the updraft velocity (factor of 1.75 or more (Gouldey, et al., 2012)), the method grossly overestimates the potential aircraft impact and resulting G-force. This, in combination with simplifications in the calculation of the maximum load factor (e.g., neglecting pitch response, neglecting lateral wing averaging effects etc.) leads to an overall load factor overestimation by a factor of around two. Consequently, the MITRE EPA can lead to incorrect predictions of non-compliant turbulence intensities and G-forces (see tables and description in (Civil Aviation Safety Authority, 2023) for further details on turbulence intensity probabilities and their acceptance). Even if the plume rise velocity drops below 4.3 m/s MITRE's EPA would conclude non-compliant turbulence intensities. That is seen to be a contradiction, as CASA's Advisory Circular states that for velocities at and below 4.3 m/s, no further assessment steps are required. The MITRE EPA includes a cutoff mean velocity, normally set at 4.3 m/s in Australia, below which the results are truncated and ignored. This appears to be a fix for the contradiction caused by the EPA model indicating unacceptable impacts at well below that velocity level. Such a truncation is seen to be necessary due to the underlying formulation issues and subsequent overestimation of the G-force probability. The alternative would be accepting the MITRE EPA results with no cutoff, and then very low updraft velocities could still lead to suggestion of unacceptable impacts. Applying a cutoff velocity to restrain the EPA prediction to be consistent with CASA's lower limit of concern (CASA are not concerned below 4.3 m/s) is hard to justify. If it is accepted that there is no problem below 4.3 m/s then the MITRE EPA should not be predicting high aircraft upset probability at much lower velocities, so a fix of the underlying formulation issues should be sought. For all stack applications including tunnel applications, it is recommended that a detailed assessment be performed as explained in Section 3 and suggested in Section 6. For some minor circular stacks, a Gaussian plume analysis might be sufficient for predicting updraft velocities. A simple method for large stack flows (e.g. cooling towers) or complex stack geometries does not currently exist.

2.4 *Criterion for CFD plume modelling*

For the Tallawarra B power station with PDD, CFD modelling was seen to be appropriate for assessing the vertical velocity of the exhaust plume. While assessing the detailed plume characteristics and the resulting aircraft response is more realistic and informative than either the EPA or Gaussian models, the criterion for plume modelling according to (Civil Aviation Safety Authority, 2023), and the Tallawarra project's requirement, was to not exceed an average plume velocity (top hat profile velocity) of 6.1 m/s (CASA's criterion for CFD assessments) at and above the relevant flight protection surface.

If the vertical velocity profile has a Gaussian distribution, the average plume rise velocity ('top hat' profile velocity) would be half of the maximum plume rise velocity (peak plume rise velocity). However, the velocity profile of a plume released by a PDD does not have a Gaussian distribution. As the real velocity profile differs from a Gaussian distribution, the equivalent top hat velocity will no longer necessarily be half of the maximum velocity. The averaging area is also uncertain, with the plume edges being indistinct. Notwithstanding this, CASA stated in the context of Tallawarra B, that it will accept half of the maximum CFD-derived plume rise velocity as the average plume velocity at the flight protection surface (700 ft above mean sea level), for the purpose of comparison with CASA's criterion of 6.1 m/s. Consequently, in the context of the peak plume rise velocity determined using a CFD model for Tallawarra B power station, CASA's criterion is effectively a peak velocity of 12.2 m/s at 700 ft.

3 CFD PLUME MODELLING OF TALLAWARRA B POWER STATION

Tallawarra B Power Station plume modelling was carried out using the CFD-software ANSYS Fluent (Release 2020 R1). The overall domain size that was adopted was, in plan, 3,700 x 3,700 m, with a height of up to 3,000 m, to capture the entire plume flow without adverse influences from the domain boundaries. The stack was positioned approximately in the middle of the domain. The PDD is mounted on top of the stack and has in total 12 openings all at the same level evenly distributed around the circumference to release the exhaust jets alternately horizontally and slightly downward.

Two wind cases were adopted for the CFD modelling as depicted by Figure 1. Firstly, wind conditions were generated by Katestone using The Air Pollution Model (TAPM), covering a 5-year period. The specific conditions that resulted in the 99.9th percentile plume average vertical velocity at 700 ft were selected during the design phase as the first wind case for CFD modelling. Following completion and commissioning of the power station and following collection of the on-site measurements, a further wind case that was obtained during the measurements was also used for the purpose of detailed model validation (see Section 5).

The stack flow parameters that were used in the design phase CFD modelling (760.9 kg/s and 639.3°C) are slightly different to those recorded by the Tallawarra B control team during the on-site measurements (756.35 kg/s and 644.2°C). The design stack flow parameters were used for the initial CFD model and the measured data were used for the post measurement validation case.

The buoyant plume behaviour was found to be transient and to be quite sensitive to ambient conditions. After establishing the flow field and refining the initial mesh to the desired final mesh resolution, the flow field was finally simulated with a transient solver and the transient behaviour was monitored during the simulation over a period of about 25 real minutes. To capture the worst conditions, the detailed plume parameters were then analysed at a time when the highest vertical plume velocity arose at the monitored elevations. The initial CFD modelling showed that the

PDD effectively disperses the plume, so that the updraft velocity was close to, but below, CASA's Critical Plume Velocity.

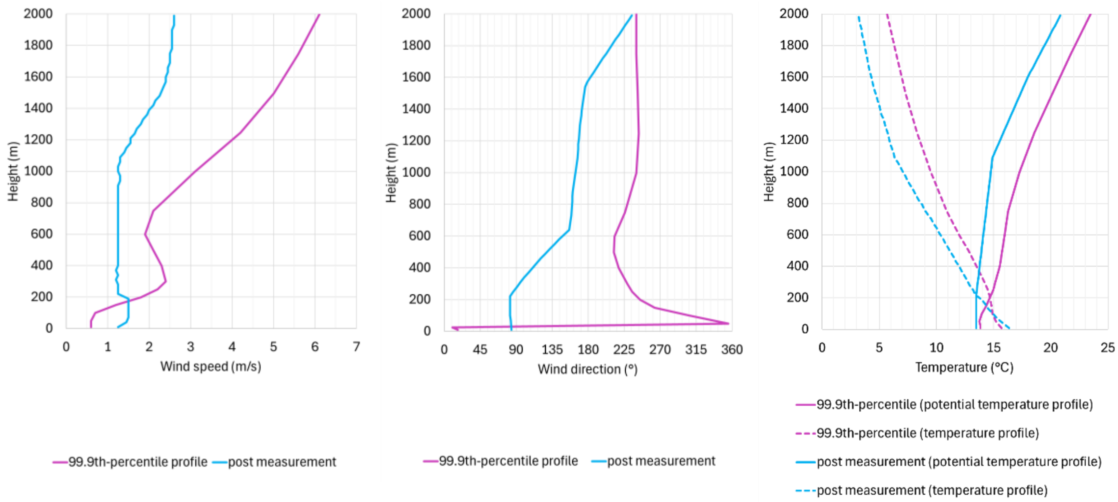

Figure 1. Profiles of wind speed (left), wind direction (middle) and temperature (right) provided by Katestone and used for the assessment.

4 ON-SITE PLUME MEASUREMENTS

The applied measuring principles and the detailed plume velocity measurement techniques were developed and performed by the Austrian company Turbulence Solutions GmbH[1]. The measurement campaign was a complex undertaking that required the expertise and assistance of different disciplines and experts and was accompanied and supported by Stacey Agnew[2], Airspeed Aviation[3], Katestone[4], Aviation Projects[5], as well as the Tallawarra B operating and project teams from Energy Australia[6]. The success of the measurement program is down to the innovative technology and expertise provided by Turbulence Solutions and the highly professional collaboration, initiative and input of the project team and all parties involved.

4.1 *Measurement principle and program*

A Cessna 172 hired by Energy Australia and operated by Airspeed Aviation was equipped with a self-contained measurement probe mounted on the strut (see Figure 2) and inertial sensors mounted in the cockpit at the centre of gravity. The flow sensor probe measured the flow field based on differential pitot pressures in three different directions, while the flight dynamics impact was measured by the inertial sensors in the cockpit as well as by acceleration and gyroscope sensors in the probe mounted on the strut. Barometric pressure and GPS signals were also logged by sensors in the cockpit. By measuring the motion of the aircraft in combination with the flow

[1] Turbulence Solutions is developing an innovative aircraft stability product to limit turbulence effects on aircraft and, in doing so, has world-leading expertise in the measurement of atmospheric turbulence, and the effect that turbulence has on aircraft.
[2] Stacey Agnew accompanied and coordinated the measurement program in detail
[3] Airspeed Aviation operated Cessna 172 and provided pilots with experience in flight instruction and aerobatics.
[4] Katestone provided expertise and processed meteorological data and forecasts from Meteomatics.
[5] Aviation Projects accompanied and coordinated measurement program from aviation safety perspective (selecting aircraft operator, organising Temporary Danger Area, NOTAM preparation, liaise with aviation authorities and local pilots etc.).
[6] Energy Australia initiated the project and coordinated overall project, companies, teams and people involved.

field and the sensor motion, it is possible, via deconvolution, to extract the flow impact on the measurement probe due to the aircraft motion, and account for that to obtain the flow field and turbulence in the atmosphere, or in this case the exhaust plume. The probe samples data at 500 Hz and the cockpit logger at 200 Hz. The result is a high-rate sampling of the flow the aircraft is flying through (500 Hz, one measurement every 2 ms, which is equivalent to 0.1 m sampling at a flight speed of 50 m/s). Further information on the measuring principle and the flow sensor can be found in (Gálffy, et al., 2021).

Figure 2. Self-contained flow sensor probe mounted on the strut (left and middle picture) and inertial sensors mounted in the cockpit (right picture)

Figure 3. Photos taken during the plume surveys on 5th of Aug. 2024 (left) and 7th of August 2024 (right)

To limit the risk to the measurement team and the experienced pilots, the expected aircraft response was assessed based on flight dynamics simulations of a Cessna 172 using the flow field obtained from the CFD simulations. This pre-assessment confirmed that the expected G-load impact and roll response were within the range of normal and safe aircraft operation. Further precautions were undertaken to reduce traffic interactions with measurement flights by arranging a Temporary Danger Area (TDFA) and a Notice to Airmen (NOTAM) issued on each day prior to test flights commencing.

An hourly average weather forecast for the local project area out to 7 days was provided four times a day by Katestone, with model data from Meteomatics[7]. The local weather forecast allowed the project team to prepare test flights and plant operations to coincide with upcoming weather conditions that were conducive to elevated vertical plume velocities (low temperature, low wind speeds, stable/neutral atmosphere and comply with Visual Meteorological Conditions minima).

In total, it was possible to conduct 2 hours and 45 minutes of calibration flights and 8 hours and 35 minutes of exhaust plume survey flights, with approximately 430 individual plume fly-throughs at different elevations (from 500 ft up to 2,600 ft) as depicted in Figure 4. Figure 3 shows photos from the aircraft flying over the stack while it is taking measurements.

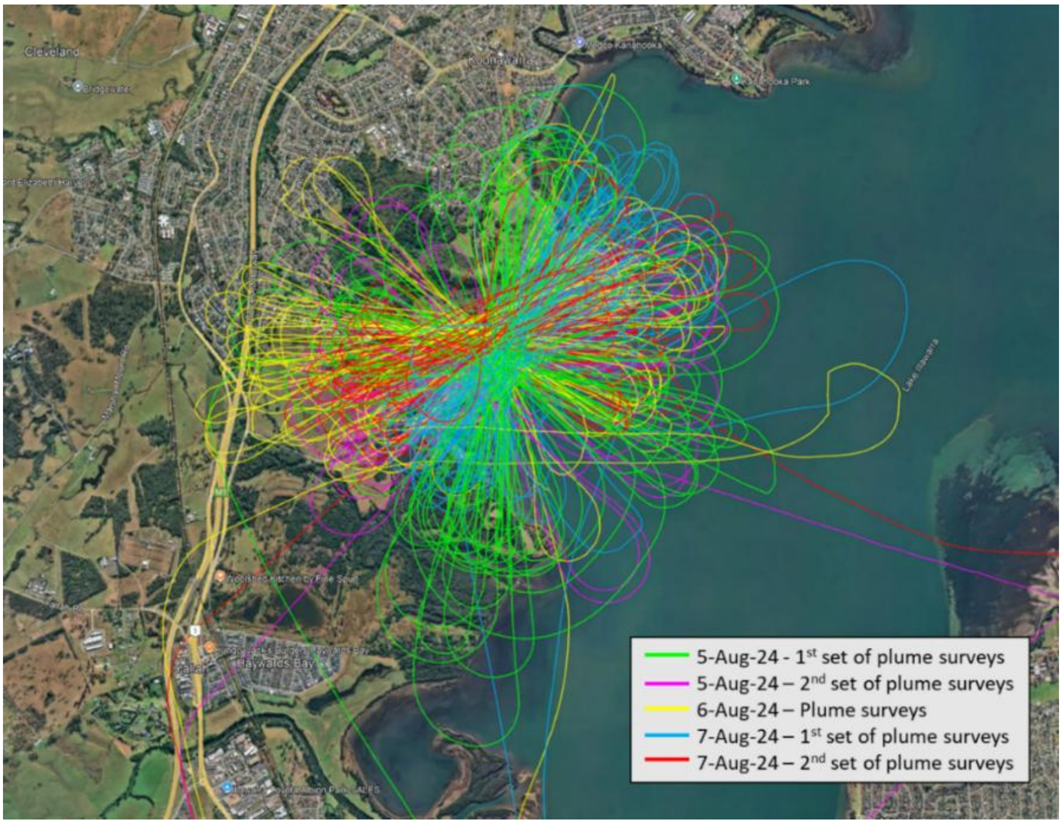

Figure 4. Figure-eight manouvres and tracks through the exhaust plume based on recorded GPS data for the individual test days and plume surveys.

4.2 *Measurement results*

Compared to the simulation results, the maximum peak plume rise velocity was slightly higher (orange squares), which is likely related to the slight differences in the meteorological conditions between simulation and observed during that test day. Of particular note, on 7-Aug-2024 between 9 am and 10 am, the wind direction changed from the west with a speed of about 1.5 m/s to from the east with a speed of about 1.9 m/s. During this wind direction change, the wind speed approached values close to zero, whereas the wind speed in the CFD model (99.9-percentile wind case) was always above 0.5 m/s.

[7] Enhanced downscaled model data based on the European Center for Medium-Range Weather Forecasts' (ECMWF) Integrated forecasting System (IFS),. The downscaling improves the coarse grid native representation down to a resolution of 90 m. This is achieved by applying high-resolution land usage data, soil, terrain data, astronomical computations & other sources.

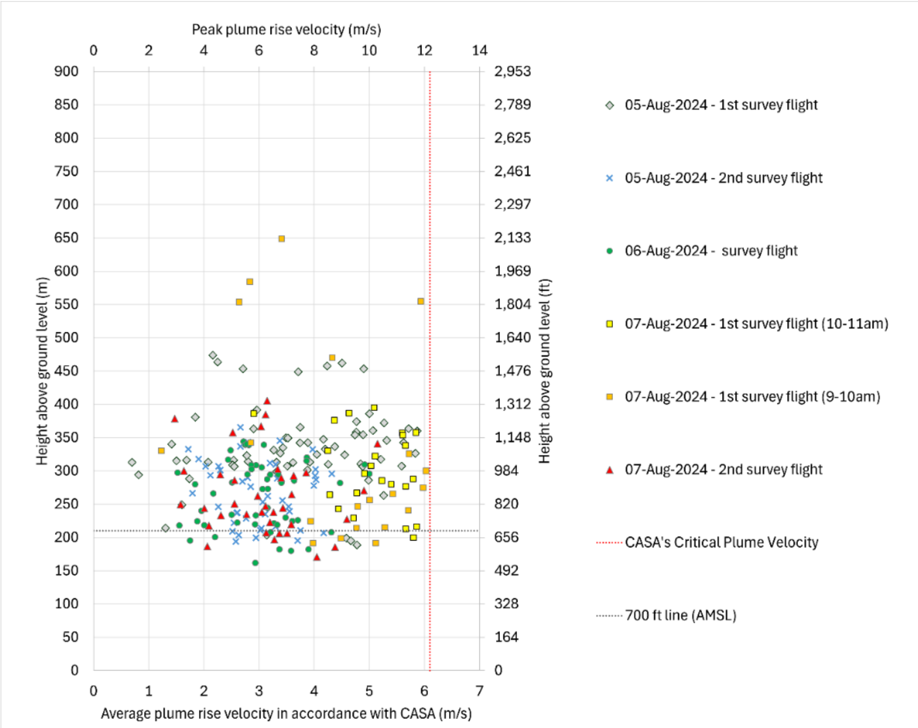

Figure 5. Highest recorded plume rise velocity per fly-through and elevation for the individual test periods. The primary x-axis at the bottom is the average plume rise velocity in accordance with CASA's criterion which is half of the peak plume rise velocity shown on the secondary x-axis at the top of the plot.

5 CFD PLUME MODEL VALIDATION

As noted in Section 3, the initial CFD model with the design wind case (99.9th percentile case) was run again with a representative wind case as obtained during the measurement campaign, to confirm observations and to validate the model assumptions and technique. For that, the profile with the lowest wind speed, was taken as new input, in combination with the lowest temperature observed during the plume surveys. Accordingly, the values of the two profiles 7-Aug-2024 at 10 am and 7-Aug-2024 at 11 am were averaged to obtain representative conditions over that measurement period. Stack flow parameters as discussed in Section 3 were also adjusted to reflect the post construction site conditions.

After establishing the flow field and refining the initial mesh to the desired final mesh resolution, the flow field was finally simulated with a transient solver over a period of 25 real minutes. Even at constant meteorological conditions, the buoyant plume showed a transient behaviour with a peak updraft velocity variation of about 3 m/s at one elevation. However, the wind conditions usually vary with time and do not remain constant for long periods. That leads to a much bigger variation in the real plume shape and exhaust plume velocities, compared to the simulation with the constant meteorological conditions. This circumstance makes it hard to compare the shape and peak of velocity profiles of individual plume passes with the simulation results. Further, the meteorological data are supported by weather stations but are based on weather modelling to obtain the local prevailing weather conditions during the measurement period. The modelled weather data are based on hourly averaged values and, therefore, detailed information about the meteorological conditions between any two hourly averaged datasets is not available. During the relevant plume survey flight (7th of August 2024, 1st survey flight between 10 am and 11 am), the wind conditions will have been changing and so, even if both measurements and simulations were 'perfect', the results from the measurement may not compare perfectly with the simulation results where the wind conditions were constant.

To evaluate the CFD plume profile at different elevations and time steps in an effective and uniform way, virtual flight trajectories were placed through the centre of the stack and were rotated in 5° intervals as illustrated by the contour plot in Figure 6.

The plot in Figure 6 compares the plume shapes obtained from the CFD simulation at stack-centred trajectories (blue lines) with the plume shapes obtained from the comparable measurement fly-throughs (red lines) at similar elevation. Measured plume profiles with an elevation close to the evaluated elevation have been bundled together in one plot.

In general, the maximum plume rise velocities obtained during the relevant test flights are about 1 m/s higher than from the CFD results with the 'post measurement' wind condition. During the plume survey on the 7th of August between 10 am and 11 am (red lines in Figure 6), the wind speed changed from 0.8 to 1.9 m/s below 400 m whereases the averaged wind speed applied in the CFD model was about 1.5 m/s below 400 m. This difference in the low-level wind speed could be the reason for the slightly lower maximum peak plume rise velocities predicted by the CFD.

Considering uncertainties in the applied modelling and measurement methodologies, as well as in the exact meteorological conditions during a plume pass, the simulation results correlate well with the measurement results and give confidence that the applied CFD modelling methodology can be used for estimating/assessing exhaust plume rise velocities with sufficient accuracy.

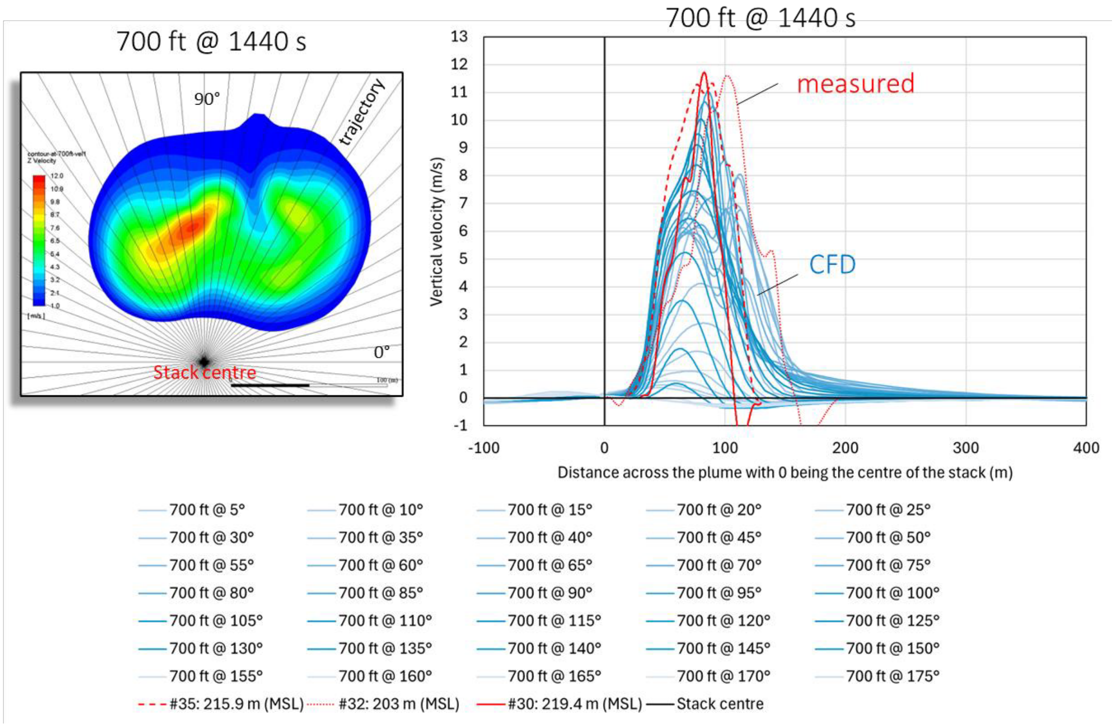

Figure 6. Trajectories through the plume in 5° intervals with the stack centre being the origin shown on a contour plot at 700 ft at a simulation time of 1440 s (colour depicts the updraft velocity and is clipped to 1 m/s). Plot on the right shows plume profiles obtained from the CFD simulations at 1440 s based on the stack-centred trajectories (blue) compared to the measured plume profiles as sampled on the 7th of August 2024 during the first plume survey flight between 10 am and 11 am at around 700 ft (red).

6 ROAD TUNNEL APPLICATIONS

Managing portal emissions of polluted air from road tunnels by exhausting parts or all of it via stacks can lead to a buoyant plume that potentially affects aircraft during normal flight. Even if the average stack exit velocity is below 4.3 m/s and passes the CASA criterion for not undertaking further assessments (Civil Aviation Safety Authority, 2023), the density deficit in the plume can cause an updraft velocity that exceeds the initial average stack exit velocity, and may exceed

CASA criteria. In such cases, it is recommended that a detailed CFD analysis be undertaken based on the methodology proposed in Section 3, followed by simulation/calculation of the aircraft response based on the plume data obtained from the CFD at relevant elevations. A detailed CFD assessment should be preferred, as flaws in the MITRE EPA lead to an overestimation of the impact/G-force (see Section 2.3). Initial plume assessments and analysis of resulting aircraft response is seen as essential to inform the early stages of tunnel projects with portals near airfields, in establishing and planning for appropriately economical mitigation measures and to reduce the project risk to stakeholders.

Given below is an example of a unidirectional 3-lane road with a tunnel air/exhaust mass flow rate of approximately 1000 kg/s (induced by the vehicle piston effect) and a temperature of approximately 20°C above the ambient temperature outside the tunnel.

By using the same model methodology as noted in Section 3, Figure 7 shows an example of a plume generated by a 100% portal flow out of a unidirectional tunnel (left) and generated from four stacks capturing 80% of that portal flow (middle). Both simulations were carried out in still air (no wind), which is the worst-case scenario in terms of maximum updraft velocities. The right plot in the figure shows the maximum updraft velocities as a function of height for the two example cases. Even if the stack exit velocity is already high (around 10 m/s) the plume further accelerates due to the density deficit and resulting buoyancy force, up to a peak updraft velocity of approximately 12 m/s. If the exit velocity were lower (e.g. 4 m/s) the final peak updraft velocity would end up at a similarly high value. In case of the 100% portal flow, the peak updraft velocity reaches approximately 4 m/s, with the potential to be higher if the exit gallery geometry did not cause the plume to be split into two parts (causing higher entrainment and lower buoyancy).

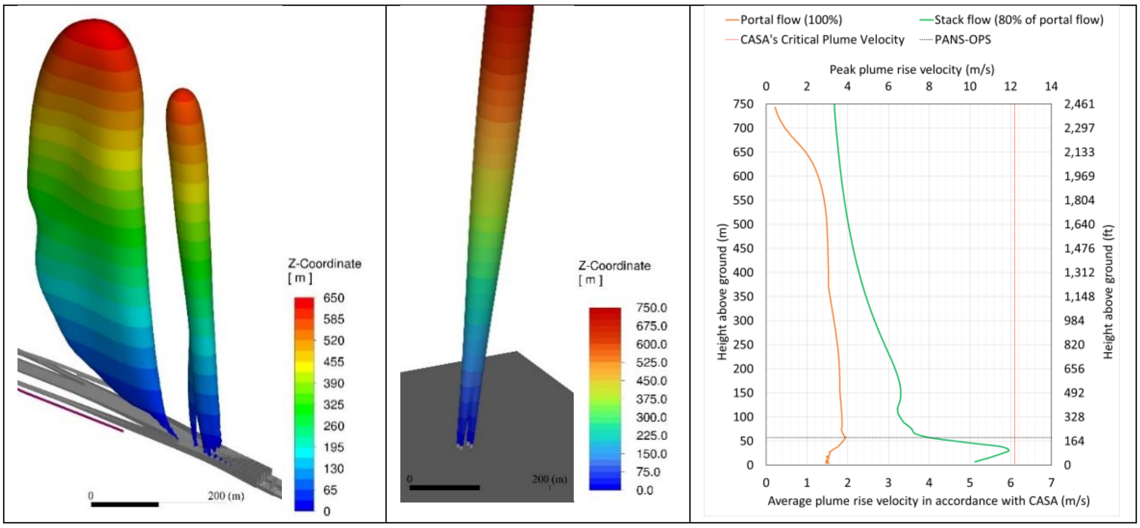

Figure 7. Iso-surface of constant vertical velocity of 2 m/s for portal emissions (left) and 3 m/s for stack emissions (middle). The plot on the right shows the maximum plume rise velocity as a function of height for the two example cases (left: 100% portal flow, middle: 80% of portal flow extracted via stacks). Colouring of the envelopes depicts the plume height.

Figure 8 shows the velocity profile generated by the four stacks at the PANS-OPS surface with the worst flight paths for the load factor impact (blue) and roll angle impact (red). Putting aside MITRE's EPA due to the flawed turbulence modelling, but using the aircraft parameters contemplated by the MITRE EPA model (weight of 1,250 kg, 10.2 m wind span and aircraft speed of 37 m/s), the resulting G-load and roll response is 0.5 g and 29°, respectively, when flying through the example plume generated by the stacks at PANS-OPS (about 50 m above ground level). To put that impact in context, CASA's Advisory Circular (Civil Aviation Safety Authority, 2023) provides a table with expected G-loads depending on the turbulence intensity and defines a G-load between 0.15 and 0.49 as light, between 0.5 and 0.99 as moderate and between 1.0 and 1.99 as severe turbulence intensity. Depending on the flight protection surface definition, the threshold is to not to encounter moderate turbulence intensity more often than once per 10^5 operations. With the probability covered by considering worst case flight path calculations, and the G-load still within the moderate band, in this example, the CASA criterion would be met.

Using aircraft parameters of the smallest aircraft that can be expected on airports (weight of 750 kg, 8 m wind span and aircraft speed of 50 m/s to 30 m/s) the resulting G-load and roll response would be 0.85 g and 40°, respectively. This result is in CASA's moderate turbulence range and so more work would need to be done on probability for such very light aircraft.

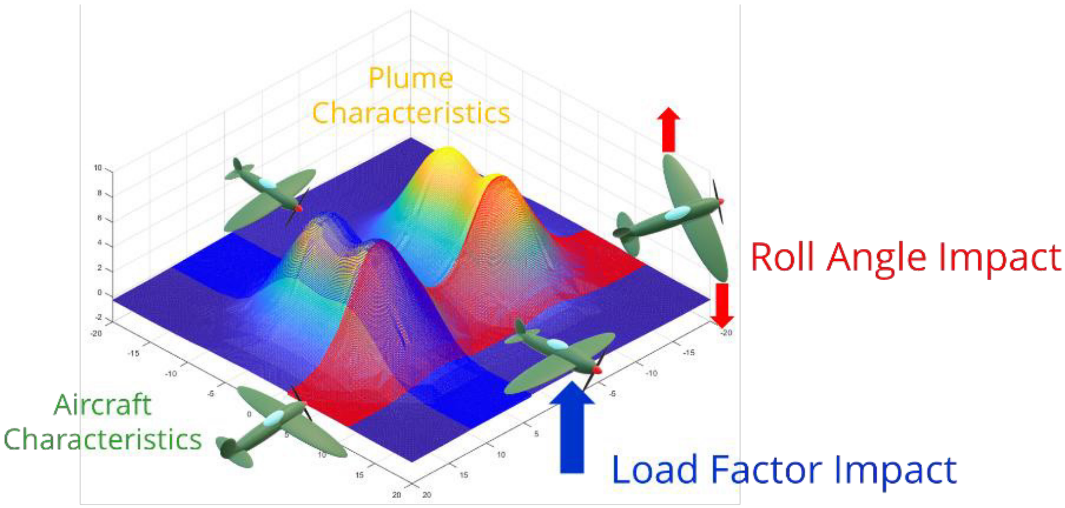

Figure 8. Velocity profile generated by the four stacks at the PANS-OPS surface with the worst flight paths for the load factor impact (blue) and roll angle impact (red). The vertical axis is the plume rise velocity (in m/s) and the horizontal axes are the horizontal distances in the x- and y-directions (in m).

ACKNOWLEDGEMENT

The authors express their gratitude to EnergyAustralia (EA) Tallawarra Project for their invaluable support and contribution to this paper. The development and operation of the Tallawarra B Power station, the subject of this study, was made possible through the dedication and commitment of EA, with significant support and funding provided by the New South Wales Government. Their professional collaboration and transparency have been critical in facilitating the research and analysis presented herein. We acknowledge the meteorological input that was essential to the project, and paper review comments, by Simon Welchman and Frank Quintarelli of Katestone Environmental, noting that we have different views on some of the content. We also acknowledge the crucial contribution of Airspeed Aviation and Aviation Projects. Their expertise and involvement were instrumental in successfully carrying out key aspects of the project, and their highly professional collaboration was greatly appreciated throughout the process.

REFERENCES

Civil Aviation Safety Authority. 2023. *Advisory Circular AC 139.E-02v1.0 - Plume rise assessments.* s.l.:Australian Government Civil Aviation Safety Authority, https://www.casa.gov.au/sites/default/files/2023-03/advisory-circular-139e-02-plume-rise-assessments.PDF.

Gálffy, A. et al. 2021. Turbulence load prediction for manned and unmanned aircraft by means of anticipating differential pressure measurements. *CEAS Aeronautical Journal,* Volume 12, https://link.springer.com/article/10.1007/s13272-021-00512-y.

Gouldey, D., Hopper, J. & Schwalbe, J. 2012. *MITRE - Expanded Model for Determining the Effects of Vertical Plumes on Aviation Safety,* s.l.: MITRE, http://www.ctcombustion.com/oxc/20130730-FAA-FOIA-Response/MITRE/September%20Plume%20Report%20Final%5B1%5D.pdf.

Papanicolaou, P. N. & List, E. J. 1988. Investigations of round vertical turbulent buoyant jets. *J. Fluid Mech.,* Volume 195, pp. 341-391, https://doi.org/10.1017/S0022112088002447.