Carbon Monoxide Dispersion in Enclosed Car Parks: Pollutant Source Modelling Methods

Jason Gaekwad[1]
[1]Mott MacDonald, Melbourne, Australia

Abstract

Adequate ventilation of enclosed car parking areas is required to ensure concentrations of vehicle exhaust pollutants (e.g. Carbon Monoxide) do not reach a level high enough to threaten the safety of car park occupants. With the development of accessible Computational Fluid Dynamics (CFD) tools, car park ventilation design is shifting from a code-based approach to a performance-based approach, with CFD methods used to confirm the safety of the proposed design. A comparison of three pollutant source modelling methods for underground car parks has been conducted using CFD methods. The modelling shows that pollutant concentration results vary considerably (maximum concentration variation of over 24 times) between modelling methods. This work represents an initial study into underground car park pollutant source modelling methods with the view of improving the safety and accuracy of performance based underground car park ventilation designs.

Introduction

Underground car parks represent a unique case where humans may be present for extended periods, ventilation is limited, and production of toxic pollutants such as carbon monoxide is high. Ventilation design of these areas is therefore critical to occupant safety. Numerical methods such as Computational Fluid Dynamics (CFD) are increasingly being used to confirm the effectiveness of car park ventilation designs however CFD methods may be sensitive to boundary conditions. In the case of car parks, pollutant concentration results may be sensitive to the method by which the pollutant source is modelled.

This paper studies three pollutant source modelling methods in order to quantify the sensitivity of pollutant concentration results to pollutant source modelling methods. A technical background is provided, outlining the methods of production and control of vehicle produced carbon monoxide (CO) as well as existing studies into modelling vehicle pollutant sources. A fundamental purpose of the research is then established, followed by a discussion of the CFD modelling methodology. Finally, the results are presented and discussed.

Background

Carbon monoxide

Carbon monoxide is a colourless, odourless, flammable gas, toxic to humans. The gas is a product of fossil fuel (hydrocarbon) combustion, refer equations (1) and (2) from Flagan and Seinfeld (1988).

Equation (1) shows the combustion of simple hydrocarbon to form stable products (water and carbon dioxide). Equation (2) provides more detail on the formation of carbon dioxide in high temperature combustion processes, revealing an equilibrium between carbon dioxide and carbon monoxide as products of combustion.

$$C_nH_m+\left(n+\frac{m}{4}\right)O_2\Rightarrow nCO_2+\frac{m}{2}H_2O \quad (1)$$

$$CO_2\Leftrightarrow CO+\frac{1}{2}O_2 \quad (2)$$

Carbon monoxide is a major environmental pollutant due to worldwide use of internal combustion engines, especially in diesel and petroleum fuelled vehicles. The World Health Organisation (WHO) stipulates a range of air quality guidelines to protect against adverse health effects due to carbon monoxide exposure. The WHO guidelines are defined at four different averaging times, as presented in Table 1.

Table 1: WHO guidelines for carbon monoxide exposure, after World Health Organisation (1999)

Averaging Period	Criterion (mg/m^3)
15 minutes	100
30 minutes	60
1 hour	30
8 hours	10

The WHO outlines a number of environments in which people are likely to encounter carbon monoxide. These include “travelling in motor vehicles, working at [their] jobs, visiting urban locations associated with combustion

https://doi.org/10.26868/25222708.2017.168

sources, or cooking and heating with domestic gas, charcoal or wood fires" (World Health Organisation (1999)). The WHO follows on to state that "among these settings, the motor vehicle is the most important for regularly encountered elevations of carbon monoxide".

Widespread vehicle emissions control regulations have led to the adoption of exhaust catalytic converter systems in vehicles to reduce carbon monoxide emissions. These systems work by catalysing the oxidation of carbon monoxide to carbon dioxide (CO_2) by equation (3), after Guyer (1988).

$$2CO + O_2 \Rightarrow 2CO_2 \quad (3)$$

However the performance of catalytic converters in converting undesirable exhaust products (such as carbon monoxide) relies on waste heat from the exhaust gas, with reaction rate (and therefore performance) increasing as the temperature of the catalytic converter increases (refer Figure 1).

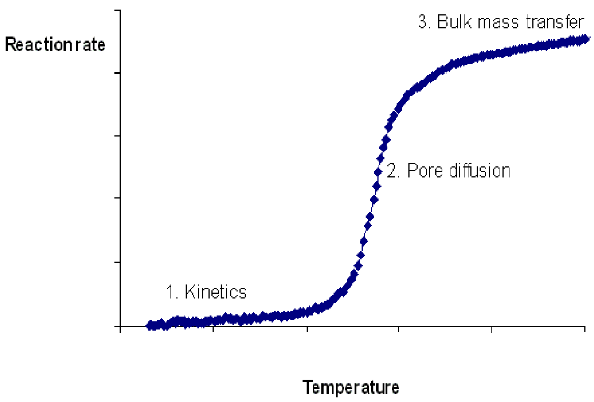

Figure 1: Typical conversion regimes and reaction rates of catalytic converters as a function of temperature, after Ehsan et al (2005)

Of note is the provision in Standards Australia (2012) of carbon monoxide emissions factors for the Australian fleet, quantifying the improvement in catalytic converter performance with temperature after a cold engine start. These emissions factors are presented in Table 2.

Table 2: Fleet average carbon monoxide emissions rates on cold engine start, after Standards Australia (2012)

Time after cold engine start	CO emission rate, g/min
First minute	25
Second minute	16
Third minute	10
Fourth minute	7
Fifth minute	5
Hot	3.2

Enclosed car parks

A critical case therefore occurs when a vehicle internal combustion engine is operating, the catalytic converter is at a low temperature (engine has just started), ventilation is limited, and humans are present. This critical case occurs in enclosed (underground) car parks.

Ventilation is the main aspect of underground car park design which can be controlled and engineered to limit human exposure to carbon monoxide. As ventilation is therefore critical for these areas, conservative "code-based" design approaches exist such as Standards Australia (2012). These approaches typically specify a coarse ventilation rate based on number of car parks and car park use (e.g. residential, retail, etc.). Some guidance may also be given on the approximate location of supply and exhaust grilles. Such simplified approaches may overestimate the supply air required and hence the scope of mechanical services plant required for maintenance of adequate air quality within the car park.

With the development of accessible, desktop computer based CFD tools, car park ventilation design is shifting from a code-based approach to a performance-based approach, with CFD methods used to confirm the safety of the proposed design. This approach is less conservative than a standardised approach, typically leading to a reduction in the mechanical ventilation requirements.

Pollutant source modelling

CFD results are extremely sensitive to several model components, including supply air diffusers, impulse ventilation (jet) fans, and vehicle pollutant sources. A range of methods have been used throughout industry to numerically model these components (typically under fire and smoke ventilation conditions) as described in Viegas (2010). These methods vary in accuracy. Of note are ASHRAE's efforts to provide simplified boundary conditions for numerical room airflow models, largely focusing on the accuracy of supply air diffuser modelling, published in Chen (2000).

For the critical underground car park case the pollutant sources are expected to have the most direct impact on pollutant levels. This paper focuses on modelling techniques for vehicle pollutant sources in car parks.

Vehicle pollutant sources have been studied in detail in mesoscale air quality applications (eg highways). For these cases turbulence provides mixing of the exhaust pollutants into the surrounding air. The widely adopted CALINE4 model (Benson (1989)) simplifies turbulence above roadways into two sources, mechanical turbulence due to the wake of the emitting vehicle and vehicles passing behind it, and thermal turbulence due to the elevated temperature and hence buoyancy of the emitted exhaust (refer Figure 2). This turbulent mixing results in a well-mixed volume source of pollutant, with dimensions some width greater than the roadway (3 metres on each side in the case of the CALINE4 model) and some height above the roadway. As mesoscale models typically address calculation domains with dimensions many hundreds of metres or several kilometres in size (i.e. in the far-field of the volume source), the pollutant volume sources tend towards line sources for these calculations.

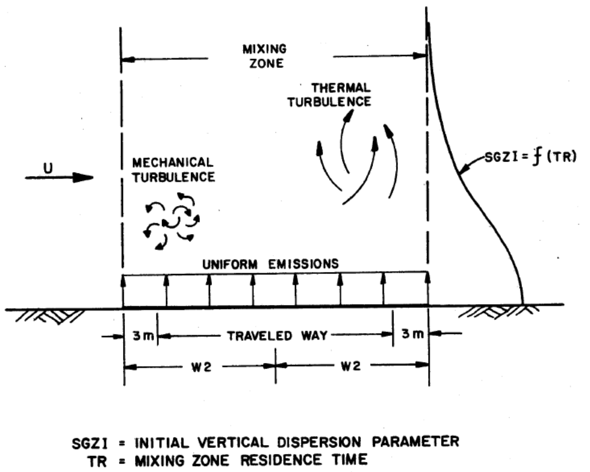

Figure 2: CALINE4 method of vehicle pollutant source modelling, after Benson (1989)

Although high speed vehicle pollutant sources have been studied and modelled for a number of decades, local flow around vehicles in low speed car park movements vary considerably from high speed mesoscale air quality applications. The level of both mechanical and thermal turbulence is unlikely to be as high due to low speed vehicle movements, fewer car movements, and cooler exhaust gas temperatures due to recent cold engine start (for cars exiting the car park). Pollutant source modelling for low speed car park applications have not previously been studied.

Purpose

The purpose of this study was to investigate various methods of representing low speed vehicle pollutant sources in underground car parks. The objectives of the study were to:

1. Identify any significant differences in pollutant source modelling methods.
2. Take the first steps in identifying which pollutant source modelling method is the most accurate and appropriate for underground car park modelling.

The fundamental objective of this work was to improve performance based methods of designing mechanical ventilation for underground car parks. This in turn will improve the air quality of car parks for occupants and will likely reduce the amount of mechanical ventilation required, with associated spatial, capital, operational, and energy savings.

Simulation

Computational method

The computational package OpenFOAM (Open Source Field Operation And Manipulation) was used to conduct a study into the sensitivity of airborne pollutant (carbon monoxide) concentrations to different methods of modelling vehicle pollution sources.

OpenFOAM is an open-source Linux-based C++ library, which provides solvers (i.e. applications) to solve specific continuum mechanics problems (Greenshield (2016)). A custom solver named transportBuoyantBoussinesqSimpleFoam was used for this work. The custom solver was based on the standard solver buoyantBoussinesqSimpleFoam, provided as part of OpenFOAM's default package of applications.

buoyantBoussinesqSimpleFoam solves the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations for momentum, pressure (using the SIMPLE algorithm (refer Ferziger and Peric (2002)) and temperature. Turbulence was also accounted for by the solver, using a 2 equation eddy viscosity based model (the k-ε model). Hence solutions for turbulent kinetic energy (k) and eddy dissipation rate (ε) were also incorporated into the solver.

The solver was customised into transportBuoyantBoussinesqSimpleFoam through addition of a solution for transport of a passive scalar variable as shown in equation (4), after Ferziger and Peric (2002). The equation is a basic scalar variable transport equation for φ in differential tensor notation, with (progressing left to right) transient, advection, diffusion and source terms.

$$\partial\frac{(\rho\phi)}{\partial t}\partial\frac{(\rho u_j\phi)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\Gamma\frac{\partial\phi}{\partial x_j}\right)+q_\phi \qquad (4)$$

The pollutant (carbon monoxide) was modelled as a passive scalar, with no physical effect on the flow. Pollutant entered the domain as a volumetric source term in the scalar transport equation (i.e. q_φ).

Although the incompressible RANS equations were solved, buoyancy forces were accounted for using the Boussinesq approximation, with density variations accounted for only in the buoyancy term of the equations. The buoyancy term was subsequently simplified from a density dependent term into a temperature dependent term, eliminating the need for a compressible solution to the RANS equations (equation (5)).

$$(\rho-\rho_0)g_i=-\rho_0 g_i\beta(T-T_0) \qquad (5)$$

This approximation is valid for flows where the difference in density is much less than the reference density ρ_0. The maximum temperature modelled was 323K, with a reference temperature of 293K. The difference in the density of air between these two temperatures was estimated to be approximately 4% of the reference density. Hence the Boussinesq approximation was considered valid for the purpose of this study.

Computational domain

The domain was a simple rectangular prism, with supply (inlet) and exhaust (outlet) located at high level along the length of the domain (i.e. +z and -z faces). Note that x is the major axis dimension, y is the vertical dimension and z is the lateral axis dimension. It is acknowledged that typical car park ventilation strategies incorporate a vertical differential between supply and exhaust to encourage mixing however the scope was to study the mixing of each type of source minimising external design-specific influences.

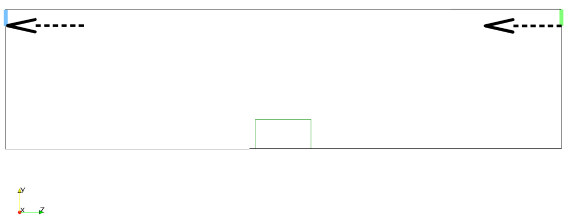

Figure 3: Section (x-plane) through domain showing supply (green) and exhaust (blue) locations. A volumetric source is also shown at the bottom of the domain.

A grid independence study resulted in a simple structured mesh of 160,000 cells. This mesh resolution provided a balance between discretisation error and solution run time. The mesh was held constant between the pollutant sources modelled.

Table 3: Domain properties

Property	Value
Size (x y z), m	40 2.2 10
Number of cells (x y z)	160 25 40
Inlet and outlet size (x y), m	40 0.3
Inlet volume flow rate, m^3/s	5
Inlet temperature, K	293
Outlet relative pressure, Pa	0

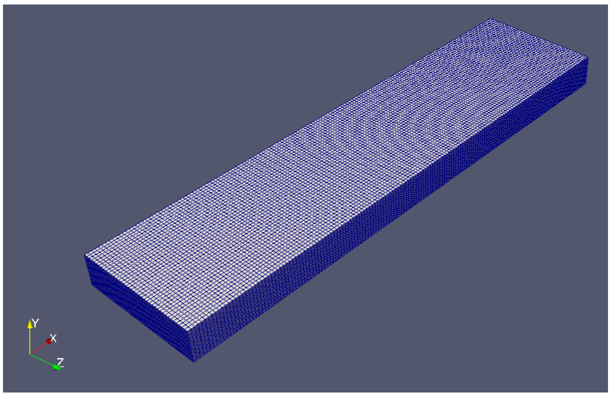

Figure 4: Isometric view of 3D domain and mesh

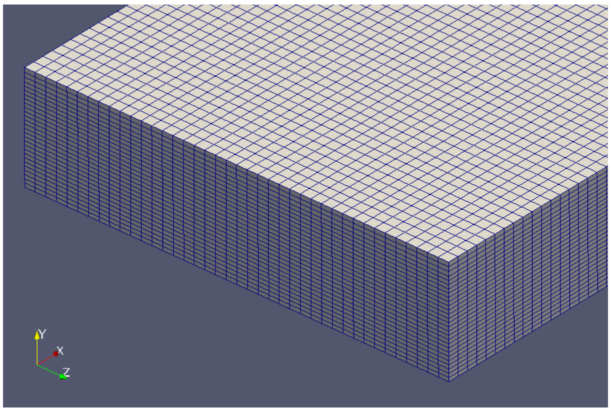

Figure 5: Detail of 3D mesh elements

Pollutant source modelling methods

Three source modelling methods were studied. These three sources vary primarily in the assumptions of turbulence and mixing made in their definition:

1. Finite width line source. This is a common method used in industry (e.g. Asimakopoulou (2009)) as it simplifies emissions calculations and is easily implemented. This is also the method used in mesoscale air quality applications, as it is a far-field approximation of a volume source. For this study, the line source was located on the car park floor in the centre of the domain, with momentum in the vertical (y) direction.

2. Distributed point sources. This is a less common method for modelling pollutant sources. This method was selected in order to eliminate assumptions about turbulence and mixing made when line and volumetric sources are used. These sources allow the numerical model to solve for turbulence associated with both buoyancy forces and shear between the exhaust jet and the free stream air. 10 distributed point sources were located in the domain, with a spacing of 2m, a temperature of 323K, and momentum in the primary horizontal (x) direction.

3. Volumetric source. This source is a similar approach to that used in the CALINE4 model (Benson (1989)), however the underground car park context

requires near-field volumetric treatment of the source, as contrasted to far-field line source treatment. This method assumes pollutant is perfectly mixed through some volume of air due to turbulence. For this study, the volumetric source was located in the centre of the domain vertically spanning between the floor of the car park and a height of 0.5m above floor level.

Properties of the sources were specified based on 10 cars with a total engine displacement of 18L (1.8L per car). This is an arbitrary assumption. However as the objective of this study was to compare the source modelling methods the assumption is valid to facilitate comparison provided modelling methods are kept constant between the three source models.

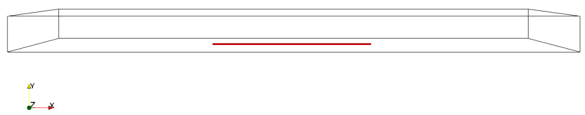

Figure 6: Finite width line source domain

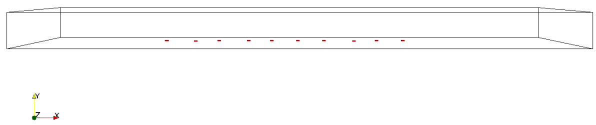

Figure 7: Distributed point source domain

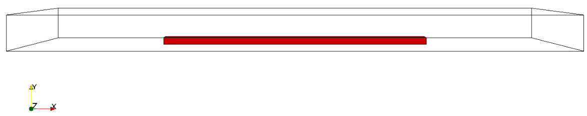

Figure 8: Volumetric source domain

Table 4: Pollutant source properties

Property	Line	Distributed point (per source)	Volumetric
Area/volume	8m^2	0.022m^2	10m^3
Volume flow rate (air), m^3/s	0.45	0.045	N/A
Temperature, K	293	323	293
Concentration of pollutant, mg/m^3	100	100	0.45

Control model

To provide an initial indication as to which pollutant source modelling method was the most accurate and appropriate for modelling of low vehicle speeds, the three source modelling methods were compared to a control model. This model consisted of a simplified vehicle body (MIRA fastback) with exhaust representing a single car after Hu et al (2015). The vehicle body was modelled with a flow velocity of 6 km/h, the average speed of vehicles moving through a car park suggested by Standards Australia (2012). The control model represented an initial attempt to study the levels of mechanical and thermal turbulence for pollutant emissions from the exhaust pipe of a slow moving car.

Similar to the pollutant source models the control model domain was a simple rectangular prism. The inlet comprised the entire -x face and the outlet comprised the entire +x face, forming a “numerical wind tunnel”. A single point source of momentum, temperature and pollutant was located at the rear of the control model along the x axis centreline, 0.1m above the ground. This source represented the exhaust pipe outlet. The properties of this source were identical to those of a single point source used in the distributed point source pollutant source model.

An unstructured mesh of 580,000 cells was created using the snappyHexMesh tool provided as part of the OpenFOAM package. This tool used a base structured hexahedral mesh to generate a refined unstructured hexahedral mesh, with mesh nodes snapped to the faces of blockage objects in the domain (i.e. MIRA car geometry).

Table 5: Control model domain properties

Property	Value
Size (x y z), m	55 5 8
Number of cells (unstructured)	580,000
Inlet velocity, m/s	1.67
Outlet relative pressure, Pa	0

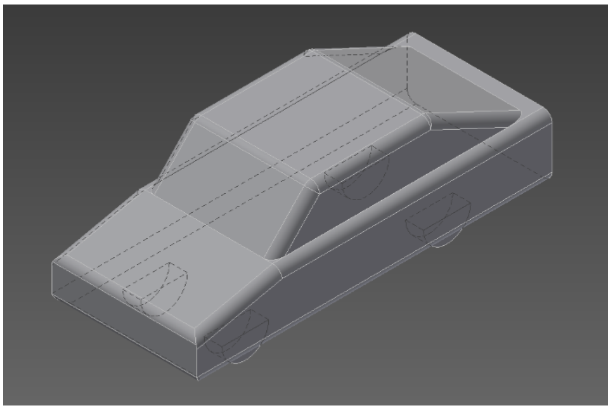

Figure 9: Isometric view of MIRA car 3D model

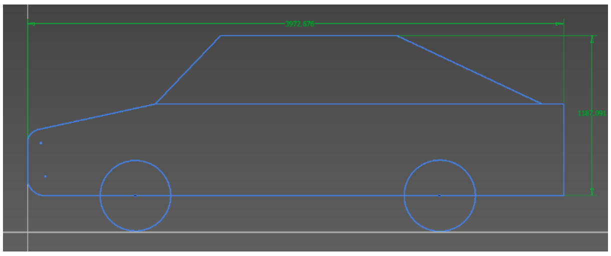

Figure 10: Sketch of MIRA car model showing major dimensions

Results

Pollutant source modelling methods

The key metric of this study was the concentration of pollutant (carbon monoxide), expressed in mg/m^3. As this study is a relative comparison, the absolute values of concentration are of limited interest. Concentration results were extracted from the models at 750mm above local floor level, identified as the lower limit of the breathing zone by Standards Australia (2012).

Two concentration values were extracted from the models. The maximum concentrations are directly related to safety based guidelines for pollutant (carbon monoxide) and hence underground car park ventilation design. These maximum concentrations typically occur across a small area and are sensitive to the flow features in the model. Measured "typical" concentrations represent concentrations represented over a large area, driven by the numerical representation of the pollutant source.

Pollutant concentrations varied considerably between the sources. The line source displayed the highest pollutant concentrations of 323 mg/m^3 (maximum). This value was measured in a thin band of flow against the wall, representing an area of recirculation and hence pollutant concentration. The typical concentration for the line source model was approximately 160 mg/m^3 dispersed over the central half of the car park.

The distributed point source method resulted in a maximum tracer concentration of 29.5 mg/m^3 measured directly above the point sources. The typical pollutant concentration was approximately 20 mg/m^3 dispersed over the central half of the car park.

The volumetric source displayed a maximum and typical concentration of 13.4 mg/m^3, across the central half of the car park. The dimensions of the volume source may have overestimated the size of the well mixed region in the wake of exiting vehicles, reducing overall concentrations.

The distributed point source showed the least sensitivity to ventilation conditions in the car park, with the pollutant flow driven by the momentum and temperature of the sources, as expected. The line source and volume source models showed sensitivity to the ventilation conditions, with the model boundary conditions encouraging some recirculating flow, and hence increasing pollutant concentrations.

Table 6: Maximum and typical pollutant concentration (mg/m^3) at 750 mm above local floor level

Result	Line	Distributed point	Volumetric
Maximum	323	29.5	13.4
Typical	160	20	13.4

Figure 11: Line source - plan view showing pollutant concentration

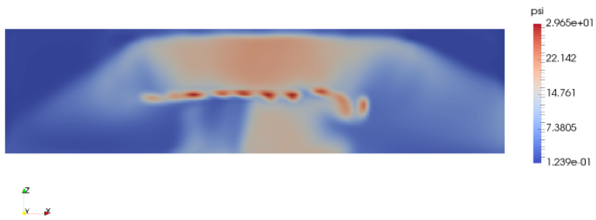

Figure 12: Distributed point source - plan view showing pollutant concentration

Figure 13: Volumetric source - plan view showing pollutant concentration

The results show that pollutant concentrations are sensitive to the source modelling method, with the largest maximum concentration modelled more than 24 times greater than the smallest maximum concentration, and the largest typical concentration more than 11 times greater than the smallest typical concentration.

Control model

The wake of the control model displayed considerable momentum in the negative y direction (downward towards the ground). This flow feature caused the exhaust plume to remain vertically unmixed and attached to the ground for a distance of approximately 40 metres, after which vertical mixing was apparent.

Figure 14 shows the rear of the control model. A contour coloured by vertical velocity (red positive y direction - upward, blue negative y direction - downward) and velocity vectors are displayed. It is apparent that significant downward momentum is induced in the wake of the car. This downward momentum is higher in magnitude than the upward momentum caused by the buoyant exhaust plume, which is identifiable as an area of upward momentum downstream of the bottom edge of the car body.

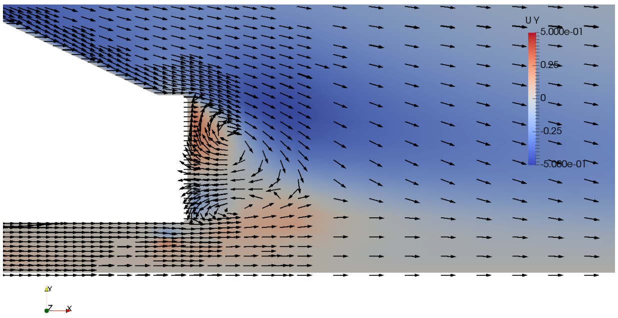

Figure 14: Control model – velocity vectors and vertical velocity contour showing recirculation and downward momentum at rear of vehicle

The maximum concentration at 750 mm above local floor level was 31 mg/m^3. This maximum concentration occurred consistently downwind of the car, with flow features generally resembling those of the line source pollutant source model. It is difficult to draw a direction comparison between numerical results of the pollutant source modelling methods and the control model due to the difference in ventilation conditions between the models. However, if a direct comparison of maximum pollutant concentration is conducted and normalised to emissions from a single car, it is found that the line source result is 4% larger and the volumetric source result 2300% smaller. The distributed point source model provides a more difficult comparison as the contribution of each source to the maximum pollutant concentration value is unknown. Based on the location of the maximum concentrations, in the emission direction and vertically upward from each point source, it may be reasonably assumed that only a single point source contributes to each area of maximum pollutant concentration. If this is the case then the distributed point source maximum concentration is 5% smaller than the control model result. These preliminary results indicate that the line source and distributed point source may provide accurate solutions for car park ventilation studies. The volumetric source may potentially be resized to match the size of the control model wake, however different car shapes and movement conditions will result in a wide range of wake sizes and behaviour. It is also worth noting that the control model did not account for mixing due to the interaction of car movement with existing wakes (i.e. a car following after another). It is anticipated that this movement will significantly increase the level of pollutant mixing, with results tending toward the well mixed volumetric source results.

Figure 15: Control model - plan view showing pollutant concentration

Although a basic preliminary study, the control model provided initial indication to which source modelling method may be the most accurate from a pollutant concentration perspective (point source) and direction to pursue further work.

Further Work

The current study has achieved its objective of demonstrating the sensitivity of carbon monoxide results to source modelling methods. Initial steps towards identifying the most accurate methods have been made, however it is acknowledged that significant further work is required in this space.

Further work includes experimental validation of the MIRA vehicle body control model under low speed applications. This will enable the most accurate modelling method to be more clearly identified. Moreover, the momentum, energy, and turbulence boundary conditions of the source modelling methods may be altered to more closely represent the flow physics occurring in exhaust plume of a slow moving vehicle. This work should also be extended to the interaction between vehicle wake plumes and turbulence induced by vehicles following behind.

A sensitivity study of the pollutant source modelling methods to momentum and temperature boundary conditions (both for the source and for the car park ventilation system) is also required to determine the robustness of the source modelling methods in a wide range of underground car park applications.

It is noted that a compressible solution to the RANS equations may be studied in the future to remove the need for the Boussinesq approximation and improve the accuracy of the numerical simulation.

Finally, the volumetric source, with refinement, may show potential for development of a basic calculation to determine maximum concentrations in linear sections of car park roadway based on limited input data (e.g. number of cars and car park layout). This approach would require a more fundamental study of mixing and turbulence of buoyant jets in a mechanically turbulent medium. A hybrid Navier-Stokes/bulk flow model may be possible. If developed this approach may represent a point on the spectrum of numerical complexity between a code based approach and a full numerical (CFD) model, which may be more accessible to Building Services/HVAC Engineers than CFD modelling.

Conclusion

A CFD study of pollutant (carbon monoxide) source modelling methods in underground car parks has been conducted. This study involved comparison of three pollutant source modelling methods: 1) finite width line source, 2) distributed point source, 3) volumetric source. The computational package OpenFOAM was used to conduct the numerical study using a custom solver named transportBuoyantBoussinesqSimpleFoam, a modification of the standard buoyantBoussinesqSimpleFoam solver to allow for advective and diffusive transport of a passive scalar variable (pollutant concentration). Modelling parameters were kept constant between the pollutant source models in order to facilitate a direct comparison between the source modelling methods.

Pollutant concentration results varied considerably between source modelling methodologies. The largest maximum concentration was more than 24 times greater than the smallest maximum concentration, and the largest typical concentration was more than 11 times greater than the smallest typical concentration.

Some methodologies were also more sensitive to the ventilation boundary conditions of the car park, with the line source and volume source models showing increased pollutant concentrations due to recirculating flow.

An initial study into identification of the most accurate pollutant source modelling method was conducted using a control vehicle model. This model approximately accounted for mechanical and thermal turbulence in the case of pollutant emissions from the exhaust pipe of a single slow moving vehicle.

The control vehicle model results pointed toward the line source and distributed point source as the most accurate methods. Flow features of the line source source closely resembled those of the developed pollutant plume in the wake of a slow moving car. The volumetric source greatly underestimated pollutant concentrations due to the assumption of a well mixed wake inherent in application of the line source. Considerable further work is required in experimental validation as well as testing momentum and energy sensitivities of the source types and wake interaction behaviour.

Acknowledgements

The author would like to acknowledge the OpenFOAM foundation and the open-source movement, without which this work would not have been possible, and Stewart Mann, for providing support, comments and review of the draft format of this paper.

References

Asimakopoulou, E. Kolaitis, D.I. Founti, M.A. (2009). CO Dispersion in a car-repair shop: An experimental and CFD modelling study. *Seventh International Conference on CFD in the Minerals and Process Industries*.

Benson, P. (1989). CALINE4 – A Dispersion Model for Predicting Air Pollutant Concentrations Near Roadways. *State of California Department of Transportation*. Sacramento.

Chen, Q. (2000). RP-1009 Simplified Diffuser Boundary Conditions for Numerical Room Airflow Models. *American Society of Heating, Refrigeration, and Air Conditioning Engineers*.

Ferziger, J.H. Peric, M. (2002). Computational Methods for Fluid Dynamics. *Springer*. Berlin.

Flagan, R.C. & Seinfeld, J.H. (1988). Fundamentals of air pollution engineering. *Prentice Hall*. New Jersey.

Greenshields, C.J. (2016). OpenFOAM User Guide version 4.0, *The OpenFOAM Foundation*. http://openfoam.org.

Guyer, H.H. (1998). Industrial Processes and Waste Stream Management. *John Wiley & Sons*. New York.

Hu, X.J. Yang, H.B. Yang, B. Li, X.C Lei, Y.L (2015). Effect of Car Rear Shape on Pollution Dispersion in near Wake Region. *Mathematical Problems in Engineering 2015*.

Standards Australia (2012). AS1668.2-2012 The use of ventilation and airconditioning in buildings – Mechanical ventilation in buildings. *SAI Global*.

Viegas, J.C. (2010). The use of impulse ventilation for smoke control in underground car parks. *Tunnelling and Underground Space Technology 25*, 42-53.

World Health Organisation (1999). Environmental Health Criteria 213 Carbon Monoxide (Second Edition). *World Health Organisation*. Geneva.