CFD Modelling for Car Park Ventilation Systems –

a guide for designers and regulators

HEVAC

Smoke Control Association

Acknowledgements

Contributions to this guide are gratefully acknowledged from the following people:

Hugh Mahoney	Advanced Smoke Technology Ltd
Ben Hume	Arup
Bernardo Vazquez	Buro Happold
Gary Walker	Buro Happold
Paul Compton	Colt International Ltd
Geoff Whittle	Simulation Technology Ltd
James Allen	Fläkt Woods Ltd
Neil Jones	Fläkt Woods Ltd
Paul Kingston	K8T Ltd
Glenn Horton	Locke Carey
Dominic Way	Locke Carey
Richard Brooks	PSB-UK Ltd
James Bertwistle	WSP Fire Engineering
Mike Duggan	HEVAC

Date of publication: February 2007

© Federation of Environmental Trade Associations 2007

All rights reserved. Apart from any fair dealing for the purposes of private study or research allowed under applicable copyright legislation, no part of the publication may be reproduced, stored in a retrieval system, or transmitted in any form by any means, electronic, mechanical, photocopying, recording or otherwise, without the prior permission of the Federation of Environmental Trade Associations, 2 Waltham Court, Milley Lane, Hare Hatch, Reading, Berkshire RG10 9TH.

FETA uses its best efforts to promulgate Standards and Guidelines for the benefit of the public in the light of available information and accepted industry practices but do not intend such Standards and Guidelines to represent the only methods or procedures appropriate for the situation discussed. FETA does not guarantee, certify or assure the safety or performance of any products, components, or systems tested, installed or operated in accordance with FETA's Standards or Guidelines or that any tests conducted under its Standards or Guidelines will be non-hazardous or free from risk.

FETA, and the individual contributors, disclaims all liability to any person for anything or for the consequences of anything done or omitted to be done wholly or partly in reliance upon the whole or any part of the contents of this booklet.

CONTENTS

1. Scope

BS 7346-7, the only detailed guide to the ventilation of enclosed or underground car parks, briefly discusses the use of Computational Fluid Dynamics (CFD) for car park ventilation design but provides little guidance on its use.

As there is a lack of suitable published guidance for either designers or approving authorities, this document sets out the information and parameters that the designer should incorporate into the design of the CFD model. It is intended that this document should be of use to the designer in producing and running the CFD model and in writing the CFD report.

It also provides recommendations on the information to be provided to the approving authority within the designer's package of supporting information when submitting the CFD analysis for information and/or approval of design intent.

The Smoke Control Association is primarily concerned with ventilation design for the removal of smoke and heat as discussed in the British Standard, but, recognising the dual use of systems, this document provides guidance on usage for both smoke and heat removal and vehicle emission ventilation.

This document is intended to support the recommendations of BS 7346-7.

2. Introduction

Ventilation of covered car parks is usually recommended in order to limit concentrations of carbon monoxide (CO) and other vehicle emissions in day to day use of the car park and to remove smoke and heat in the event of a fire. The same equipment is often used to satisfy both requirements.

Computational Fluid Dynamics (CFD) analysis is rarely used as the primary design tool for car park ventilation systems. Many systems simply comply with the prescriptive recommendations in Approved Documents B and F and do not require performance analysis. When alternative systems are proposed, for example, the use of impulse ventilation systems and in particular those designed to assist fire-fighting access or protect means of escape, the design is usually initially developed using other methods and may then be subjected to CFD analysis for fine tuning of the design and to demonstrate to approving authorities that the system is likely to perform satisfactorily.

The main advantage of using CFD for these procedures is that this can be carried out prior to installation of the ventilation system, providing confidence to all parties early in the project. It is much easier and cheaper to agree expectations and correct any problems at this stage than after testing upon completion of the installation.

Due to concerns about damage it is rare for testing to be carried out using a real fire or any significant heat source and a correctly modelled CFD analysis can provide relatively accurate simulation of what may happen within the car park.

For the design and approval process to be successful it is strongly recommended that, except perhaps in the simplest cases, the system objectives, the scenarios to be modelled, the modelling criteria, the expected reporting and the success criteria are all agreed and documented prior to commencement. CFD is too expensive and time consuming a process to be carried out without this agreement. Advice and guidance on these issues is provided in this document.

Since car park ventilation systems are usually dual purpose, providing ventilation for vehicle fume control in normal conditions and for smoke clearance or smoke control in fire conditions, consideration should be given to which operational modes require CFD analysis as the scenarios and operating conditions will be different depending upon the choice made.

It is important to note that, while CFD modelling provides highly detailed outputs, the results should be regarded as snapshots representing a likely outcome and an indication of performance and not as definitive statements of conditions in use.

Properly used, CFD is a useful tool for design and approval of car park ventilation systems. Following the guidance in this document will aid proper use and help lead to a valid outcome.

3. Definitions

The definitions include both terms that are used in this document and terms that could be contained within a CFD report.

3.1 Blocks

Blocks are used to represent solid objects in the scenario being modelled. (e.g. walls, floors, ceilings, down stands, cars, doors, etc)

3.2 Boundary

The boundary is the edges of the domain or space that the CFD calculation is being performed within.

3.3 Boundary conditions

Boundary conditions are set up by the user and characterise what happens at the edges of and in particular areas within the domain. (For example; Wall, vents etc)

3.4 CFD

Computational Fluid Dynamics (CFD) is the use of computers to solve mathematical equations that simulate the flow of fluids, heat transfer and other associated phenomena. (For the purposes of this paper, CFD modelling can be used to predict fire, smoke movement, heat, radiation, ventilation flow etc)

3.5 Comparative Analysis

This compares two or more scenarios. This is often used to show that the model is a good as the minimum requirement of a Standard or the suggested guidance to the Building Regulations.

3.6 Deterministic Analysis

This is a non comparative study. It is often used to show that the conditions satisfy the functional requirement of the Building Regulations.

3.7 Domain

The Domain is the area that is to be modelled. This may include; part of the car park, all of the car park or all of the car park and some surrounding areas.

3.8 Growing Fire

This is a fire with a varying heat release rate. Despite the name this is often used to describe the decaying period of a fire curve as well as the growing.

3.9 Impulse fan (Also known as Jet Fan or Induction Fan)

Fan designed to transfer momentum into the air as part of an impulse ventilation system and used to provide control of air direction and velocity.

3.10 Mesh (Grid)

The domain is broken down into a number of mesh cells. The computer performs calculations (Turbulence, flow, heat transfer etc) on each cell to obtain a final solution. A mesh is also known as a grid.

3.11 Output Slice

This is a two dimensional output of data across a plane in the domain and is used to show visual conditions (i.e. temperature, pressure, velocity etc). Colours are used to represent varying conditions.

3.12 Porosity

The condition of a boundary that allows a set amount of leakage that may not be proportional to the size of the vent.

3.13 Sensitivity analysis

Varying selected parameters in a model to investigate the extent of their effect (e.g. changing the mesh size).

3.14 Steady State fire

This is a fire with constant heat release rate.

3.15 Steady State model

This is a model that has no associated time period. This shows what conditions would be like if the scenario was run for infinity. (Within the scope of this document, it is usually used to show that smoke or heat is being taken out by the ventilation system at the same rate that it is being produced by the fire or demonstrates the constant flow profile of a fan or vent.)

3.16 Transient model (Also known as a time dependent model)

This is a model that is time dependent and shows how conditions vary with time.

3.17 Vector slice

This is a two-dimensional output of vector data across a plane in the domain. Arrows represent the direction with arrow size indicating the vector quantity.

4. An introduction to CFD

In more and more new building schemes it is common to have complex internal geometries and unique architectural features; many of these features may not follow standard guidance for compliance with Building Regulations. It is widely recognised that standard, simple calculation techniques may not provide adequate levels of detail to describe the flow patterns in these designed environments.

In recent years there have been major advancements in computer science, allowing more detailed calculations to be performed. The calculation methods remain based on the same validated assumptions. However, with the use of Computer Field Modelling techniques the standard equations can now be applied with much more refinement.

Computational Fluid Dynamics, or CFD, modelling is an accurate, real time analysis of interacting physical properties. It produces geometrically unique, tailored solutions, offering a greater degree of accuracy than generic hand calculations or ‘Zone Model’ solutions. It is now a widely used technique both within the building industry and in other fields such as vehicle aerodynamics, heat exchanger design, etc.

It works by allowing an object, structure or domain of interest to be split into a computational mesh containing hundreds of thousands of small volumes or cells. Software applies well validated transport equations to each of these cells, which will then describe the transfer of quantities such as heat, velocity, chemical species, density, mass, momentum, etc between cells. This builds up a real time 3D model of the specific system.

There are a number of commercially available CFD software packages, each with its own particular features depending on its intended usage. Each includes the basic transfer equations, a number of physical sub models to deal with issues not addressed by the basic transfer equations and a database of standard information.

In using CFD to analyse conditions within a car park the designer needs to make many decisions, including which CFD software package to use, what mesh size is appropriate, which sub models are appropriate, etc. These decisions all affect the quality and validity of the final output. The guidance in this document will inform these decisions and help to ensure that accurate, valid results are achieved.

5. Preparing the CFD model

5.1 Definition of the Computational Domain

The starting point for the application of CFD to the simulation of air movement, fire and smoke movement in a car park is to establish the computational domain for the simulation, i.e. the limit of the region to be modelled.

The primary considerations are itemised below.

- Three-dimensional Domain.

  Even in the simplest geometries, the air and smoke flows are three-dimensional.

- Boundaries of the Domain.

  The boundaries of the domain are a function of the objective of the CFD simulation. They should encompass the region of interest and be located where the flow conditions are known. The influence of an external flow, e.g. wind, may require that the boundaries of the domain extend beyond the area of immediate interest, i.e. the car park.

  The extent of the domain will be influenced by the nature of the car park. The boundaries of the domain for a fully enclosed, mechanically ventilated basement car park will be formed by the walls, doors (usually assumed to be closed), floors and ceiling of the car park. In an 'open' naturally-ventilated multi-storey car park, the boundaries of the domain are likely to extend beyond the car park (and may include a representation of other buildings in the immediate vicinity) in order to represent the effect of wind on internal air and/or smoke flows adequately.

- Additional Factors Affecting Domain Boundary Selection.

  - Computational Limits.

    A CFD simulation that accounts for all the possible influences on air flows or a fire and the induced flows in a car park may be too large for the computational resources (processing power, memory and time) available. In these circumstances, it is necessary to focus on the key features affecting the air and/or smoke flow whilst ensuring that the influence of the omitted factors does not compromise the objective of, or the conclusions to be drawn from, the CFD simulation.

  - Attached Volumes.

    The volume of immediate interest may be connected to, and influenced by, other volumes, e.g. floors of a multi-storey car park which are not immediately adjacent to the fire floor. In such cases, it may not be possible or necessary to model the entire car park. The boundaries of the domain need to be defined at locations where the flows between the attached volumes can be considered to be minimal or where they may be approximated by boundary conditions derived from measurement or calculation.

  - Symmetry.

    Provided that the fire, environmental and mechanical ventilation induced flows may be considered to be reasonably approximated by the application of a symmetry condition, geometric symmetry may be used as a means of reducing the size of the domain. In these circumstances the planes of symmetry provide one or more of the boundaries of the domain.

- The boundaries of the domain should be located such that they do not adversely affect simulated smoke movement. For example, open (or 'free') boundaries should not be located close to the source of the fire as smoke may be lost to the computational domain when, in reality, it might re-enter the fire affected region subsequently. (If smoke leaves the computational domain, it should be at locations sufficiently removed from any induced flow (as a result of the fire, the wind or from mechanically assisted means) which might subsequently allow the smoke to re-enter the domain.)

5.2 Details Represented in the Computational Domain

Establish what to include in, and exclude from, the geometric representation of the model.

Any object which may have a significant impact on air flows or fire induced flows and smoke movement should be represented within the model.

Typically, this should include some or all of the following items.

- Structural walls, floors and ceilings.
- Structural columns and beams.
- Services, e.g. the geometric representation of HVAC ductwork.
- Stationary vehicles.

  The size, number and distribution of vehicles represented in the car park will be a function of the car park site and the design scenario.

  In defining the number of vehicles to be included in the model, it is important to consider:

  - The objective of the CFD simulation.
  - That the potential for an accidental fire is likely to increase with an increase in the number of cars within the car park.
  - That the risk to life is likely to increase with an increase in the number of cars in the car park.
  - That the presence of vehicles will affect the fire, environmental and mechanically induced flows.

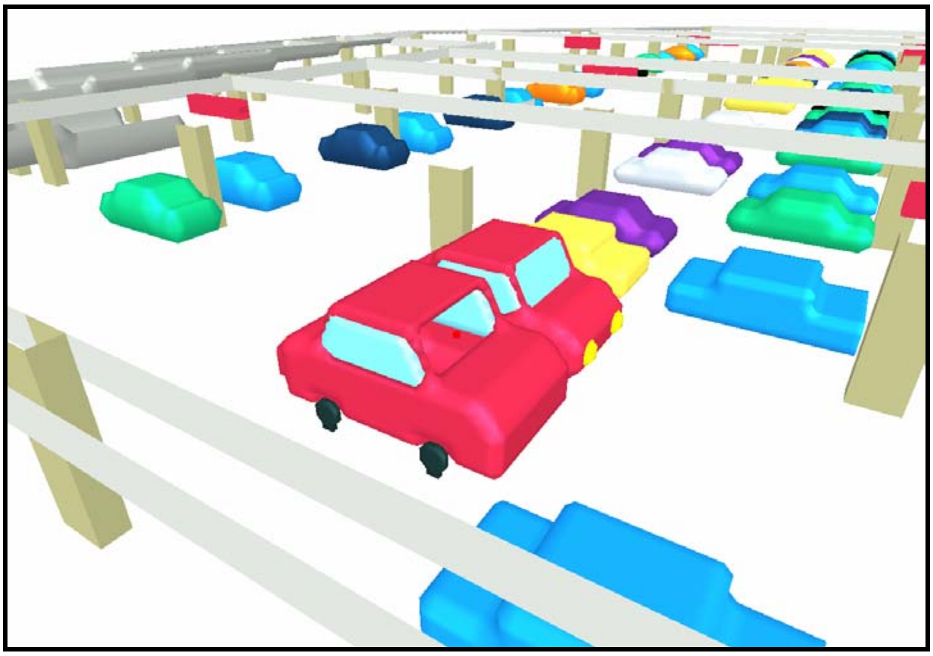

Figure 1: An example of the distribution of cars within a car park model

(note that cars may often be represented satisfactorily by simple blocks)

- Environmental flows, i.e. wind and factors / objects affecting such flows, e.g. external buildings.
- Mechanically induced flows, e.g. impulse fans and inlets / extracts.
- Significant sources of heat (other than the fire) which might create a natural convection flow that could interact with the fire induced flow.
- The fire source.

Factors affecting the geometrical representation of the model include some or all of the following.

- The location of the object / source with respect to the fire and environmental and mechanical ventilation induced flows (and the effect that the object / source may have on the induced flows).

  An object remote from the fire may have a lesser impact on the (bulk) movement of smoke than an equivalent sized object located close to the fire.
- The size of the object / source with respect to the space modelled and to the anticipated flows.
- The size of the object with respect to the computational mesh (or grid) size.

  An object that is smaller than the size of the computational mesh in the vicinity of the object cannot be represented within the model (unless a sub-mesh or porosity sub-model can be utilised).
- The computational mesh (or grid).

  The nature of the computational mesh adopted by the CFD package may affect the geometrical representation: for example, a mesh based on a rectilinear coordinate system can only approximate curved or sloping surfaces.

There are no hard and fast rules to indicate what should / should not be included within the geometrical representation of the model. The modeller should generate and justify the geometric representation of the model on a case-by-case basis. The flow should not be significantly affected by any geometric simplifications made.

It is relevant to consider that a CFD simulation is an approximation of reality (a description of what might happen in a fire event). Increasing the geometrical detail within the model will not necessarily increase the understanding of the bulk flows in the car park.

5.3 Computational Mesh

The computational domain is sub-divided into a large number of smaller cells.

The computational mesh, i.e. the size and disposition of the mesh cells, should be designed to ensure that the following requirements are satisfied.

- The geometric details are represented appropriately.
- Where a fire is being modelled, the flow phenomena driving smoke movement are resolved adequately.
  - Fire – typically, several mesh cells should be used to represent the plan dimensions.
  - Thermal plume – sufficient detail is needed to capture the rise of the hot gases.
  - The region adjacent to the ceiling – including the ceiling hot gas layer (and, particularly, ceiling layer flows) – should include:
    - A significant number of cell layers normal to the ceiling when a ‘structured mesh’ is adopted.
    - Slowly inflated mesh cell sizes normal to the ceiling when an ‘unstructured mesh is adopted.

- Mechanically induced / assisted flows are represented appropriately. Ensure that:
    - Sufficient mesh cells are used to describe the dimensions of the fan / inlet / extract in the plane normal to the direction of flow (typically, several will be required – the guidance of the product developer should be followed).
    - Changes in the dimensions of the mesh cells in the direction of the flow do not influence the flow characteristics.
- The guidance of the product developers is followed to ensure that mesh cell size selection is consistent with the modelling approach adopted.
- Mesh cells should not be subject to significant distortion, i.e. they should have low aspect ratios (preferably close to unity in the vicinity of the fire). Guidance should be sought from the product developers to assess the maximum permitted mesh cell distortion.
- Ideally, an investigation of the sensitivity of the results to the mesh cell size adopted should be undertaken.

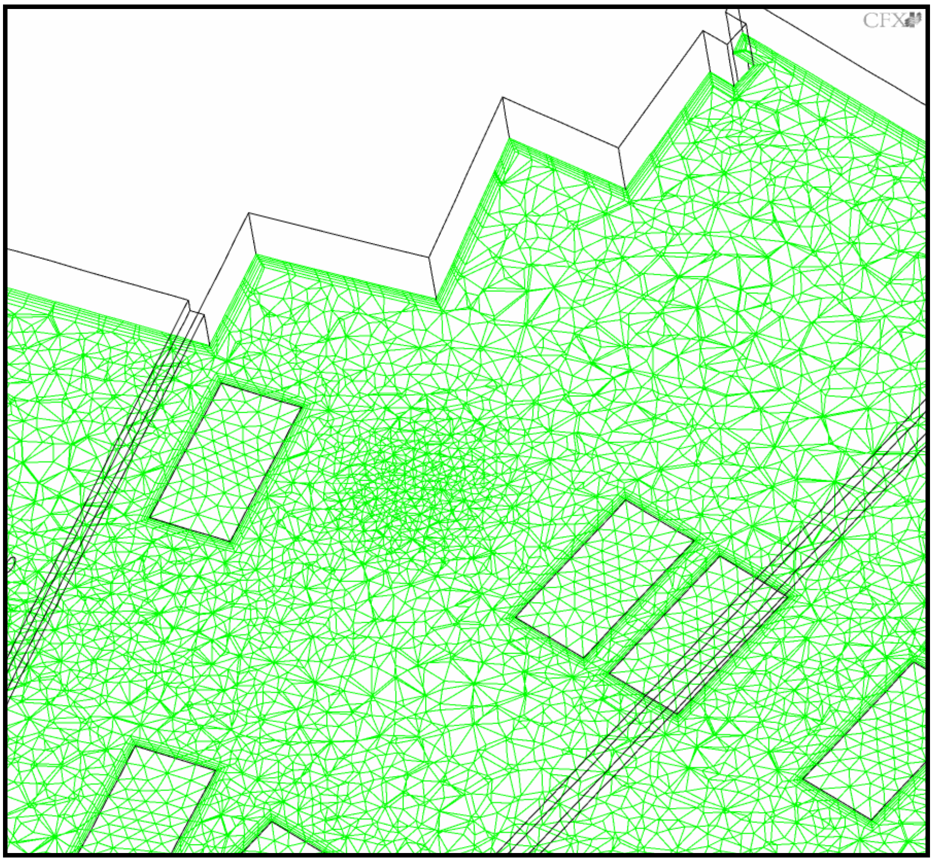

Figure 2: An example of a non-rectilinear mesh showing reduced mesh size at the fire source and an inflated mesh size at the boundaries (using CFX)

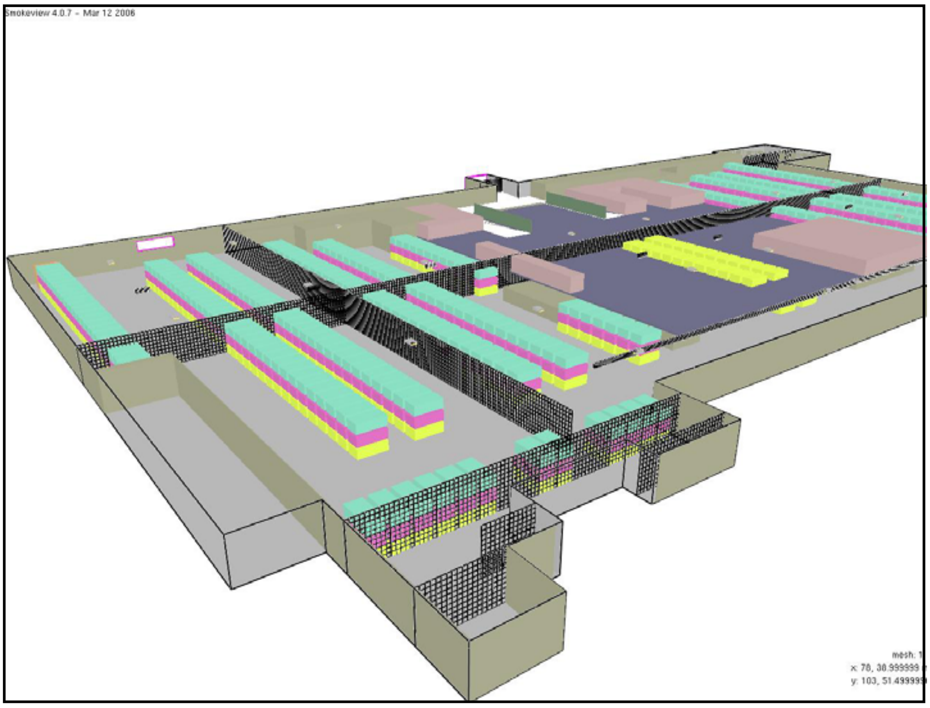

Figure 3 An example of a rectilinear mesh in a large car park (using FDS)

5.4 Physical Sub-models

The physical sub-models selected define the equations to be modelled within the CFD simulation.

The important physical mechanisms governing the flow and which need to be captured include the following.

- Combustion – the processes that are responsible for the production of heat and smoke.
- Buoyancy – heat released by a fire induces buoyant flows (natural convection) and influences turbulence.
- Turbulence – fans, air inlets and obstructions to flow create a turbulent flow that affects induction into the air flow and subsequent air and smoke transport. A fire induces a turbulent flow that influences the exchange of heat with the ambient air, affects the fuel-air mixing prior to combustion and subsequently affects smoke transport and dilution through mixing.
- Radiation – part of the heat produced by a fire and transported in the smoke is exchanged with the surroundings by radiation.
- Heat transfer at walls – fire-generated flows lose some of their heat at solid boundaries.

In all cases, guidance should be sought from the developers of the CFD simulator as to the appropriate model to adopt.

These models are discussed in more detail in Annex A.

5.5 Fire Source Specification

Where the modelling includes a fire, the location, size and characteristics of the fire need to be specified for the CFD simulation.

Modelling a fire is not usually necessary when considering ventilation for control of vehicle emissions or for smoke clearance.

Fire Scenarios

Selection of the fire scenario(s) to be investigated is a complex process requiring:

- An assessment of the objective of the investigation.

- An understanding of the likely flow processes within the car park.
- An understanding of the fire hazards, i.e. the sources of fuel and ignition.

Typically, these lead directly to the definition of the fire location.

Experimental and / or published data can then be used to define the fire size and characteristics. BS7346-7 provides recommended steady-state design fire characteristics.

For more unusual applications such as car parks containing car stacker systems there may be little data available on fire loads and fire spread. In such cases it is particularly important that any estimates for fire size and characteristics are agreed with the approving authorities while developing the CFD model.

Fire Heat Release

The rate of heat release is a prescribed input to the CFD model for both the volumetric heat source model and the combustion model.

The volume over which the heat is released (which is dependent upon the footprint and area of the burning material) is an additional input when employing a volumetric heat source model.

A combustion model predicts the heat distribution in the flaming region above the seat of the fire; the area over which the heat is released must be specified.

Fire Smoke Production

Production of smoke is dependent upon the properties and physical state of the combustible materials, the quantity available and the availability of air supply to the flame.

It is usual for the smoke production rate to be linked to the heat release rate by a 'yield factor' (representing the production of smoke) which has been determined experimentally for a wide range of materials and conditions.

Smoke is then assumed to be generated uniformly over a volume (volumetric heat source models) or an area (combustion models).

This approach is suitable except for those cases where there is a significant change in the rate of smoke production, e.g. from a well-ventilated fire to an under-ventilated fire, as the CFD model will assume that it is constant.

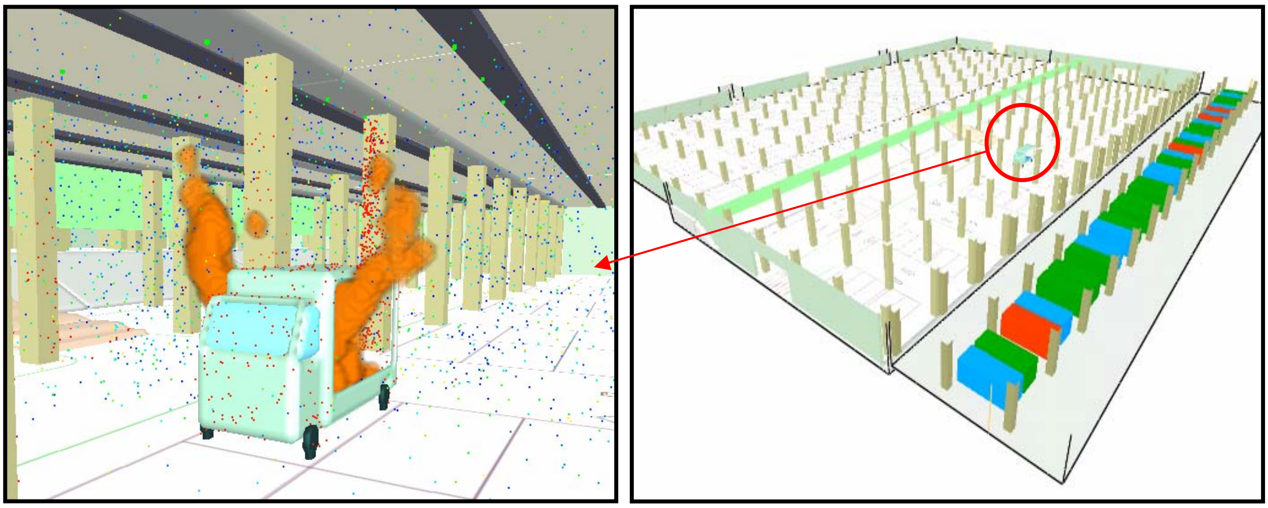

Figure 4: Detail of smoke production and sprinkler droplet distribution

Fire Spread

In some scenarios, mainly involving combustion models and growing fires, flame spread may be seen as an important factor in the total Heat Release Rate. In these incidences it is important to identify the material properties of neighbouring materials close to the seat of the fire to assess whether these materials will add to the flame spread.

5.6 Initial and Boundary Conditions

Any mechanism which is external to the computational domain but which influences the behaviour of the flow within it must be represented. Typical examples include the following.

- Initial flows present within the computational domain prior to the simulation.
- Flows in / out through doors, windows, openings, vents or mechanical inlet/extract systems.
- Change of momentum and / or energy in simplified representations of mechanical systems such as jet fans.
- Energy transfer (in the form of heat) at (to / from) walls.
- Sources of mass, momentum and / or energy, e.g. at the fire, or through the release of a suppressant.

The initial and boundary conditions must be defined by the user. Establishing representative initial and boundary conditions can be a major challenge, particularly where it is necessary to prescribe the level of turbulence associated with an initial / boundary condition.

The level of detail available will vary with the CFD program used. Not all options discussed are available in all CFD programs.

Initial Conditions

The initial flow conditions may need to be established by analysis and / or simulation prior to undertaking the simulation.

Inlets and Outlets

At an open (or 'free') boundary where the flow will be mainly influenced by what happens inside the computational domain, a constant pressure boundary (which implicitly assumes that the flow is fully developed) is applicable. Such boundaries have to be placed where the 'fully developed' assumption is either valid or has little impact on flow inside the domain, i.e. away from any fire and at locations where the flow is not expected to experience strong spatial variations.

Where flow is driven principally by mechanisms external to the computational domain, such as forced ventilation, the boundaries of the computational domain should be located where the flow conditions are known and can be specified.

Specification of the flow at the boundaries might require further analysis which can be provided either by measurements or additional modelling (including CFD). Detailed analysis is necessary when the flow across the boundary is expected to be complex, e.g. in a space partially open to the atmosphere and for which the surroundings influence the direction and velocity of the incoming wind.

Walls

In order to save computing time, universal wall laws are often applied as wall boundary conditions. These functions preclude the need to resolve in detail the large gradients of temperature and velocity near walls, which would necessitate a large number of mesh cells.

Instead, momentum and convective heat fluxes between the near-wall nodes of the computational mesh and the walls themselves are assumed to be described by 'universal laws of the wall'. These laws include parameters that account for the roughness of the walls and lead to lower velocities in the case of rough walls. For these laws to be valid, the near-wall cell mesh size must be chosen such that the first mesh nodes are specified to be at a distance from the wall which is related to the local turbulent Reynolds number.

Uncertainties associated with the use of wall laws are of two types: those due to the difficulty in complying with restrictions on the location of the near-wall nodes across the whole domain, and those due to the fact that the wall laws are strictly only valid for idealised situations.

The CFD user has to specify how heat transfer is to be modelled at the walls. One possibility is to assume nil heat transfer, i.e. an adiabatic wall. The other extreme is to assume a constant wall temperature, leading to maximum rates of heat transfer. The heating of the wall can also be modelled, by solving for thermal conduction within the wall (requiring a much finer mesh resolution near the wall and specification of the properties of the wall).

Typically, the adiabatic wall assumption leads to faster smoke propagation, with the smoke being more concentrated in the hot gas layer near the ceiling and, therefore, less smoke being predicted at lower levels, when compared with the fixed wall temperature approximation.

Radiation adds further complications, which are not dealt with here.

Fire-dependent Conditions

As the fire is growing, it may change the conditions inside the computational domain. Window and / or structural component failure will lead to a change of boundary conditions in the vicinity of the failure.

Guidance on the assessment of such behaviours is outside the scope of this document.

Summary

- Forced ventilation is usually prescribed as a boundary condition and should be remote from any fire source.
- The presence of leakage flows can have a significant effect on the flow inside the computational domain, particularly when that flow is governed by convection generated by a fire rather than by forced ventilation.
- When flow through a boundary is governed by fire-generated convection, it is usual to prescribe a pressure boundary condition located remote from the fire.
- Heat loss to the walls can have important effect on smoke movement, particularly when the fire products are in close and immediate proximity with walls. In general, a conservative approach to rates of lateral smoke propagation is to assume no heat loss at walls, i.e. an adiabatic condition. Note, however, that this approach may then under-estimate smoke concentrations at low levels. (If the latter is an important consideration it may be appropriate to prescribe a fixed temperature to the wall and undertake a second simulation for this condition.)

The results of CFD simulations are determined by the boundary conditions. It is essential, therefore, that the user specifies boundary conditions correctly and understands the key role that they play. However, usually not all of the required boundary conditions will be well-defined. For instance: turbulence parameters as flow enters the computational domain are typically unknown; there may be uncertainty in wall heat transfer coefficients; fire sources and fire growth rates or heat loading may be ill-defined; events external to the selected computational domain, such as pressure distributions arising from natural or forced ventilation, may in reality affect flow inside the domain - these couplings should ideally be encompassed. If doubt remains, the CFD user should ideally carry out a sensitivity analysis to evaluate the influence of a range of plausible values for boundary conditions on the predictions.

6. Presentation and analysis of results.

6.1 General

CFD modelling outputs are generated to inform as to the success of a comparative or deterministic analysis. Therefore solutions should be presented with evidence of their performance explicitly shown in relation to agreed acceptance criteria.

Visual and graphical results should be shown in relation to this agreed acceptance criteria and displayed with reference to location and, where applicable, orientation.

If a time dependent CFD model is used and if conditions must reach steady state in order for acceptance then a timeline should be shown at regular intervals to demonstrate that conditions have become stable. Additional images may be shown at times of unusual flow behaviour, to demonstrate the action of fans up close, etc.

The CFD output generated requires post processing before results can be presented. Care should be taken to ensure the results reflect accurately the physical scenario being investigated. Where applicable, a sensitivity analysis should be conducted.

It is recommended that where an approving authority is unfamiliar with the model used or do not have the necessary technical knowledge to assess a model that a peer review be considered.

6.2 Analysis of Results

Acceptance criteria

Before detailed modelling is completed and results presented it is vital that outputs are agreed (with respect to both the design objectives and the acceptance criteria) and approval for the modelling methodology is attained. Part of this agreement should detail a method of assessing the model's performance in relation to prescribed values.

Where a comparative approach is used, results should be compared directly to the agreed code compliant solution with variables and areas of comparison agreed.

Where a deterministic approach is used, limits for visibility, temperatures, radiation etc should be agreed.

Specific Issues

There are seven main issues requiring consideration. These relate to:

- *Vehicle emission ventilation*
- *Smoke clearance*
- *Safety of evacuating occupants*
- *Safety of fire service personnel & their ability to attack the fire*
- *Fire spread and local effects*
- *Performance throughout the car park*
- *Error checking*

Not all of these issues are necessarily relevant for all projects. The relevant issues should be selected for each project. The following sections outline the key aspects which should be considered in each area and be shown to be acceptable to the approving authority.

It is not the intent of this document to set specific acceptance criteria. Rather these should be agreed with the approving authorities based on recognised published documents, e.g. BS7974, CIBSE Guide, Approved Documents B and F.

Vehicle emission ventilation

Objective: The objective should be to show that the whole car park is adequately ventilated and that either the ventilation rate or the maximum CO level meets the recommendations of Approved Document F to the Building Regulations (or equivalent outside England and Wales).

Comparative analysis:

The Approved Document sets some basic prescriptive requirements for vehicle emission ventilation. Where these are not followed CFD analysis can be used to show equivalence.

The simplest way to show equivalence is to demonstrate that the overall ventilation rate matches the basic prescriptive requirement and that the car park has no pockets of stagnant air.

Deterministic analysis:

A deterministic approach would be to show that, under normal and peak traffic in the car park, CO levels do not exceed the recommendations in the Approved Document. This approach requires an understanding of likely traffic flows through the car park and of vehicle CO emissions.

Smoke clearance

Objective: The objective should be to show that the whole car park is adequately ventilated and that the ventilation rate meets the recommendations of Approved Document B to the Building Regulations (or equivalent outside England and Wales).

Comparative analysis:

The Approved Document sets some basic prescriptive requirements for smoke clearance. Where these are not followed CFD analysis can be used to show equivalence.

A way to show equivalence is to demonstrate that the overall ventilation rate matches the basic prescriptive requirement and that the car park has no pockets of stagnant air.

Safety of Evacuating Occupants

Objective: The objective of this stage should be to show that occupants can reach a place of relative safety during a fire.

Comparative analysis:

Primary escape routes should be included in any comparative analysis and conditions shown to be equal to or better than the agreed code compliant solution used for comparison. Comparisons should include, where appropriate, visibility, temperature and radiation.

Deterministic analysis:

Under a deterministic approach the objective should be to show that the Available Safe Egress Time (ASET) for the occupants will be greater than the Required Safe Egress Time (RSET) plus a suitable safety margin in the particular scenario being modelled.

In assessing the safety of occupants as they evacuate it is recommended that a recognised approach be used. Potential methods include for example:

- Clear layer assessment
- Tenability criteria

Clear layer assessment requires that as the occupants travel along the evacuation routes they are not exposed to smoke and that any smoke above the occupants is maintained at a temperature low enough that occupants are not affected by untenable levels of heat radiation. This can be difficult to achieve in car parks, where the headroom is usually restricted.

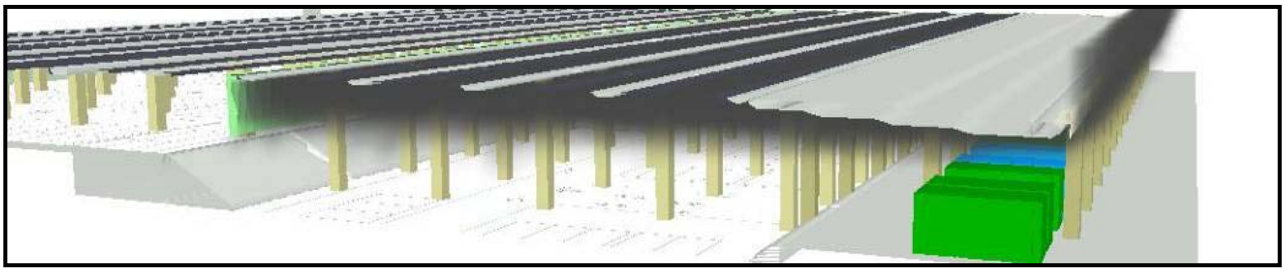

Figure 5: Maintenance of a clear layer below smoke

Tenability requires that it be shown that occupants escape in tenable conditions and that their exposure to heat and smoke is limited. This approach typically requires that visibility, temperature, radiation, CO and CO_2 be assessed and that a Fractional Equivalent Dose (FED) type analysis carried out.

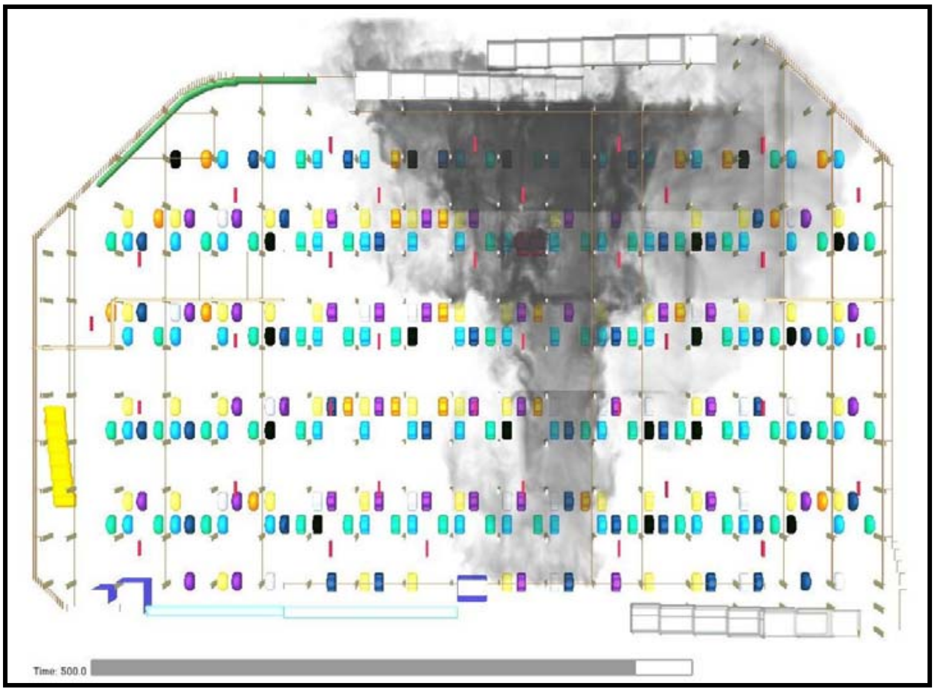

Figure 6: Limiting spread of smoke using jet fans to maintain tenable conditions

Safety of fire service personnel & their ability to attack the fire

Objective: The objective of this stage should be to show that fire service personnel will be able to enter the car park and safely reach a position where they can attack the fire.

Comparative analysis:

Primary fire service access routes should be included in any comparative analysis and conditions shown to be equal to or better than the agreed code compliant solution used for comparison. Comparisons should include where appropriate visibility, temperature and radiation.

Deterministic analysis:

Once occupants have been safely evacuated it should be shown that the proposed system will allow fire service personnel to safely enter the fire zone and attack the fire.

Unlike evacuating occupants, fire service personnel can be expected to have the additional protection provided by breathing apparatus and protective clothing. As such the tenability criteria and needs of fire service personal are very different from evacuating occupants. These needs include the ability to:

- Find the location of the fire.
- Gain safe access to a position from which to deliver water to the base of the fire.
- Retreat to an area of relative safety if necessary.

It is recommended that the tenability limits for the fire service be considered at the outset of the project and agreed with the local fire service prior to any modelling being carried out as these may be dependent on the particular equipment local fire service personnel have available.

General criteria may however be as suggested in Section 10 of BS 7346-7; designs should be such that fire fighters can move through substantially clear smoke-free air when approaching the fire up to a distance of 10m from that fire

Fire Spread and local effects

Objective: The objective is to show the effect of the fire on any near by objects including the structure, flammable materials and any required smoke control systems such as fans.

Comparative analysis:

Key nearby objects should be included in any comparative analysis and conditions shown to be equal to or better than the agreed code compliant solution used for comparison. Comparisons should include, where appropriate, temperature, radiation and the effects of flame spread.

Deterministic analysis:

Under a deterministic approach the intent should be to show that these near by objects will not be detrimentally affected or that, if they are, the safety of occupants or the fire service will not be affected.

The assessment of the effect on near by objects will depend on the type of objects that are located near by. The following outlines some of the potential areas that may need consideration. It is by no means a comprehensive list and should be used as indicative of the types of investigation that may take place.

Structural Stability: It should be shown that during an extended period of fire the surrounding structure of the car park will maintain its structural integrity to afford evacuating occupants time to escape and the fire service sufficient time to fight the fire without collapse of the building

Fire Spread: The potential for fire spread to surrounding flammable materials should be assessed. This may include the potential for the fire to spread to adjacent vehicles, fixed insulation or storage areas. The intent of this analysis is to confirm that the selected fire size is appropriate and that the fire will be unlikely to grow further than that modelled.

System Failure: It should be shown that any fire safety related system will either not be detrimentally affected by the fire or, if affected, its failure will not result in risk of life to evacuating occupants or the fire service.

Study of the above areas can be carried out using a quantitative and/or qualitative approach depending on the particular configuration within the car park. In either case, generally it will be necessary to consider conductive, convective and radiative effects from the fire and smoke.

Performance throughout the car park

Objective: The objective is to show that the proposed system will offer the necessary levels of protection throughout the entire car parking area for any credible fire location.

Comparative and Deterministic analysis:

Typically this can be achieved by carrying out one or more CFD models with the selected design fire located in the worst credible position(s). The position(s) should be selected taking into account the car park geometry, selected smoke management system and routes of escape. The objective should be to ensure tenable conditions throughout the car park for evacuating occupants and fire service personal entry.

If the required conditions (identified previously in this section based on a comparative or deterministic approach) can be shown to exist for both the escaping occupants and the entering fire service personal for a fire in the selected position(s), then it can be reasonably assumed that that the selected smoke management system will provide adequate protection for a fire in other locations within the car park. For this assumption to be made then it is critical that the fire locations selected are indeed the worst credible positions.

The selection of the worst credible fire locations can be identified through the use of quantitative and/or qualitative methods. However it is recommended that the locations be discussed and agreed with the relevant authorities at an early stage in the project.

Error checking

Objective: The objective is to allow assessment of the model in terms of error checking.

Comparative and deterministic analysis:

One of the most important aspects of any modelling presentation is that the approving authority or checking party be provided with sufficient information to allow a model to be checked for general errors. It is not intended that the modeller provide sufficient information under this heading for the specific CFD package being used to be assessed, (as the package being used should already have been assessed to determine its suitability and its use been agreed with the relevant authorities) rather that errors within the specific simulation have not occurred.

Flow patterns: It is important to confirm the flow pattern at the fans is correct, and that there are no dead zones or stagnant areas in the flow. If there are any ramps in the car park then the flow through between floors should be carefully checked for anomalies & for overly strong or weak flows. As general good working practice for the modeller it is important to highlight any abnormal results early on, before being committed to final runs.

Convergence criteria: The output of steady state models should be checked for convergence criteria if applicable (it is noted that models vary in the type of convergence criteria or convergence bounds used and hence this should be related to the specific model).

Plausibility checks: Flame heights, smoke temperatures and plume mass flow rates should be checked against historical data or hand calculations where possible to ensure that the correct fire conditions are being created.

6.4 Presentation of Results

The results of the analysis should be documented and may be provided in the form of a report, with any necessary animations attached in electronic form (CD, DVD, etc).

The documentation should include at least the following information:

- A description of the car park and the ventilation system
- The design criteria and objectives of the analysis
- The scenarios investigated
- Details of the CFD model
- The results of the analysis
- A statement as to whether the design criteria and objectives have been met

Images taken of the results must clearly explain what is being shown, and the location of the local effects.

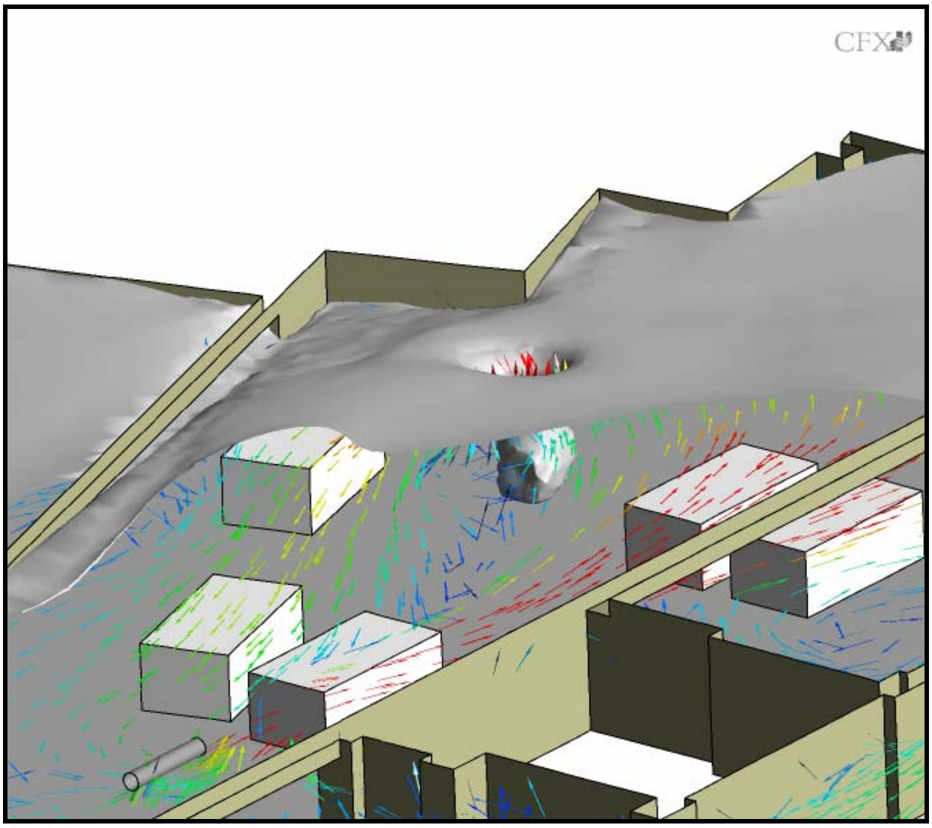

Figure 7: Image showing smoke envelope and air movement local to the fire

For time dependent analyses, graphical results should be presented wherever possible to qualitatively show conditions plotted against a timeline (although it is recognised that this is not always possible).

Scales, ranges and units should be chosen accordingly to reflect the range of data collected; velocities, temperatures etc. It is suggested that the chosen data ranges should be universally adopted for every presented image to ensure all results are directly comparable. However it is recognised that in some cases of detailed analysis this is not possible when highlighting specific phenomena.

Sensitivity analysis should be carried out and presented such that it allows important outputs between different scenarios to be easily compared.

The locations of the images taken must be relevant to the issues being investigated, showing the flow regime at areas of interest, and at relevant elevations or regions of activity.
It is useful to display images of velocity vectors at the mechanical fans to demonstrate their effects.
Images of temperature and visibility (or smoke obscuration) should be displayed with reference to the agreed tenable bounds.

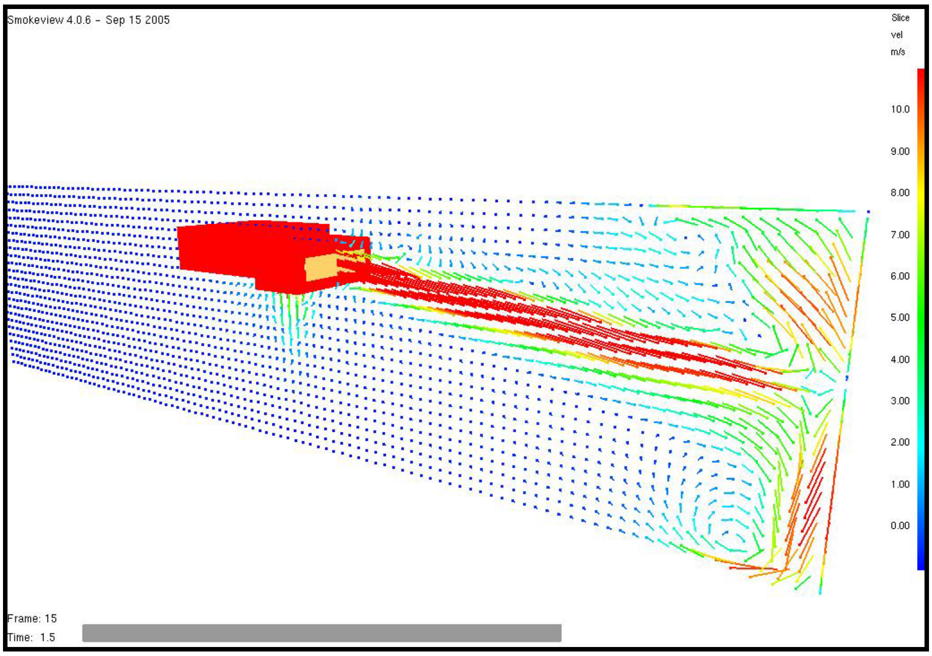

Figure 8: Section showing velocities at jet fan location

Annex A Physical sub models

Combustion Models

There are two modelling approaches that are commonly adopted to represent the combustion processes.

Volumetric Heat Source Model.

> This model does not predict the release of heat and smoke in the flame. The quantities of heat and smoke released by the fire together with the volume of flame where the releases occur are specified by the user. The distributions of heat and smoke released are assumed to be uniform over the flame volume. The model predicts the transport of heat and smoke away from the fire.
>
> The need to prescribe the volume of the flaming region is a limitation of this approach.
>
> This approach is widely used to simulate the movement of smoke in large spaces, i.e. car parks.

Combustion Model.

> Typically, these models aim to predict, in a simplified manner, the chemical reactions that happen in the flame. While the overall quantity of heat released and the area of the fire have still to be specified by the user, the non-uniform distribution of heat in the flame region is predicted and aims to take into account the influence of the local flow.

The model employed should represent the behaviour of the flaming region and the thermal plume adequately as this will affect the transport of smoke away from the fire.

Buoyancy Models

The Boussinesq approximation is commonly used to model buoyancy in flows in which the temperature gradients are small (to a maximum of a few tens of Kelvin). The model assumes that the density is constant in most of the momentum equations, adopting a linear proportionality with temperature in the gravitational term only.

Fire and smoke movement studies require the simulation of flows involving heat transfer and, therefore, the fluid properties, including density, are functions of temperature. The temperature gradients are, typically, significantly greater than those for which the Boussinesq approximation is valid.

It is recommended that an equation of state be used to represent the buoyancy effect.

Turbulence Models

Turbulent flows associated with car parks contain a wide range of length (less than a millimetre to a few metres) and time scales.

- *Large Eddy Simulation (LES)* based simulators resolve all but the smallest turbulent motions within the Navier-Stokes equations. The smallest eddies are either ignored (typically) or modelled by a sub-mesh approximation.

  Mesh resolution is, therefore, a function of the need to ensure that the turbulent flows are represented adequately within the LES model.

- *Reynolds-Averaged Navier-Stokes (RANS)* based simulators adopt a time-averaged representation of the Navier-Stokes equations. The equations so obtained do not aim to resolve the turbulent motions. They do, however, provide the time-averaged characteristic quantities of the flow.

  Since a RANS model is not trying to resolve the turbulent motions, the mesh needs only to be fine enough to capture the important time-averaged features of the flow.

The limitation of the RANS models lies in the modelling of the turbulent Reynolds stresses, i.e. the unknown terms appearing in the equations as a result of the time-averaging process.

The k-ε (eddy viscosity) turbulence model (and its variants) is the most likely model to be employed for fire and smoke movement in complex spaces (e.g. non-rectangular, multi-level / multi-storey buildings incorporating ramps) in a RANS simulation. This model is based on the assumption that the Reynolds stresses are linearly related to the local mean strain rate. This assumption is, however, not strictly valid in flows involving buoyancy, streamline curvature, acceleration / deceleration, impingement and three-dimensionality, all of which are features of fires in enclosed spaces. The equations employed in this model require that the user specifies a set of constants: these have well-established values which, although determined empirically by comparison with data from idealised experimental configurations, have proved acceptable for a wide variety of flows.

More sophisticated RANS models derive a transport equation to predict the Reynolds stresses. However, the equation obtained contains unknown terms which again need to be modelled. The higher the order of modelling applied, the lesser the effect on the results it is expected to have. The drawbacks of these more sophisticated models are that they involve solving more equations and, due to the mathematical nature of the equations, they are also more numerically unstable (requiring much more effort to obtain a ‘converged’ solution).

In principle, the second moment closure turbulence models are better suited to predict flows dominated by buoyancy forces. The computing time, however, is drastically increased compared to the eddy viscosity models, not only because of the larger number of equations to be solved, but because these models are unstable and less robust.

An acceptable alternative to second moment turbulence models, are k-ε turbulence models in which both the turbulent intensity, k, and turbulent viscosity, ε, equations are modified to take account of the buoyancy effects on turbulent mixing. Simpler eddy viscosity models that solve none, or only one, of the transport equations characterising the turbulence should not be used – without the guidance of the developers – as they are unlikely to represent the effects of buoyancy on turbulence.

Radiation Models

Radiative heat transfer occurs between the emitters and receivers (i.e. between solid surfaces, the soot / gas phase mixture of flames and smoke aerosol) and is an important feature of the heat transfer processes in combusting flows when the temperatures are above typically 600K.

The primary sources of radiation are CO, CO_2, CH_4 and H_2O (which emit energy in discrete bands) and soot (which emits radiation at all wavelengths).

There are four main modelling approaches.

- *Fractional Heat Loss Model*

  The heat output of the fire is represented by the convective heat fraction only.

  Heat loss from the fire as a result of radiation to the surroundings is, typically, ignored.

  (An alternative to ignoring the radiative fraction is to define *a priori* a prescribed heat distribution to regions of the model to represent the radiative heat transfer to the region from the fire. The uncertainties arise from the specification of the prescribed radiative heat transfer process.)

  (Radiative heat transfer between the smoke and the walls can be expressed as a function of the temperature of the wall and smoke and the emissivity of the smoke. It is applied between the wall and the fluid cell next to the wall.)

- *Six Flux Model*

This method is applicable to structured meshes using quadrilateral cells to form the mesh.

Radiation is assumed to be transmitted along the local axes of the cell such that the radiant flux across each of the six faces of the mesh cells is uniform. This simplifies the set of equations to be solved to calculate the radiative source term but the accuracy is highly directionally dependent.

- *Discrete Transfer Model*

  This model aims to solve the representative discrete radiative rays only. The directions of the rays are specified *a priori*. The solution for any particular ray is restricted to the path between two boundary walls rather than being partially reflected at walls and being tracked to extinction. The accuracy of the discrete transfer model is dependent on the ray directions chosen as well as the number of rays.

  The discrete transfer model is not ideally suited to the body fitted meshes likely to be seen in fire and smoke movement applications in complex spaces. This is because it is computationally expensive, especially for situations where a large number of rays are required to obtain an accurate solution.

- *Monte Carlo Simulation Model*

  A number of rays are ‘emitted’ in (pseudo-) random directions and are then traced until they hit an obstacle / wall or exit the computational domain.

  The quality of the heat transfer calculations is dependent on the number of rays. This is a costly approach that potentially offers the most accurate solution and flexibility for complex geometries.

All but the first approach require the calculation of local emissive powers and absorptivities, which depend on the composition of the soot / gas mixture. This is a complex process for which guidance should be sought from the developer of the CFD simulator.

When employing a volumetric heat source combustion model, the simplest method of accounting for radiation loss, i.e. the fractional heat loss model, is adequate. This approach, however, only accounts for the radiative loss of the flaming region and ignores other radiative heat transfer, in particular the transfer from hot smoke to walls and the transfer within the smoke. Including the radiative heat transfer at the walls may improve the simulation prediction, but the simulation will still ignore the radiative transfer within the gas. However, employing more sophisticated approaches to account for radiation in combination with a volumetric heat source model is unlikely to increase the accuracy of the simulation as a direct result of the assumption of uniform heat distribution in the flaming region.

Any radiation model can be combined with any combustion model.

Annex B Check list

Guidance to help the designer / CFD Engineer follow an approved procedure for Zone modelling and / or CFD modelling.

Stage 1 – Pre CFD / zone modelling

Item no.	Description	Completed
1	Define objectives of the Zone or CFD modelling.	
2	Determine type and number of simulation(s) to be prepared to demonstrate design objectives.	
3	Collate reference material and / or any previous test results for use later when checking the sensibility of results.	

Stage 2 – CFD / zone modelling

Item no.	Description	Completed
1	Selection of computational fluid domain boundary.	
2	Selection of geometric detail to be represented in the computational domain.	
3	Creation of the geometric model(s).	
4	Mesh / grid generation.	
5	Define physics for the simulation(s).	
6	Select appropriate 'Sub models' (if applicable) including definition of sources (and / or species) within the model (i.e. fire, contamination etc).	
7	Define appropriate boundary conditions.	
8	Define appropriate initial conditions.	
9	Select solver time / number of iterations, results to be obtained from the solver, monitor points etc.	
10	Plausibility check - do the results provide a reasonable representation of real life events?	

For further details of how stages 1 and 2 should be implemented please refer to section 5.1 and any section thereafter relevant to the CFD simulation being carried out.

Stage 3 – Presentation of results

Item no.	Description	Completed
1	Description of the objectives.	
2	Description of the geometric model(s) and simulation(s).	
3	Decide how the results will need to be presented to demonstrate objectives i.e. using velocity, speed, temperature, visibility, etc	
4	Description of the results for the simulation(s).	
5	Discussion of the results.	
6	Conclusions - Have the objectives been achieved? If not decide further actions to be taken.	

For further details of how stage 3 should be implemented please refer to section 6.4

Annex C Testing and Validation

Before using a CFD software product, particularly for fire modelling, it is incumbent on the user to establish the suitability of that package. The issue is whether the software has gone through a testing or validation process that shows that it is suitable for the demands being placed upon it.

To give confidence in the software there ought to be information in the literature or product documentation to support its use. Otherwise, the developers should be contacted to establish a first view of applicability. This should be followed up by tests, related to the proposed use, to compare simulation results with some expected outcome.

If documented evidence exists that the software has been used successfully, such evidence can be valuable in helping set user-controlled parameters to maximise the effectiveness of the modelling.

It should be pointed out that validation is a continuing process, and that in the present context "testing" is a more appropriate word.

The detailed capability-requirements for representing the physical processes thought important in fire modelling are discussed elsewhere in this document. But in a real-world application the question is whether the software is able to model what is required to a reasonable degree of accuracy. In this context a reasonable degree of accuracy might be +/- 20%. However, a first test is that of plausibility – do the results look right? Results ought to confirm the expectation of experienced engineers. If not, then more questions should be asked. If new or unexpected features are exposed then these features need to be understood in the light of established knowledge.

Although the emphasis is on application to fire modelling, there are often important requirements for general day-to-day ventilation for dispersal of pollutants and also for post-fire smoke clearance performance. The comments above also relate here.

A further requirement under the broad heading of validation relates to the user of the software. Does the user have sufficient knowledge and experience? It is well-known that to get the best from CFD requires both an understanding of the physical processes of thermodynamics and fluid mechanics and the careful exercise of engineering judgement. Whilst the former can be acquired in a university environment the latter requires a broader exposure to the design process. The ability to interpret practical engineering issues in the context of the sophisticated numerics implicit in a CFD package is important. If the user does not have the comprehensive experience necessary then the advice of colleagues with a complementary experience is vital.

Annex D Additional reading

1 BS 7346-7:2006, Components for smoke and heat control systems – Code of practice on functional recommendations and calculation methods for smoke and heat control systems for covered car parks.

2 EUR 18868 EN, Development of Design Rules for Steel Structures to Natural Fire in Large Compartments, European Commission, Technical Steel Research Report, 1999.

3 CIBSE Guide E. Fire Engineering Handbook. 1997 ISBN: 1903287316

4 Gupta A. K, Kumar R, Yadav P. K, Naveen M. Fire Safety Through Mathematical Modelling. Current Science, Vol 80, No 1, 10 January 2001

5 Grandison A. J, Barnett J. R, Patel M. K. Fire Modelling Standards/Benchmarks. Fire Safety Engineering Group, University of Greenwich

6 Ferziger J. H, Peric M. Computational Methods for Fluid Dynamics. 3rd Edition. ISBN 3-540-42074-6

7 Versteeg H. K, Malalasekera W. An Introduction to Computational Fluid Dynamics. The Finite Volume Method.

8 Guidance for HSE inspectors, Smoke movement in complex enclosed spaces – Assessment of Computational Fluid Dynamics

9 Fire Dynamics Simulator User Guide, Kevin McGrattan et al; www.nist.gov/fds

10 The Use of CFD Computer Models for Fire Safety Design in Buildings: Large Warehouse Case Study; Brian Hume and Mick Eady

11 Development of Standards for Field Models; Fire Research Report number 85, November 2003, Brian Hume, ODPM,

12 Assessment of Vehicle Fires in New Zealand Parking Buildings, Yuguang Li. Fire Engineering Research Report 04/02, May 2004

13 Smoke Reservoirs – an evaluation of CFD modelling as a design tool; Jacob Hagman, Fredrik Magnusson. Department of Fire Safety Engineering, Lund University, Sweden, Brandteknik, Lunds tekniska högskola, Lunds universitet, Report 5130, Lund 2004

14 A Computer Model of Fire Spread from Engine to Passenger Compartments in Post-Collision Vehicles; James A. Ierardi. May 1999

15 On the dynamics of vehicles and electrical equipment; Johan Mangs, VTT Building and Transport. Academic Dissertation. University of Helsinki.

16 Fire Statistics, United Kingdom, 2004. ODPM

17 Natural Fire Modelling of Large Spaces; E S Korhonen, 21/08/2000. Academic Dissertation, Helsinki University of Technology.

18 US Vehicle Fire Patterns and Trends; Marty Ahrens, Fire Analysis and Research Division, National Fire Protection Association, August 2005. NFPA

Published by the

HEVAC ASSOCIATION

The Building Services Division of the
FEDERATION OF ENVIRONMENTAL TRADE ASSOCIATIONS (FETA)
2 Waltham Court, Milley Lane, Hare Hatch
Reading, Berks RG10 9TH

Tel 0118 940 3416
Fax 0118 940 6258
e-mail: info@feta.co.uk
Web www.feta.co.uk