Smoke Control Association

SCA guidance on CFD analysis for Smoke Control design in Buildings

Edition 1.2: June 2021

Acknowledgements

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

Marios Alexandrou	Buro Happold
Cristina Paduano	B-Fluid Building Fluid Dynamics Ltd.
Patrick Okolo	B-Fluid Building Fluid Dynamics Ltd.
Conor Logan	Colt International Ltd
Donal Sheridan	Fire Design Solutions Ltd
Mike Duggan	HEVAC
Stewart Miles	International Fire Consultants Ltd
Ben Atkinson	Jenson Hughes
Paul Kingston	K8T Ltd
Paul Grimwood	Kent Fire and Rescue Service
Gareth Steele	London Fire Brigade
Klementina Gerova	NHBC
Maulik Katkoria	NHBC
Paul Hanson	Royal Borough of Kensington & Chelsea
Matt Salisbury	Salisbury Fire Ltd.
Nelson Godinho (Chairman of Working Group)	SE Controls Ltd
James Bertwistle	WSP Fire Engineering

Contents

1 Introduction

Computational Fluid Dynamics, or CFD, is an established technique using numerical methods to analyse problems involving fluid flows. The physical properties that define the fluid flow, such as pressure, temperature and velocity are dependent variables in a mathematical model describing the fluid flow. This mathematical model defines the flow field at any point in space and is defined as a series of partial differential equations (PDE's).

The continuous increase in computational power has made CFD a popular tool among practitioners, engineers and researchers in many fields, including fire and smoke ventilation engineering. Performing CFD simulations and obtaining the results for a specific test is not a difficult task thanks to the numerous available commercial CFD packages.

The purpose of this guide is to give an outline of the basics requirements that should be considered when developing a CFD simulation of a smoke control system. It is also hoped that this guide will give a general understanding of the challenges involved in preparing CFD simulations to help those tasked with approving such systems a better understanding of how erroneous results can be identified and what supporting information and documentation should be expected.

Above all it important that all parties are clear on the context of the CFD results – CFD software alone is not a design tool, it only predicts the performance of a design given a particular set of conditions. Other assumptions and design conditions may present an entirely different set of results. The skill of the CFD engineer is to understand the limitations of the software and present the results in such a way that all the assumptions used in the model preparation are justified and relevant and above all, traceable, without allowing the graphical nature of the outputs to conceal any shortcomings in the design.

2 Background

The use of computer models for simulating fires in enclosures has increased substantially in recent years. The increased emergence of performance-based solutions within the construction industry together with more complex and unique building geometries, has led designers, engineers and local authorities to rely on computer modelling to analyse and evaluate various elements of design within buildings, and fire safety is no exception. This is especially relevant in complex structures where the validity of simple hand calculations is limited. However, using computer modelling in fire safety engineering design is not always straightforward. The user must have a fundamental understanding of the science associated with the models in order to assess the validity and accuracy of the simulation results.

However, the ability to access CFD tools relatively easily and cheaply has led to an increase in the number of inexperienced users with insufficient knowledge of the detail that sits behind the colourful images and animations. This can be misleading, not only to the user, who may not recognise the errors embedded in the information presented but also to those tasked with approving designs which have used CFD as part of their validation.

In fire and smoke modelling, it is strongly recommended that those tasked to perform CFD modelling have a good knowledge of the underlying physics and thermodynamics associated with fire dynamics and smoke behaviour. It is also equally important to understand the implicit limitations of the CFD approach and sensitivity of a solution to assumptions relating to both model and scenario definition.

The main advantage of using CFD for validation of a system design is that this can be carried out prior to construction, 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 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 analysed building.

The increased emergence of performance-based regulations and hence solutions within the fire safety industry together with more complex and unique building geometries, have led designers, architects and local authorities to rely on computer modelling to analyse and evaluate the smoke and heat transfer within buildings. This is especially relevant in complex structures where the validity of simple hand calculations is limited. However, using computer modelling in fire safety engineering design should not be considered simple or easy. It is important that the user who creates and processes the CFD model has a fundamental understanding of the physics and chemistry associated with the models in order to assess the validity and accuracy of the simulation results.

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 of the analysis. CFD modelling is too expensive and time-consuming process to be carried out without this agreement. Advice and guidance on these issues is provided in this document.

3 Terms and 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 a method used for analytical solution of thermodynamic mathematical equations that simulate the flow of fluids, heat transfer and other associated phenomena, using computing processing power and memory. (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 based on physical relationships derived from scientific theories and empirical results, that given a set of initial conditions will always produce the same outcome. 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 building, all of the building or all of the building and some surrounding areas.

3.8 Developing Fire

Fire Development is a function of many factors including: fuel properties, fuel quantity, ventilation (natural or mechanical), compartment geometry (volume and ceiling height), location of fire, and ambient conditions (temperature, wind, etc) changing with time.

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 outcome of splitting up the computational domain (discretisation) into a number of elements or cells defining the discrete points at which the numerical solution is computed. The points are normally the cell centres or cell vertices.

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 dependant 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 Modeller Experience and Qualifications

When undertaking CFD modelling to design or validate a smoke control system, there is no single definitive approach which can be used for all buildings since interactions between the building, fire, and its occupants can be highly complex. As such, a greater degree of care and responsibility by the designer is required, therefore it is essential that application of CFD modelling be entrusted to a suitably qualified and experienced personnel. This means that that the individual or group entrusted to undertake the modelling has the relevant skills, qualifications, training, experience and professional liability cover.

Normally the individual or group appointed to carry out the CFD modelling are expected to demonstrate that they have experience of successfully working on similar schemes, that they are appropriately qualified and have the appropriate professional status or can prove they are adequately competent (e.g. showing sufficient experience), whilst working within their scope of expertise and ethical engineering practices. A designer unable to demonstrate any of these competence attributes should be mentored or supervised and their work quality assured by someone who does, following the principles as described in BS 7974.

There is a similar expectation on the individual or group tasked with reviewing the study, that they are appropriately competent or employ a third party who is competent in the relevant area. This is further discussed in Appendix E.

5 Qualitative Design Review

The early part of the design process, i.e. its conceptual stage is the ideal time to optimise the design of a building and its fire safety features whilst minimising disturbance elsewhere to the building. Although amendments to the fire safety features or their optimisation does often become necessary within the build programme, the frequency and scale of this can be minimised with the effective use of the Qualitative Design Review (QDR) technique detailed in BS 7974.

In brief, the QDR is a qualitative process that allows the Fire Engineers / CFD modeller supported by stakeholders to use their experience and knowledge to critically analyse the design problem, develop fire safety objectives and quantitative assessment criteria.

QDR team major stakeholders may include:

- Design Fire Engineer/ CFD Modeller
- Checking Fire Engineer (and/or Building Control Surveyor)
- Specialist Installing Engineer

- Fire Service
- Architect
- Mechanical & Electrical Services Engineer
- Structural Engineer
- Operational Management
- Insurer/ Surveyors
- Client/ developer/ builder

The CFD modeller, supported by the team should work towards identifying representative fire scenarios and design fires that can be regarded as worst-case fires that may affect the fire safety objectives. A structured approach is required to ensure that hazards are not missed or overlooked and that the final design and strategy is able to meet the fire safety objectives.

Items for consideration:

- Review the architectural design. There may be architectural features or floor plan layouts which cannot be justified in which case these would need to be amended.

- Establish the fire safety objectives. This may be connected to the requirements of the Building Regulations and guidance may be sought from Approved Document B or another relevant standard.

- Agree tools and software that is fit for purpose and appropriate to model and capture the physics in the resulting analysis.

- Identify fire hazards and possible risks, this should include possible ignition sources, combustible fixtures and content, materials of construction,

- Establish trial fire safety designs

- Identify acceptance criteria and methods of analysis. Acceptance criteria should be justified using established values where practicable rather than individual standalone studies.

- Establish Fire Scenarios for analysis.

- Establish CFD review process

All findings from the QDR should be documented with clear reasoning so that all stakeholders can understand it and either approve or comment upon it ahead of the final design being submitted for building regulations approval.

Note; apartment blocks are usually planned with 'Computer Aided Design' (CAD), these drawings should be made available to the CFD modeller constructing the model. The CFD model should be designed around the design intent and any conflicts with other services, fixtures or the challenging geometry can be identified (such as curved walls, columns, down-stand beams etc.) at an early stage. Similarly, 'Building Information Modelling' (BIM) is becoming widely used, where BIM exists it should be shared to encourage spatial co-ordination between all stakeholders as information can be updated from anywhere including site.

Further information on the QDR process is available in BS 7974.

6 Limitations

The application of CFD in any area of expertise requires a fair amount of knowledge and experience with both CFD and the phenomena under investigation. Until recently, only CFD researchers and design specialists held the amount of knowledge regarding building properties, installations and CFD that was needed to successfully apply these simulation techniques in building design.

However, in the past years some software and research establishments developed tools aimed at less expert users. These tools include advanced techniques that automate much of the data specification process for common situations. Some were specifically developed for use in building practice and the built environment. However, even user-friendly CFD applications still require a fair amount of input.

In addition, sufficient knowledge concerning airflow and heat transfer mechanisms is needed in order to formulate a useful CFD model.

1) CFD solutions can only be as accurate as the physical models on which they are based.

2) Solving equations on a computer invariably introduces numerical errors such as:
    a. Rounding off Error, due to finite word size available on the computer.
    b. Truncation error, due to approximations in the numerical models. These errors will tend to zero as the grid is refined but this is not always feasible due to limited resources and the software being used.

3) Boundary conditions, as with physical models, can only be as accurate as the initial and boundary parameters specified to the numerical model.

4) Physical phenomenon, a definitive understanding of physical phenomenon is needed so one can determine what assumptions can enable a convenient case through computational domain (specifications of fluids, turbulence model).

5) Experience in conducting CFD studies should make one aware of the limitations to modelling turbulence. As turbulence, there are still lots of phenomenon to discover to generate an analytic approach.

6) Numerical method is all about mathematical knowledge. Partial differential equations based mathematical topics are important. Modellers have to conflict optimal numerical method to solve those equations. They will need to understand what the problem is and develop a method to solve it.

7) Computational grid (Meshing) considers the numerical approach based on one of the several numerical approaches such as finite volume method for example. Having the wrong generated mesh structure, could results through the solution process that are non-physical.

8) CFD tool with a commercial code that should use any numerical solver to generate reasonable solution grids. However, the use of free research and shareware tools are not always subject to the same level of quality assurance.

To describe a physical phenomenon into the computational domain, it is recognised that there are several limitations that will need to be optimize. An awareness of every sub-sections from beginning to end is important. In case of a commercial program, some of these sub-sections may be directly optimized into program. There is nothing inexplicably needed to be sufficiently educated and capable of using CFD tools to apply the most appropriate analysis however, an awareness of its limitations is needed.

7 Modelling process

The following describes the modelling process for a performance-based design. These are further discussed in the sections that follow.

Stage 1 – Specification outline (QDR)

Item no.	Description
1	Define objectives of the CFD modelling.
2	Determine type and number of simulation(s) to be prepared to demonstrate design objectives.
3	Decide how the results will need to be presented to demonstrate objectives i.e. using velocity, speed, temperature, visibility, etc.
4	Collate reference material and / or any previous test results for use later when checking the credibility of results.

Stage 2 – CFD modelling

Item no.	Description
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	Run the simulation(s)
11	Interpretation of results: Sanity check – Check the results provide a reasonable representation of real-life events.
12	Technical Review – Confirm the performance objectives been achieved. Decide what further actions should be taken if objectives not achieved.

Stage 3 – Report & presentation of results

Item no.	Description
1	Description of the objectives.
2	Description of the geometric model(s) and simulation(s).
3	Description / justification of the input parameters
4	Presentation and interpretation of the results.
5	Conclusions – Confirm the objectives have been achieved.

8 Preparing the CFD Model

8.1 *Definition of Computational Domain*

The starting point for the application of CFD to the simulation of air movement, fire and smoke movement in any building 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.

  The extent of the domain will be influenced by the nature of the building being analysed. For example, 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 an enclosure 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.

- Domain boundaries.

  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, when in reality it might re-enter the fire affected region. (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).

8.2 *Details Represented in the Computational Domain*

Modeller should establish the boundary conditions 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 openings, columns and beams.
- Services, e.g. the geometric representation of HVAC ductwork.
- Stationary vehicles (Car Park).

  Note: For car parks, 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:

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

Further guidance on the inclusion of vehicles in car parks is provided under Section 12.1.7.

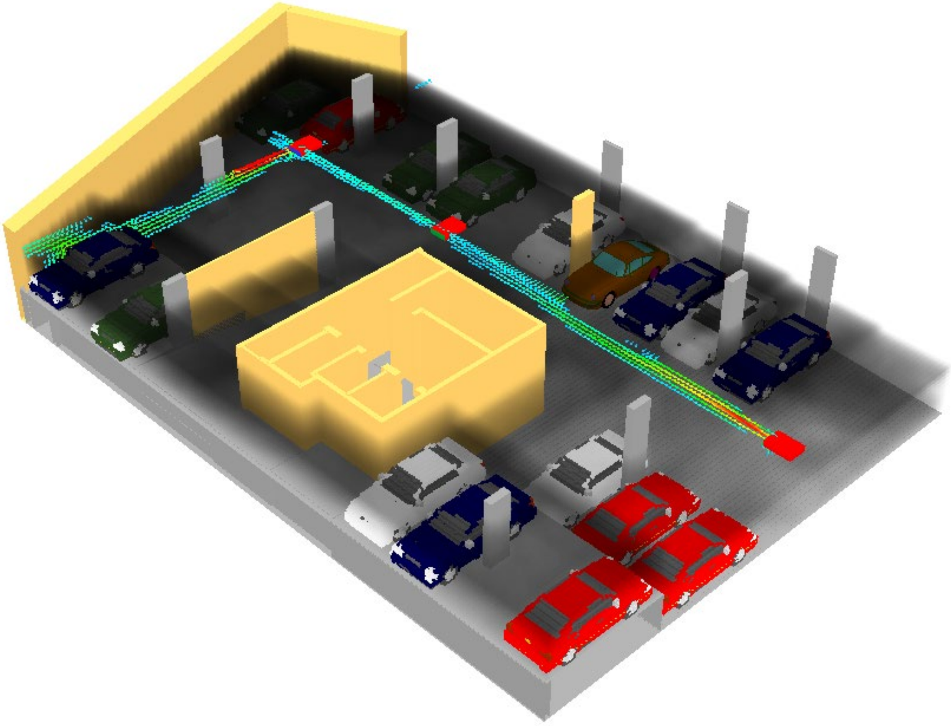

Figure 1: An example of the distribution of cars within a car park model (note that cars may often be represented satisfactorily by simple blocks)

- Environmental factors both internal and external, such as internal stack effect, wind and other objects affecting such flows, e.g. external buildings.
- Mechanically induced flows, e.g. impulse fans and inlets / extracts vents.
- Significant sources of heat (other than the fire) which might create a natural convection flow that could interact with the fire induced flow.
- The heat and smoke (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 will only approximate curved or sloping surfaces.

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 building enclosure being analysed.

8.3 *Computation Mesh*

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

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

- The geometric details (e.g. shape and size of objects) are represented appropriately.
- Where a fire is being modelled, the flow phenomena driving smoke movement are resolved adequately.
- Fire area and thermal plume-sufficient detail (fine mesh) is needed to capture the rise of the hot gases.
- Where applicable, the mesh interfacing (mesh splitting) should be avoided where large exchange of information occurs. For example, in FDS, a fire and its immediate surrounding vicinity should ideally be within a single mesh. Another example would be the internal space of an atrium with a fire at its base. This should ideally not be split horizontally due to large exchange of information at the vertical direction as buoyant hot gases and smoke rise to the top. This also applies to smoke shafts. Where it is unavoidable, the CFD user should ensure that there are no issues associated with mesh to mesh connections and demonstrate there is no impact on the results.
- 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.

Further discussion regarding mesh quality is provided in Appendix B.

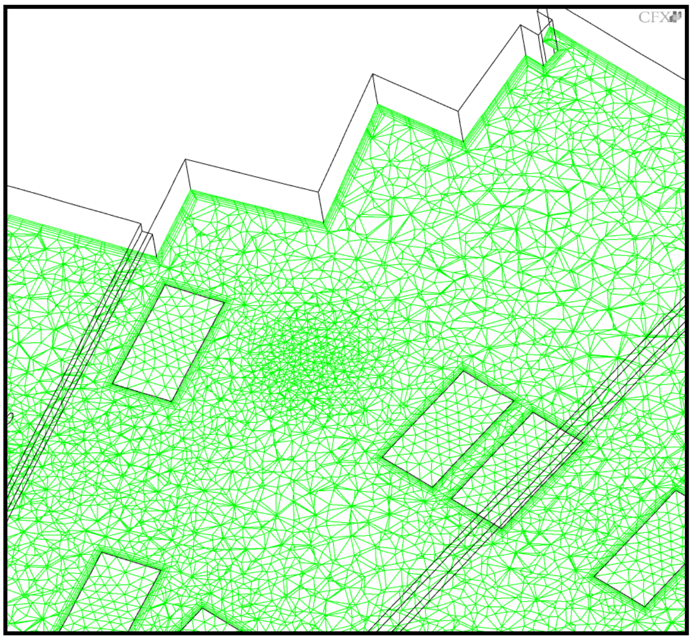

Figure 2: An example (using CFX) of a non-rectilinear mesh showing reduced mesh size at the fire source and an inflated mesh size at the boundaries

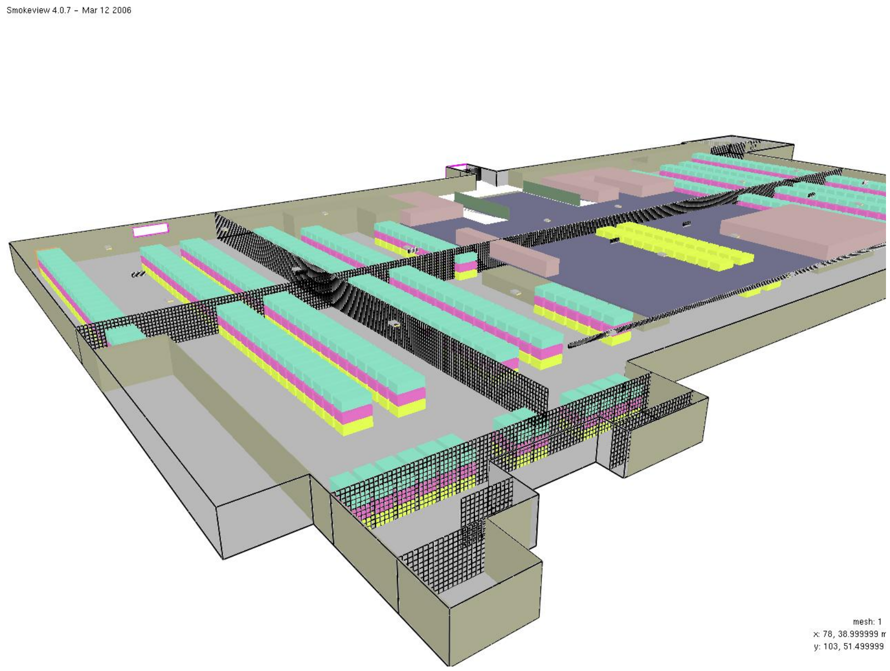

Figure 3: An example (using FDS) of a rectilinear mesh in a large car park

8.4 *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
- Buoyancy
- Turbulence
- Radiation
- Heat transfer at walls

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 Appendix A.

8.5 *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 environmental (daily) ventilation 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 area being studied.
- 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. Recommendations for suitable design fires can be found in BS7974, BS7346, SFPE Handbook and BRE publications.

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. Assessing heat release profiles (*Heat Release Rate vs Time*) in combustion models is often necessary in determining whether the fire source is ventilation or fuel controlled, when seeking specific objectives from the heat source

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 normally assume that it is constant.

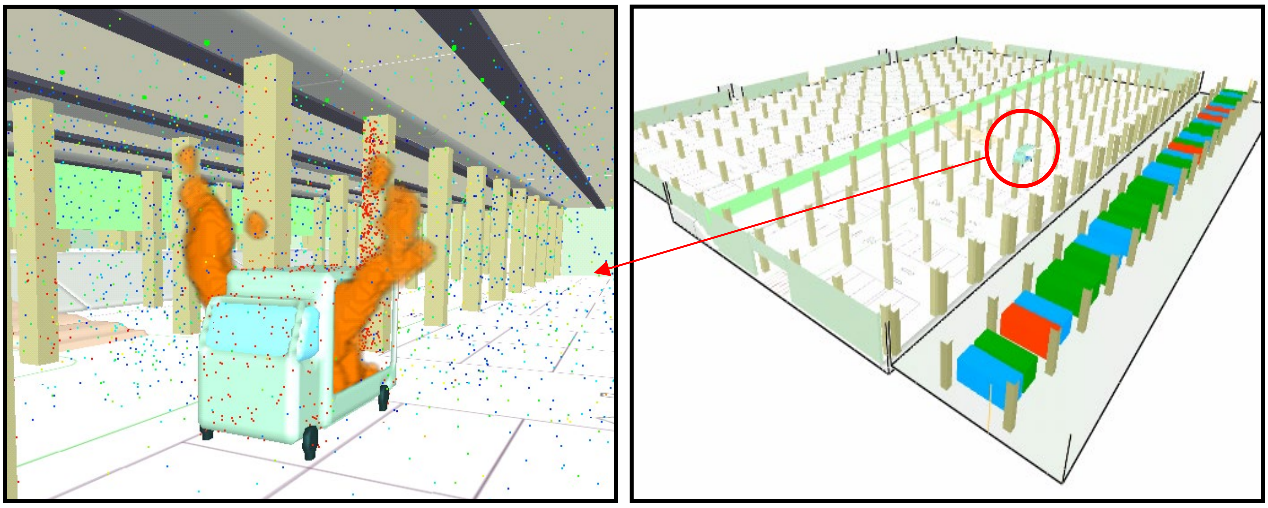

Figure 4: Detail of smoke production and sprinkler droplet distribution

Fire Spread

In some scenarios, mainly involving combustion models and developing fire, 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.

The source of fire and smoke is further discussed under Appendix C

8.6 *Initial and Boundary Conditions*

Any mechanism which is external to or within the computational domain, but which significantly 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. Appropriate assumptions should be made depending on the scenario being modelled, the objectives and acceptance criteria. The assumption made is likely to be more important in small enclosures with hot smoke than in large volume spaces where smoke temperatures may be closer to ambient.

Radiation adds further complications, which are not dealt with here. Further discussion on radiation is provided in Appendix A.

Fire-dependent Conditions

As the fire is developing, 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.

The results of CFD simulations are influenced by the boundary conditions. It is essential, therefore, that the user specifies boundary conditions appropriately 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 be encompassed if they are likely to influence the outcome of the modelling. If doubt remains, the CFD user should carry out a sensitivity analysis to evaluate the influence of a range of plausible values for boundary conditions on the predictions.

9 Presentation of Analysis and Results

9.1 *General*

The presentation of completed analysis should provide results which are clear and unambiguous. Results need to demonstrate that the design, using either comparative or absolute deterministic analysis, is acceptable. Therefore, the agreed fire scenarios should be presented with evidence the performance explicitly is shown in relation to the acceptance criteria set.

Visual and graphical results should be used in relation to the acceptance criteria with reference to the fire scenario assessed. As CFD is deterministic the use of graphical data plotted against acceptance criteria is the most useful method of demonstrating compliance.

Where a time dependent CFD model is used to achieve a steady state result, a timeline should be shown at regular intervals to demonstrate that the conditions have become stable and steady state. Additional information at specific times may be shown to highlight unusual flow behaviour, for example to demonstrate the action of fans on start-up.

Care should be taken to ensure the results reflect the fire scenario being investigated and where a sensitivity analysis is included, the comparative outcome should be clearly shown.

Key items which should be considered in the presentation of results are:

a) Images or graphical representations of key output parameters such as temperature and visibility (or smoke obscuration/distribution) should be displayed with reference to the agreed tenable bounds.
b) Results which capture the relevant fire dynamics, in particular temperature, smoke distribution and flow velocities at the relevant locations should be checked.
c) Metrics to monitor and report should be chosen accordingly to the range of data needed; velocities, temperatures etc. 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. Where the scale has change this should be clearly identified.

d) 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.

e) It is useful to display images of velocity vectors to demonstrate their effects, around fans or doorways for example.

f) Results should show mean values or where local hot spots exist;

g) Results (both graphical and deterministic) should be scaled to show relevant temperature profile for example as being fairly compared. The scales and units should be consistent across the cases analysed.

h) Use of repetitive imagery should be avoided.

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

9.2 *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 (memory stick or FTP site).

The documentation should include at least the following information:

- A simple summary description of the case being assessed.
- The design criteria and objectives of the analysis agreed in the QDR.
- The rationale for the fire scenarios investigated
- Details of the CFD model used
- The results of the analysis with discussion points
- A conclusive statement as to whether the design criteria and objectives have been met.

The reports should be fully referenced and should provide sufficient information for the reviewer to complete the analysis independently and reach the same conclusions.

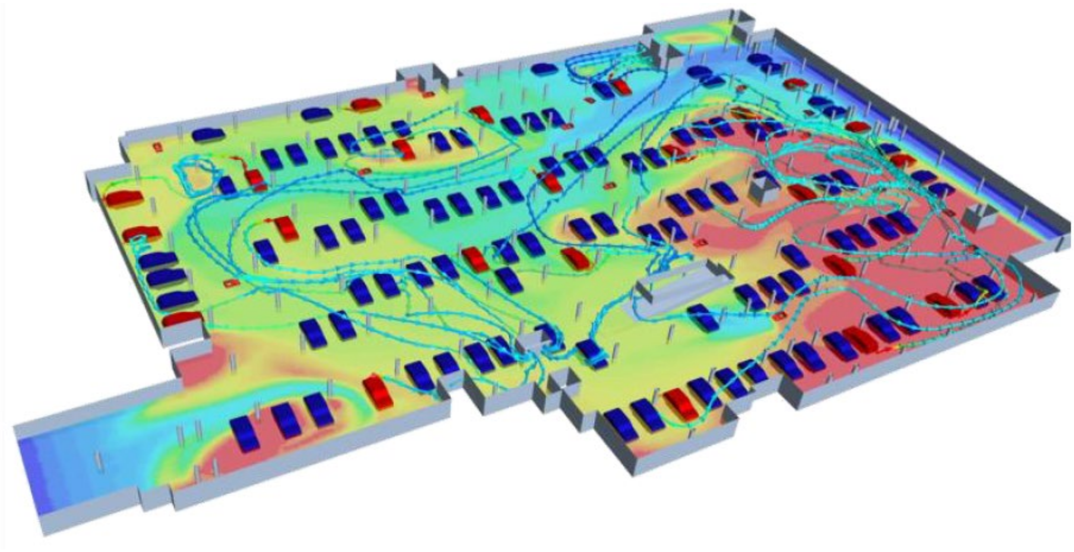

Figure 5: Image showing air residence time in a car park

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.

When necessary, a 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.

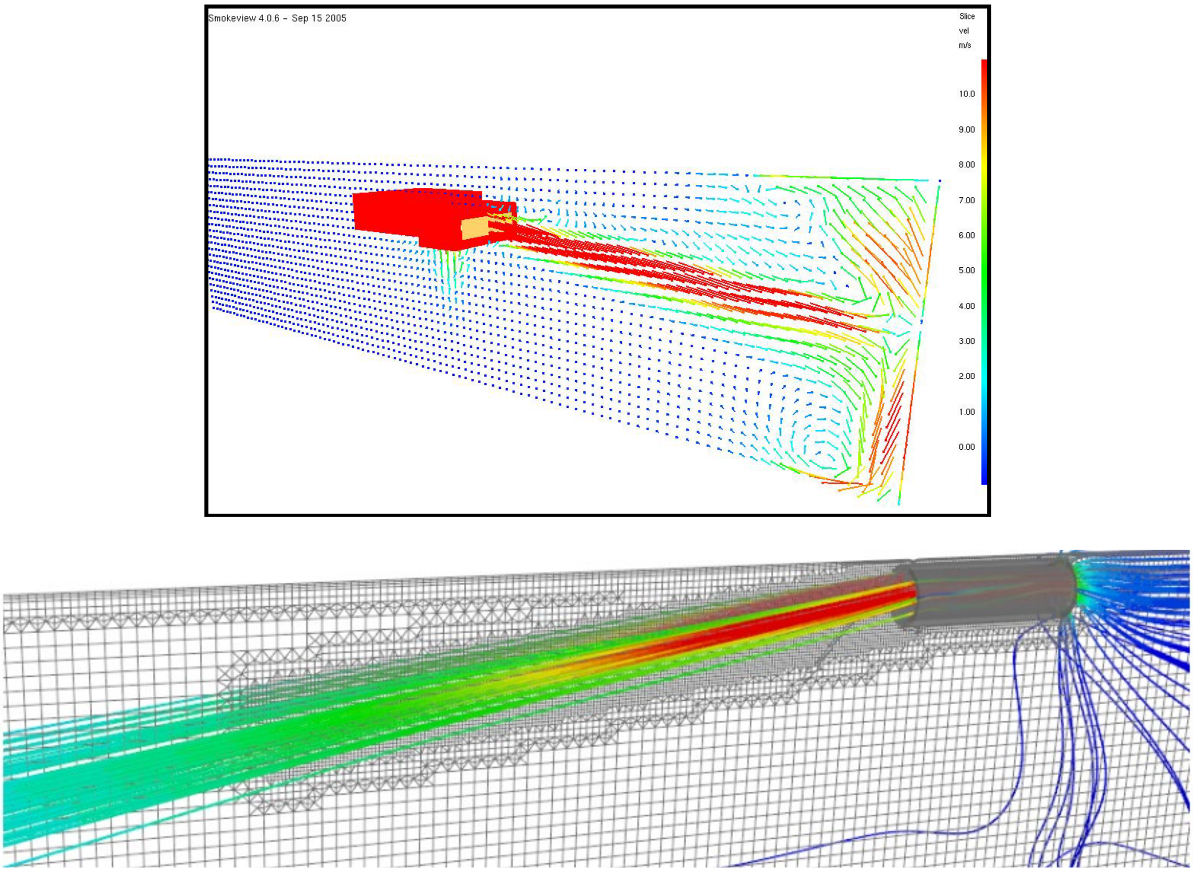

Figure 6: Section showing velocities at jet fan locations

10 Modelling Lobbies, Corridors and Stairs

The modelling of residential, commercial lobbies and corridors is undertaken to demonstrate that it satisfies the proposed design criteria outlined in the relevant guidance for the application. As there are different approaches that can be applied to demonstrate compliance when using CFD for residential corridors and commercial lobbies, the modeller must ensure that the input parameters and approach adopted forms a conservative assessment of the proposed design.

When undertaking CFD analysis, the primary objective is to protect the stairs from ingress of smoke (or maintain smoke free) or may also involve returning or providing tenable conditions to a space such as the common corridor or lobby depending on the tenability criteria outlined in the guidance that the design has adopted. Typically, acceptance criteria will be to demonstrate smoke free conditions in the stairs throughout the simulation. In cases where corridors travel distances are in excess of code requirements, tenability criteria will also be required in the corridors both escape and fire-fighting phases.

Modelling stairs, common corridors or lobbies can be referenced in the Smoke Control Association's – "*Guidance on smoke control to common escape routes in apartment buildings (Flats and Maisonettes)*". When an approach from the SCA guide is not being applied, the modeller must provide suitable rationale and justification for this deviation from the guidance.

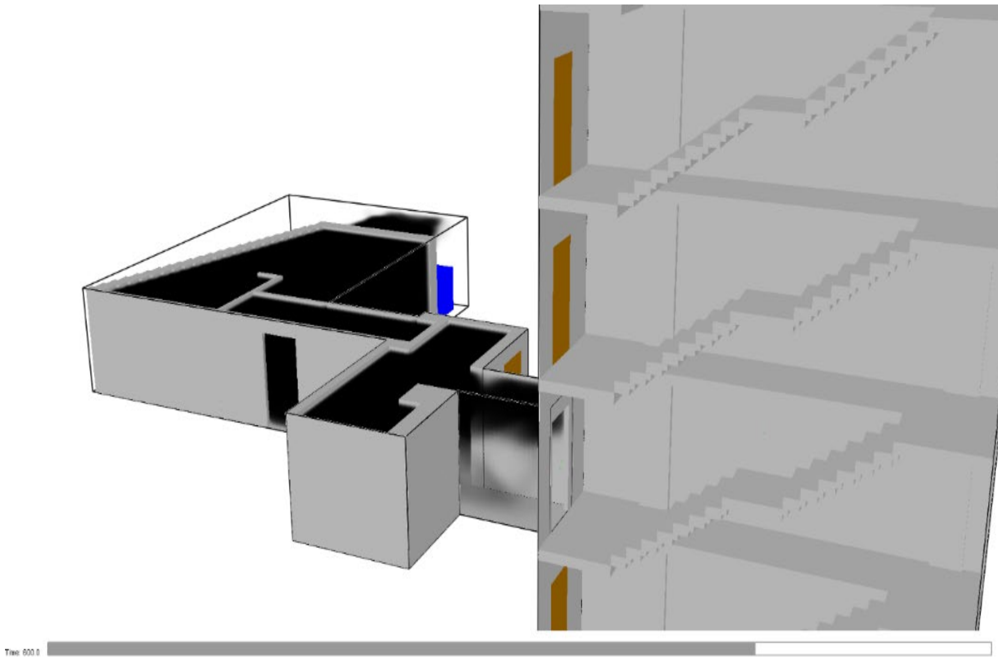

Figure 7: Section showing smoke prevented from spreading into the stair from the corridor

11 Modelling Large Enclosure Buildings

Smoke movement in large volume buildings may involve smoke from a smaller space flowing into the larger space via a spill plume at the void edge. However, in some cases it may be appropriate to model a fire on the base of the large volume with smoke flowing directly into the high-level space. Smoke control might be provided for a number of reasons including to support the means of escape strategy or to assist fire-fighting operations. The objectives of the modelling and fire scenarios to be modelled should be identified by the fire engineer prior to beginning the modelling. Examples of typical objectives for large volume buildings are discussed below:

11.1 *Storage, distribution or industrial buildings*

Such buildings often have large plan areas and mezzanine levels which are open to the levels below. The key fire strategy issue tends to be means of escape in cases where travel distances exceed the recommendations of prescriptive guidance. In such cases a CFD analysis may form part of an ASET vs RSET analysis, e.g. with the objective of showing occupants have time to escape before smoke builds down to a level at which it affects the escape routes.

Large volumes provide an inherent margin of safety and smoke control might not be necessary to support a means of escape strategy. However, smoke control might be provided for asset protection or to support firefighter access into the building.

11.2 *Atrium Buildings*

11.2.1 Open Atria

In atria which are open to some or all of the floors the main fire strategy issue is the impact of smoke spreading via the atrium to floors above the fire floor. Smoke rising though the atrium might build down below the roof of the atrium to affect occupied floors. More complex atrium geometries can result in more complex smoke movement, for example with several spill plumes. A CFD analysis may form part of an ASET vs RSET analysis, with the objective of showing occupants have time to escape before smoke builds down to a level at which it affects the escape routes. Smoke extract fans or natural smoke vents are typically required at the head of the atrium to support such a strategy.

11.2.2 Enclosed Atria

Where atria are enclosed a fire is less likely to represent an immediate risk to the levels above. However, in buildings with long evacuation times or residential buildings with stay put strategies, smoke control may be provided to control the temperature of smoke within the atrium with the objective of allowing the use of non-fire rated glazing or glazing which is fire rated for integrity only. Examples of buildings where this might be appropriate include offices with phased evacuation strategies, hospitals or residential buildings.

11.3 *Shopping Centres*

Shopping centres are generally designed such that smoke is allowed to flow out of the unit and into the mall where it is dealt with by a mall smoke control system. Mall smoke control systems are normally designed to maintain a clear layer at least 3m above the highest mall level for a steady state fire. Guidance recommends the size of mall smoke reservoirs is limited because of the risk the smoke layer will lose buoyancy and that smoke flowing under balconies is channelled by screens to limit the width of the spill plume and therefore the volume entering the reservoir. However, the prescriptive guidance was developed in the late 1980s and hasn’t kept pace with trends in shopping centre designs which include larger volume mall spaces, malls of more than two storeys, a desire to minimise the number of smoke screens and semi external malls.

CFD studies in shopping centres might be carried out to demonstrate that variations from the prescriptive guidance will not compromise the safety of occupants or firefighters.

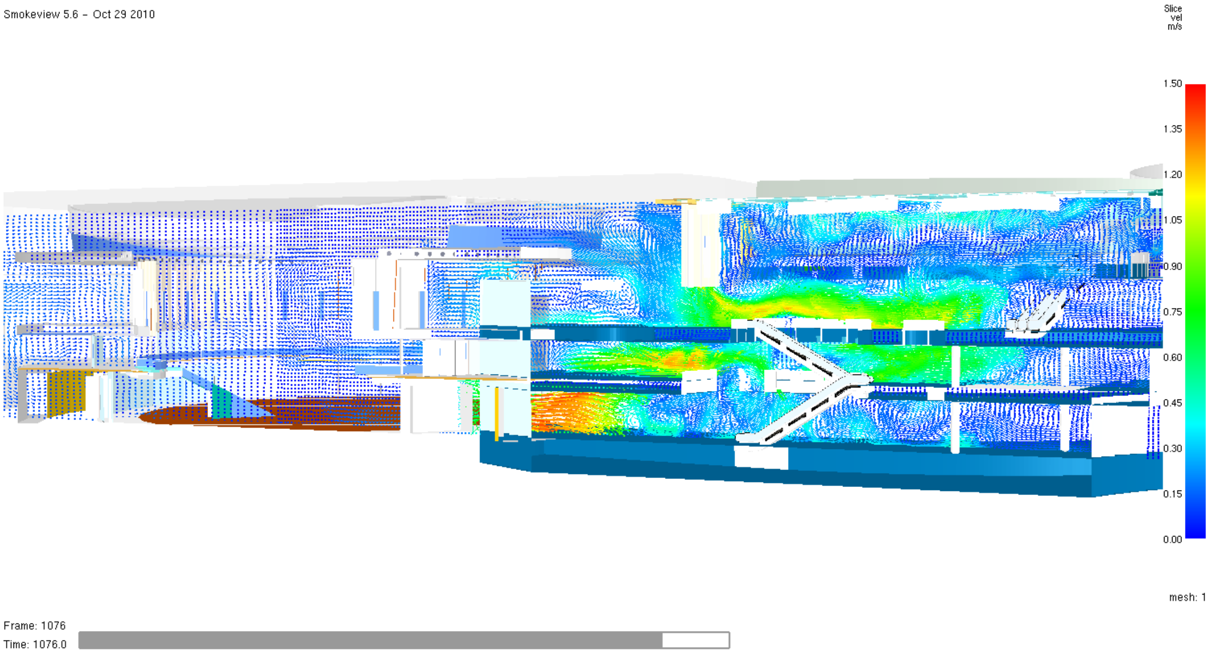

Figure 8: Section showing velocities through shopping mall

11.4 *Mesh Size*

When modelling large volumes multiple meshes are generally required in order to achieve practical processing times. For example, a relatively fine mesh should be used in the near field. For example, this might include the fire location, the fire plume and where smoke spills out of the fire room into the large volume space. Outside that region a coarser mesh is generally adequate. However, it is essential that the mesh has adequate resolution to model important features which might be remote from the fire such as vents, mechanical extract points or replacement air inlets.

11.5 *Fire*

11.5.1 Scenarios

Locations to be modelled should be identified by the fire engineer taking into consideration the objectives of the modelling. If the objective of the modelling is to demonstrate smoke does not build down to a level at which it affects escape routes, then the most complex smoke path is likely to represent the most onerous scenario.

However, if smoke temperature is the key criterion it may be appropriate to model a fire with a less complex smoke path and less entrainment into the smoke plume. This might apply, for example, to an atrium temperature control system where a fire on the atrium base or on a higher floor might result in the highest smoke temperatures.

11.5.2 Fire Size

The proposed fire size and fire growth rate should be identified by the fire engineer. Sources of guidance on design fires include BS 7974, BS 7346 and the SFPE Handbook.

If the CFD study is being carried out as part of an ASET vis RSET analysis a time dependent development fire will generally be appropriate. Where a developing fire is modelled it may be appropriate to cap the fire at a certain size to take account of any suppression system provided.

However, there may be circumstances where it is adequate to assume a steady state fire in order to simplify the modelling. For example, if carrying out an ASET vs RSET analysis in a large storage or distribution building the margin of safety might be such that modelling a steady state fire is adequate.

There are also applications where design has traditionally assumed a steady state fire. These include smoke control systems in shopping centres and atria.

11.6 *Activation of Smoke Control System*

The timeline for the activation of the smoke control system should be identified by the fire engineer prior to the modelling. Where systems operate automatically and are linked to the fire alarm system this could be informed by a separate modelling exercise such as modelling of smoke detector activation. This may also need to take into account ramp up times for smoke extract fans as well as opening of dampers and automatic opening ventilators (AOV).

However, the above factors are less critical in large volume spaces than in small volumes and it may be appropriate to make some suitably conservative assumptions. When considering steady state designs it would normally be adequate to assume the smoke control system is active immediately.

11.7 *Simulation Time*

The simulation time needs to be chosen to give results that are appropriate to the scenario being modelled. For a time dependent fire the simulation time should be based on the time a particular action occurs. For example, this might relate to evacuation time or fire brigade intervention.

For a steady state scenario, the simulation time should be sufficient to demonstrate that the model is in steady state and thermal equilibrium. This occurs when conditions such as the smoke layer height and temperature are no longer changing. In a large volume space this may require a simulation time of the order of 20 minutes.

11.8 *Acceptance Criteria*

Acceptance criteria should be defined by the Fire Strategy and ideally agreed with the approving authorities prior to carrying out the modelling. For example, acceptance criteria might be a smoke clear layer below which occupants can escape unimpeded or conditions which allow escape or fire-fighting through cool dilute smoke which remains tenable. Guidance on suitable tenability criteria is given in BS 7974.

The tenability criteria which are expected to be important should be identified prior to the modelling and the model should be developed

12 Modelling Car Parks

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.

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.

The scope on this guide is limited to car parks occupied by internal combustions vehicles and is unable to offer guidance for vehicles powered by alternative fuel sources (e.g. electrically battery powered, hydrogen fuel cell). Further consideration is required in such instances and dealt with in the QDR process.

The SCA document "*Design of Smoke Ventilation Systems for Loading Bays & Coach Parks*" offers further guidance for design of ventilation system for loading bays, service yards and coach parks and lists the options available to the design engineer.

12.1 *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. The acceptance criteria is normally defined by the Fire Strategy and agreed with the approving authorities. 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.

For car parks, there are seven specific main issues that require 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, BS7346-7, CIBSE Guides, BS EN 12101-11, Approved Documents B and F.

12.1.1 *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.

12.1.2. *Smoke clearance*

Objective: The objective should be to show that the whole car park is uniformly cleared of smoke by ventilation, without stagnate areas, and that the ventilation rate meets the recommendations of prescribed guidance (ADB, BS9991, BS9999).

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. An area with stagnant air is considered to be such areas where the age of air or rate of purging is not appropriate, for example, where the air change rate is less than 50% of the overall design value; velocity of air is less than 0.1m/s; or where the mean age of air is greater than twice the average.

12.1.3 *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:

a) Clear layer assessment
b) 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.

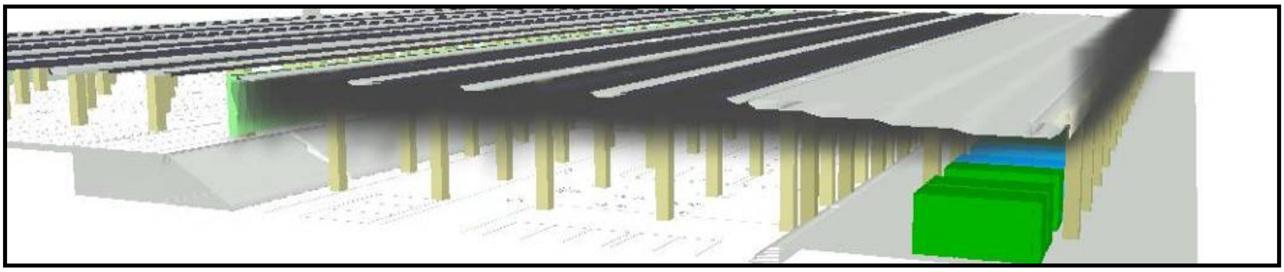

Figure 9: Maintenance of a clear layer below smoke

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.

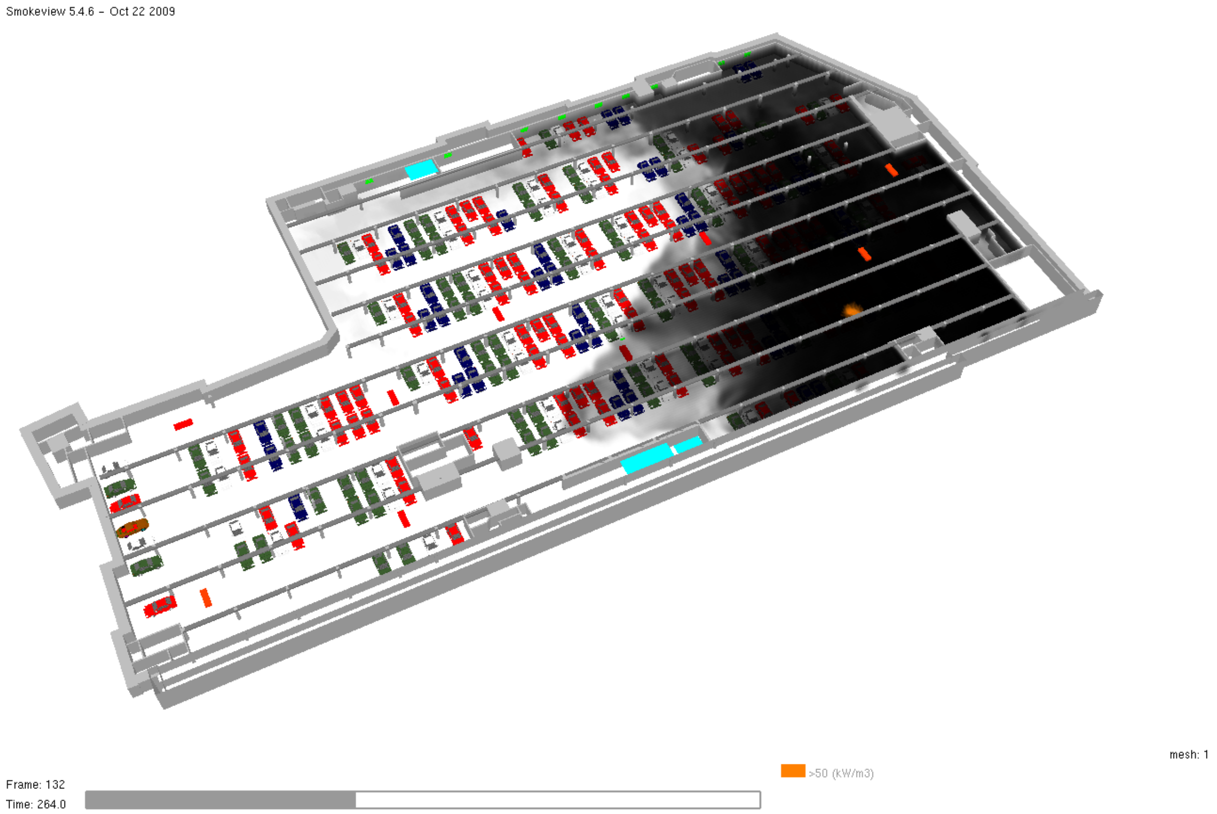

Figure 10: Limiting spread of smoke using jet fans to maintain tenable conditions

12.1.4. *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:

a) Find the location of the fire.
b) Gain safe access to a position from which to deliver water to the base of the fire.
c) 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

12.1.5 *Fire Spread and local effects*

Objective: The objective is to show the effect of the fire on any nearby 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 nearby 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 nearby objects will depend on the type of objects that are located nearby. 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.

12.1.6. *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.

12.1.7. *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 along with all the elements that could impact their flow field

	(silencers, deflection louvers tec.). The flow patterns should show that there are no dead zones or stagnant areas. 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.
Modelling cars (obstacles):	Modelling cars or other mobile obstacles is generally not required. However, when unrealistic large-scale vortices are observed, whereby large areas of the car park have air flow speeds in excess of 2.5m/s, the same simulation should be repeated with evenly distributed car obstacles occupying 50% of the available car spaces. The obstacles ought to disturb the vortex so to bring it in line with a more realistic scenario.
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.

13 Modelling guidance for designing smoke control system

The following tables are considered ‘best practice’ guidance in designing smoke control systems through use of CFD analysis. The recommendations are intended to build conservatism into the models using simplified assumptions to mitigate against known or unknown variable characteristics in buildings which may have a significant influence on the outcome of a study.

13.1 General modelling of smoke control systems:

Parameter	Description	Recommendation
Model domain(s)	Key elements, I-beams, pillars, ceiling bulk heads to be included in the model for accuracy	- Building geometry should reflect the intended or as built drawings. - Ensure software is fit for purpose for the study, particularly in high rise buildings. - Creating extended boundary domain to external vents where external conditions could influence the area of interest. - Model domain should ensure all the different zones are connected to each other.
Mesh sizing	Suitable mesh sizing selection based on fire and domain size.	- Should be sufficient to capture the physics (velocity, pressure, temperature) in the areas of interest.
Initial boundary conditions	Ambient conditions (temperature, density, etc.) Relevant external air/wind Seasonal climate consideration	- Material properties should be sufficient to capture the relevant physics of the actual areas being studied. Default values may not produce realistic results.
Fire source and location	Suitable fire source in terms of size, heat flux and heat release rate. This also includes the appropriate fire location(s).	- Heat and Energy conservation balances during development or steady state should be checked to ensure numerical validity.
Oxygen Source for fire	Ventilation opening to sustain the design fire size (artificial or actual)	- The fire size should be checked against the HHR profiles to ensure steady state fire validity. - Ensure fire development or in steady state is balanced with a suitable boundary opening to support it. Incorporating window breakage may affect the fire size, airflow dynamics and will also influence the proportion of smoke and heat lost to outside rather into the area being studied. - Further discussion on artificial ‘low-level’ vents is provided in Appendix C

Heat Release Rate	Balanced heat source (fuel / ventilation controlled)	- Observe HRR profiles to validate the design fire performance
Smoke Density (soot yield)	Suitable soot yield	- Soot yield production should be equivalent to the combination of a 10% yield with heat of combustion between 19 000 to 20 000 KJ/kg
Natural Air inlet	Natural source of replacement air	- Domains should be extended beyond the area of immediate interest where external flows may influence inlet air through an opening - Avoid confusion between Geometric and Aerodynamic area. Geometric area is represented as a hole in the wall whereas aerodynamic is related to the vents performance efficiency which is only obtainable through approved testing standards (BS EN 12101-2). - Incoming velocity from natural vents needs to be taken into account as velocities in excess of 5m/s could impede escaping occupants who need to travel past the vent(s).
Simulating occupant movement	Accounting for door opening times during escape and; or fire-fighting access	- For Fire-Fighting access, fully open doors between the fire room and the FF access route (e.g. FF stair) should be modelled. - Airflow velocity along the escape route should be observed. According to PD 7974-2, air velocity in excess of 5m/s could impede escape.
Smoke Extract / Exhaust Terminals	Modelling airflow terminals as vents	- Vents should be modelled with their actual characteristic width or as an equivalent free area using the actual width dimension. - The velocity of inlet air from a supply vent needs to be taken into account as velocities in excess of 10 m/s could impede escaping occupants who need to travel past the vent(s).
Plant start up time	Fan ramp up, damper & vent opening / closing times	- Spontaneously starting fans and opening vents is not realistic, therefore fan and damper(s) linked to mechanical extract system should include ramp up opening times of at least 10 seconds following activation. - For natural vents linked to a natural smoke exhaust system, should include a minimum opening time of 30 seconds following activation/detection.

13.2 Lobbies, Corridors and Stairs:

The guidance given below is provided in addition to Section 13.1 and is relevant to Lobbies, corridors and stairs.

Parameter	Description	Recommendation
Model domain(s)	Key elements, I-beams, pillars, ceiling bulk heads to be included in the model for accuracy	- The entire stair enclosure height linking the relevant corridors or/and lobbies should be included in the model. Where relevant, final exit routes should also be included.
Mesh sizing	Suitable mesh sizing selection based on fire and domain size.	- For smaller enclosures such as corridor and lobbies, a typical mesh size of 0.1m near field and 0.2m far field to the fire source are considered suitable.
Initial boundary conditions	Ambient conditions (temperature, density, etc.) Relevant external air/wind Seasonal climate consideration	- Extreme temperatures differentials between different interacting domains should be modelled. These should be particularly assessed in tall high-rise buildings where internal stack effect is likely to occur. - Material properties should be sufficient to capture the relevant physics of the actual areas being studied. Default values may not produce realistic results.
Fire source and location	Suitable fire source in terms of size, heat flux and heat release rate. This also includes the appropriate fire location(s).	- Fire parameters should be substantiated from available published design guides, experimental data and / or empirically calculated. SCA Residential Guide offers guidance appropriate sizing of residential building fires. Further information is provided in Appendix C. - For sprinklered fires, it acceptable to assume that the fire will develop until the suppression system activates and then remains fixed thereafter. Depending on the context of what the modelling objectives are, the cooling effect of water droplets may not need to be accounted for in certain applications. - Various locations should be considered. For corridors, a simple approach would be to test fires closest and/or furthest from the stair, particularly in extended single direction escape corridors. A minimum two fire locations should be selected where a corridor has a dead-end section (>5m) beyond and extract/inlet terminal. In such case, a fire location furthest from the dead-end section should be selected.
Oxygen Source for fire	Ventilation opening to sustain the design fire size (artificial or actual)	- Using artificial opening requires careful consideration. The opening should correlate to a steady state design fire. - The artificial vent location should be conservatively set as low as possible without having a significant impact of the airflow dynamics (unrealistic short circuit of replacement / inlet air). High level vents are not desirable for assessment. See Appendix C for more information.

Leakage	Influential leakage from doors, windows and other	- Leakage from doors and windows should be included especially where mechanical powered systems (smoke extract, pressurisation or depressurisation) are adopted. - Leakage from closed doors or windows can be deemed negligible where pressure differentials are lower than 5Pa
Natural Air inlet	Natural source of replacement air	- Natural inlet paths, such as shafts or ducts, should be included for assessment. - Natural inlet air profiles should be adjusted to account for upstream resistance e.g. pressure drop through a natural inlet shaft may affect depressurisation levels in the corridor to unsafe levels - Opening size(s) representing pressure relief vents should be closely correlated to pressure differential near the vent.
Simulating occupant movement	Accounting for door opening times during escape and; or fire-fighting access	- During initial escape, doors opening along the escape route should be modelled open for no less than 20 seconds to simulate occupants escaping through the door.
Smoke Extract / Exhaust Terminals	Modelling airflow terminals as vents	- Consideration on the viability of the smoke control system. Extract rates in corridors or lobbies that are in excess of 6m3/s, may be more hazards during escape and fire-fighting (e.g. slamming doors, drawing excessive amount of fire and smoke into protected spaces etc.) and may warrant further assessment. Furthermore, installation of systems with excessive flow rates may not be feasible to achieve. - Assumptions used for inlet air in the model need to be feasible in practice e.g. final exit door at the base of the stairs being open by automation rather than assuming FF intervention or following escape.
Simulation run-time	Model run time	- CFD runtime should run for at least 10 minutes unless steady state conditions are observed sooner. Assessment criteria should be assessed only after balanced conditions are achieved.

13.3 Large buildings and enclosures:

The guidance given below is provided in addition to Section 13.1 and is relevant to large buildings and enclosures.

Parameter	Description	Recommendation
Model domain(s)	Key elements, I-beams, pillars, ceiling bulk heads to be included in the model for accuracy	- In large open domains, all objects that may influence the flow and the performance of the system should be explicitly modelled, e.g. down-stands and beams where these divert the bulk smoke and air flow
Mesh sizing	Suitable mesh sizing selection based on fire and domain size.	- For large enclosures, a higher mesh density should be used within 2m of the fire source and a lower density throughout the remaining volume of the building
Initial boundary conditions	Ambient conditions (temperature, density, etc.) Relevant external air/wind Seasonal Climate	- Extreme temperatures differentials between different interacting domains should be modelled. These should be particularly assessed in tall open voids where internal stack effect is likely to occur.
Fire source and location	Suitable fire source in terms of size, heat flux and heat release rate. This also includes the appropriate fire location(s).	- Fire parameters should be substantiated from published design guides, available experimental data and / or empirically calculated. See Appendix C for further guidance. - For sprinklered fires, it acceptable to assume that the fire will develop until the suppression system activates and then remains fixed thereafter. Depending on the context of what the modelling objectives are, the cooling effect of water droplets may not need to be accounted for certain application. - The selection of the worst credible fire locations can be identified through the use of quantitative and/or qualitative methods. In atriums for example, a simple approach would be to test fires at the base of the atrium where smoke would spill into the atrium void from a balcony edge; and fires on the highest level that would fill the smoke reservoir to the designed clear layer height in the shortest period. - It is recommended that the locations be discussed and agreed with the relevant authorities at an early stage in the project.
Oxygen Source for fire	Ventilation opening to sustain the design fire size (artificial or actual)	- Using artificial opening requires careful consideration. The opening should correlate to a steady state design fire. - The artificial vent location should be conservatively set as low as possible without having a significant impact of the airflow dynamics (unrealistic short circuit of replacement / inlet air). High level vents are not desirable for assessment.

Smoke Density (soot yield)	Suitable soot yield	- Soot yield production should be equivalent to the combination of a 10% yield with heat of combustion of 19 000 to 20 000 KJ/kg
Leakage	Influential leakage from doors, windows and other	- Leakage from doors and windows should be included especially where mechanical powered systems are adopted. - Leakage from closed doors or windows can be deemed as negligible where pressure differentials are lower than 5Pa
Natural Air inlet	Natural source of replacement air	- Natural inlet paths, such as shafts or ducts, should be included for assessment. - - Natural inlet air profiles should be adjusted to account for upstream resistance e.g. pressure drop through a natural inlet shaft may affect depressurisation levels in the corridor to unsafe levels
Simulating occupant movement	Accounting for door opening times during escape and; or fire-fighting access	- During initial escape, doors opening along the escape route should be modelled open for less than 20 seconds to simulate occupants escaping through the door. However, longer times may need to be considered to simulate queue times for simultaneous evacuation of large buildings
Smoke Extract / Exhaust Terminals	Modelling airflow terminals as vents	- Consideration on the viability of the smoke control system. - Assumptions used for inlet air in the model need to be feasible in practice e.g. Entrance lobby doors used for inlet air should be open by automation rather than left open by escaping occupants or FF access.
Simulation run-time	Model run time	- CFD runtime should run for at least 20 minutes unless steady state conditions are observed sooner. Assessment criteria should be assessed only after balanced conditions are achieved.

13.4 Enclosed car parks:

The guidance given below is provided in addition to Section 13.1 and is relevant to enclosed car parks.

Parameter	Description	Recommendation
Model domain(s)	Key elements, I-beams, pillars, ceiling bulk heads to be included in the model for accuracy	- All objects that may influence the flow and the performance of the system should be explicitly modelled, e.g. down-stands and beams where these divert the bulk air flow.
Modelling of cars (obstacles)	Inclusion of cars as obstructions	- Cars (obstacles) are not required unless significant vortices and flow obstructions are generated. See Section 12.1.7 for further guidance.
Mesh sizing	Suitable mesh sizing selection based on fire and domain size.	- For large enclosures, a higher mesh density should be used within 2m of the fire source and a lower density throughout the remaining volume of the building
Initial boundary conditions	Ambient conditions (temperature, density, etc.) Relevant external air/wind	- Material properties should be sufficient to capture the relevant physics of the actual areas being studied. Default values may not produce realistic results. However, inclusion of material properties is not necessary for smoke clearance or vehicle emissions (pollution) studies.
Fire source and location	Suitable fire source in terms of size, heat flux and heat release rate. This also includes the appropriate fire location(s).	- Fire parameters should be substantiated from published design guides such as BS7346-7, available experimental data and / or empirically calculated. - 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.
Oxygen Source for fire	Ventilation opening to sustain the design fire size (artificial or actual)	- The model domain should match the planned geometry of the car park. This includes all external openings. Artificial openings should not be adopted unless substantiated.
Smoke Density (soot yield)	Suitable soot yield	- Soot yield production should be equivalent to the combination of a 10% yield with heat of combustion between 19 000 to 20 000 KJ/kg
Leakage	Influential leakage from doors, windows and other	- Leakage from closed doors or windows can be deemed as negligible for car park ventilation CFD studies.
Natural Air inlet	Natural source of replacement air	- Natural inlet paths, such as shafts or ducts, should be included for assessment. - Incoming velocity from natural vents and entrance ramps need to be taken into account as velocities in excess of 5m/s could impede escaping occupants who need to travel past the vent(s) or opening(s).

Simulating occupant movement	Accounting for door opening times during escape and; or fire-fighting access	- During initial escape, doors opening along the escape route should be modelled open for less than 20 seconds to simulate occupants escaping through the door. However, longer times may need to be considered to simulate queue times for simultaneous evacuation of large buildings
Smoke Extract / Exhaust Terminals	Modelling airflow terminals as vents	- Extract and inlets should be meshed with a minimum of 5 cells along each edge on the surface of the element. - Consideration on the viability of the smoke control system. - Assumptions used for inlet air in the model need to be feasible in practice e.g. Entrance shutters at the car park entrance should be open or close by automation rather assuming manual operation during FF access.
Induction / Jet Fans	Modelling jet fans and their respective airflow profiles.	- Jet fans should be modelled with all the elements which may impact their flow field e.g. silencers
Simulation run-time	Model run time	- CFD runtime should run for at least 20 minutes unless steady state conditions are observed sooner. Assessment criteria should be assessed only after balanced conditions are achieved.

Appendix A: Physical sub models

Turbulence Models

One of the most important physical models to consider when simulating relates to the treatment of turbulence as this will define to a large extent how heat and soot are transported around the computational domain.

Turbulence is generated across a wide range of length scales, but in ‘fire driven flows’ turbulence is generated across length scales typically of the order of a few meters and representative of physical lengths associated with the generation of shear layers in the flow. As the turbulent energy decays, the turbulent eddy sizes associated with this energy become smaller until they are small enough for the energy to be dissipated by viscous forces. It is at these finest length scales that fuel/air mixing takes place and at which chemical reaction occurs. Thus, fire and smoke modelling involve a large range of spatial and temporal scales, of which only a subset of these can be simulated and the rest must be modelled. it is important to understand that different choices are available for the treatment of turbulence in these problems.

A common way of modelling turbulence is termed large eddy simulation (LES), where the time-dependent flow equations are solved and the larger eddy influences are rigorously represented. This technique is aimed at extracting greater temporal and spatial fidelity from the simulations of a fire performed on the more finely meshed grids. The general philosophy behind LES simulations is that the eddies that account for most of the mixing are large enough to be calculated with reasonable accuracy from the equations of fluid dynamics. However, below the mesh size, certainly at length scales associated with the chemical reaction, fluctuations are either ignored or perhaps time averaged.

Reynolds-averaged Navier-Stokes (RANS) models were developed as a time-averaged approximation to the conservation equations of fluid dynamics. In other words, the RANS models solve only for time-averaged eddies. The smallest resolvable length scales are determined by the product of the local velocity and the averaging time rather than the spatial resolution of the computational grid. Unfortunately, the evolution and behaviour of large eddy structures characteristic of most fire and smoke plumes is lost with such approach. Over the years, the increase in CFD capabilities has shown a general trend of moving from RANS models to LES techniques.

Another computational approach is the Direct Numerical Simulation (DNS).
DNS is a simulation in computational fluid dynamics in which the Navier-Stokes equations are numerically solved without any turbulence model. This means that the whole range of spatial and temporal scales of the turbulence must be resolved in the computational mesh. In other words, all the turbulent motions within a certain flow down to the smallest turbulent scales are resolved. The computational cost of DNS is

very high also for non-turbulent flows, and as a consequence the computational resources required by a DNS would exceed the capacity of the most powerful computers currently available. Therefore, DNS techniques are not practical to fire related flows.

The difference among DNS, LES and RANS models is graphically shown below, representing the different approaches that the CFD methodology can employ for fire simulation purposes for a given variable.

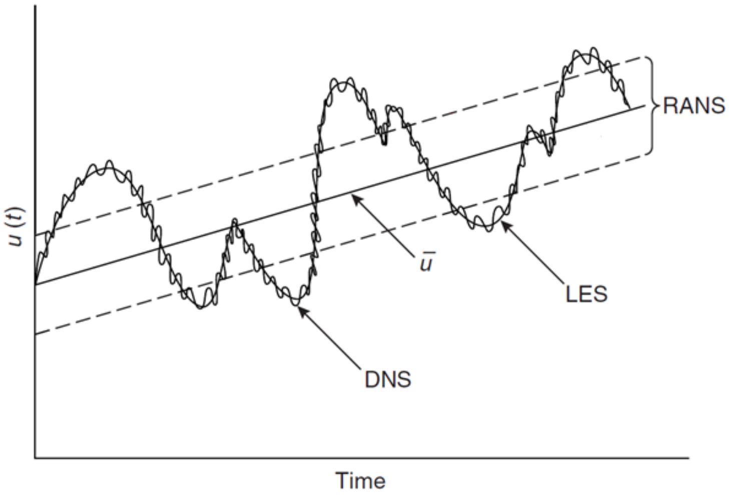

Figure 11 –Schematic representation of different turbulence resolutions among DNS, LES and RANS models

Combustion Models

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

<u>Volumetric Heat Source Model:</u>

> 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.

<u>Combustion Model:</u>

> 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.

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.

a) *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.

b) *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.

c) *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

d) *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 requires the calculation of local emissive powers and absorptivity’s, 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.

Appendix B: Quality and Compliance

The equations of fluid dynamics are discretised through the mesh division and transformed to a set of algebraic equations which are then solved through an iterative process to arrive to a numerical solution. The quality of the mesh (i.e. its density and distribution of the cells) plays an important role in the reliability and accuracy of the numerical results.

Regardless of the type of mesh used for a specific simulation, checking the size, refinement and numerical convergence of the solution obtained by using a certain mesh is essential. In many cases, poor resolution in critical regions can dramatically affect results of a CFD simulation.

This section presents an overview of the factors that can help in evaluating the quality and correctness of the numerical solution obtained.

B.1 *Mesh Quality*

The indicators which identify mesh quality are *Orthogonality, Aspect ratio* and *Skewness:*

- *Orthogonality:*
  Is a quantity computed for cells and it is in general calculated by the software package. The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher.
- *Aspect ratio:*
  The aspect ratio is a measure of the stretching of a cell. Generally, it is best to avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibit large changes or strong gradients. Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations. Rapid changes in cell volume between adjacent cells translate into larger truncation errors.
- *Skewness:*
  Defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. Highly skewed cells can decrease accuracy and destabilize the solution. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below 0.95, with an

average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties.

B.2 *Mesh Sensitivity Analysis Through Discretisation Error study*

The most recent adopted and accepted method for assessing CFD results sensitivity to the mesh size is based on calculation of the Grid Convergence Index (GCI). This method provide that the Index is calculated using Richardson Extrapolation (RE) method.

The main advantages of using this method is that it is a reliable method to all branches of flow simulations, such as fluid motion, thermal analysis, fire and smoke driven flow and furthermore, it can be applied to the results of different CFD software outputs.

Discretization errors for the CFD results can be analysed now by using the GCI (Grid Convergence Index) method.

The process of the GCI assessment is as follows:

- A representative mesh size (cell *c*) is defined, which is provided by the equation below:

$$c = [\frac{1}{N}\sum_{i=1}^{N}(\Delta Vi)]^{1/3}$$

Where:

- *c* is computational mesh size (cell size);
- *N* is the total number of cell in the mesh;
- ΔV_i is the volume of the i^{th} cell;

Three different sets of mesh (i.e. c_1, c_2 and c_3) in decreasing order of fineness (c_1 fine mesh size and c_3 coarser) are recommended for the sensitivity analysis. For these different mesh sizes, simulations are run and any variables of the flow field f_1, f_2, f_3 (i.e. temperature, or pressure, or velocity, etc.) can be used for the discretization error analysis.

For example, f_1 is the selected variable obtained for the finest mesh, while f_2 is obtained for the medium mesh and the f_3 is obtained for the coarse mesh. For these flow variables the condition $f_2 / f_1 > 1/3$ and $f_3 / f_2 > 1/3$ must always be satisfied.

The Grid Convergence Index (GCI) is calculated using the following equation:

$$GCI_{fine}^{21} = \frac{1.25\, e_a^{21}}{r_{21}^{p} - 1}$$

Where:

- $r_{21} = c_1 / c_2$ is the refinement factor (from one mesh to the other)
- $e_a^{21} = |\frac{f_1 - f_2}{f_1}|$ is a
- p is the *order of the method of the discretization scheme.* It can be found in the CFD software package details (usually range from 1-4).

<u>The calculation of GCI in relation to a certain mesh size should result to be less than 10% for the mesh to be considered reliable and the results sufficiently accurate.</u>

B.3 *Convergence check*

Discretisation of fluid dynamic equations yields to a large number of algebraic equations (one set for each cell of the computational mesh). These equations are then generally solved using an iterative method, starting with a first guess value for all variables and completing a computational cycle.

When assessing the numerical correctness of a simulation it is fundamental to check convergence of the solution. This evaluation can be done by checking the residual values (i.e. the changes in the equation over an iteration) and/or by checking that additional iterations produce negligible changes in the variable values. Information on the criteria are given as follows:

B.3.1 Convergence check through residuals

The residuals of the equations are the change in the equations over an iteration. The residual is one of the most fundamental measures of an iterative solution's convergence, as it directly quantifies the error in the solution of the system of equations.

In a CFD analysis, the residual measures the "local imbalance" of a conserved variable in each cell of the mesh. Therefore, every cell in a CFD model will have its own residual value for each of the equations being solved.

In an iterative numerical solution, the residual will never be exactly zero. However, the lower the residual value is, the more numerically accurate the solution. Each CFD code will have its own procedure for normalizing the solution residuals.

- Residuals of 10^{-4} are considered to be loosely converged,
- Residuals of 10^{-5} are considered to be well converged,
- Residuals of 10^{-6} are considered to be tightly converged.

It must be noticed that for complicated problems, however, it’s not always possible to achieve well or tightly convergence.

B.3.2 Convergence identification through flow results

Every CFD simulation has the objective of determining some quantity such as temperature, velocity, pressure, etc., of a certain flow field. In order to verify convergence, it is possible to track the values of such variables with respect to iteration and define iterative convergence when these quantities converge (i.e. remains unchanged in the following iterations). Convergence can be defined when a monitored flow value remains unchanged with respect to the number of iterations. The convergence criteria is often defined by the acceptable error in these values. Depending on the flow field simulated it can occur that certain quantities may reach convergence at a different rate than other quantities.

B.4 *Frequency for Mesh Sensitivity Testing*

Mesh density is one of the significant metrics to control accuracy of a CFD model, assuming that all the input parameters used are accurate. One of the ways to evaluate quality of the mesh is by comparing its consequent results to actual test data, empirical calculations or theoretical values. Otherwise refining mesh sizes and interpreting result deviations would provide alternative methods. The problem with multiple remeshing and rerunning the models, is that they can be time consuming, especially for complex models.

Carrying out a mesh sensitivity testing is not always necessary, especially where well-established industry norms exist. Such testing should be done in the context of the results and other necessary sanity checks (mass flow rates, heat release rates etc).

Appendix C: Fire and Smoke Source

C.1 *Introduction*

Specification of the smoke source is a key element in the application of CFD to smoke control design and analysis. This section outlines various approaches available and considers where each may be appropriate and the limitations that may apply. The content is informative only; for detailed guidance the reader should consult authoritative texts such as ‘Computational Fluid Dynamics in Fire Engineering’ by Yeoh and Yuen (published by Butterworth-Heinemann 2009) or the SFPE Handbook of Fire Protection Engineering (5th edition published by Springer 2016).

C.2 *Pyrolysis and Combustion*

In general, a fire involving flaming combustion includes the following processes:

a) heat transfer to the fuel surface, initially by the source of ignition and subsequently from the flames;
b) heating of the fuel;
c) pyrolysis (release of combustible gases);
d) mixing of the pyrolysed gases and oxygen (in the air);
e) combustion (oxidation) of the pyrolysed gases, generating products of combustion;
f) further mixing with air by entrainment and/or forced convection.

The fire may continue as a localised one with a defined smoke plume or may progress to a fully developed one with an extended combustion zone possibly involving the entire room or compartment.

For the fire to continue, the ‘fire triangle’ of fuel, heat and oxygen needs to be maintained. The fire will decay or be extinguished if the fuel ‘runs out’ (items burn out), the oxygen supply ceases (under-ventilation or suppression by water-mist etc) or the heating process terminates (by suppression or limited combustibility of the material).

C.3 Smoke and Visibility

Smoke comprises the products of combustion, any unburned fuel and the air entrained into the fire plume. It is entrained air that, in general, constitutes the main component of smoke as measured by volume; this means that away from the combustion zone smoke can be considered as contaminated air from a fluid dynamics perspective. For most fires, the primary products of combustion will include carbon dioxide and water vapour. Secondary products of combustion may include soot particles and other gases such as carbon monoxide and hydrogen cyanide.

For smoke control design and analysis, soot is often the most relevant product of combustion. The concentration of soot can provide an indication of the likely visibility through the smoke. Reduction of visibility through smoke is a complex process, caused by scattering of light by the aerosol of solid particles and liquid droplets in the smoke, augmented by irritancy effects. Visibility is a vector quantity, meaning it varies in each direction, i.e. line-of-sight. Calculation of light attenuation along a given line-of-sight needs to account for variation in concentration of smoke aerosol particles and droplets; however, this is beyond the scope of general fire engineering analyses. A simplified, and widely adopted, approach is to gauge the reduction in visibility from the concentration of soot. A 'visibility scalar' S [m] is defined using the following empirical correlation [ref: SFPE Handbook of Fire Protection Engineering 5th Ed. (2016) – Chapters 33 and 61], where ρ_{soot} is the local soot particulate density (concentration) [kg m^{-3}], K_m [m^2 kg^{-1}], is the specific light extinction coefficient and C is a constant (set to 3 for viewing light-reflecting elements of construction).

$$S = \frac{C}{K_m \rho_{soot}} \tag{1}$$

An appropriate value of K_m for mixed burning of cellulose and plastic materials is 8700 m^2kg^{-1} [ref: Mulholland and Croarkin (2000) Specific Extinction Coefficient of Flame Generated Smoke, Fire and Materials, vol. 24, pp. 227–230]. Although S is not a distance vector along the line-of-sight, it does provide an indication of the visibility at each location if it is assumed the soot concentration remains constant throughout space and visibility is isotropic. It might also be used as a qualitative indicator of exposure to irritant products of combustion.

C.4 *Modelling smoke in isolation*

In some cases, it may be appropriate to include only air and smoke transport in the CFD simulation without recourse to the fire source itself. Examples include smoke clearance from a protected corridor or a basement car park, where the space is exposed to smoke prior to the operation of the ventilation system. Two approaches that might be considered are:

Purging an initial distribution of smoke

The space to be cleared (purged) is initially filled, or part-filled, with smoke or contaminated air at a specified composition of species and temperature, e.g. warm air containing a suspension of soot or other products of combustion. The CFD model then calculates the purging of smoke or fume by air, allowing the time for the space to be returned various levels of ‘smoke-free’ or tenable conditions to be determined.

The modeller will need to establish, by reasoning and calculation as appropriate, the initial temperature and composition of smoke. A smoke-logged space might be considered as composed of gases with the same composition as a representative fire plume. Dilution of the initial smoke concentration to 1% may then be an appropriate indicator that tenable conditions are achieved [ref: Tamura (1994) Smoke Movement and Control in High-rise Buildings. NFPA, Dec 1994]. A lower level of dilution may be appropriate in other scenarios.

In some applications, e.g. post-fire smoke clearance in an enclosed car park, it may be appropriate to treat the initial smoke as ‘contaminated air’ at ambient or an elevated temperature; the progressive purging of the initial atmosphere with outside air could be examined to establish whether a required air change rate is achieved throughout the space.

The concept of residence time and mean age of air may be useful. This is an output available in some CFD models and provides a measure of how long a fluid element remains within a space. This can be used to identify whether a purge ventilation system works effectively, and there are no stagnation regions.

It may be necessary to consider whether a fire source (discussed below) is required rather than assuming an initial smoke distribution, e.g. to account for the impact of the thermal dynamics of a car fire on an escape lobby.

Smoke source specified as a boundary condition

It may be appropriate in some applications to specify the source of smoke and heat on the boundary of the computational space, e.g. at an apartment doorway on a common corridor. The modeller will need to define the flow rate, temperature and composition of smoke at the boundary. This approach may lend itself, for example, to the analysis of smoke clearance in an escape route following a limited time of exposure, or to the modelling of the external flow of smoke from an open window or shaft. Figure 12 illustrates an example of an open doorway boundary condition.

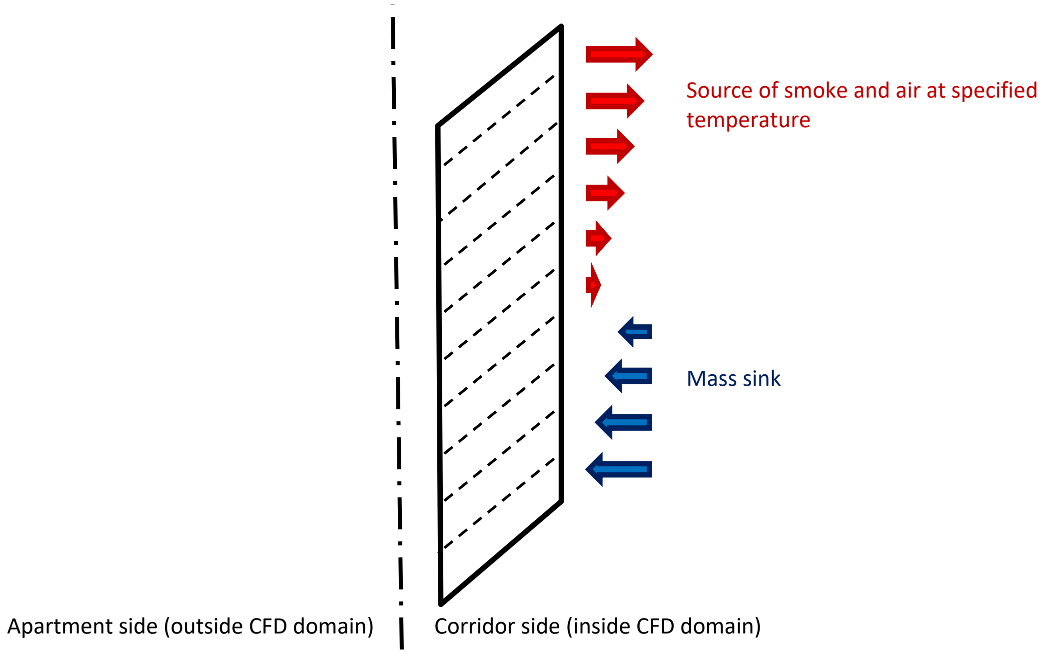

Figure 22 –Example of a doorway boundary condition

C.5 *Modelling the fire source and smoke transport*

Where the generation of heat and combustion products is required as part of the simulation, it is necessary to incorporate one or both fuel pyrolysis (material burning) and gas phase combustion. For example, the generation of heat and combustion products will be an integral part of the analysis of a smoke and heat exhaust ventilation system (SHEVS) for an atrium building or large space such as a warehouse.

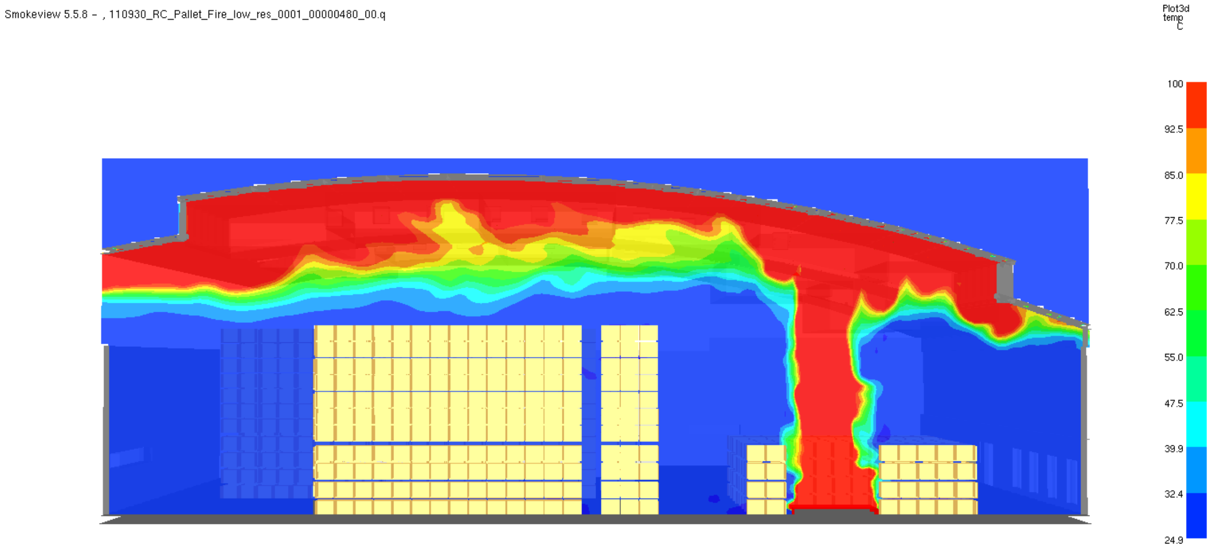

Figure 33 –Section showing temperature from a pallet fire in a warehouse

There are various approaches available, with varying levels of complexity.

Volume (3 dimensional) heat source

In scenarios where the extent of the combustion (flaming) zone can reliably be specified a-priori, then a volumetric source of heat may provide a sufficiently accurate representation of the fire source. Here the fire is prescribed simply as a source of heat; and if the calculation of smoke toxicity or visibility is required the source terms for relevant products of combustion are required too. The modeller will need to define the size and shape of the volume source, referring as necessary to correlations for fire size and flame shape.

There may be limitations to this approach, and cases where it is inappropriate; for example, where the shape of the combustion region is unknown (e.g. a leaning flame due to asymmetric airflow), or where the fire is or under-ventilated.

Single-step combustion with prescribed pyrolysis (fuel release) rate

A relatively simple approach to combustion modelling in CFD is provided by the Eddy Dissipation Concept [ref: Magnussen and Hjertager, On Mathematical Modelling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, *Proc. 17th Symposium (International) on Combustion* (1977), pp. 719–729.] whereby the rate of fuel consumption (combustion) is proportional to the concentration of reactants and the local rate of mixing with oxygen. It provides the foundation for the single step, mixing controlled combustion model available in various commonly used CFD models.

Stoichiometric mixing of air and fuel and infinite rate kinetics are assumed. The reaction takes the general form below, where the products comprises CO_2, H_2O *and* inert N_2.

$$\text{Fuel + air} \rightarrow \text{products (+ heat)} \tag{2}$$

The heat released by combustion drives the smoke transport and is a source of thermal radiation. Heat release can be expressed in a simplified form as below, where m is the fuel supply (pyrolysis) rate [kg s^{-1}], q is the rate of heat release [kW] and ΔH_c is the effective heat of combustion [J kg^{-1}].

$$q = m\Delta H_c \tag{3}$$

The modeller needs to define both m and ΔH_c. Alternatively, they may opt to specify the ‘required’ rate of heat release a priori, in which case m becomes, in effect, an output parameter. In either case, the limiting assumption is that the fire size (burning rate) is specified as an input to the model.

If the transport of secondary products of combustion such as soot is to be modelled, then an empirical fractional yield is required. For example, a 10% soot yield may be specified where visibility through smoke is to be assessed. Note importantly that where visibility is a key output, it is the combination of heat of combustion and soot yield that determines the amount of soot (and hence reduction in visibility) for a given heat release rate.

Advanced pyrolysis and combustion models

If a more detailed representation of the chemical reaction between fuel and air is required, then a more advanced combustion modelling approach will be required. This could be the case where effect of under-ventilated combustion is significant, and the modelling of *CO* production, for example, is important.
The reader should refer to specialist publications for more information on advanced pyrolysis and combustion modelling.

Note that advanced combustion modelling will not, in most cases, be necessary for CFD modelling for smoke control design and analysis.

Suppression

In the simplest approach, suppression might be represented by restricting the heat release rate such that the temperatures within the room or compartment are commensurate with a sprinkler suppressed fire, e.g. the upper layer temperature is circa 120°C (or the value considered reasonable for the scenario being modelled). In effect, this approach involves calibrating the fire heat release to achieve the required conditions. A more conservative approach might be to model a fuel bed fire with a pre-defined heat release rate, e.g.1000kW for a residential fire or 2500 kW for a retail fire. This will in general result in conditions more severe than would be generally anticipated with an operating sprinkler system but can provide a higher level of robustness as it allows for the sprinklers only part working.

A higher level of sophistication would be to include a sprinkler water particle model for example, where the interaction of the water droplets with the hot gases is incorporated and the cooling of the hot gases captured. This does not incorporate effect on fire size/fuel release, however.

At an even more advanced level, the interaction of the suppression system with the pyrolysis and combustion processes, could be modelled. This, however, remains a research topic and is generally outside the scope of a CFD modelling for smoke control design and analysis.

Fire size and ventilation openings

For many smoke control applications, a localised fire source will be most appropriate. The area of the fire source will be constrained, and flashover or fully developed fire conditions will not occur.

In general, the fuel release (pyrolysis) for a pre-flashover fire will be defined either using experimentally derived data, or by adopting a generic, 'design fire'. In the case of a steady fire, this is usually specified in terms of the surface area and the heat release associated with the combustion of the fuel.

For time-dependent fires it is common practice for many applications to adopt a so-called t-squared source (ref: ISO/TR 13387 *Fire safety engineering – Part 2:1999 Design fire scenarios and design fires*):

$$q = 1000\left(\frac{t}{t_g}\right)^2$$

Here, q (kW) is the heat release rate associated with the fire, t (s) the time since the start of the growing stage of the fire and t_g the 'characteristic time' for q to reach 1000 kW. Fires are typically classified as slow, medium, fast or ultra-fast as indicated below:

Growth rate	**Characteristic time t_g (s)**
Slow	600
Medium	300
Fast	150
Ultra-fast	75

The availability of air for combustion needs to be carefully considered when modelling smoke control systems. If the fire is not to be allowed to become under-ventilated, there will need to be sufficient opening(s) to provide the air required for ventilated combustion. Examples of ventilation openings for a compartment include open doors, windows, mechanical ventilation and louvres. Or, combustion ventilation might be formed when glazing fails. Where there is a possibility that compartment ventilation may not be readily available (e.g. basements, specialist fire glazing or specialist glazing protection sprinklers etc) then a sensitivity should be undertaken with some form of generic ventilation allowing a physically realistic fire to develop and a fire where the only ventilation is the open fire compartment door and smoke extract system running. However, in some cases, such as tall buildings, this may not be practical.

A common approach is to incorporate an ‘artificial’ low-level vent to support the fire with doors open and smoke control system running. This may be a good starting point as the approach is often very conservative however, it can lead to impractical airflow rate requirements. The low-level vent is not physically realistic but can give an upper bound on the smoke control systems airflow rates needed. Careful attention to the available ventilation in tall buildings is very important. A low-level ventilation strategy supported by mechanical ventilation may be optimistic. Here, the extract system located in the corridor/lobby, protects the staircase by drawing fresh air from the staircase enclosure. The extract system in tall buildings will prefer to draw smoke from the fire compartment as opposed to the staircase, which is made worse with large openings, like doors and windows that will allow more hot smoke to be drawn from the fire compartment. The low-level vent is considered a specialist ventilation strategy and not a generic fire ventilation strategy.

A sensitivity study of the effects of the ventilation opening in the room of fire origin is recommended under the context of the CFD objectives being studied to ensure an element of realism in the results is balanced with a margin of safety.

Fully developed and post-flashover fires

In the case of flashover and post-flashover fires, a user-defined fuel release rate (e.g. as in a design, t-squared fire) will not generally be appropriate. The pyrolysis process is more complex than for a localised fire. The availability of air for combustion is especially important, influencing whether the fire is fuel- or ventilation-controlled.

There are publications and correlations available to guide the modeller on the size of fully developed fire that can be expected in a room with given openings. For example, e.g. see Karlsson & Quintiere, Enclosure Fire Dynamics, the maximum heat release rate *q* (MW) within a room with a single opening with area *A* (m) and height *H* (m) is given approximately as:

$$q = 1.5A\sqrt{H}$$

Boundary heat transfer to walls, ceilings etc. is also an important consideration post-flashover. As the fire continues, the boundaries heat up and temperatures inside the room or compartment increase, as is reflected in the time-temperature curves employed in fire-resistance tests.

It is not possible to define, a-priori, a fire size or heat release rate for fires that reach post-flashover. The fire size, and room conditions, depends on the distribution and nature of combustible items, the availability of openings (vents) and the properties of the walls and ceiling.

Generally, modelling post-flashover or fully developed fires will be beyond the remit of the smoke control analysis. When it is required, a greater level of understanding of the fire physics and fluid dynamics is required compared to the localised fires discussed above.

Appendix D: 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.

Appendix E: CFD review process

Introduction

The review process provides an essential independent check of a design, ensuring both the analysis and documentation meet the agreed performance criteria. CFD analysis and performance-based fire design require specialist engineering rigor, particularly as the discipline and the adoption of alternative and innovative design proposals increase.

The process should be remote from the immediate design team and provide a fresh view – assessing the design intent, objectives, assumptions and acceptance criteria, as well as verifying that the input data and results reflect the documented case.

It is essential that the whole team value the review process and appreciate the wider objectives involved in it. A reviewer's scope should be technical – to confirm input and outputs for example; and non-technical such as providing a sense check on the design. It is now becoming common practice that the reviewer assists in the approval process, and in these cases the appointed reviewer is determining their support for the design.

Scope

Where a third party and external reviewer is involved, they should be appointed based on agreed scope. This may be a conceptual review of an initial design through to numeric checking of input data and confirming documented results represent the analysis provided.

It can include suitability of documentation to record the design process and conclusions made. It is generally recommended in the reviewer's scope to evaluate the qualification and experience to the design team, rather focus on the technical aspects presented. The scope of the review can include:

- Suitability of relevant codes and guidance;
- The design objectives;
- Assumptions regarding performance criteria, fire scenarios, material properties etc.
- The technical approach documented;
- Appropriateness of models and methods used;
- Input data and the suitability of models and methods used. This also includes error checking of relevant input data;
- The results presented in documentation and within analysis; and the suitability of conclusions made.

The review allows the relevant stakeholder (client, approver/ insurer etc) to make informed decisions regarding a design, with a view that the design will ultimately be agreeable by all relevant parties. It is essential the terms of reference are clear between all parties, in particular the scope and responsibility of the Reviewer. There is also an expectation on the wider team to understand and respond to the Review.

Competency

The competency of the review should reflect the scope required. Experience and judgement are essential, as well as qualifications and expertise in the area the CFD being modelled. CFD is an extensive discipline and it essential the reviewer has competencies in the fire engineering field and the application of the designs being assessed. This is discussed further in Section 4.

Process

The objective of the review is that the reviewer agrees with the design proposal, from initiation through to completion. This often means the reviewer should be appointed early in the design process. Disagreement of fundamental design parameters through late consultation with a reviewer should be avoided.

The reviewers' scope should be clearly defined, and they should be provided with all relevant documentation and analysis to conduct their review. The delivery of the review would generally be a clear itemised list of queries and clarifications sought. The reviewer should substantiate their comments through reference to relevant published information.

The designer should address the comments made. Where there is a technical disagreement the designer should clearly and unambiguously respond to the comment, with technical assessment to support the case. It is often not suitable to provide narrative responses to technical disagreements – for example if temperatures exceed an agreed limit suggesting they may be lower based on other assumptions and fire scenarios would need to be analysed and presented.

The designer should expect to revaluate their design and provide reassessments as necessary such that agreement on the design and its documentation can be provided.

Appendix F: Bibliography & Additional Reading

1 BS 7974:2019, Application of fire safety engineering principles to the design of buildings – Code of practice.

2 BS 7346-4:2003, Components for smoke and heat control systems. Functional recommendations and calculations methods for smoke and heat exhaust ventilation systems, employing steady-state design fires – Code of practice.

3 BS 7346-5:2003, Components for smoke and heat control systems. Functional recommendations and calculations methods for smoke and heat exhaust ventilation systems, employing time-dependant design fires – Code of practice

4 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.

5 BRE 368, Design methodologies for smoke and heat exhaust ventilation

6 BRE FB29, Design fires for use in fire safety engineering, Christopher Mayfield and Danny Hopkin

7 SFPE Handbook of Fire Protection Engineering. 5th Edition, Society of Fire Protection Engineers

8 CIBSE Guide E: Fire Engineering Handbook. 2019

9 BS EN 12101-11, Smoke and heat controls systems - Part 11: Horizontal flow powered ventilation systems for enclosed car parks

10 Fire Safety – Approved Document B: Volume 1 and Volume 2, HM Government

11 BS9991 - Fire safety in the design, management and use of residential buildings – Code of practice

12 BS9999 – Fire safety in the design, management and use of

11 Smoke Control Association, Guidance on smoke control to common escape routes in apartment buildings (Flats & Maisonettes) – Rev 3.1 (July 2020)

12 Smoke Control Association, Design of Smoke Ventilation systems for loading bays & coach parks – A guide for system designers

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

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

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

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

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

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

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

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

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

20 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

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

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

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

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

25 Yeoh and Yuen, Computational Fluid Dynamics in Fire Engineering – Theory, Modelling and Practice. Elsevier, 2009

26 Merci and Beji, Fluid Mechanics Aspects of Fire and Smoke Dynamics in Enclosures. CRC Press, 2016

27 Andrew Basford and Michael Spearpoint (2021) Defining a velocity criterion for stagnant air in buildings. *The journal of the Institution of Fire Engineers* (36), 32-37.