V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

SMARTFIRE – THE FIRE FIELD MODELLING ENVIRONMENT

Dr J. Ewer, Prof. E. Galea, Dr M. Patel, Dr F. Jia, Dr A. Grandison, Dr Z. Wang

Fire Safety Engineering Group, University of Greenwich, London, SE10 9LS, UK
e-mail: J.Ewer@gre.ac.uk

Key words: SMARTFIRE, CFD, Fire Field Modelling

Abstract. *SMARTFIRE is a Fire Field Modelling (FFM) environment that has an unstructured-mesh finite volume RANS Computational Fluid Dynamics (CFD) code at its heart, and incorporates embedded expertise to support the process of Fire Field Modelling. Integrated Knowledge Based Systems assist in the tasks of mesh generation and run-time solution control.*

The software also fully supports expert CFD users with a wide range of supporting tools and advanced technologies to provide modelling realism and accurate and efficient computation, such as parallel computation, intuitive building design from floor plans, and post-processing visualisation. SMARTFIRE comprises a number of intuitive and powerful support programs to allow the modelling of complex scenarios within the built environment.

The presentation will briefly describe the software capabilities and provide examples of its application to a number of real world problems. The presentation will also briefly describe some of the latest fire modelling research being undertaken by FSEG including the Experiment Engine, Toxicity modelling of additional gaseous fire effluent species using LER (with surface reactions), and coupled field-zone modelling.

Unfortunately, it is very easy to assume that because the CFD visualisations look compellingly realistic, that they are physically correct. This is not always the case. The presentation will end with a brief discussion of some of the limitations of fire field modelling and some common modelling errors that users can make when using CFD modelling techniques.

1 INTRODUCTION

This paper is concerned with the SMARTFIRE [1] Fire Field Modelling Environment developed by the Fire Safety Engineering Group (FSEG) of the University of Greenwich. This paper describes the software and details how it is used in fire safety engineering applications through a number of examples. The paper then describes some of the latest research that is being conducted by the SMARTFIRE researchers.

The paper then discusses some of the common errors and limitations associated with running CFD based Fire Field Modelling scenarios using a number of examples.

2 OVERVIEW OF SMARTFIRE

SMARTFIRE (http://fseg.gre.ac.uk) is an open architecture CFD based fire field modelling environment with an integrated knowledge based system that attempts to make fire modelling accessible to non-experts in CFD. The SMARTFIRE software has been under continuous development at FSEG since the late 1980’s and the current release version is V4.1. The software is written in C++ using object orientated principles. There are three major components to the software: the CFD code, user interfaces, and an expert system. The embedded expert knowledge aims to make fire field modelling more accessible to fire engineers with limited CFD expertise. The knowledge embedded within the software is used to support the critical task of mesh specification of fire field simulation scenarios.

The SMARTFIRE Environment consists of a number of key modules (See Figure 1). The SMARTFIRE Scenario Designer allows 2D CAD floor plans to be imported in order to create a building geometry. The building scenario is designed in a 2D CAD-like workspace. Once the building geometry for the required model has been specified, the model is passed into the SMARTFIRE Case Specification Environment. This tool allows further modification of the building geometry to allow a fire modelling scenario to be designed. This includes such details as configuration of materials, fires, boundary conditions (e.g. ventilation and fans), additional fire scenario features (e.g. sprinklers), physical models to use in the analysis (e.g. toxicity for additional fire effluent gasses, thermal radiation, smoke, etc.) and configuration of the solution control. Once a scenario design is complete, the interactive Meshing System assists the user to create a suitable control volume mesh. This process is supported by a Knowledge Based System which applies appropriate meshing rules and expert knowledge for the particular scenario that is being modelled. The CFD Engine of SMARTFIRE is also fully interactive and allows the solution state to be examined during a simulation. The CFD Engine has many of the usual capabilities of a typical RANS based unstructured mesh CFD code (e.g. K-epsilon turbulence with buoyancy modifications for stratified layers, thermal radiation) and a number of fire application specific capabilities (e.g. visibility distance computation through smoke, gaseous combustion).

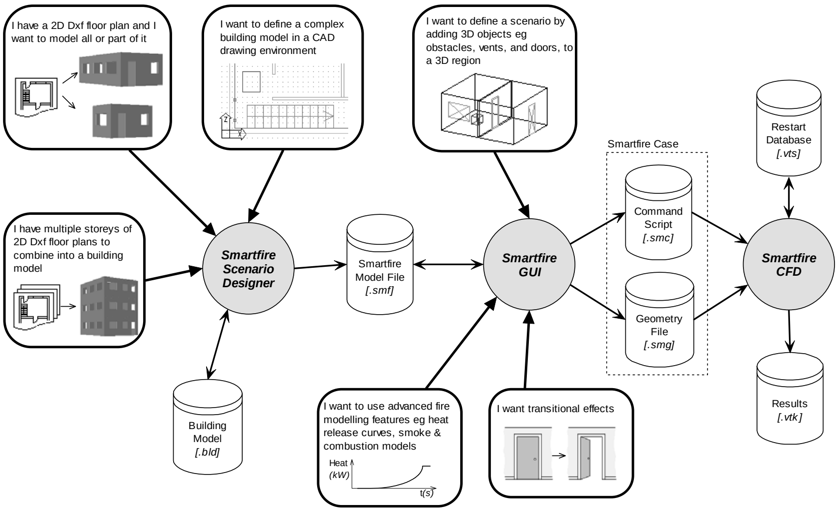


Figure 1: The architecture of the SMARTFIRE Fire Field Modelling Environment

In practice, it has been found that even complete novices to CFD have been able to quickly start to run complex built environment fire modelling scenarios with good accuracy. This has greatly enhanced the user confidence and enabled them to start to learn about the important details of fire modelling that make the scenarios appropriate to simulate real world scenarios.

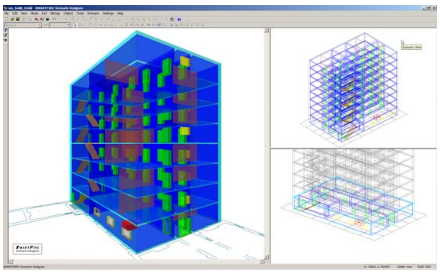
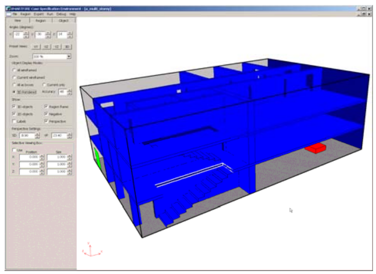
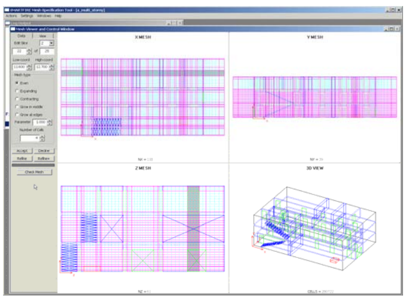
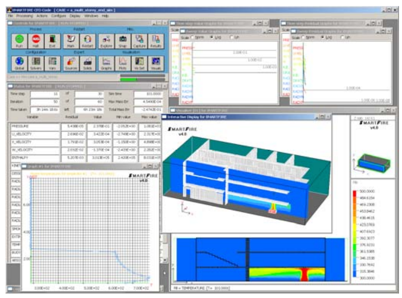
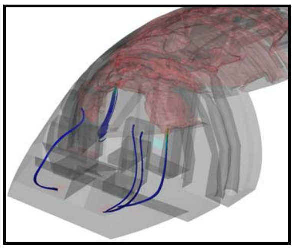
	
The SMARTFIRE Scenario Designer showing a walkthrough for a multi-storey building design	The SMARTFIRE Case Specification Environment allows the user to configure a fire scenario with the ability to set boundaries and physics choices
	
The SMARTFIRE Interactive Meshing System generates a structured mesh which can then be edited by experienced CFD users	The SMARTFIRE CFD Engine showing run-time visualization of solution and interactive controls

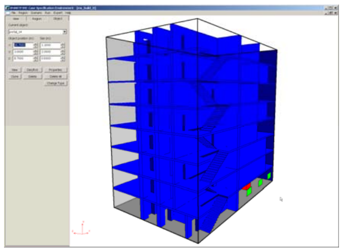
Figure 2 : The components of the SMARTFIRE Environment

3 EXAMPLES OF FIRE APPLICATIONS USING SMARTFIRE

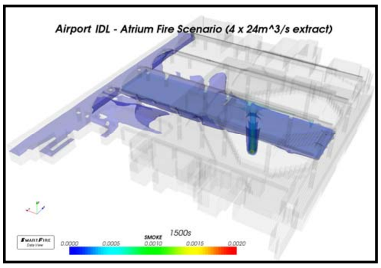
SMARTFIRE has been used on a wide variety of diverse fire field modelling simulation scenarios. The following examples depict the complexity of some of these modelling scenarios (See Figure 3) and their application to real-world fire situations.



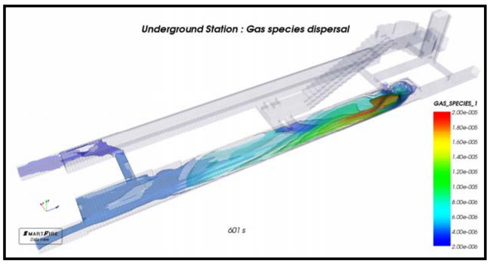
Swissair MD11 Aircraft fire/crash investigation[8] into fire above the cockpit for a forensic study of possible cause of the crash



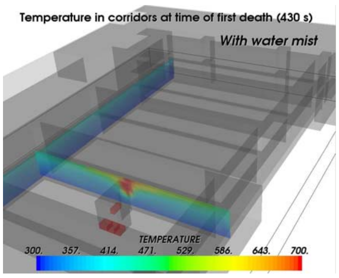
Modelling multi-floor building fires using the Scenario Designer is simplified by the ability to clone storeys of the building



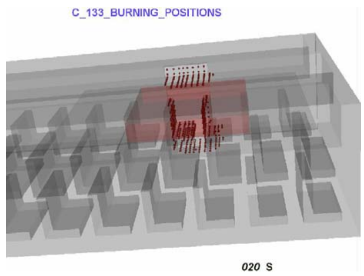
Airport terminal fire showing smoke control design tests for an extraction system linked into the barrel vaulted ceiling



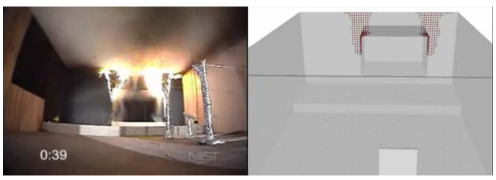
Underground station fire showing the spread of a gaseous concentration from the fire due to the simulated approach of a train through a tunnel



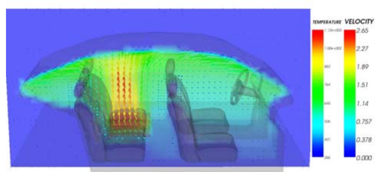
Modelling sprinklers as a fire control strategy for passenger ships. Uses coupled evacuation modelling in EXODUS to test the effectiveness of the sprinkler design



Modelling fire spread inside the body of an aircraft[4] due to a fuel fire outside of an exit door. The model tracks fire spread over burnable surfaces such as seats and overhead luggage compartments



Modelling the rapid fire spread for a fire in a Station night club[3] – comparison with video of full size experimental study performed by NIST



Fire in the passenger compartment of a car – scenario testing a complex unstructured geometry defined by a 3D surface mesh

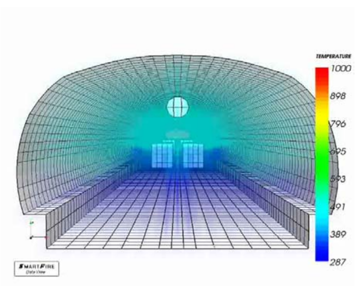
Figure 3 : Examples of applications of SMARTFIRE

4 SMARTFIRE RESEARCH

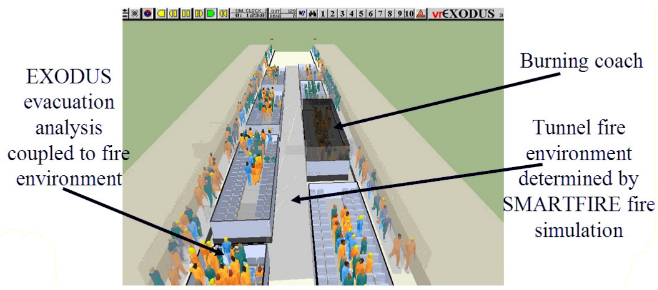
The SMARTFIRE group is investigating a number of research projects aimed at improving the capabilities and realism of fire field modelling (e.g. LER modelling of Toxicity and additional fire effluent gaseous species), improving the reliability, accuracy and performance of the CFD computations (e.g. Coupled Zone/CFD modelling, parallel computation and Experiment Engine CFD control system) and examining fire safety scenarios to improve the understanding of risk factors and mitigation strategies (e.g. Investigating factors such as sprinklers, material usage, forced ventilation to mitigate fire severity).

4.1 Coupled Fire Field And Evacuation Modelling

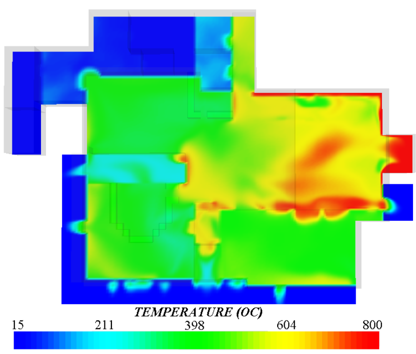
SMARTFIRE was the first fire field model to link to an evacuation modelling software environment, the EXODUS suite of software [3,14-16]. This coupling takes the end of time-step field data for critical fire field properties (such as temperatures, smoke, thermal radiation, and the toxic gaseous species produced by combustion, e.g. CO_2, CO, HCl, HCN, etc.) as zone averaged data which is passed into an EXODUS evacuation simulation. This means that the population in the evacuation simulation experience and react to the fire environment produced in the fire field model. The effects of the various gaseous species and physical properties are determined within the evacuation simulation software using a Fractional Effective Dose model [17].



SMARTFIRE model of coach fire in a road tunnel. Impact analysis looked at effectiveness of fans and sprinkler systems on fire/smoke control



Evacuation simulation uses the Fire Field Model simulation results to set the time-varying environmental conditions that are experienced by the tunnel population



Station nightclub fire[3] in Rhode Island simulated in SMARTFIRE. Temperatures at head height at 100 seconds from the start of the fire

Station nightclub evacuation simulated within the buildingEXODUS evacuation model with fire environment imported from SMARTFIRE Fire Model. Predicted number of fatalities are in close agreement with the real incident

Figure 4 : Coupled Fire Field and Evacuation Modelling

The modelling of coupled fire and evacuation simulation is under continual research and development to provide greater realism in terms of human interaction with fire effluents and ease of use in terms of unified problem set-up.

4.2 Experiment Engine

The Experiment Engine (EE) is a control system built above the CFD engine that attempts to make the software operate (semi-) automatically in a similar way to that in which a CFD expert would use it (See Figure 5). Using the EE, the software performs a shortened, simplified, coarse mesh simulation and then uses the results to help configure the meshing and controls for the production run. The system also monitors the simulation for possible problems. Any detected problems will initiate performing experiments to try to fix the problems, e.g. re-meshing, changes of solution control, etc.

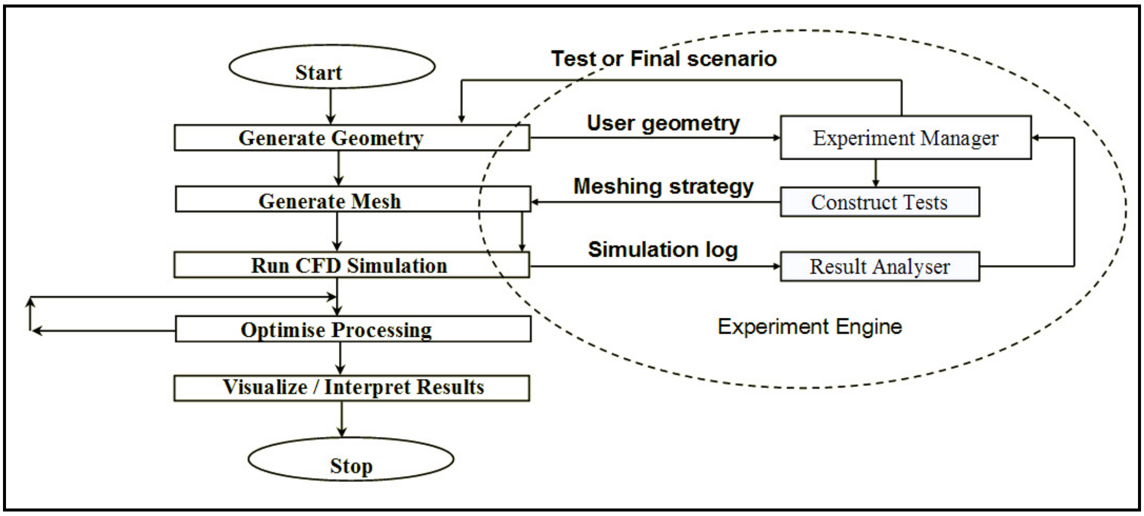


Figure 5 : Diagram showing flow chart of the Experiment Engine interacting with the CFD Engine

The overall aim is to ensure that a solution is reached and that it is a good quality solution. The EE is also fault tolerant because it can (generally) recover from failed runs. The EE can also be configured to perform mesh dependency studies by progressively refining the mesh.

4.3 Understanding Smoke Visibility

It is often the case that post processing visualisation of Fire Field Modelling results will present the smoke concentrations as a volumetric fill that represents the density of the smoke. Such displays can be misleading as they do not always represent what an observer would "see" when subjected to those smoke conditions. To facilitate the user understanding of visibility, the SMARTFIRE developers have produced the visibility distance through smoke visualization tool (See Figure 6). This generates rays from an arbitrary eye-point and tracks the distance that an observer should be able to see based on a calibration factor that takes account of the nature of the smoke and the computed smoke concentration fields.

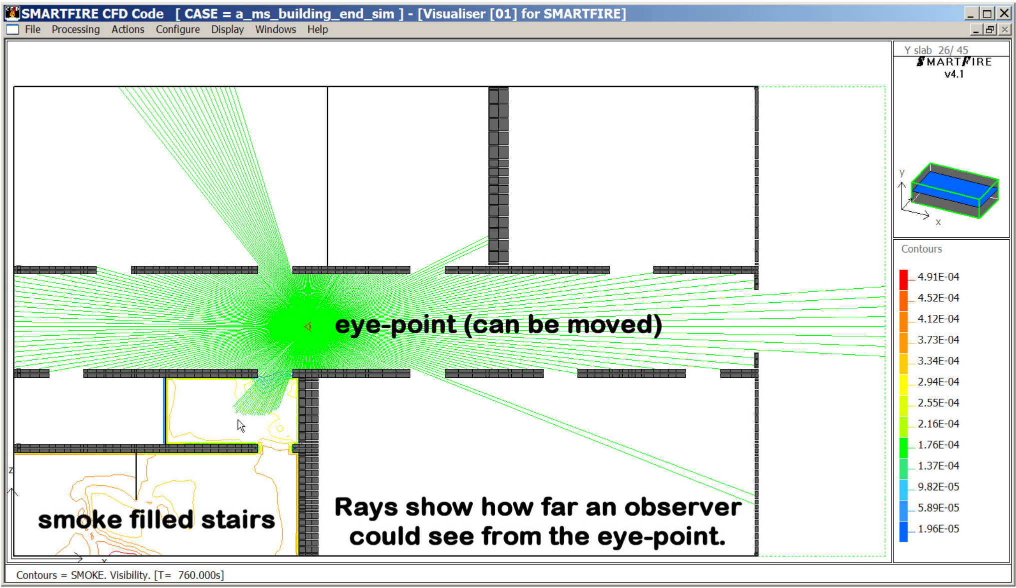


Figure 6 : Visualisation of visibility through smoke

4.4 SMARTFIRE Toxicity Model

The SMARTFIRE toxicity model[7] assumes that combustion conditions within a control volume, with a value of the Local Equivalence Ratio[9] (LER) and a value of temperature, are close to that in small-scale fire experiments (e.g. W14B3 Fire Tests) with the same values of Global Equivalence Ratio (GER) and temperature. Essentially this uses small-scale toxicity date to model real-scale fires.

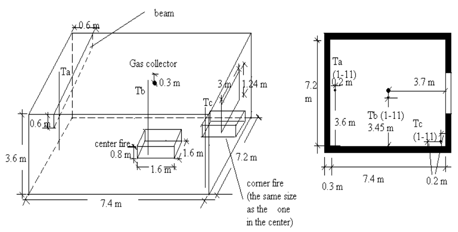
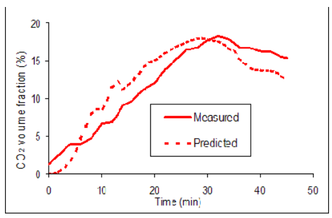
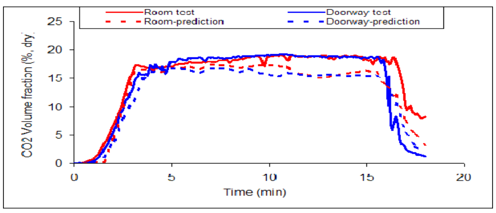
	
The W14B3 Experimental set-up	Example results graph showing comparison of the CO_2 from the experiment with the predicted CO_2 from the Toxicity model
	
The UoG/RockWool room test fire	Example results graph showing comparison of the CO_2 from the experiment with the predicted CO_2 for the both the room and the doorway

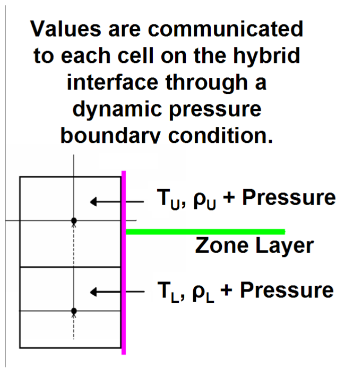
Figure 7 : The SMARTFIRE Toxicity Model

The toxicity model has performed very well on a variety of different test scenarios, generally providing good agreement with experimental measurements taken from calorimetry data.

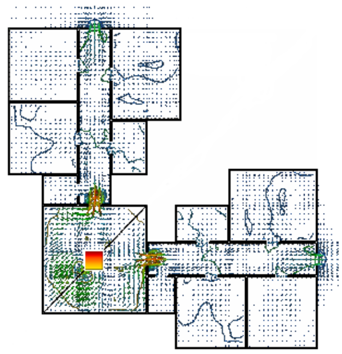
The SMARTFIRE Toxicity model has also been extended to handle additional fire effluent toxic gas species such as HCl[5] and HCN. These additional species can present considerable modelling issues where the gas can react with certain types of surface or if the surfaces are moist and the gas is easily absorbed by water.

4.5 Coupled Field-Zone Modelling

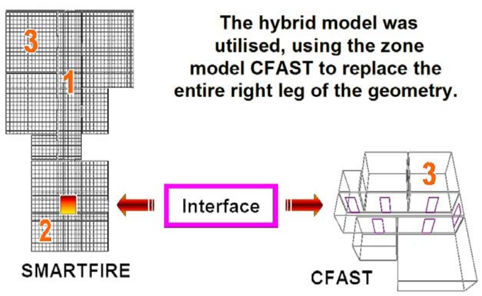
Field modelling (CFD) can have unacceptably high computational time requirements for large or very large scenarios. Generally, not all areas of geometry need to be accurately modelled with CFD, due to having minimal flows, e.g. in areas well away from the fire. Research is now exploring a hybrid approach[6] to fire modelling using a field model only for the most important parts of the geometry and Zone modelling elsewhere. The field model then sees the zone model as an accurate set of boundary conditions at the interfaces between the field and zone modelled regions (See Figure 8). This hybrid approach uses SMARTFIRE and the CFAST zone model produced by NIST.



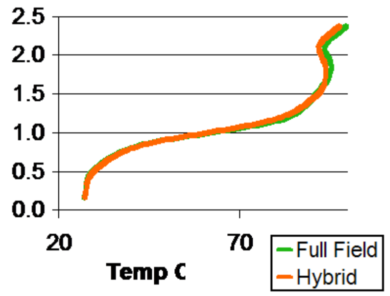
Coupling of the Zone model into the boundary conditions for interface cells in the CFD code



Test scenario showing floor plan of symmetrical building simulated fully with a Field model



Test scenario run in hybrid mode with the upper left area (1) and the fire room (2) modelled in the Field model and the lower right area run in CFAST[10] Zone model



Comparison of Field only and Field-Zone hybrid results in the fire compartment shows only minimal differences in the solution and the same trends.

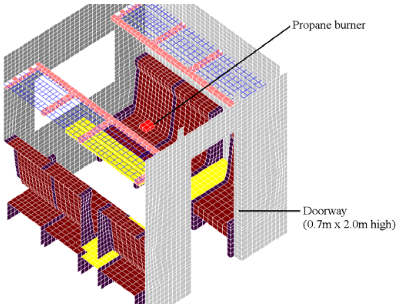
Figure 8 : SMARTFIRE Coupled Field-Zone Model

Since the Zone model is so fast in comparison to the Field model, the percentage time savings are very similar to the percentage cell budget savings from the replacement of parts of the original field model with their zone model equivalents.

4.6 Fire Spread, Suppression and Smoke Production/Transport

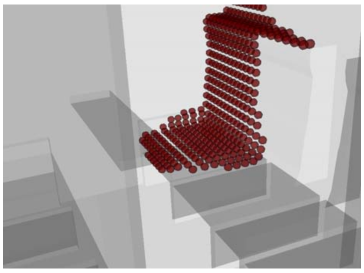
The SMARTFIRE researchers are heavily involved in researching Fire Spread (See Figure 9), Fire Suppression by Sprinklers and Smoke Generation from gaseous combustion/pyrolysis and for Smoke Transport.

A rail car compartment

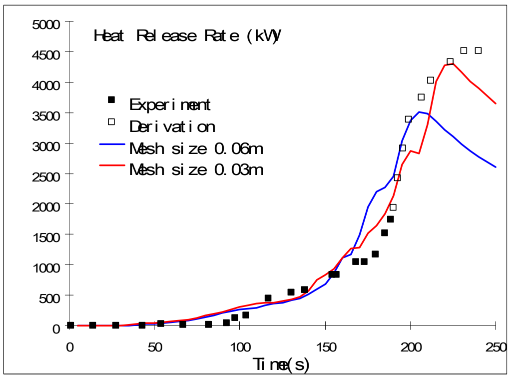


The SMARTFIRE model set up

Observed fire conditions at 132s



Predicted burning locations at 132s



Comparison of experimental and predicted HRR using fine and coarse meshes. The flame spread model has refined the burnable surfaces so that the results are less mesh dependent

Figure 9 : Fire Spread Modelling

Recent research has investigated fire spread model over burnable fuel surfaces in a rail carriage compartment[2] and has attempted to ensure that this is independent of the mesh resolution. The model performs well in predicting the extent of the burning surface over time.

4.7 Modelling Positive Pressure Ventilation (PPV) Attack of Building Fires

Recent investigations have looked at Positive Pressure Ventilation attack of fires in buildings. This uses a high powered fan at the building entrance to blow clean air into the fire compartment and to displace smoke through the fire compartment windows. It should be noted that PPV needs a well managed and cautious approach to ensure that a free through path is available, that local wind conditions will not render the PPV attack ineffective and that the attack will not accelerate the fire spread or flare-up a ventilation controlled fire.

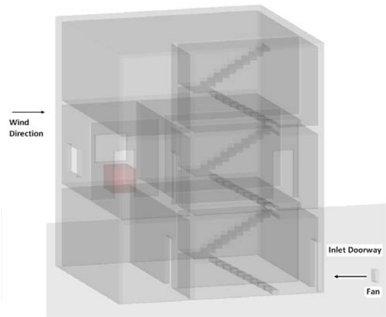
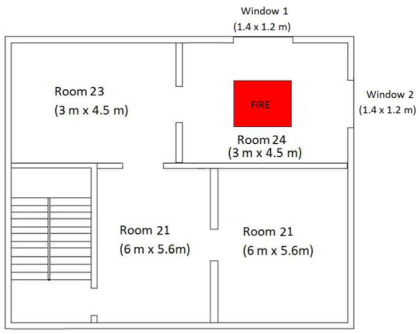
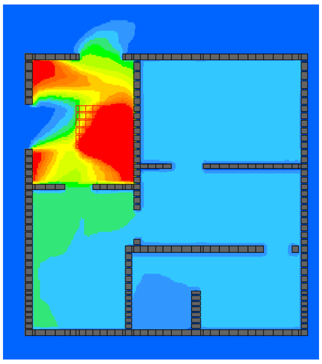
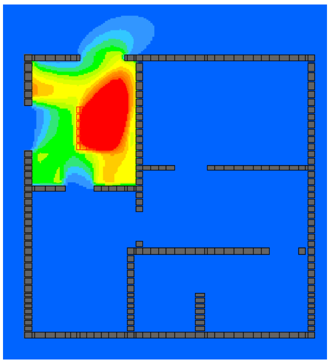
	
Full 3D view of Fire Fighter Training Building	First floor layout
	
First floor temperatures at 1.22m high without PPV fan active	First floor temperatures as 1.22m high with PPV fan active

Figure 10 : Using PPV fan to provide smoke clearance from building fire

5 FIRE FIELD MODELLING ERRORS AND LIMITATIONS

Fire Field Modelling (FFM) uses complex physics, chemistry and mathematics. Users need good understanding of these issues to use FFM effectively. Using general purpose CFD software (for FFM) requires the user to have an even better level of understanding of these skills as it may be necessary to modify equations, write code or set various options that would not normally be used in general purpose CFD, e.g. for the buoyancy terms in K-Epsilon turbulence equations. Even fire specific software can be challenging to use correctly, since there are a multitude of possible boundary conditions, control strategies and solution parameters that can be modified and adjusted by the user.

The following sections identify some of the common sources of error that the user can introduce into fire simulations. These errors can lead to the generation of results that, while they seem to be reasonable or plausible when viewing colour temperature output graphics or animations, suffer from potentially quite serious shortcomings when the results are studied in detail. Wherever possible, the example scenarios are kept relatively small and simple to show the fundamental principles which are affecting the solutions.

5.1 Extended Regions Beyond Vents (i.e. Doors and Windows)

FFM solution will critically depend on the ventilation conditions within a structure and having accurate prediction of all external vent flows (e.g. through open doors). In SMARTFIRE, using a VENT object, to define an external door or window, will automatically give an extended region beyond the vent with free-surface boundary conditions (i.e. fixed pressure). However, most CFD software does not have this capability and it is up to the user to set the boundary conditions appropriately. A common error is to use an outlet (uni-directional outflow) or an inlet (uni-directional inflow), at the vent surface (e.g. doorway). This will typically produce incorrect flow conditions at the boundary. Another common error is placing the ER free-surface too close to the vent which can create reasonable looking flows (i.e. bi-directional) but the flow detail/magnitude may be incorrect. The severity of these errors will depend on relative location of vent and the fire.

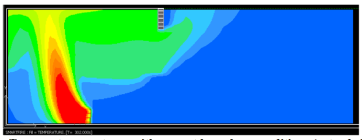
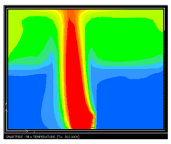
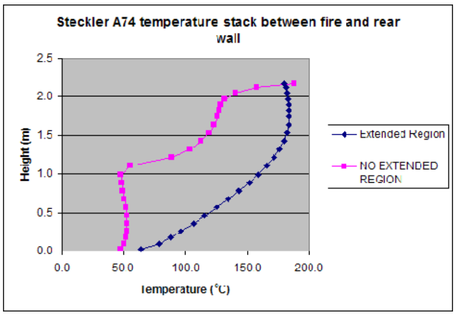
 Temperature contours with correct boundary conditions (extended region).  • Angle of plume recline is different. Temperature contours with zero extended region.	
Simple room modelled with and without an extended region beyond the single door	Vertical temperatures between the fire and the back wall are very different

Figure 11 : Errors from using no extended region

5.2 Ignoring the Effects of Thermal Radiation

Performing a simulation with a large heat output fire or a small fire compartment can give rapidly rising temperatures. As temperatures exceed 500-600K, thermal radiation becomes the dominant heat transfer process due to Stefan–Boltzmann's law, where the radiant energy is proportional to T^4.

In fire calculations, there is often a strong temptation to ignore radiation as it is (i) difficult to calculate (i.e. some codes do not have a thermal radiation model); (ii) expensive to calculate (i.e. using radiation can extend computation time considerably).

Where significant temperatures are observed, but thermal radiation is not calculated, the temperatures tend to be over predicted and can become unrealistically high. To some extent, the buoyancy driven flow will attempt to compensate for increased temperatures but this can lead to unrealistically large flows.

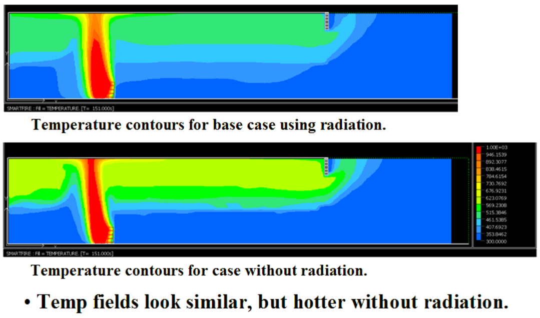
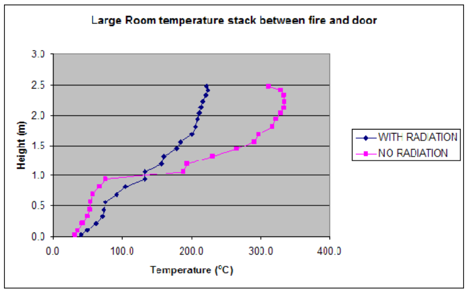
<table>
<tr>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>Long room modelled with and without thermal radiation enabled</td>
<td>Vertical temperature stack shows marked temperature elevation when radiation is not used</td>
</tr>
</table>

Figure 12 : Errors from using no radiation model

5.3 Errors From Using An Inappropriate Radiation Model

A number of different radiation models can be used to represent radiative heat transfer. Unfortunately, the best radiation models tend to use considerably more processing time and so are often less desirable.

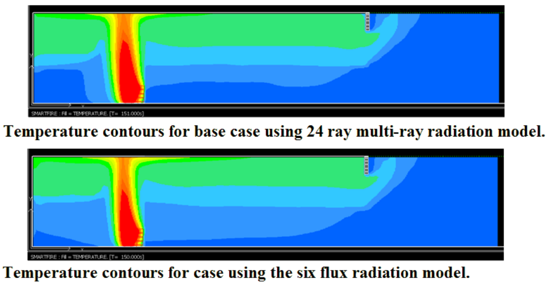
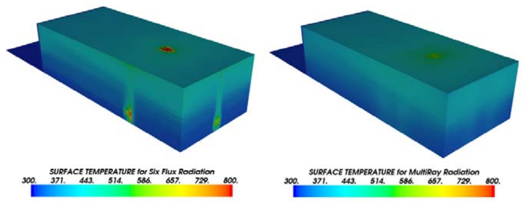
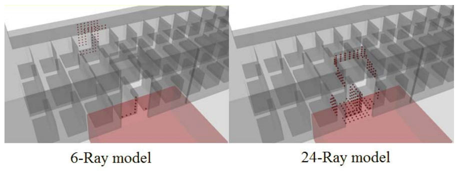
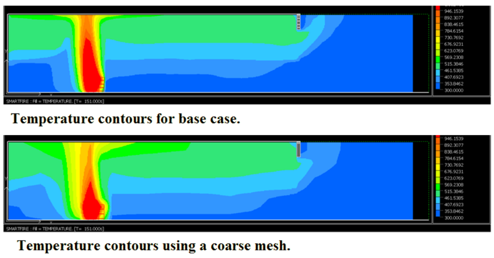
<table>
<tr>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>Long room modelled with a 24 ray radiation model and the six flux radiation model, show hardly any differences in air temperatures or layering</td>
<td>However, the surface temperatures show localised hot spots for the six flux radiation model</td>
</tr>
<tr>
<td></td>
<td>
</td>
</tr>
<tr>
<td>Fire spread into an aircraft from a fuel pool fire outside an exit door[4]</td>
<td>Markedly different burning positions when simulating the fire spread using two different radiation models</td>
</tr>
</table>

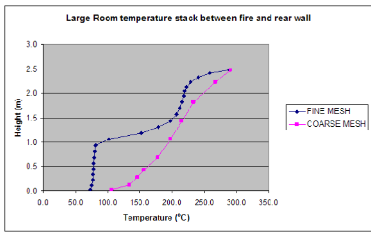
Figure 13 : Errors from using an inappropriate radiation model

Whilst most radiation models will give a fair to good representation of the total radiated heat leaving the fire, they do not tend to give a very accurate measure of thermal radiation arriving at an arbitrary surface. This can be critical in some applications, (e.g. fuel source ignition). The six flux radiation model is not particularly good for this application as it over-predicts the radiative fluxes in the six co-ordinate directions (combines all of the thermal radiation losses into only six coordinate directions) and underestimates the radiative fluxes in the other directions – but it is relatively quick. Multiple-ray/discrete transfer radiation models are much better for this application as they represent the fire (or hot layer) as a point source radiating in many directions.

5.4 Errors From Using An Overly Coarse Computational Mesh

For practicing fire engineers, coarse meshing is very attractive as it can dramatically reduce simulation times thereby improving project turnaround times. Unfortunately, many important complex and normally observed flow structures (e.g. re-circulations, plumes, jets and opposing layer flows) will not occur if the mesh is too coarse or may occur in incorrect locations. Ideally, a thorough CFD fire analysis will include mesh dependency checks (using different mesh resolutions) to ensure that a suitably refined mesh has been used – but this is not always the case. Coarse mesh simulations are useful to get a quick idea of likely fire conditions, however, they should not be relied upon for engineering decisions and should never be considered the final solution. It is often difficult to detect poor nature of solution unless experienced in CFD fire simulation.



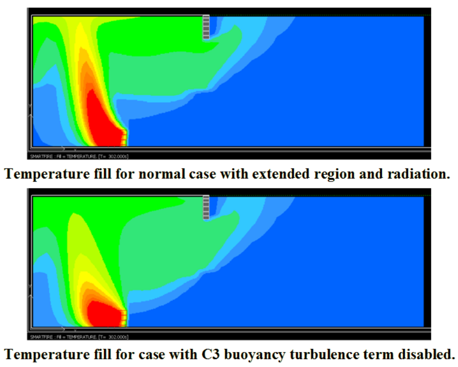


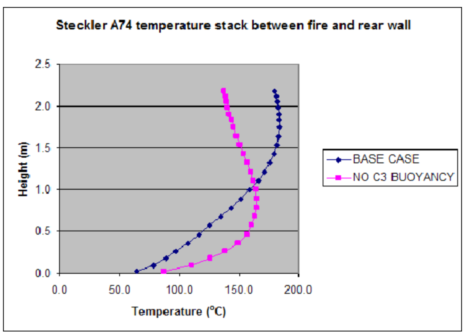
Long room modelled using a moderately refined and a coarse mesh	Vertical temperature stack shows lack of stratification and smeared temperature profile for the coarse mesh

Figure 14 : Errors from using an overly coarse computational mesh

5.5 Errors From Not Using Buoyancy Modification Of Turbulence

The KE turbulence model needs an added "C3" buoyancy modification term to allow the model to correctly predict the temperature stratification in the y-direction. This term is essential for fire simulations because of the large range of temperatures encountered.





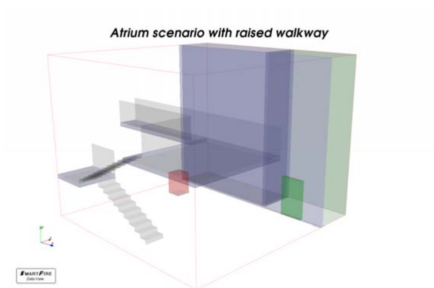
Small room modelled with and without the C3 buoyancy modifier in the KE turbulence model	Vertical temperature stack shows lack of hot layer stratification for the case without C3 modifier

Figure 15 : Errors from using no C3 buoyancy modifier

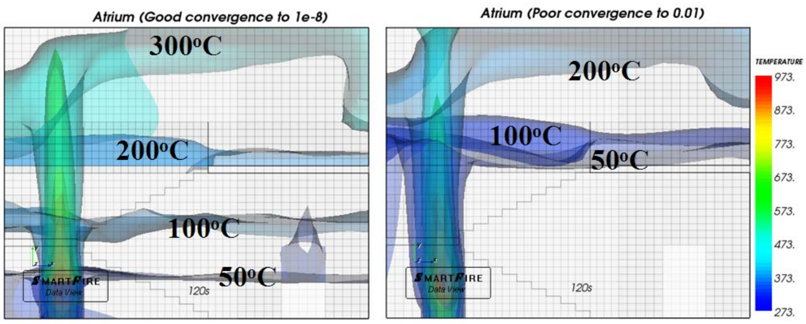
In many general purpose CFD codes, the C3 modifier is either absent or inactive by default. In which case, the user must either write code to add the term to the equations (e.g. include as a source term) or "set" a switch in order to activate the term.

5.6 Errors From Poor Convergence

Poor convergence can produce results which are highly misleading – although they look plausible seen in isolation. In a hypothetical atrium scenario, poor convergence greatly delayed the development of the high temperature regions, suggesting that the environment was tenable for far longer – producing longer Available Safe Evacuation Time (ASET). Poor convergence could equally have lead to higher temperatures producing shorter ASET.



Atrium geometry with raised walkway to be tested for evacuation purposes



Comparison of the well converged and poorly converged simulations showing marked differences in the temperatures at the walkway at 120 seconds

Figure 16 : Errors from using poor levels of convergence

It should be noted that the poorly converged cases required considerably less computer time than the well converged case, which could be seen as an incentive for producing poorly converged results. Using a tolerance of 1e-5 produced results which were almost identical to those using convergence to 1e-8. While the differences in the predicted results were produced by using poor convergence tolerance, a similar effect can be produced if the maximum number of sweeps per time step is set too low.

5.7 Final Considerations About Errors

There are a few special instances where poor modelling decisions can produce poor outcomes (for example giving poor ASET time for a building) without there being any actual "error" or omission in the modelling.

One such example is when using a field model to predict the temperatures, thermal radiation, smoke and toxic gaseous species generated by the fire and modelling their dispersion through the building BUT failing to combine the various factors to give the cumulative effect that a building occupant would experience on their escape route.

All of the examples that have been discussed have assumed that the scenario models are fully representative of the actual real world fire cases.

6 CONCLUDING COMMENTS

FSEG is one of world's largest fire and evacuation modelling research groups and has:

- undertaken research into CFD fire modelling for more than 25 years,
- been developing its SMARTFIRE CFD fire simulation software for more than 20 years,
- been supporting users of its SMARTFIRE software around the world for more than 15 years, and
- supported the development of validation standards for Fire Field Modelling[11][12][13].

In this paper we have presented an overview of the SMARTFIRE Fire Field Modelling Environment and described some of the recent application areas. Recent research undertaken by FSEG into fire simulation and fire modelling has also been described. Also presented, in this paper, are examples of some of the common poor modelling choices that engineers may make when running CFD fire simulation software and the impact these may have on the quality of their fire predictions.

FSEG continues to develop, support and apply the SMARTFIRE software to fire safety and fire engineering problems around the world. For more information concerning FSEG research and its two main software tools, SMARTFIRE and EXODUS, interested readers are referred to the FSEG web pages at http://fseg.gre.ac.uk.

REFERENCES

[1] J. Ewer, F. Jia, A. Grandison, E. Galea and M.K. Patel, SMARTFIRE v4.1 Technical Reference Manual, FSEG, University of Greenwich (2008)

[2] X. Hu, Z. Wang, F. Jia, E.R. Galea, Simulating A Rail Car Fire Using A Flame Spread Model, Interflam 2010, to be published in conference proceedings (2010)

[3] E.R. Galea, Z. Wang, A. Veeraswamy, F. Jia, P. Lawrence, J. Ewer, Coupled Fire/Evacuation Analysis of the Station Nightclub Fire, J. Proceedings of 9th IAFSS Symposium Karlsruhe, Germany, ISNN 1817-4299, pp 465-476. DOI:10.3801/IAFSS.FSS.9-465 (2008)

[4] Z. Wang, E.R. Galea, and F. Jia, A computational study of the characteristics of aircraft post-crash fires, Proceedings, 5th Triennial International Fire and Cabin Safety Research Conference, Atlantic City, publishers US FAA (2007)

[5] Z. Wang, F. Jia, E.R. Galea, M.K. Patel and J. Ewer, Predicting HCl concentrations in fire enclosures using an HCl decay model coupled to a CFD-based fire field model, Fire and Materials, 31, pp443-461, DOI: http://dx.doi.org/10.1002/fam.942 (2007)

[6] D.J. Burton, A.J. Grandison, M.K. Patel, E.R. Galea, J. Ewer, Introducing a Hybrid Field/Zone Modelling Approach for Fire Simulation, Proceedings of the 11th

International Fire Science & Engineering Conference, Interflam 2007, 3-5th September 2007, Royal Holloway College, University of London, UK, Volume 2, pp. 1491-1497. ISBN 978 0 9541216-9-3 (2007)

[7] Z. Wang, F. Jia, E.R. Galea, M.K. Patel, Predicting Toxic Species Concentrations in a Full-Scale Vitiated Fire, Proceedings of the 11th International Fire Science & Engineering Conference, Interflam 2007, 3-5th September 2007, Royal Holloway College, University of London, UK, Volume 2, pp. 1047-1058. ISBN 978 0 9541216-9-3 (2007)

[8] F. Jia, M.K. Patel, E.R. Galea, A.J. Grandison, J. Ewer, CFD Fire Simulation of the Swissair Flight 111 In-Flight Fire – Part 2: Fire Spread analysis, The Aeronautical Journal. Vol 110, Number 1107, pp 303-314 (2006)

[9] E.R. Galea, F. Jia, and Z. Wang, Predicting toxic gas concentrations resulting from enclosure fires using local equivalence ratio concept linked to fire field models. Fire and Materials, Vol 31, Issue 1 pp 27-51 Jan/Feb (2007)

[10] R.D. Peackock, W.W. Jones, P.A. Reneke, G.P. Forney, CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 6), User's Guide, NIST Special Publication 1041 (2008)

[11] A.J. Grandison, M.K. Patel, E.R. Galea, SMARTFIRE Verification and Validation Report, Report Version 4.1.1, Report Revision Date 05/2007, CMS, University of Greenwich (2007)

[12] A.J. Grandison, M.K. Patel, E.R. Galea, Development of Standards for Fire Field Models. Report on Phase 1 Simulations, Office of the Deputy Prime Minister, Fire Research Division, Research Report 2/2003 (2003)

[13] A.J. Grandison, M.K. Patel, E.R. Galea, Development of Standards for Fire Field Models. Report on SMARTFIRE Phase 2 Simulations, Office of the Deputy Prime Minister, Fire Research Division, Research Report 1/2003 (2003)

[14] E.R. Galea, J.M. Perez Galparsoro, EXODUS: An Evacuation Model for Mass Transport Vehicles. UK CAA Paper 93 006 ISBN 0 86039 543X (1993)

[15] E.R. Galea, J.M.P. Galparsoro, A Computer Based Simulation Model for the Prediction of Evacuation from Mass Transport Vehicles. Fire Safety Journal Vol 22 pp 341 - 366 (1994)

[16] S. Gwynne., E.R. Galea, P.J. Lawrence, L. Filippidis Modelling Occupant Interaction with Fire Conditions Using the buildingEXODUS model, Fire Safety Journal, 36, pp327-357 (2001)

[17] E.R. Galea, M. Owen, P. Lawrence. Computer Modelling of Human behaviour in aircraft fire accidents. Toxicology 115, pp. 63-78, Publ. Elsevier (1996)