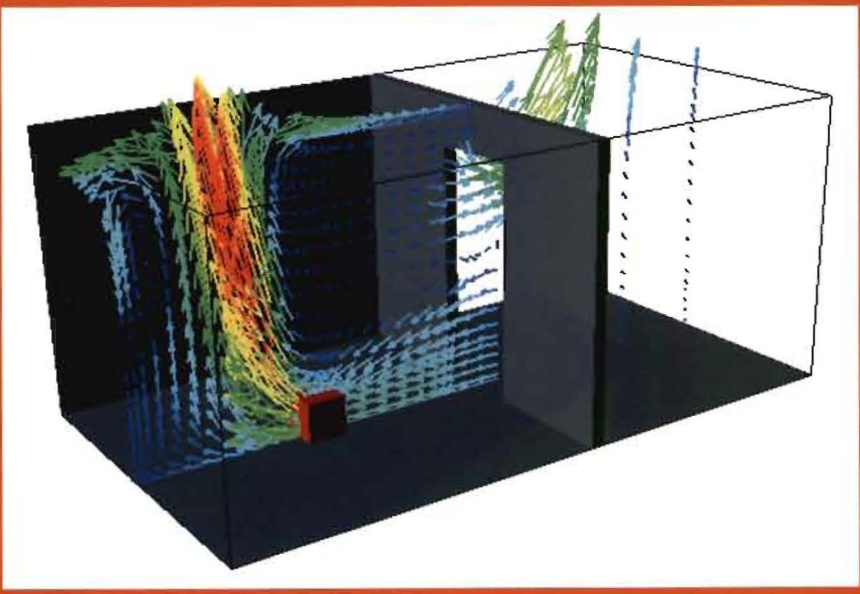
OFFICE OF THE
DEPUTY PRIME MINISTER

# Development of Standards for Fire Field Models



Fire Research Report Number 85

OFFICE OF THE
DEPUTY PRIME MINISTER

# Development of Standards for Fire Field Models

## Fire Research Report Number 85

December 2003

Brian Hume
Fire Research Division
Office of the Deputy Prime Minister: London

Following the reorganisation of the government in May 2002, the responsibilities of the Home Office in this area were transferred to the Office of the Deputy Prime Minister (ODPM).

Office of the Deputy Prime Minister
Eland House
Bressenden Place
London SW1E 5DU
Telephone 020 7944 4400
Web site www.odpm.gov.uk



For further information, please contact the Head of Fire Research Division, ODPM, Fire Research Division, Zone 18/D, Portland House, Stage Place, London SW1E 5LT.

Further copies of this publication are available from:
Office of the Deputy Prime Minister Publications
PO Box 236
Wetherby LS23 7NB
Tel: 0870 1226 236
Fax: 0870 1226 237
Textphone: 0870 1207 405
E-mail: odpm@twoten.press.net

Online: www.publications.odpm.gov.uk

A copy of this report is also available on the ODPM website at: www.odpm.gov.uk/fireresearch

ISBN 1 85112 679 1

Printed in Great Britain on material containing 75% post-consumer waste and 25% ECF pulp

December 2003

Reference no. 03LRGG01797

# Development of Standards for Fire Field Models

## INTRODUCTION

Computer models of fire growth are useful tools for assessing the fire safety of building designs and the results of these models are sometimes included in submissions of building proposals to fire brigades. In this project, carried out on behalf of Fire Research Division of the ODPM (formerly DTLR) by the Fire Safety Engineering Group (FSEG) at the University of Greenwich, the basis of a standard for fire field models has been developed. To meet the standard, a fire model would need to achieve a sufficient level of accuracy in predicting the results of a set of specified test cases, when used under controlled conditions. The proposed standard was tested on three models (CFX, PHOENICS and SMARTFIRE) with the participation of the model developers.

## BACKGROUND

There are two main types of computer models used for predicting fire growth in buildings: zone and field models. This project concerns field models that are able to model a building fire in more detail than zone models. They do this by defining a ***grid*** which divides the building volume into a large number of smaller volumes (called ***cells***) and carrying out repeated calculations on each cell to estimate the physical conditions at each location (see Figure 1). In all, many millions of calculations are performed to model one fire.

Fire field models are based on a technique called 'Computational Fluid Dynamics' or CFD, which is used in a number of other fluid flow applications such as weather forecasting and industrial plant design. Although fluid flow, i.e. the movement of gases and liquids, can be predicted with some confidence with field models, other processes such as flame spread and thermal radiation are more difficult to predict.

There are several field models now available for prediction of fire development in buildings, and although they are all based on the same principles, at least for fluid flow, the actual computer codes are different and different sub-models may be used for modelling processes such as thermal radiation. The results obtained by different models are therefore likely to vary.

In fact, the accuracy of the results of a fire model will depend on the following:

- the accuracy of the model used when applied to the case in question
- how much knowledge the user has of fire field modelling and of the specific model they are using and
- the appropriateness of the input data (eg fire heat release rate) used to define the case, which will depend on the user's knowledge of fire engineering.

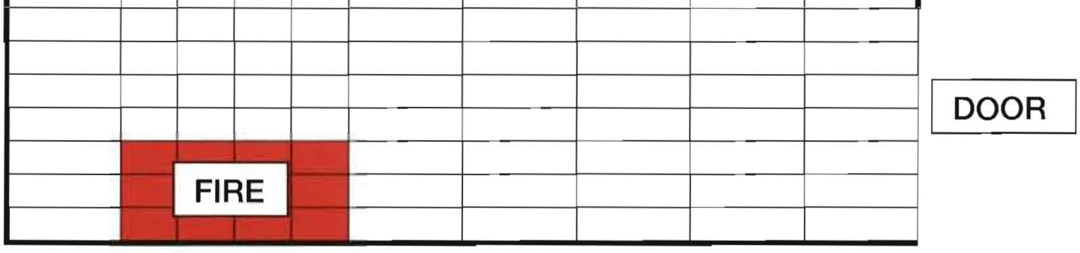



*Figure 1. Example of grid used in CFD*

The purpose of the standard developed in this project is specifically to assess the accuracy of the model. The influence of the other two factors is removed by ensuring that the user has sufficient knowledge of the model to use it correctly and by defining a set of input data for each test case to be used in each model. This input data includes all set up parameters, such as the grid that divides up the building into cells. Any differences in the results obtained from the different models should therefore only be due to differences within the models themselves.

## THE MODELS USED

Several developers of well-known fire field models currently used in the UK were approached to participate in this project and of these, three agreed to do so. These are listed in Table 1.

*Table 1. The Models Used*

| Field Model Name | Developer |
|---|---|
| CFX version 4.2 | AEA Technology, Harwell |
| PHOENICS version 3.1 | Cham Ltd, London |
| SMARTFIRE version 2.01 | University of Greenwich |

Both CFX and PHOENICS are general purpose CFD codes, while SMARTFIRE was developed specifically as a fire field model.

## THE TASK GROUP

A task group was set up to review the work as it proceeded. The group consisted of representatives from the software developers (AEA Technology, Cham and University of Greenwich), one independent user of fire field models drawn from the user community (Arup Fire) and a representative from the ODPM Fire Research Division.

## METHOD

A set of 10 test cases was defined and agreed by the task group, which could be used as the basis of a standard. To meet the standard, the model would need to be able to predict the conditions in the test cases to within a specified range. The set of test cases is shown in Table 2. They are divided into two sets: 5 CFD (non-fire) cases and 5 fire cases. The cases were designed to test the basic features of the models and to ensure that these functioned correctly.

*Table 2. The 10 proposed test cases*

| | |
|---|---|
| **CFD Cases** | |
| 1/1 | Two-dimensional turbulent flow over backward facing step. |
| 1/2 | Turbulent flow along a long duct. |
| 1/3 | Symmetry boundary condition. |
| 1/4 | Turbulent buoyancy flow in a cavity. |
| 1/5 | Radiation in a three-dimensional cavity. |
| **Fire Cases** | |
| 2/1 | Steckler[1] Room (heat source). |
| 2/2 | Steckler Room (combustion model). |
| 2/3 | Fire in a completely open compartment with lid (heat source). |
| 2/4 | CIB W14 fire (combustion model). |
| 2/5 | Large fire (combustion model) |

Full details of these test cases may be found in Appendices B and C of the report on the Phase 1 simulations [1].

For a model under test, the proposed standard requires two types of simulation to be performed, referred to here as Phase 1 and Phase 2 simulations. In the Phase 1 simulations the input data and set-up conditions have been rigidly defined by FSEG in consultation with the task group.

1 This case is taken from a set of 55 full-scale fire tests carried out by Steckler of the US National Bureau of Standards (now National Institute of Standards and Technology) [3]. The purpose of these tests was to study the flow induced by a simulated pool fire in a compartment and they are often used by fire model developers to check their results.

This includes the grid specification, which defines the cells as well as the physical models to be activated, such as the type of radiation model. Also specified are the parameters for which results are to be generated, eg the temperature at location X for the first 600 seconds from ignition time.

In the Phase 2 simulations, the organisation that developed the model was asked to repeat the simulation using whatever specification of the case they desired. Phase 2 simulations therefore allowed the developers to demonstrate the full capabilities of their model. Any changes made to the set-up of the case would have to be declared to ensure that they were valid.

In this project, FSEG ran Phase 1 simulations on all test cases using each of the three models. As a check, the participants (Cham, AEA Technology and Arup) were asked to perform Phase 1 simulations using just their own model on at least two test cases, one CFD case and one fire case, without informing FSEG which cases they had chosen.

On completing the Phase 1 simulations the participants were invited to undertake Phase 2 simulations but in the event none could find the time to do this within the period allowed in the project. However, FSEG carried out Phase 2 simulations with their own model, Smartfire, on those cases where there was thought to be room for improvement.

***Results of Phase 1 (Prescribed input data)***

*Case 1/4: Buoyant, turbulent flow in a duct.* As an example of a CFD case (i.e. non-fire case), Case 1/4 is a 2-dimensional case where a buoyant, turbulent flow is created by the heating and cooling effect of two facing walls (see Figure 2). The air heated by the hot wall rises due to buoyancy, whilst air cooled by the cold wall falls, and the resulting flow pattern in the duct causes turbulence. The walls at the top and bottom of the duct are adiabatic, in other words they are perfect insulators and so there is no heat flow in or out of them. The purpose of this case is to test the ability of the model to predict buoyancy and turbulence in the absence of other factors.

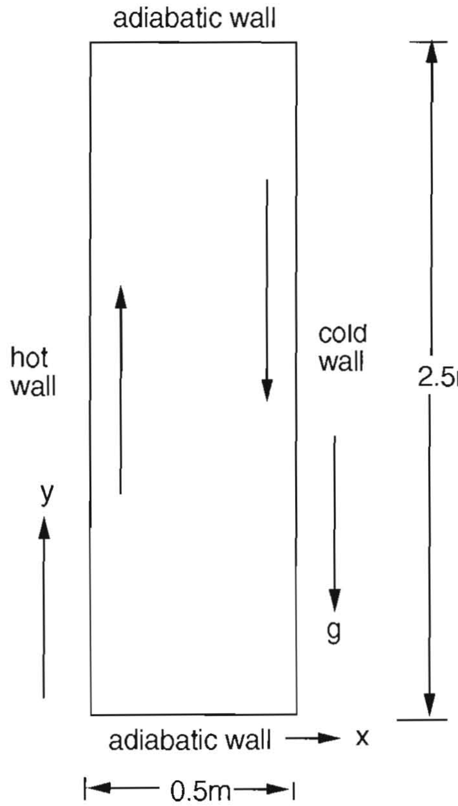



*Figure 2. Case 1/4 configuration*

Figure 3 shows a comparison of the results of the three models and experimental data. The parameter shown here is the turbulent fluctuation half way up the length of the duct as it varies across the width. The turbulent fluctuation is measured by $\sqrt{k}$ where $k$ is the turbulent kinetic energy in kg m $s^{-2}$. All models are in reasonable agreement with each other and with the experimental data.

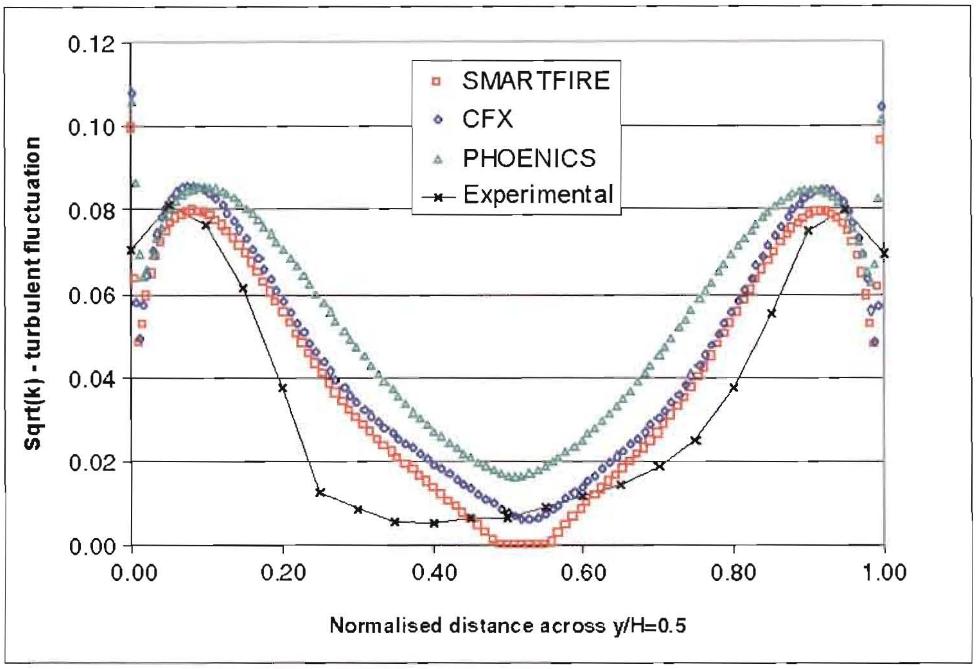



*Figure 3. Case 1/4 – Turbulent fluctuation across duct*

*Case 2/1 – Steckler room with volumetric heat source* and *Case 2/2 – Steckler room with combustion model.*

This test case is a standard fire model test case used by a number of field and zone model developers. Its primary purpose is to test the fire model's capability in predicting temperature and flow distributions in a small compartment subjected to a steady, non-spreading fire. Model predictions are also compared with experimental results.

The non-spreading fire was created using a centrally located methane burner (position A in Figure 4) of 62.9kW heat output. The experiments were conducted in a compartment measuring 2.8m ( 2.8m in plan and 2.18m in height (see Figure 4). The walls and ceiling were 0.1m thick and they were covered with a ceramic fibre insulation board to establish near steady state conditions within 30 minutes.

In Phase 1, this test case was specified such that, within the models, the walls were all assumed to be thermal insulators and perfect radiative reflectors. The case was run for 200 seconds of simulated time using 200 time steps of 1 second at which point steady state conditions were achieved in the simulation.

The methane burner was modelled using two methods:

1) Using a simple volumetric heat source (case 2/1), where the known heat output of the burner is injected into a defined volume in the centre of the compartment.

2) Using a combustion model (case 2/2), where the known fuel mass release rate is injected into a defined volume and the mixing and combustion are calculated.

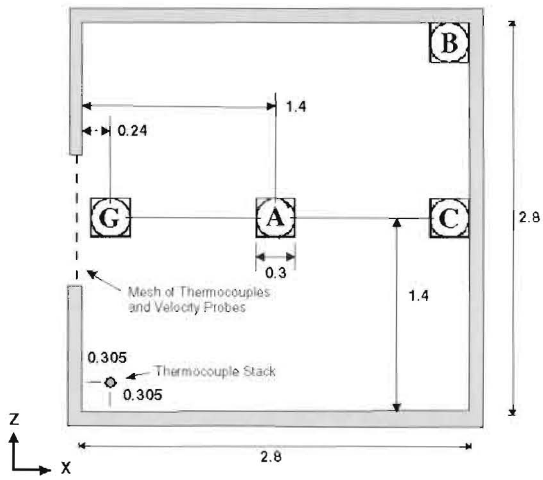



*Figure 4. Configuration of Steckler room*

Comparisons of temperatures from floor to ceiling in one corner of the compartment are shown in Figure 5 for cases 2/1 and 2/2. Results of case 2/2 using the combustion model are marked with a 'C' in the key. Results from PHOENICS version 3.3 using the volumetric heat source model are also shown. The results presented are after 200 seconds of simulated time at which point the conditions have reached a steady state.

As can be seen from the graph, all models predicted higher temperatures in the upper layer[2] than the experimental data. This can be expected as the walls have been treated as perfect insulators and so no heat would be lost through them, which is not the case in reality.

It is also notable that there are differences between the models with Smartfire predicting significantly higher temperatures in the upper layer than the other models.

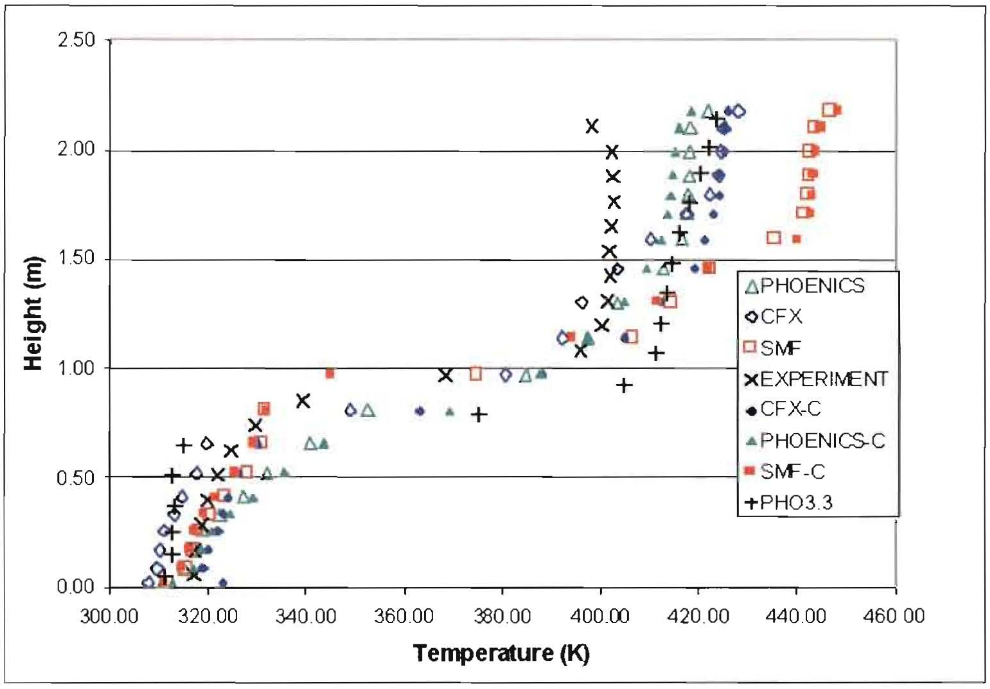



*Figure 5. Corner Stack temperatures produced using heat source model and combustion model.*

Figures 6 – 8 show the temperature contours of a slice through the centre of the compartment, predicted by the three models using combustion modelling. It can be seen that each model produces a similar picture.

2 The layer of hot smoky gases which forms beneath the ceiling

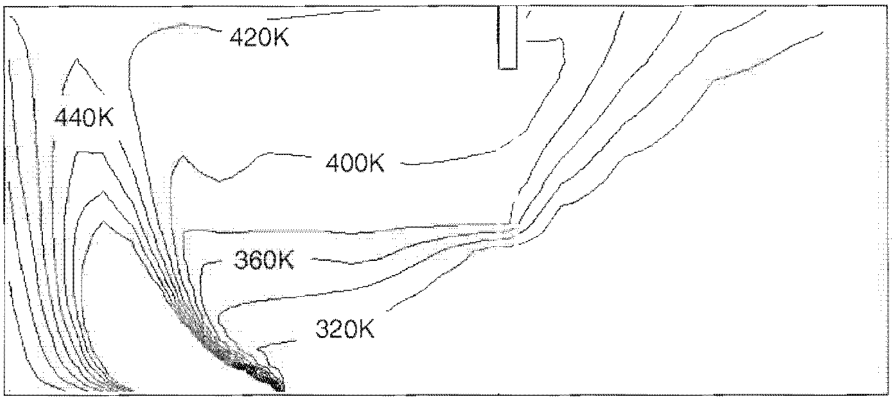



*Figure 6. Case 2/2 – Temperature contour plot produced by PHOENICS using the combustion model.*

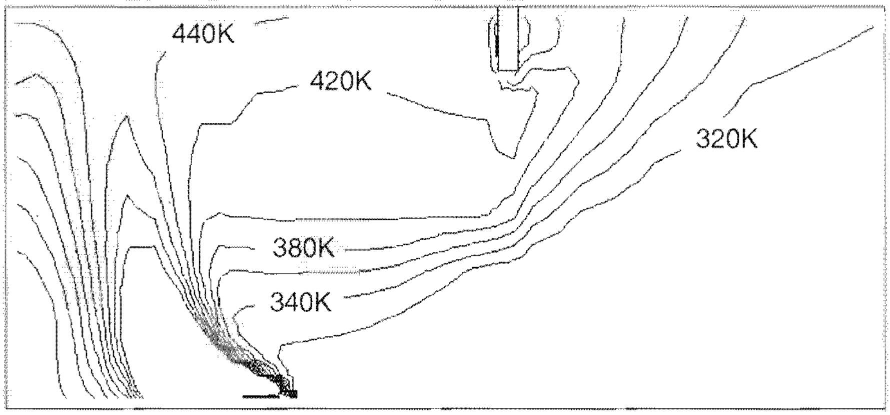



*Figure 7. Case 2/2 Temperature contour plot produced by CFX using the combustion model.*

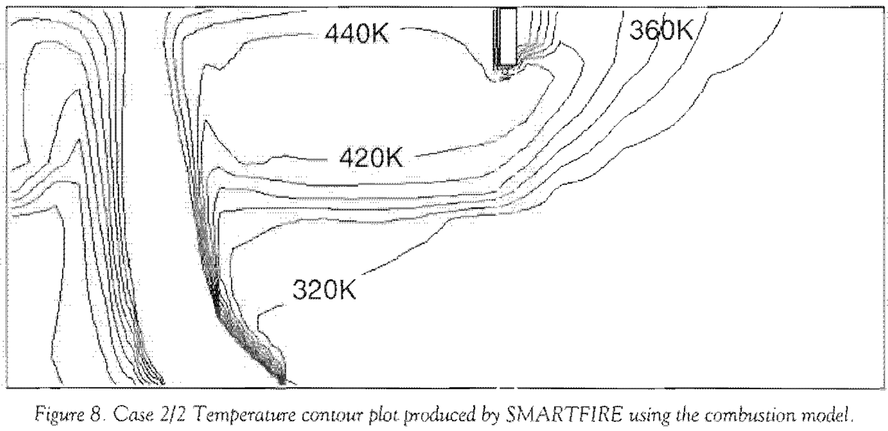



*Figure 8. Case 2/2 Temperature contour plot produced by SMARTFIRE using the combustion model.*

***Results of Phase 2 (Developer's own input data).***

In the Phase 2 simulations, the organisation who developed the model were asked to repeat the simulation using whatever specification of the case they desired, to obtain a more accurate representation of the case. However, only FSEG were able to carry out this work in the time available and so only the results from the Smartfire model are available for Phase 2 [2].

FSEG carried out Phase 2 simulations on a selection of the original test cases, where it was thought likely that an improvement could be achieved.

*Case 2/1 – Steckler room with volumetric heat source.* One of the cases chosen for Phase 2 was the Steckler room with volumetric heat source. The volumetric heat source was chosen as there was little difference between the results generated by the combustion model (Case 2/2) and the volumetric heat source model in Phase 1.

From the Phase 1 results it was apparent that all the models over-predicted the upper layer

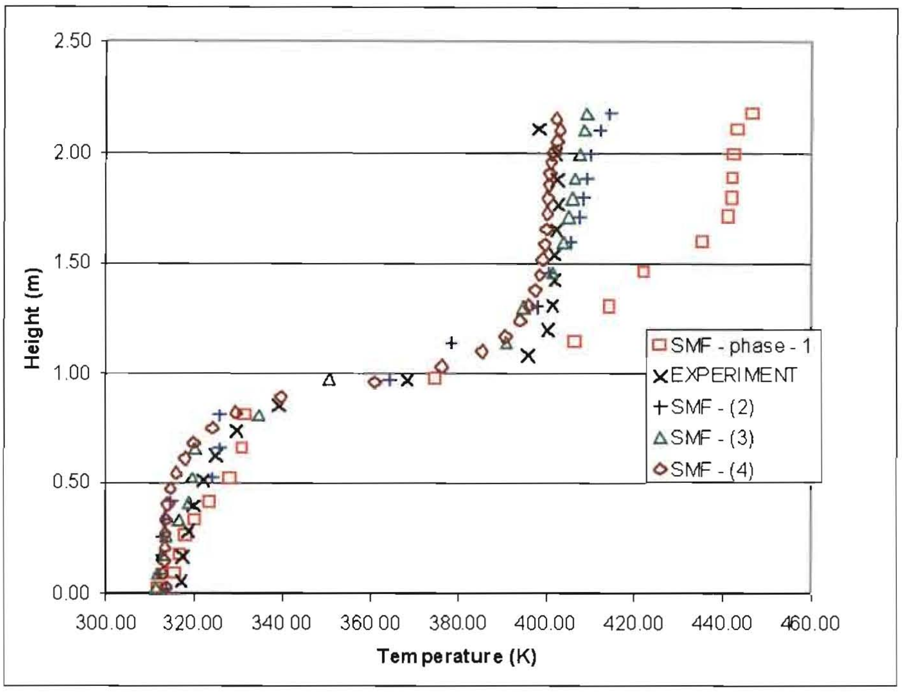



*Figure 9. Corner stack temperatures produced by the Phase 2 set-ups for Smartfire*

temperatures generated by the fire. This was expected as all the models assumed that the walls were adiabatic and perfect radiative reflectors. In Phase 2 of the validation process, three further simulations were performed with Smartfire for this test case with modified case specifications as follows:

- SMF (2) – as for Phase 1 but with improved modelling of compartment boundary conditions: heat transmission at the walls was modelled using the correct thermophysical properties of the wall material, which was common brick.
- SMF (3) – as SMF (2) but with the multi-ray radiation model with 24 rays replacing the basic six-flux radiation model[3].
- SMF (4) – as SMF (2) but with refined grid and taking advantage of symmetry. In Phase 2 a total of 49,980 cells were used to model half the domain and symmetry was used to model the other half. The total number of cells was therefore 99,960 compared to 13,020 in Phase 1.

An example of the results of these simulations is shown in Figure 9 along with the results for Phase 1. As can be seen, all three simulations produce a much better prediction of the temperatures of the corner stack than the original Phase 1 predictions. Improving the physical properties and the wall boundary conditions produce the most significant improvement in the results. This brought the upper layer temperatures very close to the observed values. On the other hand, the use of the multi-ray radiation model and the refined grid have not significantly improved the accuracy of the predictions.

3 When modelling compartment fires, a radiation model is usually required to account for the effect of thermal radiation around the compartment. CFD models are not ideally suited to modelling radiation as it is not a fluid flow but a direct transmission across space. In the six-flux model, radiation is propagated out of a cell in each of the six directions formed by the perpendicular to each of the six faces of the cell. Radiation along any other directions, e.g. diagonals, is ignored. This accounts for a general propagation around the compartment but does not give accurate prediction of radiation levels at specific points. The more sophisticated 24-ray radiation model is likely to give a more accurate result because of the greater number of rays used, but requires more computer resources because of the greater number of calculations.

## CONCLUSIONS

From the Phase 1 test cases, it is clear that when identical physical models are activated, identical computational grids used and similar convergence criteria applied, all of the software products tested are capable of generating similar results. This is an important observation and suggests that – within the limitations of the tests undertaken – the three models tested are capable of achieving similar results and similar levels of accuracy on a set of standard cases.

The one area that showed relatively poor agreement with theoretical results concerned the radiation model performance. The six-flux radiation model while capable of representing the average trends within the compartment, does not produce an accurate representation of local conditions. When using the six-flux model, it is possible that target fuel surfaces would not be preheated by radiation to the extent that would occur in reality, thereby slowing the flame spread process.

Phase 2 simulations were only performed with the Smartfire model. Results from these simulations showed how improvements in the accuracy of the results could be obtained by using for instance a multi-ray radiation model instead of a six-flux model and a more realistic treatment of the wall boundary conditions.

In summary it is concluded that the concept and testing protocols developed as part of this project have been shown to be a valuable tool in providing a verifiable method of benchmarking both the basic and advanced capabilities of CFD-based fire models on a level playing field.

## RECOMMENDATIONS

It is recommended that the principles and procedures developed in this project be adopted in some form as a quality measure of fire modelling software.

Putting a model through the test regime described in this report involves a significant amount of work and every new release of a model should ideally be tested separately. One possibility, suggested by the University of Greenwich, might be for a test organisation to be made responsible for testing all models but this would be a very large amount of work. To reduce the burden on a testing organisation, it is recommended that the software developers should be required to run all the test cases and submit details to the test organisation and that the test organisation then only need perform random testing on some of their results as a check.

To further improve the capabilities of the approach, it is recommended that additional test cases are developed and several of the fire cases are refined.

Finally in addition to the accuracy of the model, which is the subject of this project, the importance of the knowledge and skill of the model user should be emphasised. The ODPM are currently looking at this issue.

## REFERENCES

1. Grandison, A. J., Galea, E.R. and Patel, M.K., *Development of Standards for Fire Field Models – report on Phase 1 Simulations*, University Of Greenwich report for Fire Research Division, Department for Transport, Local Government and the Regions, 2/2002.

2. Grandison, A. J., Galea, E.R. and Patel, M.K., *Development of Standards for Fire Field Models – report on Smartfire Phase 2 Simulations*, University Of Greenwich report for Fire Research Division, Department for Transport, Local Government and the Regions, 3/2002.

3. Steckler, K.D, Quintiere, J.G and Rinkinen, W.J., *Flow induced by fire in a compartment*, NBSIR 82-2520, National Bureau of Standards, 1982.

Computer models of fire growth are useful tools for assessing the fire safety of building designs and the results of these models are sometimes included in submissions of building proposals to fire brigades. This report describes a project to develop the basis for a standard for fire models. To meet the standard a fire model would need to achieve a sufficient level of accuracy in a set of defined cases.

£5.00

ISBN 1 85112 679 1

ISBN 1-851126-79-1

9 781851 126798