

## Computational Fluid Dynamics Modelling of a Fire Resistance Furnace

N. Bressloff<sup>Ⓛ</sup>, P. Rubini<sup>Ⓛ</sup>, G. Cox<sup>Ⓜ</sup>

<sup>Ⓛ</sup> School of Mechanical Engineering, Cranfield University, Cranfield, UK

<sup>Ⓜ</sup> Fire Research Station, Garston, Watford, UK

### ABSTRACT

The fire resistance "rating" of a building component or element is determined by its performance in a standard furnace test. For these "ratings" to be acceptable across the European Union it is important that specimens be subject to the same standard test wherever it is conducted. Unfortunately, since existing methods only standardise on a furnace thermocouple temperature / time curve, substantial differences can occur in measured ratings when tested in different member states. This is because there are substantial differences in the design of standard furnaces used across Europe. They differ in size, geometry, number and position of burners, fuel type, lining and flue gas extraction. As a consequence it is most probable that different furnaces expose the test specimen to different thermal conditions even though the same standardised time / temperature control curve is employed.

Computational Fluid Dynamics (CFD) is now routinely used for fire science modelling, see for example [1]. Often referred to as *field modelling*, typical applications include smoke movement and heat transfer, in circumstances where traditional zone models are inappropriate.

The current paper presents the results from an investigation into the thermal characteristics of a fire resistance furnace using the techniques of Computational Fluid Dynamics. The furnace investigated, the BRE *One Metre<sup>3</sup> furnace*, is a small scale test furnace and as such is very suitable for the initial validation and proof of concept for the application of CFD to full scale fire resistance furnaces.

The numerical predictions were carried out using **SOFIE**, (Simulation of Fires in Enclosures), a CFD code written at Cranfield University with support from a number of European fire research laboratories, including the Fire Research Station (England), SP Boras (Sweden), University of Lund (Sweden), VTT (Finland), CSTB (France), **ITSEMAP** (Spain), [2].

SOFIE employs a finite volume procedure to solve the governing **Navier-Stokes** equations in a general curvilinear coordinate system, for more details see for example [3]. The standard **k-ε** turbulence model is employed with buoyancy modifications [4]. Combustion is accounted for by assuming that the rate of heat release is limited by turbulent mixing of the fuel and oxidant, as modelled by the **Magnussen** combustion model [5]. Radiation heat transfer is modelling using the ray tracing based approach, discrete transfer [6]. Heat transfer with the combustion products and within the solid walls of the furnace and test specimen is accounted for simultaneously.

The BRE One Metre<sup>3</sup> fire resistance furnace is constructed from concrete walls, approximately 10cm thick. The test specimen, a 10mm thick steel plate, represents one whole wall, for the tests under consideration this was the top of the furnace. The furnace is fired by two parallel gas burners, symmetrically placed about the centre of the furnace on one wall. The fuel was assumed to be pure methane.

The furnace was modelled using a computational grid of 17x27x11 nodes, of which 5 nodes were placed across the thickness of the test specimen. The furnace was assumed to be symmetrical about a line midway between the burners. The furnace was simulated for the equivalent of a 40 minute run time with a numerical time step equal to 15 seconds. The calculations were run on a DEC 3000/600 workstation. The time / temperature curve followed during the experimental trials was simulated by varying the fuel flow rate of the burners.

The numerical predictions displayed good agreement with the limited experimental data available. The predicted gas temperature, at locations close to the test specimen, closely followed a normalised time/temperature curve. However the most significant feature of the numerical simulations is that a considerable variation in surface, and local gas, temperature was predicted over the test specimen. This is of paramount importance when assessing the relative performance of full scale fire resistance furnaces. The simulation clearly indicates that although a time / temperature curve may be followed, the actual test specimen is exposed to a varying heat flux across its surface.

Although detailed improvements to the simulation can obviously be incorporated, for example improved grid resolution, the results illustrate the potential of Computational Fluid Dynamics for assessing the thermal performance of fire resistance furnaces. The capability to model transient phenomena over a significant period of time has been demonstrated within a fully three dimensional problem, taking into account both the fluid dynamics and coupled radiation/convection/conduction heat transfer.

## References

- 1) Cox, G. "Compartment Fire Modelling", in Combustion Fundamentals of Fire, ed. Cox G. Academic Press, 1995.
- 2) Cox G., Rubini P. "Development of a new fire simulation model". Nordic Fire Safety Engineering Symposium, Espoo, Finland, 1993.
- 3) Melaaen M. C. "Analysis of curvilinear non-orthogonal coordinates for numerical calculation of fluid flow in complex geometries". Dr. Ing. thesis, University of Trondheim, 1990.
- 4) Rodi W. "Turbulence models and their applications in hydraulics - a state of the art review". IAHR Monograph, 1993.
- 5) Magnussen B. F., Hjertager B. H. "On mathematical modelling of turbulent combustion with special emphasis on soot formation and combustion". Proc. Sixteenth Symposium (International) on Combustion. The Combustion Institute. 1976.
- 6) Lockwood F. C., Shah N. G. Proc. Eighteenth Symposium (International) on Combustion, 1981, pp.1405-1414.

• **DEVELOPMENT OF A NEW CFD FIRE SIMULATION MODEL**

G Cox  
Fire Research Station  
Borehamwood  
WD6 2BL  
UK

and

P A Rubini  
Cranfield Institute of Technology  
Cranfield  
MK43 0AL  
UK

SUMMARY

The paper provides a brief outline of **SOFIE**, a new computational fluid dynamics fire simulation model, being developed by the Fire Research Station, Cranfield Institute of Technology, SP Boras, the University of Lund and **VTT**. The need for the development of a new model is explained and its specification given. Some preliminary testing work with the base code is presented.

INTRODUCTION

Computational fluid dynamics has made an increasingly significant contribution to many branches of engineering since its emergence in the mid 1970's as a practical design and analysis tool. Examples of its application can be found in many areas of endeavour ranging from **airframe**, ship hull and car body design through to analyses of the efficiency of gas turbines, cement kilns and glass furnaces.

By solving, numerically, the full partial differential equation set describing the principles of local conservation of mass, momentum, energy and species, subject to the particular boundary conditions of the problem, models of this type have enjoyed widespread development in many combustion applications. Until comparatively recently, however, their use in fire research has been limited, tending to concentrate more on validation of the methodology than on its development and application.

A growing confidence in predicting far-field conditions gained from these studies, in addition to the availability of increasing computer power at reducing cost is encouraging greater interest in their practical application to the assessment and design of smoke control systems in buildings. The safety cases for the designs utilised within the Channel Tunnel for example, have benefited from application of this new technology. With an increasing awareness of the potential of CFD for the design environment a synergism between it and Computer Aided Design is also growing.

The assessment of smoke movement problems represents, however, just one of two distinct branches of active development in what has become known in fire science as field modelling. The detailed modelling of turbulent buoyant diffusion flames, particularly in under-ventilated conditions, and the full coupling of the thermal radiation field to the rate of gasification of solid fuels also remain important areas of active research and development.

For the treatment of these and the other phenomena peculiar to fire, effort has primarily been concentrated on development not of the details of the enabling technology of the core 'Navier Stokes' equation solver but on what might be termed the fire science sub-models. It is these which ensure an appropriate treatment for the phenomena of buoyancy driven recirculating flow, turbulent combustion, thermal radiation and boundary effects that are unique to enclosure fire problems.

The core numerical solver has often of necessity been untouched because it has remained the property of a CFD software vendor who has jealously guarded access to its contents for good (to him) commercial reasons. This had been the case until recently with the FRS model known as JASMINE[1] which exploits an early version of the commercial PHOENICS package[2] at its heart. Only after many years of discussion and negotiation have FRS acquired the source code of the PHOENICS core solver for its own use.

Although the JASMINE model has been subjected to many years of development and validation, FRS has had to use alternative software when it has undertaken investigations of for example, the accuracy of various numerical algorithms or of the potential of parallel processing.

Investigations of various numerical differencing schemes [3] for the treatment of the convection ~~terms~~ and of **MIMD** (multiple instruction multiple data) parallel processing had of necessity to be undertaken with alternative '**open-source**' programs (the earlier two **dimensional**, steady-state fire model **MOSIE** [4] or the TEACH derivative **CINA** [5]) with the results of this work extremely difficult to implement in the preferred **tried-and-tested** JASMINE model.

Although the details of the numerical solver may be of little interest to the practitioner - after all which of us concerns ourselves with the details of the FORTRAN compiler - not to have such access cannot be satisfactory for those laboratories responsible for providing advice to regulatory authorities on safety matters. Based on experience of these problems with JASMINE and its forerunner MOSIE, FRS decided, in addition to continuing development of JASMINE for application purposes, to embark, along with others, upon the development of a new '**open source**' CFD model. This would subsume the current fire science sub-models and validation experience of JASMINE and would employ the latest techniques and developments in numerical methods. It has joined together with the **Cranfield** Institute of Technology, Lund University, SP Boras and **VTT** to develop the **SOFIE** (Simulation Of Fire In Enclosures) model based upon a completely new '**Navier Stokes**' solver.

## **SPECIFICATION**

Although most building fire problems can be considered to occur within essentially '**Cartesian**' structures, there are problems that can benefit from the use of a more general coordinate system. Particularly with the expectation of incorporating adaptive grids which can move with a spreading flame, it was decided to develop the new model using a general curvilinear coordinate system. The velocity vectors are represented, see figure 1, in terms of Cartesian base vector components (u,v,w) within the general curvilinear coordinate system ( $\xi, \eta, \zeta$ .)

Furthermore all variables are evaluated and stored at the same locations on the numerical grid. There is no '**\*staggering\***' of the grid between velocity and scalar variables as is the practice in the JASMINE model. Although grid staggering is particularly convenient for

eliminating problems associated with determining the pressure field when exploiting Cartesian grids, it creates many difficulties when used with a curvilinear grid. Instead the method developed by Rhie and **Chow**[6] which interpolates pressure and momentum fields between grid nodes is used to evaluate mass fluxes across control volume faces.

Not only does the collocation of velocity and scalar variables offer advantages for the use of curvilinear coordinates it also permits ease of application of numerical acceleration techniques such as the **multigrid method**[7].

The form of the transport equation for the general variable  $\phi$ , familiar in its Cartesian form,

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial(\rho u_i \phi)}{\partial x_j} - \frac{\partial}{\partial x_j} \left[ \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right] = S_\phi \quad (1)$$

now becomes,

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial(\rho U_j \phi)}{\partial \xi_j} - \frac{\partial}{\partial \xi_j} \left[ J \Gamma_\phi g^{jk} \frac{\partial \phi}{\partial \xi_k} \right] = JS_\phi \quad (2)$$

where  $\phi$  takes on the same meaning in both cases ( $1, h, m, k, \epsilon$ ) and the momentum equations are expressed in terms of the Cartesian velocity components  $u, v, w$ .  $\Gamma_\phi$  and  $S_\phi$  are exchange coefficient and source terms for the variable  $\phi$ . However, very importantly in the curvilinear treatment, the fluxes are calculated by using the **contravariant** velocity components  $U, V, W$ . These are simply related to the velocity components normal to each face of the curvilinear control volume.  $J$  and  $g^{jk}$  are geometrical properties.  $J$  is the Jacobian of the transformation matrix relating the Cartesian to the curvilinear coordinate system. It expresses the ratio of control volumes expressed in either coordinate system.

$$g^{ik} = \frac{\partial \xi_j}{\partial x_i} \frac{\partial \xi_k}{\partial x_j}$$

With these decisions made for the basic form of the new model, the specification for the core solver is:

- (i) to incorporate various numerical differencing schemes (upwind, hybrid, power **law**, second order **upwind**, **QUICK**[8] and a **TVD** [9] scheme) to continue to improve accuracy and preserve stability of the solution procedure. Each of these schemes, with the exception of the total variation diminishing, TVD scheme had been the subject of the earlier study with **MOSIE** [3].
- (ii) to allow for embedded grids to resolve smaller scale physical features within a larger scale solution eg. resolving a small fire source within a large enclosure
- (iii) to write the code in modular form capable of exploiting parallel processors.
- (iv) to exploit **state-of-art** numerical methods to ensure rapid solution convergence (eg **multigrid** acceleration and conjugate gradient **methods**[10]).

In addition to this new '**core**' solver the specification requires for the existing fire science sub-models currently implemented within JASMINE to be incorporated including,

- (v) advanced models of turbulent combustion - eg laminar **flamelet models**[11]
- (vi) advanced thermal radiation models - eg discrete transfer **models**[12].

and also

- (vii) liquid droplet dispersion to allow study of extinguishing sprays

## **DEVELOPMENT**

The development of **SOFIE** is in its early stages. The base code has been written and

tested against a number of standard cases: flow over a backward-facing step, within a driven cavity, within a buoyant cavity and finally within an unbounded buoyant turbulent diffusion flame.

The results of these are promising. Some results for the fire simulation are illustrated in figure 2. Here only very simple fire science **sub-modelling** has been used (no thermal radiation model, but a simple eddy-break-up treatment for combustion) within **SOFIE** and **predictions** compared with experimental data and with predictions from some early simulations of this test case made using the parabolic **GENMIX** code.

The buoyant fire simulation analysed by **Crauford** [13] has been modelled using a right angled segment of the **axisymmetric** flow field with control volume coordinates generated by cylindrical **polars**. The boundary conditions used were those of measured mean temperature, velocity, fuel mass fraction and turbulent kinetic energy at 0.14 m above the burner surface as used in the GENMIX simulation.

The results of a 42 x 42 x 5 grid simulation using SOFIE are shown. Both GENMIX and SOFIE simulations greatly **overpredict** mean gas temperatures just downstream of the source but further downstream the two computer simulations span the experimental measurements. Clearly the absence of a radiation heat exchange model in SOFIE helps explain its **overprediction** a long way downstream of the source but the **overprediction** demonstrated by both computer simulations immediately downstream of the source is more likely to be due to a poor prediction of turbulent mixing in this critical region.

At this stage of model development this level of agreement can be regarded as reasonably acceptable but it is expected that improvement will result from incorporation of improved fire science sub-models (see, for example reference 14).

Another aspect of this early development phase has been the production of a user-friendly front end. Based on the public domain, tool command language, TCL [15], a front end has been erected which allows the user to specify his problem simply by clicking on appropriate boxes from a variety of menus. Currently these boxes include choices of boundaries and blockages, grid specification, boundary conditions and choice of type of

calculation required eg laminar, turbulent, combustion model etc.

## CONCLUSIONS

This paper has attempted to summarise the needs for and current progress in the development of **SOFIE**, a new CFD model for the prediction of **fire** behaviour. A brief outline is given of the basis for the model together with an example of a test against fire data of the base code.

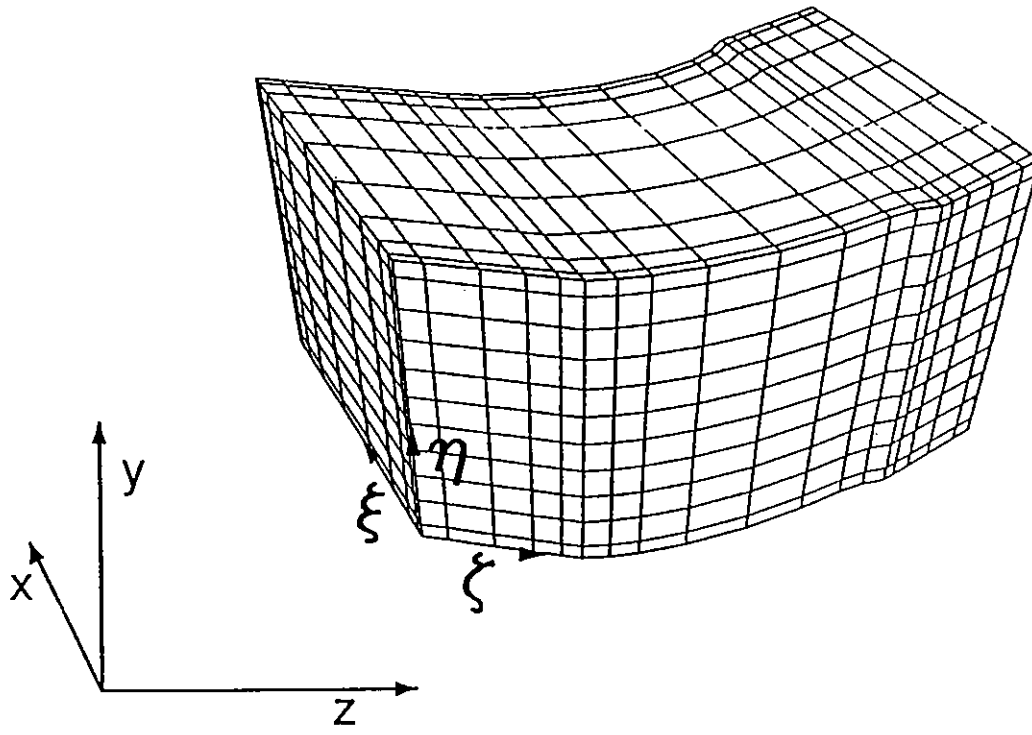
It is intended that the model currently being developed jointly by FRS, **Cranfield**, Lund, SP Boras and VTT will be developed further, first to subsume and then to add to the **fire** science sub-model currently incorporated within JASMINE.

## REFERENCES

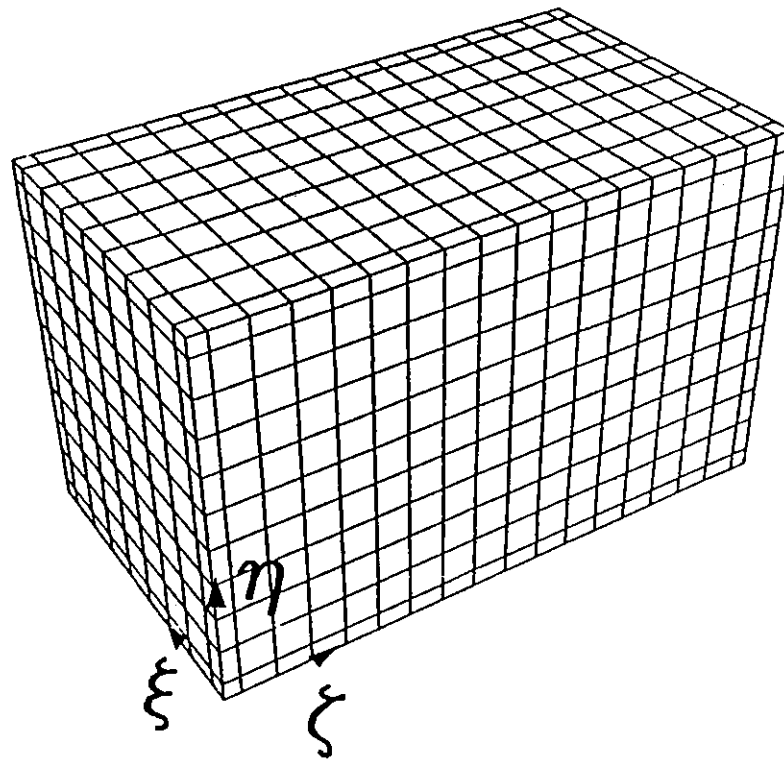
1. Cox G and **Kumar S**. Field Modelling of Fire in Forced Ventilated Enclosures, *Combustion Science and Technology* **5**, 7 (1987).
2. Spalding D B. A General Purpose Computer Program for Multi-Dimensional One and Two Phase Flow, *Mathematics and Computers in Simulation*, North Holland Press, **Vol XXIII**, pp 267-276 (1981).
3. **Patel M K**. On the False Diffusion Problem in the Numerical Modelling of Convection-Diffusion Processes, PhD. Thesis, Thames Polytechnic, 1986.
4. **Markatos**, N C, **Malin M R** and Cox, G. Mathematical Modelling of Buoyancy Induced Smoke Flow in Enclosures. *Int. J. Heat Mass Transfer* **25**, 63 (1982).
5. Cox G, **Cumber P**, **Lockwood F C**, **Papadropoulos C** and **Taylor K**. On the Field Modelling of Fire using Parallel Processors, P 544, *Heat Transfer in Radiating and Combusting Systems* (eds **Carvalho M G**, **Lockwood F C** and **Taine J**) **Springer-Verlag**, 1991.

6. Rhie, C M and Chow W L. A Numerical Study of the Turbulent Flow Past an Isolated Aerofoil with Trailing Edge Separation, **AIAAJ** 21, 1525, (1983).
7. Rubini P A, Becker H A, **Grandmaison** E W, Pollard A, **Sobiesiak** A and **Thurgood** C. **Multigrid** Acceleration of Three Dimensional **Turbulent**, Variable Density Flow, Numerical Heat Transfer, part B, 22. 163, (1992).
8. Leonard B P. A Stable and Accurate **Convective** Modelling Procedure Based on Quadratic Upstream Interpolation, Computer Methods in Applied **Mechs.** and Engineering, 19, 59, (1979).
9. Van Leer B. Towards the Ultimate Conservative Difference Scheme III. Upstream Centred Finite Difference Schemes for Ideal Compressible **Flow**, **Jnl Computational Physics** 23, 262 (1977).
10. Noll B and **Wittig** S. Generalised Conjugate Gradient method for the Efficient Solution of Three Dimensional Fluid Flow Problems, Numerical Heat Transfer, part **B**, 20, 207, (1991).
11. **Askari-Sardhai** A. Liew S **K** and Moss J B. Bamelet Modelling of Propane-Air Chemistry in Turbulent Non premixed Combustion, Comb. Sci. Tech 44. 89, (1985).
12. Lockwood F C and Shah N G. A New Radiation Solution Method for Incorporation in General Combustion Prediction Procedures, Eighteenth Symposium (International) on Combustion, the Combustion Institute, p 1405, (1981).
13. <sup>u</sup>**Cranford** N L, Liew S K, and Moss J B, Experimental and Numerical Simulation of a Buoyant Fire, Comb. Flame, 61, 63, 1985.
14. Syed K J. Soot and Radiation Modelling in Buoyant Fires, PhD, Thesis, **Cranfield** Institute of Technology 1990.

15. **Ousterlout J K.** An Introduction to TCL and **TK**, To be published by Addison-Wesley, 1993.

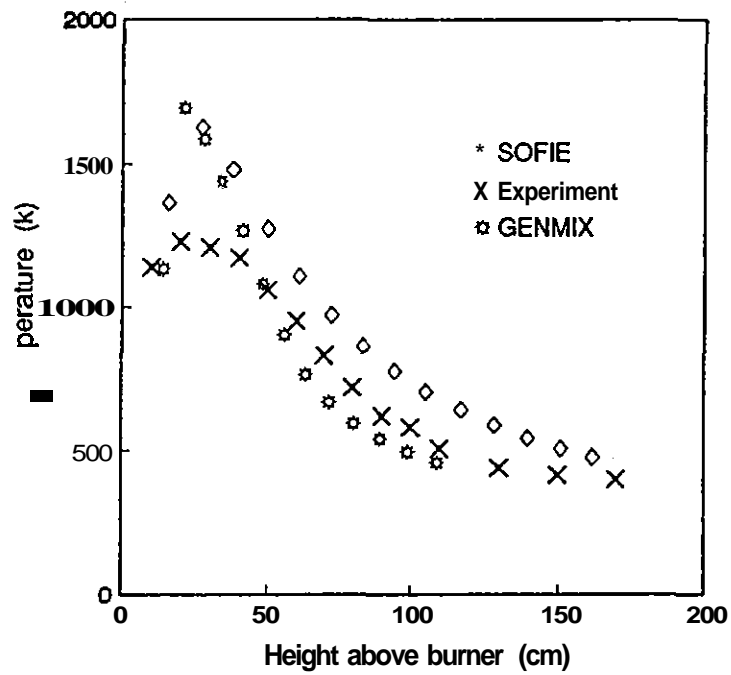


(a) Curvilinear grid in Physical Space



(b) Grid transformed into Cartesian Space

**Figure 1** Computational Grid



**Figure 2** Variation of axial temperature with height above burner