

# Computer Models For Fire and Smoke

<i>Model Name:</i>	SMARTFIRE
<i>Version:</i>	2.01 (Next release, version 3.0 due second half 2000)
<i>Classification:</i>	Fire Field Model using CFD techniques.
<i>Very Short Description:</i>	A PC based fire field model with auto mesh generation and interactive graphical interface providing dynamic control of the CFD software. Capable of simulating fires within multiple compartment geometries.
<i>Modeler(s), Organization(s):</i>	SMARTFIRE development Team, FSEG, The University of Greenwich, key members consist of Dr John Ewer, Prof Ed Galea, Dr Mayur Patel, Dr Fuchen Jia, Mr Angus Grandison.
<i>User's Guide:</i>	SMARTFIRE V2.0 User Guide and Technical Manual, Doc Rev 1.0, July 1998.
<i>Technical References:</i>	SMARTFIRE User Guide
<i>Validation References:</i>	<p>“SMARTFIRE Verification and Validation Report”, Report Version 1.01, Report Revision Date 23/08/99.</p> <p>“SMARTFIRE KBS Grid Generator Verification Report”, Report Version 1.2, Report Revision Date 07/06/99.</p> <p>“Simulating one of the CIB W14 Round Robin Test cases using the SMARTFIRE fire field model.” Wang, Jia, Galea, Patel and Ewer. To Appear 2000, Fire Safety Journal.</p> <p>“SMARTFIRE: An Intelligent CFD Based Fire Model”, J Ewer, E R Galea, M K Patel, S Taylor, B Knight, M Petridis, Journal of Fire Protection Engineering, Vol 10 (1), 1999, pp. 13-27</p>

“The Development and Application of Group Solvers in the SMARTFIRE Fire Field Model”, J Ewer, E Galea, M Patel and B Knight, Proceedings of the 8th International Fire Science and Engineering Conference: Interflam'99, Vol. 2, pp 939-951, Edinburgh, Scotland, June 29 – July 1st 1999, published by Interscience Communications Ltd, London, UK, 1999. ISBN 0 9532312 3 2.

“SMARTFIRE: An Integrated Computational Fluid Dynamics Code and Expert System for Fire Field Modelling”. S Taylor, E Galea, M Patel, M Petridis, B Knight, J Ewer. Proceedings of the 5th International Symposium IAFSS, Melbourne, Australia 1997, Ed: Y. Hasemi, pp. 1285-1296.

“SMARTFIRE: An Intelligent Fire Field Model”. S Taylor, E R Galea, M K Patel, M Petridis, B Knight, J Ewer. Proceedings of the 7th International Fire Science and Engineering Conference: Interflam'96, pp 671-680, St John's College Cambridge, England, March 1996, compiled by C Franks and S Grayson, Published by Interscience Communications Ltd, London, UK, 1996. ISBN 0 9516320 9 4.

*Availability:* Contact Professor E. Galea, FSEG, The University of Greenwich, 35 Park Row, Greenwich, UK SE10 9LS, [e.r.galea@gre.ac.uk](mailto:e.r.galea@gre.ac.uk), <http://fseg.gre.ac.uk/smartfire/>, phone: +44 (0)20-8331-8730.

*Price:* UK£3000 annual commercial license, discounts available for education, fire brigades, local authorities, etc.

*Necessary Hardware:* Intel architecture running Windows 95, 98 or NT 4.0. Not yet validated under Windows 2000. Workstations using Motif supported by request.

*Computer Language:* C++

*Size:* Approximately 40MB of disk space and a minimum of 32MB of free memory.

*Contact Information:* Dr John Ewer FSEG, The University of Greenwich, 35 Park Row, Greenwich, UK SE10 9LS, [smartfire@gre.ac.uk](mailto:smartfire@gre.ac.uk) +44 (0)20-8331-8658.

*Detailed Description:*

SMARTFIRE is an open architecture CFD environment with an integrated knowledge based system that attempts to make fire field modelling accessible to non-experts in CFD. There are three major components to the software: CFD code, User Interfaces, and Expert System. By embedding expert knowledge into the CFD software, it is hoped that fire field modelling is made more accessible to the fire engineer with limited CFD expertise. The expertise currently embedded within the code is used to support the critical task of mesh specification of fire field simulation scenarios. More information about SMARTFIRE can be found at <http://fseg.gre.ac.uk/smartfire/>.

### **1.0 The CFD engine**

The CFD engine of SMARTFIRE is written in C++ and has been developed from an existing in-house FORTRAN code. It uses validated numerical methods from the legacy FORTRAN code enhanced by object oriented developments in C++ with additional physics features that are required for fire field modelling. These include the addition of a six-flux radiation model, provision for heat transfer through walls and buoyancy modifications to the turbulence equations. It uses three-dimensional unstructured meshes, enabling complex irregular geometries to be meshed without the need for body fitted coordinate grids. The current release of the software is V2.01. In this version, the software represents fires as a transient volumetric heat and mass source. Standard models for gaseous combustion (i.e. diffusion and eddy-dissipation models, while not enabled in the current general release software, are implemented within V3.0  $\beta$  and undergoing final testing. The code uses the SIMPLE pressure-correction algorithm and can solve turbulent (two equation K-Epsilon closure with buoyancy modification) flow problems under transient or steady state conditions. Radiation is represented through the use of an enhanced six-flux radiation model.

### **2.0 The Main User Interface**

The main user interface (UI) is used to specify the problem. Through the UI the user sets the geometry, specifying the location of walls, wall materials, internal compartments, obstacles, the location of vents, specifies the fire, activates the radiation model, specifies gaseous properties such as absorption coefficients, meshes the geometry and starts the CFD engine. In specifying the geometry, the user specifies the compartment(s) dimensions. On each wall of the compartment it is possible to position vents. Here, the generic term vent is used to represent any opening to the outside such as a door or window. The software does not currently include the provision of forced vents. Each vent's size and location is also specified. The user must also specify the wall material. The material type is specified using user familiar terms such as 'brick' or 'insulator', and the conversion to material properties and boundary conditions is performed internally. The fire is specified as a volumetric heat source. The user must specify the physical size of the fire, its location and heat output. The fire can be specified either as a constant heat release rate or as a transient heat release rate. If the transient option is selected, the user is presented with a menu, which allows the specification of a range of functional forms. The heat release rate can also be specified through an input table.

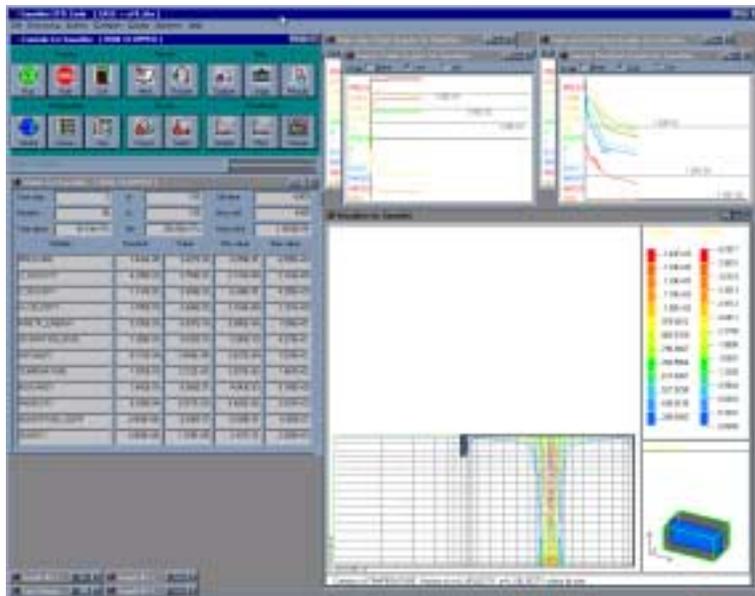
### **3.0 The Mesh Generator**

Once the problem has been specified, the mesh generation expert system is launched from the main user interface to generate the CFD grid for the problem. It is important here to note that the intelligent mesh generator component of SMARTFIRE will determine a cell budget appropriate for a *reasonable* solution to the problem. This will be

reported back to the user who is given the opportunity to accept the recommended cell budget or modify the cell budget. Once the mesh is generated the CFD engine can be launched.

#### 4.0 The CFD User Interface

The CFD code has its own unique windows based user interface depicted in figure 1. Unlike traditional fire field models, this allows the user to interact with the solution through *observation* of the developing solution and by allowing the user to make *adjustments* to control parameters while the code is in operation. Adjustment such as this in traditional CFD codes involves stopping the simulation, editing input files (possibly searching through large files containing nothing but floating point numbers), and restarting the simulation. With SMARTFIRE, this form of dynamic user control is considerably easier. All that is required is point and click of various buttons on the user interface.



**FIGURE 1: SMARTFIRE CFD Engine User Interface**

#### 5.0 SMARTIFRE V3.0 FEATURES

Version 3.0 of SMARTFIRE is planned for release in the second half of 2000. This will include a range of new modelling and user interface features. Some of the additional features include:

- Standard gaseous combustion models for diffusion controlled combustion and eddy dissipation rate combustion
- Horizontal vents
- Forced ventilation
- Manual meshing tool to allow expert users to augment the KBS meshing