

This document has been superseded
by a later version. For the latest
version go to the web site:

<http://fire.nist.gov/fds>

NISTIR 6761

User's Guide for Smokeview Version 2.0 – A Tool for Visualizing Fire Dynamics Simulation Data

Glenn P. Forney
Kevin B. McGrattan



NIST
National Institute of Standards and Technology
Technology Administration, U.S. Department of Commerce

/

NISTIR 67611

User's Guide for Smokeview Version 2.0 – A Tool for Visualizing Fire Dynamics Simulation Data

Glenn P. Forney
Kevin B. McGrattan
Building and Fire Research Laboratory

July 2001



U.S. Department of Commerce
Donald L. Evans, Secretary

National Institute of Standards and Technology
Karen H. Brown, Acting Director

Disclaimer

The US Department of Commerce makes no warranty, expressed or implied, to users of Smokeview, and accepts no responsibility for its use. Users of Smokeview assume sole responsibility under Federal law for determining the appropriateness of its use in any particular application; for any conclusions drawn from the results of its use; and for any actions taken or not taken as a result of analysis performed using this tools.

Users are warned that Smokeview and the companion program FDS is intended for use only by those competent in the fields of fluid dynamics, thermodynamics, combustion, and heat transfer, and is intended only to supplement the informed judgment of the qualified user. These software packages may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions could lead to erroneous conclusions with regard to fire safety. All results should be evaluated by an informed user.

Throughout this document, the mention of computer hardware or commercial software does not constitute endorsement by NIST, nor does it indicate that the products are necessarily those best suited for the intended purpose.

Contents

| | | |
|----------|---|-----------|
| 1 | Introduction | 1 |
| 1.1 | Overview | 1 |
| 1.2 | What's New in Version 2.0 | 3 |
| 2 | Getting Started | 4 |
| 3 | Using Smokeview | 5 |
| 3.1 | Data Files | 5 |
| 3.1.1 | Particle | 5 |
| 3.1.2 | Slice | 6 |
| 3.1.3 | Boundary | 6 |
| 3.1.4 | Isosurface | 8 |
| 3.1.5 | Plot3D | 8 |
| 3.2 | Visualizing Output | 10 |
| 3.2.1 | Manipulating the Scene | 10 |
| 3.2.2 | Setting data bounds | 10 |
| 3.2.3 | Plot3D Viewing Options | 12 |
| 3.3 | Examining Input | 12 |
| 3.3.1 | Debugging | 14 |
| 3.3.2 | Editing Blockages | 14 |
| 4 | Summary | 16 |
| A | Smokeview Command Line Options | 18 |
| B | Menu Options | 18 |
| B.1 | Main Menu Items | 19 |
| B.2 | Load/Unload | 19 |
| B.3 | Show/Hide | 20 |
| B.3.1 | Geometry Options | 20 |
| B.3.2 | Animated Surface | 21 |
| B.3.3 | Particles | 21 |
| B.3.4 | Boundary | 21 |
| B.3.5 | Animated Vector Slice | 22 |
| B.3.6 | Animated Slice | 22 |
| B.3.7 | Plot3D 2D Contours | 22 |
| B.3.8 | Plot3D 3D Contours | 22 |
| B.3.9 | Heat detectors, Sprinklers, Thermocouples | 22 |
| B.3.10 | Labels | 22 |
| B.4 | Options | 23 |
| B.4.1 | Shades | 23 |
| B.4.2 | Units | 23 |
| B.4.3 | Rotation | 23 |
| B.4.4 | Max Frame Rate | 23 |
| B.4.5 | Render | 24 |
| B.4.6 | Viewpoint | 24 |
| B.4.7 | Font Size | 24 |

| | | |
|----------|--|-----------|
| B.4.8 | Aperture | 24 |
| B.4.9 | Stereo 3-D | 24 |
| B.4.10 | Set Bounds | 24 |
| B.4.11 | Movement | 24 |
| B.4.12 | Edit Geometry | 25 |
| C | Keyboard Shortcuts | 25 |
| D | File Formats | 26 |
| D.1 | Smokeview Preference File Format (.ini files) | 26 |
| D.1.1 | Color parameters | 27 |
| D.1.2 | Size parameters | 28 |
| D.1.3 | Time and value bound parameters | 29 |
| D.1.4 | Data loading parameters | 31 |
| D.1.5 | Viewing parameters | 32 |
| D.2 | Smokeview Parameter Input File (.smv file) | 34 |
| D.2.1 | Geometry Keywords | 34 |
| D.2.2 | File Keywords | 36 |
| D.2.3 | Sensor Keywords | 37 |
| D.2.4 | Miscellaneous Keywords | 39 |
| D.3 | Data File Formats (.iso, .part, .sf, .bf and .q files) | 39 |
| D.3.1 | Isosurface File Format | 39 |
| D.3.2 | Particle File Format | 40 |
| D.3.3 | Slice File Format | 41 |
| D.3.4 | Boundary Files | 41 |
| D.3.5 | Plot3D Data | 42 |
| E | Sample FDS Input Files | 42 |
| E.1 | Plume Case | 42 |
| E.2 | Townhouse Case | 43 |
| F | Frequently Asked Questions | 45 |
| F.1 | How can I keep up with new releases and other information about FDS and Smokeview? | 45 |
| F.2 | Smokeview doesn't look right on my computer. What's wrong? | 45 |
| F.3 | How do I make a movie of a Smokeview animation? | 45 |
| | References | 46 |
| | Acknowledgements | 47 |

List of Figures

| | | |
|----|---|----|
| 1 | Diagram illustrating data files and programs used in the NIST Fire Dynamics System (FDS). | 2 |
| 2 | Particle file snapshots of an isolated plume at t=0.0, 1.0 and 2.0 seconds. | 7 |
| 3 | Slice file snapshots of shaded temperature contours in a vertical plane centered at the fire origin at t=30, 60 and 240 seconds after ignition. | 7 |
| 4 | Vector slice file snapshots of shaded vector plots in vertical plane centered at the fire origin at t=30, 60 and 240 seconds after ignition. | 7 |
| 5 | Boundary file snapshots of shaded wall temperatures at t=30, 60 and 240 seconds after ignition. | 9 |
| 6 | Isosurface file snapshots of mixture fraction levels at t=30, 60 and 240 seconds after ignition. | 9 |
| 7 | Plot3D contour and vector plot examples. | 11 |
| 8 | Plot3D isocontour example. | 11 |
| 9 | Motion Dialogue Box | 11 |
| 10 | Bounds Dialogue Box | 13 |
| 11 | Ceiling jet visualization | 13 |
| 12 | Blockage Edit Dialogue Box | 17 |
| 13 | Modifying the base FDS case in steps. | 17 |

List of Tables

| | | |
|---|---|---|
| 1 | Output Quantities used by Slice and Plot3D Visualizations | 6 |
| 2 | Output Quantities used by Boundary File Visualizations. | 8 |

User's Guide for Smokeview Version 2.0 - A Tool for Visualizing Fire Dynamics Simulation Data

Glenn P. Forney
Kevin B. McGrattan

Abstract

Smokeview is a software tool designed to visualize numerical predictions generated by the NIST Fire Dynamics Simulator (FDS), a computational fluid dynamics (CFD) model of fire-driven fluid flow. These two tools are typically used in concert to respectively simulate and visualize the flow of smoke induced by a fire. Smokeview visualizes FDS modelling results by animating tracer particle flow, 2D or 3D shaded contours of data such as temperature and flow vectors showing flow direction and magnitude. Smokeview also visualizes static data at particular times again using 2D or 3D contours of data such as temperature and flow vectors showing flow direction and magnitude. Examples of each visualization type are given.

Windows PC versions of Smokeview and FDS and their documentation may be obtained at the web site <http://fire.nist.gov/smokeview>. Versions for other computer platforms such as Redhat Linux 6.x/7.x, Silicon Graphics and some other UNIX systems are also available and may be obtained by contacting the authors at glenn.forney@nist.gov or kevin.mcgrattan@nist.gov.

1 Introduction

1.1 Overview

Smokeview is a software tool designed to visualize numerical predictions generated by the NIST Fire Dynamics Simulator (FDS), a computational fluid dynamics (CFD) model of fire-driven fluid flow. This report documents version 2.0 of Smokeview updating material found in Ref. [1]. FDS[2, 3, 4, 5] and Smokeview are used in concert to respectively model and visualize fire phenomena. Smokeview performs this visualization by presenting animated tracer particle flow, animated contour slices of computed gas variables and animated surface data. Smokeview also presents contours and vector plots of static data anywhere within a simulation scene at a fixed time. An analysis of the Cherry Road townhouse fire analyzed with FDS and Smokeview using some of these visualization techniques is given in Ref. [6].

New visualization features implemented in Smokeview version 2.0 are animated isosurfaces and animated flow vector fields. An isosurface animation of a flame sheet and a constant smoke density surface occurring in another townhouse kitchen fire may be viewed in the lower left corner by shuffling the odd pages of this report from back to front. A shaded temperature contour animation of an outdoor tank farm fire may be viewed in the lower right corner by shuffling the even pages of this report from front to back. The frame and time are indicated in each image. The input files used to generate the data for these animations are found in Appendix E.

Normally Smokeview is used in a post-processing step to visualize FDS data after a calculation has completed. Smokeview may also be used during a calculation to monitor a simulation's progress and before a calculation to visualize blockage, vent, sprinkler and/or heat detector placement *etc.* in order to set up correct

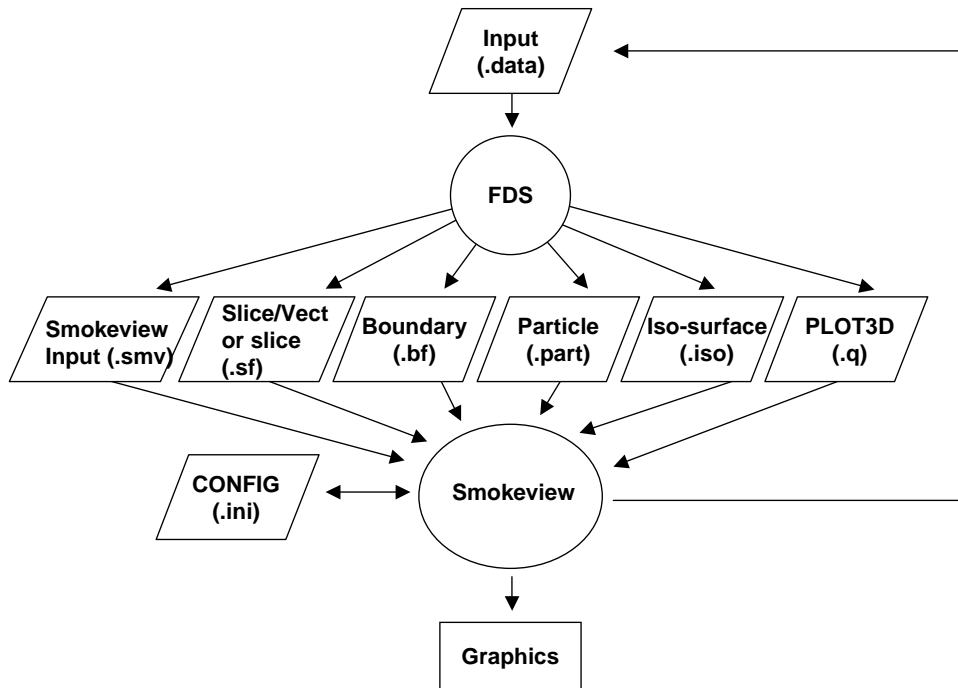


Figure 1: Diagram illustrating data files and programs used in the NIST Fire Dynamics System (FDS).

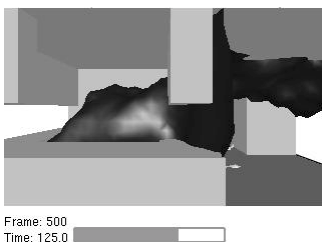
FDS input files more quickly. Smokeview version 2.0 allows one to create new blockages and modify existing ones.

Figure 1 gives an overview of how data files used by the NIST Fire Dynamics Simulator and Smokeview are related. A typical procedure for using FDS and Smokeview is to:

1. Set up an FDS input file, a file with a .data extension (e.g., a file named “casename.data”).
2. Run FDS. (e.g. by typing `fds < casename.data` at a command line). The fire model FDS then creates one or more output files with extensions .smv, .part, .sf, .q, .bf, .iso. These files are the Smokeview file, particle file, slice file, Plot3D file, boundary file and isosurface file, respectively.
3. Run Smokeview to analyze the output files generated by step 2. (e.g. by either double-clicking (on the PC) on the file named `casename.smv` or by typing `smokeview casename` at a command line.) Smokeview may also be used to create new blockages and modify existing ones. The blockage changes are saved in a new FDS input data file.

This report documents step 3. Steps 1 and 2 are documented in reference [4].

Data files may be visualized with Smokeview by clicking the right-mouse button and selecting the desired Load/Unload menu option. Other menu options are discussed in Appendix A. Many menu commands may also be executed using keyboard shortcuts. These shortcuts are listed in Smokeview’s help menu and are described in Appendix C. Visualization features not controllable through the menus may be changed by using the Smokeview customization file, `smokeview.ini`, discussed in Appendix D.1.



Smokeview is written using the 3D graphics library OpenGL[7, 8] and the Graphics Library Utility Toolkit (GLUT)[9]. Smokeview uses the GLUT software library so that most of the development work can be spent implementing the visualizations rather than creating an elaborate user interface.

1.2 What's New in Version 2.0

Smokeview version 1.0 is documented in Ref.[1]. The major new features in Smokeview version 2.0 are the two new ways of visualizing fire dynamics data: animated isosurfaces and animated flow vectors. These and other new features are detailed below.

Visualizations Two new methods for visualizing fire dynamics data have been added to Smokeview, animated isocontours and animated flow vectors. Isosurface animations may be used to represent flame boundaries, layer interfaces and/or regions in the simulation where sprinklers or heat detectors would have activated. Multiple isocontours may be stored in one file, allowing one to simultaneously visualize several levels for the same variable. Isosurfaces may also be generated for any gas variable such as temperature, density, heat release rate per unit volume (HRRPUV) *etc.* To reduce storage requirements, isosurface data stored in files with the extension .iso may be compressed using the standard compression program gzip (supplied with Smokeview).

Flow vector animations, though similar to slice file animations (the vector colors are the same as the corresponding slice file colors), are better than slice file animations at highlighting flow changes. Data needed to visualize animated flow vectors may be generated by selecting (in an FDS input file) U, V and/or W velocity slice files in addition to whatever other slice files are desired.

Menus The Smokeview menus have been reorganized to accommodate new features. Commands affecting the appearance of a visualization are now grouped under the `Option` Menu.

Dialogue boxes Several dialogue boxes have been added, allowing for easier Smokeview input. These are: an "open file" dialogue box for opening other Smokeview cases, a dialogue box for changing data bounds, a dialogue box for moving/rotating the scene and a dialogue box for creating and editing FDS blockages. These dialogue boxes may be accessed from the `Option` menu.

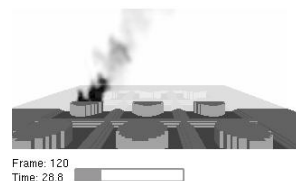
Colorbar/Timebar Several options have been added for altering the color and time bar in order to allow one to more easily identify features and behaviors found in the simulation data. One may now flip or reverse the order of colors in the colorbar. One may also click in the colorbar and slide the mouse to highlight data values in the scene. These options may be found under `Options/Shades` menu.

Similarly, the user may now click in the timebar and slide the mouse to change the simulation time displayed. One use for the timebar and colorbar selection modes might be to determine when smoke of a particular temperature enters a room.

A large font is now available to display text for both the colorbar and timebar.

Visualizing blockage and vent opening/closing New Smokeview keywords, `HIDE_OBST`, `OPEN_VENT` and `CLOSE_VENT`, have been added to the .smv file to allow Smokeview to visualize when a blockage should be hidden and when a vent should be opened or closed. These keywords are added to the .smv file automatically by FDS version 2.0. They may be manually added to .smv files generated with FDS versions 1.0 and 1.01.

New smokeview.ini keywords The viewpoint and orientation coordinates may be saved in the smokeview.ini file so that one may restore a scene to a desired location. Keywords for specifying data bounds have been expanded so that databounds may be specified independently for each variable



type (e.g. one may now specify min's and max's for oxygen, temperature, or any other quantity independently).

Units Smokeview and FDS use SI units by default. Alternate units may be displayed for temperature (K and F instead of C) and for velocity (f/s and mph instead of m/s) by using the Options/Units menu item. Smokeview may also be configured at run time to display other alternate units by using the UNITS keyword described in Appendix D.1.

Scene rendering Smokeview scenes are now rendered using version 2.0 of the GD graphics library [10, Appendix 4]. This library supports “full color” conversion allowing for more realistic scene representations. The former version of the library (used in Smokeview version 1.0) allowed only 256 distinct colors to be saved into an image file. This resulted in banding or contouring in the saved image where none was present in the original display. Images in Smokeview 2.0 are saved in either JPEG or PNG formats. GIF support was dropped in GD 2.0 and is therefore not available in Smokeview 2.0.

Scene clipping Complicated geometries such as those found in cases involving rack storage are difficult to visualize due to the number of obstructed surfaces. These interior obstructed surfaces may now be seen more easily by “clipping” part of the scene away.

Line drawing Line drawing quality has improved by using anti-aliasing (by drawing lines more smoothly).

Max Frame Rate A “real time” option has been added to the max frame rate menu, so that animations can be displayed in real time or multiples of real time if desired.

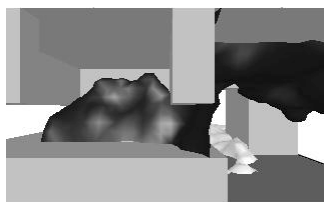
Stereo 3-D Smokeview can now produce stereo 3-D images by drawing a “left” and a “right” version of each frame displayed. This feature has been tested on Silicon Graphics workstations and PC's with video cards that support the use of quad buffering or equivalently stereo with OpenGL.

2 Getting Started

Smokeview may be obtained at the web site <http://fire.nist.gov/smokeview> . This site gives links to a Setup program for PC installation. It also contains documentation for Smokeview and FDS, sample FDS calculations, software updates and mechanisms for requesting feedback about the software.

After obtaining the setup program, install Smokeview on the PC by either entering the setup program name from the Windows Start/Run . . . menu or by double-clicking on the downloaded Smokeview setup program. The setup program then steps through the program installation. It copies the Smokeview executable, the GLUT graphics libraries and the Smokeview preference file `smokeview.ini` to the default directory `c:\nist\fds` . The setup program also defines PATH variables and associates the `.smv` file extension to the Smokeview program so that one may either type Smokeview at any command line prompt or double click on any `.smv` file. The installation program assumes that OpenGL has been installed which is true for all but the very first release of Windows 95. Though the OpenGL libraries could be found on the internet and installed on a PC running the first release of Windows 95, it is recommended that Windows be upgraded to a more current release. Windows 98, 2000, NT 4.0 and all later versions of Windows 95 contain the OpenGL libraries.

Most computers purchased today (May 2001) are perfectly adequate for running Smokeview with the caveat that additional memory should be obtained to bring the memory size up to at least 512mb in order to display results without “swapping” to disk and hence slowing down the visualizations. For Smokeview it is more important to obtain a fast graphics card than a fast CPU. If the computer will run both FDS and Smokeview then it is important to obtain a fast CPU as well. For example, the townhouse case found in Appendix E consisting of over 110,000 grid cells and used in many of the Figures in this report, required 8



Frame: 480
Time: 120.0

hours of CPU time on a Pentium III 1GHZ Linux system. Cases with more grid cells and longer simulation times (the townhouse case simulated 180 seconds of smoke flow) would clearly benefit from a faster CPU which are now becoming available.

3 Using Smokeview

Smokeview may be started on the PC by double-clicking the file named `casename.smv` where `casename` is the name specified by the `CHID` keyword defined in the FDS input data file. The `Load/Unload` menu may be used to read in the data files to be visualized. The `Show/Hide` menu may be used to change how the visualizations are presented. For the most part, the menu choices are self explanatory. Menu items exist for showing and hiding various simulation elements, creating screen dumps, obtaining help *etc.* Menu items are described in Appendix A.

To use Smokeview from a “command line”, open a DOS (if running on a PC) or UNIX shell, change to the directory containing the FDS case to be viewed and type:

```
smokeview casename
```

where again `casename` is the name specified by the `CHID` keyword defined in the FDS input data file. Data files may be loaded and options may be selected by clicking the right mouse button and picking the appropriate menu item.

Smokeview opens two windows, one displays the scene and the other displays status information. Closing either window will end the Smokeview session. Multiple copies of Smokeview may be run simultaneously if the computer has adequate resources.

Normally Smokeview is run during an FDS run or after it has completed. Smokeview may also be used as an aid in setting up FDS cases by visualizing geometric components such as blockages, vents, sensors, *etc.* One can then verify that these modelling elements have been defined and located as intended. One may select the color of these elements using color parameters in the `smokeview.ini` to help distinguish one element from another. `Smokeview.ini` file entries are described in section D.1.

3.1 Data Files

FDS generates several data files visualized by Smokeview. Each file type may be loaded or unloaded using the `Load/Unload` menu described in Appendix B.2. These data files are described below.

3.1.1 Particle

Particle files contain the locations of the tracer particles used to visualize the flow field. Particles may be colored with various gas properties. Colors are selected from a user definable color palette defined using the `COLORBAR` keyword documented in Appendix D.1. Figure 2 shows several snapshots of a developing fire plume visualized using particles where particles are colored according to their temperature. Sprinkler water droplets are colored blue. Particles are stored in files ending with the extension `.part` and are displayed by selecting the desired file from the `Load/Unload` menu.

To specify particles colored by temperature add the line:

```
&PART QUANTITY='TEMPERATURE' /
```

to the FDS input file found in Appendix E.1. To specify heat release rate per unit volume use `HRRPUV` above instead of `TEMPERATURE`.

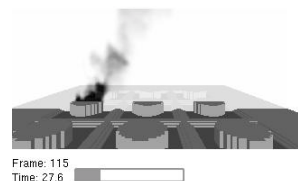


Table 1: Output Quantities used by Slice and Plot3D Visualizations

| Quantity | Description | Units |
|-------------|--------------------|-------------------|
| DENSITY | density | kg/m ³ |
| TEMPERATURE | gas temperature | C |
| U-VELOCITY | velocity component | m/s |
| V-VELOCITY | velocity component | m/s |
| W-VELOCITY | velocity component | m/s |
| VELOCITY | velocity magnitude | m/s |

3.1.2 Slice

Slice files contain data recorded within a rectangular array of grid points at each recorded time step. Continuously shaded contours are drawn for simulation quantities such as temperature, gas velocity and heat release rate. All or part of a plane is selected when setting up the FDS input data file. Figure 3 shows several snapshots of a vertical animated slice where the slice is colored according to gas temperature. Slice files have file names with extension `.sf` and are displayed by selecting the desired file from the Load/Unload menu.

To specify a vertical slice 1.5 m from the $y = 0$ boundary colored by temperature, add the line:

```
&SLCF PBX=1.5 QUANTITY='TEMPERATURE' /
```

to the FDS input file found in Appendix E.2 Some other quantities that can be specified are given in Table 1. A more complete list may be found in [2].

Animated vectors are displayed using data contained in two or more slice files. The direction and length of the vectors are determined from the U , V and/or W velocity slice files. The vector colors are determined from the file (such as temperature) selected from the Load/Unload menu. Figure 4 shows a sequence of vector slices corresponding to the shaded temperature contours found in Figure 3.

Recall that the FDS input line:

```
&SLCF PBZ=1.50, QUANTITY='TEMPERATURE' /
```

was used to generate the shaded contours in Figure 3. The additional input lines:

```
&SLCF PBZ=1.50, QUANTITY='U-VELOCITY' /
&SLCF PBZ=1.50, QUANTITY='V-VELOCITY' /
&SLCF PBZ=1.50, QUANTITY='W-VELOCITY' /
```

would then be used to record the U , V and W velocity data used to create the vector plot snapshots found in Figure 4. Alternatively, adding `VECTOR=.TRUE.` to `&SLCF` line in a FDS 2.0 data file as in:

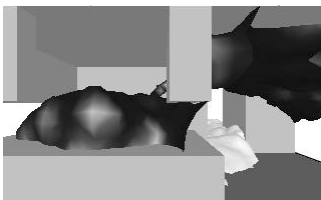
```
&SLCF PBZ=1.50, QUANTITY='TEMPERATURE', VECTOR=.TRUE. /
```

may be used to record temperature data and three components of velocity for 3D simulations.

3.1.3 Boundary

Boundary files contain simulation data recorded at boundary or wall surfaces. Continuously shaded contours are drawn for quantities such as wall surface temperature, radiative flux, *etc.* Figure 5 shows several snapshots of a boundary file animation where the surfaces are colored according to their temperature. Boundary files have file names with extension `.bf` and are displayed by selecting the desired file from the Load/Unload menu.

A boundary file containing wall temperature data may be generated by adding:



Frame: 460
Time: 115.0

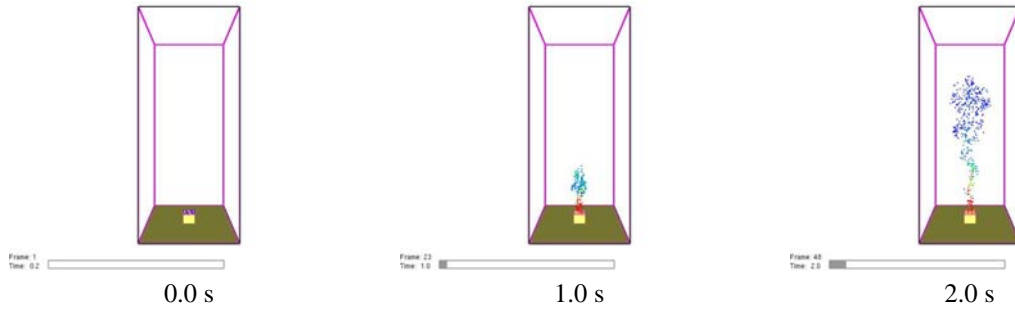


Figure 2: Particle file snapshots of an isolated plume at $t=0.0$, 1.0 and 2.0 seconds. The data file used to generate these snapshots was generated by adding the line: “&PART QUANTITY= "TEMPERATURE" /” to the FDS input file found in Appendix E.1.

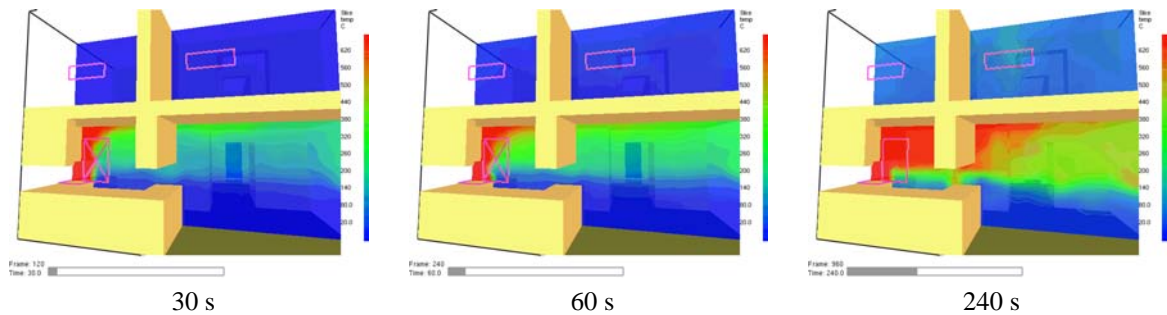


Figure 3: Slice file snapshots of shaded temperature contours in a vertical plane centered at the fire origin at $t=30$, 60 and 240 seconds after ignition. The data file for these contours was generated by adding the line, “&SLCF PBX=1.5, QUANTITY= "TEMPERATURE" /” to the FDS input file found in Appendix E.2.

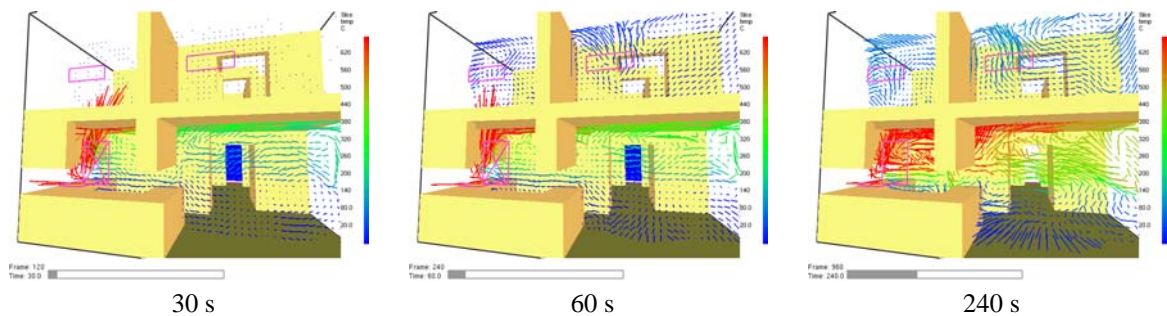


Figure 4: Vector slice file snapshots of shaded vector plots in a vertical plane centered at the fire origin at $t=30$, 60 and 240 seconds after ignition. The data files for these vector plots were generated by adding the line, “&SLCF PBX=1.5, QUANTITY= "TEMPERATURE" , VECTOR= .TRUE. /”, to the FDS input file found in Appendix E.2.

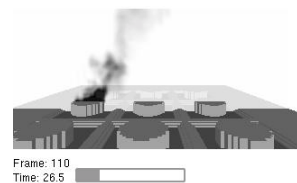


Table 2: Output Quantities used by Boundary File Visualizations.

| Quantity | Description | Units |
|------------------|------------------|-------------------|
| RADIATIVE_FLUX | radiative flux | kW/m ² |
| CONVECTIVE_FLUX | convective flux | kW/m ² |
| WALL_TEMPERATURE | wall temperature | C |

```
&BNDF 'WALL_TEMPERATURE' /
```

to the FDS input file found in Appendix E.2. Table 2 gives other quantities that can be specified on a &BNDF line. A more complete list of quantities is given in Ref. [2].

3.1.4 Isosurface

The surface where a quantity such as temperature attains a given value is called an isosurface. An isosurface is sometimes also called a level surface or 3D contour. Isosurface files contain data specifying isosurface locations for a given variable at one or more levels. These surfaces are represented as triangles. Figure 6 shows a snapshot of an isosurface file animation at several time steps for a fire (mixture fraction level=0.3) and a layer interface (mixture fraction level = 0.001). Isosurface files have file names with extension .iso and are displayed by selecting the desired file from the Load/Unload menu.

Isosurfaces are specified in the FDS input file (version 1.01 or higher) with the &ISOF keyword. To specify isosurfaces for mixture fraction with values 0.03 and 0.001 as illustrated in Figure 6 add the line:

```
&ISOF QUANTITY='MIXTURE_FRACTION',VALUE(1)=0.03,VALUE(2)=0.001 /
```

to the input file found in Appendix E.2. Other quantities may be specified using values given in Table 1. A more complete list may be found in [2]

3.1.5 Plot3D

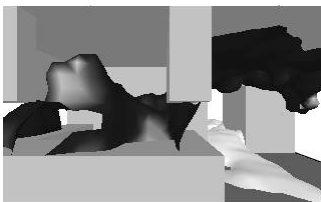
Data stored in Plot3D[11] files use a format developed by NASA and used by many CFD programs for representing simulation results. Plot3D files store five data values at each grid cell. FDS uses Plot3D files to store temperature, three components of velocity (U, V, W) and heat release rate by default. Other quantities may be stored if desired. In addition the Plot3D format allows one to specify internal blockages so that contour data need not be displayed where the physical domain is blocked off.

An FDS simulation will typically create Plot3D files at several times throughout the simulation. Plot3D data is visualized in three ways: as 2D contours, vector plots and isosurfaces. Figure 7a shows an example of a 2D Plot3D contour. Vector plots may be viewed if one or more of the U,V and W velocity components are stored in the Plot3D file. The vector length and direction show the direction and relative speed of the fluid flow. The vector colors show a scalar fluid quantity such as temperature. Figure 7b shows vectors in a doorway. Note the hot flow (red color) leaving the fire room in the upper part of the door and the cool flow (blue color) entering fire room in the lower part of the door. Figure 8 gives an example of isosurfaces for two different temperature levels. Plot3D data are stored in files with extension .q.

To specify a 30 seconds time interval between Plot3D files stored with default gas variables add the line:

```
&PL3D DTSAM=30.0 /
```

to the FDS input file. To change the gas variables stored use the QUANTITIES keyword picking quantities from Table 1, for example:



Frame: 440
Time: 110.0

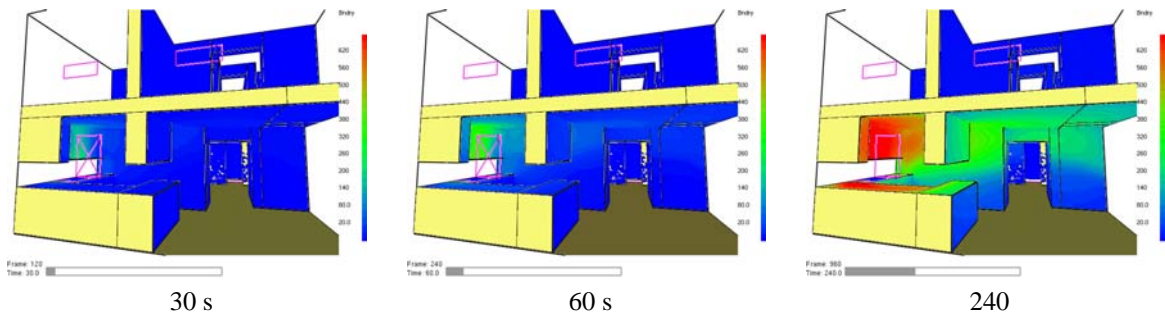


Figure 5: Boundary file snapshots of shaded wall temperatures at $t=30$, 60 and 240 seconds after ignition. The data file for these snapshots was generated by adding the line:
`&BNDF QUANTITY="WALL_TEMPERATURE" /` to the data file found in Appendix E.2.

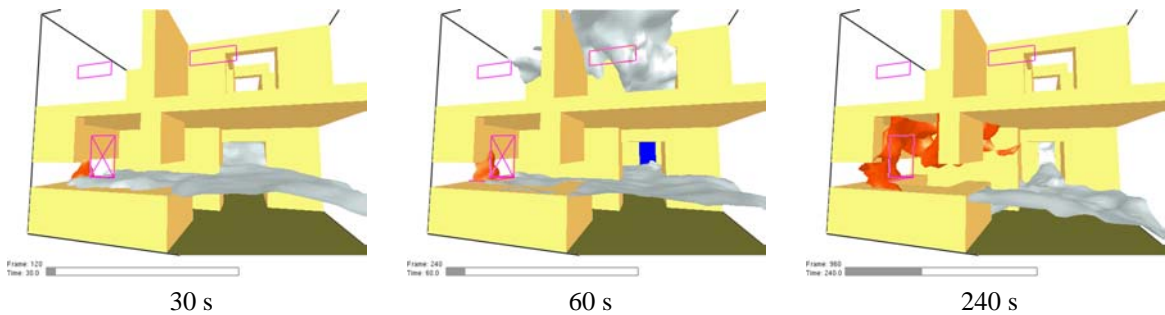
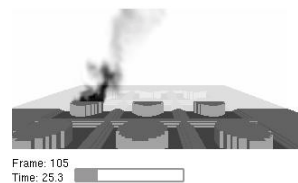


Figure 6: Isosurface file snapshots of mixture fraction levels at $t=30$, 60 and 240 seconds after ignition. The data file for these snapshots was generated by adding the line:
`&ISOFS QUANTITY='MIXTURE_FRACTION', VALUE(1)=0.03, VALUE(2)=0.001 /`, to the FDS input file found in Appendix E.2.



&PL3D QUANTITIES="DENSITY" , "PRESSURE" , "WMPUV" , "VELOCITY" , "TEMPERATURE" /

It is recommended however that U-VEL, V-VEL, W-VEL be stored so that vector plots may be drawn.

The format used to store each of the above data files is given in Appendix D.3 so that Smokeview may be used to visualize data generated by other fire models.

3.2 Visualizing Output

3.2.1 Manipulating the Scene

The scene may be rotated or translated either directly with the mouse or by using the scene movement dialogue box as illustrated in Figure 9. Clicking on the scene and dragging the mouse horizontally, vertically or a combination of both results in scene rotation or translation depending upon whether the SHIFT, CTRL or ALT keys are depressed or not during mouse movement. These effects are described below.

Smokeview uses a standard right handed coordinate system. Left and right scene motion occurs along the X axis, up and down scene motion occurs along the Y axis while in and out scene motion occurs along the Z axis. Axis labels are displayed, if desired, by selecting the *Axis label* menu item under the *Show/Hide* menu.

The modifier keys effect scene movement in the following ways:

No modifier key depressed Horizontal(vertical) mouse movement results in scene rotation parallel to the XY (YZ) plane. Note that rotation parallel to the YZ plane is disabled while *eye view* is in effect. The view is switched between *eye view* and *world view* by either depressing the *e* key or by selecting the *Show/Hide view* menu item. Equivalently, horizontal(vertical) mouse movement results in scene rotation about the Z(X) axis.

SHIFT key depressed Horizontal(vertical) mouse movement results in scene rotation parallel to the XZ(YZ) plane. Equivalently, horizontal(vertical) mouse movement results in scene rotation about the Y(X) axes respectively.

CTRL key depressed Horizontal mouse movement results in scene translation from side to side along the X axis. Vertical mouse movement results in scene translation “in and out” of the computer screen along the Y axis.

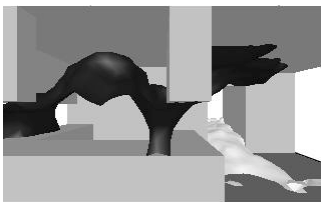
ALT key depressed Vertical mouse movements results in scene translation along the Z axis. Horizontal mouse movement has no effect on the scene.

The scene motion dialogue box may be used to move the scene in a more controlled manner. For example, clicking on the “Rotate X” button in Figure 9 and dragging the mouse results in scene rotation about the “X” (and only the “X”) axis. Similarly, clicking the mouse in the “Translate Y” button results in scene translation along the “Y” axis. This dialogue box is opened from the *Option* menu.

The *Reset View* menu may be used to reset the scene back to either an external viewpoint, an internal (to the scene) viewpoint or a previously saved viewpoint.

3.2.2 Setting data bounds

Normally, Smokeview determines data bounds automatically when it loads data. Sometimes, however, it is desirable to override Smokeview’s choice, for example so that several loaded files can have the same bounds. This allows for consistent shading when displaying several files simultaneously.



Frame: 420
Time: 105.0

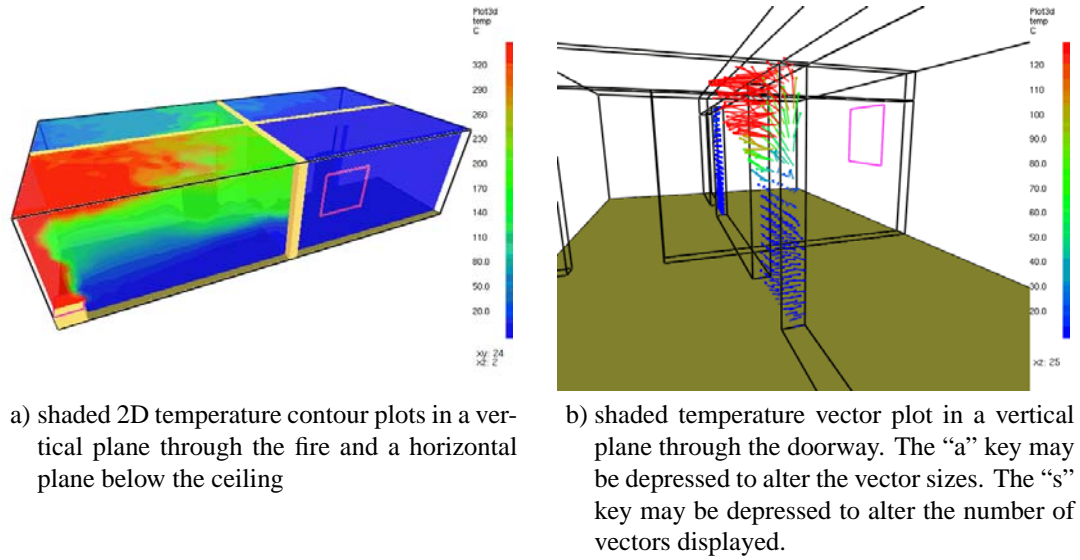


Figure 7: Plot3D contour and vector plot examples.

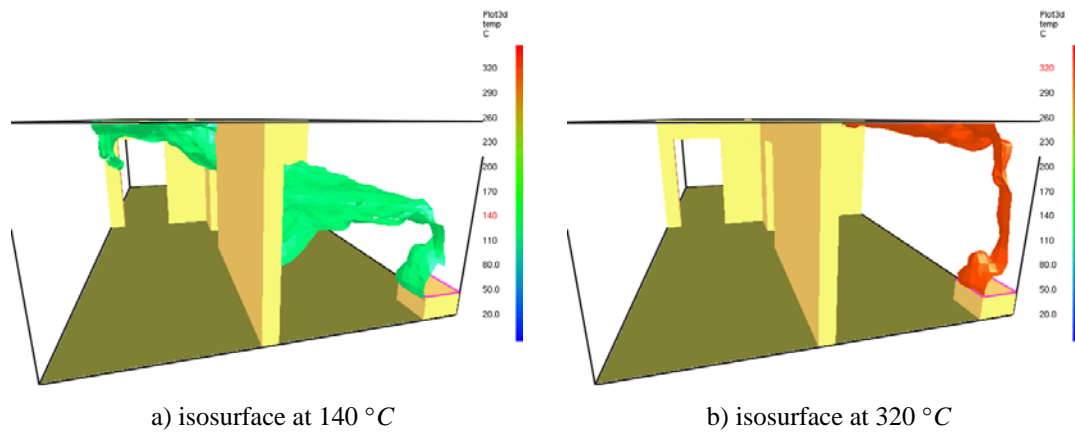


Figure 8: Plot3D isocontour example.

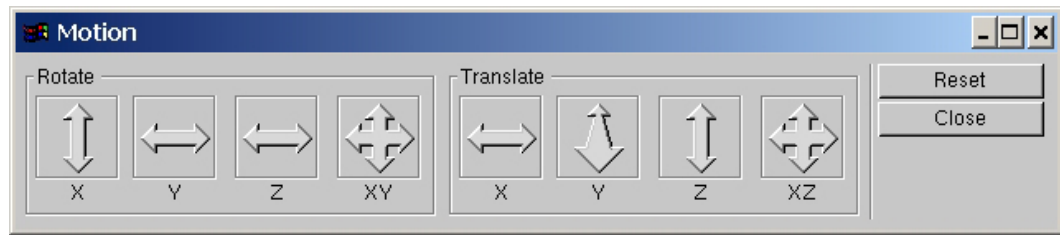
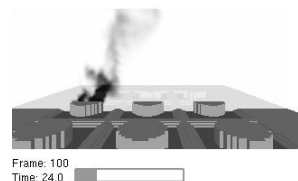


Figure 9: Motion Dialogue Box. Rotate or translate the scene by clicking an “arrow” and dragging the mouse.



Data bounds may be set using either the dialogue box in Figure 10 or using `smokeview.ini` keywords. Using the configuration file `smokeview.ini` to set data bounds is explained in Appendix D.1.

The `Bounds Dialogue Box` is opened from the `Options` menu. Each file type in Figure 10 (slice, particle, Plot3D *etc*) has a set of “radio buttons” for selecting the variable type that data bound are to be applied too. These variable types are determined from the files generated by FDS and are automatically recorded in the `.smv` file. The data bounds are set in a pair of edit boxes. Check boxes adjacent to the edit boxes determine whether or not Smokeview should use the entered data bounds. The new bounds become activated after the data has been reloaded. The `Plot3D` portion of the dialog box uses radio buttons instead of check boxes in order to implement an additional loading option. Portions of the plot as illustrated in Figure 11 are not drawn when this option, labelled “chop”, is selected.

Data bounds may be saved by creating a `smokeview.ini` file using the `Init` menu found under `Load/Unload`

3.2.3 Plot3D Viewing Options

Plot3D files are more complicated to visualize than time dependent files such as particle, slice or boundary files. For example, only the transparency and color characteristics of a time file may be changed. With Plot3D files however, many attributes may be changed. One may view 2D contours along the X, Y and/or Z axis of up to six¹ different simulated quantities, view flow vectors and iso or 3D contours. Plot3D file visualization is initiated by selecting the desired file from the `Load/Unload Plot3D` sub-menu and as with time files one may change color and transparency characteristics.

2D contours Smokeview displays a 2D contour slice midway along the Y axis by default when a Plot3D file is first loaded, To step the contour slice up by one grid cell along the Y axis, depress the “space bar”. Similarly to step the contour slice down by one grid cell along the Y axis, depress the “-” key. To view a contour along either the X or Z axis, depress the x or z keys respectively. Depressing the x, y or z keys while the contour is visible will cause it to be hidden. The Plot3D variable viewed may be changed by either depressing the “p” key or by selecting the `Solution Variable` sub-menu of the `Show/Hide` menu.

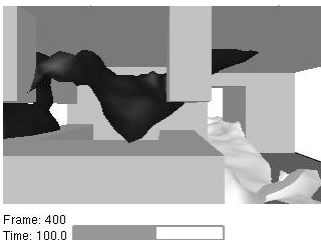
Iso-Contours Iso-contours also called 3D contours or level surfaces may be viewed by depressing the “i” key or by selecting the `Plot3D 3D Contours` sub-menu of the `Show/Hide` menu.

Flow vectors If at least one velocity component is present in the Plot3D file then the “v” key may be depressed in order to view flow vectors. The length and direction of the vector indicates the flow direction and speed. The vector color indicates the value of the currently displayed quantity. A small dot is drawn at the end of the line to indicate flow direction. The vector lengths as drawn may be changed by depressing the “a” key. Vector plots may be very dense when the grid is finely meshed. The “s” key may be depressed in order to skip vectors. For example, all vectors are displayed by result. If the “s” is depressed then every other vector is skipped.

3.3 Examining Input

The previous section describes how Smokeview is used to visualize FDS output. This section explains how Smokeview may be used to setup FDS input files.

¹The FDS software stores temperature, three components of velocity (denoted u , v and w) and heat release per unit volume. If at least one velocity component is stored in a Plot3D file then Smokeview adds speed to the Plot3D variable list.



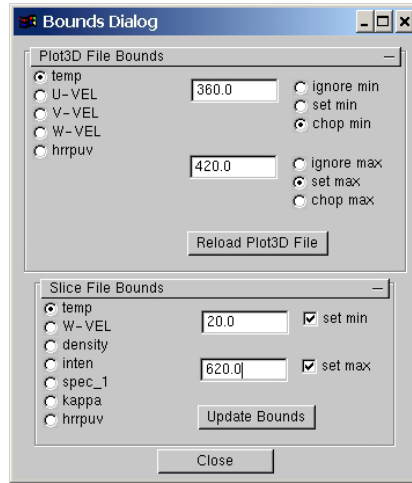


Figure 10: Bounds Dialogue Box. Select a variable, select a bounds type “check box/radio button”, then enter a bound. Then either reload the data file or update the bounds.

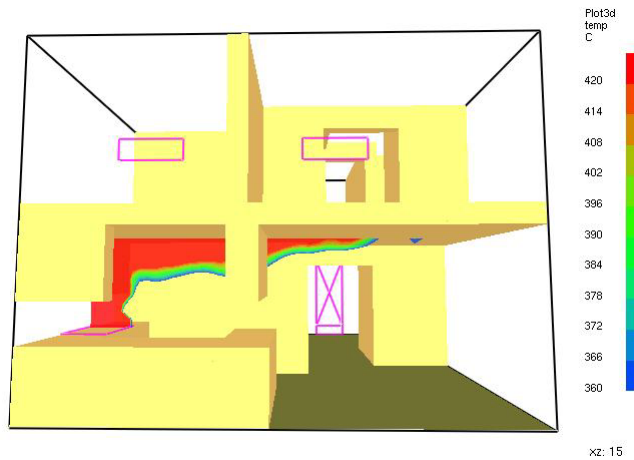
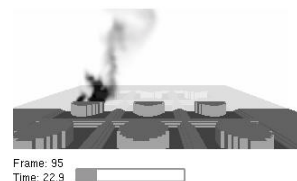


Figure 11: Ceiling jet visualization created by “chopping data” below a specified temperature using a V_PLOT3D keyword entry of: “1 2 360.0 1 420.0” for the first (temperature) Plot3D variable. The same effect can be achieved by using the Bounds Dialogue Box as illustrated in Figure 10.



One of the most difficult tasks in setting up an FDS input file is defining the geometry (blockage, vent locations *etc*) properly. Smokeview may be used to debug input FDS files by making short runs and observing whether blockages, vents and other geometric features of a run are located correctly. A new feature in Smokeview version 2.0 is the ability to create new blockages and edit existing ones. This is done with the “Edit Blockage” dialogue box called from the `Option` menu. The next two sections indicate how these tasks may be performed.

3.3.1 Debugging

The following is a general procedure for identifying problems in FDS input files. Assume that the FDS input data file is named `testcase1.data`.

1. Add `SET_UP=.TRUE.` to the `&MISC` line. This causes FDS to read the input file and create a `.smv` file without running the case.
2. After the input file has been modified, run the FDS model by typing:

```
fds < testcase1.data
```

FDS creates a file named `testcase1.smv` containing information that Smokeview uses to visualize data.

3. To visualize the input file, open `testcase1.smv` with Smokeview by either typing `smokeview testcase1` at a command shell prompt or if on the PC by double-clicking the file `testcase1.smv`. If Smokeview was already running (*i.e.* from an earlier iteration in this debugging cycle) re-read `testcase1.smv` into Smokeview with the `Reload testcase1.smv` menu item.
4. Make corrections if necessary. Use the `BLOCK_COLOR` option² of the FDS keyword `OBST` to more easily identify blockages to be edited. For example, to change a blockage’s color to red use:

```
&OBST XB=0.0,1.0,0.0,1.0,0.0,1.0 BLOCK_COLOR='RED' /
```

Save `testcase1.data` file and go back to step 2.

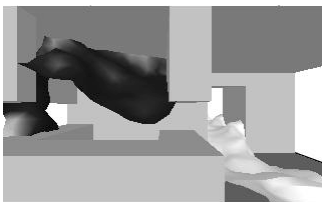
5. If corrections were not necessary, then change the `TWFIN` keyword back to the desired final simulation time, remove any unnecessary FDS `BLOCK_COLOR`, `VENT_COLOR` and `SET_UP` keywords and run the case.

3.3.2 Editing Blockages

The Blockage Edit dialogue box shown in Figure 12 may be used to create new blockages and edit existing ones. This dialogue box is opened from the `Options` menu. Blockage modifications are saved to a file named `casename.data_new` where `casename.data` is the original file name.

The following is a short tutorial on how to create the `roomfire.data` example case distributed with FDS and Smokeview using the Blockage edit dialogue box to create the blockages. The results of each step are illustrated in Figure 13

²The FDS `BLOCK_COLOR` and `VENT_COLOR` keywords may have values: RED, BLUE, GREEN, MAGENTA, CYAN, YELLOW, WHITE or BLACK.



Frame: 380
Time: 95.0

Step 1 - Create base case: Start by adding the following line to a file named `casel.data`.

```
&HEAD CHID='casel',TITLE='Sample fire in multiple room layout' /
```

Run this file with FDS by typing:

```
fds < casel.data
```

The resulting output is illustrated in Figure 13a. Note that by default FDS assumes that the input domain is a unit cube.

Step 2 - Defining the Geometry and the Grid: Assume that the input domain dimensions are $9.6\text{ m} \times 4.8\text{ m} \times 2.4\text{ m}$ and that each grid cell is 0.1 m on a side. The FDS input lines to implement these assumptions are given by:

```
&HEAD CHID='casela',TITLE='Sample fire in multiple room layout' /
&GRID IBAR=96, JBAR=48, KBAR=24 /
&PDIM XBAR=9.6, YBAR=4.8, ZBAR=2.4 /
&MISC SETUP=.TRUE. /
&TIME TWFIN=0. /
```

These input lines result in output illustrated in Figure 13b.

Step 3 - Adding Blockages: The next step is to add blockages for the fire and the two walls. Assume that the blockage for the fire has dimension $0.5\text{ m} \times 0.5\text{ m} \times 0.3\text{ m}$ and is located at the front, left, bottom corner. Open the Blockage Edit Dialogue Box and click in the **New** button. This creates a new blockage one grid cell in size located in the lower, left, front part of the simulation domain. Expand the blockage to the desired size ($0.5\text{ m} \times 0.5\text{ m} \times 0.3\text{ m}$) by either 1) typing in the proper dimension into the edit boxes or by 2) clicking in the stretch/move arrows to stretch/move the blockage to the proper location/size.

Repeat this process twice more, once for each wall. Each wall is 0.2 m thick. The first wall is $9.6\text{ m} \times 2.4\text{ m}$ and is located in the middle of the short ('Y') side from 2.4 m to 2.6 m . The second wall is $4.8\text{ m} \times 2.4\text{ m}$ and is located in the middle of the long ('X') side from 4.8 to 5.0 m . Again these blockages may be entered by either 1) typing in the proper dimension into the edit boxes or 2) clicking in the stretch/move arrows to stretch/move the blockage to the proper location/size.

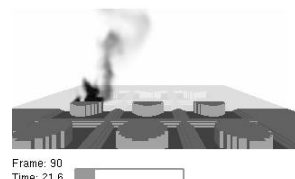
The FDS input lines to complete these steps are generated by clicking the "Update" button and are given by:

```
&HEAD CHID='caselb',TITLE='Sample fire in multiple room layout' /
&GRID IBAR=96, JBAR=48, KBAR=24 /
&PDIM XBAR=9.6, YBAR=4.8, ZBAR=2.4 /
&MISC SETUP=.TRUE. /
&TIME TWFIN=0. /

&OBST XB=0.000000, 0.500000, 0.000000, 0.500000, 0.000000, 0.300000/
&OBST XB=0.000000, 9.600000, 2.400000, 2.600000, 0.000000, 2.400000/
&OBST XB=4.800000, 5.000000, 0.000000, 4.800000, 0.000000, 2.400000/
```

The results of these input lines are illustrated in Figure 13c.

Step 4 - Adding Doors ("subtracting blockages") Doors and windows are "added" by removing portions of pre-existing blockages. To add the first door, select the longer of the two walls by clicking on it. Next click on the radio button labelled 'Openings' and then click on the 'New' button. Type in the door coordinates in the edit boxes or click in the stretch/move arrows to stretch/move the door to the proper location/size.



Step 5 - Completing the case To complete the case add input lines for vents (&VENT commands) and keywords for various output files as shown in the input file:

```
&HEAD CHID='caselc',TITLE='Sample fire in multiple room layout' /
&GRID IBAR=96, JBAR=48, KBAR=24 /
&PDIM XBAR=9.6,YBAR=4.8,ZBAR=2.4 /
&TIME TWFIN=0. /

&OBST XB=0.000000, 0.500000, 0.000000, 0.500000, 0.000000, 0.300000/
&OBST XB=0.000000, 3.000000, 2.400000, 2.600000, 0.000000, 2.000000/
&OBST XB=0.000000, 9.600000, 2.400000, 2.600000, 2.000000, 2.400000/
&OBST XB=4.000000, 7.000000, 2.400000, 2.600000, 0.000000, 2.000000/
&OBST XB=8.000000, 9.600000, 2.400000, 2.600000, 0.000000, 2.000000/
&OBST XB=4.800000, 5.000000, 0.000000, 3.500000, 0.000000, 2.000000/
&OBST XB=4.800000, 5.000000, 4.500000, 4.800000, 0.000000, 2.000000/
&OBST XB=4.800000, 5.000000, 0.000000, 4.800000, 2.000000, 2.400000/

&SURF ID='BURNER',HRRPUA=1000. /

&VENT XB=0.00,0.50,0.00,0.50,0.30,0.30, SURF_ID='BURNER' /
&VENT XB=5.50,6.60,0.00,0.00,1.00,2.00, SURF_ID='OPEN' /

&PART QUANTITY='TEMPERATURE',NPSAM=2/
&ISOF QUANTITY='MIXTURE_FRACTION',VALUE(1)=0.03,VALUE(2)=0.001 /
&SLCF PBX=3.60,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBX=7.50,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBY=0.30,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBY=1.30,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=3.90,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=2.30,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /

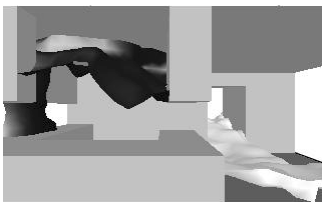
&BNDF QUANTITY='WALL_TEMPERATURE' /
&PL3D DTSAM=20. /
```

and illustrated in Figure 13e. Figure 13f shows the sample case run to 10 seconds with a mixture fraction isosurface representing the a layer of constant smoke density.

4 Summary

This report documented the use of Smokeview version 2.0 to visualize predictions generated by the NIST Fire Dynamics Simulator. Various visualization techniques such as animated tracer particle flow, contour plots and isosurfaces were implemented to enable the user to better understand fire and smoke behavior under investigation.

The next step in this research project will be to investigate additional visualization techniques for characterizing and understanding fire dynamics data. For example, the isosurface algorithms presently used in Smokeview calculate only the location of a level surface such as that corresponding to a flame sheet. By coloring the level surface according to the value of a second variable such as temperature, one could see the effects of temperature on combustion. In general, techniques that help one better understand the simulated environment will be investigated.



Frame: 360
Time: 90.0

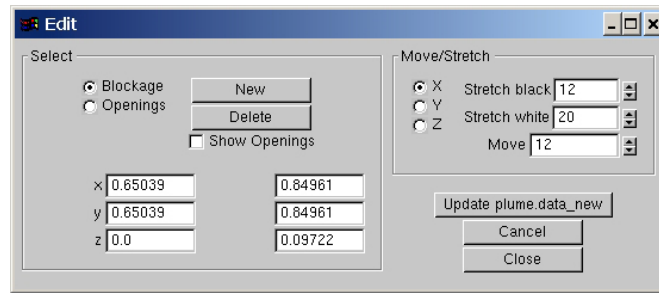


Figure 12: Blockage Edit Dialogue Box. Click on 'New' to create a new blockage or click on an existing blockage to change its size or location. Change the size or location by entering blockage bounds directly or clicking on an 'arrow' and/or direction buttons. Windows or doors may be 'carved' out of existing blockages by selecting the Opening rather than Blockages 'radio button'.

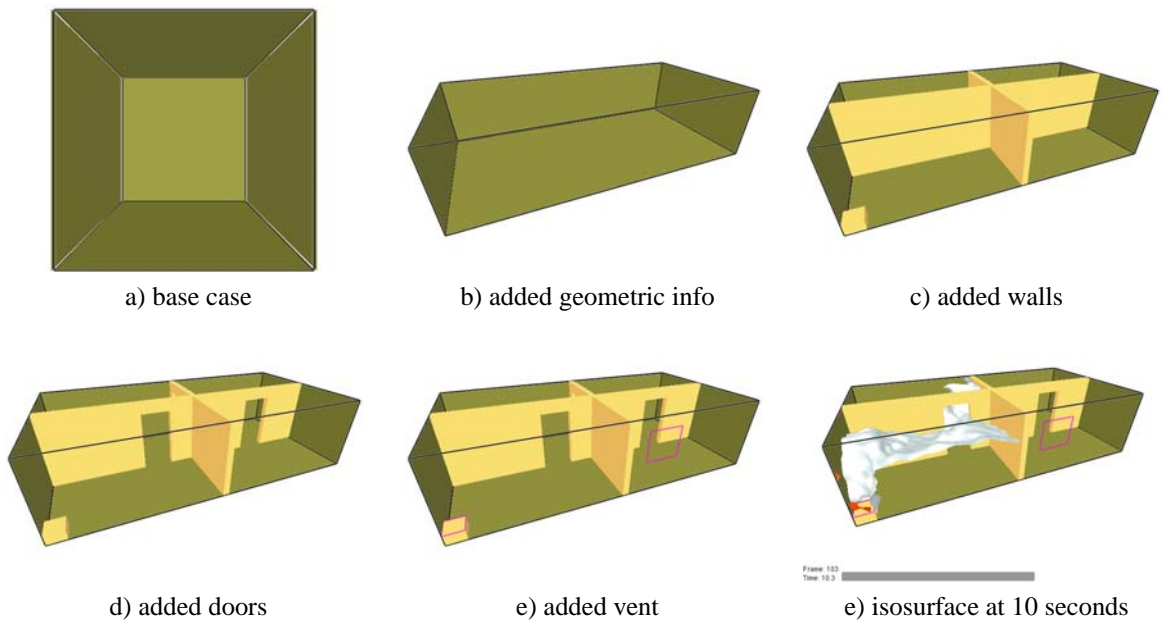
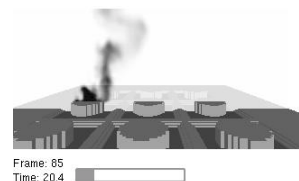


Figure 13: Modifying the base FDS case in steps.



A Smokeview Command Line Options

Smokeview may be run from a command shell. Further, command line options may be invoked in order to alter Smokeview's startup behavior. Normally these options are not necessary. However, they may be used for cases with very large particle files or to generate a preference or customization file. To obtain a list of command line options, type:

```
smokeview -help
```

without any arguments which results in output similar to:

The `-nopart` option is used to prevent a particle file from being loaded at startup as in:

```
smokeview casename -nopart
```

The `-points` or `-frames` options may be used to load more than the default 5,000,000 points or 500 frames where a frame is data for one time step. To load up to 6,000,000 points and 1000 frames then type:

```
smokeview casename -points 6000000 -frames 1000
```

where in both cases `casename` is the basename of the FDS output files. The same effect may be achieved by using:

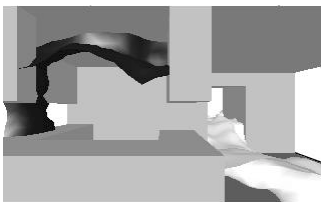
```
MAXPOINTS
6000000
MAXFRAMES
1000
```

in the `smokeview.ini` or `casename.ini` file. This file may be created with the `-ini` option and contains many other customizable Smokeview parameters. The `-benchmark` option is used to measure the performance of Smokeview. When particle files are displayed Smokeview outputs the rate that points (particles) are displayed per second. Similarly, when isosurfaces (3D Plot3D contours) are displayed the `-benchmark` option causes Smokeview to output the rate that triangles are displayed per second.

B Menu Options

The design philosophy used to develop Smokeview has been to avoid complicated, non-portable user interfaces that are costly to implement. As a result, most of the development effort has gone into the visualization (display of particle flow, contour plots *etc*) rather than user interface design and implementation. The graphics library utility toolkit GLUT[9] based on OpenGL[7, 8] is used to implement the "pop up" menus used by Smokeview to manipulate various features of the graphical scenes. The user interacts with Smokeview *via* 1) menus, 2) keyboard shortcuts and 3) the `.ini` file.

A "pop up" menu is displayed when the right mouse is clicked anywhere within the scene. The main menu options are: Load/Unload, Show/Hide, Max Frame Rate, Render, Tour, Reset View, Help and Quit. Several of these menu options have sub-menus. These menus are described in the following sections.



Frame: 340
Time: 85.0

B.1 Main Menu Items

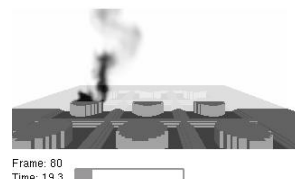
- Load/Unload** This menu option allows one to load or unload particle, slice, Plot3D or boundary files. The three time dependent files (particle, slice and boundary) may be viewed simultaneously, but not concurrently with the time independent Plot3D files. However, only one particle, one boundary and one Plot3D file may be viewed at a time. Multiple slices for the same variable may be viewed simultaneously. This menu may also be used to load and create preference files `.ini` files and to re-read the `.smv` file. For more details see Appendix B.2.
- Show/Hide** This menu item allows one to show or hide the loaded data files and various scene attributes such as time/color bars, internal blockages *etc.* As a file type is shown or hidden (or loaded and unloaded), the color and time bars are changed to reflect the currently visible data files. More details are given in Appendix B.3.
- Options** This menu allows one to specify various smokeview options such as specifying frame rates, dumping the screen to a PNG or JPEG file, changing font sizes, selecting dialogue boxes *etc.*
- Tour** Causes the scene to rotate about its center.
- Reset** Resets the simulation scene back to an alternate view. The three choices are 1) exterior view, 2) interior view and 3) saved view. A viewpoint may be saved using the `Viewpoint` sub-menu of the `Options` menu. If a time file (particle, slice and/or boundary) is visible then two sub-menus occur allowing one to reset the view back to the original position or the time bar back to the initial time.
- Help** Displays a list and explanation of keyboard equivalent commands.
- Quit** Quit smokeview.

B.2 Load/Unload

Five types of files may be visualized with Smokeview. They are particle, slice, boundary, isosurface and Plot3D files. Note that vector slice animations use two or more slice files to display the animated vector slices. The format of the data contained in these files is described in Appendix D.3. A sub-menu is present under `Load/Unload` for each file type generated for the simulation. Selecting one of the files appearing in the sub-menu causes it to be loaded and then displayed. The data may be unloaded or freed by selecting an `Unload` menu item appearing under the file list. Selecting `Unload All` as expected will unload all files. If the wish is to temporarily hide a data file, then the appropriate `Show/Hide` menu option (depending on the file type to be hidden) should be selected.

The Smokeview input data file contains information about all data files appearing in the `Load/Unload` menu item. The FDS field modelling software creates this file. (See Appendix D.2 for documentation on the format of this file).

The character “*” occurring before a file name in one of the sub-menus indicates that the file has already been loaded. If the file below is loaded but not visible, then use the appropriate `Show/Hide` option to make it visible.



Open Smokeview (.smv) This menu item allows one (on the PC) to use a dialogue box to open other FDS cases.

Isosurface File (.iso) This menu item gives the name of all isosurface files and also the option to unload the currently loaded isosurface file.

Particle File (.part) This menu item gives the name of the particle file and also the option to unload the currently loaded particle file.

Vector Slice File (.sf) This menu item gives the name of all slice files that have one or more associated U, V and/or W velocity slice files. These slice files must be defined for the same region (or slice) in the simulation.

Slice File (.sf) This menu item gives the name of all slice files and also the option to unload the currently loaded slice files. New in Smokeview version 2.0 is the ability to view several slice files simultaneously.

Plot3D File (.q) This menu item gives the name of all Plot3D files and also the option to unload the currently loaded Plot3D file.

Boundary File (.bf) This menu item gives the name of all boundary files and also the option to unload the currently loaded boundary file.

Init File (.ini) The init or preference file contains configuration parameters that may be used to customize Smokeview's appearance and behavior. This menu item lets one create (or overwrite) a preference file named either `smokeview.ini` or `casename.ini`. A preference file contains parameter settings for defining how Smokeview visualizes data. This file may be edited and re-read while Smokeview is running.

Smokeview parameter input file (.smv) This menu item is used when debugging FDS input files. It allows one to reload a Smokeview case (.smv file). A debugging cycle can then be performed consisting of 1) editing FDS input file, 2) running FDS and 3) reloading an updated Smokeview input file.

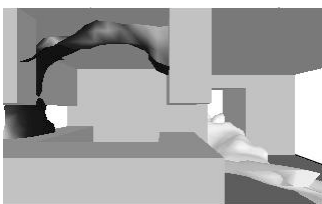
Unload All This option causes all data files to be unloaded.

B.3 Show/Hide

The Show/Hide menu item allows one to show or hide various parts of the simulation. This menu item contains sub-menus for Particles, Boundary, Animated Slice, Plot3D 2D and 3D contours, sensors (thermocouples, heat detectors and sprinklers), Color and time Bars and Geometry. These menu items only appear if they pertain to the simulation. For example the `Particles` sub-menu only appears if a particle file has been loaded. Similarly, the `Plot3D contouring` sub-menus only appear if a Plot3D file has been loaded. The "*" character is used to indicate whether the visualization feature corresponding to the menu item is set or active.

B.3.1 Geometry Options

Blockages The three blockage viewing options are `Show All`, `Show Frame` and `Hide`. The first and last option are self explanatory. The `Show Frame` option allows one to view only a frame or



Frame: 320
Time: 80.0

outline of each blockage. This makes it easier to see other simulation features when the scenario has many blockages. Blockages may be displayed in different colors. This allows one to more easily set up FDS input files by identifying blockages and verifying that they have been defined as intended.

- Grid** This option allows one to visualize the grid used to perform the numerical calculations. One selects `xy plane`, `xz plane` or `yz plane` to visualize a single plane or `Show All`, `Hide All` to show or hide all grids. (Keyboard shortcut: `g`)
- Outline** Show or hide the outline that frames the simulation scene.
- Floor** Show or hide the floor.
- Ceiling** Show or hide the ceiling
- Vents** Show or hide the vents.
- Walls** Show or hide the walls.
- Show All** Show all geometric features described above.
- Hide All** Hide all geometric features described above.

B.3.2 Animated Surface

An isosurface is represented in Smokeview as a collection of triangles. Each triangle consists of three edges and three vertices.

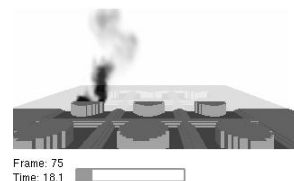
- Solid** Display the isosurface by shading the triangles.
- Outline** Display the isosurface by only showing the triangle edges.
- Points** Display the isosurface by only showing the triangle vertices.
- “quantity” levels** Display the desired isosurface level (when more than one isosurface is stored in an isosurface file).
- Smooth** Display the isosurface using smoothed vertex normals.

B.3.3 Particles

- Smoke (tracer)** Toggle the visibility of the particles. If water/sprinkler droplets are present in the particle file then sub-menus exist for both smoke and water to allow one to show or hide smoke and water droplets independently.
- Sprinkler** Toggle the visibility of sprinkler (water droplets).

B.3.4 Boundary

- Exterior** Show or hide all data contained in a boundary (`.bF`) file pertaining to the exterior walls.
- Interior** Show or hide all data contained in a boundary (`.bF`) file pertaining to interior blockages.
- Front, Back, Left, Right, Up, Down** Toggle the visibility of whatever exterior boundary surface is selected. Note an exterior boundary menu option only appears if its data is present in the boundary file.



B.3.5 Animated Vector Slice

Toggle the visibility of the animated vector slice file.

B.3.6 Animated Slice

Toggle the visibility of the animated slice file.

B.3.7 Plot3D 2D Contours

Solution Variables A Plot3D data file contains five solution variables. If one or more of the velocity components denoted u , v and w are present in the Plot3D file then speed, calculated using $\sqrt{u^2 + v^2 + w^2}$, appears in the menu. Any velocity components missing from the Plot3D file are set to 0.0 when calculating speed. This menu item allows one to select the Plot3D solution variable to be visualized. (Keyboard shortcut: p)

xy, xz, yz planes These three menu items, appearing beneath the **Solution Variables** menu item, allow one to select which plane (xy, xz or yz) is displayed. (Applicable keyboard shortcuts: space bar, -, left/right cursor, up/down cursor, page up/down, 1 . . . 9)

Flow vectors Toggle visibility of flow vectors. The magnitude and direction of the vectors are determined by the U, V and W components of velocity. The vector color is determined by the solution variable selected. (Applicable keyboard shortcuts: a, s, v)

Continuous Display contours as smooth continuous shades or as stepped constant shades. (Keyboard shortcut: c)

Show All Show all three (xy, xz and yz) Plot3D planes at once.

Hide All Hide all three Plot3D planes.

B.3.8 Plot3D 3D Contours

Solution Variables Same as for 2D contours. This menu item allows one to select the solution variable used to generate the 3D or iso-contour to be displayed.

Solution Value Select the 3D contour level to display. The axis label shown in red corresponds to the 3D contour level displayed.

Block Size To increase the drawing speed, adjacent grid cells may be combined when viewing 3D contours. Selecting 1 will result in highly resolved contours but may take longer to draw. Selecting 5 will have the opposite effect.

Hide Hide the 3D contour.

B.3.9 Heat detectors, Sprinklers, Thermocouples

Toggle the sensor visibility. The currently implemented sensors are heat detectors, sprinklers and thermocouples.

B.3.10 Labels

Color Bars, Time Bars, Title, Axis labels, Frame Rate Show or hide the individual scene element.



Frame: 300
Time: 75.0

Show All Show all scene elements.

Hide All Hide all scene elements.

B.4 Options

B.4.1 Shades

Flip Background Flip the background color between a dark and light shade. A dark background shade looks better on a computer monitor while a light shade looks better on the printed page.

Flip Colorbar Reverse the order of the colors displayed in the colorbar.

Cycle Colorbar Cycle the colors in the colorbar.

Reset Colorbar Return the colorbar to the original display.

Color Toggle scene colors between color and shades of gray. This option currently does not convert all scene colors to shades of gray. It was felt that contours and blockages were the most important to convert.

Transparent Toggle contours between opaque and transparent. Transparent colors allow one to view the scene behind the contours giving the one a better sense of scale. Transparent colors, however, may make the scene look too confusing when the geometry is complex.

B.4.2 Units

Select alternate temperature or velocity units. The `UNIT` keyword described in Appendix D.1 may be used to incorporate additional unit changes into Smokeview.

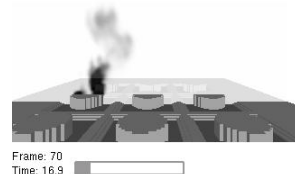
B.4.3 Rotation

Eye Centered Rotate and move the scene relative to the observer's "eye". Eye centered views make it easier to move around within the scene itself as in modern computer games. (Keyboard shortcut: `e`. The "e" keyboard shortcut toggles the view between an *eye centered* and a *world centered* perspective.)

World Centered Rotate and move the scene relative to the scene's center. (Keyboard shortcut: `e`. The "e" keyboard shortcut toggles the view between an *eye centered* and a *world centered* perspective.)

B.4.4 Max Frame Rate

This option controls the rate at which image frames are displayed. The sub-menus allow one to specify a maximum frame rate. The actual frame rate may be slower if the scene is complex and the graphics card is unable to draw the scene sufficiently fast. The `unlimited` menu item allows one to display frames as rapidly as the graphics hardware permits or to step through the frames manually. The `Real Time` menu item allows one to draw frames so that the simulation time matches real time. The `step` menu item allows one to step through the simulation one time step at a time. This menu item may be used in concert with the `Render` menu item described below to create images at the desired time and view orientation for inclusion into reports. This is how figures were generated in this report.



B.4.5 Render

This option allows one to create PNG or JPEG image files of the currently displayed scene. The graphics library, GD 2.0, was used to create the rendered versions. GD 1.0 is documented in reference [10, Appendix 4]. GD 2.0 now creates images using “full color” allowing for more realistic scene representations eliminating the color banding that occurred with the previous version because of the limited number (256) of colors used to represent images. Due to patent disputes, GD 2.0 has dropped support for the GIF file format and uses JPEG or PNG instead.

The `Render` sub-menus allow one to specify an integer between 1 and 20 indicating the number of frames between rendered images. This allows one to generate images encompassing the entire time duration of the simulation which in turn can be converted into movie files (`mpeg`, `mov`, `avi` etc) using software available on the internet. This is how movies were displayed <http://fire.nist.gov/smokeview> were generated. Rendering may be stopped by selecting `Cancel`.

Smokeyview scenes are now rendered using version 2.0 of the GD graphics library [10, Appendix 4]. This new library supports “full color” conversion allowing for more realistic scene representations. The former version of this library (used in `Smokeyview` version 1.0) allowed only 256 distinct colors to be saved into an image file. This resulted in banding or contouring in the saved image where none was present in the original display. Due to patent disputes the library GD 2.0 has dropped support for the GIF file format. Images in `Smokeyview` 2.0 are saved in either JPEG or PNG formats. The keyboard shortcut for the render option is `r`.

B.4.6 Viewpoint

This option allows one to save the location and orientation relative to the observer of the scene.

B.4.7 Font Size

This option allows one to display text in either a normal or a large font.

B.4.8 Aperture

This option allows one to change the perspective used to display the scene which is equivalent to changing the focal length of a camera lens.

B.4.9 Stereo 3-D

Provides stereo 3-D output on systems equipped with supported video cards. Stereo 3-D requires a computer graphics card, monitor, software drivers and shuttered glasses. This option only appears if `Smokeyview` is started from the command line using the `''-stereo''` command line option.

B.4.10 Set Bounds

Open the dialogue box for specifying data bounds.

B.4.11 Movement

Open the dialogue box for controlling scene movement. To use the movement arrows, click and hold the mouse in one of the arrows while simultaneously moving the mouse to achieve the desired movement.



Frame: 280
Time: 70.0

B.4.12 Edit Geometry

Open the dialogue box for editing FDS blockages.

C Keyboard Shortcuts

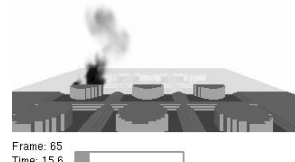
Many menu commands have equivalent keyboard shortcuts. These shortcuts are described here and are also briefly described under the Help menu item from within Smokeview itself.

- a** Alter the arrow size used to display velocity vectors.
- b** To increase the drawing speed, adjacent grid cells may be combined into coarser pseudo-grid cells when generating and viewing 3D contours. The “b” key is used to increment the number of grid cells that are combined (between one and five grid cells) when generating 3D contours. Using one grid will result in highly resolved contours but may take longer to draw. Using five grid cells will have the opposite effect, *i.e.* will result in coarse contours that can be drawn quickly.
- c** Toggle the Plot3D 2D contour display between solid banded contours and continuously shaded contours.
- e** Toggle how the scene is manipulated between an “eye view” where scene motion is relative to the observer and a “world view” where the scene motion is relative to the scene center.
- g** Toggle the grid visibility. When the grid display option is active, the x, y and z keys may be used to show or hide the grid perpendicular to the x, y and z axes respectively.
- i** Toggle the visibility of iso-contours (3D surface contours).
- o** Reset a time dependent animation to the initial time.
- p** Show the next variable in the Plot3D data set.
- r** Render the current scene as a JPEG or a PNG file which can be viewed in a web browser or inserted into a word processing document.
- s** Increment the number of vectors skipped. Useful for making vector displays more readable when grids are finely meshed.
- t** Toggle the time stepping mode. Time stepping mode allows one to step through the simulation one time step at a time.
- v** Toggle vector visibility. This option is only active when there are U, V and/or W velocity components present in the Plot3D data set.
- w** Toggle window clipping on and off. If window clipping is on then the x/X, y/Y, z/Z keys may be used to clip the scene along the min/max x, y and z axes respectively. The “<” and “>” keys are used to control the location of the selected clipping plane by decreasing or increasing the plane location respectively.
- x, y, z** Toggle the visibility of the Plot3D data planes perpendicular to the x, y and z axes respectively (parallel to the yz, xz and xy planes).

Left/Right Cursor Increment/decrement the Plot3D plane location displayed in the xz plane.

Page Up, Page Down Increment/decrement the Plot3D plane location displayed in the xy plane.

Up/Down Cursor Increment/decrement the Plot3D plane location displayed in the yz plane.



- Decrement Plot3D data planes, Plot3D iso-contour levels or time step displayed.
- space bar** Increment Plot3D data planes, Plot3D iso-contour levels or time step displayed.
- 1..9** Increment amount for time frame or Plot3D slice.

D File Formats

D.1 Smokeview Preference File Format (.ini files)

Smokeview uses preference files to set input parameters not settable using menus or the keyboard and to save the “state” of a visualization. Smokeview looks for preference files in three locations in the following order:

1. a file named `smokeview.ini` in a global directory defined by the `SMOKEVIEWINI` environment variable. On the PC, the directory `C:\nist\fds\smokeview.ini` is the default location for this preference file. The `SMOKEVIEWINI` environment variable may be defined on the PC by adding the line:


```
set SMOKEVIEWINI=dir
```

 to the `c:\autoexec.bat` file where `dir` is the directory where the global customization file is stored. This step is performed automatically by the Smokeview installation program. This environment variable may be defined on a UNIX workstation by adding the line:


```
setenv SMOKEVIEWINI dir
```

 to a `.login` or `.cshrc` start up file again where `dir` is the directory containing the global customization file. Changes to this `smokeview.ini` file apply to all cases visualized on the computer unless overridden by preference files named or located in directories named in steps 2. and 3.
2. a file named `smokeview.ini` in the directory containing the case being visualized. Changes to this `smokeview.ini` file apply to all cases in the current directory unless overridden by the `casename.ini` file contained in this directory.
3. a file named `casename.ini` in the directory containing the case being visualized where `casename` is the name of the case.

The `smokeview.ini` file may be created by typing:

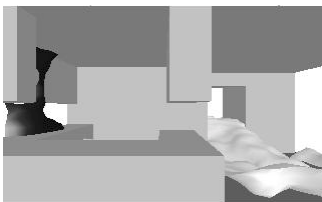
```
smokeview -ini
```

from the command line or by selecting the `Write smokeview.ini` menu item. The `casename.ini` preference file can be created *via* the menus or by copying a previously created `smokeview.ini` file.

Smokeview reads the global `smokeview.ini` file first (step 1. above), followed by the local `smokeview.ini` file (step 2. above), followed by the `casename.ini` file. The global `smokeview.ini` file is used to customize parameters for all Smokeview runs. The local `smokeview.ini` file is used to customize parameters for just those Smokeview runs contained in the local directory. The `casename.ini` file is used to customize parameters for only those Smokeview runs with the prefix `casename`.

All preference file parameters unless otherwise noted consist of a `KEYWORD` followed by a value, as in:

```
KEYWORD
value
```



Frame: 260
Time: 65.0

D.1.1 Color parameters

All colors are specified using a 3-tuple: r g b where r, g and b are the red, green and blue components of the color respectively. Each color component is a floating point number ranging from 0.0 to 1.0 where 0.0 is the darkest shade and 1.0 is the lightest shade. For example the 3-tuple 1.0 0.0 0.0 is bright red, 0.0 0.0 0.0 is black and 1.0 1.0 1.0 is white.

AMBIENTLIGHT Sets the color used for specifying ambient light. (default: .6 .6 .6)

BACKGROUNDCOLOR Sets the color used to visualize the scene background. (default: 0.0 0.0 0.0)

BLOCKCOLOR Sets the color used to visualize internal blockages. (default: 1.0 .8 4.0)

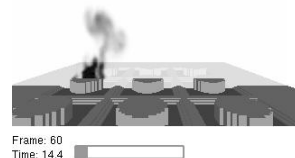
BOUNDCOLOR Sets the color used to visualize floors, walls and ceilings. (default: 0.5 0.5 0.2)

COLORBAR Entries for the color palette in rgb (red, green, blue) format where each color component ranges from 0.0 to 1.0 . The default values (rounded to 2 digits) are specified with:

```
COLORBAR
12
0.00 0.00 1.00
0.00 0.28 0.96
0.00 0.54 0.84
0.00 0.76 0.65
0.00 0.91 0.41
0.00 0.99 0.14
0.14 0.99 0.00
0.41 0.91 0.00
0.65 0.76 0.00
0.84 0.54 0.00
0.96 0.28 0.00
1.00 0.00 0.00
```

COLOR2BAR Miscellaneous colors used by Smokeview. The default values are specified using:

```
COLOR2BAR
8
1.0 1.0 1.0 :white
1.0 1.0 0.0 :yellow
0.0 0.0 1.0 :blue
1.0 0.0 0.0 :red
0.0 1.0 0.0 :green
1.0 0.0 1.0 :magenta
0.0 1.0 1.0 :cyan
0.0 0.0 0.0 :black
```



where the 8 indicates the number of colors defined and the character string after the ``:`` are ignored.

COLORBARFLIP Specifies whether the colorbar is flipped (1) or not flipped (0) (default: 0)

DIFFUSELIGHT Sets the color for specifying diffuse light (default: 0.5 0.5 0.5)

FLIP Specifies whether to flip (1) or not to flip (0) the foreground and background colors. By default the background color is black and the foreground color is white. Setting FLIP to 1 has the effect of having a white background and black foreground. (default: 0).

FOREGROUNDCOLOR Sets the color used to visualize the scene foreground (such as text labels). (default: 1.0 1.0 1.0)

ISOCOLORS Colors and parameters used to display animated isocontours. Default:

```
ISOCOLORS
10.000000 0.800000 : shininess, transparency
0.700000 0.700000 0.700000 : specular
3
0.960000 0.280000 0.000000
0.750000 0.800000 0.800000
0.000000 0.960000 0.280000
```

SENSORCOLOR Sets the color used to visualize sensors. (default: 1.0 1.0 0.0)

SETBW The parameter used to set whether color shades (0) or shades of grey (1) are to be used for coloring contours and blockages. (default: 0)

SPRINKOFFCOLOR Sets the color used to visualize sprinklers before they activate. (default: 1.0 0.0 0.0)

SPRINKONCOLOR Sets the color used to visualize sprinklers after they activate. (default: 0.0 1.0 0.0)

TIMEBARCOLOR Sets the color used to visualize the timebar. (default: 0.6 0.6 0.6)

VENTCOLOR Sets the color used to visualize vents. (default: 1.0 0.0 1.0)

D.1.2 Size parameters

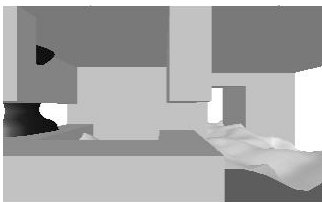
The parameters described in this section allow one to customize the size of various Smokeview scene elements.

LINEWIDTH Defines the width of lines³ in pixels. (default: 2.0)

PARTPOINTSIZE Defines the size in pixels of smoke or tracer particles. (default: 1.0)

SENSORABSSIZE Defines the sensor size drawn by smokeview using the same units as used to specify the grid coordinates. (default: 0.038)

³Except lines used to draw vents



- SLICEOFFSET** Defines an offset distance⁴ animated slices are drawn from adjacent solid surfaces. (default: 0.10)
- SMOOTHLINES** Specifies whether lines should be drawn (1) or not drawn (0) using anti-aliasing (default: 1).
- SPRINKLERABSSIZE** Defines the sprinkler size drawn by smokeview using the same units as used to specify the grid coordinates. (default: 0.076)
- VECTORLENGTH** Defines the length of Plot3D vectors. A vector length of 1.0 fills one grid cell. Vector lengths may also be changed from within Smokeview by depressing the “a” key. (default: 4.0)
- VECTORPOINTSIZE** Defines the size in pixels of the point that is drawn at the end of a Plot3D vector. (default: 3.0)
- VENTLINEWIDTH** Defines the width of lines used to draw vents in pixels. (default: 2.0)
- VENTOFFSET** Defines a distance used to offset vents drawn from adjacent surfaces. (default: 0.10 (units of fraction of a grid cell width))
- WINDOWHEIGHT** Defines the initial window height in pixels. (default: 480)
- WINDOWWIDTH** Defines the initial window width in pixels. (default: 640)

D.1.3 Time and value bound parameters

This section describes parameters used by Smokeview to 1) modify the time intervals used to load data (keywords beginning with T_) and 2) over-ride the minimum and maximum data values (keywords beginning with V_) used to convert data to color values. By default, Smokeview reads in data for all time values and does not over-ride minimum and maximum data values. Each time and data bound keyword (except for V_PLOT3D) has the format:

```
KEYWORD
minflag minvalue maxflag maxvalue
```

where minflag can be either 0 or 1. If it is 1 then, the subsequent number, minvalue is used by Smokeview to scale the data otherwise if minflag is 0 then minvalue is ignored. The next two parameters maxflag and maxvalue are defined similarly. The V_PLOT3D keyword contains data bound entries for each variable in the Plot3D data file. If a Plot3D “speed” variable was constructed by Smokeview then the V_PLOT3D keyword will contain six entries instead of five.

T_BOUNDARY Defines the minimum and maximum times used to include particles in a visualization. To load boundary data between 20 and 150 seconds use:

```
T_BOUNDARY
1 20. 1 150.
```

(default: 0 1.0 0 0.0)

T_PARTICLES Defines the minimum and maximum times used to include particles in a visualization. To load particle data between 20 and 150 seconds use:

⁴distance is relative to the maximum grid cell width




```
T_PARTICLES
1 20. 1 150.
```

T_SLICE Defines the minimum and maximum times used to include particles in a visualization. To load slice data between 20 and 150 seconds use:

```
T_SLICE
1 20. 1 150.
```

(default: 0 1.0 0 0.0)

V_BOUNDARY Defines the minimum and maximum data bounds used to convert boundary data values to color indices. (default: 0 1.0 0 0.0)

The **V_BOUNDARY** keyword has an optional parameter allowing one to specify which type of data the bounds should apply to. For example, to specify boundary file bounds for temperature (30.0 °C, 600.0 °C) use:

```
V_BOUNDARY
1 30.000000 1 600.000000 TEMP
```

where **TEMP** is the Smokeview colorbar labels displayed when showing the boundary file.

These suffixes are added automatically when the Bounds dialogue box is used to set data bounds.

V_PARTICLES Defines the minimum and maximum data bounds used to convert particle data values to color indices. (default: 0 1.0 0 0.0)

V_PLOT3D Defines the minimum and maximum data bounds used to convert Plot3D data values to color indices. The default values are given by:

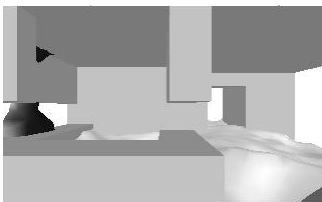
```
V_PLOT3D 5 1 0 1.0 0 0.0 2 0 1.0 0 0.0 3 0 1.0 0 0.0 4 0 1.0 0 0.0
5 0 1.0 0 0.0
```

where the initial 5 indicates the number of subsequent entries, the first integer on each line indicates the Plot3D data variable being specified and all other parameters on each line are defined as above.

To cause Smokeview to set the minimum and maximum data values to for the first quantity (usually temperature) to 20 and 600 use:

```
V_PLOT3D 5 1 1 20.0 1 600.0 2 0 1.0 0 0.0 3 0 1.0 0 0.0 4 0 1.0 0
0.0 5 0 1.0 0 0.0
```

The integer “1” occurring before the “20” or “600” causes Smokeview to use the next number



Frame: 220
Time: 55.0

as a minimum or maximum value respectively otherwise if “0” is specified then Smokeview ignores the subsequent min/max value.

In addition to 0 and 1, the `V_PLOT3D` keyword may use 2 as a bound indicator. In the above example, if 2 rather than 1 is used to define Plot3D bounds, then Smokeview does not draw contour⁵ levels smaller than 20 or contours greater than 600. The `PLOT3D` bound line `1 2 360.0 1 420.0` indicates that temperatures below 360 °C are not drawn and that temperatures above 420 °C are drawn with the “highest” color (black in black and white mode, red in color mode). This bound line could also be implemented with the “Set Bounds” dialog box as illustrated in Figure 10 resulting in a contour plot as illustrated in Figure 11.

V_SLICE Defines the minimum and maximum data bounds used to convert slice data values to color indices. (default: 0 1.0 0 0.0)

The `V_SLICE` keyword has an optional parameter allowing one to specify which type of data the bounds should apply to. For example, to specify separate slice file bounds for temperature (30.0 °C, 600.0 °C) and the U component of velocity (-1.0 m/s and 1.0 m/s) use:

```
V_SLICE
1 30.000000 1 600.000000 TEMP
V_SLICE
1 -1.0 1 1.0 U-VEL
```

where `TEMP` and `U-VEL` are the Smokeview colorbar labels displayed when showing the slice file.

These suffixes are added automatically when the Bounds dialogue box is used to set data bounds.

D.1.4 Data loading parameters

The keywords in this section may be used to reduce the memory required to visualize FDS data. Keywords exist for limiting particles and frames. Other keywords exist for compressing particle data and skipping particles and frames.

BOUNDFRAMESTEP Defines the number of intervals or steps between loaded boundary file frames. (default: 1)

ISOFRAMESTEP Defines the number of intervals or steps between loaded isosurface file frames (default: 1)

MAXFRAMES Specifies the maximum number of particle frames that can be read in. (default: 501)

MAXPOINTS Specifies the maximum number of particle points that can be read in from a particle file. (default: 5000000)

NOPART Indicates that a particle file should not (1) or should (0) be loaded when Smokeview starts up. This option is used when one wants to look at other files besides the particle file. (default: 0)

PARTFRAMESTEP Specifies the interval or steps between loaded particle frames. A particle frame is all the particles loaded at one particular time step. For example, if `PARTFRAMESTEP` is set to

⁵This is true for “stepped” or discrete contours. If “continuous” contours are drawn, then “2” and “1” have the same effect.



3 then every 3rd particle frame is read in. This and the `PARTPOINTSTEP` options may be used together to dramatically reduce the memory required to view particle files. These options should be used when displaying Smokeview files on computers with limited memory. (default: 1)

PARTPOINTCOMPRESS Specifies how Smokeview stores particle data. Each particle is represented using three position coordinates and one data value. Using full precision (`PARTPOINTCOMPRESS` value of 0), Smokeview uses four bytes per position coordinate and one byte for the data value or 13 bytes to represent each particle point. If this keyword is set to 1 then Smokeview uses one byte per position coordinate and one byte for the data value or 4 bytes per particle point. Finally if this keyword has value 2 then 2 bytes are used to store each position coordinate and one byte for the data value or 7 bytes per particle point. (default: 2) The recommended value is 2. Using 0 results in almost twice the memory usage without any extra precision in positioning particles. Using 1, though using less memory than the “2” option, results in granularity effects when displaying particles which one may find annoying. (default: 2)

PARTPOINTSTEP Specifies the interval or steps between loaded particle points. For example, if `PARTPOINTSTEP` is set to 5 then every 5th particle or only 20 per cent of the particle file is read in. (default: 1)

SLICEFRAMESTEP Specifies the number of interval or steps between loaded slice file frames. (default: 1)

D.1.5 Viewing parameters

The keywords in this section define how a scene is viewed. Keywords exist for showing or hiding various scene elements and for modifying how various scene elements appear.

AXISNUM Specifies whether Smokeview should (`AXISNUM` set to 1) or should not (`AXISNUM` set to 0) calculate data minimums and maximums based upon statistical limits. For example, if this parameter is set to 1 then Smokeview will set the minimum and maximum data bound to be the 1st and 99th percentile of the data respectively. (default: 1)

AXISSMOOTH Specifies whether axis numerical labels should be smoothed (`AXISSMOOTH` set to 1) or not smoothed (`AXISSMOOTH` set to 0). (default: 1)

CULLFACES Hide (1) or show (0) the back side of various surfaces.

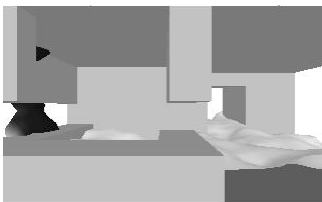
EYEVIEW Specifies whether the scene should be rotated relative to the observer (`EYEVIEW` set to 1) or the scene center (`EYEVIEW=0`). (default: 0)

EYEX, EYEX, EYEX The parameters `EYEX`, `EYEX`, `EYEX` specify the x , y and z location respectively of the viewing origin (where your eyes are). (default: 0.5 -0.9 1.5)

FRAMERATEVALUE Specifies the maximum rate (frames per second) that Smokeview should display frames. This value is an upper bound. Smokeview will not display frames faster than this rate but may display frames at a slower rate if the scene to be visualized is complex. (default: 1000000)

P3CONT2D The parameter `P3CONT2D` may be set to 0, 1 or 2. If `P3CONT2D` is set to 0 then Plot3D color contours are drawn by coloring each node and letting OpenGL interpolate colors between nodes. If `P3CONT2D` is set to 1 then discrete or stepped shaded contours are drawn. If `P3CONT2D` is set to 2 then contour lines are drawn. (default: 1)

P3DSURFACETYPE Specifies how Plot3D isosurfaces should be drawn. If `P3DSURFACETYPE` is set to 1 then Plot3D isosurfaces are drawn using shaded triangles. If `P3DSURFACETYPE` is



Frame: 200
Time: 50.0

set to 2 or 3 then Plot3D isosurfaces are drawn using triangle outlines and points respectively. (default: 1)

P3DSURFACESMOOTH When drawing Plot3D isosurfaces using shaded triangles, this option specifies whether the vertex normals should be averaged (P3DSURFACESMOOTH set to 1) resulting in smooth isosurfaces or not averaged resulting in isosurfaces that have sharp edges (P3DSURFACESMOOTH set to 0). (default: 1)

RENDERFILETYPE Specifies whether PNG (RENDERFILETYPE set to 0) or JPEG (RENDERFILETYPE set to 1) should be used to render images. (default: 1)

SHOWBLOCKS Specifies how a blockage should be drawn. A value of 0, 1 or 2 indicates that the blockages are invisible, drawn normally or drawn as outlines respectively. (default: 1)

SHOWCEILING Specifies whether the ceiling (upper bounding surface) should be drawn (1) or not drawn (0). (default: 0)

SHOWCOLORBARS Specifies whether the color bars should be drawn (1) or not drawn (0) (default: 1)

SHOWFLOOR Specifies whether the floor (lower bounding surface) should be drawn (1) or not drawn (0) (default: 1)

SHOWFRAMERATE Specifies whether the frame rate label should be drawn (1) or not drawn (0). (default: 0)

SHOWSMOKEPART Specifies whether smoke or trace particles should be drawn (1) or not drawn (0) (default: 1)

SHOWSPRINKPART Specifies whether sprinkler droplet particles (if present in the particle file) should be drawn (1) or not drawn (0). (default: 1)

SHOWTIMEBAR Specifies whether the timebar should be drawn (1) or not drawn (0). (default: 1)

SHOWTITLE Specifies whether the title should be drawn (1) or not drawn (0). (default: 1)

SHOWWALLS Specifies whether the four walls (four vertical bounding surfaces) should be drawn (1) or not drawn (0). (default: 1)

SURFINC If not zero, this parameter specifies the increment between displayed Plot3D surface values. That is, two Plot3D isosurfaces are displayed and the SURFINC value specifies the interval between the two surfaces. (default: 0)

TRANSPARENT Specifies whether 2D and 3D contours should be drawn with solid colors (0) or transparent colors(1). (default: 1)

UNIT This keyword defines how to calculate colorbar label values for non-default units. Note that in the following example all text to the right of the ‘//’ characters is not part of the keyword but is used to explain the format.
default:

```
UNIT
temperature // text appearing in smokeview unit menu
1           // number of FDS variables that have temperature units
temp       // FDS variable, appears in colorbar heading
3 1       // number of units, default unit number
C         // first unit
1.0 0.0   // newunitval = 1.0*oldunitval + 0.0
```

```

F          // second unit
1.8 32.0
K          // third unit
1.0 273.15

UNIT
velocity  // text appearing in smokeview unit menu
4         // number of FDS variables that have velocity units
U-VEL    // first variable
V-VEL    // second variable
W-VEL    // third variable
Speed    // fourth variable
3 1      // number of units, default unit number
m/s
1.0 0.0
mph
2.236932 0.0
f/s
3.280833 0.0

```

USEVIEWPOINT Specifies whether the stored viewpoint parameters should (1) or should not (0) be used to restore the viewpoint of a scene. (default: 0)

VECTORSKIP Specifies what vectors to draw. For example, if this parameter is set to 2 then every 2nd vector is drawn when displaying vectors. (default: 1)

VIEWPOINT Specifies the internal Smokeview parameters used to record a scene's viewpoint and orientation. This parameter is set automatically by Smokeview when a .ini file is created. (default: none)

D.2 Smokeview Parameter Input File (.smv file)

The FDS software outputs simulation results into the Smokeview input file with extension .smv and various output data files whose format is documented in the next section. A .smv file is a formatted ascii file (*ie.* a regular text file) consisting of a set of KEYWORD followed by DATA describing an FDS case's geometry, file name and contents, sensor information *etc.*

D.2.1 Geometry Keywords

GRID This keyword specifies the number of grid cells in the X, Y and Z directions. For example,

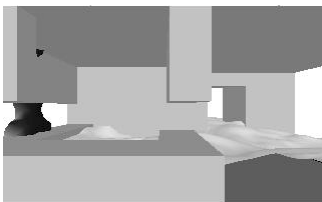
```

GRID
10 20 30

```

specifies that there are 10, 20 and 30 grid cells in the X, Y and Z directions respectively.

PDIM This keywords specifies the physical length of the simulation domain in the X, Y and Z directions. For example,



Frame: 180
Time: 45.0

```
PDIM
6.0 12.0 5.0
```

specifies that the length of the X, Y and Z directions is 6.0, 12.0 and 5.0 m respectively.

HIDE_OBST This keyword specifies when a blockage should be opened. For example,

```
HIDE_VENT 10 120.1
```

specifies that the tenth blockage should be opened at 120.1 seconds. This keyword is automatically added to the .smv file by FDS version 2.0. This (and the OPEN_VENT, CLOSE_VENT) keyword may be added by hand to FDS version 1.0 files.

OBST The keyword specifies internal blockages. A FORTRAN 90 code segment describing the format of OBST data is given by:

```
read(5,*)nb
do i = 1, nb
  read(5,*)dummy
end do
do i = 1, nb
  read(5,*)ib1(i), ib2(i), jb1(i), jb2(i), kb1(i), kb2(i)
end do
```

where ib1, ib2, jb1, kb1, kb2 are integer arrays containing boundary nodes.

TRNX,TRNY,TRNZ The TRNX, TRNY, TRNZ keywords specify grid nodes in the X, Y, Z coordinate directions. A FORTRAN 90 code segment describing the format of TRNX data is given by:

```
read(5,*)nx
do i = 1, nx
  read(5,x)idummy
end do
do i = 1, nx
  read(5,*)xplt(i)
end do
```

TRNY and TRNZ data entries are defined similarly. The first nx data items are not required by Smokeview.

VENT The keyword specifies vent coordinates. A FORTRAN 90 code segment describing the format of VENT data is given by:



```

read(5,*)nv
do i = 1, nv
  read(5,*)dummy
end do
do i = 1, nv
  read(5,*)iv1(i), iv2(i), jv1(i), jv2(i), kv1(i), kv2(i)
end do

```

where *iv1*, *iv2*, *jv1*, *kv1*, *kv2* are integer arrays containing vent coordinates.

OPEN_VENT(CLOSE_VENT) These keywords specifies when a vent should be opened(closed). For example,

```

CLOSE_VENT
3 15.6

```

specifies that the third vent should be closed at 15.6 seconds.

XYZ The XYZ keyword defines the .xyz or Plot3D grid file name. A FORTRAN 90 code segment describing the format of XYZ data is given by:

```

read(5,"(a)")xyzfilename

```

where *xyzfilename* is a character variable containing the name of the .xyz file.

D.2.2 File Keywords

BNDF The BNDF keyword defines the .bf file name along with character labels used to describe the data contents of the boundary file. A FORTRAN 90 code segment describing the format of BNDF data is given by:

```

read(5,"(a)")longlabel
read(5,"(a)")shortlabel
read(5,"(a)")unit

```

where *longlabel*, *shortlabel*, *unit* are character variables of length 30.

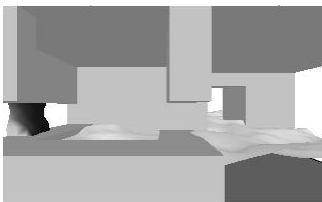
INPF The INPF keyword specifies a file containing a copy of the FDS input file.

ISOF The ISOF keyword defines the .iso file name along with character labels used to describe the data contents of the isosurface file. A FORTRAN 90 code segment describing the format of ISOF data is given by:

```

read(5,"(a)")longlabel

```



Frame: 160
Time: 40.0

```
read(5,"(a)")shortlabel  
read(5,"(a)")unit
```

where `longlabel`, `shortlabel`, `unit` are character variables of length 30.

PART The **PART** keyword defines the `.part` file name along with character labels used to describe the data contents of the particle file. A FORTRAN 90 code segment describing the format of **PART** data is given by:

```
read(5,"(a)")longlabel  
read(5,"(a)")shortlabel  
read(5,"(a)")unit
```

where `longlabel`, `shortlabel`, `unit` are character variables of length 30.

PL3D The **PL3D** keyword defines the `.q` file name along with character labels used to describe the data contents for each Plot3D variable. A FORTRAN 90 code segment describing the format of **PL3D** data is given by:

```
do i = 1, 5  
  read(5,"(a)")longlabel(i)  
  read(5,"(a)")shortlabel(i)  
  read(5,"(a)")unit(i)  
end do
```

where the character arrays `longlabel`, `shortlabel`, `unit` each have length 30.

SLCF The **SLCF** keyword defines the `.sf` file name along with character labels used to describe the data contents of the slice file. A FORTRAN 90 code segment describing the format of **SLCF** data is given by:

```
read(5,"(a)")longlabel  
read(5,"(a)")shortlabel  
read(5,"(a)")unit
```

where `longlabel`, `shortlabel`, `unit` are character variables of length 30.

D.2.3 Sensor Keywords

HEAT The **HEAT** keyword defines heat detector location data. A FORTRAN 90 code segment describing the format of **HEAT** data is given by:

```
read(5,*)nheat
```



```

do i = 1, nheat
  read(5,*)xheat(i),yheat(i),zheat(i)
end do

```

where `nheat` is the number of heat detectors and `xheat`, `yheat`, `zheat` are the `x`, `y`, `z` coordinates of the heat detectors.

HEAT_ACT The `HEAT_ACT` keyword defines heat detector activation data. A FORTRAN 90 code segment describing the format of `HEAT_ACT` data is given by:

```

read(5,*)iheat, heat_time

```

where `heat_time` is the activation time of the `iheat`'th heat detector.

SPRK The `SPRK` keyword defines sprinkler location data. A FORTRAN 90 code segment describing the format of `SPRK` data is given by:

```

read(5,*)nsprink
do i = 1, nsprink
  read(5,*)xsprink(i),ysprink(i),zsprink(i)
end do

```

where `nsprink` is the number of sprinklers and `xsprink`, `ysprink`, `zsprink` are the `x`, `y`, `z` coordinates of the sprinklers.

SPRK_ACT The `SPRK_ACT` keyword defines sprinkler activation data. A FORTRAN 90 code segment describing the format of `SPRK_ACT` data is given by:

```

read(5,*)isprink, sprink_time

```

where `sprink_time` is the activation time of the `isprink`'th sprinkler.

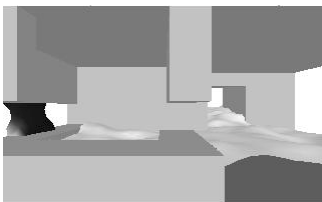
THCP The `THCP` keyword defines thermocouple location data. A FORTRAN 90 code segment describing the format of `THCP` data is given by:

```

read(5,*)ntherm
do i = 1, ntherm
  read(5,*)xtherm(i),ytherm(i),ztherm(i)
end do

```

where `ntherm` is the number of thermocouples and `xtherm`, `ytherm` and `ztherm` are the `x`, `y` and `z` coordinates of the thermocouples.



D.2.4 Miscellaneous Keywords

ENDIAN The **ENDIAN** keyword defines whether a big or little endian⁶ system is used to represent the data. The PC (both Windows and LINUX) uses the little endian system for representing numbers while most UNIX workstations use the big endian system. The PC version of Smokeview uses this keyword to determine whether data should be converted between these two systems before it is visualized. The valid **ENDIAN** parameters are **BIG** or **LITTLE**.

SYST The **SYST** keyword defines which system the FDS case was run on. The PC version of Smokeview uses this information (if the **ENDIAN** keyword is absent) to swap data bytes if the FDS case was run on a computer with a different “endian” format than the PC.

TITLE1/TITLE2 The **TITLE1** and **TITLE2** keywords allow one to specify extra information documenting a Smokeview case. These keywords and associated labels are added by hand to the smokeview (.smv) file using the format:

```
TITLE1
first line of descriptive text
TITLE2
second line of descriptive text
```

D.3 Data File Formats (.iso, .part, .sf, .bf and .q files)

Smokeview visualizes fire dynamics data using isosurface, particle, boundary, slice and Plot3D data files described in this section along with a Smokeview parameter input file described in the previous section. This section documents the data format of each data file by showing FORTRAN program code segments used to write the data out. All files are unformatted and are opened using **OPEN** statements of the form:

```
OPEN (UNIT=IUNIT, FILE=FILENAME, FORM="UNFORMATTED")
```

where **IUNIT** is the FORTRAN unit number used for output and **FILENAME** is a character variable containing the file name. In the following code segments the standard FORTRAN naming convention of associating variables beginning with I through N to integer variables and all other alphabetic characters to **REAL** variables is used. It is assumed that all variables take up four bytes. Variables having **DUMMY** in the name indicates that they are not used by Smokeview.

D.3.1 Isosurface File Format

Isosurface files are used to store one or more surfaces where each surface represents a region where the solution take one a particular value. The FDS software outputs isosurface frames at fixed time intervals. A “pseudo-code” representation of the isosurface file is given by:

```
version                               (int)
len1, len2, len3                       (3*int)
label1, label2, label3                 ((len1+len2+len3+4)*char)
```

⁶Big-endian and little-endian are terms that describe the order in which a sequence of bytes are stored in computer memory. Big-endian is an order in which the *big end* (most significant value in the sequence) is stored first (at the lowest storage address). Little-endian is an order in which the *little end* (least significant value in the sequence) is stored first. The terms big-endian and little-endian are derived from the Lilliputians of Gulliver’s Travels, whose major political issue was whether soft-boiled eggs should be opened on the big end or the little end.

```

nlevels                                (int)
level_1, level_2, ..., level_nlevels  (nlevels*float)
for each time:
  time                                  (float)
  for each level
    nvertices                           (int)
    ntriangles                          (int)
    vertices_1, ..., vertex_nvertices  (3*short*nvertices)
    triangles_1, ..., triangle_ntriangles (3*short*ntriangles)
  end level
end time

```

D.3.2 Particle File Format

Particle files are used to store smoke/tracer particles and/or sprinkler droplet information. The particle file consists of a header and a series of particle frames. Each frame contains particle/droplet positions and data values for each particle in that frame. The FDS software outputs particle frames at fixed time intervals but there is no requirement by Smokeview for this to happen. Smokeview reads in the data one frame at a time noting the time read in for each frame. The header can be described using the code segment:

```

WRITE(IUNIT) DUMMY, DUMMY, DUMMY, IPART, NPPS
WRITE(IUNIT) IBAR, JBAR, KBAR
WRITE(IUNIT) (DUMMY, I=0, IBAR), (DUMMY, J=0, JBAR), (DUMMY, K=0, KBAR)
WRITE(IUNIT) NB
DO N=1, NB
  WRITE(IUNIT) IB1(N), IB2(N), JB1(N), JB2(N), KB1(N), KB2(N), 1
END DO
WRITE(IUNIT) NV
DO N=1, NV
  WRITE(IUNIT) IV1(N), IV2(N), JV1(N), JV2(N), KV1(N), KV2(N), 2
END DO
WRITE(IUNIT) NSPR
DO N=1, NSPR
  WRITE(IUNIT) XSP0(N), YSP0(N), ZSP0(N)
END DO

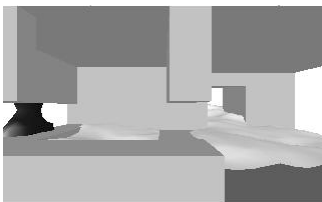
```

where DUMMY is written by FDS but not needed by Smokeview, IPART is the index of the scalar quantity associated with the particles, NPPS is the maximum number of particles per frame, IB1, IB2, etc are the indices of blocked grid cells, IV1, IV2, etc indicate vent cell nodes, and XSP0, YSP0, ZSP0 are the coordinates of the sprinklers. Grid coordinates are obtained by Smokeview from the Smokeview parameter input file. A particle frame is written using:

```

WRITE(IUNIT) TIME, NP, IDUMMY, (ISPR(N), N=1, NSPR)
WRITE(IUNIT) (XP(I), I=1, NP),
.           (YP(I), I=1, NP),
.           (ZP(I), I=1, NP),
.           (BP(I), I=1, NP)
IF (NASPR.GT.0) THEN

```



Frame: 120
Time: 30.0

```

WRITE(IUNIT) NSP
WRITE(IUNIT) (XSP(I), I=1, NSP), (YSP(I), I=1, NSP), (ZSP(I), I=1, NSP)
ENDIF

```

where NP is the number of particles in this frame, ISPR denotes whether the sprinkler has activated, NSPR is the number of sprinklers, XP, YP, ZP are the particle coordinates, BP is the particle value, NASPR is the number of active sprinklers, NSP is the number of sprinkler droplets, and XSP, YSP, ZSP are the droplet coordinates.

D.3.3 Slice File Format

Slice files are unformatted. The slice file header is written out using:

```

WRITE(IUNIT) CDUMMY
WRITE(IUNIT) CDUMMY
WRITE(IUNIT) CDUMMY
WRITE(IUNIT) I1, I2, J1, J2, K1, K2

```

where CDUMMY is character strings of length 30. Smokeview obtains this information from the Smokeview parameter input file and not from the slice file. The sextuple (I1, I2, J1, J2, K1, K2) denotes the bounding grid cell nodes of the slice to be displayed. The sextuple indices correspond to grid cell nodes, or corners, thus the entire grid would be represented by the sextuple (0, IBAR, 0, JBAR, 0, KBAR). The grid node positions are obtained from the Smokeview parameter input file.

The FDS software outputs slice frames at fixed time intervals but there is no requirement by Smokeview for this to happen. Each slice frame is written using:

```

WRITE(IUNIT) TIME
WRITE(IUNIT) ((QQ(I, J, K), I=I1, I2), J=J1, J2), K=K1, K2)

```

where TIME is the time in seconds when the data is output and QQ are the data values.

D.3.4 Boundary Files

The boundary files are unformatted. The header is written out using:

```

WRITE(IUNIT) QUANTITY
WRITE(IUNIT) SHORT_NAME
WRITE(IUNIT) UNITS
WRITE(IUNIT) NPATCH
DO 10 I=1, NPATCH
    WRITE(IUNIT) IP1(I), IP2(I), JP1(I), JP2(I), KP1(I), KP2(I)
10 CONTINUE

```

where QUANTITY, SHORT_NAME and UNITS are character strings of length 30, NPATCH is the number of planes (or “patches”) that make up the solid boundaries plus the external walls. The sextuple arrays (IP1, IP2, JP1, JP2, KP1, KP2) defines the cell nodes of each patch. The user does not prescribe these.

The FDS software outputs boundary frames at fixed time intervals but there is no requirement by Smokeview for this to happen. The boundary file frame is written using:

```

WRITE(IUNIT) TIME
DO 10 I=1, NPATCH
    WRITE(IUNIT) ((QQ(I, J, K), I=IP1(I), IP2(I)), J=JP1(I), JP2(I)), K=KP1(I), KP2(I)
10 CONTINUE

```

D.3.5 Plot3D Data

Field data is dumped periodically by FDS in a format used by the graphics package **Plot3D**[11]. The **Plot3D** data sets are single precision (32 bit reals), whole and unformatted. Note that there is blanking, that is, blocked out data points are not plotted. The grid data is written out to a file called `casename.xyz` using:

```
WRITE(IUNIT)  IBAR+1,JBAR+1,KBAR+1
WRITE(IUNIT)  ((X(I),I=0,IBAR),J=0,JBAR),K=0,KBAR),
.              ((Y(J),I=0,IBAR),J=0,JBAR),K=0,KBAR),
.              ((Z(K),I=0,IBAR),J=0,JBAR),K=0,KBAR),
.              (((IBLK(I,J,K),I=0,IBAR),J=0,JBAR),K=0,KBAR)
```

where `IBAR+1`, `JBAR+1`, `KBAR+1` are the number of grid cells along the `I`, `J`, `K` coordinate directions, `X`, `Y` and `Z` are the physical coordinates of the cell corners, and `IBLK` is an indicator of whether or not the cell is blocked. If the point `(X, Y, Z)` is completely embedded within a solid region, then `IBLK` is 0. Otherwise, `IBLK` is 1. The flow variables are written to a file using:

```
WRITE(IUNIT)  IBAR+1,JBAR+1,KBAR+1
WRITE(IUNIT)  ZERO,ZERO,ZERO,ZERO
WRITE(IUNIT)  (((Q(I,J,K,N),I=0,IBAR),J=0,JBAR),K=0,KBAR),N=1,5)
```

The five channels `N=1, 5` are by default the temperature (C), the u , v and w components of the velocity (m/s), and the heat release rate per unit volume (kW/m^3).

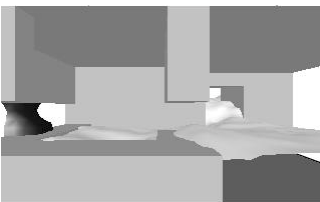
E Sample FDS Input Files

E.1 Plume Case

```
&HEAD CHID='plume',TITLE='Single Fire Plume' /
&GRID IBAR=32,JBAR=32,KBAR=72 /
&PDIM XBAR=1.5,YBAR=1.5,ZBAR=3.5 /
&TRNX IDERIV=0,CC=0.75,PC=0.75 /
&TRNX IDERIV=1,CC=0.75,PC=0.5 /
&TRNY IDERIV=0,CC=0.75,PC=0.75 /
&TRNY IDERIV=1,CC=0.75,PC=0.5 /
&TIME DT=0.05,TWFIN=20. /

&SURF ID='burner',HRRPUA=600.,TBO=0.20 /
&OBST XB=0.65,0.85,0.65,0.85,0.00,0.15 /
&VENT XB=0.65,0.85,0.65,0.85,0.15,0.15,SURF_ID='burner' /
&VENT CB='XBAR',SURF_ID='OPEN' /
&VENT CB='XBAR0',SURF_ID='OPEN' /
&VENT CB='YBAR',SURF_ID='OPEN' /
&VENT CB='YBAR0',SURF_ID='OPEN' /
&VENT CB='ZBAR',SURF_ID='OPEN' /

&PART QUANTITY='TEMPERATURE',NPSAM=5 /
&ISOF QUANTITY='MIXTURE_FRACTION',VALUE(1)=0.03,VALUE(2)=0.001 /
&PL3D DTSAM=5. /
```



Frame: 100
Time: 25.0

```

&SLCF PBX=0.75,QUANTITY='DENSITY',VECTOR=.TRUE. /
&SLCF PBY=0.75,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=0.75,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=1.50,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=2.50,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=3.00,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /

```

E.2 Townhouse Case

```

&HEAD CHID='thouse',TITLE='Town House Kitchen Fire' /
&GRID IBAR=48,JBAR=64,KBAR=36 /
&PDIM XBAR=6.4,YBAR=8.0,ZBAR=4.8 /
&TIME TWFIN=180.0 /
&MISC TMPA=20.,SURF_DEFAULT='GYPSUM BOARD',NFRAMES=720 /

```

```

&SURF ID='GYPSUM BOARD'
      HRRPUA=100.
      RAMP_Q='GB'
      KS=0.48
      ALPHA=4.1E-7
      DELTA=0.013
      TMPIGN=400. /

```

```

&RAMP ID='GB',T= 0.0,F=0.0 /
&RAMP ID='GB',T= 1.0,F=0.5 /
&RAMP ID='GB',T= 2.0,F=1.0 /
&RAMP ID='GB',T=10.0,F=1.0 /
&RAMP ID='GB',T=20.0,F=0.0 /
&RAMP ID='GB',T=30.0,F=0.0 /

```

```

&SURF ID='THINPINE'
      HRRPUA=200.
      KS=0.14
      ALPHA=8.3E-8
      DELTA=0.008
      TMPIGN=390. /

```

```

&SURF ID='BURNER',HRRPUA=1000. /

```

```

&VENT XB=0.0,0.6,1.0,2.00,0.9,0.9,SURF_ID='BURNER' /
&VENT XB=3.4,4.2,0.0,0.00,3.2,3.4,SURF_ID='OPEN' /
&VENT XB=1.2,2.0,0.0,0.00,3.2,3.4,SURF_ID='OPEN' /
&VENT XB=3.0,3.8,8.0,8.00,3.2,3.4,SURF_ID='OPEN' /
&VENT XB=1.2,2.0,8.0,8.00,3.2,3.4,SURF_ID='OPEN' /
&VENT XB=4.0,4.8,8.0,8.00,0.0,0.2,SURF_ID='OPEN' /
&VENT XB=4.0,4.8,8.0,8.00,0.2,2.2,T_OPEN=60.,SURF_ID='OPEN' /

```

```

&OBST XB=0.00,1.08,0.00,0.23,1.50,2.4, SURF_ID='THINPINE' /
&OBST XB=0.00,0.23,0.25,2.75,1.50,2.4, SURF_ID='THINPINE' /
&OBST XB=0.00,3.63,2.75,3.75,0.00,2.4 /

```

```

&OBST XB=2.533330, 2.933330, 0.000000, 1.500000, 1.466670, 2.400000, SURF_ID='THINPINE' /
&OBST XB=2.40,3.10,0.00,1.75,0.00,0.9, SURF_ID='THINPINE' /
&OBST XB=0.00,2.40,0.00,0.75,0.00,0.9, SURF_ID='THINPINE' /
&OBST XB=0.00,0.60,0.75,2.75,0.00,0.9, SURF_ID='THINPINE' /
&OBST XB=3.466670, 4.933330, 3.500000, 3.750000, 2.000000, 2.400000/
&OBST XB=4.933330, 5.200000, 3.500000, 5.500000, 0.000000, 2.400000/
&OBST XB=5.2,6.4,6.50,6.75,0.00,0.20/
&OBST XB=5.2,6.4,6.25,6.50,0.00,0.40/
&OBST XB=5.2,6.4,6.00,6.25,0.00,0.60/
&OBST XB=5.2,6.4,5.75,6.00,0.00,0.80/
&OBST XB=5.2,6.4,5.50,5.75,0.00,1.00/

&OBST XB=5.2,6.4,5.25,5.50,0.00,1.20/
&OBST XB=5.2,6.4,5.00,5.25,0.00,1.40/
&OBST XB=5.2,6.4,4.75,5.00,0.00,1.60/
&OBST XB=5.2,6.4,4.50,4.75,0.00,1.80/
&OBST XB=5.2,6.4,4.25,4.50,0.00,2.00/
&OBST XB=5.2,6.4,4.00,4.25,0.00,2.20/
&OBST XB=5.2,6.4,3.50,4.00,0.00,2.50/
&OBST XB=5.2,6.4,2.25,3.50,2.40,4.8/
&OBST XB=4.933330, 5.200000, 4.250000, 8.000000, 2.400000, 4.800000/
&OBST XB=5.2,6.4,6.25,8.00,2.40,4.8/
&OBST XB=5.2,6.4,6.00,6.25,2.60,4.8/
&OBST XB=5.2,6.4,5.75,6.00,2.80,4.8/
&OBST XB=5.2,6.4,5.50,5.75,3.00,4.8/
&OBST XB=5.2,6.4,5.25,5.50,3.20,4.8/
&OBST XB=0.000000, 5.200000, 0.000000, 8.000000, 2.400000, 2.666670/
&OBST XB=2.533330, 2.800000, 0.000000, 2.500000, 2.666670, 4.800000/
&OBST XB=0.000000, 4.000000, 3.500000, 4.500000, 2.666670, 4.800000/
&OBST XB=2.800000, 3.866670, 2.250000, 2.500000, 2.666670, 4.800000/
&OBST XB=5.200000, 6.400000, 0.000000, 2.250000, 2.400000, 2.666670/
&OBST XB=4.000000, 4.933330, 4.250000, 4.500000, 4.400000, 4.800000/
&OBST XB=3.866670, 4.000000, 2.250000, 3.500000, 4.400000, 4.800000/
&OBST XB=4.00,5.20,2.25,2.50,4.40,4.8/

&ISOF QUANTITY='MIXTURE_FRACTION',VALUE(1)=0.03,VALUE(2)=0.001 /

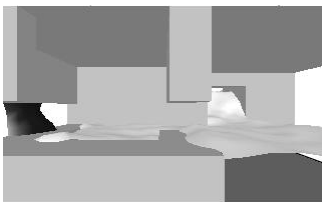
&PART QUANTITY='TEMPERATURE',NPSAM=5 /

&PL3D DTSAM=30. /

&BNDF QUANTITY='WALL_TEMPERATURE' /

&SLCF PBX=1.5,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=2.2,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBZ=4.6,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBX=4.4,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /
&SLCF PBX=5.8,QUANTITY='TEMPERATURE',VECTOR=.TRUE. /

```



Frame: 80
Time: 20.0

F Frequently Asked Questions

This appendix contains a list of frequently asked questions concerning Smokeview. An updated list may be found at <http://fire.nist.gov/smokeview>.

F.1 How can I keep up with new releases and other information about FDS and Smokeview?

An e-mail list has been set up to alert users to changes, upgrades, *etc.* To subscribe to the list, send an email to listproc@nist.gov and put the line

```
subscribe fdsusers Jane Doe
```

in the text where Jane Doe is your name. Subject line is unimportant. To get off the list, send an email to the same address and write

```
unsubscribe fdsusers
```

Also, you can check the web page <http://fire.nist.gov/download> to see the date of the most recent update.

F.2 Smokeview doesn't look right on my computer. What's wrong?

Some video cards use too few bits to represent the "depth buffer". The depth buffer is used to determine when objects are hidden and therefore should not be drawn. A problem then can occur when the video card reduces the number of depth buffer bits from 32 to 16 in order to speed up its performance (less data to transport). This results in straight lines or polygonal edges appearing to be scalloped. Often reducing the acceleration setting for the video card can eliminate this problem. To do this:

- click your mouse on the desktop to bring up the "Display Properties" dialog box.
- click on the "Advanced..." button
- Find the dialog panel that lets you select the "Hardware acceleration". Reduce the acceleration (re-booting if necessary) until the scalloping disappears.

If you have upgraded your PC recently, make sure you have loaded the latest drivers for your video card.

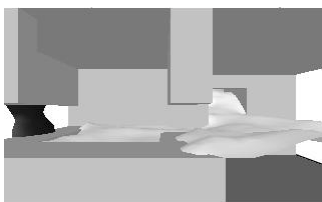
F.3 How do I make a movie of a Smokeview animation?

You make a movie of a Smokeview animation by converting the visualized scene into a series PNG or JPEG files, one file for each time. More specifically:

- Set up Smokeview orienting the scene and loading the desired data files.
- Select the Options/Render menu and pick the desired frame rate. The more frames you include in the animation, the smoother it will look. Of course more frames results in larger file sizes.
- Use a program such as the Antechinus Media Editor or Platypus Animator available at <http://www.c-point.com>

References

- [1] G. P. Forney and K. McGrattan. User's Guide for Smokeview Version 1.0: A Tool for Visualizing Fire Dynamics Simulation Data. NISTIR 6513, National Institute of Standards and Technology, 1999.
- [2] K.B. McGrattan and G.P. Forney. Fire Dynamics Simulator Version 2.0- User's Guide. Technical Report NISTIR xxxx, National Institute of Standards and Technology, Gaithersburg, Maryland, March 2001.
- [3] K.B. McGrattan, H.R. Baum, R.G. Rehm, A. Hamins, and G.P. Forney. Fire Dynamics Simulator - Technical Reference Manual. Technical Report NISTIR 6467, National Institute of Standards and Technology, Gaithersburg, Maryland, January 2000.
- [4] K.B. McGrattan and G.P. Forney. Fire Dynamics Simulator - User's Guide. Technical Report NISTIR 6469, National Institute of Standards and Technology, Gaithersburg, Maryland, January 2000.
- [5] K.B. McGrattan, H.R. Baum, and R.G. Rehm. Large Eddy Simulations of Smoke Movement. *Fire Safety Journal*, 30:161–178, 1998.
- [6] D. Madrzykowski and R.L. Vettori. Simulation of the Dynmaics of the Fire at 3146 Cherry Road NE, Washington, DC May 30, 1999. Technical Report NISTIR 6510, Gaithersburg, Maryland, April 2000.
- [7] Mason Woo, Jackie Neider, Tom Davis, and Dave Shreiner. *OpenGL Programming Guide*. Addison-Wesley, Reading, Massachussets, 3 edition, 1999.
- [8] Richard S. Wright Jr. and Michael Sweet. *OpenGL Super Bible*. Waite Group Press, Indianapolis, Indiana, 2 edition, 2000.
- [9] Mark J. Kilgard. *OpenGL Programming for the X Window System*. Addison-Wesley Developers Press, Reading, Massachussets, 1996.
- [10] Thomas Boutell. *CGI Programming in C & Perl*. Addison-Wesley Publishing Co., Reading, Massachussets, 1996.
- [11] Pamela P. Walatka and Pieter G. Buning. PLOT3D User's Manual, version 3.5. NASA Technical Memorandum 101067, 1989.



Frame: 60
Time: 15.0

Acknowledgements

The authors are grateful to the many people who have contributed to this work providing comments and suggesting improvements, many of which have been incorporated into version 2.0 of Smokeview. Further suggestions or comments are encouraged and may be sent to glenn.forney@nist.gov or to the web site with URL: <http://fire.nist.gov/smokeview>.