This document has been superseded by a later version.   For the latest version go to the web site: http://fire.nist.gov/fds

User's Guide for Smokeview Version 1.0 - A Tool For Visualizing Fire Dynamics Simulation Data

Glenn P. Forney
Kevin B. McGrattan

User's Guide for Smokeview Version 1.0 - A Tool For Visualizing Fire Dynamics Simulation Data

Glenn P. Forney
Kevin B. McGrattan

Building and Fire Research Laboratory

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 analyses performed using these tools.

Users are warned that Smokeview 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. The software package is a computer model that may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions by the model 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

List of Figures

User's Guide for Smokeview Version 1.0 - A Tool for Visualizing Fire Dynamics Simulation Data

Glenn Forney
Kevin B. McGrattan

Abstract

Smokeview is a software tool designed to visualize numerical predictions generated by the NIST Fire Dynamics Simulator (FDS). FDSis a computational fluid dynamics (CFD) model of fire-driven fluid flow. These two tools are used in concert to respectively model and analyze scenarios involving features such as fire, wind, sprinklers, complex geometry *etc*. Smokeview visualizes FDS modeling results in many ways, some of which are 1) animating flow of smoke or tracer particles, 2) animating 2D contours of data slices, 3) plotting 2D and 3D static contours or 4) plotting flow vectors .

Smokeview requires a fast CPU, fast video card, a substantial amount of random-access memory (RAM) and a large hard disk. Smokeview is portable and can be run on a variety of computing platforms. For a Windows-based PC, the processor should be at least a 450 MHz Pentium II, with at least 256 Mbytes of memory. Of course, more is better, but this configuration should allow one to do fairly large calculations in a reasonable time period. For UNIX-based workstations, the processor and memory should be at least as fast and as large as the PC specs.

Smokeview and FDS may be obtained at the web site **http://fire.nist.gov/smokeview** or by contacting the authors at `glenn.forney@nist.gov` or `kevin.mcgrattan@nist.gov`.

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. FDS[1, 2, 3] and Smokeview are used in concert to respectively model and visualize fire scenarios of interest. Smokeview performs this visualization by presenting: smoke or tracer particle flow (particle files); animated (in time) contour slices of computed gas quantities (slice files) and surface data (boundary files); and static contour and vector plots (PLOT3D files) anywhere within a simulation scene at a single fixed time.

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 locations *etc*. in order to set up correct FDS input files more quickly.

Figure1 illustrates how the 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. Reference [2] gives documentation on FDS input data files.

2. Run FDS. The fire model FDS then creates one or more output files with extensions `.smv`, `.part`, `.sf`, `.q`, `.bf` .

3. Run Smokeview to analyze the output files generated by step 2.

This report documents step 3. Steps 1 and 2 are documented in reference [2].
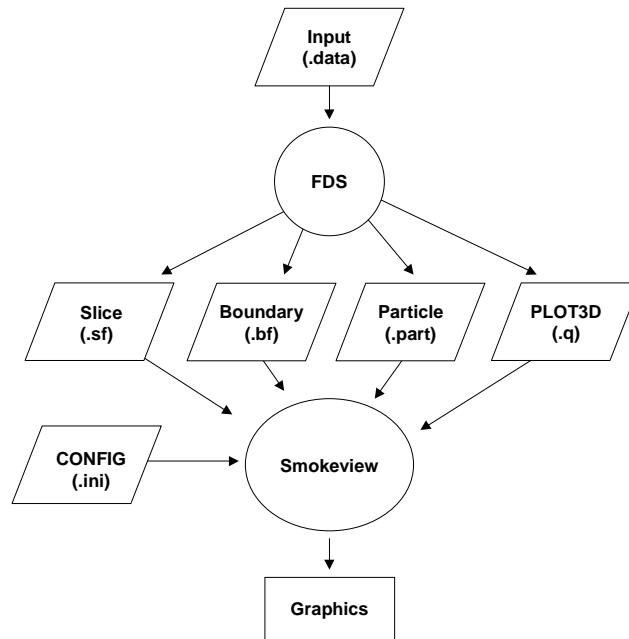
Figure 1: Diagram illustrating data files and programs used in the NIST Fire Dynamics System (FDS)

Particle, slice, boundary and PLOT3D files may be loaded into Smokeview by clicking the right-mouse button and selecting the appropriate `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 and explained in Smokeview's help menu item and Appendix C. Visualization attributes not controllable through the menus (for example, data minimums and maximums) may be changed by using the Smokeview customization file named `smokeview.ini` which is discussed in Appendix D.1.

Smokeview is written using the 3D graphics library OpenGL[4] and the Graphics Library Utility Toolkit (GLUT)[5]. Implementing a full fledged graphical user interface (GUI) is a daunting task. Using GLUT allows one to create simple GUI applications with minimal effort so that most of the development can be spent on implementing the visualizations rather than the creating an elaborate user interface. In addition, GLUT applications are portable. Currently, Smokeview runs on the PC under Windows 95/98/NT/2000 and on several UNIX workstations including SGI/IRIX, IBM/AIX, SUN/Solaris, HP/UX and Redhat LINUX.

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 providing 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.[1] Windows 98, 2000, NT 4.0 and all later versions of Windows 95 contain the OpenGL libraries.

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` 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 GIF 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.

Normally Smokeview is run after an FDS case 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 modeling elements have been defined and located as intended. One may select the color of these elements using color parameters in the `smokeview.ini` file to help distinguish one element from another.

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:

Particle files   Particle files contain the locations of the thermal elements (or particles) used to introduce heat from the fire or to visualize the flow field. Particles are colored using criteria such as: 1) temperature, 2) burning state: burning, burned out, or 3) whether they are water (sprinkler droplets). Temperature colors are selected from a user definable color palette defined with the `COLOR` keyword documented in Appendix D.1. Figure 2 shows a snapshot of particle flow where particles are colored according to their temperature. Default colors for other quantities are yellow for burning, white for non-burning, and blue for water droplets. Particles are stored in files ending with the extension `.part`. Particle files are displayed by selecting the desired particle file from the `Load/Unload` menu.

Slice files   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 a snapshot of a slice file where the slice is colored according to gas temperature. Slice files have file names with extension `.sf`. Slice files are displayed by selecting the desired slice file from the `Load/Unload` menu.

Boundary Files   Boundary files contain simulation data recorded at interior or exterior boundary surfaces. Continuously shaded contours are drawn for quantities such as wall surface temperature, radiative flux *etc*. Figure 4 shows a snapshot of a boundary file animation where the surfaces are colored according to their temperature. Boundary files have file names with extension `.bf`.

---

[1]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.

NIST Smokeview 1.0.0

Part
TEMP
C

215
195
175
155
135
115
95.0
75.0
55.0
35.0
15.0

Frame: 5
Time: 2.5

Figure 2: Particle File Example.

NIST Smokeview 1.0.0

Slice
TEMP
C

215
195
175
155
135
115
95.0
75.0
55.0
35.0
15.0
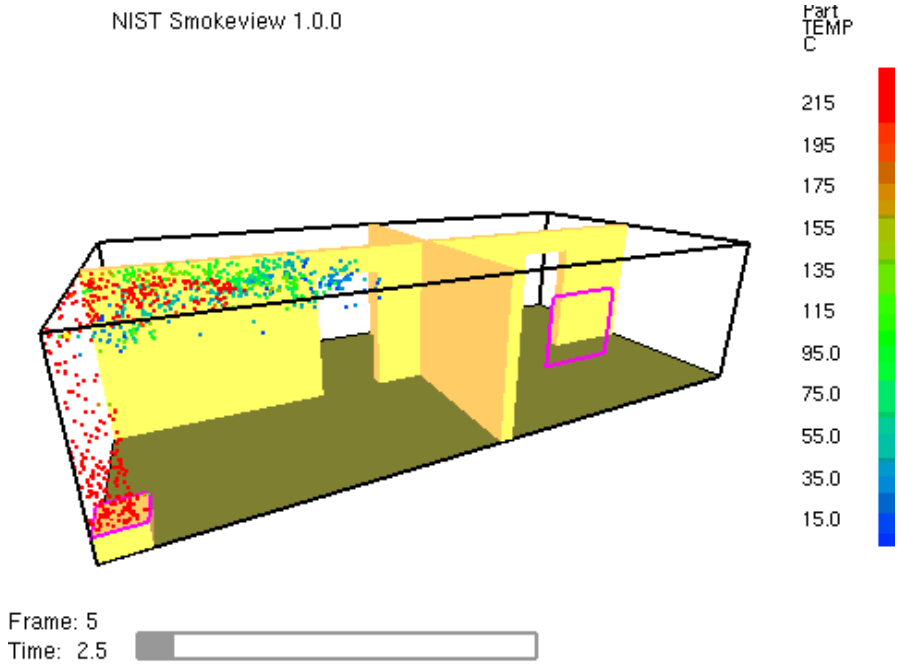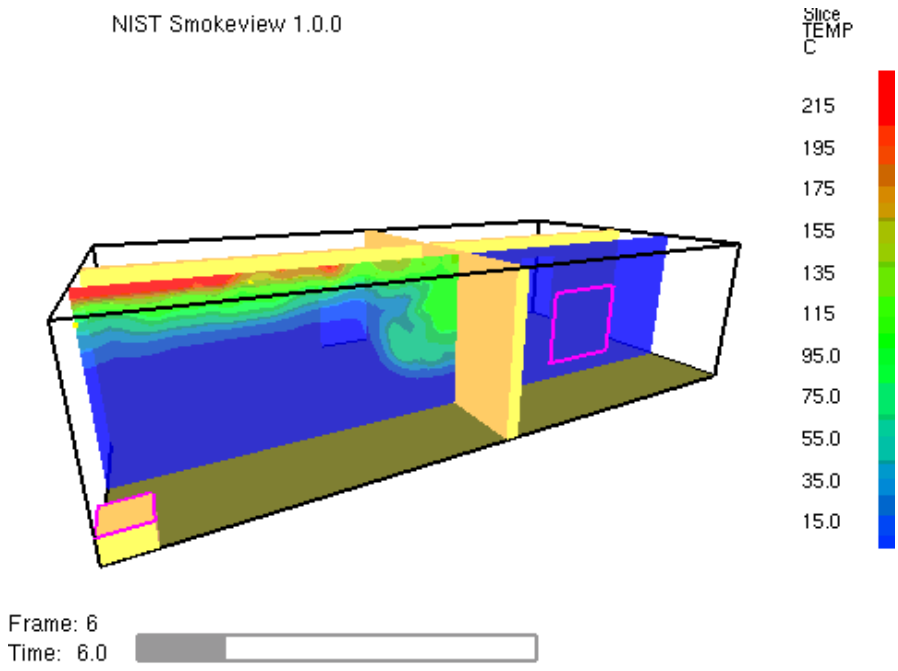
Frame: 6
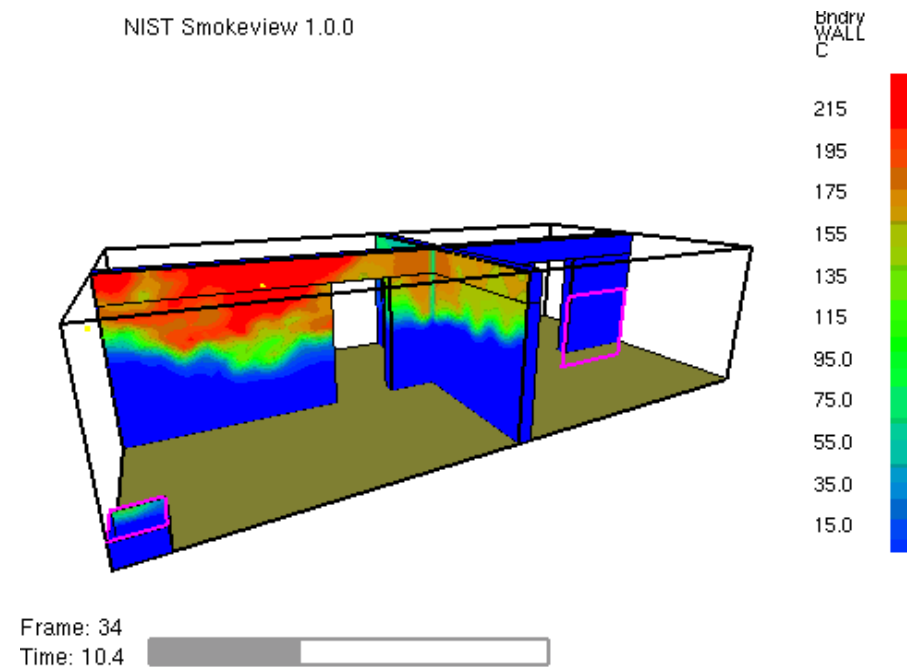Time: 6.0

Figure 3: Slice File Example

Figure 4: Boundary File Example

Boundary files are displayed by selecting the desired boundary file from the `Load/Unload` menu.

PLOT3D files   PLOT3D files store five data values at each grid cell. FDS stores 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. PLOT3D[6] files store data in a standard format developed by NASA and used by many CFD programs for representing simulation results. An FDS simulation will typically create PLOT3D files at several times. PLOT3D data is visualized in three ways: as 2D contours, vector plots and iso-surfaces. 2D contours[2] along either of the XZ, XY or YZ planes. In the default Smokeview view, the X and Z coordinate axes are parallel to the horizontal and vertical screen edges respectively while the Y coordinate axis occurs front to back "within" the screen. Figure 5 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 speed of fluid flow travel while the vector colors show the solution value. Figure 6 show vectors which indicate flow direction. 3D contours or iso-surfaces show where the solution attains one particular value. Figure 7 gives an example of an iso-surface. PLOT3D data are stored in files with extension `.q`.

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

---

[2]2D contours may be shaded continuously as are slice and boundary file data or shaded in discrete shades of uniform color.
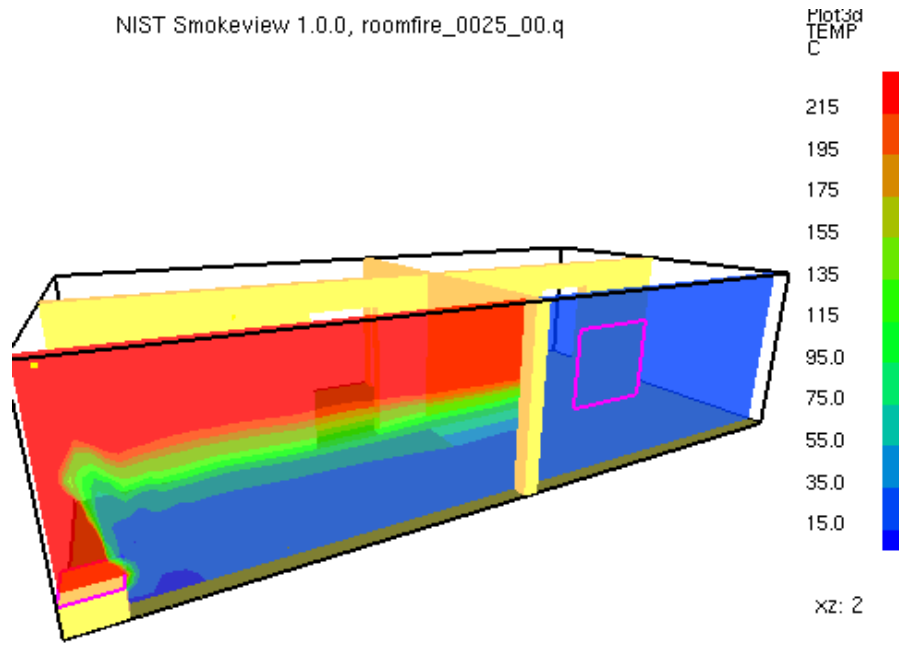
Figure 5: PLOT3D Contour Example
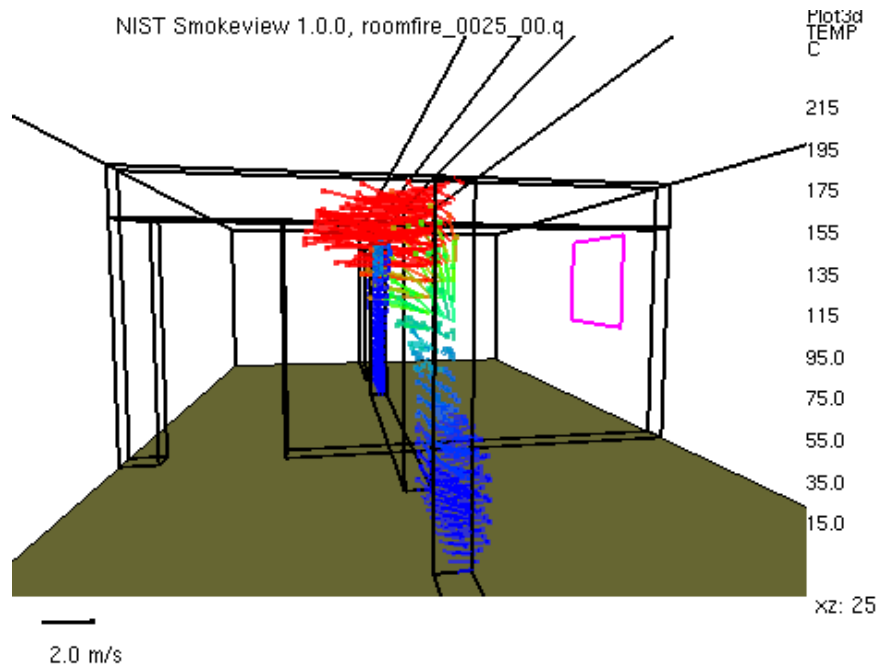
Figure 6: PLOT3D Vector Plot through a door
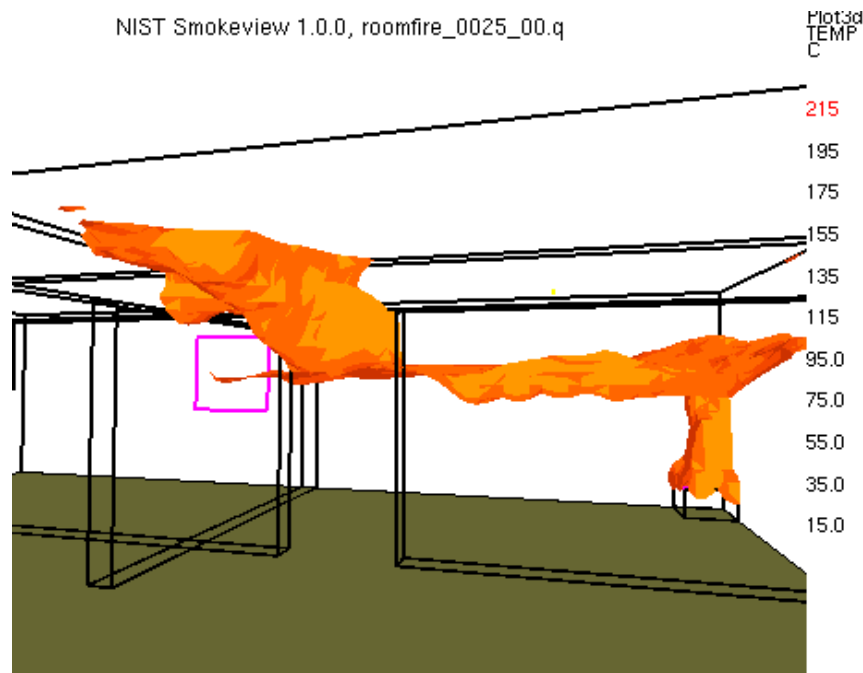
Figure 7: PLOT3D Iso-contour Example

3.2 Visualizing FDS Output

3.2.1 Mouse movement

The left mouse button is used to either rotate or translate the scene depending upon whether the `SHIFT`, `CTRL` or `ALT` keys are depressed or not during mouse movement. Figure 8 shows an outline view of a Smokeview scene with X, Y and Z axes labeled. Again note that in the default Smokeview view, the X and Z coordinate axes are parallel to the horizontal and vertical screen edges respectively while the Y coordinate axis occurs front to back "within" the screen. 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 th Z axis. These labels are used to describe how the mouse effects scene movement in the following way:

No modifier key depressed Horizontal(vertical) mouse movement results in scene rotation parallel to the XY (YZ) plane. 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.

7
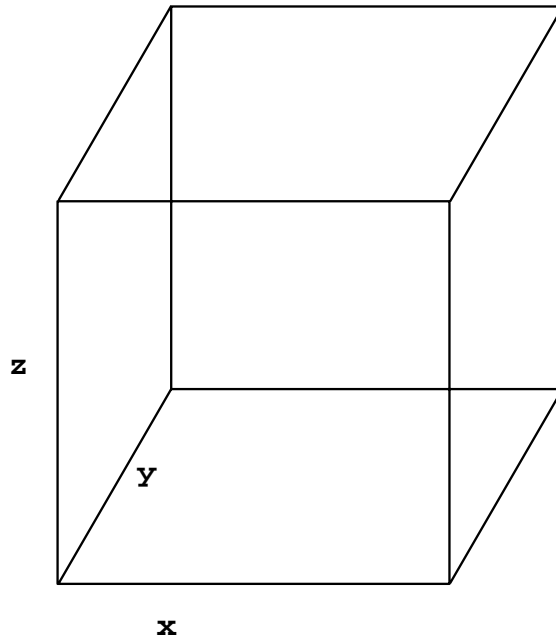
Figure 8: Smokeview coordinate system.

Note that the Reset View menu item resets the scene back to the original view.

3.2.2 Particle, slice, boundary file visualization

Particle, slice and boundary files are straightforward to visualize. Once Smokeview has started up, simply select the desired file from the Load/Unload menu and view the scene.

Particle files are typically much larger than other output files. Several strategies exist for reducing the memory required to display particle files. All of these strategies involve modifying various keywords found in the Smokeview preference files. The Smokeview preference file smokeview.ini may be created by typing "smokeview -ini" at a command line or by selecting the Init File sub-menu of the the Load/Unload menu. These preference file keywords with default values are given below and are documented in Appendix D.1.

```
MAXFRAMES
501
MAXPOINTS
5000000
PARTPOINTSTEP
1
PARTPOINTCOMPRESS
2
PARTFRAMESTEP
1
```

The MAXFRAMES and MAXPOINTS keywords determine the maximum number of data frames and data points used to read in particles. These two parameters may also be set from the command line when starting up Smokeview. The PARTPOINTSTEP and PARTFRAMESTEP parameters allow one to skip particles or particle frames. Finally, the PARTPOINTCOMPRESS allows one to store particle data in a compressed format. Setting PARTPOINTSTEP and PARTFRAMESTEP to

8

```
PARTPOINTSTEP
3
PARTFRAMESTEP
3
```

reduces the memory needed by a factor of nine since every third point and every third particle frame instead of all points and frames are loaded.

Sometimes it is of interest to change the minimum and maximum axis labels, for example to use the same min and max for each loaded data file. Data value bounds may be set with the V_PARTICLES, V_SLICE and V_BOUNDARY keywords. The default values are given by

```
V_PARTICLES
0 1.000000 0 0.000000
V_SLICE
0 1.000000 0 0.000000
V_BOUNDARY
0 1.000000 0 0.000000
```

To cause Smokeview to set the min and max labels to 30 and 600 respectively use

```
V_PARTICLES
1 30.000000 1 600.000000
V_SLICE
1 30.000000 1 600.000000
V_BOUNDARY
1 30.000000 1 600.000000
```

The integer "1" occuring before the "30" or "600" causes Smokeview to use the next number as a minimum or maximum value respectively. If this integer is 0 then Smokeview ignores the subsequent min/max value.

3.2.3 Visualizing PLOT3D data files

PLOT3D files are more complicated to visualize than files containing time dependent data 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 six[3] 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 the "middle" 2D contour 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. The "x", "y" or "z" keys toggle contour visibility along the X, Y and Z axes respectively. 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. Minimum and maximum PLOT3D values may be set using the V_PLOT3D smokeview.ini keyword. The default V_PLOT3D keyword value is given by

```
V_PLOT3D
```

---

[3]The 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.

```
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 bound lines. Each bound line begins with
an integer (indicating the PLOT3D variable number) followed by 4 numbers defined the same
as with the V_PARTICLE, V_SLICE and V_BOUNDARY keywords. To set the minimum and
maximum bound for the first PLOT3D variable to 20 and 620 use

```
V_PLOT3D
5
1 1 20.0 1 620.0
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
```

Smokeview then uses these bounds to label the PLOT3D color bar from 20 to 620. It draws con-
tours smaller than 20 with the "smallest" color and contours bigger than 620 with the "biggest"
color. 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 contour levels smaller than 20 or contours greater than 620. This is true for "stepped" or
discrete contours. If "continuous" contours are drawn then "2" and "1" have the same effect.
Figure 9 shows a ceiling jet contour plot drawn using the following V_PLOT3D values

```
V_PLOT3D
5
1 2 300.0 1 350.0
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 bound line 1 2 300.0 1 350.0 indicates that temperatures below 300 ° C are not
drawn and that temperatures above 350 ° C are drawn with the "highest" color (black in black
and white mode, red in color mode).

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" key is depressed, then every other vector is skipped.
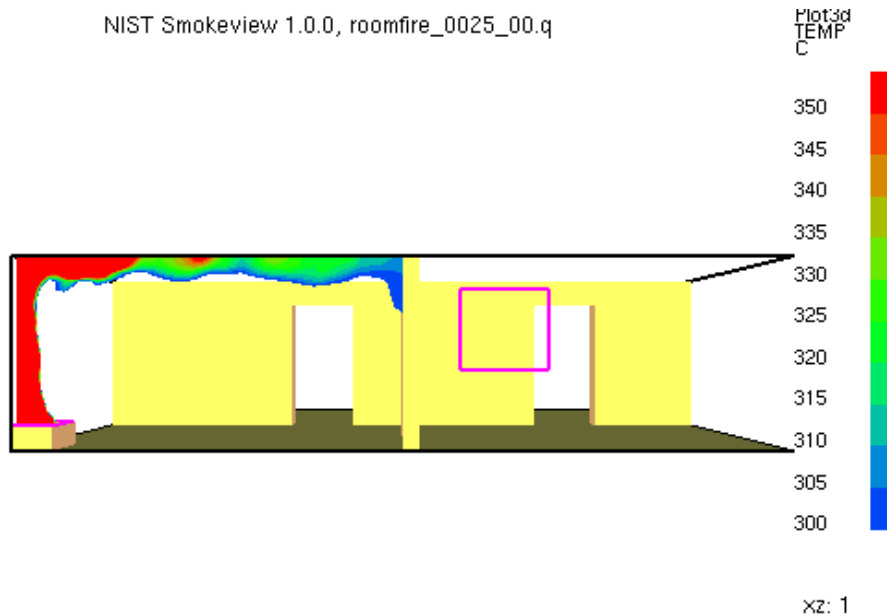
Figure 9: Contour plot illustrating a ceiling jet by using a `V_PLOT3D` keyword entry of: "`1 2 300.0 1 350.0`" for the first (temperature) PLOT3D variable.

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.

3.3   Checking FDS Input

Consider the following FDS input file. It describes a small space with a burner adjacent to the near wall, a small vent on the opposite wall and an interior wall containing a door. Figure 10 is a Smokeview visualization of this input file. Using Smokeview one may conclude that a blockage and vent are not in their intended locations.

```
&HEAD CHID='testcase1',TITLE='Single Fire Plume' /
&GRID IBAR=32,JBAR=32,KBAR=32 /
&PDIM XBAR=1.0,YBAR=1.0,ZBAR=1.0 /
&TIME DT=0.05,TWFIN=0.05 /
&SURF ID='burner',VEL=-.2,PARTICLES=.TRUE.,PARTICLE_COLOR='RED' /
&SURF ID='inlet',VEL=-.2,PARTICLES=.TRUE.,PARTICLE_COLOR='GREEN' /
&OBST XB=0.65,0.85,0.00,0.2,0.1,0.2,BLOCK_COLOR='GREEN' /
&OBST XB=0.00,0.30,0.45,0.5,0.0,1.0 /
&OBST XB=0.70,1.00,0.45,0.5,0.0,1.0 /
&OBST XB=0.30,0.60,0.45,0.5,0.7,1.0,BLOCK_COLOR='BLUE' /
&VENT XB=0.70,0.8,0.05,0.15,0.2,0.2,SURF_ID='burner' /
&VENT XB=0.65,0.85,1.00,1.0,0.2,0.4,SURF_ID='inlet' /
```

This input file contains two errors as can be noted by examining Figure 10. First, the door soffit (the blue blockage) contains an unintended gap, second the burner vent does not completely cover the underlying (GREEN) blockage. Note that it is certainly permissible to define a vent this way but for the purposes of this example it was intended that the vent cover the entire blockage. Blockages may be individually colored
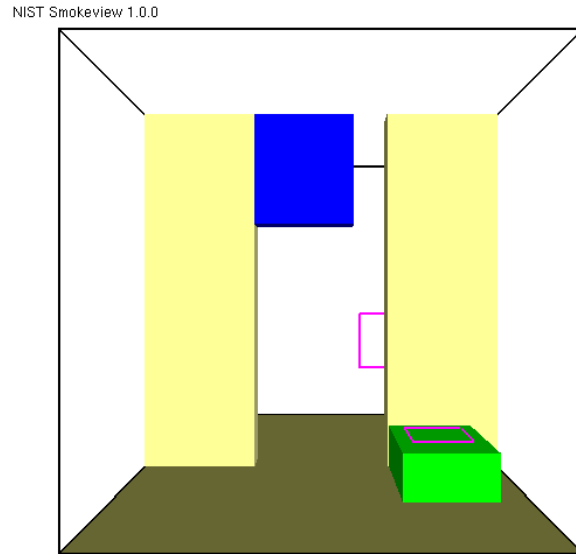
Figure 10: Using Smokeview to verify model input. This example has a gap in the door soffit and a burner vent boundary that is too small.

in order to associate `&OBST` lines in an FDS data file with blockages in the Smokeview scene. The `blue` blockage may be corrected replacing the above `blue OBST` line with

```
&OBST XB=0.30,0.70,0.45,0.5,0.7,1.0,BLOCK_COLOR='BLUE' /
```

The burner vent may be corrected by matching the horizontal vent coordinates with the horizontal "GREEN" blockage coordinates and replacing the 'burner' vent line in the above case with

```
&VENT XB=0.65,0.85,0.00,0.20,0.2,0.2,SURF_ID='burner',VENT_COLOR='GREEN' /
```

Figure 11 shows a view of the corrected test case.

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. Set `SET_UP=.TRUE.` on 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.
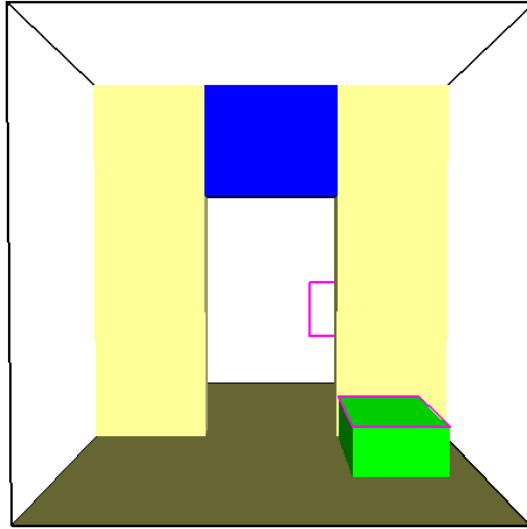
Figure 11: Corrected case

4. Make corrections if necessary. Use the BLOCK_COLOR option[4] of the FDS keyword OBST to more easily identify blockages to be edited. 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.

---

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

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
```

without any arguments which results in output similar to:

```
Smokeview 1.0.0 - Release Jan  1 2000 - NIST/BFRL
Visualize fire/smoke flow simulations.  All parameters are optional.

Usage:

smokeview casename -points m -frames n -ini -nopart -benchmark

where

  casename = project id (file names without the extension)
         m = maximum number of particles.  Default=5000000
         n = maximum number of particle frames.  Default=501
      -ini = output default smokeview parameters to smokeview.ini
   -nopart = do not load particle file
-benchmark = display graphics performance indicators
```

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 iso-surfaces (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[5] based on OpenGL[4] 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 options are explained in the following sections.

B.1   Main Menu Items

Load/Unload   This 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 slice, one boundary and one PLOT3D file may be viewed at a time. 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.

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. The `unlimited` menu item allows one to display frames as rapidly as the graphics hardware permits or to step through the frames manually. 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.

Render   This option allows one to create GIF files of the currently displayed scene. The `Render` sub-menus allow one to specify an integer between 1 and 20 indicating the number of frames between rendered GIF images. This allows one to generate GIF 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`. (Keyboard shortcut for the render option: `r`)

Tour   Causes the scene to rotate about its center.

Reset   Resets the simulation scene back to the original view. 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   Exit or quit Smokeview.

B.2 Load/Unload

Four types of files may be visualized with Smokeview. They are particle, slice, boundary and PLOT3D files. 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 modeling software creates this file. (See Appendix D.2 for documentation on the format of this file).

    The character "*" occuring 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.

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

Slice File (`.sf`)   This menu item gives the name of all slice files and also the option to unload the currently loaded slice file.

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 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 (ther-

mocouples, 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 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.

Vents   Show or hide the vents.

Walls   Show or hide the walls.

B.3.2 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.3.3 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.3.4 Animated Slice

Toggle the visibility of the animated slice or slice file.

B.3.5 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.3.6 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.

Level 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.3.7 Heat detectors, Sprinklers, Thermocouples

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

B.3.8 Shades

Flip 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.

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.3.9 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.3.10   Labels

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

Show All   Show all scene elements.

Hide All   Hide all scene elements.

C   Keyboard Commands

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 visiblity. 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 visiblity 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 GIF 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.

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).

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.

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.

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 if 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`

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.

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.

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

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 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`)

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 lines in pixels except lines used to draw vents. (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)

SLICEOFFSET   Defines an offset distance between animated slices and adjacent solid surfaces. The offset distance is relative to the maximum grid cell width. (default: 0.10)

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. The default values are specified with

```
T_BOUNDARY
0 1.0 0 0.0
```

To load boundary data between 20 and 150 seconds use

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

T_PARTICLES   Defines the minimum and maximum times used to include particles in a visualization. (default: `0 1.0 0 0.0`)

T_SLICE   Defines the minimum and maximum times used to include particles in a visualization. (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`)

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.

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`)

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)

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 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 % 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)

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, EYEY, EYEZ   The parameters `EYEX, EYEY, EYEZ` 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)

SHOWBLOCKS   Specifies how a blockage should be drawn. A value of 0, 1 or 2 indicates that the blockages are invisible, drawn normaly 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: 1)

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)

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

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)

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

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.

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.

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
```

27

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, ztherm` are the `x, y, z` coordinates of the thermocouples.

D.2.4   System Keywords

ENDIAN   The `ENDIAN` keyword defines whether a big or little endian[5] 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` paramters 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.

---

[5]Big-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 side or the little side.

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

Smokeview visualizes fire dynamics data using 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` statments 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 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
       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.3.2 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=11,I2),J=J1,J2),K=K1,K2)
```

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

D.3.3 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.3.4 PLOT3D Data

Field data is dumped periodically by FDS in a format used by the graphics package **PLOT3D**[6]. 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) (((((QQ(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).

References

[1] 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.

[2] 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.

[3] K.B. McGrattan, H.R. Baum, and R.G. Rehm. Large Eddy Simulations of Smoke Movement. *Fire Safety Journal*, 30:161–178, 1998.

[4] Mason Woo, Jackie Neider, and Tom Davis. *OpenGL Programming Guide*. Addison-Wesley Developers Press, Reading, Massachussets, 1996.

[5] Mark J. Kilgard. *OpenGL Programming for the X Window System*. Addison-Wesley Developers Press, Reading, Massachussets, 1996.

[6] Pamela P. Walatka and Pieter G. Buning. PLOT3D User's Manual, version 3.5. NASA Technical Memorandum 101067, NASA, 1989.

<table>
<tr><td colspan="2">

U.S. Department of Commerce
National Institute of Standards and Technology

Manuscript Review and Approval

</td><td colspan="4">(ERB Use Only - Version 2.10)</td></tr>
</table>

U.S. Department of Commerce / National Institute of Standards and Technology — Manuscript Review and Approval	(ERB Use Only - Version 2.10)	
	ERB Control Number	Division
	Publications Report Number	Category Code
	Publication Date	No. Printed Pages
Attach Original of this Form to One Copy of Manuscript and Send to: WERB Secretariat, Mail Stop 2500 (Admin. 101/E212)	HOLDREC#	Changes Required

Title and Subtitle (Cite in full)
User's Guide for Smokeview Version 1.0 - A Tool for Visualizing Fire Dynamics Simulation Data

Author(s) (Last Name, First Initial, Second Initial)	Performing Organization:
Forney, G P (Primary) ; McGrattan, K B	**NIST/GAITHERSBURG**

Laboratory and Division
864 - Fire Safety Engineering Division

Sponsoring Organization Name and Complete Address (Street, City, State, Zip)
NIST

Type of Publication Proposed
NISTIR

Proposed for Non-NIST Publication (Cite fully)

Publishing Medium:
Paper

Abstract (A 2000-character or less factual summary of most significant information. If document includes a significant bibliography or literature survey, cite it here. Spell out acronyms on first reference.)
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 used in concert to respectively model and analyze scenarios involving features such as fire, wind, sprinklers, complex geometry et. cetera. Smokeview visualizes NIST FDS modeling results in many ways, some of which are 1) animating flow of smoke or tracer particles, 2) animating 2D contours of data slices, 3) plotting 2D and 3D static contours or 4) plotting flow vectors . Smokeview requires a fast CPU, fast video card, a substantial amount of random-access memory (RAM) and a large hard disk. Smokeview is portable and can be run on a variety of computing platforms. For a Windows-based PC, the processor should be at least as fast as a 450 Mhz Pentium II, with at least 256 Mbytes of memory. Of course, more is better, but this configuration should allow one to do fairly large calculations in a reasonable time period. For UNIX-based workstations, the processor and memory should be at least as fast and as large as the PC specs. Smokeview and NIST FDS may be obtained at the website http://fire.nist.gov/smokeview or by contacting the authors at glenn.forney@nist.gov or

kevin.mcgrattan@nist.gov.

Supplementary Notes

Key Words (Maximum of 9; 28 characters and spaces each; Alphabetic order; Capitalize only proper names)

1. cfd models	**2.** field models
3. fire simulation	**4.** fluid flow
5. scientific visualization	**6.** simulation
7. visualization	**8.**
9.	

Availability:	Manuscript Announcement:
Unlimited	**Announce Only After Publication.**

Signature Page of MRA-OPS Form