

This document has been superseded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

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

NIST Special Publication 1017

User's Guide for Smokeview Version 4
- A Tool for Visualizing Fire Dynamics
Simulation Data

Glenn P. Forney
Kevin B. McGrattan

NIST Special Publication 1017

User's Guide for Smokeview Version 4
- A Tool for Visualizing Fire Dynamics
Simulation Data

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

August 2004



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

Technology Administration
Phillip J. Bond, Under Secretary for Technology

National Institute of Standards and Technology
Arden L. Bement, Jr., Director

Certain commercial entities, equipment, or materials may be identified in this document in order to describe an experimental procedure or concept adequately. Such identification is not intended to imply recommendation or endorsement by the National Institute of Standards and Technology, nor is it intended to imply that the entities, materials, or equipment are necessarily the best available for the purpose.

National Institute of Standards and Technology Special Publication 1017
Natl. Inst. Stand. Technol. Spec. Publ. 1017, 96 pages (August 2004)
CODEN: NSPUE2

U.S. GOVERNMENT PRINTING OFFICE
WASHINGTON: 2004

For sale by the Superintendent of Documents, U.S. Government Printing Office
Internet: bookstore.gpo.gov – Phone: (202) 512-1800 – Fax: (202) 512-2250
Mail: Stop SSOP, Washington, DC 20402-0001

Preface

Smokeview is a software tool designed to visualize numerical calculations generated by the NIST Fire Dynamics Simulator (FDS), a computational fluid dynamics (CFD) model of fire-driven fluid flow. For details on setting up and running FDS cases read the FDS User's guide[1]. These two tools are typically used together to respectively simulate and visualize the flow of smoke induced by a fire. Smokeview visualizes the smoke using traditional scientific methods such as displaying tracer particle flow, 2D or 3D shaded contours of gas flow data such as temperature and flow vectors showing flow direction and magnitude. Smokeview also visualizes smoke realistically so that one can "experience" the fire. This is done by displaying a series of partially transparent planes where the transparencies in each plane (at each grid node) are determined from soot densities computed by FDS. 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.

Windows PC versions of Smokeview and FDS and associated documentation may be downloaded from the web site <http://fire.nist.gov/smokeview> at no cost. Versions for Linux and SGI/IRIX may also be downloaded from the same location.

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.

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.

Acknowledgements

Smokeview has been under development for over five years. A number of people have made significant contributions. In trying to acknowledge those that have contributed, we are inevitably going to miss a few people - let us know and we will include those missed in the next version of this guide.

The original version of Smokeview was inspired by Frames, a visualization program written by James Sims for the Silicon Graphics workstation. This software was based on visualization software written by Stuart Cramer for an Evans and Sutherland computer. Frames used tracer particles to visualize smoke flow computed by a pre-cursor to FDS. Judy Devaney made the NIST multi-screen eight foot Rave facility available allowing a stereo version of Smokeview to be built that can display scenes in “3D”. Both Steve Satterfield and Tere Griffin on many occasions helped me demonstrate Smokeview cases on the Rave inspiring many people to the possibility of using Smokeview as a “virtual reality-like” fire fighter training facility.

Many conversations with Nelson Bryner, Dave Evans, Anthony Hamins and Doug Walton were most helpful in determining how Smokeview could be adapted for use in fire fighter training applications.

Smokeview would not be possible without the use of a number of software libraries developed by others. Mark Kilgard while at Silicon Graphics developed GLUT, the basic tool kit for interfacing OpenGL with the underlying operating system on multiple computer platforms. Paul Rademacher while a graduate student at the University of North Carolina developed GLUI, the software library for implementing the user friendly dialog boxes.

Significant contributions have been made by those that have used Smokeview to visualize complex cases; cases that are used to perform both applied and basic research. The resulting feedback has improved Smokeview as a result of their interaction with me, pushing the envelope and not accepting the status quo.

For applied research, Daniel Madrzykowski, Doug Walton and Robert Vettori of NIST have used Smokeview to analyze fire incidents. Steve Kerber has used Smokeview to visualize flows resulting from PPV fans. David Stroup has used Smokeview to analyze cases for use in fire fighter training scenarios. Conversations with Doug Walton have been particularly helpful in identifying needed features and clarifying how best to make their implementation user friendly. David Evans, William (Ruddy) Mell and Ronald Rehm used Smokeview to visualize “urban-wildland interface” fires. For basic research, Greg Linteris has used Smokeview to visualize fire simulations involving the cone calorimeter. Anthony Hamins has used Smokeview to visualize the structure of CH₄/air flames undergoing the transition from normal to micro-gravity conditions and fire suppression in a compartment. Jiann Yang has used Smokeview to visualize smoke or particle number density and saturation ratio of condensable vapor.

This user’s guide has improved through the many constructive comments of the reviewers Anthony Hamins, Doug Walton, Ronald Rehm, and David Sheppard. Chuck Bouldin helped port Smokeview to the Macintosh.

Many people have sent in multiple comments and feedback by email, in particular Adrian Brown, Scot Deal, Charlie Fleischmann, Jason Floyd, Simo Hostikka, Bryan Klein, Davy Leroy, Dave McGill, Brian McLaughlin, Derek Nolan, Steven Olenick, Stephen Priddy, Boris Stock, Jason Sutula, Javier Trelles, and Christopher Wood.

Feedback is encouraged and may be sent to glenn.forney@nist.gov .

Contents

Preface	i
Disclaimer	iii
1 Introduction	1
1.1 Overview	1
1.2 Features	2
1.3 What's New	3
2 Getting Started	5
3 Basics	7
3.1 Manipulating the Scene	7
3.2 Tracer Particles - Particle Files	10
3.3 3D Smoke	10
3.4 2D Shaded Contours - Slice Files	11
3.5 2D Shaded Contours on Solid Surfaces - Boundary Files	12
3.6 3D Contours - Isosurface Files	15
3.7 Static Data - Plot3D Files	15
4 Advanced Features	19
4.1 Manipulating the Scene Automatically - The Touring Option	19
4.2 Clipping Scenes	23
4.3 Setting Data Bounds	24
4.4 3D Smoke Options	27
4.5 Plot3D Viewing Options	27
4.6 Display Options	27
4.7 Texture Maps	28
4.8 Debugging FDS Input Files	29
4.9 Making Movies	31
4.10 Annotating the Scene	32
5 Summary	35
References	38
Appendices	38
A Command Line Options	39

B	Menu Options	41
B.1	Main Menu Items	41
B.2	Load/Unload	42
B.3	Show/Hide	44
	B.3.1 Geometry Options	44
	B.3.2 Animated Surface	45
	B.3.3 Particles	45
	B.3.4 Boundary	45
	B.3.5 Animated Vector Slice	46
	B.3.6 Animated Slice	46
	B.3.7 Plot3D	46
	B.3.8 Heat detectors, Sprinklers, Thermocouples	47
	B.3.9 Textures	47
	B.3.10 Labels	47
B.4	Options	47
	B.4.1 Shades	49
	B.4.2 Units	49
	B.4.3 Rotation	49
	B.4.4 Max Frame Rate	49
	B.4.5 Render	51
	B.4.6 Font Size	51
	B.4.7 Zoom	51
B.5	Dialogs	51
B.6	Tours	52
C	Keyboard Shortcuts	53
D	File Formats	55
D.1	Smokeview Preference File Format (.ini files)	55
	D.1.1 Color parameters	56
	D.1.2 Size parameters	58
	D.1.3 Time, Chop and value bound parameters	58
	D.1.4 Data loading parameters	61
	D.1.5 Viewing parameters	62
	D.1.6 Tour Parameters	66
	D.1.7 Realistic Smoke Parameters	67
D.2	Smokeview Parameter Input File (.smv file)	68
	D.2.1 Geometry Keywords	68
	D.2.2 File Keywords	70
	D.2.3 Sensor Keywords	70
	D.2.4 Miscellaneous Keywords	71
D.3	Data File Formats (.s3d, .iso, .part, .sf, .bf and .q files)	72
	D.3.1 3D Smoke File Format	72
	D.3.2 Isosurface File Format	72
	D.3.3 Particle File Format	72
	D.3.4 Slice File Format	73
	D.3.5 Boundary Files	74
	D.3.6 Plot3D Data	74

E	Frequently Asked Questions	77
E.1	How can I keep up with new releases and other information about FDS and Smokeview? . . .	77
E.2	Can I run Smokeview on other machines?	77
E.3	Smokeview doesn't look right on my computer. What's wrong?	77
E.4	How do I make a movie of a Smokeview animation?	78
E.5	Smokeview is running much slower than I expected. What can I do?	78
E.6	I loaded a particle file but Smokeview is not displaying particles. What is wrong?	78

List of Figures

1.1	Diagram illustrating data files and programs used in the NIST Fire Dynamics Simulator (FDS).	2
3.1	Motion Dialog Box	8
3.2	Particle file snapshots of a townhouse kitchen fire.	10
3.3	Smoke3D file snapshots of a townhouse kitchen fire.	11
3.4	Slice file snapshots of shaded temperature contours.	12
3.5	Vector slice file snapshots of shaded vector plots.	13
3.6	Boundary file snapshots of shaded wall temperatures contours.	14
3.7	Isosurface file snapshots of temperature levels.	16
3.8	Plot3D contour and vector plot examples.	17
3.9	Plot3D isocontour example.	18
4.1	Overhead view of the townhouse example showing the default “Circle” tour and a user defined tour.	20
4.2	Touring dialog boxes	21
4.3	Tutorial examples for Tour option	22
4.4	Clipping dialog box	23
4.5	Clipped scene corresponding to Clip Dialog box given in Figure 4.4	24
4.6	Bounds dialog box showing PLOT3D file options	25
4.7	Bounds dialog box showing slice file options	25
4.8	Ceiling Jet Visualization.	26
4.9	Dialog Box for setting 3D smoke options	26
4.10	Dialog Box for setting miscellaneous Smokeview scene properties	28
4.11	Texture map example.	29
4.12	Blockage Edit Dialog Box	31
4.13	Annotation example using the TICKS and LABEL keyword.	33
B.1	Main Menu	42
B.2	Load/Unload Menu	43
B.3	Geometry Menu	44
B.4	Label Menu	47
B.5	Option Menu	48
B.6	Shades Menu	48
B.7	Render Menu	50
B.8	Dialogs Menu	51
B.9	Tour Menu	52

List of Tables

3.1	Keyboard mappings for “eye centered” scene movement	9
3.2	Some output quantities used to generate slice and Plot3D visualizations.	13
3.3	Some output quantities used to generate boundary file visualizations.	15

Chapter 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[2]. This report documents version 4 of Smokeview updating material found in Ref. [3]. For details on setting up and running FDS cases read the FDS User's guide[1].

FDS and Smokeview are used in concert to respectively model and visualize fire phenomena. However, FDS and Smokeview are not limited to fire simulation. For example, one may use FDS and Smokeview to model other applications such as contaminant flow in a building. Smokeview performs this visualization by displaying time dependent tracer particle flow, animated contour slices of computed gas variables and surface data. Smokeview also presents contours and vector plots of static data anywhere within a simulation scene at a fixed time. Several examples using these techniques to investigate fire incidents are documented in Refs. [4, 5, 6, 7].

Normally Smokeview is used in a post-processing step to visualize FDS data after a calculation has been completed. Smokeview may also be used during a calculation to monitor a simulation's progress and before a calculation to setup FDS input files more quickly, one can then use Smokeview to edit or create blockages by specifying the size, location and/or material properties.

Figure 1.1 gives an overview of how data files used by both FDS and Smokeview are related. A typical procedure for using FDS and Smokeview is to:

1. Set up an FDS input file.
2. Run FDS. FDS then creates one or more output files used by Smokeview to visualize the case.
3. Run Smokeview to analyze the output files generated by step 2. by either double-clicking the file named `casename.smv` with the mouse (on the PC) 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 publication documents step 3. Steps 1 and 2 are documented in the FDS User's Guide [1].

Menus in Smokeview are activated by clicking the right mouse button anywhere in the Smokeview window. Data files may be visualized by selecting the desired `Load/Unload` menu option. Other menu options are discussed in Appendix B. Many menu commands have equivalent 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 customized by using the Smokeview preference file, `smokeview.ini`, discussed in Appendix D.1.

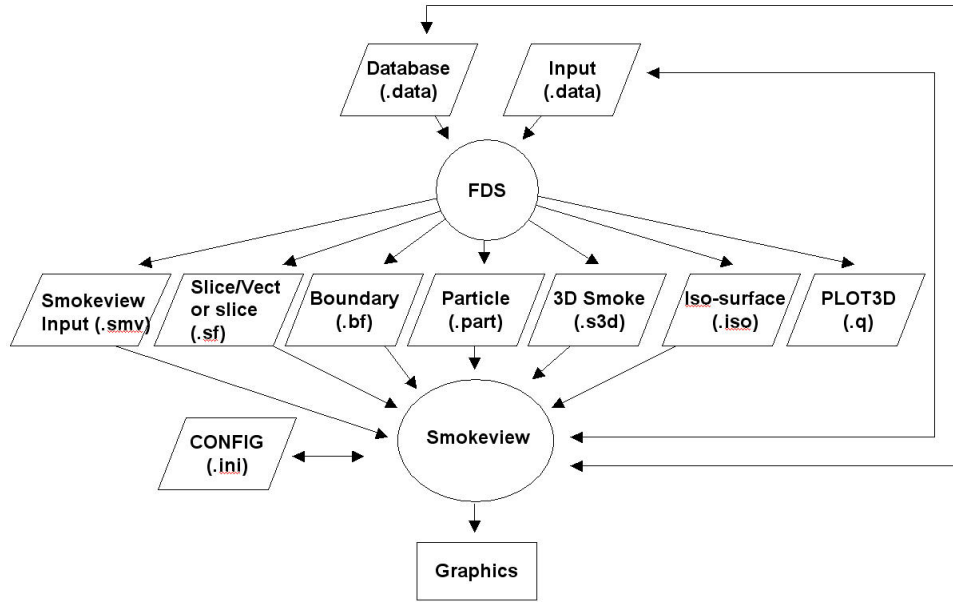


Figure 1.1: Diagram illustrating data files and programs used in the NIST Fire Dynamics Simulator (FDS).

Smokeview is written in C and Fortran 90 and consists of about 45,000 lines of code. The C portion of Smokeview visualizes the data while the Fortran 90 portion reads in the data generated by FDS (also written in Fortran 90). Smokeview uses the 3D graphics library OpenGL[8] and the Graphics Library Utility Toolkit (GLUT)[9]. Smokeview uses the GLUT software library so that most of the development effort can be spent implementing the visualizations rather than creating an elaborate user interface. Smokeview uses a number of auxiliary libraries to implement image capture (GD[10, 11], PNG[12], JPEG[13]), image and general file compression (ZLIB[14]) and dialog creation (GLUI[15]). Each of these libraries is portable running under UNIX, LINUX and Windows 9x/2000/XP allowing Smokeview to run on these platforms as well.

1.2 Features

Smokeview is a program designed to visualize numerical calculations generated by the Fire Dynamics Simulator. Smokeview visualizes both dynamic and static data. Dynamic data is visualized by animating particle flow (showing location and “values” of tracer particles), 2D contour slices (both within the domain and on solid surfaces) and 3D level surfaces. 2D contour slices can also be drawn with colored vectors that use velocity data to show flow direction, speed and value. Static data is visualized similarly by drawing 2D contours, vector plots and 3D level surfaces. Smokeview features in more detail include:

Animated Isosurfaces Isosurface or 3D level surface animations may be used to represent flame boundaries, layer interfaces or various other gas phase variables. Multiple isocontours may be stored in one file, allowing one to simultaneously view several isosurface levels for the same variable.

Color Contours Animated 2D shaded color contour plots are used to visualize gas phase information, such as temperature or density. The contour plots are drawn in horizontal or vertical planes along any coordinate direction. Contours can also be drawn in shades of gray.

Animated 2D shaded color contour plots are also used to visualize solid phase quantities such as radiative flux or heat release rate per unit area.

Animated Flow Vectors Flow vector animations, though similar to color contour animations (the vector colors are the same as the corresponding contour colors), are better than solid contour animations at highlighting flow features.

Particle Animations Lagrangian or moving particles can be used to visualize the flow field. Often these particles represent smoke or water droplets.

Data Mining The user can analyze and examine the simulated data by altering its appearance to more easily identify features and behaviors found in the simulation data. One may flip or reverse the order of colors in the colorbar and also click in the colorbar and slide the mouse to highlight data values in the scene. These options may be found under `Options/Shades`.

The user may click in the time bar and slide the mouse to change the simulation time displayed. One use for the time bar and color bar selection modes might be to determine when smoke of a particular temperature enters a room.

Multimesh Geometry Smokeview shows the multi-mesh geometry new in FDS 3, and gives the user control over viewing certain features in a given mesh. Using the `LOAD/UNLOAD` > `MULTI-SLICE` menu item one may now load multiple slices simultaneously. These slices are ones lying in the same plane (within a grid cell) across multiple meshes.

Scene Clipping It is often difficult to visualize slice or boundary data in complicated geometries due to the number of obstructed surfaces. Interior portions of the scene may now be seen more easily by “clipping” part of the scene away.

Scene Motion The motion dialog box has been enhanced to allow more precise control of scene movement and orientation. Cursor keys have been mapped to scene translation/rotation to allow easy navigation within the scene.

Texture Mapping Jpeg or PNG image files may be applied to a blockage, vent or enclosure boundary. This is called texture mapping. This allows one to view Smokeview scenes more realistically. These image files may be obtained from the internet, a digital camera, a scanner or from any other source that generates these file formats. Image files used for texture mapping should be “seamless.” A seamless texture as the name suggests is periodic in both horizontal and vertical directions. This is an especially important feature when textures are tiled or repeated across a blockage surface.

Blockage Editing The Blockage Editing dialog box has been enhanced to allow one to assign material properties to selected blockages. Comment labels found in the FDS input file may be viewed or edited. Any text appearing after the closing “/” in an &OBST line is treated as a comment by Smokeview.

Annotating Cases The two keywords, LABEL and TICKS are used to help document Smokeview output. The LABEL keyword allows one to place colored labels at specified locations at specified times. The TICK keyword places equally spaced tick marks between specified bounds. These marks along with LABEL text may be used to document length scales in the scene.

1.3 What’s New

Realistic Smoke Smoke, fire and sprinkler spray are displayed realistically using a series of partially transparent planes. The transparencies for smoke are determined by using smoke densities computed by FDS. The transparencies for fire and sprinkler spray are determined by using a heuristic based on heat release rate data and water density data, again computed by FDS. Various settings for the 3d smoke option may be set using the new “3D Smoke” dialog box found in the `Dialogs` menu.

Virtual Tour A series of checkpoints or key frames specifying position and view direction may be specified. A smooth path is computed using Kochanek-Bartels splines[16] to go through these key frames so that one may control the position and view direction of an observer as they move through the simulation. One can then see the simulation as the observer would. This option is available under the `Tour` menu item. Existing tours may be edited and new tours may be created using the “Tour” dialog box found in the `Dialogs` menu. Tour settings are stored in the local configuration file (casename.ini).

Data Chopping - The `Set Bounds . . .` dialog box has been enhanced to allow one to chop or hide data in addition to setting bounds. One use of this feature would be to more easily visualize a ceiling jet by hiding ambient temperature data (data below a prescribed temperature).

Chapter 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 Smokeyview and FDS, sample FDS calculations, software updates and links for requesting feedback about the software.

After obtaining the setup program, install Smokeyview on the PC by either entering the setup program name from the Windows `Start/Run...` menu or by double-clicking the downloaded Smokeyview setup program. The setup program then steps through the program installation. It copies the FDS and Smokeyview executables, sample cases, documentation and the Smokeyview 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 Smokeyview program so that one may either type Smokeyview at any command line prompt or double click on any `.smv` file. Smokeyview uses the OpenGL graphics library which is a part of all Windows distributions except for the first release of Windows 95.

Most computers purchased today are perfectly adequate for running Smokeyview with the caveat that additional memory should be obtained to bring the memory size up to at least 512 MB in order to display results without “swapping” to disk and hence slowing down the visualizations. For Smokeyview it is more important to obtain a fast graphics card than a fast CPU. If the computer will run both FDS and Smokeyview then it is important to obtain a fast CPU as well. For example, the townhouse case found at <http://fire.nist.gov/fds/svsamples/thouse3d.data> consists of about 180,000 grid cells and is used in many of the Figures in this report, required 6.5 hours of CPU time on a 2.8 GHZ Pentium IV Windows XP system. Cases with more grid cells and longer simulation times (the townhouse case simulated 300 S of smoke flow) would clearly benefit from a faster CPU and more memory which are now relatively inexpensive.

Chapter 3

Basics

Smokeyview 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. Menus are accessed by clicking with the right mouse button. 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 B.

To use Smokeyview 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 `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.

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

Normally Smokeyview is run during an FDS run, after the run has completed and 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.

Although specific video card brands cannot be recommended, they should be *high-end* due to Smokeyview’s intensive graphics requirements. These requirements will only increase in the future as more features are added. A video card designed to perform well for “fancy” computer games should do well for Smokeyview. Some apparent bugs in Smokeyview have been found to be the result of problems found in video cards on older computers.

3.1 Manipulating the Scene

The scene may be rotated or translated either directly with the mouse or by using the scene movement dialog box illustrated in Figure 3.1. This dialog box is opened from the `Dialogs>Motion/View` menu item. Clicking on the scene and dragging the mouse horizontally, vertically or a combination of both results in scene rotation or translation depending upon whether `Ctrl` or `Alt` is depressed or not during mouse movement.

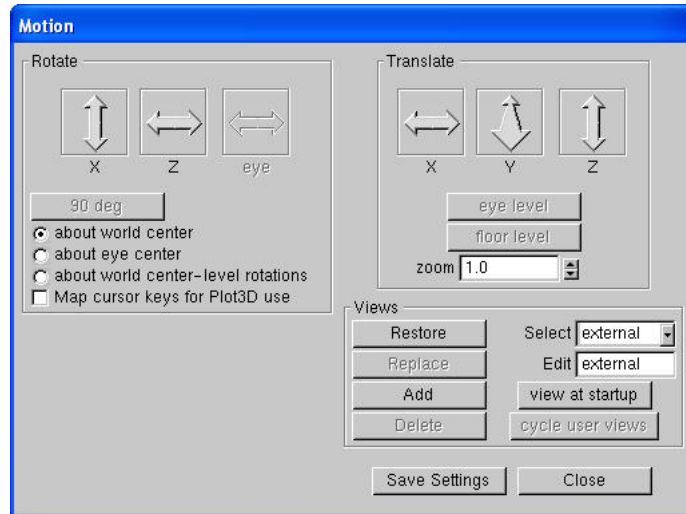


Figure 3.1: Motion Dialog Box. Rotate or translate the scene by clicking an “arrow” and dragging the mouse. The Motion Dialog Box is invoked by selecting `Dialogs>Motion/View`

The modifier keys affect scene movement in the following ways:

No modifier key depressed Horizontal or vertical mouse movement results in scene rotation parallel to the XY or YZ plane, respectively. Note that rotation parallel to the YZ plane is disabled while *eye view* mode is in effect. The view is switched between *eye view* and *world view* by either depressing the e key or by selecting `Options>Rotation`. Equivalently, horizontal or vertical mouse movement results in scene rotation about the Z or X axis, respectively.

SHIFT key depressed The shift key no longer affects scene movement. It was found that the rotation mode was not needed and could cause confusion.

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 into and out of the computer screen along the Y axis.

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

The scene motion dialog box may be used to move the scene in a more controlled manner. For example, clicking the `Rotate X` button in Figure 3.1 and dragging the mouse results in scene rotation about the X (and only the X) axis. Similarly, clicking the mouse on the `Translate Y` button results in scene translation along the Y axis.

The `zoom` edit box allows one to change the perspective or magnification one uses to view the scene. `View` may be used to reset the scene back to either an external, internal (to the scene), or previously saved viewpoint.

Select Rotation Center A pull down list appears in multi-mesh cases allowing one to change the rotation center. Therefore one could rotate the scene about the center of the entire physical domain or about the center of any one particular mesh. This is handy when meshes are defined far apart.

Table 3.1: Keyboard mappings for “eye centered” scene movement

Key	Description
Left/Right cursor	rotate left/right
CTRL + Left/Right cursor	rotate left/right but slower
ALT + Left/Right cursor	slide left/right
SHIFT + Left/Right cursor	rotate 90 degrees clockwise
Down/Up cursor+CTRL	slide backward/forward
CTRL + Down/Up cursor	slide backward/forward but slower
ALT + Down/Up cursor	slide down/up
Insert or Page Down	look down
Home	look level
Page Up	look up
F1	decrease speed
F2	reset speed
F3	increase up
F4	look left
F5	look forward
F6	look right

Select how the scene moves Radio buttons allow one to specify whether the scene is moving (world center or world center - level rotation) or whether the observer is moving (eye center). These modes may also be changed as before by toggling the “e” key. When in “eye center” mode, several key mappings have been added, inspired by popular computer games, to allow easier movement through the scene. For example, the up and down cursor keys allow one to move forward or backwards in the scene while the left and right cursor keys allow one to rotate left or right in the scene. Other keyboard mappings are described in Table 3.1.

Rotation Buttons Rotation buttons are enabled or disabled as appropriate for the mode of scene motion. For example, if “about world center - level rotations” has been selected, then the Rotate X and Rotate eye buttons are disabled. A rotation button labelled 90 deg has been added to allow one to rotate 90 degrees while in “eye center” mode. This is handy when one wishes to move down a long corridor precisely parallel to one of the walls. The first click of “90 deg” snaps the view to the closest forward or side direction while each additional click rotates the view 90 degrees clockwise.

Viewpoint Buttons A viewpoint is the combination of a location and a view direction. Several new buttons have been added to this dialog box to save and restore viewpoints. The scene is manipulated to the desired orientation then stored the **Add** button to add the new viewpoint or the **Replace** button to replace this viewpoint with the currently selected one. To change the view to a currently stored viewpoint, use the **Select** listbox to select the viewpoint and the **Restore** button to activate it. The **Delete** button may be used to remove a viewpoint from the stored list. The **Edit** text box may be used to change the name or label for the selected viewpoint. The **view at startup** button is used to specify the viewpoint that should be set when Smokeview first starts up.

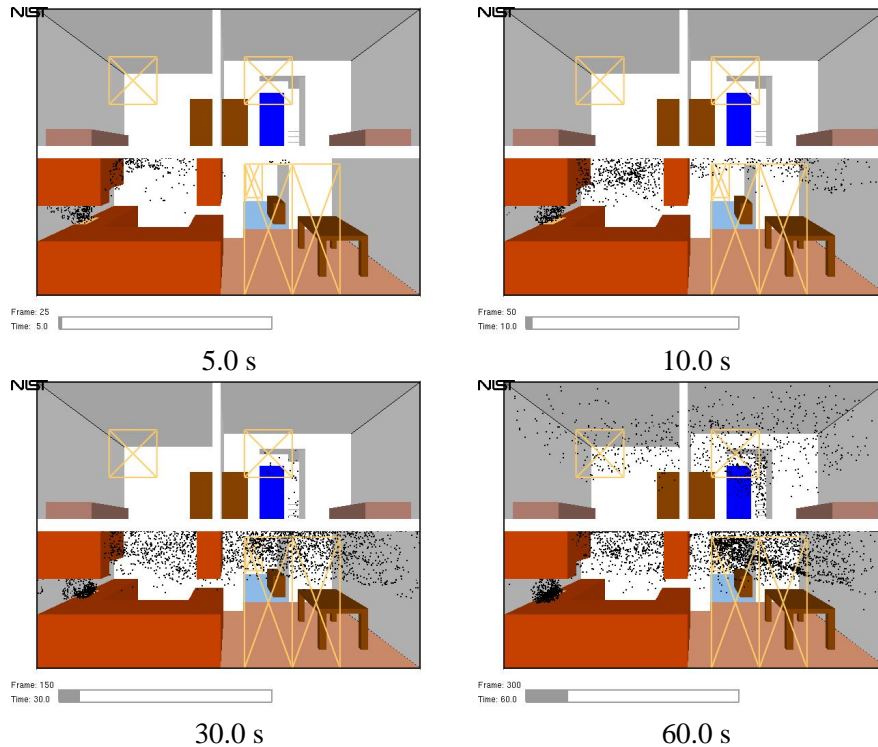


Figure 3.2: Cross section of a townhouse showing first and second floor. Particle file snapshots at four times during the simulation of a townhouse kitchen fire.

3.2 Tracer Particles - Particle 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. Visualizations produced by these data files are described below. The format used to store each of the data files is given in Appendix D.3. The FDS input data file used to generate the following examples may be found at <http://fire.nist.gov/fds/svsamples/thouse3d.data>.

Particle files contain the locations of tracer particles used to visualize the flow field. Colors are selected from a user definable color palette defined using the `COLORBAR` keyword documented in Appendix D.1. Figure 3.2 shows several snapshots of a developing kitchen fire visualized by using particles where particles are colored black. If present, sprinkler water droplets would be colored blue. Particles are stored in files ending with the extension `.part` and are displayed by selecting the entry from the `Load/Unload` menu.

3.3 3D Smoke

Visualizing smoke realistically is a daunting challenge for at least three reasons. First, the storage requirements for describing smoke can easily exceed the disk capacities of present 32 bit operating systems such as Linux, *i.e.* file sizes can easily exceed 2 gigabytes. Second, the computation required both by the CPU and the video card to display each frame can easily exceed 0.1 s, the time corresponding to a 10 frame/s display rate. Third, the physics required to describe smoke and its interactions with itself and surrounding light sources is complex and computationally intensive. Therefore, approximations and simplifications are required to display smoke rapidly.

Smoke visualization techniques such as tracer particles or shaded 2D contours are useful for quantitative

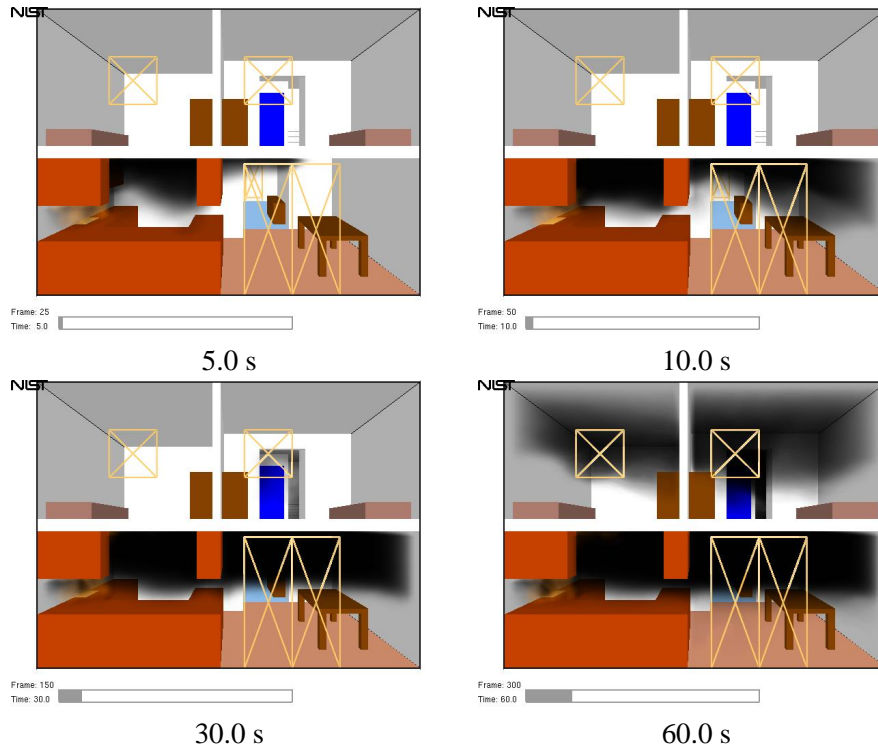


Figure 3.3: Cross section of a townhouse showing first and second floor. Smoke3d file snapshots at various times in a simulation of a townhouse kitchen fire.

analysis but not suitable for virtual reality applications, where displays need to be realistic and fast as well as accurate. The approach taken by Smokeview is to display a series of parallel planes. Each plane is colored black (for smoke) with transparency values pre-computed by FDS using time dependent soot densities computed by FDS corresponding to the grid spacings of the simulation. The transparencies are adjusted in real time by Smokeview to account for differing path lengths through the smoke as the view direction changes. The graphics hardware then combines the planes together to form one image. For some view angles the smoke “planes” may be visible. Changing the viewing angle a small amount will normally make the smoke appear more uniform.

Fire by default is colored a dark shade of orange wherever the computed heat release rate per unit volume exceeds a user-defined cutoff value. The visual characteristics of fire are not automatically accounted for. The user though may use the **3D Smoke** dialog box to change both the color and transparency of the fire for fires that have “non-standard” colors and opacities.

3D smoke files are generated by default in FDS 4. To turn off, add `SMOKE3D= .FALSE.` to the `&MISC` line. Figure 3.3 illustrates this technique for visualizing smoke.

3.4 2D Shaded Contours - Slice Files

Slice files contain results 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 3.4 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 entry from the

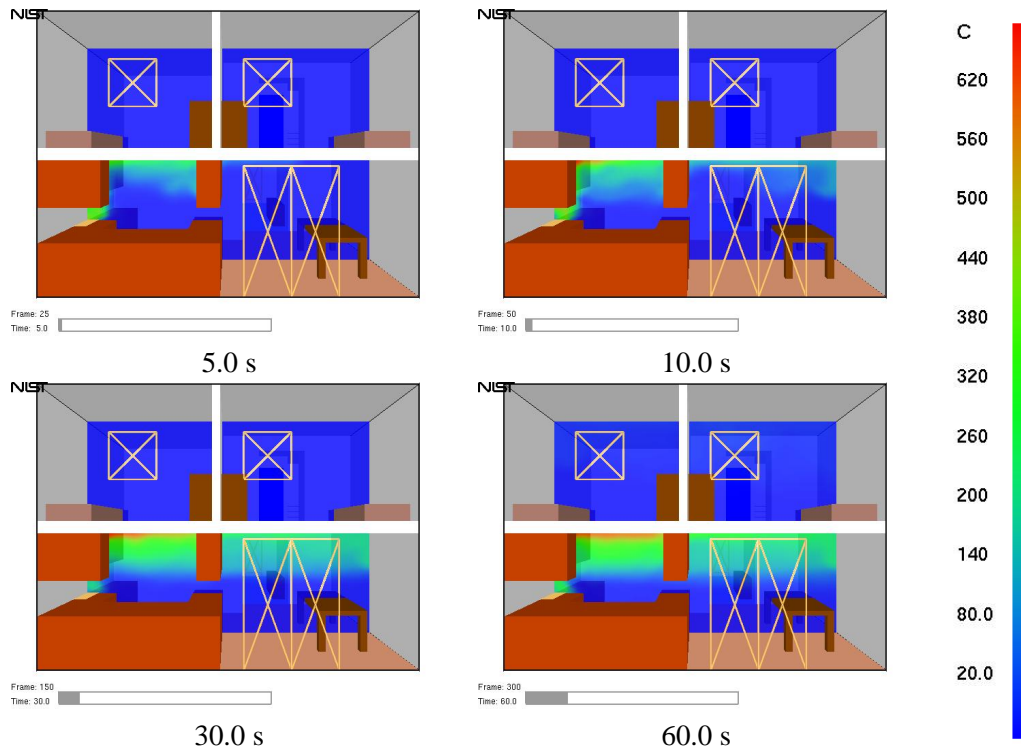


Figure 3.4: Slice file snapshots of shaded temperature contours at various times in a simulation. These contours were generated by adding “&SLCF PBY=1.5,QUANTITY='TEMPERATURE' /” to the FDS input file.

Load/Unload menu.

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

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

Other quantities that can be specified are given in Table 3.2. A more complete list may be found in Ref. [1].

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 3.5 shows a sequence of vector slices corresponding to the shaded temperature contours found in Figure 3.4.

To generate the extra velocity files needed to view vector animations, add `VECTOR=.TRUE.` to the above `&SLCF` line to obtain:

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

3.5 2D Shaded Contours on Solid Surfaces - Boundary Files

Boundary files contain simulation data recorded at blockage or wall surfaces. Continuously shaded contours are drawn for quantities such as wall surface temperature, radiative flux, *etc.* Figure 3.6 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 entry from the **Load/Unload** menu.

Table 3.2: Some output quantities used to generate slice and Plot3D visualizations. A more complete list may be found in Ref.[1]

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

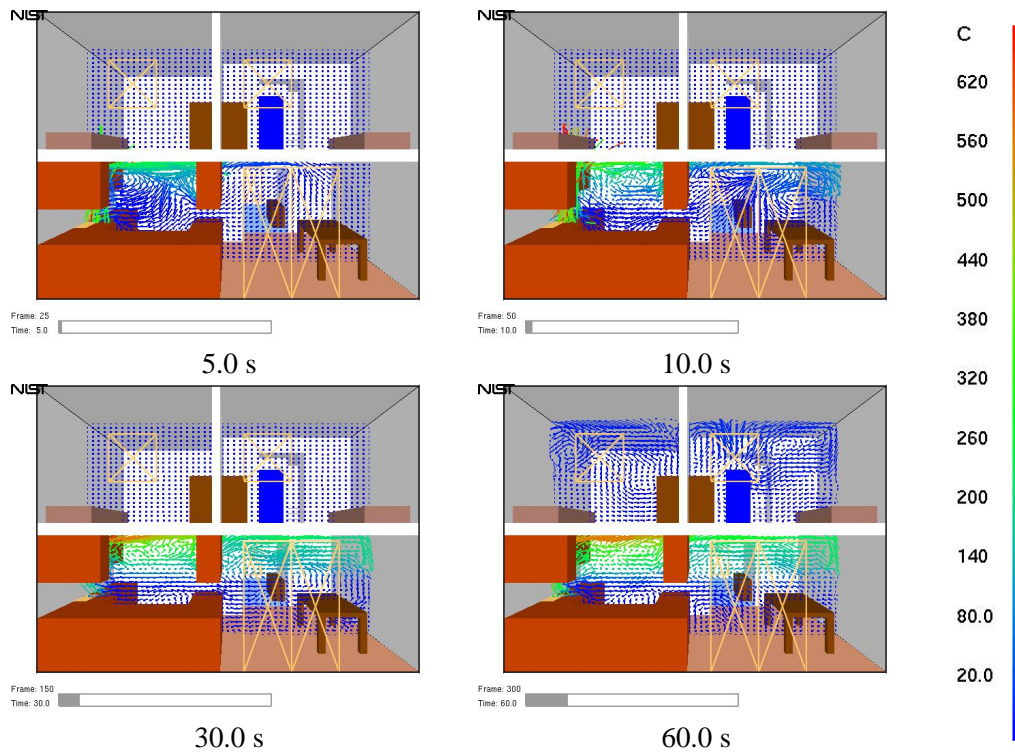


Figure 3.5: Vector slice file snapshots of shaded vector plots. These vector plots were generated by using “&SLCF PB Y=1.5, QUANTITY= 'TEMPERATURE', VECTOR= .TRUE. /”.

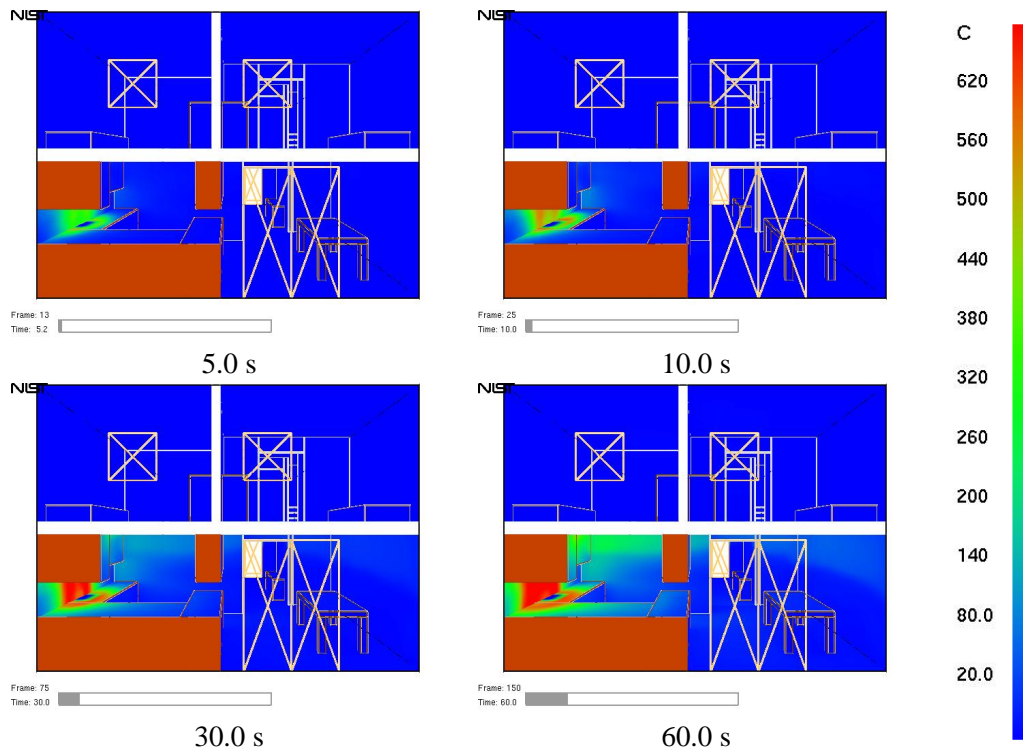


Figure 3.6: Boundary file snapshots of shaded wall temperatures. These snapshots were generated by using “&BNDF QUANTITY= 'WALL_TEMPERATURE' /”.

Table 3.3: Some output quantities used to generate boundary file visualizations. A more complete list may be found in Ref.[1]

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

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

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

Loading a boundary file is a memory intensive operation. The entire boundary file is read in to determine the minimum and maximum data values. These bounds are then used to convert four byte floats to one byte color indices. To drastically reduce the memory requirements, simply specify the minimum and maximum data bounds using the `Set Bounds` dialog box. This should be done before loading the boundary file data. When this is done, memory for the boundary file data is allocated for only one time step rather than for all time steps.

Table 3.3 gives other quantities that can be specified on a &BNDF line. A more complete list of quantities is given in Ref. [1].

3.6 3D Contours - Isosurface Files

The surface where a quantity such as temperature attains a given value is called an isosurface. An isosurface is also called a level surface or 3D contour. Isosurface files contain data specifying isosurface locations for a given quantity at one or more levels. These surfaces are represented as triangles. Isosurface files have file names with extension .iso and are displayed by selecting the desired entry from the `Load/Unload` menu.

Isosurfaces are specified in the FDS input file with the &ISOF keyword. To specify isosurfaces for temperatures of 30°C and 100°C as illustrated in Figure 3.7 add the line:

```
&ISOF QUANTITY='TEMPERATURE',VALUE(1)=30.0,VALUE(2)=100.0 /
```

to the FDS input file. Other quantities may be specified using values given in Table 3.2. A more complete list may be found in Ref. [1]

3.7 Static Data - Plot3D Files

Data stored in Plot3D files use a format developed by NASA [17] and are 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. Other quantities may be stored if desired.

An FDS simulation will typically create Plot3D files at several specified times throughout the simulation. Plot3D data is visualized in three ways: as 2D contours, vector plots and isosurfaces. Figure 3.8a 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 3.8b

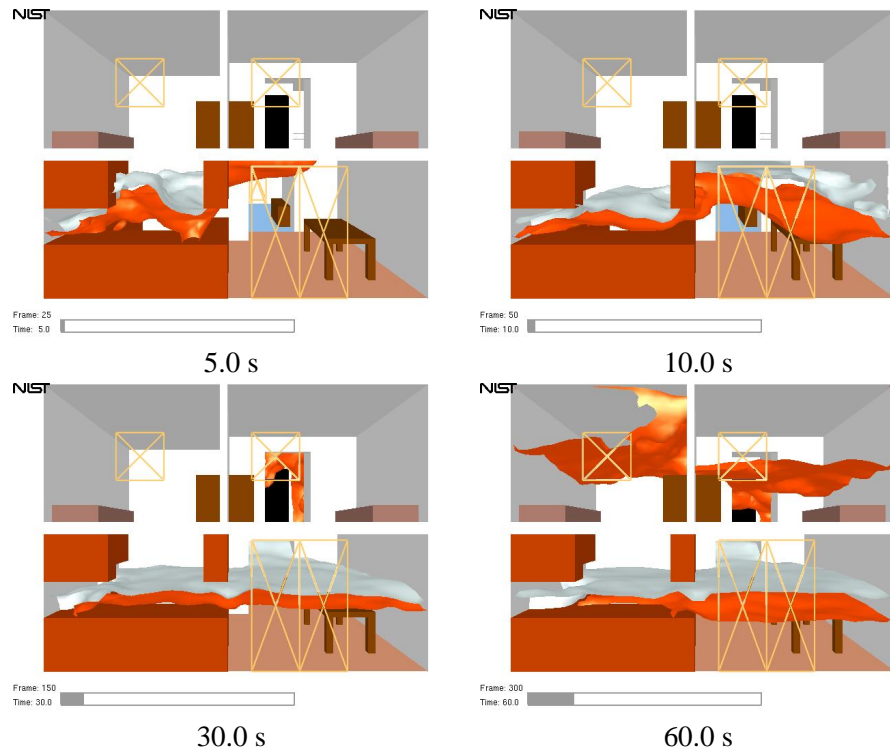
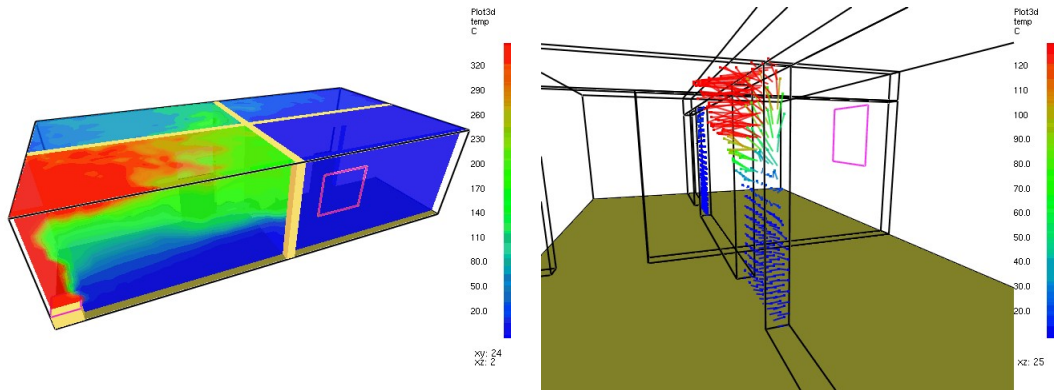


Figure 3.7:

Isosurface file snapshots of temperature levels. The orange surface is drawn where the air/smoke temperature is 30 °C and the white surface is drawn where the air/smoke temperature is 100 °C. These snapshots were generated by adding “&ISO F QUANTITY= 'TEMPERATURE' , VALUE (1)=30 . 0 , VALUE (2)=100 . 0 /” to the FDS input file.



a) shaded 2D temperature contour plots in a vertical plane through the fire and a horizontal plane below the ceiling

b) shaded temperature vector plot in a vertical plane through the doorway. The “a” key may be depressed to alter the vector sizes. The “s” key may be depressed to alter the number of vectors displayed.

Figure 3.8: Plot3D contour and vector plot examples.

shows vectors in a doorway. The vector lengths may be adjusted by depressing the “a” key. 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 3.9 gives an example of isosurfaces for two different temperature levels. Plot3D data are stored in files with extension .q .

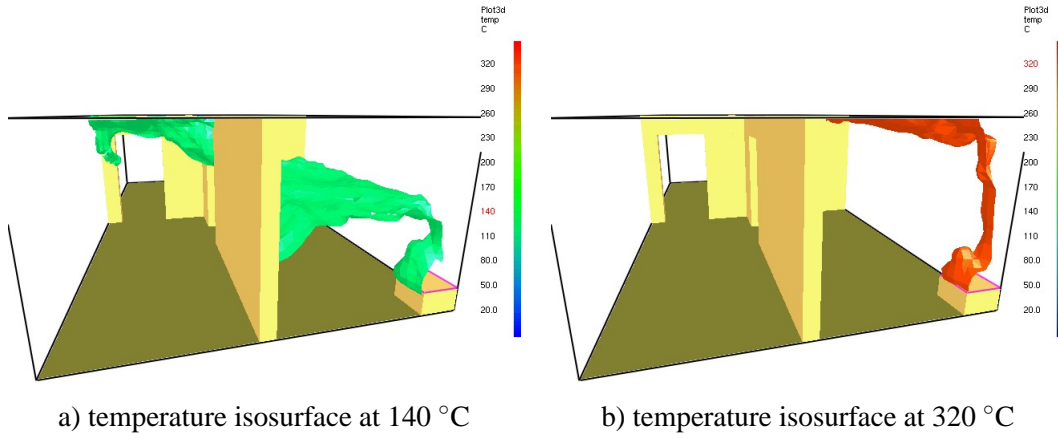


Figure 3.9: Plot3D isocontour example.

Chapter 4

Advanced Features

4.1 Manipulating the Scene Automatically - The Touring Option

The touring option allows one to specify arbitrary paths or tours through or around a Smokeview scene. One may then view the scenario from the vantage point of an observer moving along one of these paths. Alternatively, one may observe the motion along all of the paths simultaneously from an external vantage point. These paths may be used to represent the movement of building occupants or first responders during a fire incident, or to highlight key portions or features of a scenario during presentations. A tour may also be used to observe time dependent portions of the scenario such as blockage/vent opening and closings. The default view direction is towards the direction of motion. The view direction and path tension may be changed with the **Advanced Settings** dialog box.

When Smokeview starts up it creates a tour, called the “circle tour” which surrounds the scene. The “circle” tour and a user defined tour are illustrated in Figure 4.1. The circle tour is similar to the Tour menu option found in earlier versions of Smokeview. The user may modify the “circle” tour or define their own tours by using the **Tour** dialog box illustrated in Figure 4.2. The user places several points or keyframes in or around the scene. Smokeview creates a smooth path going through these points.

Tour Settings An existing tour may be modified by selecting it from the **Select tour:** listbox found in the Tour dialog box. A new tour may be created by clicking the **Add Tour** button. A newly created tour goes through the middle of the Smokeview scene starting at the front left and finishing at the back right. A tour may also be modified by editing the text entries found in the “local” preference file, casename.ini under the **TOUR** keyword.

The speed traversed along the tour is determined by the time value assigned to each keyframe. If the **Constant Speed** checkbox is checked then these times are determined given the distance between keyframes and the velocity required to traverse the entire path in the specified time as given by the “start time” and “stop time” entries found in the **Advanced Settings** dialog box.

Three different methods for viewing the scene may be selected. To view the scene from the point of view of the selected tour, check the **View From Tour Path** checkbox. To view the scene from a keyframe (to see the effect of editing changes), select the **View From Selected Keyframe** checkbox. Unchecking these boxes returns control of scene movement to the user.

Keyframe Settings A keyframe may be selected for editing by either clicking it with the mouse or by using the **Next** or **Previous** buttons. Keyframe positions may then be modified by changing data in the t, X, Y or Z edit boxes. A different view direction may be selected using the **Advanced Settings** dialog box.

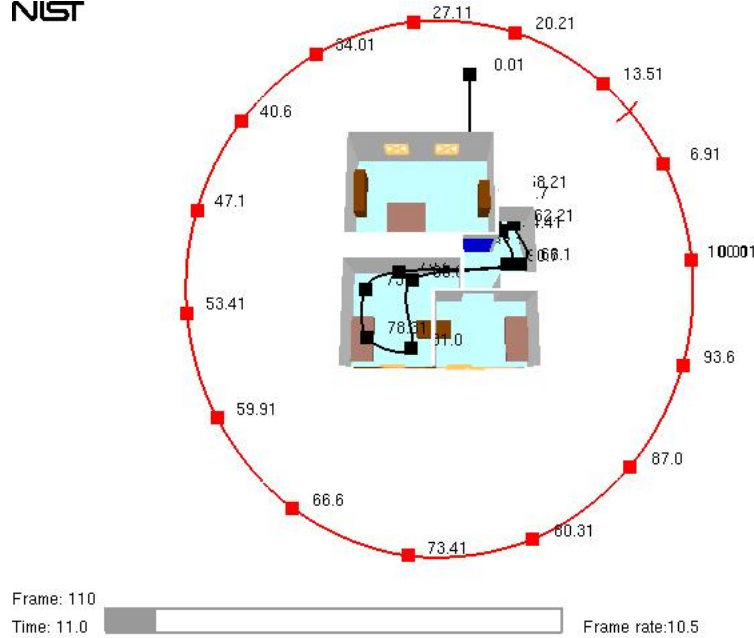


Figure 4.1: Overhead view of townhouse example showing the default “Circle” tour and a user defined tour. The square dots indicate the keyframe locations. Keyframes may be edited using the Touring or Advanced Touring dialog boxes.

A new keyframe is created by clicking the **Add** button. It is formed by averaging the positions and view directions of the current and next keyframes. If the selected keyframe is the last one in the tour then a new keyframe is added beyond the last keyframe.

A keyframe may be deleted by clicking the **Delete** button. There is no **Delete Tour** button. A tour may be deleted by either deleting all of its keyframes or by deleting its entry in the casename.ini file.

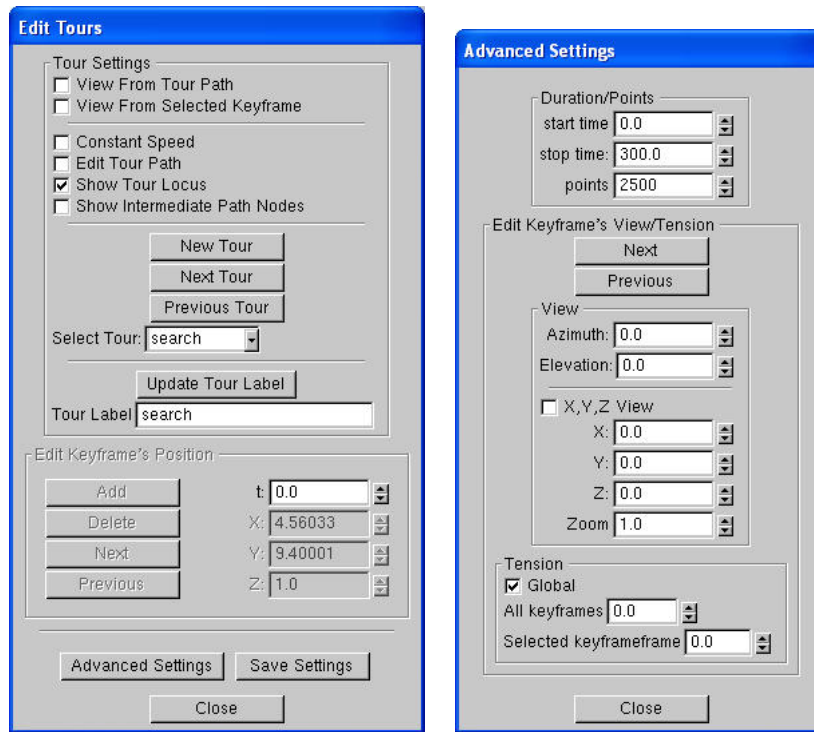
Advanced Settings The **Advanced Settings** dialog box is only necessary if one wishes to override Smokeview’s choice of view direction (along the direction of motion) or tension settings. This dialog box is opened by clicking the **Advanced Settings** button contained in the **Keyframe Settings** dialog box.

A view direction may be defined at each keyframe by either setting direction angles relative to the path (an azimuth and an elevation angle) or by setting a direction relative to the scene geometry (a cartesian (X,Y,Z) view direction).

Path relative view directions are enabled by default. To define a cartesian view direction, select the **X,Y,Z View** check box and edit the X, Y and Z View edit boxes to change the view location. To define an path relative view direction, uncheck the **X,Y,Z View** check box and edit the Azimuth and Elevation edit boxes. Checking the **View From Selected Keyframe** checkbox allows one to see the effects of the above view changes from the keyframe being edited. To see the effect of a change in one of the keyframe’s parameters, uncheck the **View From Selected Keyframe** checkbox and position the tour locator (vertical and horizontal red lines) near the keyframe. The horizontal red line always points in the view direction.

Spline tension settings may also be changed using the **Advanced Settings** dialog box, though normally this is not necessary except when one wishes abrupt rather than smooth path changes. Kochanek-Bartels[16] splines (piecewise cubic Hermite polynomials) are used to represent the tour paths.

The cubic Hermite polynomials for each interval are uniquely specified using a function and a derivative



a) basic options

b) advanced options

Figure 4.2: The Touring dialog boxes may be used to select tours or keyframes, change the position or view direction at each keyframe and change the tension of the tour path.

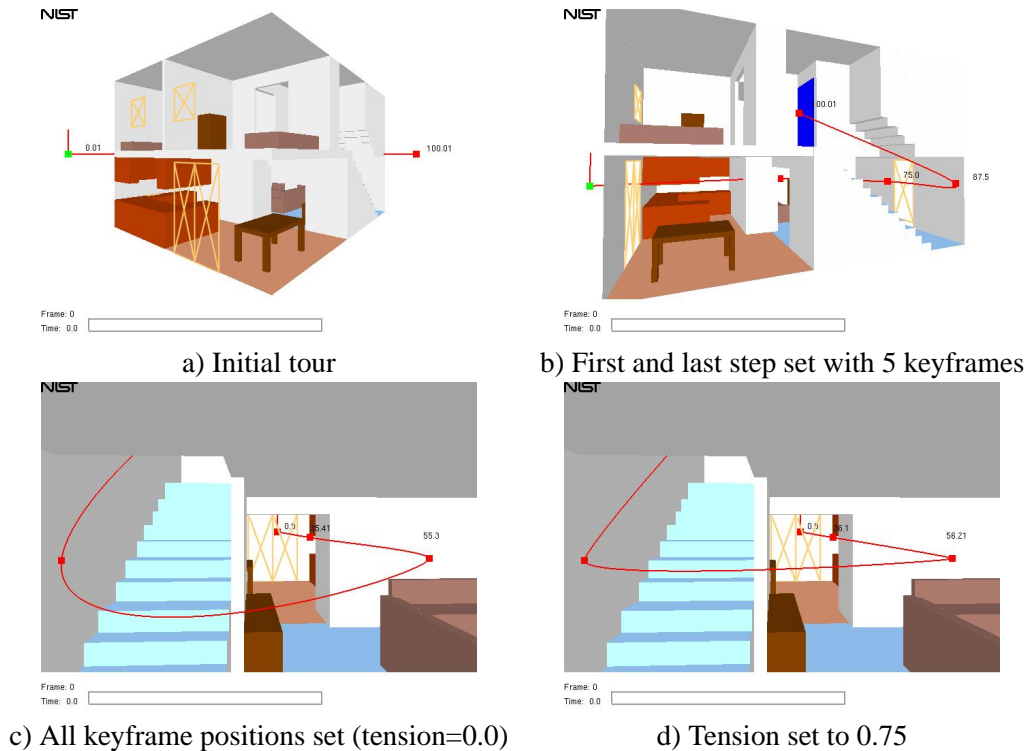


Figure 4.3: Tutorial examples for Tour option.

at both endpoints of the interval (*i.e.* 4 data values). These derivatives are computed in terms of three parameters referred to as *bias*, *continuity* and *tension*. Each of these parameters range from -1 to 1 with a default value of 0. The tension value may be set for all keyframes at once (by checking the **Global** checkbox) or for each keyframe separately. The bias and continuity values are set to zero internally by Smokeview. A tension value of 0 is set by default, a value of 1 results in a linear spline.

Setting up a tour The following steps give a simple example of setting up a tour in the townhouse scenario. The tour will begin at the back of the house, go towards the front door and then end at the top of the stairs. These steps are illustrated in Figure 4.3.

1. Start by clicking the `Dialog>Tours...` menu item which opens up the `Edit Tours` dialog box.
2. Click on the `New Tour` button in the Tour dialog box. This creates a tour, illustrated in Figure 4.3a, starting at the front left of the scene and ending at the back right. This tour has two keyframes. The elevation of each keyframe is halfway between the bottom and top of the scene.
3. Click on the `Edit Tour Path` checkbox. This activates buttons that allows the user to edit the properties of each individual keyframe. Click on the square dot at the back of the townhouse. This is the first keyframe. Click on the square dot at the back of the townhouse. This is the first keyframe. Change the “Z” value to 1.0. Click on the second dot and change its “Z” value to 1.0.
4. Click on the `Add` button, found inside the “Edit Keyframe’s Position” panel, three times. This will add three more key frames to the tour which will be needed so that the path bends up the stairs. You should now have five keyframes.

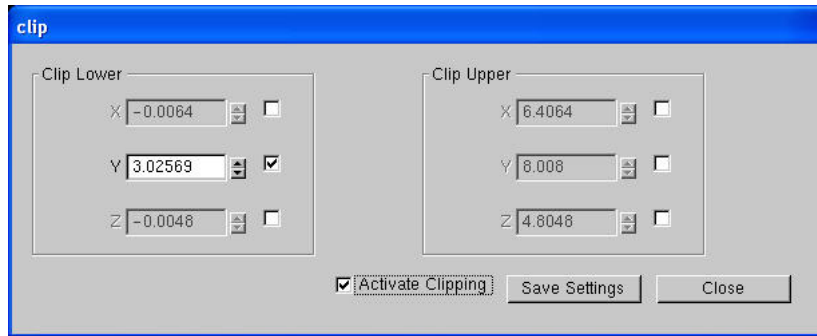


Figure 4.4: Clipping dialog box

5. Move the first keyframe at the back of the townhouse near the double door by setting X, Y, Z positions to (3.8,-1.0,1.6). Move the last keyframe to the top of the steps by setting X, Y, Z positions to (6.0,3.6,4.1). The path should now look like Figure 4.3b.
6. Move the second, third and fourth keyframes to positions (4.0,4.0,1.6), (4.0,6.8,1.6) and (6.0,6.8,1.6). The path should now look like Figure 4.3c.
7. Click on the `Advanced Settings` button. Check the `Global` checkbox and set the `All keyframes` edit box to 0.5. This “tightens” up the spline curve reducing the dip near the stairs that occurs with the tension=0.0 setting. The path should now look like Figure 4.3d.
8. Click on the `Save Settings` button to save the results of your editing changes.
9. To see the results of the tour, click on the “View From Tour Path” check box.

The point of view of the observer on this path is towards the direction of motion. Next the view direction will be changed to point to the side while the observer is on the first floor.

1. Click on the `Advanced Settings` button if it is not already open.
2. Uncheck the “View From Tour Path” checkbox in the “tour” dialog box and make sure that the `X, Y, Z View` checkbox is unchecked.
3. Click on the dot representing the first keyframe. Then change “Azimuth” setting to 90 degrees. To see the results of the change, go back and check the “View From Tour Path” checkbox.
4. Uncheck the “View From Tour Path” checkbox again. Now select the second and third keyframes and change their azimuth settings to 90 degrees.

With this second set of changes, the observer will look to the side as they pass through the kitchen and living room. The observer will look straight ahead as they go up the stairs.

4.2 Clipping Scenes

It is difficult to view interior portions of a scene when modelling complicated geometries. Scene clipping allows one to hide portions of the scene by defining up to six clipping planes. All portions of the scene with “x” coordinates smaller the “x lower” clipping plane will be hidden. Portions of the scene with “x”

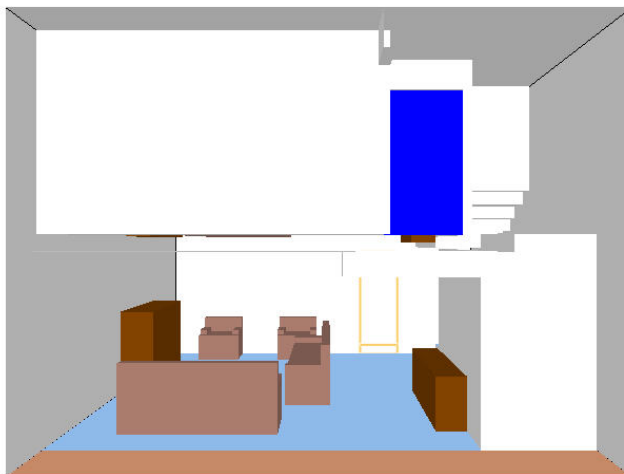


Figure 4.5: Clipped scene corresponding to Clip Dialog box given in Figure 4.4

coordinates larger than the “x upper” clipping plane will be hidden. The “y lower”, “y upper”, “z lower” and “z upper” clipping planes behave similarly. These clipping planes are specified by using the clipping dialog box which is opened by selecting the `Dialogs>Clip Geometry` menu item. Figure 4.4 shows this dialog box with the “y lower” clipping plane activated. The corresponding clipped scene is illustrated in Figure 4.5. Note that this clipped scene is the same townhouse kitchen fire scenario used to generate various plots presented earlier in this report.

4.3 Setting Data Bounds

Normally, Smokeview determines data bounds automatically when it loads data. Sometimes, however, it is desirable to override Smokeview’s choice. This allows for consistent color shading when displaying several data files simultaneously.

The `Set Bounds...` dialog box is opened from the `Dialogs` menu. Each file type in Figure 4.6 (slice, particle, Plot3D *etc*) has a set of “radio buttons” for selecting the variable type that data bounds are to be applied to. 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. Radio buttons adjacent to the edit boxes determine what type of bounds should be applied. The `Update` and `Reload` buttons are pressed to make the new bounds take effect.

The `Plot3D` and `Slice File` portions of the bounds dialog box have additional controls used to chop or hide data. The settings used in Figure 4.7 were used to generate the ceiling jet visualized in Figure 4.8. Data values less than 140°C are chopped or not drawn in the figure.

The **Boundary File** portion of the dialog box has an **ignition** checkbox which allows one to visualize when and where the blockage temperature exceeds its ignition temperature.

The bounds dialog for `PLOT3D` display allows one to select between three different types of contour plots: shaded, stepped and line contours.

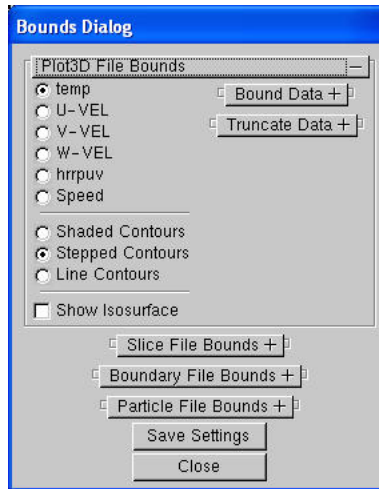


Figure 4.6: Bounds dialog box showing PLOT3D file options. Select a variable and a bounds type “check box/radio button”, then enter a lower and/or upper bound. Data may be excluded from the plot by selecting a “Truncate” bound. Select type of contour plot to be displayed. Press “Reload...” or “Update” for the new bounds to take effect.

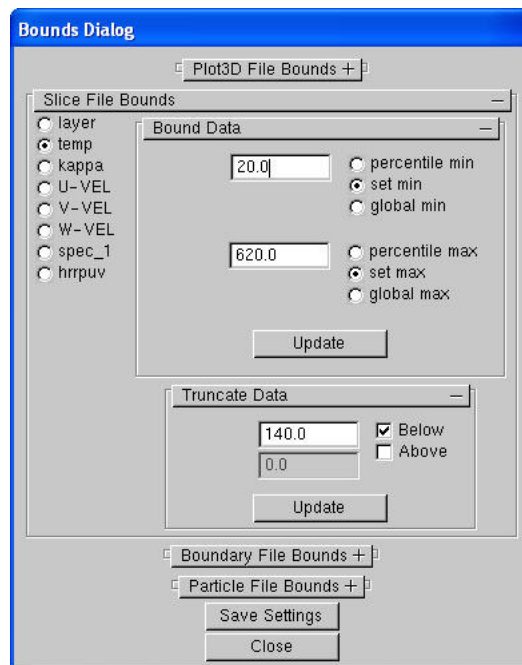


Figure 4.7: Bounds dialog box showing slice file options. Select a variable and a bounds type “check box/radio button”, then enter a lower and/or upper bound. Data may be excluded from the plot by selecting a “Truncate” bound. Press “Reload...” or “Update” for the new bounds to take effect.

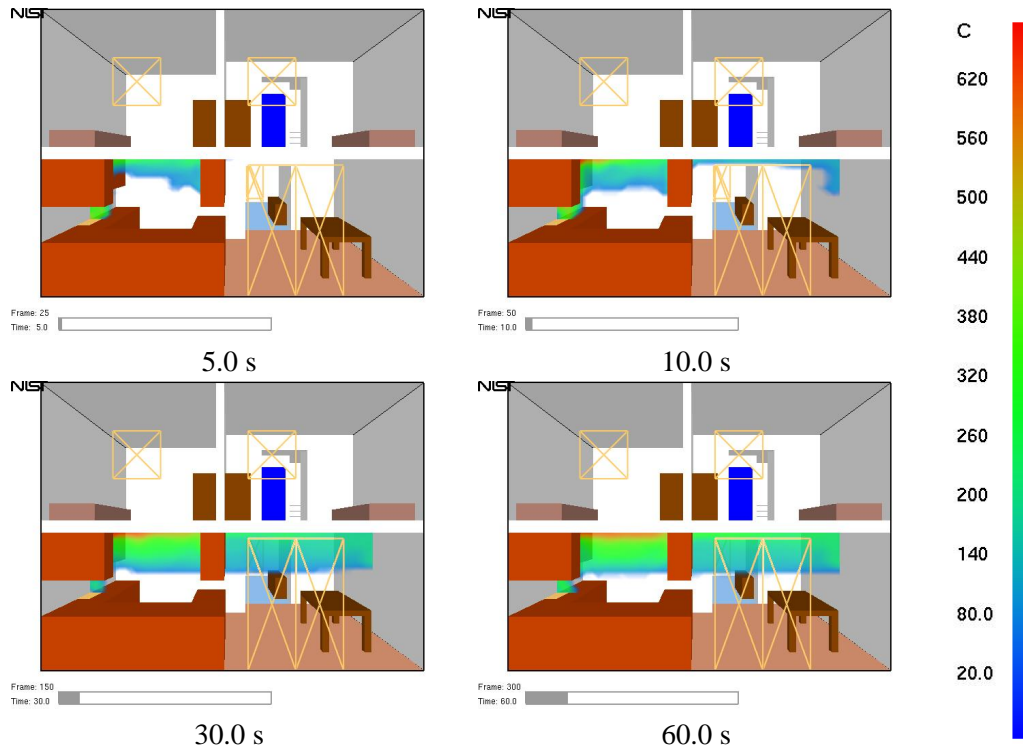


Figure 4.8: Ceiling jet visualization created by “chopping data” below 140 °C using the Bounds Dialog Box as illustrated in Figure 4.7.

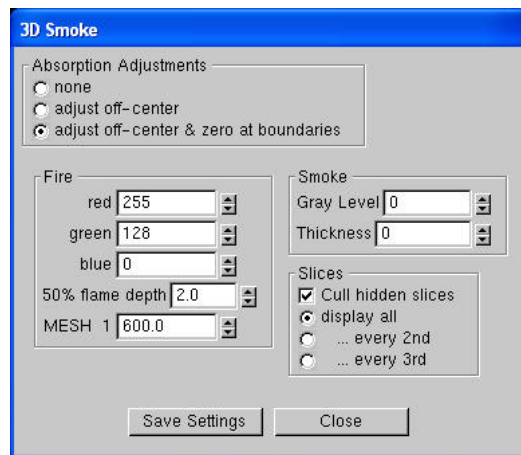


Figure 4.9: Dialog box for setting 3D smoke options.

4.4 3D Smoke Options

Figure 4.9 allows one to override Smokeview’s choice for several of the 3D smoke parameters. The user may specify the color of the fire and the gray level of the smoke. A gray level of n where n ranges from 0 to 7 results in a color of $(2^n, 2^n, 2^n)$ where the three components represent red, green and blue contributions. The **hrrpv cutoff** input refers to the heat release rate required at a node before Smokeview will color the node as fire rather than smoke. The **50% flame depth** allows one to specify the transparency or optical thickness of the fire (for visualization purposes only). A small value results in opaquely drawn fire while a large value results in a transparently drawn fire. The **Absorption Parameter** setting refers to how the smoke slices are drawn. The **adjust off-center** setting causes Smokeview to account for non-axis aligned paths. The **adjust off-center + zero at boundary** accounts for off center path lengths and zeros smoke density at boundaries in order to remove graphical artifacts.

4.5 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 entry 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 default. If the “s” is depressed then every other vector is skipped.

4.6 Display Options

The display dialog box, illustrated in Figure 4.10, allows one to set various options to control the display or “look” of the Smokeview scene. It may be invoked by selecting the `Dialogs>Display` menu item.

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.

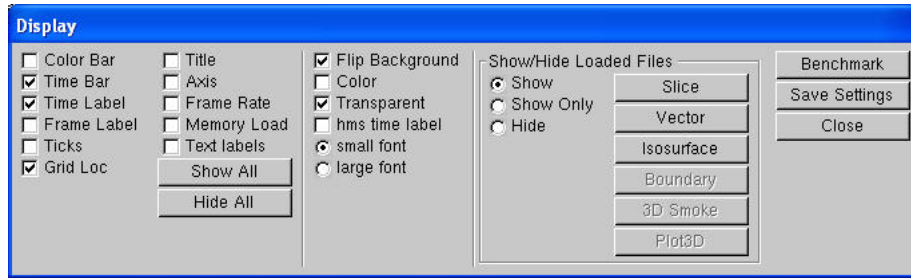


Figure 4.10: Dialog Box for setting miscellaneous Smokeview scene properties

This dialog box also allows one to show or hide loaded data files.

4.7 Texture Maps

Texture mapping is a technique used by Smokeview to make a scene appear more realistic by pasting images onto obstructions or vents. For example, to apply a wood paneling image to a wall, add the keywords `TEXTURE_MAP='paneling.jpg'`, `TEXTURE_WIDTH=1.`, `TEXTURE_HEIGHT=2.` to the `&SURF` line where **paneling.jpg** is the JPEG file containing the texture map (SGI users should use RGB files instead of JPEG) and **TEXTURE_WIDTH** and **TEXTURE_HEIGHT** are the characteristic dimensions of the texture map in meters. Note that the image will not appear when Smokeview first starts up. The user must select the texture maps using the Show/Hide menu.

One can create texture maps using a digital camera or obtain them commercially. The maps should be *seamless* so that no breaks or seams appear when the maps are tiled on a blockage or vent. This is important, because Smokeview replicates the image as often as necessary to cover the blockage or vent.

When the texture does have a pattern, for example windows or bricks, the keyword `TEXTURE_ORIGIN` may be used to specify where the pattern should begin. For example,

```
&OBST XB=1.0,2.0,3.0,4.0,5.0,7.0,SURF_ID='wood paneling',
      TEXTURE_ORIGIN=1.0,3.0,5.0 /
```

will apply paneling to an obstruction whose dimensions are 1 m by 1 m by 2 m, such that the image of the paneling will be positioned at the point (1.0,3.0,5.0). The default value of `TEXTURE_ORIGIN` is (0,0,0), and the global default can be changed by added a `TEXTURE_ORIGIN` statement to the `MISC` line.

Figure 4.11 shows a simple application of a texture applied to two different blockages and a vent. The same jpeg file was used in two different `&SURF` lines so that the texture could be stretched by differing amounts (using the `TEXTURE_WIDTH` parameter.) The FDS data file used to create this Figure follows.

```
&HEAD CHID='sillytexture',TITLE='Silly Test Case' /
&MISC TEXTURE_ORIGIN=0.1,0.1,0.1 /

&GRID IBAR=20,JBAR=20,KBAR=20 /
&PDIM XBAR=1.0,YBAR=1.0,ZBAR=1.0 /
&TIME TWFIN=0. /
&SURF ID      = 'TEXTURE 1'
      TEXTURE_MAP= 'nistleft.jpg'
      TEXTURE_WIDTH=0.6
      TEXTURE_HEIGHT=.20
```



Figure 4.11: Texture map example. The same texture was applied to two different blockages and a vent (with different widths) by assigning different TEXTURE_WIDTH parameters in the input file.

```

    rgb=0.0,0.0,1.0
/

&SURF ID      = 'TEXTURE 2'
  TEXTURE_MAP= 'nistleft.jpg'
  TEXTURE_WIDTH=0.4
  TEXTURE_HEIGHT=.20
  rgb=1.0,0.0,1.0
/

&OBST XB=0.1,0.30, 0.1,0.7, 0.1,0.3, SURF_ID='TEXTURE 1' /
&OBST XB=0.5,0.9, 0.3,0.7, 0.1,0.5 SURF_ID='TEXTURE 2'
    rgb=1.0,0.0,0.0 TEXTURE_ORIGIN=0.5,0.3,0.1 /

&VENT XB=0.00,0.00,0.2,0.8,0.20,0.40,SURF_ID='TEXTURE 1'
    rgb=0.0,1.0,0.0 TEXTURE_ORIGIN=0.0,0.2,0.2 /
&VENT XB=0.30,0.90,0.0,0.0,0.30,0.50,SURF_ID='TEXTURE 1'
    TEXTURE_ORIGIN=0.90,0.0,0.3 /

```

4.8 Debugging FDS Input Files

One of the most difficult tasks in setting up an FDS input file is defining the geometry (blockages, vent locations *etc*) properly. Smokeview may be used to debug FDS input files by making short runs and observing whether blockages, vents and other geometric features of a run are located correctly. Blockages may then be created or changed. This is done with the “Edit Blockage” dialog box called from the Option menu.

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 the stop time to 0.0 using `TWFIN=0.0` on the `&TIME` line. This causes FDS to read the input file and create a `.smv` file without performing lengthy startup calculations.
2. After the input file has been modified, run the FDS model (for details see the FDS User's Guide[1])
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`.
4. Make corrections if necessary. Use the `COLOR` or `RGB` option² of the `OBST` keyword 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 COLOR='RED' /
```

or

```
&OBST XB=0.0,1.0,0.0,1.0,0.0,1.0 RGB=1.0,0.0,0.0 /
```

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

5. If corrections are unnecessary, then change the `TWFIN` keyword back to the desired final simulation time, remove any unnecessary FDS `COLOR` keywords and run the case.

Examining and/or Editing Blockages Blockages may be examined or changed by selecting `Dialogs/Edit Geometr` which opens up the dialog box illustrated in Figure 4.12. Text edit boxes allow one to define comments (any text to the right of the closing `/` on an `&OBST` line and pull down menus for linking surfaces (`&SURF` lines) to obstructions. The surface choices are obtained from the input file or the database file if one is specified in the input file.

Note, the blockage editor does not work when clipping planes are active.

The user may associate one, three or six surfaces with an obstruction. If the “3 wall” radio button is selected then materials for the ceiling (upper surface), walls (vertical surfaces) and floor (lower surface) may be specified. The “color” check box allows the inputted or default color (manilla) to be displayed on blockage surfaces.

Associating unique colors with each surface allows the user to quickly determine whether blockages are defined with the proper surfaces. One can then verify that these modelling elements have been defined and positioned as intended.

The blockage editor works on one mesh at a time. For multi-mesh cases, use the Mesh list-box to select the desired mesh or simply click on a blockage in the mesh to switch to that mesh. The pull-down menu is necessary for meshes without blockages.

Position coordinates entered into the text boxes are “snapped” to the nearest grid line. Materials appearing under “Surface Type” come from those materials defined in either the input data file or the database file. Either one, three or six wall properties can be entered by selecting the appropriate radio button.

²The FDS `COLOR` keyword may have values: `RED`, `BLUE`, `GREEN`, `MAGENTA`, `CYAN`, `YELLOW`, `WHITE` or `BLACK`. The `RGB` keyword uses three floating point inputs ranging from 0.0 to 1.0 to specify the red, green and blue components of color. `COLOR` replaces `BLOCK.COLOR` and `VENT.COLOR`.

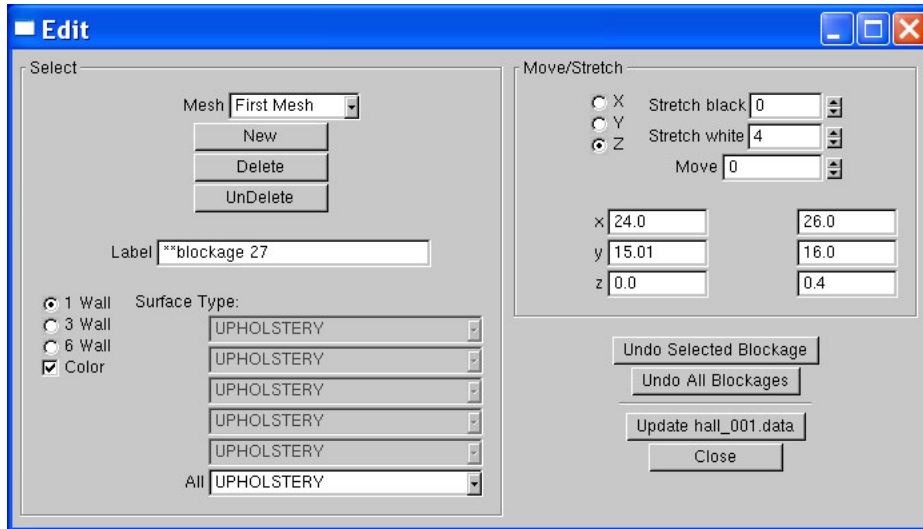


Figure 4.12: Blockage Edit Dialog Box. Click on ‘New’ to create a new blockage or click on an existing blockage in the Smokeview scene to change its size or location. Change the size or location by entering blockage bounds directly or clicking an ‘arrow’ and/or direction buttons. Note, the blockage editor does not work when clipping planes are active.

4.9 Making Movies

A movie can be made of a Smokeview animation by converting the visualized scene into a series of PNG or JPEG files, one file for each time step and then combining the individual images together into one movie file. More specifically:

1. Set up Smokeview by orienting the scene and loading the desired data files.
2. Select the `Options/Render` menu and pick the desired frame skip value. The more frames you include in the animation, the smoother it will look. Of course more frames result in larger file sizes. Choose fewer frames if the movie is to appear on a web site.
3. Use a program such as the Antechinus Media Editor available at <http://www.c-point.com> to assemble the JPEGs or PNGs rendered in the previous step into a movie file.

The default Smokeview image size is 640×480 . This size is fine if the movie is to appear in a presentation located on a local hard disk. If the movie is to be placed on a web site then care needs to be taken to insure that the generated movie file is a reasonable size. Two suggestions are to reduce the image size to 320×240 or smaller by modifying the `WINDOWWIDTH` and `WINDOWHEIGHT` smokeview.ini keywords and to reduce the number of frames to 300 or less by skipping intermediate frames *via* the `Options/Render` menu.

Sometimes when copying or “capturing” a Smokeview scene it is desirable, or even necessary, to have a margin around the scene. This is because the capturing system does not include the entire scene but itself captures an indented portion of the scene. To indent the scene, either press the “h” key or select the `Option>Viewpoint>Offset Window` menu item. The default indentation is 45 pixels. This may be changed by adding/editing the `WINDOW_OFFSET` keyword in the smokeview.ini file.

4.10 Annotating the Scene

Smokeview rendered JPEG or PNG image files may be annotated with any package that edits these types of files. Tick marks and label annotation can be placed within the 3D scene and moved around as the scene is moved. This would be difficult to duplicate with an external image editor. In addition, axis labels are displayed, if desired, by selecting `Show/Hide` > `Labels` > `Axis labels`.

The current release of FDS places tick marks and labels documenting the scene dimensions. To replace or customize these annotations add the `TICK` keyword to a `.smv` file using the following format:

```
TICKS
xb yb zb xe ye ze nticks
ticklength tickdir r g b tickwidth
```

where `xb`, `yb`, and `zb` are the x, y and z coordinates of the first tick; `xe`, `ye` and `ze` are the x, y and z coordinates of the last tick and `nticks` is the number of ticks. The coordinate dimensions are in physical units, the same units used to set up the FDS geometry. The parameter `ticklength` specifies the length of the tick in physical units. The parameter `tickdir` specifies the tick direction. For example 1(-1) places ticks in the positive(negative) x direction. Similarly, 2(-2) and 3(-3) place ticks in the positive(negative) y and positive(negative) z directions.

The color parameters `r`, `g` and `b` are the red, green and blue components of the tick color each ranging from 0.0 to 1.0. The foreground color (white by default) may be set by setting any or all of the `r`, `g` and `b` components to a negative number. The `tickwidth` parameter specifies tick width in pixels. Fractional widths may be specified.

The `LABEL` keyword allows a text string to be added within a Smokeview scene. The label color and start and stop appearance time may also be specified. The format is given by

```
LABELS
x y z r g b tstart tstop
label
```

where (`x`, `y`, `z`) is the label location in cartesian coordinates and `r`, `g`, `b` are the red, green and blue color components ranging from 0.0 to 1.0. Again, if a negative value is specified then the foreground color will be used instead (white is the default). The parameters, `tstart` and `tstop` indicate the time interval when the label is visible. The text string is specified on the next line (`label`).

Figure 4.13 shows how the `LABEL` and `TICKS` keywords can be used together to create a “ruler” with major and minor tick marks. These ticks and labels were created using:

```
TICKS
0.0 0.0 0.0 8.0 0.0 0.0 5
0.5 -2.0 -1. -1.0 -1.0 4.0
TICKS
1.0 0.0 0.0 9.0 0.0 0.0 5
0.25 -2.0 -1. -1.0 -1.0 4.0
TICKS
0.0 0.0 0.0 0.0 0.0 2.0 3
0.5 -1.0 -1. -1.0 -1.0 4.0
TICKS
0.0 0.0 0.0 0.0 4.0 0.0 5
0.5 -1.0 -1. -1.0 -1.0 4.0
LABEL
0.0 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
```

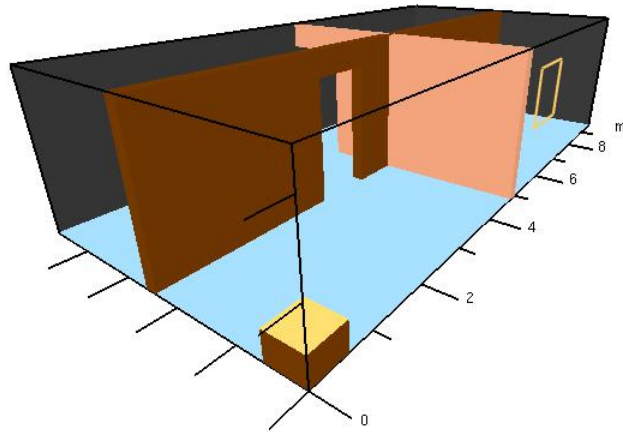


Figure 4.13: Annotation example using the TICKS and LABEL keyword.

```

0
LABEL
2.0 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
2
LABEL
4.0 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
4
LABEL
6.0 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
6
LABEL
8.0 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
8
LABEL
9.5 -0.6 0.0 -1.0 0.0 0.0 0.0 20.0
m

```


Chapter 5

Summary

Often fire modeling is looked upon with skepticism because of the perception that eye-catching images shroud the underlying physics. However, if the visualization is done well, it can be used to assess the quality of the simulation technique. The user of FDS chooses a numerical grid on which to discretize the governing equations. The more grid cells, the better but more time-consuming the simulation. The payoff for investing in faster computers and running bigger calculations is the proportional gain in realism manifested by the images. There is no better way to demonstrate the quality of the calculation than by showing the realistic behavior of the fire.

Up to now, most visualization techniques have provided useful ways of analyzing the output of a calculation, like contour and streamline plots, without much concern for realism. A rainbow-colored contour map slicing down through the middle of a room is fine for researchers, but for those who are only accustomed to looking at real smoke-filled rooms, it may not have as much meaning. Good visualization needs to provide as much information as the rainbow contour map but in a way that speaks to modelers and non-modelers alike. A good example is smoke visibility. Unlike temperature or species concentration, smoke visibility is not a local quantity but rather depends on the viewpoint of the eye and the depth of field. Advanced simulators and games create the illusion of smoke or fog in ways that are not unlike the techniques employed by fire models to handle thermal radiation. The visualization of smoke and fire by Smokeview is an example of the graphics hardware and software actually computing results rather than just drawing pretty pictures. A common concern in the design of smoke control systems is whether or not building occupants will be able to see exit signs at various stages of a fire. FDS can predict the amount of soot is located at any given point, but that doesn't answer the question. The harder task is to compute on the fly within the visualization program what the occupant would see and not see. In this sense, Smokeview is not merely a "post-processor," but rather an integral part of the analysis.

Bibliography

- [1] K.B. McGrattan and G.P. Forney. Fire Dynamics Simulator (Version 4), User's Guide. NIST Special Publication 1019, National Institute of Standards and Technology, Gaithersburg, Maryland, July 2004.
- [2] K.B. McGrattan (editor). Fire Dynamics Simulator (Version 4), Technical Reference Guide. NIST Special Publication 1018, National Institute of Standards and Technology, Gaithersburg, Maryland, July 2004.
- [3] G. P. Forney and K. B. McGrattan. User's Guide for Smokeview Version 3.1: A Tool for Visualizing Fire Dynamics Simulation Data. NISTIR 6980, National Institute of Standards and Technology, April 2003.
- [4] D. Madrzykowski and R.L. Vettori. Simulation of the Dynamics of the Fire at 3146 Cherry Road NE, Washington, DC May 30, 1999. Technical Report NISTIR 6510, Gaithersburg, Maryland, April 2000. URL: <http://fire.nist.gov/6510>.
- [5] D. Madrzykowski, G.P. Forney, and W.D. Walton. Simulation of the Dynamics of a Fire in a Two-Story Duplex, Iowa, December 22, 1999. Technical Report NISTIR 6854, Gaithersburg, Maryland, January 2002. URL: <http://www.fire.nist.gov/bfrlpubs/duplex/duplex.htm>.
- [6] R.L. Vettori, D. Madrzykowski, and W.D. Walton. Simulation of the Dynamics of a Fire in a One-Story Restaurant – Texas, February 14, 2000. Technical Report NISTIR 6923, Gaithersburg, Maryland, October 2002.
- [7] R.G. Rehm, W.M. Pitts, Baum H.R., Evans D.D., K. Prasad, K.B. McGrattan, and G.P. Forney. Initial Model for Fires in the World Trade Center Towers. Technical Report NISTIR 6879, Gaithersburg, Maryland, May 2002.
- [8] Mason Woo, Jackie Neider, Tom Davis, and Dave Shreiner. *OpenGL Programming Guide*. Addison-Wesley, Reading, Massachusetts, 3 edition, 1999.
- [9] Mark J. Kilgard. *OpenGL Programming for the X Window System*. Addison-Wesley Developers Press, Reading, Massachusetts, 1996.
- [10] Thomas Boutell. *CGI Programming in C & Perl*. Addison-Wesley Publishing Co., Reading, Massachusetts, 1996.
- [11] Thomas Boutell. GD version 2.0.7, <http://www.boutell.com/gd/>, November 2002.
- [12] Guy Eric Schalnat, Andreas Dilger, and Glenn Randers-Pehrson. libpng version 1.2.5, <http://www.libpng.org/pub/png/>, November 2002.
- [13] jpeg version 6b, <http://www.ijg.org/>.

- [14] Jean loup Gailly and Mark Adler. zlib version 1.1.4, <http://www.gzip.org/zlib/>, November 2002.
- [15] Paul Rademacher. GLUI version 2.1, <http://www.cs.unc.edu/rademach/glui/>.
- [16] Tomas Akenine-Moller. *Real-Time Rendering*. A K Peters, Ltd., 2002.
- [17] Pamela P. Walatka and Pieter G. Buning. PLOT3D User's Manual, version 3.5. NASA Technical Memorandum 101067, NASA, 1989.

Appendix A

Command Line Options

Smokeyview may be run from a command shell. Furthermore, command line options may be invoked in order to alter Smokeyview'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:

```
Smokeyview 4.0 - Jul 23 2004
Visualize fire/smoke flow simulations. All parameters are optional.
```

Usage:

```
smokeview casename -points m -frames n -ini -part -nopart -stereo
```

where

```
casename = project id (file names without the extension)
m = maximum number of particles. Default=5000000
n = maximum number of particle frames. Default=5001
-ini = output default smokeview parameters to smokeview.ini
-part = load particle file if present
-nopart = do not load particle file
-stereo = activate stereo mode (if supported)
```

The `-nopart` option is used to prevent a particle file from being loaded at startup. The `-points` and `-frames` options are 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. The `-benchmark` option causes Smokeview to produce precise timings by outputting frame rates based upon one cycle through the timing loop rather than using moving averages. The `-stereo` option may be used to access stereo hardware (often called quad buffering) if it is available.

Appendix 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. Smokeview's pop-up menus are implemented with GLUT[9], the graphics library utility toolkit. The user interacts with Smokeview via 1) menus, 2) keyboard shortcuts and 3) the preference file (`smokeview.ini` or `casename.ini`).

A “pop up” menu is displayed when the right mouse is clicked anywhere within the scene. The main menu options as illustrated in Figure B.1 are: `Load/Unload`, `Show/Hide`, `Options`, `Dialogs`, `Tours`, `View`, `Help` and `Quit`. Several of these menu options have sub-menus. These menus are described in the following sections.

B.1 Main Menu Items

Load/Unload This menu option allows one to load or unload particle, isosurface, vector slice, slice, Plot3D or boundary files. These time dependent files may be viewed simultaneously, but not concurrently with the time independent Plot3D files. However, one boundary or one Plot3D file per mesh 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 dialog boxes *etc*.

Dialogs This menu allows one to open/close various dialog boxes for setting data bounds, controlling “look” of the 3D smoke, specifying clip planes *etc*.

Tour This menu displays a list of tours stored in the `casename.ini` preference file. The manual tour gives control back to the user. Tours may be defined using the Tour dialog box.

View Resets the simulation scene to an alternate view. The three choices are 1) exterior view, 2) interior view or a 3) user defined view. A viewpoint may be saved by using this menu or by



Figure B.1: Main Menu

using the `Viewpoint` sub-menu of the `Options` menu. If a time file 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

Six types of files may be visualized with Smokeview. These files are loaded using the `LOAD/UNLOAD` menus as illustrated in Figure B.2. They are particle, vector slice, 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. To hide a data file, select the `Show/Hide` menu option corresponding to the file type to be hidden.

The Smokeview `.smv` 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 an open file dialog box to open a Smokeview scene (file with `.smv` extension). This menu is only active when one starts up Smokeview without first specifying a file. (One cannot open a new Smokeview file after one has already been opened.)

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

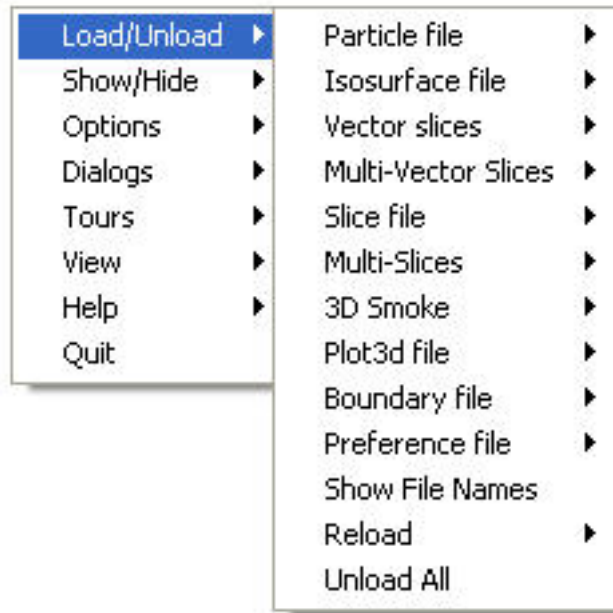


Figure B.2: Load/Unload Menu

the currently loaded particle file.

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

Multi-Vector Slice File (.sf) This menu item allows one to load all vector slices occurring in one plane (within a grid cell) simultaneously. It also gives the option to unload the currently loaded multi-slices.

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

Multi-Slice File (.sf) This menu item allows one to load all slices occurring in one plane (within a grid cell) simultaneously. It also gives the option to unload the currently loaded multi-slices.

3D Smoke File (.s3d) This menu item allows one to load realistic smoke, fire or water spray files.

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.

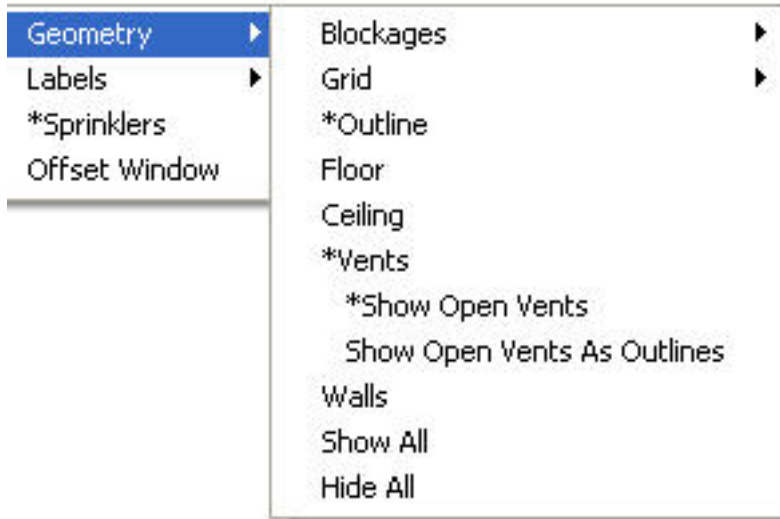


Figure B.3: Geometry Menu

Preference File (.ini) The INI or preference file contains configuration parameters that may be used to customize Smokeview’s appearance and behavior. This menu item allows one to 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.

Show File Names Load and Unload menus by default are specified using the location and type of visual to be displayed. This menu item adds file names to the Load and Unload menus.

Reload This menu item allows one to reload files at intervals of 1, 5 or 10 minutes. This is useful when using Smokeview to display a case that is currently running in FDS.

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

The `Geometry` menu is illustrated in Figure B.3.

Blockages Blockage sub-menus are divided into two groups. The first group allows the user to change how blockage appear (`Defined in Input File`, `Normal`, `Outline`, `Hidden`).

The second group allows the user to change where blockages are located (actual or requested). The `Actual` sub-menu positions blockages as computed by FDS (along grid lines). The `Requested` sub-menu positions blockages at locations as specified in the input file. If the dx2dxf conversion program was used to convert a Cad drawing to input compatible with FDS and Smokeview then a third menu option appears, `Cad`. This option displays the Smokeview scene in a form similar to the original Cad drawing.

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

This menu allows one to control the way isosurfaces are displayed. 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 (.b \bar{f}) 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.

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.

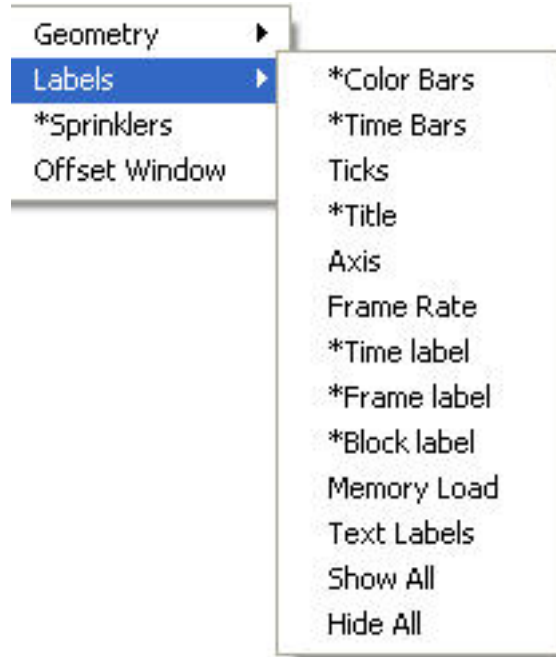


Figure B.4: Label Menu

B.3.8 Heat detectors, Sprinklers, Thermocouples

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

B.3.9 Textures

Toggle the visibility of each or all textures.

B.3.10 Labels

The label menu is illustrated in Figure B.4

Color Bars, Time Bars, Title, Axis, Frame Rate, Time Label, Frame label, Mesh label, Memory Load, Text Labels
Show or hide the individual scene element.

Show All Show all scene elements.

Hide All Hide all scene elements.

B.4 Options

The option menu is illustrated in Figure B.5 and detailed below.

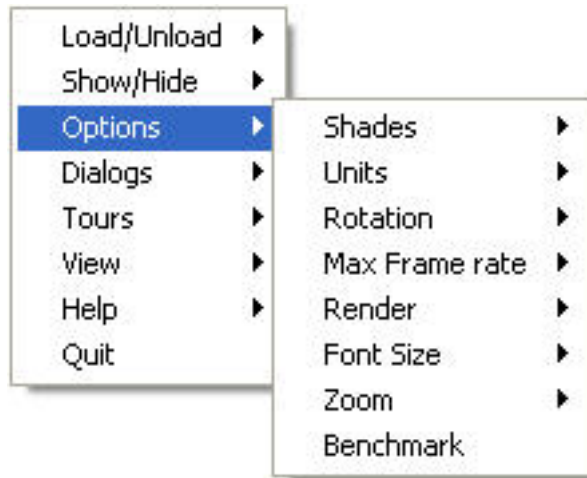


Figure B.5: Option Menu

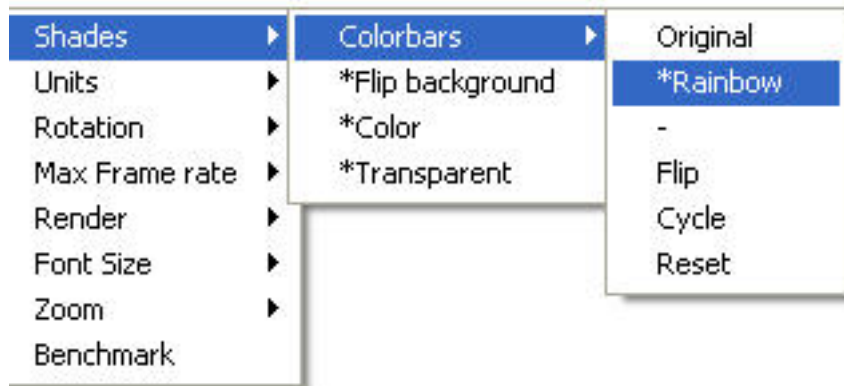


Figure B.6: Shades Menu

B.4.1 Shades

The shades menu is illustrated in Figure B.6. This menu contains a new `colorbar` sub-menu which allows the user to choose and manipulate colorbars.

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. Contours and blockages are the most important Smokeview objects to convert.

Transparent Toggle contours between opaque and transparent. Transparent colors allow one to view the scene behind the contours giving the user 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.)

World Centered - Level Rotation As expected, this is the same as `World Centered` but with level rotations.

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. 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.

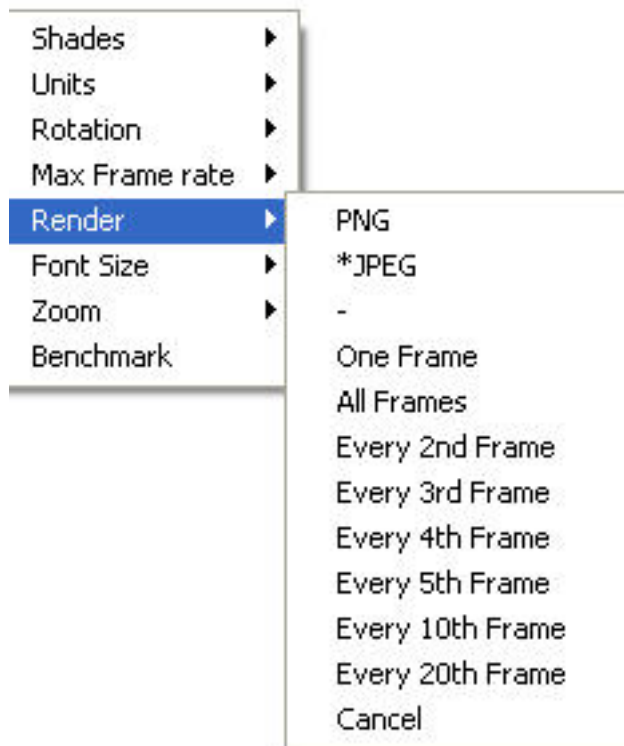


Figure B.7: Render Menu

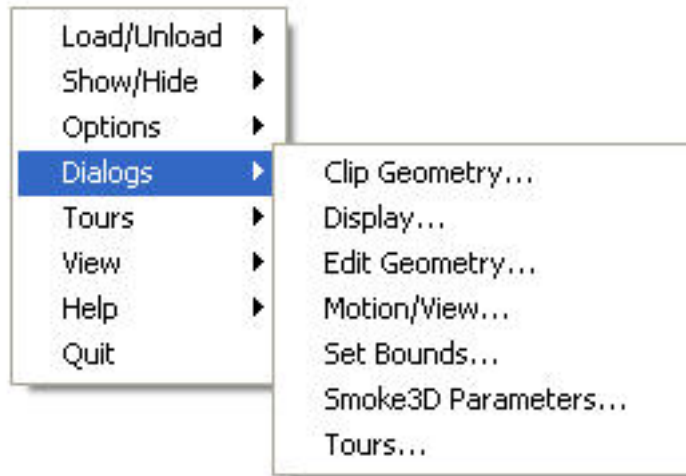


Figure B.8: Dialogs Menu

B.4.5 Render

The **Render** menu, illustrated in Figure B.7, allows one to create PNG or JPEG image files of the currently displayed scene. The graphics library, GD 2.0.15, was used to create the rendered versions. GD 1.0 is documented in reference [10, Appendix 4]. GD 2.0.7 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 2.0.15 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. Rendering may be stopped by selecting **Cancel**.

The keyboard shortcut for the render option is *r*.

B.4.6 Font Size

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

B.4.7 Zoom

This menu item allows one to change the perspective by altering the magnification used to display the scene.

B.5 Dialogs

The **Dialogs** menu, illustrated in Figure B.8, allows one to select dialog boxes for setting various Smoke-view features.

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

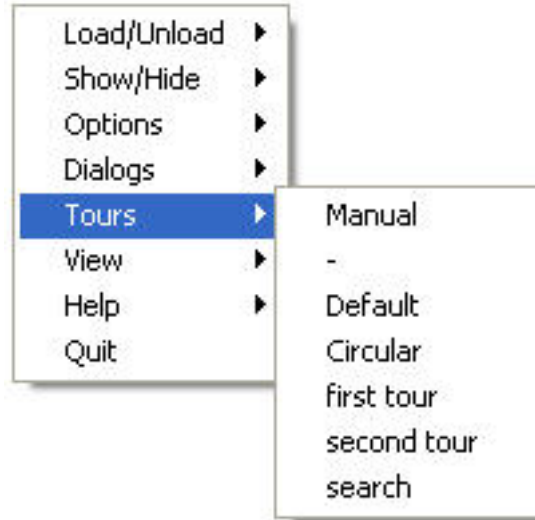


Figure B.9: Tour Menu

option only appears if Smokeview is started from the command line using the `''-stereo''` command line option and if a video card supporting quad buffering (left and right front buffer, and left and right back buffer) is supported.

Clip Geometry Open the dialog box for clipping the geometry allowing one to see past exterior portions of the scene.

Edit Geometry Open the dialog box for editing FDS blockages.

Move Scene Open the dialog box for controlling scene movement. To use the movement arrows, click and hold the mouse in one of the arrows then move the mouse to achieve the desired movement.

Properties Open the dialog box for setting various parameters that control how the Smokeview scene appears.

Set Bounds Open the dialog box for specifying data bounds.

Smoke3D Parameters Open the dialog box for specifying controlling the look of the 3D smoke.

Tours This menu item opens up the **Edit Tour** dialog box and shows the path over which the tours occur.

B.6 Tours

The Tour menu illustrated in Figure B.9 allows one to show and hide available tours.

Manual This menu item turns “touring” off, allowing the user to control the scene with the mouse.

Default This menu item activates the original “tour” used in previous versions of Smokeview.

List of tours Each tour defined in the `.ini` file is listed under the Tour menu. A “circular” tour is defined automatically by Smokeview.

Appendix 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 resulting 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.
- h** Toggle window indentation, for use with window capturing.
- i** Toggle the visibility of iso-contours (3D surface contours).
- k** Toggle the visibility of the time bar.
- m** Switch between blocks in a case that has more than one mesh.
- o** Reset a time dependent animation to the initial time.
- p** Show the next variable in the Plot3D data set.
- P** The cursor keys and the page up/down keys are needed to both move grid and Plot3D planes around the scene and to move through the scene while in “eye centered” mode. The upper case “P” key is then used to switch the way these keys are used between grid/Plot3D movement mode and scene movement mode.
- q** Switch between blockage views. These views are blocks that are aligned on grid lines, blocks as specified by the user (in the .data file) and blocks as generated by a CAD (computer aided drawing) package.
- 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. This is 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** When the *eyevew* mode is **eye centered** then these keys rotate the scene to the left or right otherwise they 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.

Appendix D

File Formats

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

Smokeview uses preference files to set input parameters not settable by 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 preference 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 preference 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
```

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: `0.6 0.6 0.6`)

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 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)

HEATOFFCOLOR Sets the color used to visualize heat detectors before they activate. (default: 1.0 0.0 0.0)

HEATONCOLOR Sets the color used to visualize heat detectors after they activate. (default: 0.0 1.0 0.0)

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)

STATICPARTCOLOR Sets the color used to visualize static particles (particles displayed in frame 0). (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 lines¹ in pixels. (default: 2.0)
- PARTPOINTSIZ** 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 distance² 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)
- VECTORPOINTSIZ** 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)
- WINDOWOFFSET** Defines a margin offset around the Smokeview scene for use when capturing images to video. (default: 45)

D.1.3 Time, Chop 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_), 2) eliminate or chop data from being displayed (beginning with C_) and 3) override 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 override 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

¹Except lines used to draw vents

²distance is relative to the maximum grid cell width

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.

C_PARTICLES Defines the minimum and maximum values used to discard particle data in a visualization. To drop particle data below 70°C and above 200°C use:

```
C_PARTICLES 1 70. 1 200.
```

C_PLOT3D Defines the minimum and maximum data values used to drop or chop Plot3D. To cause Smokeview to set the minimum and maximum chop values for the first PLOT3D quantity (usually temperature) to 100 and 300 use:

```
C_PLOT3D 5
1 1 100.0 1 300.0
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 “100” or “300” causes Smokeview to use the next number as a minimum or maximum chop value respectively.

C_SLICE Defines the minimum and maximum values used to discard slice file data in a visualization. To drop slice data below 70°C and above 200°C use:

```
C_SLICE 1 70. 1 200.
```

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

```
T_ISO
1 20. 1 150.
```

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:

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

T_SLICE Defines the minimum and maximum times used to include slice frames 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 to which type of data the bounds should apply. 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 dialog 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 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 contour³ 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 ?? resulting in a contour plot as illustrated in Figure 4.8.

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 to which type of data the bounds should apply. 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 dialog 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 (de-

³This is true for “stepped” or discrete contours. If “continuous” contours are drawn, then “2” and “1” have the same effect.

fault: 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. 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 the user 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)

SLICEDATAOUT When set to 1 will output data corresponding to any loaded slice files whenever the scene is rendered.

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.

APERTURE Specifies the viewing angle used to display a Smokeview scene. Viewing angles of 30, 45, 60, 75 and 90 degrees are displayed when **APERTURE** has the value of 0, 1, 2, 3 and 4 respectively. (default: 2)

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)

BLOCKLOCATION Specifies the location or method used to draw blockages. Blockages are drawn either snapped to the nearest grid (5), drawn at locations as specified in the FDS input file (6) or drawn as specified in a compatible CAD package (7). A program used to convert DXF formatted CAD files to files compatible with FDS and Smokeview is available at <http://fire.nist.gov/fds/refs/tools.html> (default: 5)

CLIP Specifies the near and far clipping plane locations. Dimensions are relative to the longest side of an FDS scene. (default: 0.001 3.000)

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

FONTSIZE Specifies whether small (0) or large (1) fonts should be used to display text labels. (default: 0)

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 (essentially unlimited))

MSCALE Specifies how dimensions along the X, Y and/or Z axes should be scaled. (default: 1.0 1.0 1.0)

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 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)

SHOWAXISLABELS Specifies whether axis labels should be drawn (1) or not drawn (0) drawn. (default: 0)

SHOWBLOCKLABEL Specifies whether a label identifying the active mesh should be drawn (1) or not drawn (0). (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)

SHOWFRAME Specifies whether the frame surrounding the scene should be drawn (1) or not drawn (0). (default: 1)

SHOWFRAMELABEL Specifies whether the frame number 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)

SHOWLABELS Specifies whether labels should be drawn (1) or not drawn (0). Labels are specified using the **LABEL** keyword described in subsection 4.10. (default: 0)

SHOWMEMLOAD Specifies (when run on a PC) whether a label giving the memory used should be drawn (1) or not drawn (0). (default: 0)

SHOWNORMALWHENSMOOTH Specifies that smooth blocks should be drawn as normal blocks (1) or drawn as smooth blocks (0). (default: 0)

SHOWOPENVENTS Specifies that open vents should be drawn (1) or not drawn (0). (default: 0)

SHOWDUMMYVENTS Specifies that dummy vents (vents created by FDS) 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)

SHOWTICKS Specifies whether labels should be drawn (1) or not drawn (0). Ticks are specified using the **TICK** keyword described in subsection 4.10. (default: 0)

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

SHOWTIMELABEL Specifies whether the time label should be drawn (1) or not drawn (0). (default: 1)

SHOWHMSTIMELABEL Specifies whether the time label should be drawn (1) or not drawn (0) using the format “h:m:s” where “h” is hours, “m” is minutes and “s” is seconds. (default: 0)

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

SHOWVENTS Specifies whether vents 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)

STARTUPVIEW Used by Smokeview to record the view to be displayed at startup (as defined in a previous Smokeview run).

SURFINC Smokeview allows one to display two Plot3D isosurfaces simultaneously. The SURFINC parameter specifies the interval between displayed Plot3D surface values. (default: 0)

TIMEOFFSET Specifies that offset time in seconds added to the displayed simulation time. Along with the **SHOWHMSTIMELABEL** keyword, the **TIMEOFFSET** keyword allows one to display “wall clock” rather than simulation time. (default: 0.0)

TITLESAFE Amount in pixels to offset titles when displaying scene in “title safe” mode. (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
```

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)

XYZCLIP Specifies clip planes in physical coordinates. There are six clipping planes, a minimum and maximum X, a minimum and maximum Y, a minimum and maximum Z. Each clipping plane may be used or not. The first parameter is 1 or 0 and specifies whether clipping is turned on or off. The next three lines specify clipping parameters for the X, Y and Z axes. Each line has the format

```
minflag min-clipval maxflag max-clipval
```

where the two flags, minflag and maxflag are 1 if turned on or 0 if turned off. Clipping is specified with the clipping dialog box found under the Options menu item. (default:

```
0
0 0.0 0 0.0
0 0.0 0 0.0
0 0.0 0 0.0
```

ZOOM Specifies the zoom amount used to display a Smokeview scene using two parameters, an integer zoom index and a floating point zoom amount. If the zoom index is 0-4 then the zoom amount is 0.25, 0.5, 1.0, 2.0 and 4.0 respectively. If the zoom index is negative then the second parameter is used to specify the zoom amount. (default: 0 1.0)

D.1.6 Tour Parameters

GLOBALTENSION Specifies whether the tour spline tension parameter found in the dialog box should (1) or should not (0) be used for ALL tour keyframes. (default: 1)

SHOWPATHNODES Specifies whether the path nodes should (1) or should not (0) be drawn. This is a debugging parameter, not normally used. (default: 0)

SHOWTOURROUTE Specifies whether the tour route should (1) or should not (0) be drawn. (default: 0)

TOURS Keyword used to specify the tours. The format is

```
TOURS
ntours - number of tours
label for tour
nkeyframes - number of keyframes for first tour
time xpos ypos zpos 1 az elev bias continuity tension zoom localspeedflag
...
time xpos ... for last keyframe
nkeyframes for 2nd tour
...
...
```

If a cartesian view direction is specified then instead of 1 az elev above use 0 xview yview zview where xview, yview, zview are the coordinates of the view direction. The “Circle” tour is not stored in the .ini file unless it has been changed by the user. The tour entry created by using the **Add** button in the **Tour** dialog box is given by

```
TOURS
1
```

```

Added Tour 1
2
0.0 -1.0 -1.0 -1.0 0 0.0 0.0 0.0 0.0 0.0 1.0 0
100.0 7.4 9.0 7.4 0 0.0 0.0 0.0 0.0 0.0 1.0 0

```

TOURCOLORS Keyword used to specify the tour colors. The colors as before consist of a red, green and blue component ranging from 0.0 to 1.0 . One can override Smokeview's choice for the path, the path knots for both the selected and un-selected case. One may also specify the color of the time labels and the location of the object or avatar on the tour at the current time. The foreground color is used when a color component less than 0.0 is specified. Default:

```

TOURCOLORS
1.000000 0.000000 0.000000 :selected path line
1.000000 0.000000 0.000000 :selected path line knots
0.000000 1.000000 0.000000 :selected knot
-1.000000 -1.000000 -1.000000 :path line
-1.000000 -1.000000 -1.000000 :path knots
-1.000000 -1.000000 -1.000000 :text
1.000000 0.000000 0.000000 :avatar

```

TOURCONSTANTVEL Specifies whether the avatar should (1) or should not (0) traverse the path with a constant velocity. (default: 1)

VIEWALLTOURS Specifies whether all (1) tours should be drawn. (default: 0)

VIEWTOURFROM Specifies whether the scene should (1) or should not (0) be observed from the selected tour.

VIEWTIMES Specifies the tour start time, tour stop time and number of points to specify a tour. (default: 0.0 100.0 1000)

D.1.7 Realistic Smoke Parameters

ADJUSTALPHA The ALPHA parameter in OpenGL/Smokeview is used to specify the transparency of an object. The ALPHA's are computed in FDS and adjusted in SMOKEVIEW according to value of the ADJUSTALPHA keyword. If ADJUSTALPHA is zero, then no adjustments to ALPHA are made. If ADJUSTALPHA is one, then ALPHA's are adjusted for non-orthogonal view directions (ALPHA is increased to account for longer path lengths along non-orthogonal view directions. If ADJUSTALPHA is two, then ALPHA's are adjusted as in the ADJUSTALPHA=1 case and are also set to zero on wall at blockage boundaries (this reduces graphical artifacts). (default: 1)

FIRECOLOR Specifies the color of the fire in red, green, blue coordinates. Each color component is an integer ranging from 0 to 255. (default: 255, 128, 0)

FIREDEPTH Specifies the depth at which the fire is 50 percent transparent. (default: 2.0)

SMOKECULL Cull (or do not draw) smoke if it is outside of the viewing frustum. (default: 1)

SMOKESHADE Grey level of smoke, may range from 0 to 255. (default: 255)

SMOKESKIP To speed up smoke drawing, spatial frames may be skipped. Allowable parameters are (0, 1, 2). (default: 0)

SMOKETHICK The ALPHA transparency parameter may be divided by a power of two ($2^{\text{SMOKETHICK}}$) to make the smoke thinner. Parameters may range from 0 to 7. (default: 0)

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 text file consisting of a set of KEYWORDS followed by DATA describing the FDS case's geometry, data file names 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,

```
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.

SHOW_OBST(HIDE_OBST) This keyword specifies when a blockage should be shown(hidden). For example,

```
SHOW_OBST 2
10 120.1
```

specifies that the tenth blockage in mesh 2 should be opened at 120.1 seconds. This keyword is automatically added to the .smv file by FDS.

OBST This 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*, *jb2*, *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,*)idummy
end do
do i = 1, nx
  read(5,*)xp1t(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*, *jv2*, *kv1*, *kv2* are integer arrays containing vent coordinates.

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

```

OPEN_VENT 2
3 15.6

```

specifies that the third vent in mesh 2 should be opened at 15.6 S.

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.
- 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.
- PART** The `PART` keyword defines the `.part` file name along with character labels used to describe the data contents of the particle file.
- PL3D** The `PL3D` keyword defines the `.q` file name along with character labels used to describe the data contents for each Plot3D variable.
- SLCF** The `SLCF` keyword defines the `.sf` file name along with character labels used to describe the data contents of the slice file.

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 endian⁴ 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
```

⁴Big-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.

D.3 Data File Formats (.s3d, .iso, .part, .sf, .bf and .q files)

Smokeview visualizes Fire Dynamics Simulator 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 3D Smoke File Format

D.3.2 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. Note that char's are one byte, short's are two bytes and "int's" and float's are four bytes. 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)
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*(byte/short/float)*ntriangles)
  end level
end time
```

The length of each triangles_i node is one byte if the number of triangles, ntriangles, is between zero and 255 (inclusive), two bytes if ntriangles is between 256 and 65536 (inclusive) and four bytes if ntriangles is greater than or equal to 65536. Note that the isosurface files are written using the C programming language. These files should be read the same way unless the programming language of choice supports binary I/O.

D.3.3 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.3.4 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 a 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.3.5 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.3.6 Plot3D Data

Field data is dumped periodically by FDS in a format used by the graphics package **Plot3D**[17]. 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).

Appendix E

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> .

E.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.

E.2 Can I run Smokeview on other machines?

Smokeview is more complicated to compile on other platforms because it uses several freeware libraries that need to be compiled and installed on that particular machine. We have compiled Smokeview on the PC, SGI (IRIX), and Red Hat Linux, but with the exception of the PC executable, there's no guarantee that something we compile on one of our machines is going to work on yours. Worse comes to worse, use a PC to visualize the output of a calculation run on your other machine.

E.3 Smokeview doesn't look right on my computer. What's wrong?

Some video cards do not use enough 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.

E.4 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 of 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>

E.5 Smokeview is running much slower than I expected. What can I do?

A likely explanation is that your video drivers are not hardware accelerated for OpenGL. You need to download the latest drivers for your video card. You may also try playing around with the hardware acceleration settings (see E.3).

E.6 I loaded a particle file but Smokeview is not displaying particles. What is wrong?

This is a bug in your video card. To see particles, reduce your hardware acceleration setting (see E.3).