



403 Poyntz Avenue, Suite B
Manhattan, KS 66502
USA
+1.785.770.8511
www.thunderheadeng.com

Results User Manual

PyroSim 2019

Disclaimer

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

This software is intended only to supplement the informed judgment of the qualified user. The software package is a computer model that may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions by the model could lead to erroneous conclusions. All results should be evaluated by an informed user.

Trademarks

All other product or company names that are mentioned in this publication are tradenames, trademarks, or registered trademarks of their respective owners.

Table of Contents

1. About this Manual	1
Additional Support Resources	1
Please Give Us Feedback	1
2. Installing and Licensing	2
System Requirements	2
Results Licensing	3
3. Configuration and Files	5
Properties File	5
Persistent Session Data	5
4. Using Results	6
Loading results	6
Results Timelines	7
Example	8
Navigating through a model	9
Scene Camera	9
Views	9
Displaying geometry input	9
Viewing multi-floor problems	11
Animation playback	14
Refreshing Results	15
File Streaming	15
Controlling drawing detail/speed	15
Creating Screenshots	18
Single	19
Multiple	19
From File	21
Creating Movies	22
High Quality Movies	22
Real-time Movies	23
5. Views, Tours, and Clipping	25
Viewpoints	25
Creating a Viewpoint	25
Recalling a Viewpoint	26
Editing a Viewpoint	26
Camera Tours	26
Tour Editor Dialog	27
Tour Properties	28
Creating a Tour	28
Editing a Camera Tour	29
Section Boxes	31
Creating a section box	32
Editing a section box	33
Remove a section box	33
Duplicating Views	34
Views Files	34

Importing Views	34
6. Annotations	35
Dimensions	35
Custom Labels	37
7. Viewing FDS Results	40
Results Organization	41
Activating Results	41
Colorbars	43
Highlight region	44
Global FDS Properties	44
3D Smoke and Fire	47
3D Slices	51
Plot3d	52
Data3d	52
Data3d Volumetric Rendering	52
Data3d Points	54
Data3d Slices	55
Data3d Isosurfaces	56
Isosurfaces	58
2D Slices	58
2D Slice Vectors	59
Boundaries	61
Realistic Visualization of Soot Deposition	62
Particles	63
8. Viewing Pathfinder Results	66
Displaying occupants	66
Selecting Occupants	67
Viewing occupant paths	67
Occupant Contours/Heat Maps	68
Adding/Removing Contours	69
Adding Contour Filters	70
Importing Contours	70
Duplicating Contours	70
Global Contour Properties	71
Individual Contour Properties	72
Activating/Deactivating Contours	74
Optimizing Contours	75
9. Technical Reference	77
Occupant Contours	77
Density	78
Level of Service	79
Speed	79
Normalized Speed	79
Usage [Instantaneous]	79
Usage [Accumulated]	79
Time to Exit	79
Average	80
Maximum	80

Integrate	80
10. Troubleshooting	81
Floating License Server Problems	81
Video Display Problems.....	81
Results crashes on startup	81
Results application crashes when results are loaded.....	82
Results don't display correctly	82
Stuttering FDS Results	82
Unresponsive Results	82
Known Issues.....	82
Contacting Technical Support	83
11. References	84

1. About this Manual

This guide tells you how to use the results visualization software that accompanies PyroSim and Pathfinder. It assumes a basic understanding of the work flow when using these platforms to model fire and movement. For introductory guides to PyroSim and Pathfinder, refer to the support materials for each product online at <http://www.thunderheadeng.com>

Additional Support Resources

This guide is the primary source of documentation for the Results visualization software and describes how to perform common tasks. For additional information describing how different parts of the software work, refer to the Technical Reference on page 77.

For documentation on other Thunderhead Engineering applications, refer to the product-specific support materials online at <http://www.thunderheadeng.com>

Please Give Us Feedback

If you need assistance, or wish to provide feedback about any of our products, contact support@thunderheadeng.com

2. Installing and Licensing

The results visualization software will be installed automatically with PyroSim and Pathfinder. There is no need to download or install a separate application. For more information about installing PyroSim or Pathfinder, refer to the support materials for each product online at <http://www.thunderheadeng.com>

System Requirements

The following table lists system requirements necessary to run results visualization.

Operating System	<ul style="list-style-type: none">• Microsoft Windows 7 or newer• Microsoft Windows 7 N or newer (with Media Feature Pack)• Microsoft Windows Server 2008 R2 or newer (with Desktop Experience) <p>Desktop Experience and Media Feature Pack are required due to a dependency on Windows Media Player (wmvcore.dll).</p>
Memory	<ul style="list-style-type: none">• 4 GB (required)• 8 GB (recommended)
Display Adapter	<ul style="list-style-type: none">• Support for OpenGL 1.2 (required)• Support for OpenGL 3.2 (recommended)

Results visualization performance (measured in frames per second, or FPS) is controlled by the number of occupants in the model, the complexity of the geometry, and the resolutions of the FDS solution meshes. In one Pathfinder experiment, a single-room model containing 3,000 occupants and 4 geometry triangles operated at 15 FPS. By comparison, a complex model with 50,000 occupants and 1,300,000 geometry triangles operated at 5 FPS.

Performance can be improved in several ways:

- Upgrade the display adapter (GPU). The Results application has been optimized for use with gaming-level GPUs. There is little to no benefit to using a workstation GPU over a good gaming-level GPU such as an NVIDIA GeForce GTX 1050 or equivalent.
- Upgrade the processor (CPU). The Results does a fair amount of processing on the CPU to generate contours, interpolate data, etc. It also takes advantage of multi-core and multi-CPU systems.
- For FDS, a fast hard drive, such as an SSD, is also recommended, especially for viewing results of multi-million cell models. This is especially important for file streaming as discussed in File Streaming on page 15.

Results Licensing

Results visualization requires a PyroSim or Pathfinder license. Additional information on license activation is located on the corresponding software download page.

To locate a license, a search is performed in standard locations for a valid license. In addition, license locations listed in the PROPS files for PyroSim and Pathfinder are searched. The search order is shown below:

Location listed in Results.props or -rlmpath argument
Install folder where the results executable resides

The following additional locations will be searched by default. To prevent default search locations, see the -rlmstrict program argument.

Current Working Folder
%PROGRAMDATA%\Application Data\Results\license
Location listed in %APPDATA%\PyroSim\PyroSim.props
%PROGRAMDATA%\PyroSim\license
Location listed in %APPDATA%\Pathfinder\Pathfinder.props
%PROGRAMDATA%\Application Data\Pathfinder\license

Results will operate if any valid license is found. It is recommended to run PyroSim or Pathfinder before running the results software to ensure a valid license location is present in the corresponding PROPS file.

For advanced use and license troubleshooting, two command-line options are available:

-RLMPATH <path>	<p>Specify a custom RLM license location. This location will be prepended to the list of license search locations.</p> <p>To add a local license folder to the search path:</p> <pre>PyroSimResults.exe -rlmpath "C:\path to\licenses"</pre> <p>To add a floating license server to the search path (port@computer):</p> <pre>PyroSimResults.exe -rlmpath "52100@servername"</pre> <p>Results will save the value of <code>-rlmpath</code> to the PROPS file entry "RlmLicense.Path" for later use. To remove, it must be manually deleted from the PROPS file.</p>
-RLMSTRICT	<p>Prevent results from appending default search locations to the RLM license search path. When this option is set, only the custom path and the executable folder will be searched for licenses.</p>

	<p>To use only the executable folder and (optionally) a license path stored in the PROPS file entry "RlmLicense.Path":</p> <pre>PyroSimResults.exe -rlmstrict</pre> <p>To use this option in combination with <code>-rlmpath</code>:</p> <pre>PyroSimResults.exe -rlmstrict -rlmpath "C:\license"</pre>
--	---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------

The examples in the table above assume a PyroSim naming convention for the results EXE (i.e. `PyroSimResults.exe`). These options are present in all versions of the results executable.

3. Configuration and Files

Properties File

The Results application stores data related to user preferences in a file called Results.props. By default, this file can be found in one of:

%APPDATA%\Pathfinder\Results.props

%APPDATA%\PyroSim\Results.props

The PROPS file is stored in a plaintext format, and can be viewed or edited with any conventional text editor. While it is not recommended to edit the file directly, some troubleshooting techniques may involve deleting the PROPS file so that a new one can be created from scratch by the Results application.

Persistent Session Data

Some session data is stored in a file ending with the extension PFRMETA. This data allows the Results application to remember settings specific to a single set of Pathfinder results. The PFRMETA file is stored in the same location as the PFR file that it is associated with. The PFRMETA file may be safely deleted to reset the session state for the results the next time they are opened.

4. Using Results

The Results application can be used to visualize results from PyroSim and Pathfinder. It operates much like a video player in that it allows users to play, pause, stop, slide, and speed up and down time. It is completely 3D and allows users to navigate through a model.

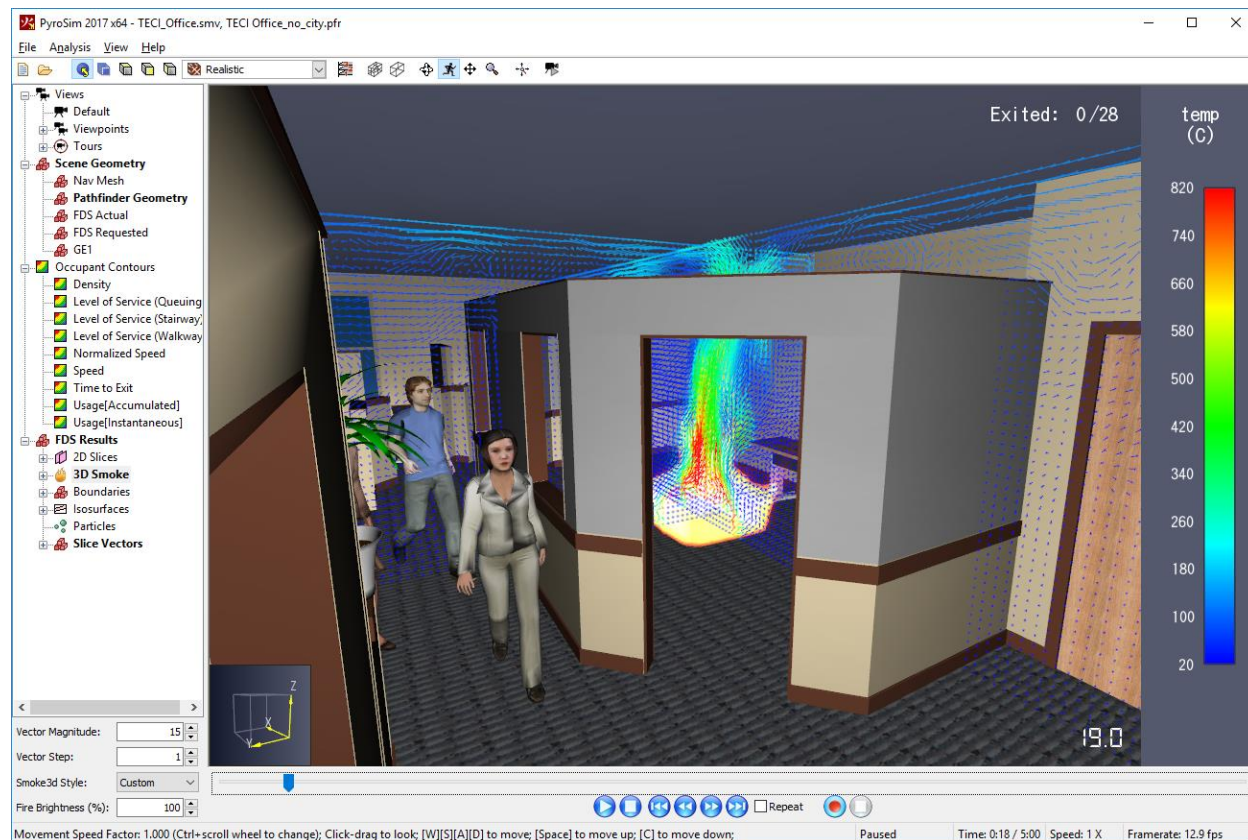


Figure 4-1: 3D Results of the Thunderhead office

In the center of the image is the **3D View**, which is the graphical representation of the scene. At the bottom of the **3D View** are the time controls for adjusting playback of the results. At the far right is a color bar that provides a legend for the colors of the temperature vectors being displayed. At the far left is the **Navigation View**, which lists objects in the model that can be activated and deactivated. At the bottom left are the **Quick Settings**, which allow fine-tuning control over the currently active results. At the top is the main toolbar for the application.

NOTE: The navigation view can be hidden by going to the **View** menu and de-selecting **Show Navigation View**.

Loading results

The Results application may show exactly one set of results from Pathfinder and one set from PyroSim/FDS at the same time. If results are opened that are of the same type as those already loaded,

the current results of that type will be replaced with those from the new file. So for instance, if Pathfinder results A are opened and then PyroSim results B, both A and B will be shown. If Pathfinder results C are then loaded, only B and C will be shown.

To clear all of the results, either click the **Create new results** button (📄) on the toolbar or in the **File** menu click **New Results**.

When the Results application is started from Pathfinder or PyroSim it will automatically show the results of the open model in Pathfinder or PyroSim, respectively. At any time, a different results set can be loaded by clicking the **Open results** button (📁) on the toolbar or by going to the **File** menu and clicking **Open Results**. Choose a PFR file to open Pathfinder results or choose an SMV file to open FDS results.

Results Timelines

Each result set (Pathfinder or FDS) has its own timeline. The **Timelines** dialog, as shown in Figure 4-2, can be used to control the timelines, set a reference clock time, and set the time range displayed in the results as controlled by the time slider.

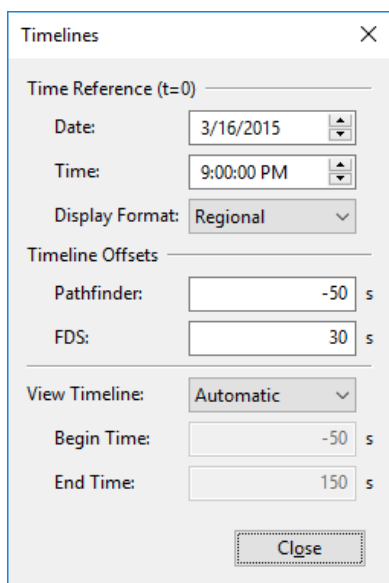


Figure 4-2 Timelines dialog

To open the timelines dialog, perform one of the following:

- In the **Analysis** menu, select **Time Settings**.
- Click the **Timelines** button (🕒) in the main toolbar.

There may be up to three timelines in the results. There is always a **Global timeline**, and depending on which results are loaded, there may also be a **Pathfinder timeline** and an **FDS timeline**. The Global timeline has a **Time Reference** that is used to synchronize the Pathfinder and FDS results. The time reference has an associated calendar **Date** and a clock **Time**, such as June 3, 2018 at 3:47 PM. This value

defaults to the date/time at which either the Pathfinder or FDS results was first created. It may be changed, however, to correspond with a real date/time either for event recreation or scenario planning.

The date/time may optionally be shown in the 3D View by going to the **View** menu and under the **Time Display** sub-menu, selecting either the **Date/Time** or **Time** option. The display of the date/time will automatically adjust depending on the current playback time. The **Display Format** defines how the date/time will be displayed. The following formats are available:

- **Regional** - displays the date/time in a format that depends on the regional settings of the operating system. For example, in the US, it might display as 7/3/2018 3:47:00 PM. In Germany, however, it might display as 03.07.2018 15:47:00.
- **ISO 8601** – displays the date/time using the ISO 8601 international standard. In this format, the date/time will appear in all regions as 2018-07-03 15:47:00.

NOTE: In either format, the current time zone is ignored.

The **Timeline Offsets** are used to change the beginning time of the FDS and Pathfinder results relative to the global time reference in order to synchronize the timelines properly. This is necessitated by the fact that Pathfinder and FDS simulations are often run as though the beginning time is at some arbitrary value, usually $t=0$. When loading both Pathfinder and PyroSim/FDS results together at the same time, however, the result sets might need to be offset from one another.

The **View Timeline** specifies the global time range displayed in the results. It can be one of these values:

- **Automatic** (default): The time range is automatically calculated based on the active tour and currently active results. If a view with a tour is active (see Camera Tours on page 26), the time range is set to the tour's time range. Otherwise, the time range is the union of all time ranges for currently active results (see Activating Results on page 41 for more information).
- **Custom**: The timeline can be entered manually using the **Begin Time** and **End Time**.

Example

To demonstrate the timeline parameters, suppose a hypothetical fire and evacuation scenario is recreated in PyroSim and Pathfinder. It is known that the fire alarms tripped on March 23, 2003 at 3:00 PM. It is believed that the fire started 2 minutes earlier. It is also known that the occupants started evacuating 3 minutes after the alarms tripped.

The fire is simulated such that the fire starts immediately at $t=0$. The Pathfinder simulation is also simulated with the evacuation taking place at $t=0$.

If both these result sets were to be loaded with no changes to the timelines, the occupants would leave as soon as the fire started. In order to synchronize the results properly, the date, March 23, 2003 at 3:00 PM is entered for the reference time. The FDS time offset is set to -120 s to represent that the fire started 2 minutes earlier. The Pathfinder time offset is set to 180 s to represent the 3 minutes of pre-movement time. Now when played back, the results will be synchronized properly and show the correct date/time in the 3D View.



Navigating through a model

Once a model is loaded, the navigation tools work similarly to those found in PyroSim and Pathfinder. The user can rotate the model (⤴), zoom in and out (🔍), and roam (🚶).

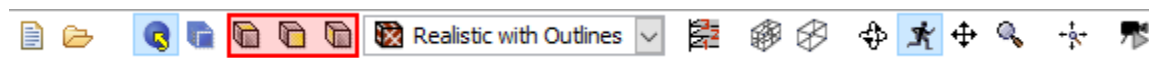
At any time, press Ctrl+R to reset the camera view or in the **View** menu click **Reset Camera**.

Scene Camera

There are two main cameras that can be used to view the model. When switching between cameras, the Results application tries to maintain the same viewpoint if possible.

-  **Perspective View:** A camera that views the scene using a perspective projection. This tries to mimic the way a real camera would view the model. Perspective views can be used to view the model from any viewpoint including inside and outside buildings. All of the navigation tools can be used with the perspective view.
-  **Orthogonal View:** This camera uses an orthographic projection to view the model. Orthographic projections make edges that are parallel in the world coordinate system also appear parallel on the screen. The orthogonal view is limited to viewing the model from the outside. The roam tool cannot be used with the orthogonal view.

Either camera can be reset to several pre-defined viewpoints by selecting the appropriate toolbar button. These viewpoints include the **Top**, **Front**, and left **Side**.



Views

Views can be used to store and retrieve various view state in the Results and to create animated tours of the model. For more information on using and creating views and tours, see Views, Tours on page 25.

Displaying geometry input

There are several geometry sets that may be available with Pathfinder and PyroSim results. When results are loaded, the available geometry sets are shown under the **Scene Geometry** group of the **Navigation View**.

By default, with the exception of the Pathfinder **Navigation Mesh**, only one geometry set may be shown at a time. This can be changed in the **Preferences** dialog, by unchecking **Scene Geometry->Only show one at a time**. To go to the Preferences dialog, in the **File** menu select **Preferences**.

To switch between the geometry sets, either double-click the desired geometry set in the Navigation View or right-click it and select **Show**. The labels for geometry sets that are active become **bold** in the Navigation View.

The following geometry sets may be available for Pathfinder results:

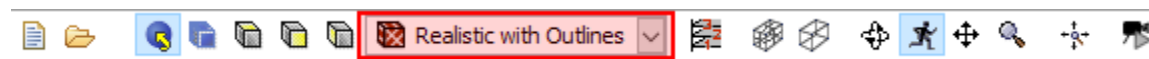
- **Pathfinder Geometry:** This includes CAD data that was imported into Pathfinder, such as from DWG or PyroSim files. It also includes any background images imported into Pathfinder.
- **Nav Mesh:** This represents the walkable space for Pathfinder occupants and may be shown with any other geometry set. It may either be shown on top of other geometry or underneath to fill in missing pieces of geometry, such as an elevator floor. To change this preference, either click the **View** menu and select **Show Nav Mesh on Top** or in the **Navigation View** right-click the geometry set that should be on top and select **Show on Top of...**

The following geometry sets may be available for PyroSim results:






- **FDS Actual:** The FDS obstructions, vents, and mesh boundaries that FDS used in the simulation. This geometry will be exactly aligned with the solution meshes.
- **FDS Requested:** The FDS obstructions, vents, and mesh boundaries that were requested in the FDS input file. These may not line up with the solution meshes.
- **PyroSim Geometry:** This includes CAD data that was imported into PyroSim. This format is richer than the GE1 format in that it can support arbitrary texture coordinates and it may better represent the actual CAD data.
- **GE1:** A file format used by Smokeview to represent arbitrary CAD data.

The default geometry set shown in the results depends on which results type (Pathfinder or FDS) was loaded first and which geometry sets are available. For the results type that is loaded first, the geometry is chosen by selecting the first available set as listed above for each type.

With the exception of the Nav Mesh, geometry data can be shown in several different modes. To change the mode, click the mode drop-down box in the main toolbar.



The following modes are available:

-  **Wireframe Rendering:** Displays only the outlines of geometric objects in the scene.
-  **Solid Rendering:** Shows objects as solid with no materials/textures applied.
-  **Solid with Outlines:** Shows objects as solid with no materials/textures applied but with outlines turned on.
-  **Realistic:** Shows objects with their materials/textures.
-  **Realistic with Outlines:** Shows objects with their materials/textures and outlines turned on.

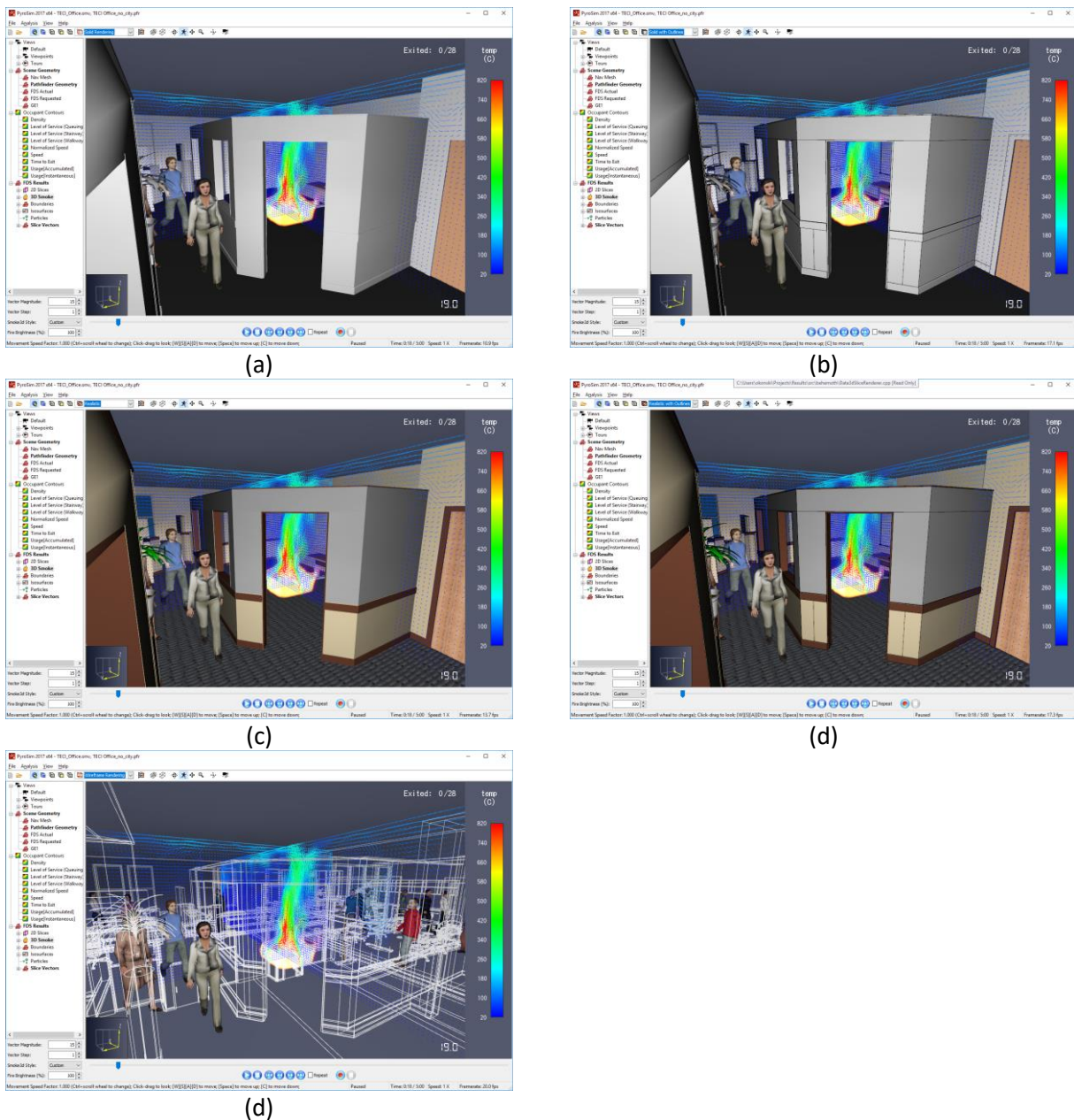


Figure 4-3: The rendering modes for scene geometry (a) Solid (b) Solid with outlines (c) Realistic (d) Realistic with outlines (e) Wireframe

Viewing multi-floor problems

Some models are composed of several floors, causing each floor to obscure the one below it as shown in Figure 4-4 (a). The Results application provides a variety of options for viewing the model such that building contents can be easily observed. One option is to view the model with the floors stacked vertically but separated by a settable distance. Another option is to view the model with the floors lying in the XY plane so that they can all be viewed from overhead. Both options allow a wall clip-height to be specified and for certain floors to be selectively shown or hidden.

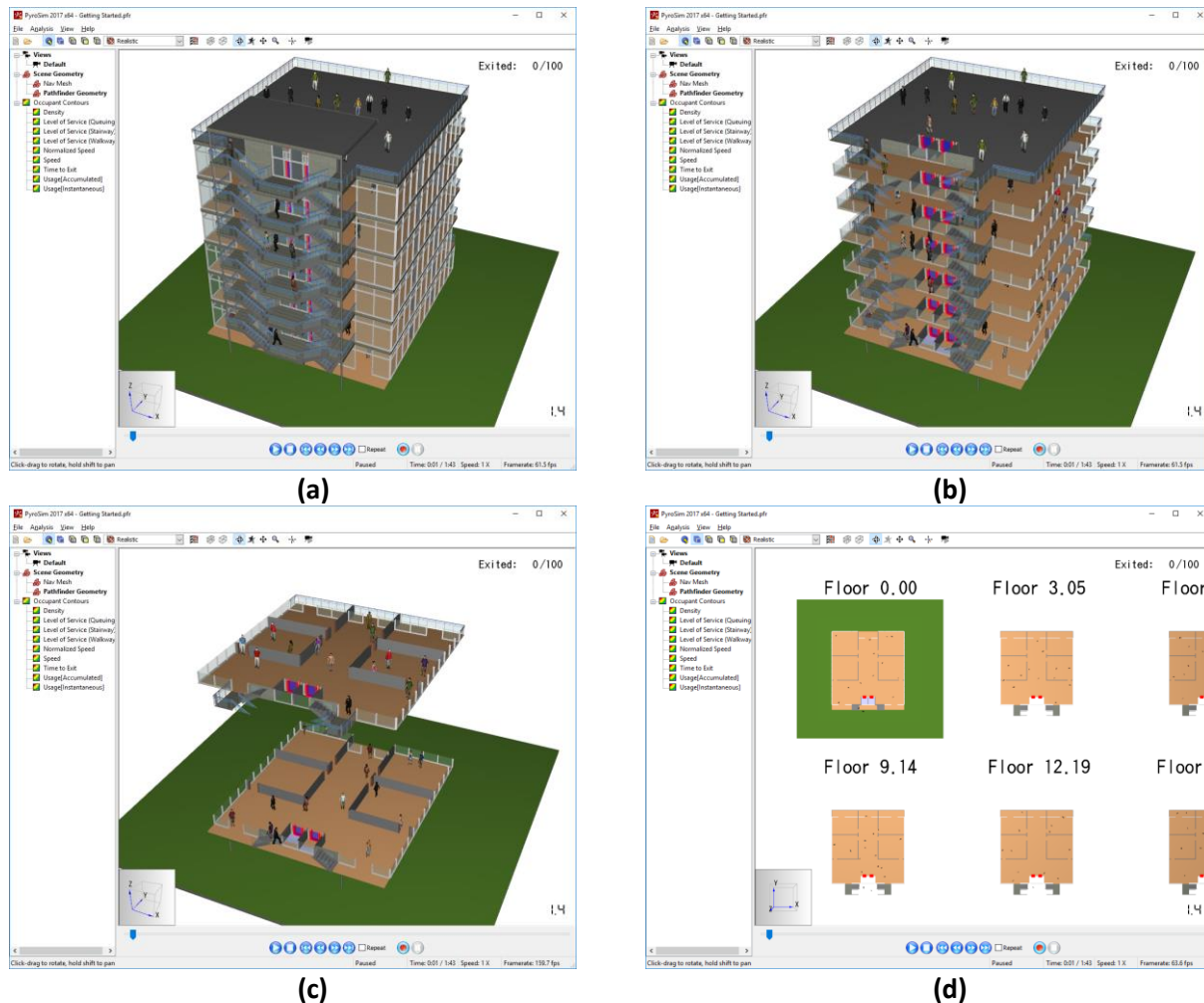



Figure 4-4: Multi-floor layout options using the Pathfinder Getting Started model

To use any of the layout options, floors must first be defined. If the floors were defined in PyroSim or Pathfinder, they will carry through to the 3D visualization and no additional input is needed. If they have not been defined or they need to be modified, this can be done as follows:

1. Press the **Edit Floor Settings** button () on the toolbar or in the **View** menu, select **Floor Settings**. This will open the **Floor Settings** dialog as shown in Figure 4-5.
2. Next to **Floors** click the **Edit** button. A dialog will appear that shows a list of the Z locations of the floors.
3. In this list, add, remove, and edit floor locations.
4. Click **OK**.

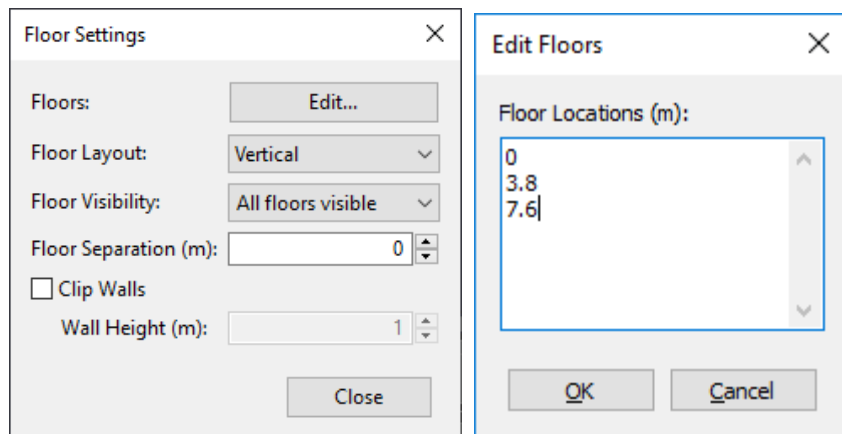


Figure 4-5: Floor Settings and Edit Floors dialogs

The Results application will partition the geometry among the floors such that the bottom floor contains all geometry from $-\infty$ to the next floor's location, the top floor contains all geometry from its location to $+\infty$, and every other floor contains geometry from its location to the next floor's location.

Once floors are defined, there are several ways to lay out the floors and clip the walls, which can all be done from the **Floor Settings** dialog.

The **Floor Layout** controls how the floors are laid out in the view. The following options are available:

- **Vertical:** Stacks the floors vertically along the Z axis. When viewed in this manner, floors will appear naturally such that each floor is stacked on top of the one below it. The floors can then be separated by empty space to more easily view inside the model as shown in Figure 4-4 (c). The **Floor Separation** controls the amount of space between floors.
- **Horizontal:** Lays out the floors in a square grid and in one plane. They will appear in order from left to right and top to bottom such that the lowest floor is in the upper left corner and the highest floor is in the lower right corner. When laid out in this manner, the **Floor Separation** will affect the gap between floors in both the X and Y directions. An example of this type of layout is shown in Figure 4-4 (d). By default, labels appear above each floor so the floors can be easily identified, but these labels can be hidden by opening the **View** menu and deselecting **Show Labels**.

Wall clipping can be applied to shorten the walls of each floor yet maintain a sense of obstruction placement as shown in Figure 4-4 (b). As shown in this figure, more of the model can be seen than in Figure 4-4 (a). To enable wall clipping, first check **Clip Walls** in the **Floor Settings** dialog. Then set **Wall Height** to the desired value. The geometry on every floor will be clipped above the floor's Z location plus the wall height.

Floor Visibility provides a way to show or hide floors. By default, all floors are visible. Visible floors will appear either in Vertical or Horizontal layout, based on the choice in Floor Layout control. One or multiple floors can then be observed easily inside the model. Similarly, the **Floor Separation** controls the

amount of space between visible floors. To show a single floor, click the down arrow and then click the desired floor. To show multiple floors, hold **Ctrl** on the keyboard while selecting the desired floors. To show all floors, click **Select All**.

In addition to showing and hiding floors in the **Edit Floor Settings** dialog, floors can also be isolated based on a selection. To do so, select the desired objects whose floors you want to isolate. Then right click and choose **Show Floors for Selection**. This will only show the floors containing the selected objects. The floors are chosen as follows:

- A single floor will be displayed if the Z bounds of the selected objects is contained entirely within one floor.
- Several related floors will be displayed if the Z bounds of the selected objects span multiple floors.
- If the selection contains objects spanning multiple, non-consecutive floors, only the floors containing the objects are shown and not those in between.

To easily show all the floors again, either right-click on empty space in the scene view and choose **Show All Floors** or in the **View** menu, choose **Show All Floors**. The floors can also be shown again using the Floor Settings dialog.

Animation playback








To begin animation playback, press the play button  at the bottom of the 3D View. As the animation is playing, the time slider will move, and the animation can be paused  or stopped . The buttons  and  will move the time slider to the beginning and end, respectively. The buttons  and  will slow down and speed up the animation by factors of two. In addition, the playback can be looped by checking the **Repeat** button.



Figure 4-6: Playback controls

The status bar at the bottom of the screen shows information about the current navigation tool and about playback status. The first section from left to right shows instructions for using the active navigation tool. The next shows the current playback status. This will display “Playing,” “Paused,” or “Stopped.” The following section shows the current playback time as minutes:seconds. The next section shows the current playback speed. The last section shows the current rendering speed in frames per second – a common speed measurement in computer graphics. This can help determine how well a user’s computer handles 3D playback. Some ways to increase this are discussed in the section, Controlling drawing detail/speed on page 15.

Refreshing Results

Results can be viewed as a simulation is running, but they will only show the amount of time available at the last load or refresh. To refresh the results at any time, press *F5* on the keyboard or select **Refresh Results** in the **File** menu. The current playback time will be restored after refreshing the results, making the refresh as seamless as possible. If the results are launched from PyroSim or Pathfinder before the simulation is finished, the Results application will automatically refresh and come to the front of the screen when the simulation completes.

File Streaming

The Results application streams files from disk rather than loading the entire results files into memory. File streaming allows results to load quickly and enables very large results to be loaded even if there is limited system memory.

In order for file streaming to work well, the hard drive that contains the results data needs to be fast enough to load the results data at the desired playback speed. With some very large FDS results, however, even a fast disk may not be fast enough to stream the data. These models may cause the results to stutter or pause intermittently during playback or when moving the time slider. In these extreme cases, the results can be loaded into memory. To do so:

1. Activate the problematic FDS results as discussed in Activating Results on page 41.
2. Right-click the problematic FDS results and select **Cache results for smooth playback**. This will ensure the results file will fit in memory and if so, start caching the file while showing a progress bar to indicate its progress.

Controlling drawing detail/speed

There are several options controlling the drawing quality and speed in the Results application, including level-of-detail rendering, hardware versus software rendering, and vertex buffers. Almost all of the options are chosen automatically depending on the capabilities reported by the graphics adapter (GPU).

To view the rendering options, in the **File** menu select **Preferences**. This will open the **Preferences** dialog. Next select the **Rendering** tab.

At any time, the **Max Quality** button may be pressed to load settings that will give the highest quality image available, which are automatically loaded by default. There are some cases where these settings can be problematic for the graphics adapter and can produce incorrect renderings, such as black geometry, incorrect fire, etc. If this is the case, pressing the **Max Compatibility** button will load settings that should work even on these problematic GPUs, at the expense of rendering quality and speed.

The following rendering options are available:

- **Enable level of detail:** Occupants far from the camera will be rendered with less detail than those closer to the camera. This greatly increases rendering speed when viewing occupants while only slightly sacrificing render quality.

- **Enable vertex buffers:** Allows memory buffers used for graphical objects to be stored on the GPU rather than in system memory, reducing the amount of data that must be sent to the graphics card per frame. This increases rendering speed, but on some chipsets (e.g. older Intel graphics or drivers) we have found that using vertex buffers may lead to crashes, so it may need to be disabled for these chipsets.
- **Load animation frames asynchronously:** Allows results animation frames to be loaded from disk in a separate thread. This is useful when the hard drive is too slow to stream the results at the desired playback speed (see File Streaming on page 15). In this case, the results will appear to pause, but the application to remain responsive to user clicks and maintains interaction with the scene. With it off, the UI may become unresponsive while frames are loaded.
- **Occupant animation:** Specifies the algorithm used to animate Pathfinder occupants (see Displaying occupants on page 66).
 - **Animate in Hardware:** Uses a special GPU vertex shader to animate the occupants. This option, coupled with **Enable vertex buffers** can vastly improve occupant-rendering performance on some GPUs.
 - **Animate in Software:** Uses the CPU to animate the occupants.
- **FDS slice rendering:** Specifies the algorithm used to animate 2D FDS slices (see 2D Slices on page 58).
 - **Use Hardware Shaders:** Uses GPU shaders to render the slices. This gives the highest quality rendering of 2D slices and may be the fastest option on modern GPUs.
 - **Use Compatibility Renderer:** Uses legacy OpenGL features to render the slices. This will be automatically chosen if the GPU fails to support shaders.
- **FDS data3d slice rendering:** Specifies the algorithm used to animate 2D Data3d slices (see Data3d Slices on page 55).
 - **Use Hardware Shaders:** Uses GPU shaders to render the slices. This gives the highest quality rendering.
 - **Use Compatibility Renderer:** Uses legacy OpenGL features to render the slices. This will be automatically chosen if the GPU fails to support shaders but still supports 3D textures.
 - **Disabled:** Disables the rendering of these slices. This will be automatically chosen if the GPU fails to support shaders or 3D textures.
- **FDS Smoke/Fire->Render method:** Specifies the algorithm used to render fire and smoke (see 3D Smoke and Fire on page 47). A comparison of the algorithms is shown in Figure 4-7.

NOTE: Using hardware shaders produces the most accurate color representation for fire when using **Radiant** rendering. The other algorithms have restrictions that limit the amount of light that can penetrate and accumulate through the smoke.

 - **Use Hardware Shaders:** Uses GPU shaders to render smoke and fire. This produces the highest quality rendering and allows the use of more complex colormaps that may have colored smoke. It works by performing ray marching, which is a process where a ray is cast from each pixel away from the camera. The smoke/fire data is sampled at regular intervals along the ray and combined to produce the final color.

NOTE: This method may be slower than the other algorithms on older GPUs, especially if the smoke/fire mesh covers a significant portion of the pixels on the screen. The quality of this method can be reduced to increase speed by changing the **Sampling Factor** as discussed in 3D Smoke and Fire on page 47.

- **Use Compatibility Renderer:** Stores the smoke/fire data in 3D textures and draws many translucent slices through the data that are parallel with the view screen. This algorithm does not require the use of GPU shaders, but it has some disadvantages. It may produce visible artifacts where the slices intersect with scene geometry or the edge of the solution mesh. It will also produce incorrect renderings if using non-black smoke combined with the **Radiant Fire Rendering** technique (see 3D Smoke and Fire on page 47). It also has reduced quality and somewhat reduced rendering speed.
- **Use Smokeview Renderer:** Uses an algorithm similar to that used in Smokeview and the compatibility renderer where many slices are drawn through a 3D dataset. It also has the same limitations as the compatibility renderer except that it is generally much faster for static viewpoints. This technique also does not require 3D textures, which improves its compatibility with older GPUs, but the slices are only drawn through the data at fixed angles. This becomes more visible as the camera is rotated around the scene and can produce slight, sudden shifts in the rendering.
- **FDS Smoke/Fire->Reduce banding artifacts:** This option is only available when using the hardware shaders algorithm. It reduces banding artifacts that may be visible when the ray-marching sampling frequency is too low for the fire data as seen in Figure 4-8. It counters the banding effect by using a dithering pattern that is generally more visually pleasing. It negatively impacts the rendering speed very slightly, but is still much faster than the alternative, which is to increase the **Sampling Factor**.

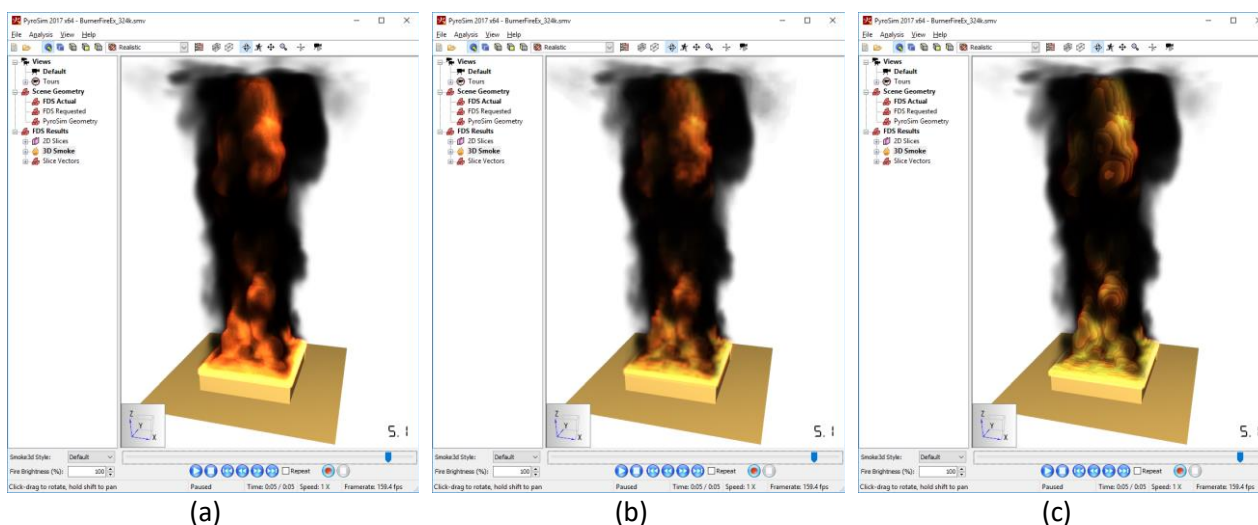


Figure 4-7: Fire rendering algorithms: (a) Hardware Shaders (b) Compatibility Renderer (c) Smokeview Renderer

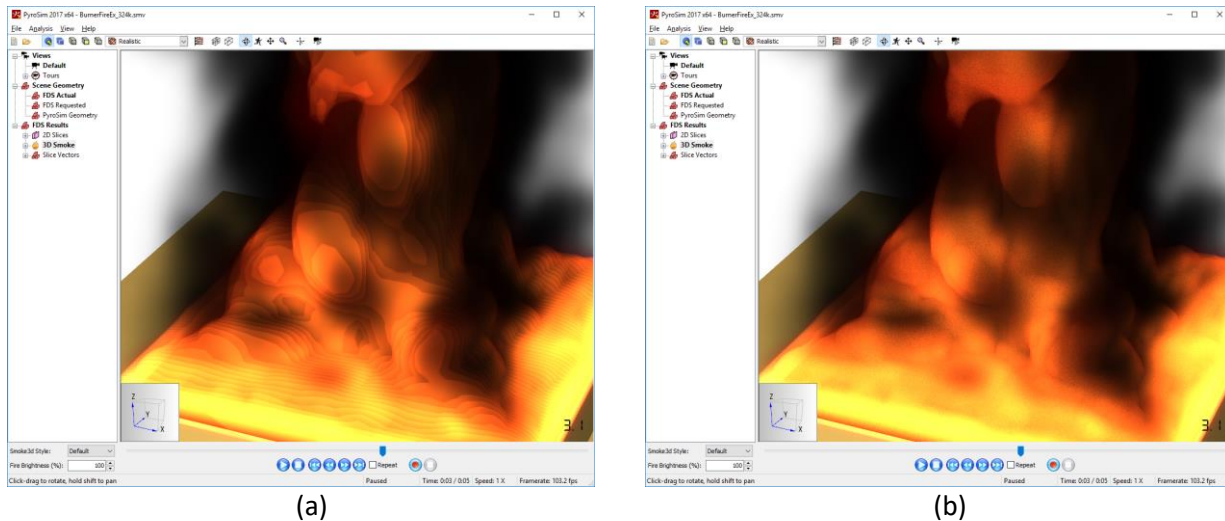


Figure 4-8: Reduce banding artifacts option: (a) off (b) on

- **FDS Volumetric Data3d->FDS data3d rendering:** Specifies the algorithm used to create volumetric rendering of 3D datasets, such as Plot3D and 3D Slices.
 - **Use Hardware Shaders:** Uses GPU shaders to perform volumetric rendering. As with fire/smoke, this uses a ray-marching algorithm to produce the highest quality rendering. NOTE: The quality of this method can be reduced to increase speed by changing the **Sampling Factor** as discussed in Data3d Volumetric Rendering on page 52.
 - **Use Compatibility Renderer:** As with fire/smoke rendering, this algorithm draws many slices through 3D textures to generate the volumetric effect.
 - **Disabled:** This is automatically chosen if the GPU does not support shaders or 3D textures.
- **FDS Volumetric Data3d->Reduce banding artifacts:** This works the same way as for fire/smoke rendering.

Creating Screenshots

A screenshot is a still image of the scene. The Results application can create a single screenshot or multiple screenshots from different views and playback times. To create a screenshot or series of screenshots, perform the following:

1. Activate the desired viewpoint or tour or position the camera at the desired viewpoint from which to take the screenshot.
2. Activate all desired result sets that should be shown in the screenshots.
3. Set any desired view options, such as whether the clock or occupant counts should be shown.
4. In the **File** menu, select **Create Screenshot**. A file chooser will ask for the file name of the resulting screenshot. If multiple screenshots will be created, this is a base name that will be used for each resulting file.
5. The **Screenshot** dialog will appear as shown in Figure 4-9. The following options are available for screenshots:

- **Screenshot Size:** Specifies the dimensions of the resulting screenshots. The size is entered as **width x height**.
 - **Enable Level of Detail:** Determines whether level-of-detail should be used for occupants as discussed in Controlling drawing detail/speed on page 15. Checking this box will reduce the quality of the screenshots but will be faster to render.
 - **Show screenshots when finished:** If checked, when the screenshots are finished, the folder containing the screenshots will open in a Windows Explorer window, with the new screenshots selected. If only one screenshot is created, the screenshot will also be opened in the default image application.
 - **Creation:** Specifies how the screenshots are created, including from which views and at what playback times. The different creation methods are defined below.
6. Press **OK** to create the screenshots.

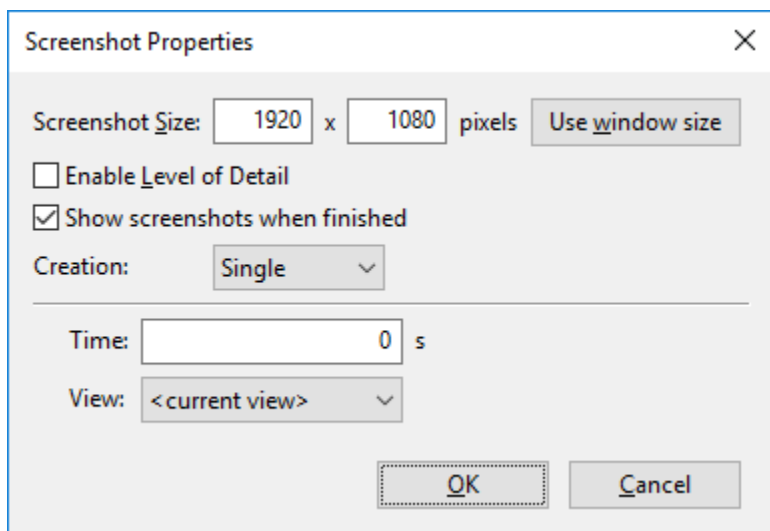


Figure 4-9: Screenshot dialog

Single

The `Single` creation method allows one screenshot to be created. The following options are available:

- **Time:** Specifies the playback time of the screenshot. This defaults to the current playback time when the screenshot dialog was opened.
- **View:** Specifies which view to take the screenshot from. `<current view>` indicates that the screenshot should be taken from whatever is currently displayed in the 3d view.

Multiple

The `Multiple` creation method allows multiple screenshots to be created from several different playback times and views. The following options are available:

- **Start Time:** The first playback time at which to take a screenshot from each view.
- **End Time:** No screenshots will be taken after this playback time.

- **Time Interval:** The amount of playback time between screenshots.
- **Views:** The views from which to take screenshots. Clicking the link will open the **Select Views** dialog as shown in Figure 4-10.

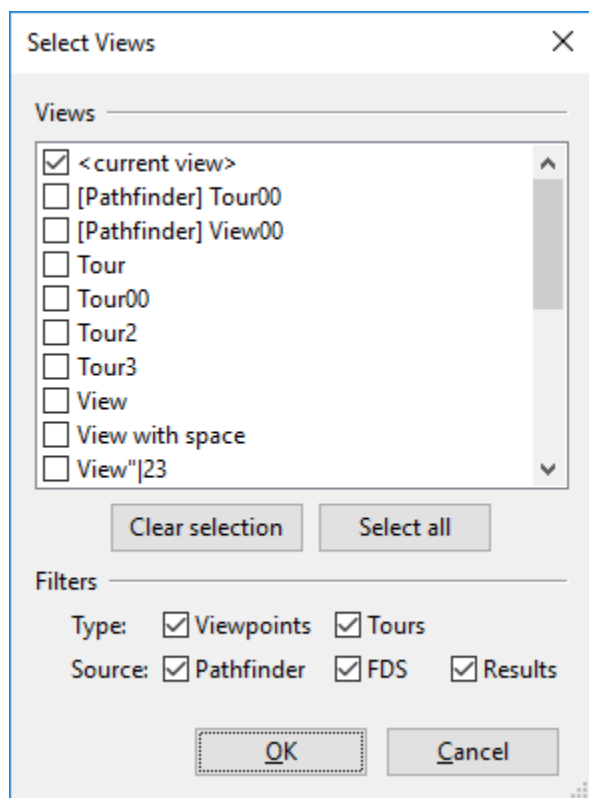


Figure 4-10: Select Views dialog

The **Select Views** dialog can be used to select which views the screenshots will be taken from. The list of views can be limited by unchecking the boxes next to the various filters at the bottom of the dialog.

NOTE: Unchecking these boxes does not uncheck views. Views will still be checked even if hidden from the list.

When using the `Multiple` creation method, the screenshots are created in the following way: for each playback time between the start and end times using the time interval, a screenshot will be created for each selected view. This means that there may be up to $m \cdot n$ screenshots taken, where m is the number of times between the start and end time, and n is the number of selected views. The actual number of screenshots may be fewer than this, as screenshots will only be taken for a tour if the playback time is inside the time range of the tour. The names of the screenshots follow this format:

name_time_viewname.ext

Where *name* is the name of the screenshot as chosen when the screenshot dialog was opened, *time* is the playback time for the screenshot as an offset in seconds from the start of the results, *viewname* is

the name of the view from which the screenshot was taken, and *ext* is the image extension. If `<current view>` is chosen as one of the views, `_viewname` is excluded from the file path.

From File

Screenshots may also be created from a specification file. This method gives the most flexibility and allows the re-use of that specification between results sets, but it also requires the user to create the file.

To do so, create a file with the extension, `.tsv`, which stands for “tab-separated values”. This is a plain-text file, meaning that it can be created in a text editor, such as Notepad or Wordpad. The format of the file is as follows:

```
time1  viewname1    viewname2
time2  viewname1    viewname3    viewname4
time3  viewname2
...
```

Each row specifies a different playback time, with the playback time in the first column. Each column after that specifies the name of a view. If there are multiple views from different sources (e.g. Pathfinder or PyroSim) that have the same name, prepend the name with “[Pathfinder] ” for views created in Pathfinder or “[FDS] ” for views created in PyroSim or Smokeview (do not include the quotation characters). Note the space after the closing bracket.

NOTE: There must be tabs between each column rather than spaces. In addition, if a view name contains a tab character, the tab must be immediately preceded by a backslash character (`\`). Alternatively, the name of the view can be surrounded by quote (") characters. If the name contains a quote character, however, the quote must also be preceded by a backslash (`\`).

Here’s a snippet from a file named “`screenshots_for_model1.tsv`”:

```
0.0    Top    Front
30.0   \"Floor 0\"    Left
120.0  Top
```

Selecting this file with a screenshot name of “`model1.png`” would create the following screenshots:

- `model1_000.000_Top.png`
- `model1_000.000_Front.png`
- `model1_030.000__Floor 0_.png`
- `model1_030.000_Left.png`
- `model1_120.000_Top.png`

Notice that the quotation marks in “Floor 0” turned into underscores (`_`). The Results application does this to special characters that are not allowed in file paths.

Creating Movies

Movies can be created from the Results application for playback on machines that do not have the Results application or for presentation purposes. Movies can be created in two different ways. One way is to create a high-quality movie that is rendered offline. The other is to create a lower-quality movie in real-time that allows the user to interact with the scene while the movie is recording.

High Quality Movies

High quality movies are useful for presentation purposes, and play back smoothly at any resolution and with any high-quality settings turned on, regardless of the complexity of the scene. The disadvantage to this type of movie is that the camera must either remain static or its movements must be pre-planned using camera tours. In addition, the user cannot interact with the scene while the movie is being created (such as changing the selected occupant, turning on and off paths, etc.).

To create a high-quality movie:

1. Either activate the desired camera tour to be used for the movie by double-clicking it in the navigation view or position the camera to the desired viewpoint.
2. Activate all desired result sets that should be shown in the movie.
3. Set any desired view options, such as whether the clock or occupant counts should be shown.
4. In the **File** menu select **Create Movie**. A file chooser will ask for the file name of the resulting movie. From this file chooser, the type of movie can be specified as an **Audio-Video Interlaced** (AVI) file or a **Windows Media Format** (WMV) file. AVI files can be rendered as *uncompressed* or *compressed*. WMV files can only be compressed. Uncompressed files will be very large but will be of the highest-quality. Compressed files are much smaller in size but with some reduced quality. To create compressed AVI files, video *codecs* must be installed on both the computer that will create the file and the computer that will play back the file. Some popular AVI codecs include Xvid, Divx, and ffdshow. WMV files, however, do not require special codecs to be installed on either the source or destination computers, but they will only work on Windows machines unless special software is installed on another platform.
5. A dialog will show some movie making options as in Figure 4-11 (a). The options are as follows:
 - **Video Size** specifies the resulting video's dimensions. This is only limited by the maximum size of a viewport on the computer's video hardware; with some codecs, however, each dimension must be a multiple of 2 or 4. If you have trouble creating a movie, try making the size a multiple of 4. The **Use window size** button will set the size to the 3D window's current size.
 - **Framerate** determines how smoothly the video will play back. The higher this number is, the smoother the playback at the expense of longer rendering time to create the movie and larger file size. 30 frames/second provides reasonably smooth video playback.
 - **Speed** specifies the speed at which the results will play. 1 X will play the movie at real-time speed, 2 X will play the movie twice real-time, etc.
 - **Time Range**: Specifies the beginning and end time of the results over which to create the movie.

- **Enable Level of Detail** specifies that level of detail rendering should be turned on while creating the movie. This will lower the resulting quality, but it will decrease the time to make the movie on scenes with many occupants.
- **Watch Movie when Finished** specifies that the resulting movie should open in the default video player for the chosen video type when the movie is completed.
- **Compressor** shows which video codec is currently selected and allows the codec to be chosen and configured. If **Configure...** is pressed, another dialog will show as in Figure 4-11 (b). This dialog will show a list of the codecs currently installed on the computer. Some codecs allow certain properties, such as the **Quality**, **Data Rate**, and **Keyframe interval** to be specified. These properties control the resulting movie's quality, file size, and seeking performance (how well a movie can be skipped around in the video player), respectively. Some codecs do not expose these properties through this dialog, however. To configure these codecs, you must press the additional **Configure...** button if it is enabled.

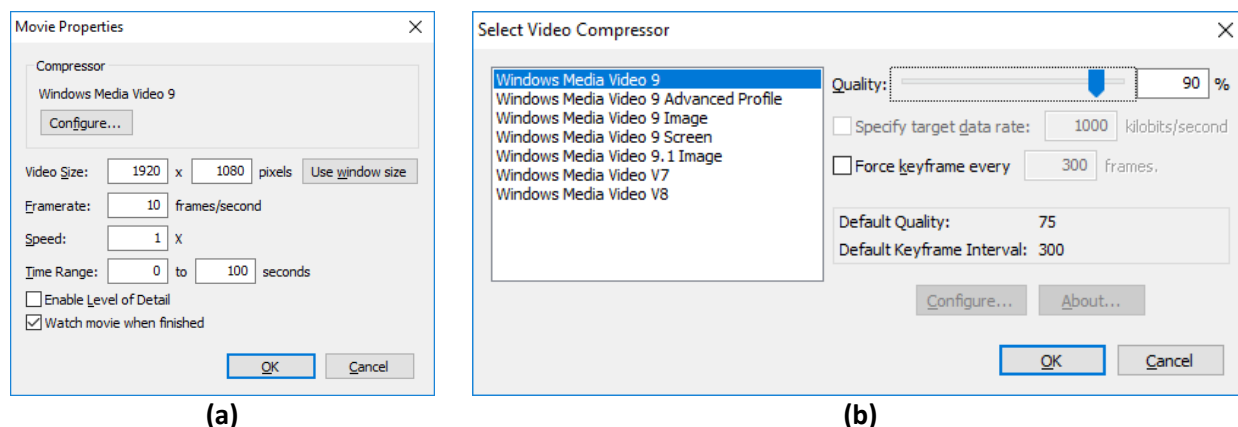


Figure 4-11: Movie option dialogs

6. Press **OK** to start making the movie. While the movie is being created, a dialog will show the progress. If the movie is cancelled before it is completed, there will still be a valid video; it just will not play the entire results.

Real-time Movies

Movies can also be created in real-time. This means that it will record while the user is interacting with the results and it will record everything the user sees except for the cursor. The disadvantage of this type of movie is that it will play back only as smooth and of the same quality as that which the user who created the movie saw. So if the scene is very complex and the movie creator's computer cannot view the results smoothly, the resulting movie's play back will suffer as well.

To create a movie of this type, press the record button (). A file chooser dialog will prompt for the file name of the movie. For real-time movies the only allowable type is WMV. Once the file name is chosen, the codec configuration dialog will be shown as in Figure 4-11 (b). When **OK** is pressed on this dialog, the movie will begin recording. At this point, anything the user sees in the 3D View will be recorded in the movie. Now the user can change the camera angle, start and stop playback, select occupants, change view settings, etc. and the movie will capture everything. At any time, the movie can be paused

or stopped. Pausing the movie will stop the recording but leave the movie file open so that recording can be resumed to the same file. This allows the user to perform some action that they do not want to be recorded. To pause movie recording, press the pause movie button (⏸). To resume recording, press the resume movie button (▶). Stopping the movie will end recording and disallow any further recording to the file. To stop recording, press the stop movie button (⏹).

5. Views, Tours, and Clipping

Views provide the capability to store and retrieve various view state in the Results, to create animated tours of the model, and to clip the model and results using section boxes.

There may only be one active view at a time. There is always a **Default** view, which contains no saved state and is the active view by default. Whenever a view is activated, the stored view state will be restored. View state may optionally include any of the following:

- **Section Box:** A box that can be used to clip away portions of the scene. The clipping is applied to all results, including occupants, though occupants will either be fully visible or fully hidden; they will not be partially clipped as with other scene geometry.
- **Viewpoint:** Positioning information for viewing the scene from one of the cameras.
- **Tour:** A camera animation that changes the viewing perspective over time.

A view may have a viewpoint or tour, but not both. A section box can be used in combination with either, however.

To activate one of the views, either double-click the desired view in the navigation view or right-click it and select **Set Active**. This will make the view active and restore the view state associated with that view. If there is a tour or section box, the tour/section box becomes active and remains active as long as the view is active. If there is a viewpoint instead of a tour, the camera shows the viewpoint, but the navigation tools can still be used to modify the camera. Modifying the camera does not modify the saved viewpoint, however.

To deactivate the view, either double-click the active view or right-click it and select **Set Inactive**. This activates the **Default** view. Alternatively, another view can be made active, which will automatically deactivate the currently active view and make the new one active.

If any views or tours were created in PyroSim, Pathfinder, or Smokeview, they will appear in the navigation view, with a label indicating their origin, such as [Pathfinder] or [FDS]. While these views cannot be edited, they can be duplicated, and the duplicates can be edited. For more information, see Duplicating Views on page 34.

Viewpoints

At any time, the state of the active camera can be saved to a view, including its position, orientation, and zoom. A viewpoint can be recalled later to view the scene from that perspective.

Creating a Viewpoint

A viewpoint can be created in one of the following ways:

- In the **View** menu, select **Views and Tours->New View**.
- In the navigation view, right-click the **Views** group and select **New View**.
- Right-click empty space in the 3D view and select **New View**.

This will add a new view to the model, saving the current camera viewpoint, section box, navigation tool, and camera type (Perspective versus Orthographic) into the new view. The new view will become the active view.

To delete a view, select it in the navigation view and press Delete on the keyboard. If the deleted view was active, the default view becomes active instead.

Recalling a Viewpoint

To recall a viewpoint, perform any of the following:

- Activate the view that contains the viewpoint.
- In the navigation view, right-click the view and select **Show viewpoint**.
- In the 3D view, right-click empty space and select **Reset to Viewpoint**. This will show the viewpoint of the active view.
- Under the **Views** menu, select **Views and Tours->Reset to Viewpoint**. This will also show the viewpoint of the active view.
- While editing a tour, double-click a view containing the viewpoint. See Camera Tours below for more information on editing tours.

Recalling a viewpoint will switch the scene camera to either the perspective camera or orthographic camera and will initialize it with the state of the saved viewpoint. In addition, the saved navigation tool will also become active.

Editing a Viewpoint

If the orientation of a viewpoint needs to be changed, perform the following:

1. Position the scene camera into the new desired position.
2. Perform one of the following:
 - In the navigation view, right-click the view and select **Save viewpoint**.
 - In the 3D view, right-click empty space and select **Save viewpoint**.

This will save both the current camera orientation and navigation tool into the view.

Camera Tours

The Results application provides the capability to create and view camera tours, which can be played back or recorded to video. Tours are useful in the following scenarios:

- Create more cinematic views of the results, such as showing a dynamic panning view of the model.
- Simulate security cameras by having the camera pan/tilt back and forth on a timed loop.
- Show several static views of the scene that change periodically.

Conceptually, camera tours consist of a number of viewpoints at different playback times, referred to as **keyframes**. When a tour is played back, the scene camera smoothly interpolates between viewpoints at the keyframes. Tours may repeat and can start at any time.

When a tour is active, the results playback time range is set to the time range of the tour. See Results Timelines on page 7 for more information.

Tour Editor Dialog

Tour creation and editing is performed through the **Tour Editor Dialog**, as shown in Figure 5-1. The tour editor dialog allows various tour properties to be modified and allows keyframes to be added, removed, and modified.

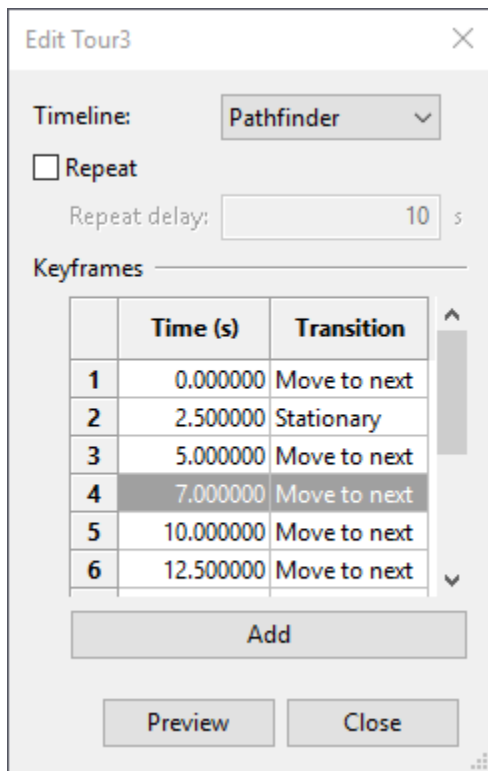


Figure 5-1 Tour editor dialog

While the tour editor dialog (Figure 5-1) is open, the scene camera will stop tracking with the tour. This is to allow positioning of the camera in order to add or edit keyframe positions. To preview changes made to the tour, click the **Preview** button at the bottom of the tour editor dialog. This will lock the scene camera back to the tour and begin playback at the current time. If the current time is outside the tour time or at the end of it, the playback time is changed to the beginning time of the tour. To end the preview, click **End Preview**. The scene camera will once again unlock, allowing changes to the keyframes.

While the tour editor dialog is open, clicking a keyframe in the table will also set the playback time of the results to that keyframe time and show the viewpoint of that keyframe in the 3D view. Similarly, changing the scene playback time using the time slider will automatically select the most recent keyframe in the table.

While the tour editor dialog is open, another view can be double-clicked in the navigation view to show the viewpoint of that view.

Tour Properties

Tours have the following properties:

- **Timeline:** Specifies which results timeline the tour belongs in. The value may either be *Pathfinder* or *FDS*. If the value is *FDS*, the keyframe times are stored relative to the **FDS Time Offset** as described in Results Timelines on page 7. That way, if the **FDS Time Offset** changes, the tour will still remain aligned with the FDS results data set.
- **Repeat:** Specifies whether the keyframes should repeat if the playback time is past the last keyframe time.
- **Repeat delay:** If **Repeat** is checked, this value indicates how much time should be spent on the last keyframe before repeating the first keyframe.
- **Keyframes:** Provides a list of all the keyframes in time order. Keyframes can be added and modified in several ways as described below.

Keyframes have these properties:

- **Time:** The keyframe's time. This is the time at which the scene camera should be in the recorded position.
- **Viewpoint** (not shown in the keyframes table): The desired position, orientation, and zoom of the camera when the keyframe is reached.
- **Transition:** Specifies how the scene camera will transition to the *next* keyframe. If this is the last keyframe and **Repeat** is checked, the next keyframe is the first keyframe. The transition value can be one of the following:
 - **Move to next:** The scene camera will smoothly move to the next keyframe.
 - **Stationary:** The scene camera will remain stationary until the next keyframe.

Creating a Tour

To create a tour, perform the following:

1. Add an empty tour to the results by performing one of the following (NOTE: These actions will create an empty tour, set it active, and open the tour editor dialog shown in Figure 5-1):
 - In the **View** menu, selected **Views and Tours->New tour**.
 - In the navigation view, right-click under the Views group and select **New tour**.
 - In the 3D view, right-click empty space and select **New tour**.
2. In the tour editor dialog, choose the desired **Timeline**, but leave **Repeat** unchecked. If it is desired for the tour to repeat, only check the repeat box after all the keyframes for one loop have been added.
3. Move the time slider to the desired start time of the tour.
4. Position the camera to the desired start location of the tour. This can be done either with the navigation tools or by double-clicking another view to show its viewpoint.

5. Click the **Add** button in the tour editor dialog. This will record the camera position and playback time as a new keyframe. The time can be adjusted by typing in a new time into the keyframe table.
6. Decide what kind of transition should take place to the next keyframe and choose the appropriate value in the **Transition** column of the keyframes table.
7. Repeat steps 3 through 6 for subsequent keyframes.
8. Check the **Repeat** box if the tour should repeat.
9. Click the **Close** button to close the tour editor dialog.

NOTE: While adding keyframes, it may, at times, be desirable to check a transition to make sure it looks correct. This can be done by clicking the keyframe row that starts the transition and clicking the **Preview** button at the bottom of the tour editor dialog.

Editing a Camera Tour

There will be times when changes need to be made to a tour, such as the start time, camera position during a transition, adding a repeated section of the tour, etc. Tours can be edited in several ways. To begin editing the tour, open the tour editor dialog in one of the following ways:

- In the navigation view, right-click the view with the tour, and click **Edit tour**. This will activate the view and open the tour editor dialog.
- With a tour active, in the 3D view, right-click empty space and select **Edit tour**. This will begin editing the tour in the active view.
- With a tour active, in the **View** menu, select **Views and Tours->Edit tour**. This will also begin editing the tour in the active view.

With the tour editor dialog open, several changes can be made, including the following:

Fix the viewpoint of a keyframe

Sometimes a keyframe doesn't have quite the right orientation or camera position. This can be changed by doing the following:

1. Select the keyframe that needs to be updated in the **Keyframes** table.
2. Position the camera to the desired location.
3. Right-click the keyframe and select **Record camera position**.

Fix a viewpoint in the middle of the tour

Perhaps all the keyframes look correct, but there is a point in the tour when the camera does not look quite right. This can be fixed by doing the following:

1. Preview the tour.
2. Move the time slider to the time when the camera looks bad.
3. End the preview.
4. Adjust the camera until it looks correct.

5. Click the **Add** button in the tour editor dialog. This will insert a new keyframe at the current playback time.
6. Preview the tour again and make sure everything looks correct.

Delete a portion of the tour

A portion of the tour can be removed by deleting keyframes. To do so, select the keyframes and either press Delete on the keyboard or right-click the frames and select **Remove**.

Shift all or part of the tour to a new time

There might be scenarios where all or part of a tour needs to be moved to a different time. If the time of only one keyframe needs to be corrected, edit the time value directly in the **Keyframes** table. If all or a portion of the tour needs to be shifted:

1. In the **Keyframes** table, select the keyframes whose times you would like to shift.
2. Right-click the keyframes and select **Shift Times**. This will open the **Shift Keyframes** dialog as shown in Figure 5-2.
3. In the dialog, enter the new **Time** for the first of the selected keyframes or enter an **Offset**.
4. Click **OK**.

The selected keyframe times will be shifted by the specified amount. NOTE: The keyframes can only be shifted within the window of times surrounding the selection. If all the keyframes are selected, there are no limits on the amount of time to shift.

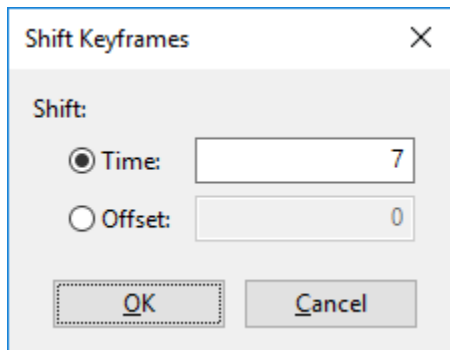


Figure 5-2 Shift Keyframes dialog

Duplicate all or part of the entire tour

If you want to duplicate some or all of the tour so that the section will play again at another time within the same tour, perform the following:

1. Select the keyframes to be duplicated.
2. Right-click the keyframes and select **Duplicate**. This will open the **Duplicate Keyframes** dialog as shown in Figure 5-3.
3. The following options may be chosen to determine when the duplicated keyframes will take place:

- **Insert at end of tour:** This will cause the duplicated keyframes to be inserted after the last keyframe of the tour. The **Last keyframe delay** specifies how much time there is between the last keyframe and the first duplicated keyframe.
- **Insert at specific time:** This will cause the duplicated keyframes to be inserted at a specific time. If the chosen time is in the middle of the tour, the subsequent keyframes' times are shifted so that the first subsequent keyframe takes place after the last duplicated keyframe. The amount of time between the last duplicated keyframe and first subsequent keyframe is the same as the time between the last selected keyframe and its subsequent keyframe or 10 seconds if the last selected is also the last keyframe.
- **Insert at end of the selected keyframes:** This will cause the duplicated keyframes to be inserted at the same time as the next unselected keyframe. The subsequent unselected keyframes' times are shifted the same way as when inserting at a specific time.

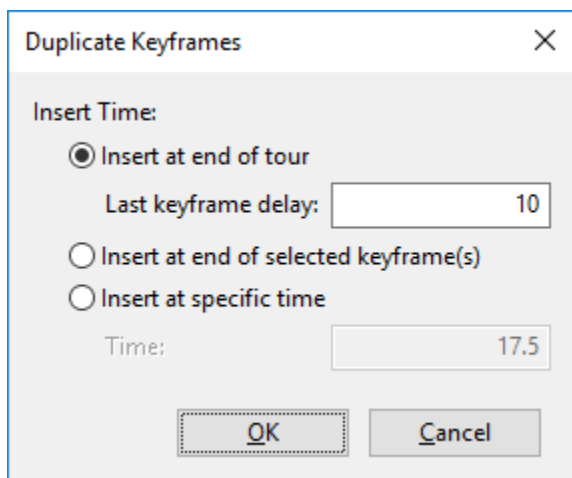


Figure 5-3 Duplicate Keyframes dialog

Hold a viewpoint for some duration in the middle of the tour

After creating a tour, it may be decided that the camera should remain stationary at a particular viewpoint for some length of time before moving on to the next keyframe. This can be done by duplicating the keyframe that should be held and selecting **Insert at end of selected keyframe** as described above.

The new keyframe will have the same viewpoint as the previous keyframe, which keeps the camera stationary before moving on to the next keyframe. While not completely necessary, it may also be desirable to set the old keyframe transition to **Stationary** to make it obvious in the keyframes table which keyframe is being held.

Section Boxes

A section box is a convex region defined by up to six planes and is used to limit the scope of visible geometry and results data. All data outside the box are clipped from view. An example of the clipping is shown in Figure 5-4.

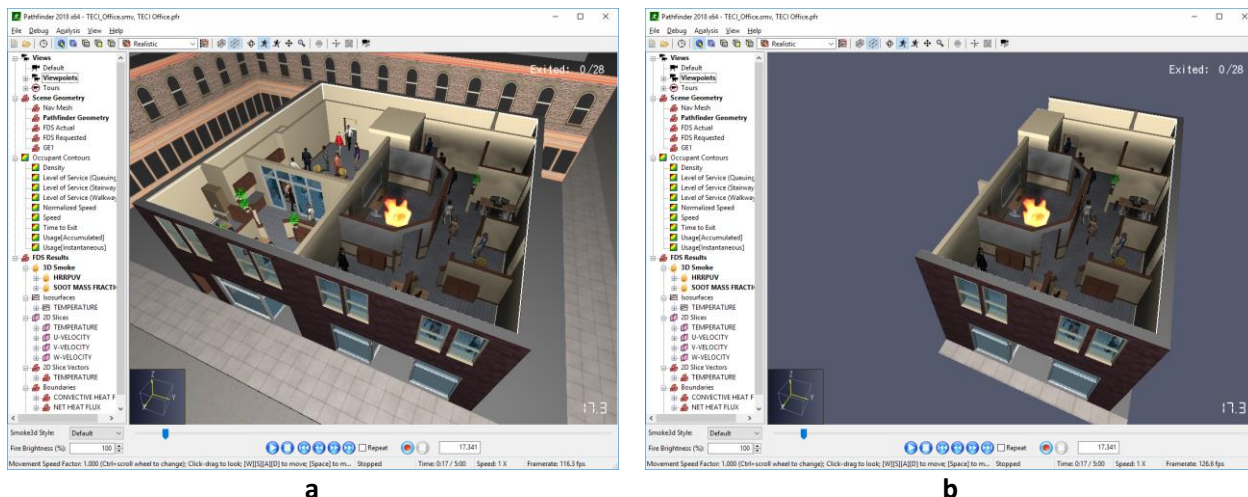


Figure 5-4 Effect of a Section Box (a) section box disabled (b) section box enabled

NOTE: When viewing a section box with the floors of the model spread out, each floor is clipped by the intersection of the section box and the Z clipping planes of the floor. See Viewing multi-floor problems on page 11 for more information on floors.

The active view's section box can be disabled by going to the **View** menu and de-selecting **Views and Tours->Enable Section Boxes**.

Creating a section box

A section box can be created in several ways:

- In the navigation view, right-click a view that does not already have a section box and select **Add section box**. This will add a section box that encompasses all visible geometry.
- In the **View** menu select **Add section box**. This will add a section box to the active view if it does not already have one.
- In the 3D view, right-click in empty space and select **Add section box**. This will also add a section box to the active view.
- Select several geometric objects, right-click one of them, and select **Add section box**. If the active view does not already have a section box, one will be added that encompasses only the selection.

After a section box is added, the **Section Box Dialog** appears as shown in Figure 5-5.

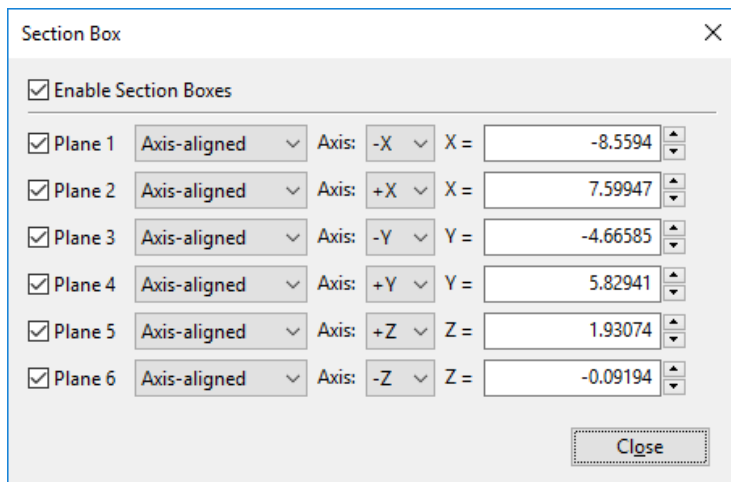


Figure 5-5 Section Box dialog

While the section box dialog is open, the section box is displayed as a dashed outline of the region with translucent faces. The individual planes of the section box can be added and removed by checking and unchecking the box next to each plane.

NOTE: The Section Box dialog also has a shortcut to the **View** menu item, **Enable Section Boxes**, which enables or disables the active view's section box. This option does not alter the section box in any way.

Editing a section box

A section box can be edited in several ways:

- In the navigation view, right-click a view and select **Edit section properties**. If the view is not already active, it will be made active, and the section box dialog will appear.
- In the 3D view, right-click in empty space and select **Edit section properties**. This will edit the active view's section box.
- In the **View** menu, select **Views and Tours->Edit section properties**. This will also edit the active view's section box.

NOTE: If the view does not have a section box, the section box dialog will show all planes as unchecked. The values for each plane are initialized to the bounds of the model.

Remove a section box

A section box can be removed from a view in any of the following ways:

- In the navigation view, right-click a view with a section box and select **Remove section box**.
- In the 3D view, right-click in empty space and select **Remove section box**. This will remove the section box from the active view.
- In the **View** menu, select **Views and Tours->Remove section box**. This will also remove the active view's section box.

Duplicating Views

With the exception of the default view, any view can be duplicated, including those from Pathfinder, PyroSim, or Smokeview. To duplicate a view or several views:

1. In the navigation view, select the views.
2. Right-click the selected views, and select **Duplicate view**.

This will create a copy of each view. All duplicated views can be modified and deleted.

Views Files

Views are stored in several different files including the following:

- *modelname_views.json* – Views file created in Pathfinder. This is saved in a JSON file format.
- *modelname.views* – Views file created by PyroSim. This is also saved in a JSON file format.
- *modelname.ini* – File created by PyroSim or Smokeview that contains viewpoints and tours created for use in Smokeview.
- *modelname_resultsviews.json* – Views file created by the Results application for storing views created in the results, including those duplicated from PyroSim, Pathfinder, or Smokeview views. This is also stored in a JSON file format.

NOTE: *modelname* is the name of the originating Pathfinder, PyroSim, or FDS file. If both Pathfinder and FDS results are opened in the results at the same time, the *resultsviews.json* filename is based on the filename of the first opened results. For this reason, when loading both Pathfinder and FDS results, they should always be opened in the same order.

The *resultsviews.json* file is saved automatically whenever changes are made to the views or when the current results set is closed, either by creating new results or by opening another results set.

Importing Views

Views can be imported into the results, for instance, from another results data set. To do so:

1. In the **File** menu, select **Import->Import Views**.
2. Choose the views file to open. This can be any of the above types except for Smokeview views.
3. Click **Open**.

The views are imported as editable results views and are saved into the *resultsviews.json* file.

6. Annotations

The Results application provides several different ways to annotate results, including the ability to add **dimensions** and **custom labels**.

Annotations are added by drawing them directly into the model using one of the provided tools, detailed below. This can be accomplished in either the Perspective View or Orthographic View, though in some cases it may be easier to use one over the other.

All annotations are automatically saved to a file called “x_session.json”, where “x” is the name of the results file. Annotations are automatically restored when the results are reloaded. If multiple result sets are loaded (for example, both Pathfinder and FDS results), the annotations are saved with and restored from the first loaded set.

Dimensions

Dimensions may be added to any result set to annotate the length of features in the model or distances as shown in Figure 6-1. Dimensions may be shown or hidden by going to the **View** menu and selecting or deselecting **Show Dimensions**.

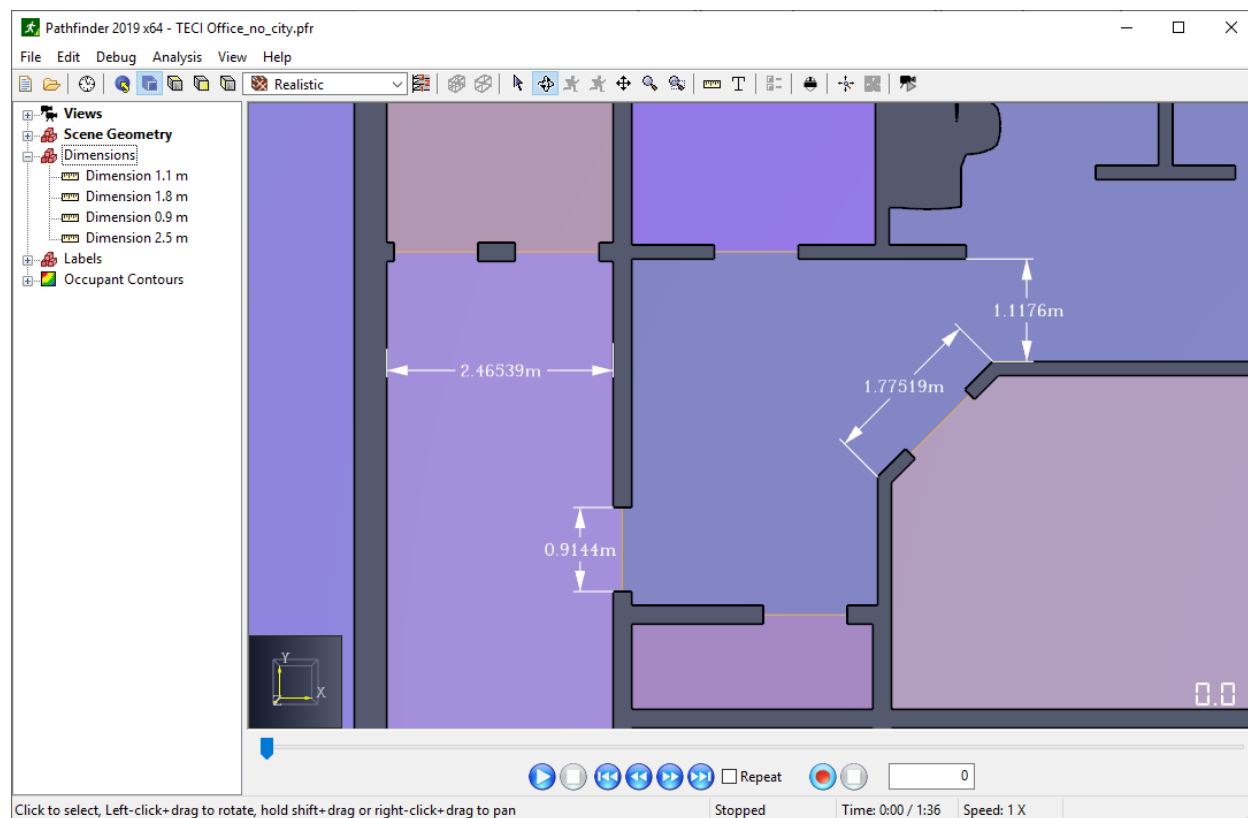



Figure 6-1 Dimensions

To draw a dimension, perform the following steps:

1. Orient the view so that the feature to be dimensioned is visible.
2. Click the **Dimension Tool** () button in the toolbar above the 3D view to start drawing.
NOTE: The Dimension Tool button can be clicked again to pin the tool as in PyroSim and Pathfinder. When pinned, the dimension tool will stay active until **Esc** is pressed on the keyboard, allowing multiple dimensions to be drawn, one after the other. If the tool is unpinned, the most recently used navigation tool will become active after a dimension is drawn.
3. Single-click the first endpoint of the feature to be dimensioned.
4. Single-click the second endpoint of the feature.
5. If the dimension does not need further adjustments, either press **Enter** on the keyboard or click the second point again, and the dimension will be added. Skip the remaining steps.
6. If the dimension needs adjustments, such as to offset it from the dimensioned feature or represent its length along an axis, move the mouse away from the second point. The dimension preview will show any adjustments to the dimension. The dimension can be modified in the following ways as demonstrated in Figure 6-2 on page 37:
 - a. **Offset from feature** – The resulting dimension is parallel to the feature and matches its length as shown in Figure 6-2 (b). To dimension in this manner, move the cursor along an imaginary line extending from the second dimension point and perpendicular to the dimensioned feature.
NOTE: This is the only available dimension adjustment if the feature is already parallel to the X, Y, or Z axis.
 - b. **Length along Y axis** – The resulting dimension shows the length of the feature when projected along the Y Axis as shown in Figure 6-2 (c). To dimension in this manner, move the cursor away from the second dimension point along the X axis.
 - c. **Length along X axis** – The resulting dimension shows the length of the feature when projected along the X axis as shown in Figure 6-2 (d). Add this dimension by moving the cursor away from the second dimension point along the Y axis.
 - d. **Length along Z axis** – The resulting dimension shows the length of the feature when projected along the Z axis. Add this dimension by moving the cursor away from the second dimension point along the Z axis.
7. Click a third time to finalize the dimension with its adjustment and add it to the model.

Dimensions may be selected and deleted from either the 3D view or the **Navigation View** under the **Dimensions** group.

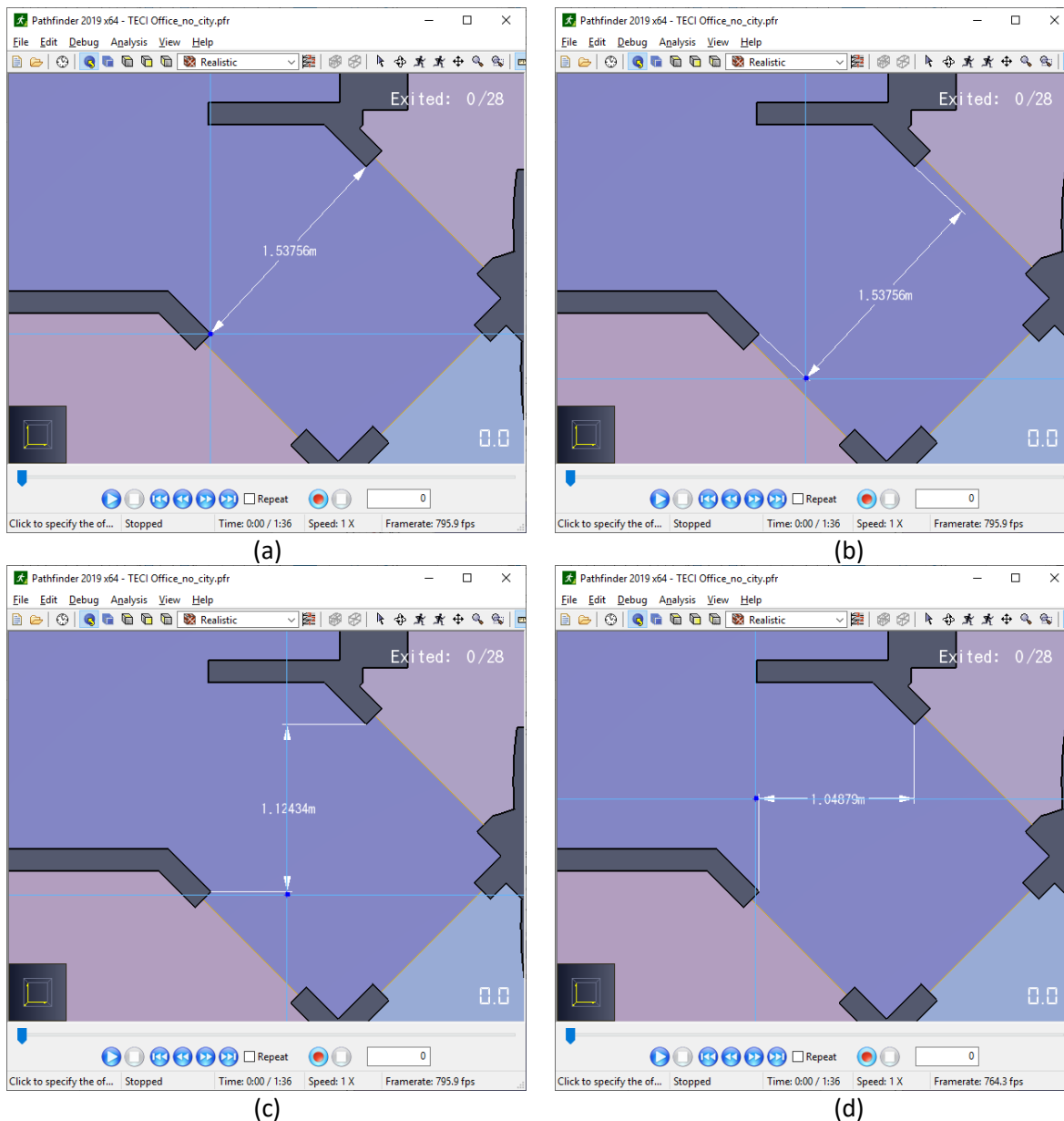
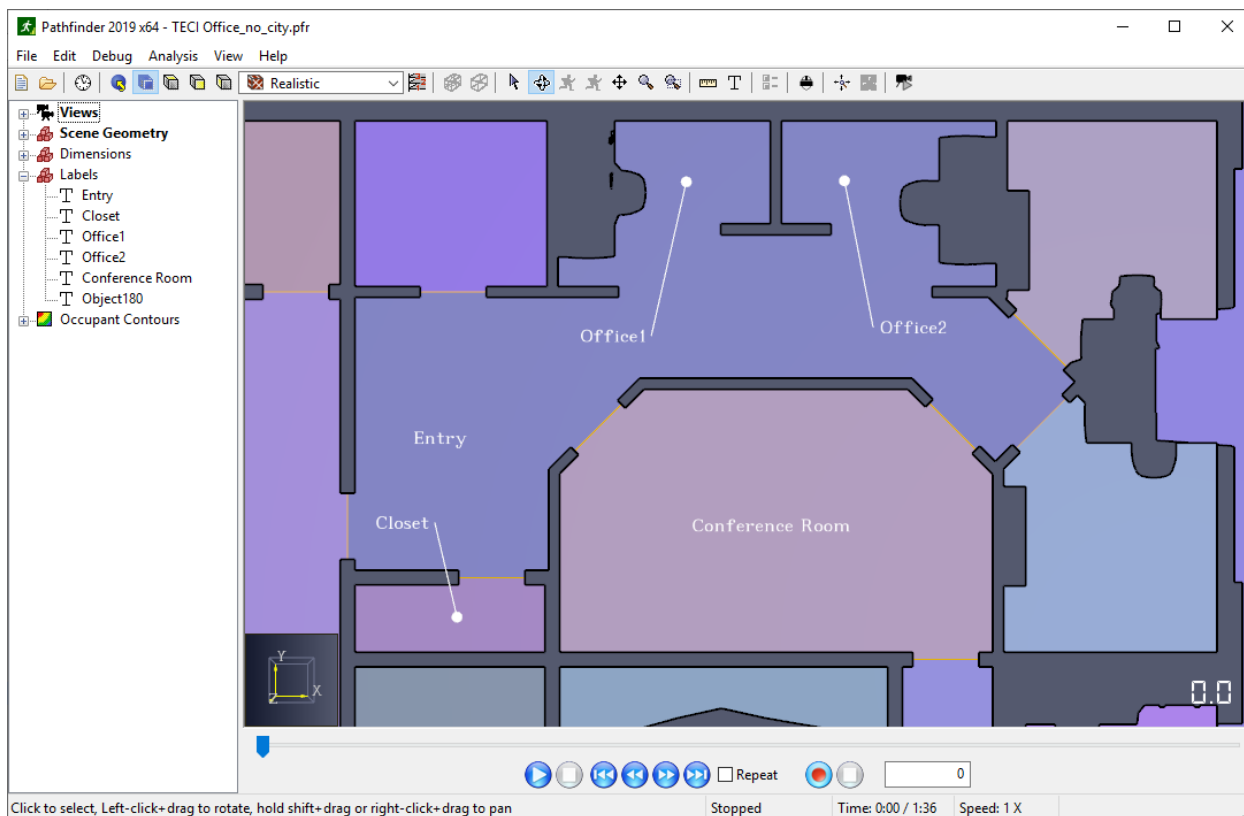



Figure 6-2 Types of dimension offsets. (a) Shows an unmodified dimension. (b) Shows a dimension offset from its feature. (c) Shows the feature's length along the Y axis. (d) Shows the feature's length along the X axis.

Custom Labels

Another way to annotate results is to add custom labels as shown in Figure 6-3. Custom labels can be used to add any text to the model and can be added with or without a leader. Labels may be shown or hidden by going to the **View** menu and checking or unchecking **Show Labels**.

**Figure 6-3 Custom Labels**

To create custom labels, perform the following steps:

1. Orient the view so that the feature to be labeled is visible.
2. Click the **Label Tool** () button in the toolbar above the 3D view to start drawing.
NOTE: The Label Tool button can be clicked again to pin the tool as in PyroSim and Pathfinder. When pinned, the label tool will stay active until **ESC** is pressed on the keyboard, allowing multiple labels to be drawn, one after the other. If the tool is unpinned, the most recently used navigation tool will become active after a label is drawn.
3. To add a label in place without a leader line, double-click the location for the label or single-click and then press **Enter** on the keyboard. Then skip to step 5.
4. To add a label with a leader line, single-click the point to be labelled. Then move the mouse to the desired label location and click again. A line is drawn between the labelled point and the label.
5. In the dialog that appears, enter the desired label text. The label defaults to the name of the object where the first point was picked, if one exists. In addition, a label may be split into multiple lines if desired.
6. Click **OK** to create the label.

As with dimensions, labels appear in the **Navigation View** under the **Labels** group and can be deleted or renamed if necessary.

7. Viewing FDS Results

The Results application provides support for viewing results from the Fire Dynamics Simulator (FDS) from NIST. It is the default results viewer for PyroSim, which is a graphical user interface for FDS. FDS results may optionally be shown along with Pathfinder results. When FDS results are loaded, the **Navigation View** will list all of the supported FDS output as shown in Figure 7-1.

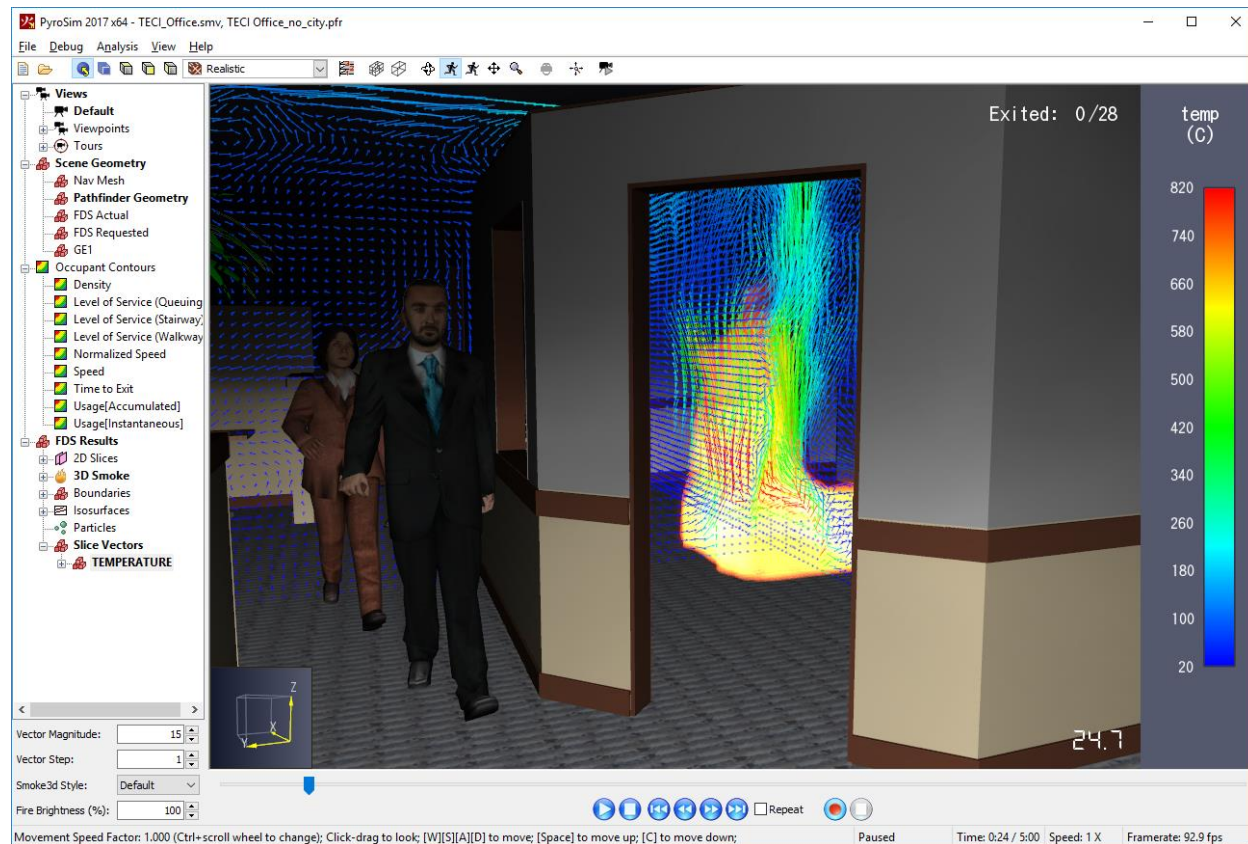


Figure 7-1: PyroSim and Pathfinder results shown with fire and temperature vectors and occupants

The Results application supports the following FDS output:

- **3D Smoke:** Represents the smoke and fire and can be rendered realistically using radiant lighting or volumetrically.
- **3D Slices (SLCF):** Contain volumetric data of gas-phase quantities. This can be viewed as a volumetric rendering or it may have 2D slices or isosurfaces created from it.
- **Plot3d (PL3D):** Like 3D Slices, contains volumetric data from which volumetric renderings, 2D slices, and isosurfaces may be created.
- **Isosurfaces (ISOF):** Represent 3D contours of gas-phase quantities at specific values.
- **2D Slices (SLCF):** Represent planar slices of gas-phase data.
- **2D Slice Vectors (SLCF):** Represent the direction of air flow in planar slices and can be colored by gas-phase quantities.

- **Boundaries (BNDF):** Represent solid- or gas-phase quantities at the boundaries of obstructions.
- **Particles (PART):** May be emitted by devices or vents.

Results Organization

Each FDS output may be split into multiple objects depending on how many meshes the object resides in. Figure 7-2 show an example model where there are multiple meshes. In this example, there are five slices of `SOOT DENSITY` at different locations. The slice at `X=13.319` only goes through one mesh, so there is only one entry in the Navigation View. The slices, `X=13.5` and `Y=17.104`, however, go through multiple meshes, so they have multiple entries in the Navigation View.

The hierarchy in this view makes it easy to activate an entire slice as it goes through multiple meshes or only activate the slice in one or more meshes. It also demonstrates how individual results objects have been divided among multiple meshes.

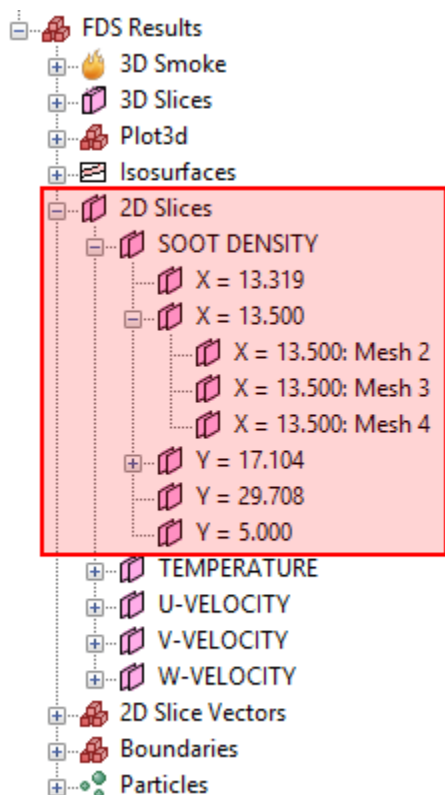


Figure 7-2: FDS Results Organization

Activating Results

When an FDS results set is loaded, none of the results are visible. They must be selectively activated to be shown. To do so, in the Navigation View either right-click the desired results object and select **Show** or double-click the object. To deactivate the object, right click it and select **Hide** or double-click it.

Results objects can be activated or deactivated at any level in the hierarchy. For instance, in Figure 7-2, all `SOOT DENSITY` slices can be activated by double-clicking `SOOT DENSITY` in the navigation view. In addition, all `SOOT DENSITY` slices at `X=13.5` can be activated by double-clicking `X = 13.5000`.

Note that only one FDS quantity can be active at a time. For instance, a `TEMPERATURE` slice may NOT be shown along with a `SOOT DENSITY` slice. You can, however, mix different result types as long as they are the same quantity. For instance, you can show a `TEMPERATURE` slice and a `TEMPERATURE` boundary. The exception to this rule is that 3D Smoke and Particles colored with the `Default` color can be shown in combination with any quantity. This is because these outputs have no associated quantity.

When an object is activated the Results application performs several actions:

- In the Navigation View, the label for the object becomes **bold**. In addition, all the parent nodes of the object become bold as well. For example, if `SOOT DENSITY` slice `X = 13.5000`: `Mesh 3` is activated, the Navigation View will appear as shown in Figure 7-3. This makes it easy to see which items in a hierarchy are currently active even if a node is collapsed.
- The first time the object is activated, the file associated with that object will be read from disk, and information will be gathered about the data and cached in a new file that ends with the extension, “`tcframes`.” For instance, if particles are loaded from a file named “`carport.prt5`”, the caching file will be named “`carport.prt5.tcframes`”.
- Subsequent activation of the object will load the cached information from the “`tcframes`” file. This occurs much faster than loading the first time. In addition, the application checks to see if there are new frames in the object’s file, which can occur if the FDS simulation is still running. If there are more frames, information is gathered for only the new frames and added to the cache file.
- If necessary, a colorbar is added to the right side of the main window as shown in Figure 7-4. The colorbar can be edited as described in [Colorbars](#) on page 43.
- Other activated objects are deactivated if they do not have a compatible quantity with that of the newly activated objects. For instance, if a `SOOT DENSITY` slice is already active and a `TEMPERATURE` slice is being activated, the `SOOT DENSITY` slice is deactivated.
- The time range is changed to be the union of the time ranges of all activated objects.
- The newly activated objects are shown in the 3D View.
- Under **File->Preferences**, if the **Begin playback on activation** option is checked and the results are currently paused, the following will occur:
 - If the current playback time is outside the time range of the newly-activated results, the playback time is set to the beginning of the new results’ time range.
 - If the new results’ time range has a length greater than zero, playback will resume.

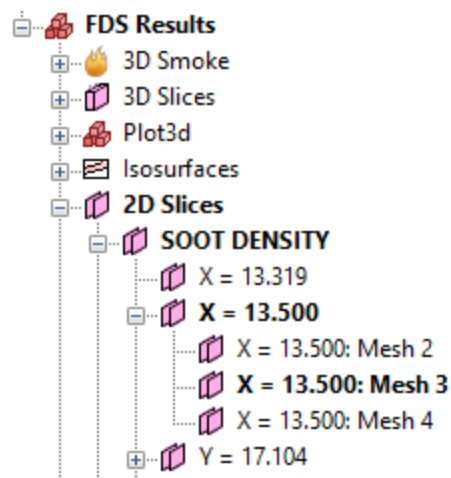


Figure 7-3: Activated 2D slice in the Navigation View

Colorbars

With the exception of 3D Smoke and Particles viewed with the `Default` coloring, all FDS results data is colored with a colormap. A colormap provides a way to map data values to a color that can be displayed on the screen. When a colormap is in use, it will be shown as a colorbar on the right side of the 3D View as shown in Figure 7-4.

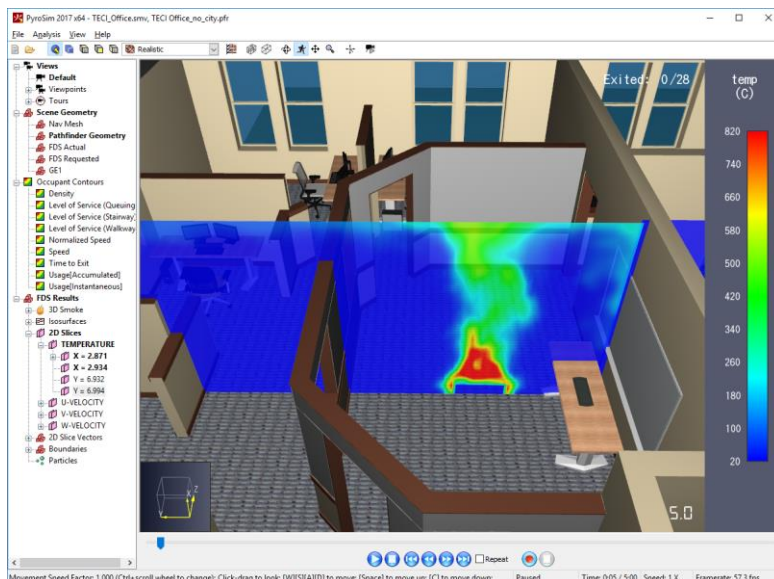


Figure 7-4: Results colorbar

The colorbar shows the colormap as well as which data values are mapped to the colors. At the top, it also shows an abbreviation of the quantity and the quantity's unit in parentheses. In Figure 7-4, the label "temp" corresponds with the quantity, `TEMPERATURE`, and the unit is "C" for degrees Celsius.

Highlight region

The colorbar can be left-clicked to add a highlight region. A highlight region changes all of the values of a limited range to a specific color to make it easier to see as shown in Figure 7-5. By default, a highlight region covers 5% of the data values surrounding the clicked location on the colorbar (2.5% above and 2.5% below). The size of the region can be changed by using the scroll wheel on the mouse. In addition, the highlight region can be removed by right-clicking the colorbar. The highlight region can also be moved by clicking and dragging the left mouse button on the colorbar.

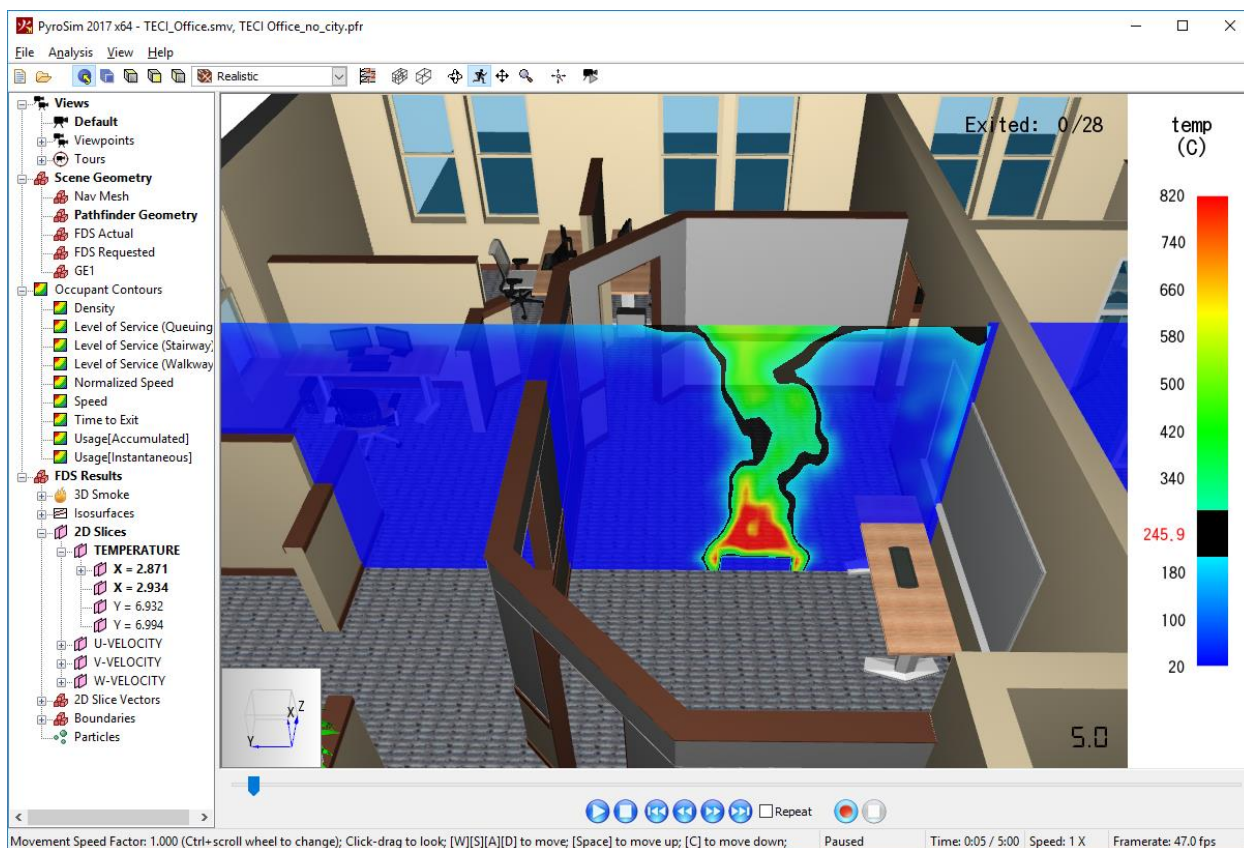


Figure 7-5: Colorbar highlight region

More fine-grained control over the colorbar can be achieved through the **FDS Preferences** dialog as discussed in Global FDS Properties below.

Global FDS Properties

There is a set of global properties for FDS results that controls how the results are presented, including colors, time offsets, and interpolation. These settings are located in the **FDS Preferences** dialog. To open this dialog, either double-click the colorbar if results are already active or under the **Analysis** menu, select **FDS Preferences**. The FDS Preference dialog is shown in Figure 7-6.

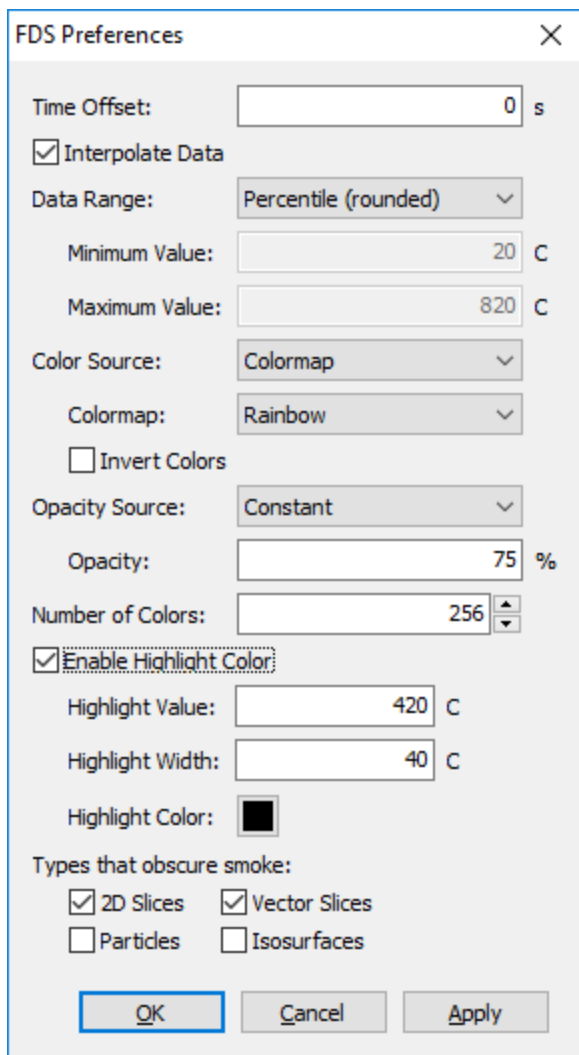


Figure 7-6: The FDS Preferences dialog

- **Time Offset:** Provides a shortcut to the **FDS Time Offset** as described in Results Timelines on page 7.
- **Interpolate Data:** Indicates that results should be linearly interpolated over time. This causes the animation of results to look smoother as they are played back.
NOTE: Interpolation takes extra processing power. It may be turned off to improve rendering performance. In addition, isosurfaces created by FDS are never interpolated.
- **Data Range:** Specifies the data range for mapping results data values to colors in the colormap. The **Minimum Value** is mapped to the first (bottom) color in the colormap. The **Maximum Value** is mapped to the last (top) color in the colormap. Colors in-between are mapped linearly between the first and last colors. Values outside this range are clamped to the first or last colors. There are several preset **Data Range** values depending on the active results:
 - **Percentile (rounded):** When the results are loaded, a histogram is created from the data. The middle 98% of all data values over all time are rounded and then used for the minimum and maximum values of the data range.

- **Percentile:** The middle 98% of all data values are used for the data range.
- **Global (rounded):** When the results are loaded, the absolute minimum and maximum data values are calculated. These values are rounded and used for data range.
- **Global:** The absolute minimum and maximum data values for the active results are used for the data range.
- **Custom:** Allows the user to type in the desired data range.
- **Color Source:** Specifies the source of the colors. This can be one of the following values:
 - **Colormap:** Uses a pre-defined colormap that is a sequence of varying colors.
 - **Constant Color:** A single color is used for all data. This is usually used in combination with a **Gradient Opacity Source**.
- **Colormap:** The name of the colormap to use when the **Color Source** is set to **Colormap**.
- **Invert Colors:** Whether to reverse the colors in the colormap when the **Color Source** is set to **Colormap**.
- **Color:** The color to use when the **Color Source** is set to **Constant Color**.
- **Opacity Source:** Specifies the source of opacity values. Opacity controls how opaque results are.
 - **Constant:** The opacity values are always the same.
 - **Gradient:** A gradient function will be used to linearly map the data values to opacity values. This can be used to highlight either the lower or upper range of the results data as the other end will be more translucent.
- **Opacity:** Specifies how opaque the results are when the **Opacity Source** is set to **Constant**.
- **Opacity at Minimum¹:** The opacity at the **Minimum Value** as specified in the **Data Range**. This can be any value and does NOT have to lie between 0 and 100%.
- **Opacity at Maximum¹:** The opacity at the **Maximum Value** as specified in the **Data Range**. This can be any value and does NOT have to lie between 0 and 100%.
- **Number of Colors:** Specifies how many samples should be taken from the colormap to generate the final color function. The samples are taken at evenly-spaced intervals. Using a value of 256 will create a smooth gradient of colors. Using a low value, such as 8, will create discrete colors, leaving visible color bands in the visualization.
- **Enable Highlight Color:** Enables the highlight region as discussed in Highlight region on page 44.
- **Highlight Value:** The center value of the highlight region.
- **Highlight Width:** The size of the highlight region. Half of this will be above the Highlight Value and half will be below.
- **Highlight Color:** The color of the highlighted values.
- **Types that obscure smoke:** Controls which results types appear easily on top of smoke when both are active. When this option is enabled, the result object will be easier to see when it is embedded in thick smoke, but the depth cues may be more difficult to understand. Figure 7-7 shows this option enabled and disabled for a 2D Slice.

¹ The Opacity at Minimum can be either less than or greater than the Opacity at Maximum.

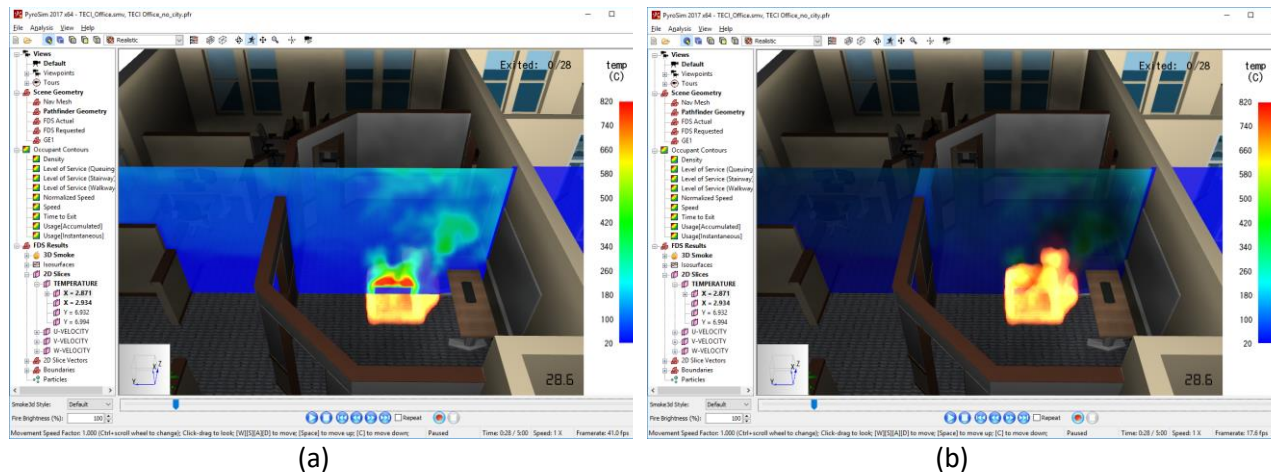


Figure 7-7: 2D slices obscure smoke option (a) enabled (b) disabled

3D Smoke and Fire

By default, all FDS simulations that contain fire will output two s3d files for each mesh. Together these files can be viewed in the Results application as smoke and fire as shown in Figure 7-8. When these files are present, the Results application will show a **3D Smoke** group in the Navigation View. In the 3D Smoke group, **SOOT MASS FRACTION** represents the smoke information and **HRRPUV** (heat release rate per unit volume) represents the fire information.

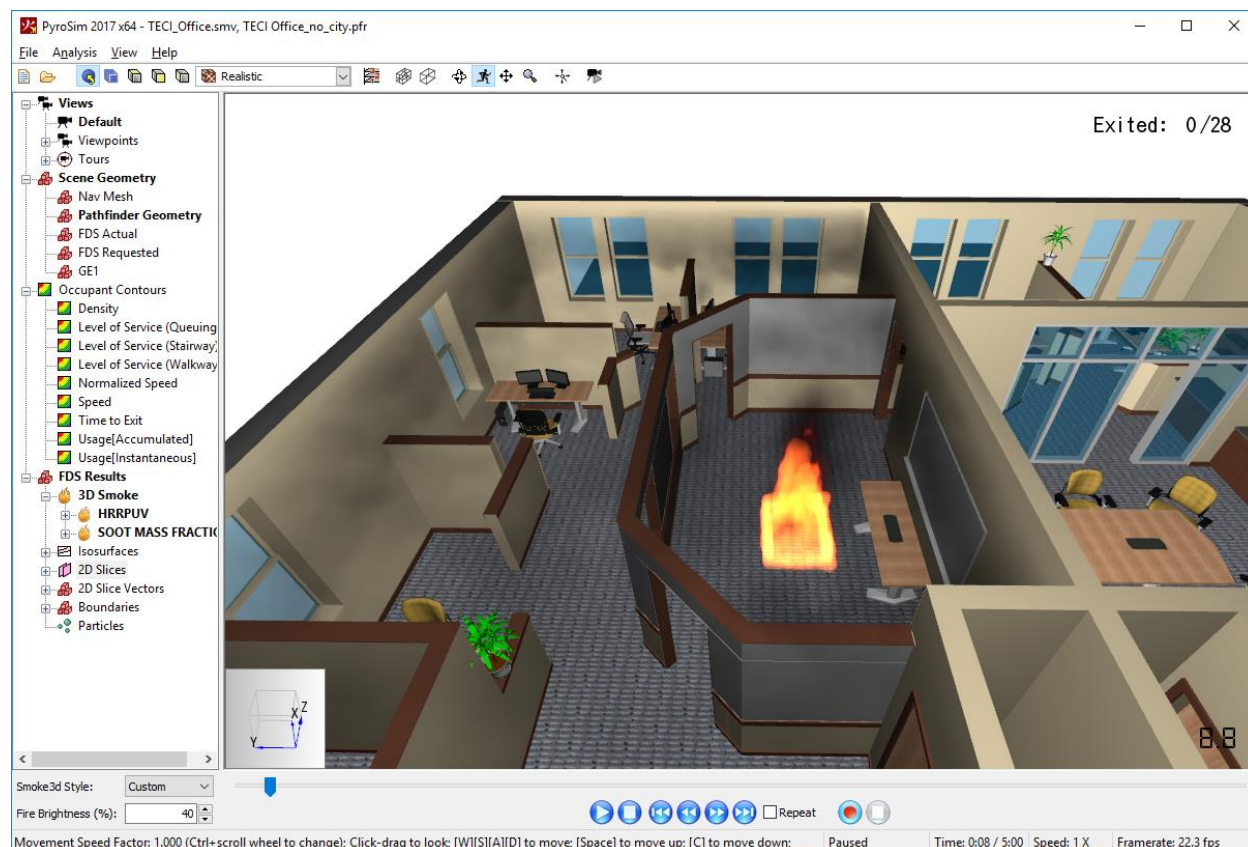


Figure 7-8: Smoke and fire

The smoke and fire visualization is merely an approximation of the actual smoke and fire. The visualization has the following limitations:

- The smoke data only simulates the absorption of light along each view ray from the camera. It does not simulate the scattering of light as would occur in actual smoke.
- The smoke is completely unlit. In actual smoke, there would be multiple lighting influences on the smoke, including ambient lighting and light from the fire and other light sources.
- The fire location, size, and colors are approximate. In actual fires, these fire properties are determined by a number of factors, including the flame temperature, black-body colors, chemical reactions of elements in the fire, etc. This information is not present, however. Instead, FDS outputs the HRRPUV of the fire. In the Results application, these HRRPUV values are used along with an HRRPUV cutoff value to determine the fire location and size. These values are also fed into a colormap based on the black-body diagram to determine the fire color.

There are a number of properties that affect the appearance of the smoke and fire. These properties can be edited by right-clicking the **3D Smoke** group in the Navigation View and selecting **Properties**. This will show the **Smoke3D Properties** dialog in Figure 7-9.

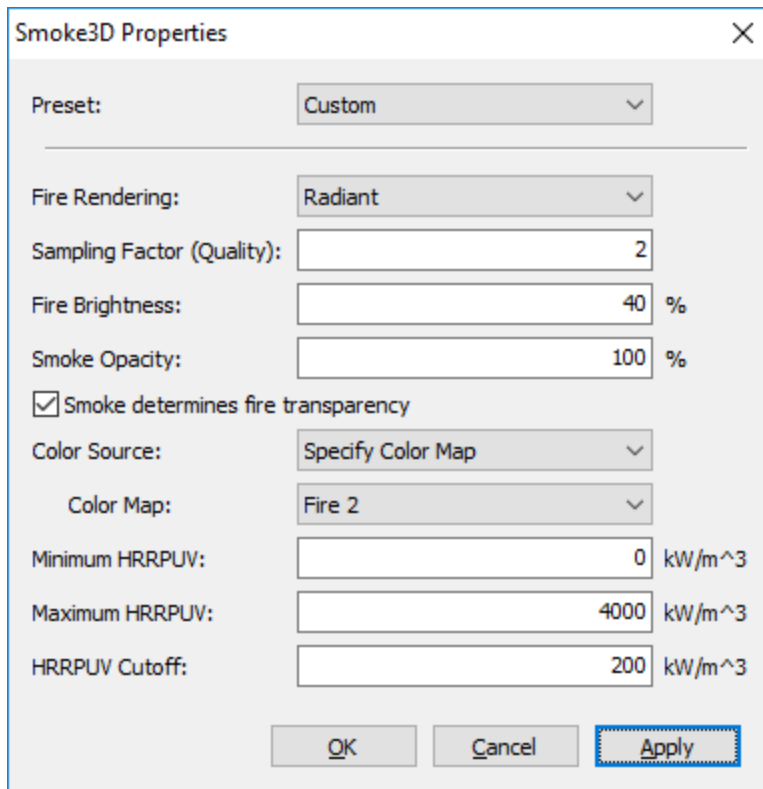


Figure 7-9: Smoke3D Properties dialog

- **Preset:** Specifies a pre-defined set of properties to be applied to the values in the dialog. The following presets are available and are demonstrated in Figure 7-10:
 - **Default:** The default preset that uses radiant rendering for the fire and only maps the HRRPUV data to a limited range to approximate a fire with a maximum temperature of about 2000° C.
 - **Smokeview Default (Pre-6.4.4):** Chooses properties that attempt to make the fire look like that in Smokeview before version 6.4.4. This uses volumetric rendering with a constant fire color.
 - **Smoke Default:** Chooses properties that attempt to make the fire look like that in Smokeview after version 6.4.4. This uses volumetric rendering with a varying fire color and maps the entire HRRPUV range to the entire fire colormap.
 - **Custom:** Allows custom values to be entered for the fire and smoke.
- **Fire Rendering:** Controls the appearance of the fire. It may be one of these values:
 - **Radiant:** Approximates real fire, where the fire itself emits light and accumulates along the view ray toward the camera, while being absorbed by smoke between the viewer and the fire. Usually when using this option, **Smoke determines fire transparency** should also be checked.
 - **Volumetric:** Treats fire similarly to smoke where it absorbs and reflects light passing through it toward the camera rather than emitting light itself. The **50% Transparency Distance** controls how much light is absorbed by the fire.

- **Sampling Factor (Quality):** This controls the rendering quality of the smoke and fire and depends on the smoke and fire rendering method. See Controlling drawing detail/speed on page 15 for more details on the available methods. Generally, the higher the value, the better the quality. If the render method is **Use Hardware Shaders**, this value controls the number of samples along the view ray used to generate each pixel color. If the render method is **Use Compatibility Renderer**, it changes the number of slices through the smoke/fire data. When using the compatibility renderer, there is an upper limit to the number of slices that should be used. Exceeding this limit may actually reduce the quality and possibly cause the fire and smoke to disappear. For more information, see the *Smokeview Technical Reference* in the section titled, *A Limitation of the Slice Solution Method* (Forney, 2017). For either render method, the distance between samples or slices is computed as the average cell size divided by the sampling factor.
- **Fire Brightness:** If **Fire Rendering** is set to **Radiant**, this controls how bright the fire appears. This is similar to adjusting the exposure on a camera, except that it only affects the light from the fire. If the fire appears almost completely white, this may be because it is either a very large fire or has little smoke. In this case, lower the brightness. If the fire can barely be seen, it may be because of very thick smoke. In this case, increase the brightness. NOTE: It is possible for the smoke to completely obscure the fire. In this case, turning up the brightness will not make a difference in the obscured regions.
- **Smoke Opacity:** Controls how opaque the smoke appears.
WARNING: Changes to this value should be carefully considered. It should only be changed for evaluation purposes to either see the effects of more or less smoke or to make it easier to see scene geometry and other results. Images and videos that show smoke with opacity other than 100% should be carefully annotated to indicate the modified opacity value.
- **Smoke determines fire transparency:** This option is only available if **Fire Rendering** is set to **Radiant** and should usually be checked in this case. If checked, the amount of light that each fire cell obscures behind it is controlled by the amount of smoke in that cell. With this option unchecked, the light absorption is controlled by the parameter, **50% Transparency Distance**.
- **50% Transparency Distance:** Controls the amount of light absorbed by fire cells. This distance sets the point at which 50% of the light will be absorbed by sequential fire cells.
- **Color Source:** Controls the source of the smoke and fire colors. Cells are colored as smoke when HRRPUV is less than **HRRPUV Cutoff**. Otherwise, cells are colored as fire. The color source value can be one of the following:
 - **Specify Color:** Allows a single color to be set each for fire and smoke.
 - **Specify Color Map:** Allows a colormap to control the color of the fire and smoke.
NOTE: The following colormaps are split into two regions. The lower half is used to color smoke and the upper half is used to color fire:
 - Blue->Red Split
 - Fire
 - Fire 2
 - Fire line (level set)
 - Split

- Cool.
 - **Fire Color:** The color of the fire when **Color Source** is set to **Specify Color**.
 - **Smoke Color:** The color of the smoke when **Color Source** is set to **Specify Color**.
 - **Color Map:** The colormap to use to color both smoke and fire.
 - **Minimum HRRPUV:** The HRRPUV value that corresponds with the lowest color in the colormap.
 - **Maximum HRRPUV:** The HRRPUV value that corresponds with the highest color in the colormap.
 - **HRRPUV Cutoff:** Controls how cells are determined to be smoke or fire cells. If the HRRPUV value is less than HRRPUV Cutoff, the cell is treated as a smoke cell and only absorbs and reflects light (reflection color is the smoke color). All other cells are treated as fire cells. If **Fire Rendering** is set to **Radiant**, these cells will absorb light and emit light as determined by the colormap.
- NOTE: For split colormaps, the HRRPUV Cutoff also determines the HRRPUV value at the center of the colormap. All values below the cutoff map linearly into the bottom half of the colormap from Minimum HRRPUV to HRRPUV Cutoff. All values above the cutoff are mapped linearly into the upper half of the colormap from HRRPUV Cutoff to Maximum HRRPUV.

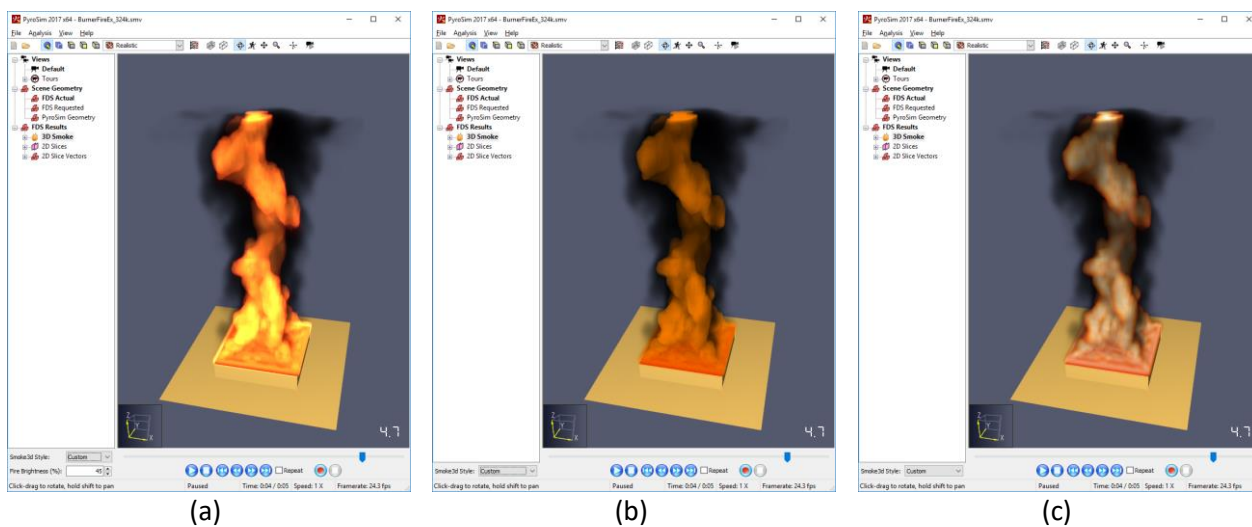


Figure 7-10: Comparison of smoke/fire property presets with smoke opacity set to 25% (a) Default (b) Smoke Default (Pre-6.4.4) (c) Smokeview Default

3D Slices

In FDS, a 3D slice represents an animated volumetric region that contains data at every intersecting cell for a single gas-phase quantity such as TEMPERATURE or SOOT DENSITY. There can be any number of 3D slices of the gas-phase quantities.

When an FDS simulation contains 3D slices, they will appear under the **3D Slices** group in the Navigation View. The hierarchy appears in the Navigation View as **FDS Results->3D Slices->Quantity->Mesh**.

For more information on the visualization capabilities of 3D slices, refer to Data3d on page 52.

Plot3d

FDS has the capability of creating Plot3d files. Like 3D slices, a Plot3d file contains 3D data sets. There are a few key differences between the file format for Plot3d and 3D slices:

- The Plot3d file format is a known format compatible with other applications. 3D slices are specific to FDS.
- FDS Plot3d files always contain information for exactly five quantities, even if fewer than five are specified in the FDS input file. 3D slices only have one quantity and only output exactly the quantities requested in the input file.
- Plot3d files do not contain animation information. Instead, each Plot3d file contains data for exactly one time step. 3D slice files contain animated data for all time steps.
- Plot3d files contain data for an entire mesh. 3D slices can be a subset of a mesh.

Despite the differences, the Results application hides these details from the user and presents Plot3d output exactly like 3D slices, in that each quantity is represented as a separate object in the Navigation View. In addition, the Results application will treat all of the Plot3d files of a given quantity as if they were one animated file.

When an FDS simulation contains Plot3d data, they will appear under the in the Navigation View with this hierarchy: **FDS Results->Plot3d->Quantity->Mesh**.

For more information on the visualization capabilities of Plot3d, refer to Data3d below.

Data3d

Both Plot3d and 3D slice files contain 3D datasets (Data3d) of gas-phase quantities. Though they require fairly significant storage space, Data3d are perhaps the most flexible output formats from FDS and can be visualized in several ways. They can be viewed directly using volumetric rendering, and arbitrary **isosurfaces**, **slices**, and **points** be created from them in the Results application without being required to re-run the FDS simulation.

Data3d Volumetric Rendering

Volumetric rendering of a Data3d object enables viewing it as a 3D form as if it was a cloud of colored gas as shown in Figure 7-11. In order to understand the shape and depth of the volume, portions of the dataset must be made translucent.

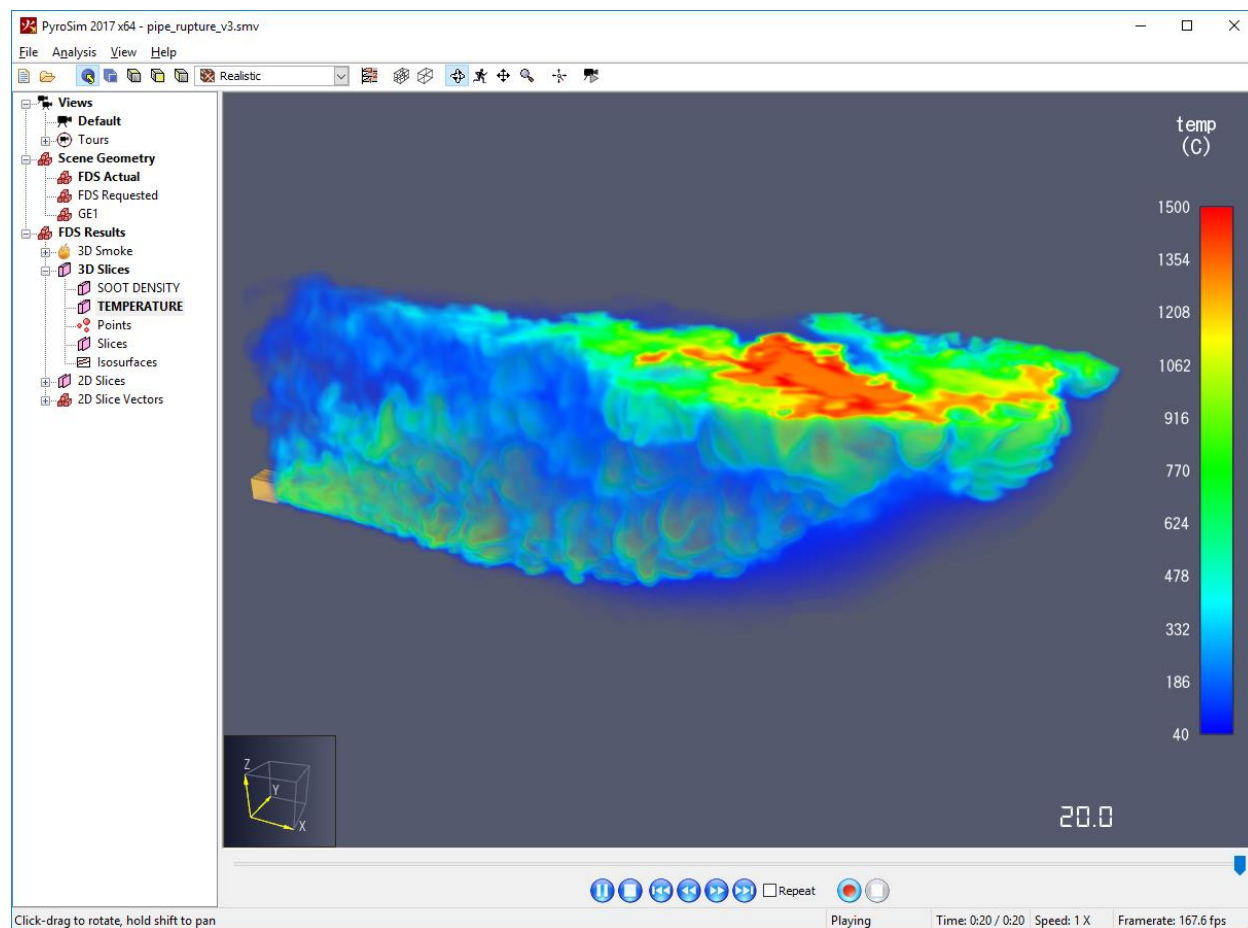


Figure 7-11 Volumetric rendering of a temperature 3D Slice

By default, the minimum value in the data range (see Global FDS Properties on page 44) has an opacity of 0% making the lower values completely transparent. The maximum value in the data range has an opacity of 75%. The range of opacity values can be changed by either choosing the constant opacity value or by choosing a **Gradient Opacity Source** in the FDS Preferences dialog and setting the desired opacity mapping.

NOTE: Even if a constant opacity value is chosen, the Results application will turn this into a gradient opacity mapping from 0% for the minimum data value to the specified opacity value for the maximum data value.

Volumetric renderings are created similarly to fire and smoke. To change their quality, in the Navigation View expand either **3D Slices** or **Plot3d**, right-click one of the quantities, and select **Properties**. This will open the **Data3d Properties** dialog as shown in Figure 7-12.

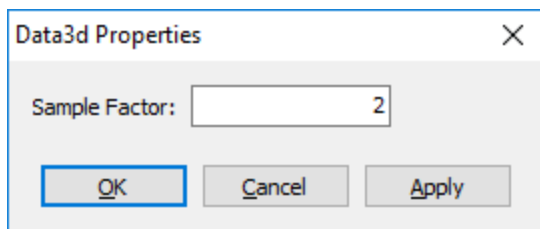


Figure 7-12: Data3d Properties dialog

When using hardware shaders, the **Sample Factor** controls the number of samples taken from the data set for each view ray. When using the compatibility renderer, the **Sample Factor** controls the number of slices through the 3d texture. Increasing this value will generally increase the quality of the volumetric rendering at the expense of rendering speed.

Data3d Points

From Data3d objects, any number of points can be created that are colored according to the colorbar and Data3d values as shown in Figure 7-13.

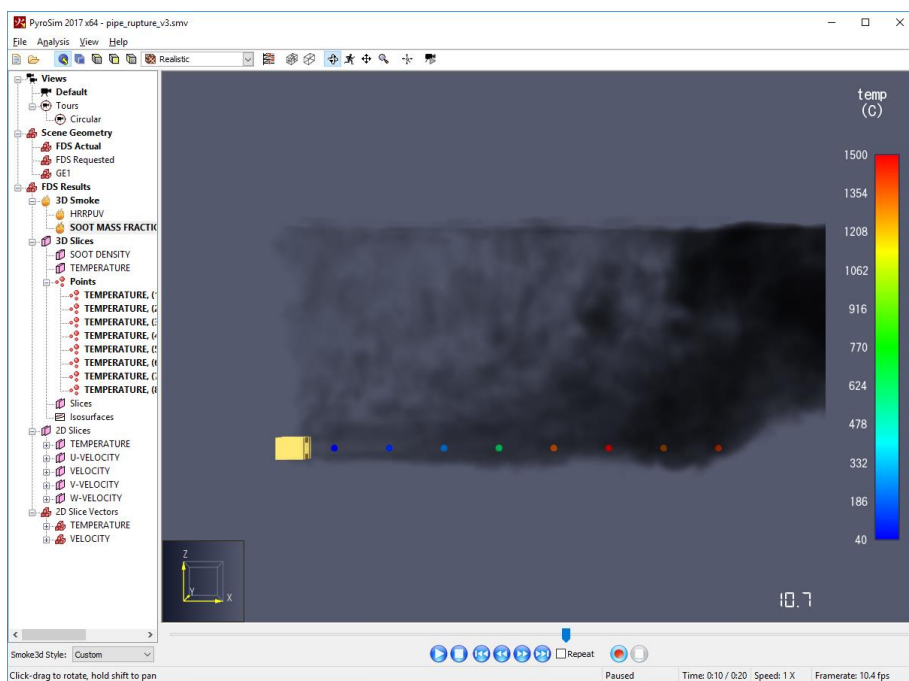


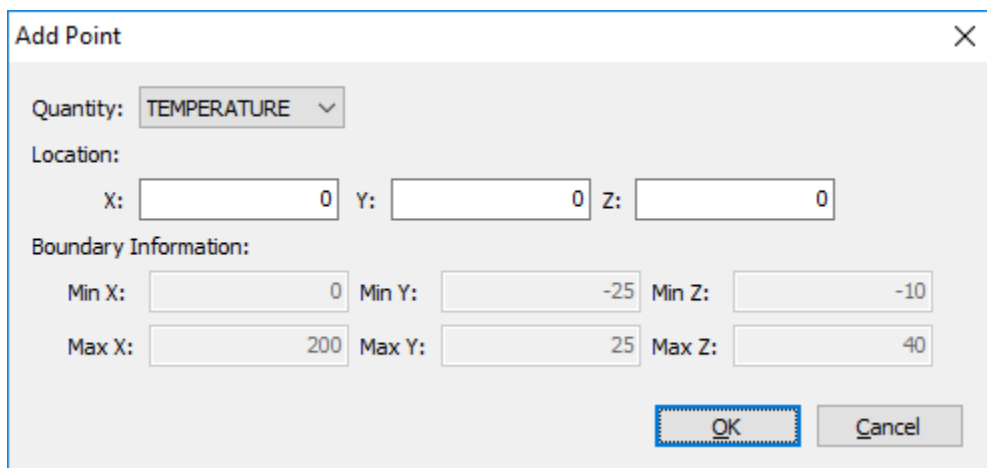
Figure 7-13: Data3d points

To create a Data3d point:

1. In the Navigation View, expand the appropriate Data3d group (**3D Slices** or **Plot3d**).
2. Right-click either **Points** or one of the quantities.
3. Select **Add point**. The **Add Point** dialog will open as shown in Figure 7-14.

- Choose the desired quantity and location of the point and click OK. The point will be added under the **Points** group.

NOTE: The point must be located inside one of Data3d regions for the given quantity.



The 'Add Point' dialog box is shown with the following fields:

- Quantity:** A dropdown menu set to 'TEMPERATURE'.
- Location:** Three input fields for X, Y, and Z coordinates, all set to 0.
- Boundary Information:** Six input fields for Min X, Min Y, Min Z, Max X, Max Y, and Max Z. The values are: Min X: 0, Min Y: -25, Min Z: -10, Max X: 200, Max Y: 25, Max Z: 40.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom right.

Figure 7-14: Add Point dialog

To delete a Data3d point, select it in the Navigation View and press the DELETE key.

Data3d Slices

Arbitrary 2D slices can also be created from Data3d objects without rerunning the simulation. These slices do not have the same restrictions as regular FDS slices in that they do not have to be axis-aligned. Instead, they can be in any plane. Figure 7-15 shows a 2D slice created from a 3D slice.

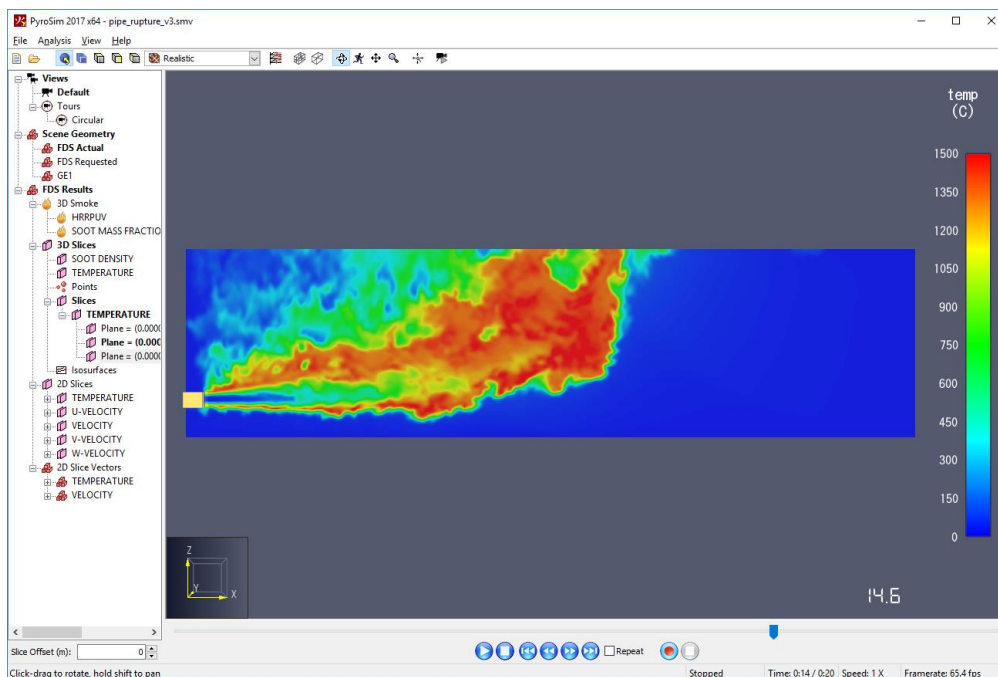
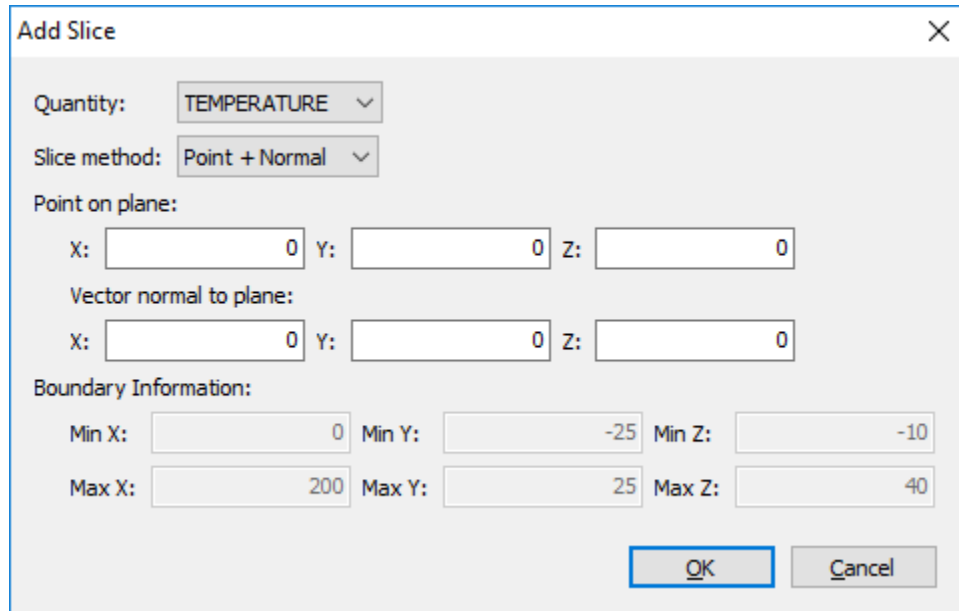


Figure 7-15: 2D slice from a 3D slice

To add a Data3d slice:

1. In the Navigation View, expand the appropriate Data3d group (**3D Slices** or **Plot3d**).
2. Right-click either **Slices** or one of the quantities.
3. Select **Add slice**. The **Add Slice** dialog will open as shown in Figure 7-16.
4. Choose the desired quantity and enter the slice information and click OK. The slice will be added under the **Slices** group.

When Data3d slices are active, they can be quickly moved along their normal in the Quick Settings.

The image shows a software dialog box titled "Add Slice" with a close button (X) in the top right corner. The dialog contains several input fields and dropdown menus. At the top, there is a "Quantity:" dropdown menu set to "TEMPERATURE". Below it is a "Slice method:" dropdown menu set to "Point + Normal". The "Point on plane:" section has three input fields for X, Y, and Z, all set to 0. The "Vector normal to plane:" section also has three input fields for X, Y, and Z, all set to 0. The "Boundary Information:" section contains six input fields: "Min X:" (0), "Min Y:" (-25), "Min Z:" (-10), "Max X:" (200), "Max Y:" (25), and "Max Z:" (40). At the bottom right, there are two buttons: "OK" and "Cancel".

Add Slice

Quantity:

Slice method:

Point on plane:

X: Y: Z:

Vector normal to plane:

X: Y: Z:

Boundary Information:

Min X: Min Y: Min Z:

Max X: Max Y: Max Z:

Figure 7-16: Add Slice dialog

To delete a Data3d slice, select it in the Navigation View and press the DELETE key.

Data3d Isosurfaces

Arbitrary 3D isosurfaces can also be created from Data3d objects without rerunning the simulation. An isosurface shows the contour of a gas-phase quantity at a specific value. Some isosurfaces created from a 3D slice are shown in Figure 7-17.

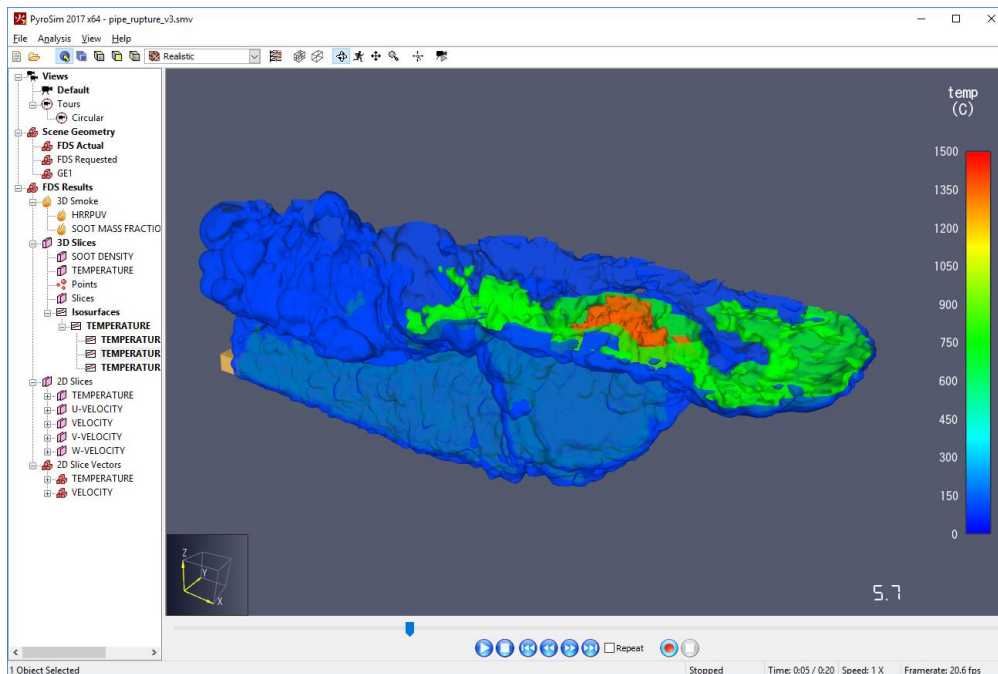


Figure 7-17: Data3d isosurfaces

To add a Data3d isosurface:

1. In the Navigation View, expand the appropriate Data3d group (**3D Slices** or **Plot3d**).
2. Right-click either **Isosurfaces** or one of the quantities.
3. Select **Add isosurface**. The **Add Isosurface** dialog will open as shown in Figure 7-18.
4. Choose the desired quantity and enter the isosurfaces value. The isosurface will be added under the **Isosurfaces** group.

If only one Data3d isosurfaces is active, its value can be quickly changed in the Quick Settings.

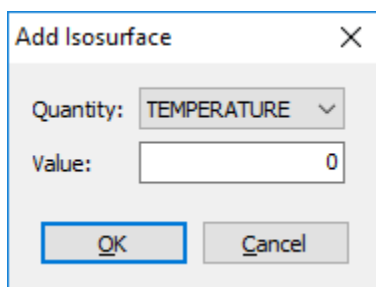


Figure 7-18 Add Isosurface dialog

To delete a Data3d isosurface, select it in the Navigation View and press the DELETE key.

Isosurfaces

FDS supports the generation of animated 3D isosurfaces. An isosurface shows the contour of a gas-phase quantity at a specific value, as shown in Figure 7-19.

Unlike Data3d isosurfaces, FDS isosurfaces cannot be added or changed in the Results application. Instead, they must be specified in the FDS input file or in PyroSim.

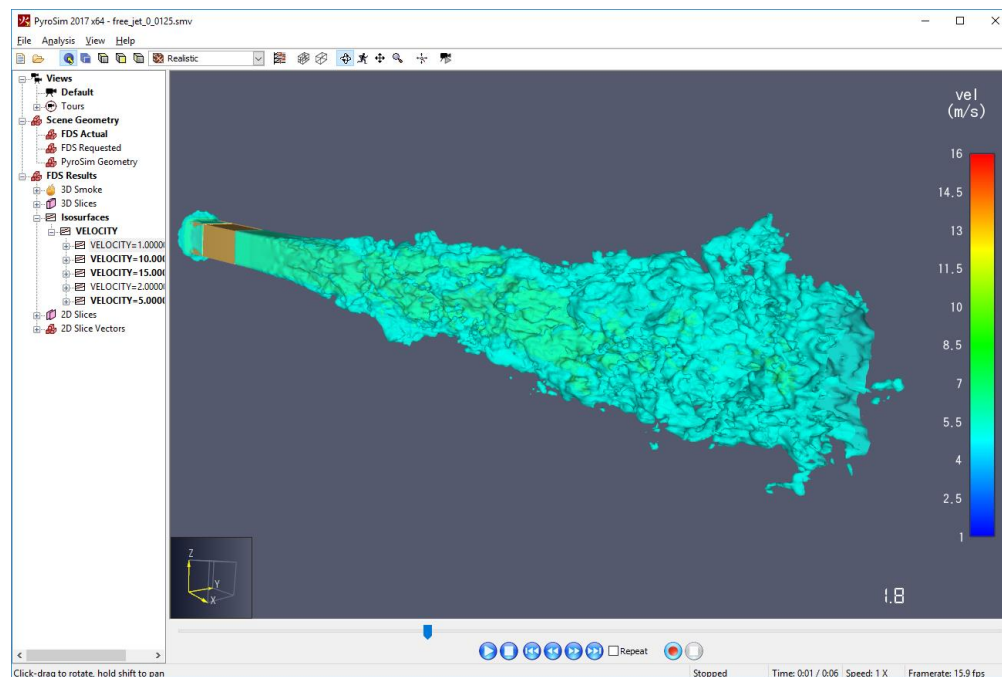


Figure 7-19: FDS Isosurfaces

When an FDS simulation contains isosurfaces, they will appear under the **Isosurface** group in the Navigation View. The hierarchy appears in the Navigation View as **FDS Results->Isosurfaces ->Quantity->Value->Mesh**.

2D Slices

FDS also supports the generation of animated 2D slices. A 2D slice shows the values of a gas-phase quantity in a single plane as shown in Figure 7-20.

Unlike Data3d slices, FDS 2D slices cannot be added or changed in the Results application. Instead, they must be specified in the FDS input file or in PyroSim.

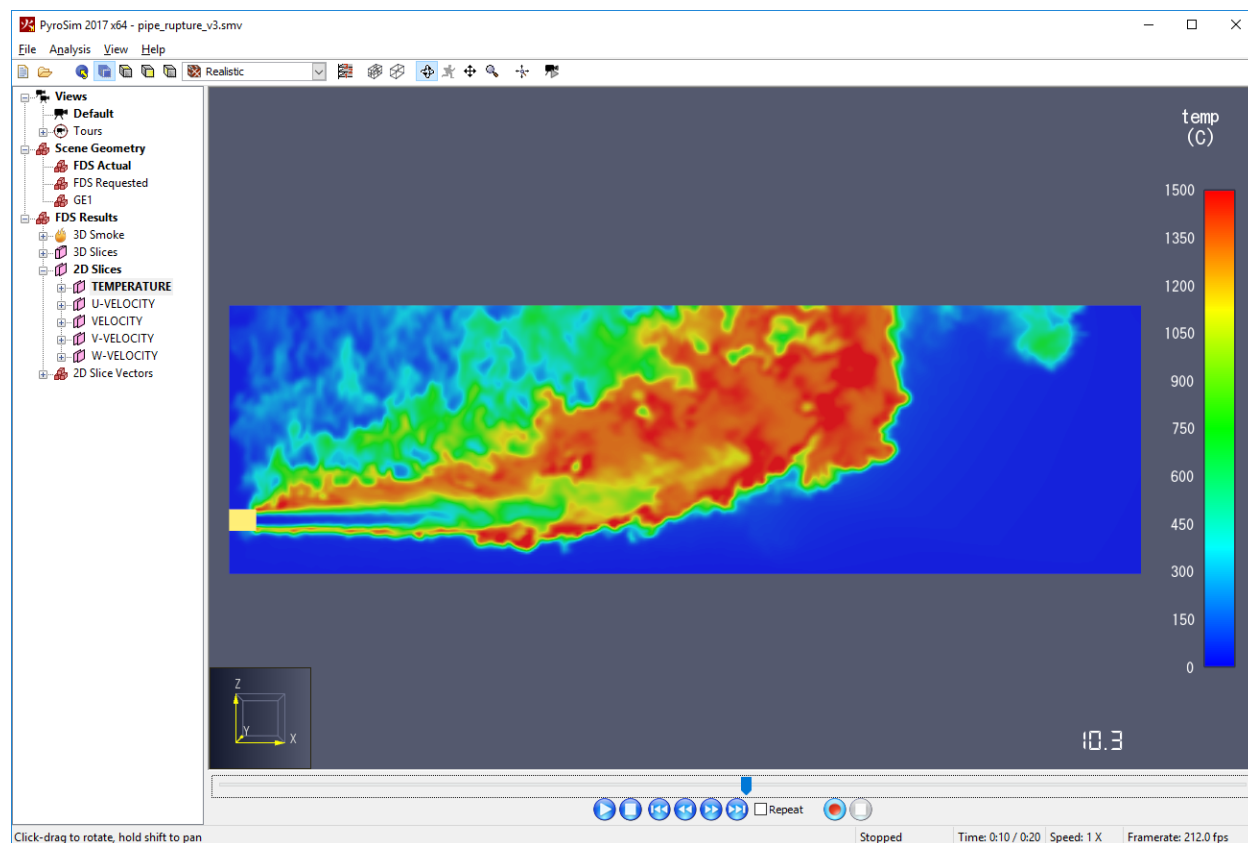


Figure 7-20: FDS 2D Slice

When an FDS simulation contains 2D slices, they will appear under the **2D Slices** group in the Navigation View. The hierarchy appears in the Navigation View as **FDS Results->2D Slices ->Quantity->Slice Location->Mesh**.

2D Slice Vectors

In FDS, 2D slices may optionally be created with the VECTORS option. When this option is enabled, FDS creates three additional 2D slice files containing the U,V,W coordinates of the air velocity. The Results application can combine these additional slice results to create 2D Slice Vectors.

2D Slice Vectors show the velocity of the air as vectors that point in the direction of the air flow and are scaled by the air speed. The velocity and color of each vector is determined at the center of the vector. Figure 7-21 shows 2D slice vectors that are colored by TEMPERATURE.

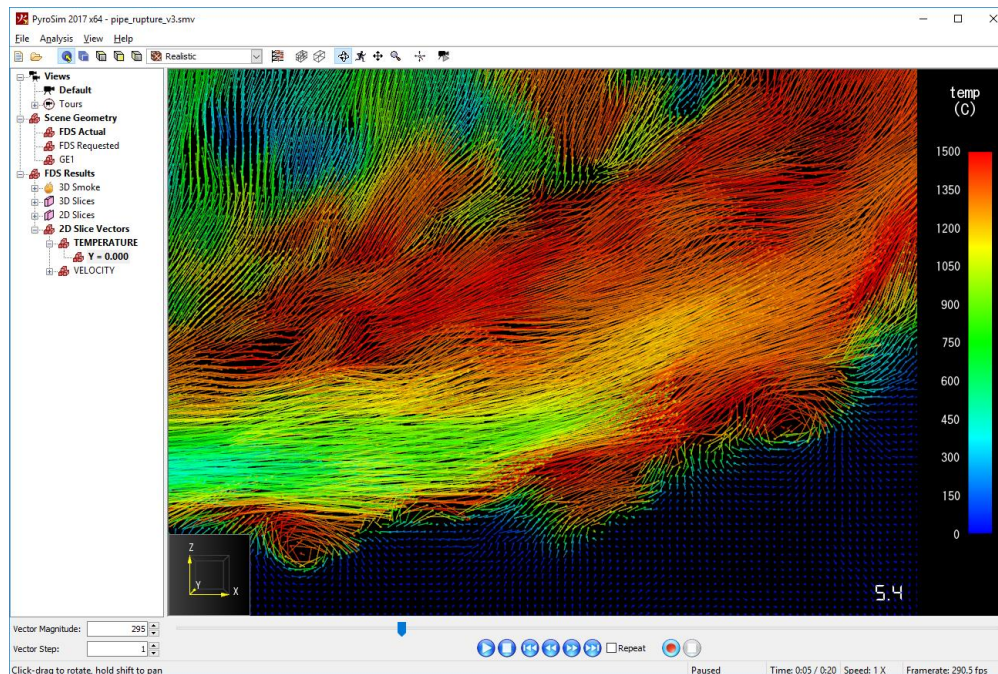


Figure 7-21: FDS 2D Slice Vectors

When an FDS simulation contains 2D slice vectors, they will appear under the **2D Slice Vectors** group in the Navigation View. The hierarchy appears in the Navigation View as **FDS Results->2D Slice Vectors - >Quantity->Slice Location->Mesh**.

To change the magnitude or density of the slice vectors, in the Navigation View, right-click the desired 2D slice vectors and click **Properties**. The **Slice Vector Properties** dialog will open as shown in Figure 7-22.

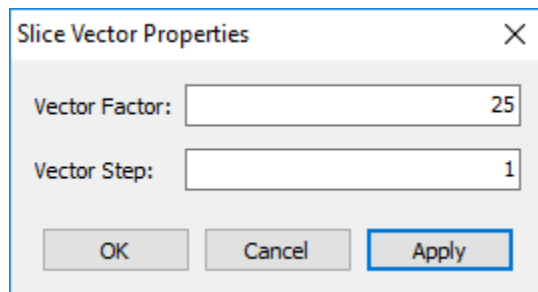


Figure 7-22: FDS Slice Vector Properties Dialog

- **Vector Factor:** Controls the length of the vectors. Larger values produce longer vectors.
- **Vector Step:** Controls how many vectors are drawn. Larger values produce fewer vectors.

These options are also available in the Quick Settings when 2D slice vectors are active.

Boundaries

Boundaries show solid-phase and some gas-phase data at the boundaries of solid surfaces as shown in Figure 7-23.

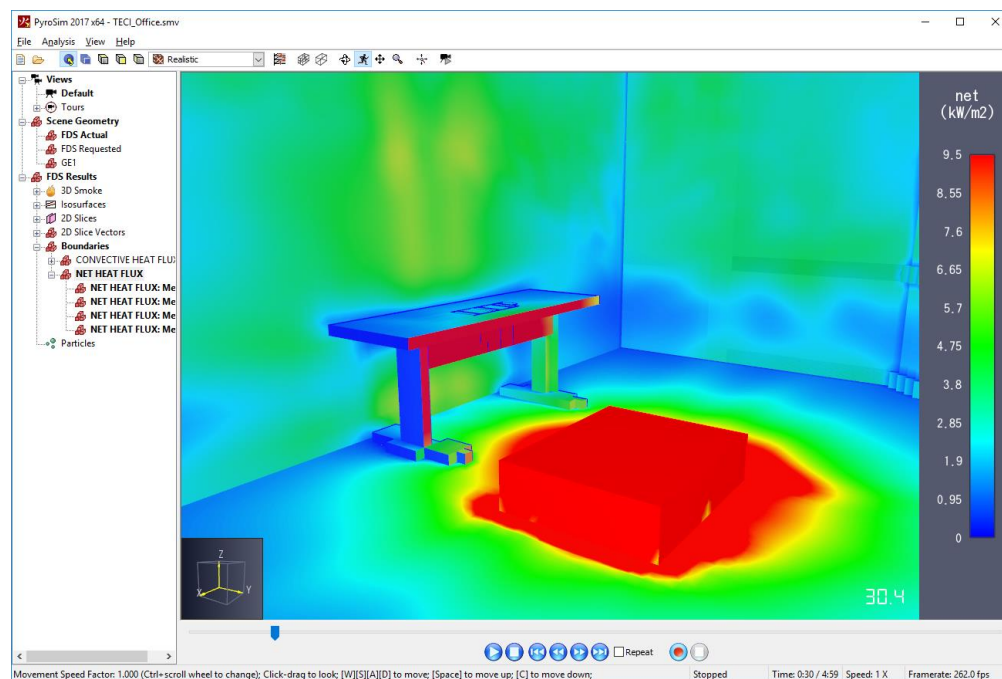


Figure 7-23: FDS Boundaries

When an FDS simulation contains boundaries, they will appear under the **Boundaries** group in the Navigation View. The hierarchy appears in the Navigation View as **FDS Results->Boundaries ->Quantity->Mesh**.

The colors of the boundaries are blended with those of the active geometry set. The amount of blending can be controlled with the **Opacity** setting in the **FDS Preferences** dialog (see Global FDS Properties on page 44). The default opacity setting of 75% causes 75% of the color to come from the boundary and 25% from the geometry set. Setting this value to 100% would disable blending and only show the boundary colors. In addition, a gradient opacity function can be used to highlight specific boundary values as shown in Figure 7-24.

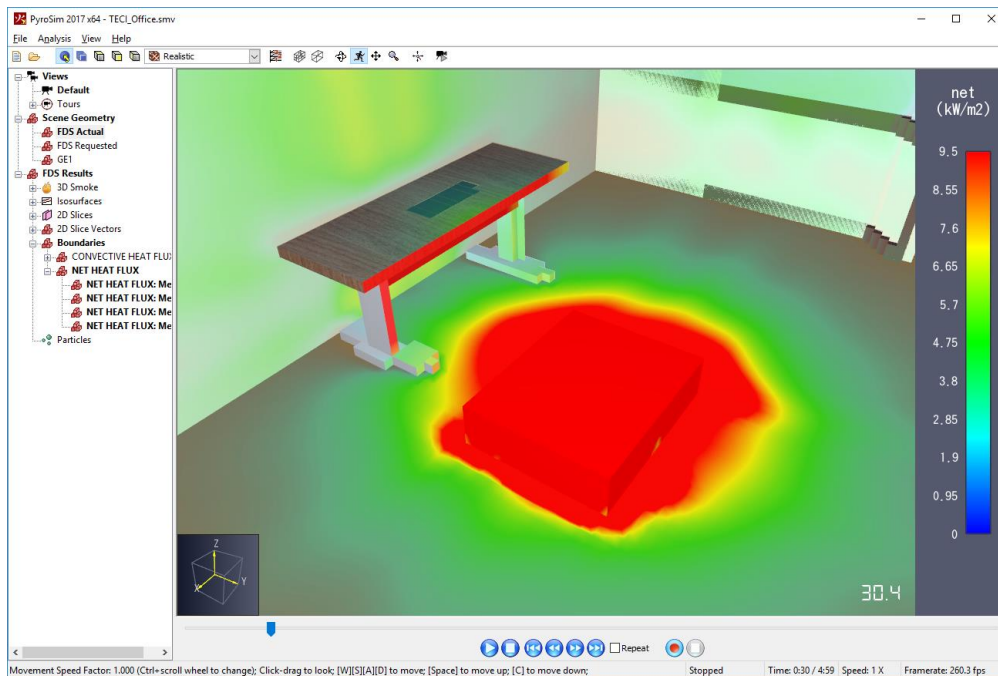


Figure 7-24: Boundary color mixed with scene color using a gradient opacity function

NOTE: Unlike other FDS output, boundaries should almost always be viewed along with the **FDS Actual** geometry set (see Displaying geometry input on page 9). If viewed in combination with other geometry sets, the boundaries might not properly align with the geometry, creating distracting display artifacts.

Realistic Visualization of Soot Deposition

FDS supports the simulation of aerosol deposition, which can be used to track the deposition of various species on the boundaries of obstructions, such as soot gathering on walls and ceilings. FDS stores the output information in boundary files. By setting specific color parameters, the Results application can visualize soot deposition fairly realistically as shown in Figure 7-25.

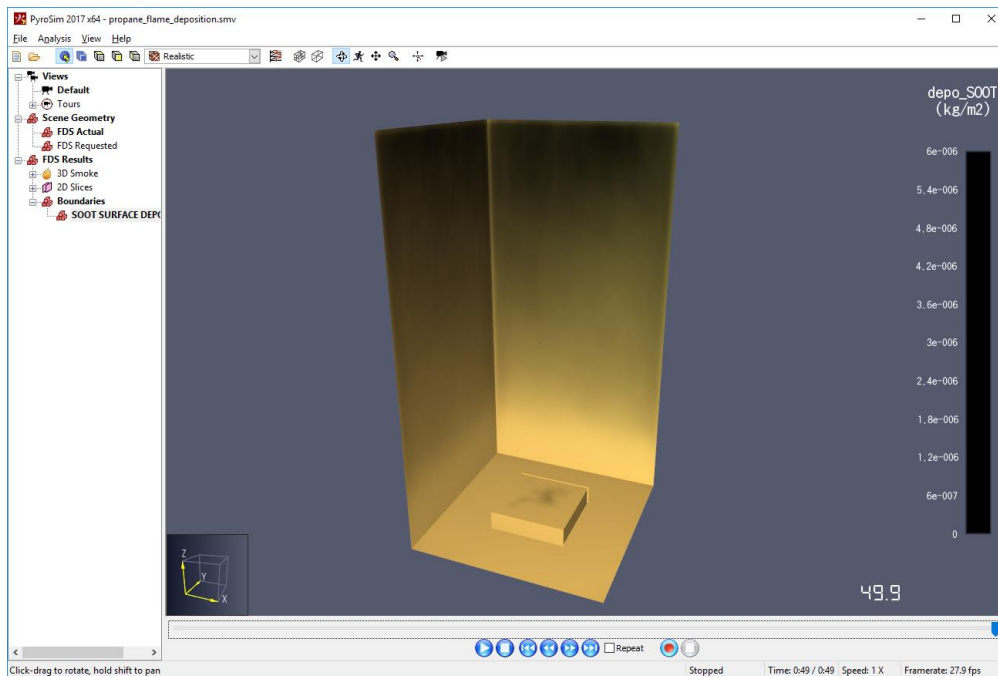


Figure 7-25: Realistic soot deposition

To recreate this visualization, perform the following:

1. In the Navigation View, under the **FDS Results->Boundaries** group, activate **SOOT SURFACE DEPOSITION**.
2. In the **Analysis** menu, select **FDS Preferences** to open the **FDS Preferences** dialog.
3. Set **Color Source** to **Constant Color**.
4. Set **Color** to the soot color (e.g. black).
5. Set **Opacity Source** to **Gradient**.
6. Set **Opacity at Minimum** to 0% and **Opacity at Maximum** to 100%.
7. Adjust the **Maximum Value** of the data range to be the value required for the soot to look completely opaque on a surface assuming complete coverage of the surface.
8. Click **OK**.

Particles

FDS supports the injection of particles in a simulation. Particles can be used to trace air flow or to represent liquids, such as water or fuel. There can be several classes of particles in one simulation. Figure 7-26 shows tracer particles flowing through the model.

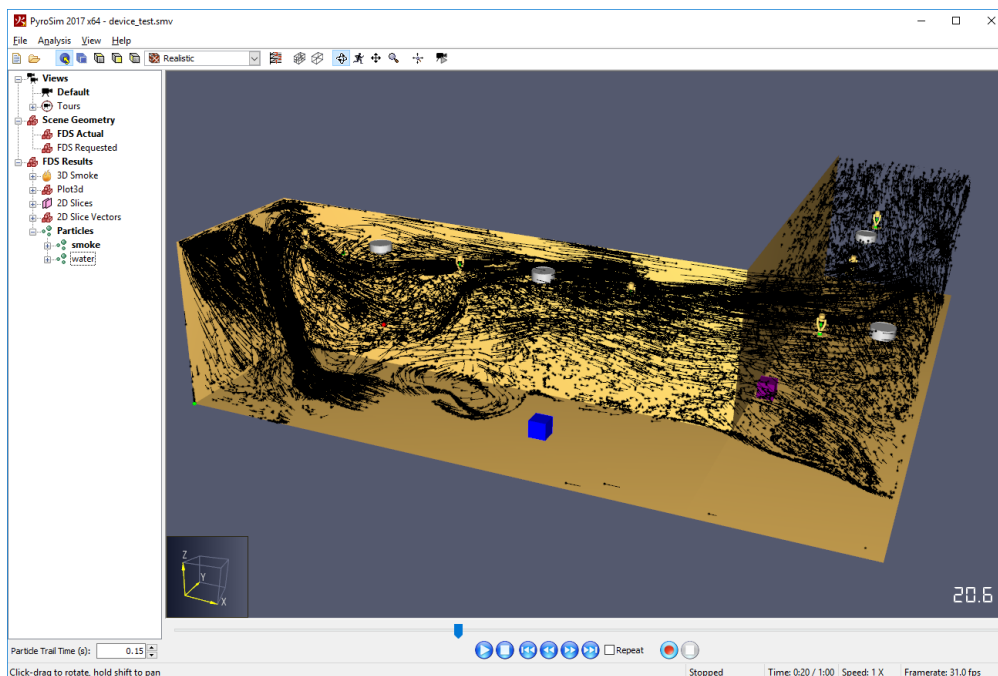


Figure 7-26: FDS Particles with trails

When an FDS simulation contains particles, they are found in the Navigation View and are organized as follows: **FDS Results->Particles->Particle Class->Mesh**.

By default, particles appear as dots and are a single color, but they may also be visualized with trails that show the particle motion. In addition, they may be colored by various particle quantities, if enabled in PyroSim or the FDS input file. To change the particle properties, in the Navigation View, right-click the desired particle class and select **Properties**. This will open the **Particle Property Dialog** as shown in Figure 7-27.

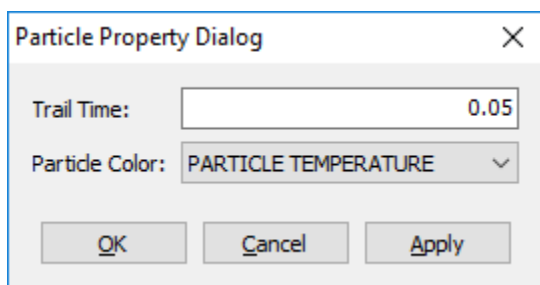


Figure 7-27: Particle Property Dialog

- **Trail Time:** Controls the length of the particle trails by setting how far back in time in seconds to trace the path of the particle. If this value is zero, trails are turned off.
NOTE: Large values may significantly lower rendering performance and increase system memory usage.

- **Particle Color:** Controls the color of the particles. <Default> will color all the particles with a uniform color as specified in PyroSim or the FDS input file. Other values will cause a colorbar to appear and will color the particles individually according to their values. Figure 7-28 shows particles colored by their temperature.

NOTE: Choosing a **Particle Color** other than <Default> will cause other active FDS results to deactivate if their quantity is incompatible with the selected particle quantity. If there are active particle classes that do not support the chosen quantity, they will remain active but they will be hidden.

Both the **Trail Time** and **Particle Color** may also be changed in the Quick Settings for the active particles.

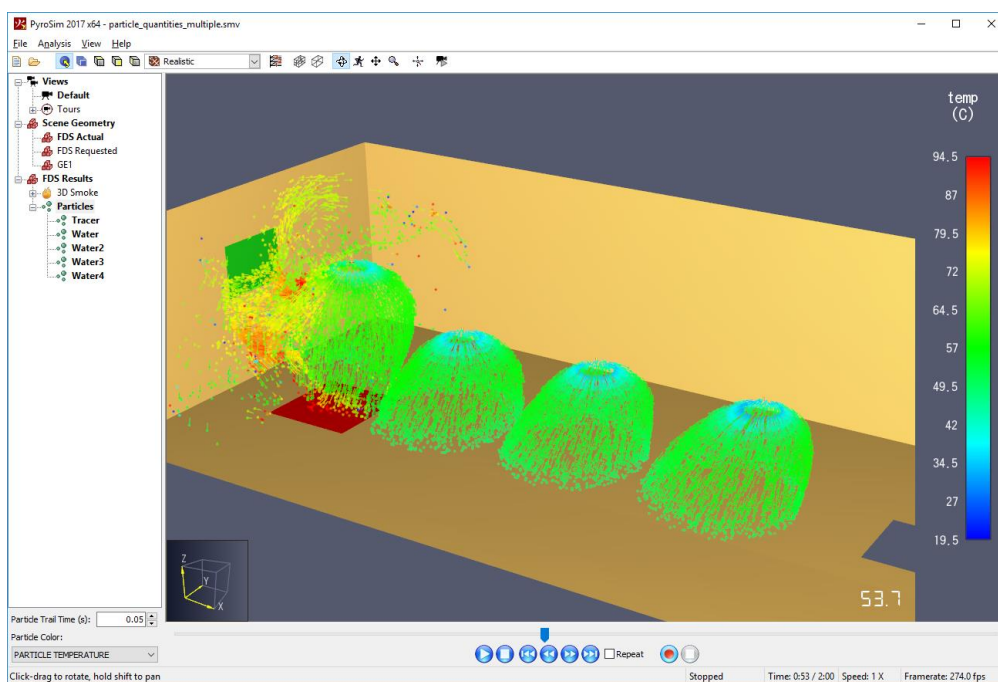


Figure 7-28: Particles colored by PARTICLE TEMPERATURE

8. Viewing Pathfinder Results

Displaying occupants

Occupants can be displayed in several ways. They can be shown as simple shapes, as realistic people, or as the artist's wood mannequin as shown in Figure 8-1. All display options are found under the **View->Occupant Display** menu. Occupants can be displayed as follows:

- **Show as Disks:** Represents the occupants as flattened disks. The size of each disk represents the collision size of the occupant. A triangle points in the direction in which the occupant is facing.
- **Show as Cylinders:** Represents the occupants as cylinders that represent their collision volume. A triangle on top of each cylinder points in the direction in which the occupant is facing.
- **Show as Generic:** Represents the occupants as animated mannequins.
- **Show as People:** Represents the occupants as realistic animated people. The avatar selected in Pathfinder determines which avatar is shown in the Results application.

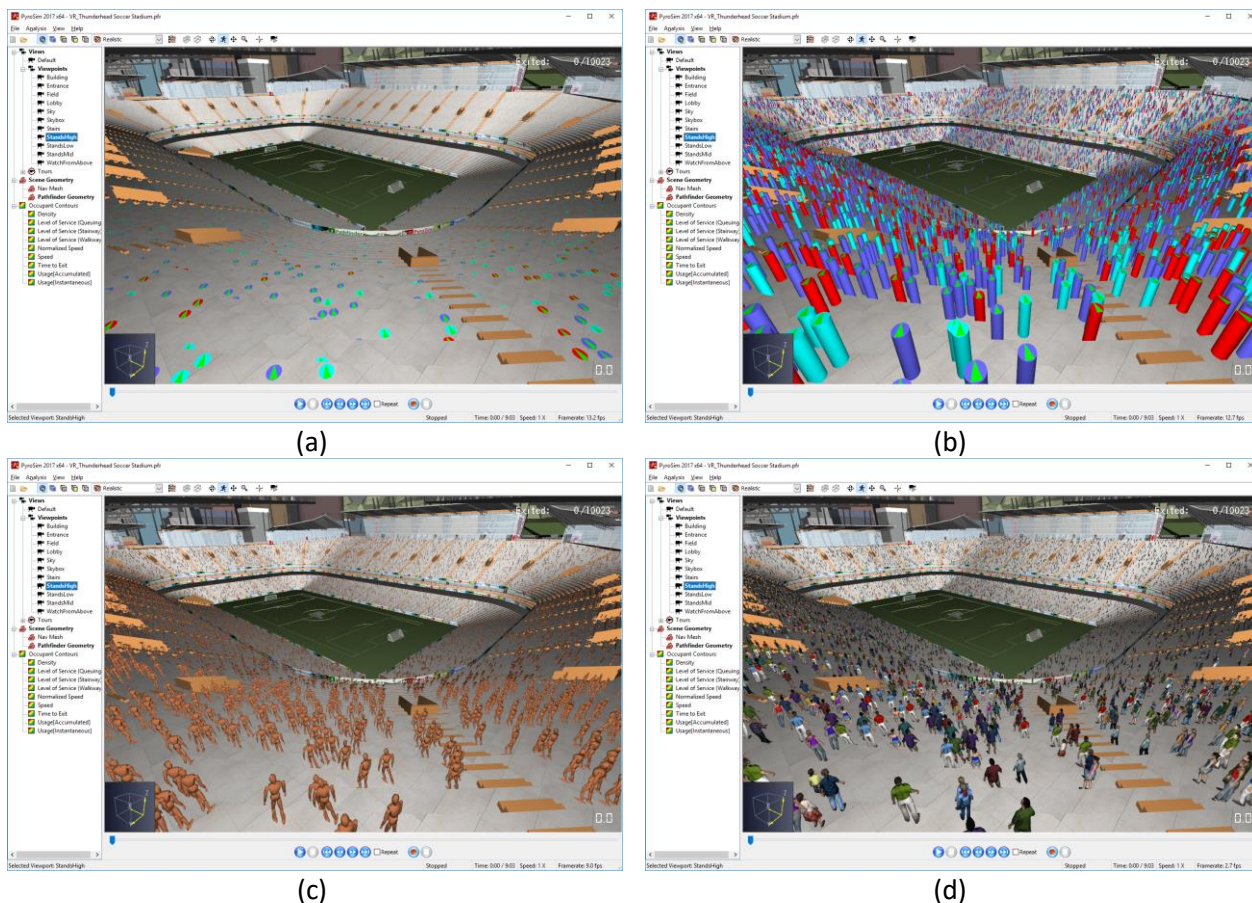


Figure 8-1: Occupant Display options (a) Disks (b) Cylinders (c) Mannequins (d) People

The color of the occupants depends on the occupant display. When shown as generic or people, the occupants will be textured according to their avatars. When shown as disks or cylinders, the occupants

will be colored according to the current **Occupant Color** option. The occupant color options are found in the **View->Occupant Color** menu. The following options are available:

- **Default:** If an occupant has an individually-set color, the occupant is colored with this; otherwise, the occupant's profile color is used.
- **By Movement Group:** If the occupant is part of a movement group, this colors all the occupants in their group the same color. The group color is set in the properties panel for the selected movement group. If the occupant is not in a movement group, their default color is used.
- **By Movement Group Template:** If the occupant is part of a movement group that was created from a movement group template, this option colors the occupant using the color of the movement group template; otherwise, their default color is used.
- **By Behavior:** Colors the occupant according to the occupant's behavior color as set in Pathfinder.
- **By Profile:** Colors the occupant according to the occupant's profile color as set in Pathfinder.
- **By View Direction:** Colors the occupant according to their view direction at the current playback time. Yellow is North, green is East, blue is South, red is West. This is useful for identifying counterflow and crossflow.
- **By Speed:** Colors the occupant according to their speed at the current playback time. The color map uses a fixed range of 0 m/s to 1.5 m/s.
- **By Normalized Speed:** Colors the occupant according to their normalized speed at the current playback time. The normalized speed is their current speed/max speed. This is useful for determining how well occupants are traveling in populations of varied maximum occupant speeds.
- **By Time to Exit:** Colors the occupant according to their remaining time until they leave the simulation. The value changes depending on the current playback time. The color map uses a fixed range of 0 s to 200 s.

Selecting Occupants

Occupants can be selected for tracking purposes by clicking on the occupant. When selected, the occupant and their path (if displayed) will turn yellow. Multiple occupants can be selected by holding CTRL when clicking them. They can be de-selected by clicking anywhere else in the model.

Viewing occupant paths

Occupant paths may be enabled to help visualize flow in an evacuation as shown in Figure 8-2. They can be shown by checking **Show Occupant Paths** in the **View** menu. When turned on, each occupant's path will be drawn from the time they entered the simulation to the current playing time.

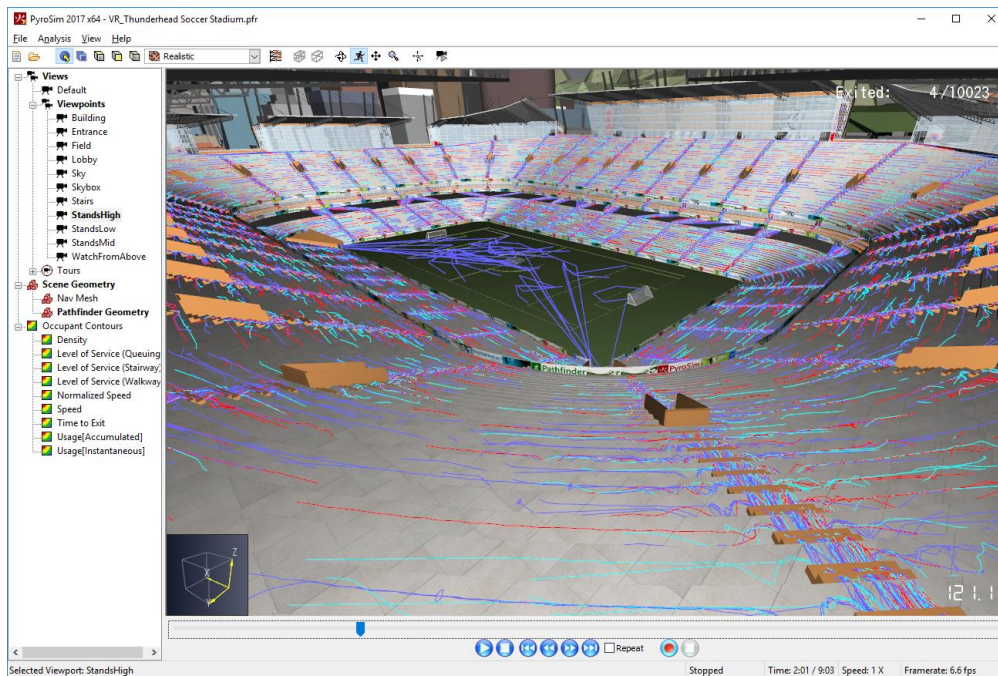


Figure 8-2: Occupant paths

Occupant Contours/Heat Maps

Pathfinder supports the visualization of dynamic occupant data on the floor areas of the model as shown in Figure 8-3. Referred to as occupant contours or heat maps, this animated data can give a qualitative visualization of the model's performance.

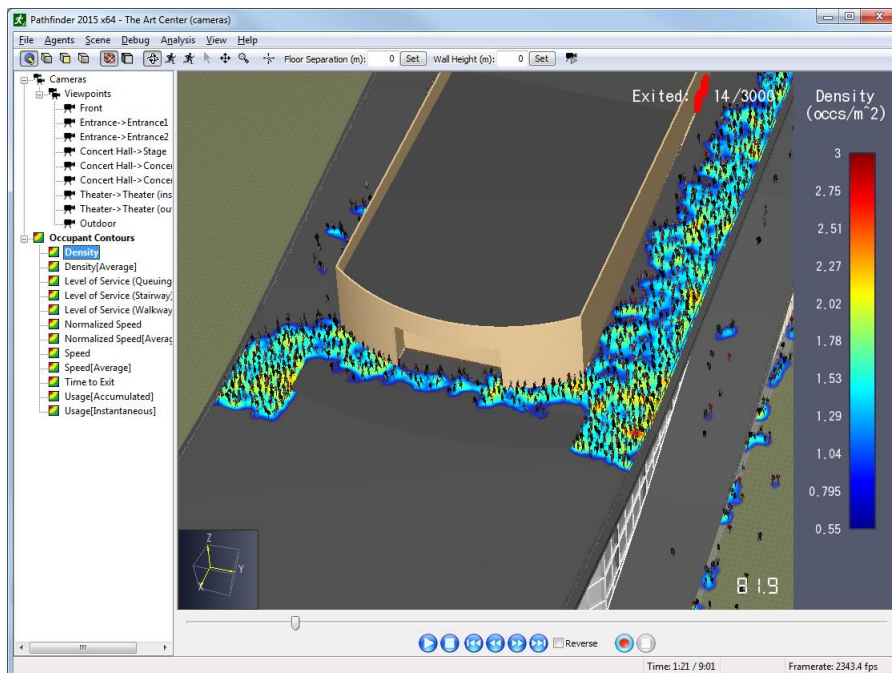


Figure 8-3: Occupant Contours

Pathfinder supports the visualization of several types of occupant contours, including the following:

- **Density:** Shows how densely packed occupants are in occupants/m².
- **Level of Service:** Shows Level of Service (Fruin, 1987) defined by surface type, including stairways, walkways, and queuing areas.
- **Speed:** Shows occupant speed.
- **Normalized Speed:** Shows occupant speed normalized against each occupant's maximum speed.
- **Time to Exit:** Shows the amount of time it will take an occupant to exit from each point on the floor given the current playback time.

NOTE: There will only be data from occupants that have a simulated exit time. If this quantity is viewed before a simulation is complete, the results may be incomplete. The contour will update, however, when results are refreshed.

- **Usage [Instantaneous]:** Shows where occupants currently exist on the floor. This can be used as an alternative to viewing individual occupants, which is useful when viewing large models from afar.
- **Usage [Accumulated]:** Shows a time-integrated version of **Usage**, which allows the user to see how much time is spent on each area of the floor.

In addition to the basic contours, Pathfinder also provides the ability to filter any contour as follows:


- **Average:** Averages the values of the base contour over a trailing time interval (default = 30 s). This can be useful for smoothing data over time and to act as a low-pass filter.
- **Maximum:** Calculates the maximum contour value over a trailing time interval (default = unlimited).

The ability to filter any contour gives the user a great deal of flexibility in visualizing contours. For instance, a contour could be created that displays the maximum average density. The averaging filter would act as a low-pass filter that reduces high-density peaks that may be short lived. The maximum filter would then be used to find the maximum of this smoothed data. This would be created by adding an average filter to the density contour and then adding a maximum filter to this averaging filter.

Adding/Removing Contours

By default, a results file will contain several contours. More can be added or removed at any time.

Adding a contour can be done in one of two ways:

- On the **Analysis** menu, select **New occupant contour...** () or
- Right-click the "Occupant Contours" group in the navigation view and select, **New occupant contour....**

Performing either of these actions will open the **New Occupant Contour** dialog. In this dialog, select the desired contour quantity, and click **OK**. This will create the new contour and show its properties dialog as discussed below.

Contours can be removed at any time by selecting the desired contour in the navigation view, and either pressing DELETE on the keyboard or selecting **Delete...** (✖) from the right-click menu.

Adding Contour Filters

Contour filters can be added at any time in one of two ways:

- On the **Analysis** menu, select **New contour filter...** (🎨) or
- In the navigation menu, right-click the contour to filter and select **Add filter...**

This will show the **New Filtered Occupant Contour** dialog as shown in Figure 8-4. Select the desired properties and press **OK** to create the filter and show its properties dialog.

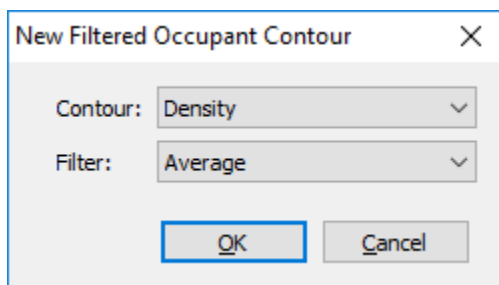


Figure 8-4: New Filtered Occupant Contour dialog

- **Contour:** The contour that will be filtered (any changes to this contour will also affect the filtered contour).
- **Filter:** The filter to apply to the contour.

Importing Contours

Contours can also be imported from other results. This will import the list of available contours as well as their generation and visualization properties, but it will not import the contour data. This can be useful for users who want to view the same set of contours across several result sets. To import contours, under the **File** menu, select **Import Occupant Contours....** In the file chooser dialog, choose the PFRMETA file that corresponds to the desired results set, and choose **Open**. This will add the contours in the specified result set into the current result set.

Duplicating Contours

Contours can also be duplicated. This is useful for creating several versions of a particular quantity, such as showing different averaging intervals or creating several snapshots of a contour at different times.

To duplicate a contour, in the navigation view, right-click the contour and select **Duplicate**. This will create and add a duplicate of the selected contour. Several contours may be duplicated at once by selecting several in the tree before duplicating. **NOTE:** If the selection contains both a filter and its base contour, the duplicate filter will be filtering the base contour's duplicate and not the original.

Global Contour Properties

There are several global properties that control how each contour is generated and displayed. To modify these properties, right-click the “Occupant Contours” group in the navigation view as shown in Figure 8-5, and select **Properties...**

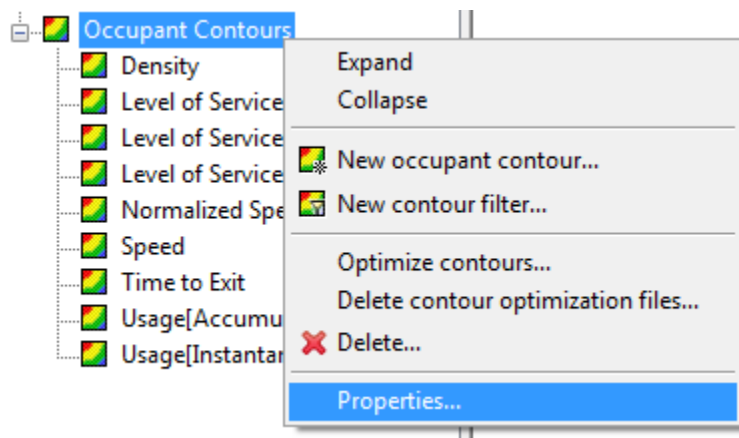


Figure 8-5: Occupant contours context menu

This will show the **Occupant Contours Properties** dialog as shown in Figure 8-6.

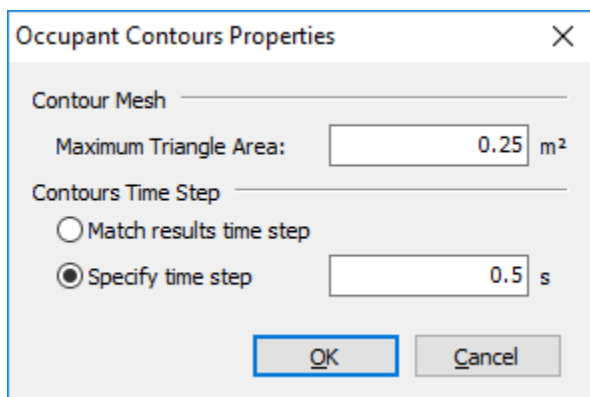
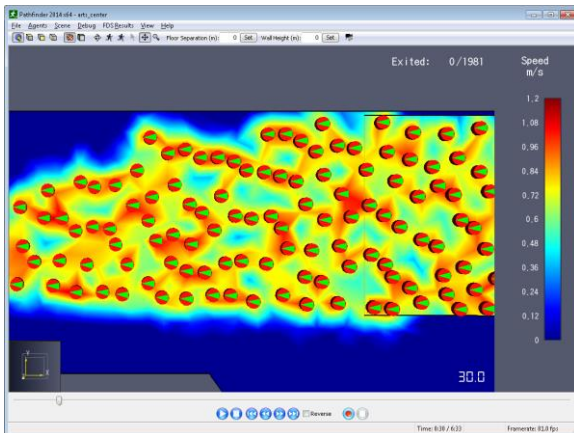


Figure 8-6: Occupant Contours Properties dialog

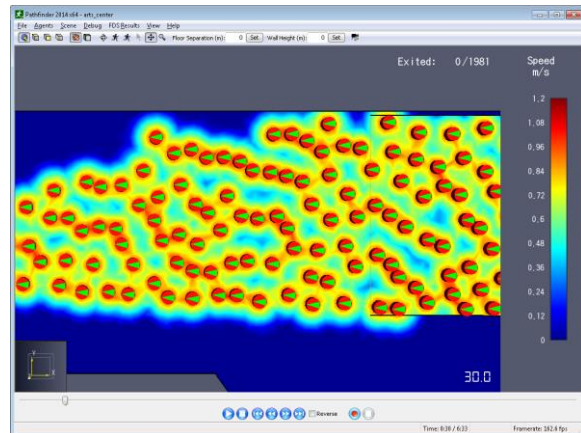
- **Maximum Triangle Area:** This controls the quality of the generated contours. Smaller values result in higher quality contours at the expense of larger optimization files and longer calculation times. Specifically, this controls the resolution of the underlying computation mesh for the contours. The effect of this parameter can be seen in Figure 8-7.
- **Match results time step:** Selecting this causes the contours to be generated at the same frequency as the Pathfinder results. This provides the most time accuracy but also takes the longest to generate and creates the largest files.
- **Specify time step:** Allows the contour time step to be specified. Values higher than the time step of the Pathfinder results will decrease the time accuracy of the generated contours but will

also decrease the calculation time and disk space requirements for the contour files. The specified time step will be rounded to the nearest multiple of the Pathfinder results time step.

NOTE: Changing one of the preceding properties invalidates all existing optimization files, requiring them to be recalculated. See Optimizing Contours on page 75 for more information.



Triangle area=.25m² (default)



Triangle area=.05m²

Figure 8-7: Effect of contour mesh resolution

Individual Contour Properties

Each contour also contains a set of properties that control the generation of the contour as well as its visualization. To set the properties of a contour, right-click the contour in the navigation view and select **Properties....** This will show the properties dialog for that contour. Each contour has a potentially different set of properties. The properties dialog for density is shown in Figure 8-8.

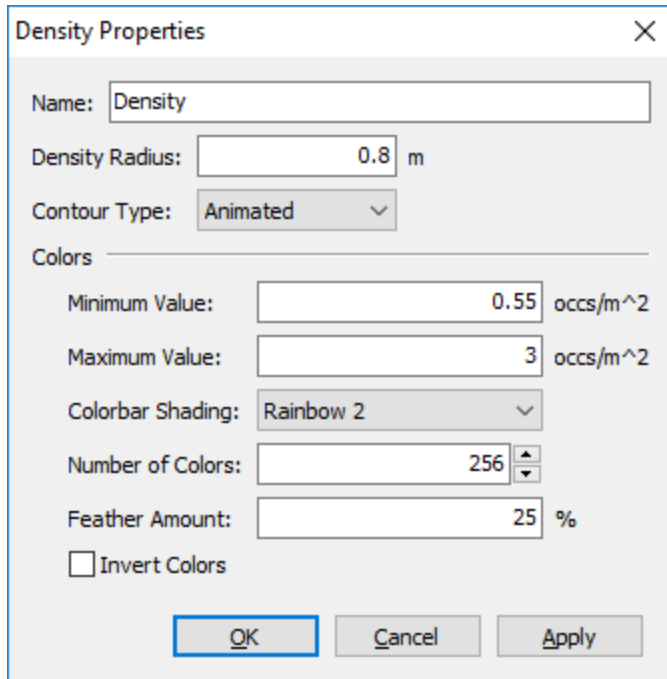


Figure 8-8: Density contour properties dialog

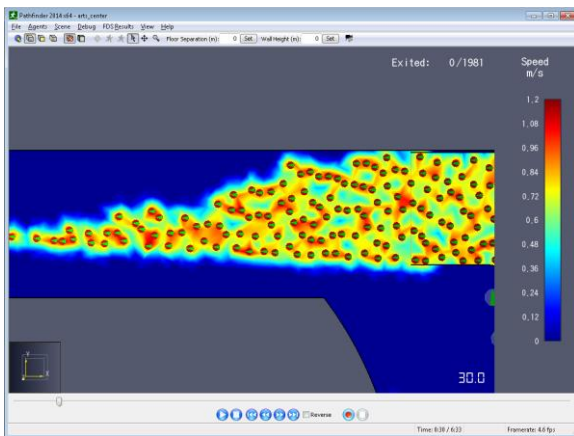
The following is composite list of all contour properties that control the creation of the contours:

- **Name:** The name displayed in the navigation view for the contour.
- **Density Radius:** Controls creation of the density contour. This radius determines the area in which occupants are searched on the solution mesh to determine density.
- **Influence Radius:** Controls the size of the area that each occupant's value has on the contour. The value is tapered from the full value where the occupant exists down to zero at this radius. The effect of this property is shown in Figure 8-9.
- **Contour Type:** Specifies the type of contour. Contours may either be **Animated** or **Snapshots**. If they are animated, their values will change as results are played back. If they are snapshots, however, they will always display the same contour value as calculated at a particular time or time range.
- **Limit Time Interval:** For contours that require a time range, such as **Average** and **Maximum** filters and **Usage [Accumulated]**, this option limits the time range to a fixed interval.
 - If this option is checked for an animated contour, the time interval is trailing. For instance, if this is checked and a time interval of 30 s is entered for an average density contour, the average will only be calculated for the trailing 30 seconds of density data at each time step. So at t=120s, the average is calculated over the time range of [90,120] s. If this option is left unchecked, the calculation will take place over the entire time range preceding each time step. So at t=120s, the time range would be [0,120] s.
 - If this option is checked for contour snapshots, the time interval is explicitly specified using **Minimum Time** and **Maximum Time**. If this option is unchecked, the time interval

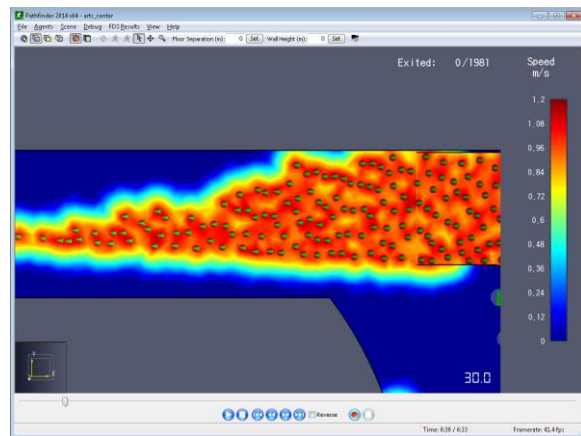
is set to the entire results playback time. So for instance, if there is currently 400 s of results data available, the time interval will be [0,400] s.

- **Snapshot Time:** For contour snapshots, this specifies the calculation time that will be displayed during playback.

NOTE: Changing one of the preceding properties also invalidates the optimization file for the modified contour if one exists, requiring it to be re-calculated. See [Optimizing Contours](#) on page 75.



Influence radius of 1m



Influence radius of 2m

Figure 8-9: Effect of influence radius on contours

The following are visualization properties of contours:

- **Minimum Value:** Sets the minimum value that is used to color the contour.
- **Maximum Value:** Sets the maximum value that is used to color the contour.
- **Colorbar Shading:** Sets the color scheme that is used to color the contours.
NOTE: For **Level of Service** contours, this controls the floor type, which is one of **Queuing**, **Stairway**, or **Walkway**.
- **Number of Colors:** Sets the number of colors used to shade the contours. This must range from 2 to 256.
- **Feather Amount:** Fades the lowest $\%$ of contour values to transparent so that the underlying geometry or navigation mesh may be seen.
- **Invert Colors:** Reverses the colors used to shade the contour.

NOTE: Changing one of the visualization properties will NOT invalidate contour optimization files.

Activating/Deactivating Contours

One contour may be activated/shown at a time. To activate a contour, either double-click it in the navigation view, or right-click it and select **Show** from the context menu. The first time this is performed, a dialog will appear asking whether to optimize the contour. Choosing **Yes** will cause a progress bar to appear while the contour is optimized and stored in an optimization file. See [Optimizing Contours](#) below for more information about optimization. Check **Remember my decision** to prevent being asked in the future. This can be changed in the preferences by going to **File->Preferences....**

Once a contour is active, it will appear bold in the navigation view. In the model view, the contour will appear overlaid on the navigation mesh or imported geometry and a colorbar legend will appear to the right.

The active contour can be hidden by either double-clicking it in the navigation view or right-clicking it and selecting **Hide**.

Optimizing Contours

Each contour may optionally be optimized for playback. Optimizing a contour is especially helpful with complex models, as it typically allows for much smoother playback and faster seeking when the contour is active, especially for contour filters. Contour optimization only has to be performed once per contour, as the optimization is stored on disk, allowing it to be accessed during later sessions.

The optimization file for each contour is stored in the same directory as the results and will have a filename based on the results filename and the contour's quantity and filter. For instance, a model named `test.pth` will have a density file name `test_density.octranim`.

There are several ways to optimize a contour. The first two methods optimize the contour ahead of time. This is the best option, but it comes at the cost of requiring time up-front to calculate the contour. Here are all the options for optimizing a contour:

- **Manually**¹ – Contours can be optimized manually by right-clicking the contours to optimize in the navigation view and selecting **Optimize ...** from the context menu.
- **On Activation**² – Contours can be optimized when they are activated (shown). By default, when a contour is activated and the contour has any unoptimized frames, Pathfinder will prompt the user to optimize. The decision can be remembered or this behavior can be manually changed by going to the **File Menu** and choosing **Preferences...**, which will show the **Preferences** dialog. Under the **Occupant Contours** section, choose an option for **Optimize on activation**.
- **On Playback** – If a contour has not been optimized using one of the two previous options, contours may instead be optimized dynamically during playback and while seeking. To enable this option, open the preferences dialog as described previously, and check the box next to **Enable dynamic optimization**. This will cause the contour frames to be optimized as requested either during playback or during seeking of Pathfinder results. Once a frame of contour data has been optimized and saved to disk, it will be retrieved from disk whenever that frame is needed again.

NOTE: While this option may improve performance somewhat, it can also produce very large files, especially if many frames are skipped during seeking. The reason is that Pathfinder reserves enough space in the optimization file for each missing frame such that it can store the frame uncompressed at a later time. In addition, this option works best if the results are stored on a disk volume formatted as NTFS. The previous two optimization options, however, calculate

¹ If the process of optimizing a contour is cancelled or skipped, the partial results are still retained in the optimization file and used to display the contour. In addition, these partial results are remembered the next time the contour is requested to be optimized unless some other action invalidates the file.

frames sequentially, producing compressed frames of data. These methods result in much smaller files and work well on any file system.

Once a contour has been optimized, there are some actions that may cause the optimization to become invalid. Pathfinder will re-optimize if necessary when the user requests that it be optimized again using one of the above methods. The following actions may cause a contour optimization to become invalid:

- Re-running the simulation.
- Re-writing the input file from the user interface.
- Modifying one of the global contour properties as described previously.
- Modifying an individual contour's generation properties as described previously.
- Modifying the properties of a contour that another contour depends on. For instance, modifying the properties of the density contour will invalidate the optimization for any filter that has been applied to that density contour, such as an average density or maximum density filter.

Contour optimization files may be very large for models with large navigation areas and/or long simulation times. These optimization files may be deleted at any time to preserve disk space in one of two ways:

- In the navigation view, right-click the contours for which their optimization files should be deleted and select **Delete optimization file....**
- In the **Analysis** menu select **Delete contour optimization files....** This will delete the optimization files for all contours still present in the results.

9. Technical Reference

Occupant Contours

The Results application provides some post-processing capabilities, including the ability to generate contours that show dynamic occupant data overlaid on the navigation mesh or imported geometry as shown in Figure 9-1.

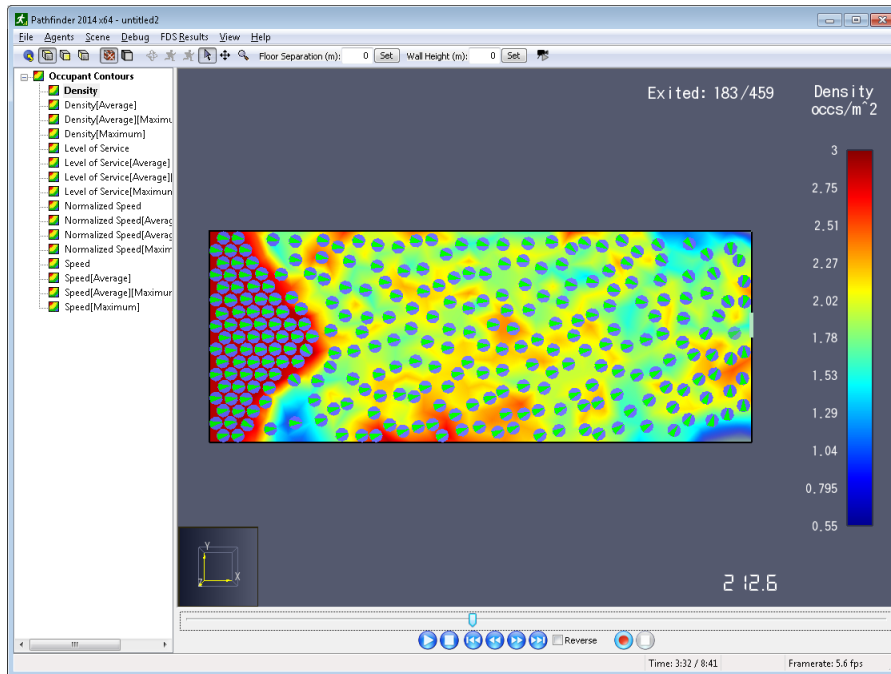


Figure 9-1: Occupant contours

Contours can either be calculated as results are played back or can be optimized and stored on disk for later retrieval. Optimization is discussed further in the User Manual. The calculation of the contours is described in this guide.

Contours are calculated using a sub-divided version of the navigation mesh as seen in Figure 9-2.

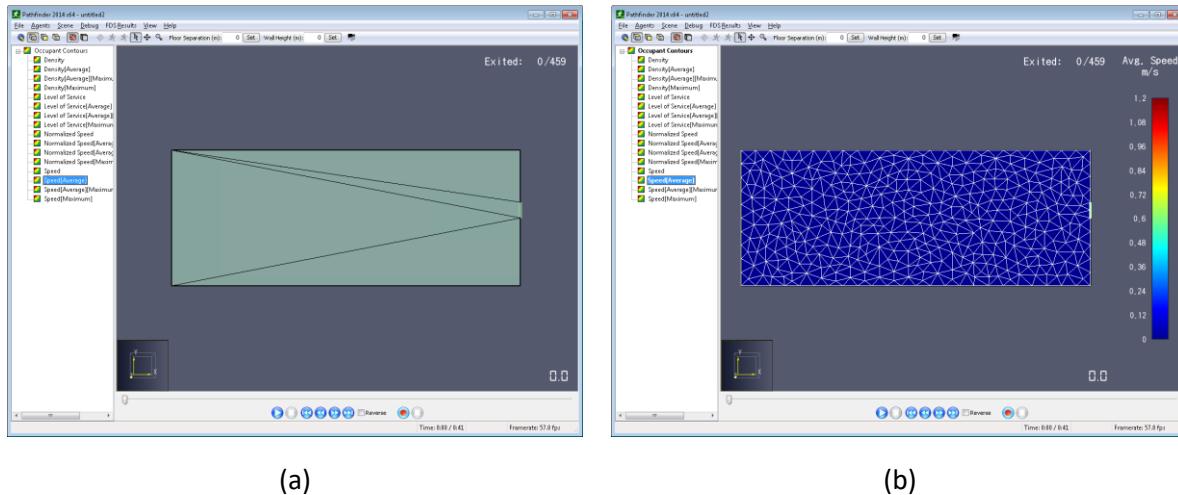


Figure 9-2: Navigation Mesh vs. Occupant Contour Mesh (a) Navigation Mesh (b) Occupant Contour Mesh

Contour values are calculated at the vertices of the triangles in the contour mesh and are linearly interpolated across triangles to produce the shading. The contour mesh is used only to display occupant contours and is independent from the navigation mesh used during the simulation. The spatial resolution of the contour mesh as well as the update frequency can be controlled using the **Occupant Contours Properties** dialog. Smaller values for space and time resolution offer smoother contours at the cost of greater calculation time and memory requirements.

Frames of contour data are produced throughout time depending on both the Pathfinder results data frames and the time step specified in the **Occupant Contours Properties** dialog as discussed in the User Manual. Each contour frame lines up in time with a frame of occupant animation data, though there does not have to be a one-to-one correspondence. This ensures that contour data is only produced when occupants are at known positions rather than interpolated positions. To view contour data at times between contour frames, the data is linearly interpolated between frames.

Density

The density calculated at each vertex of the contour mesh is dependent on the location of nearby occupants and the value of the **Density Radius** parameter. Any occupant intersecting the circle around a vertex contributes to the density at that vertex. Occupants who partially fall within the circle around a vertex contribute partially based on intersection area.

Increasing the **Density Radius** parameter smooths out contours by eliminating peaks and valleys in the density calculation. This effect will tend to be more visible near the edges of dense flows and more stable within uniformly dense regions due to partial attribution of occupants intersected by the density measurement radius.

The following steps describe in detail how the density values that control contour colors are calculated.

1. For each vertex of the contour mesh, an area is calculated. The area is at most πr_d^2 , where r_d is the user-settable parameter, **Density Radius**, as described in the User Manual. The area will be less if there are boundaries closer than r_d .
2. For each vertex, the mesh is traversed to find occupants whose bodies are at least partially within r_d .
3. The number of occupants within this radius are summed, with occupants fully within the radius counting as 1 and occupants partially within the radius counting as follows:

$$C = 1 - \frac{d - r_d + r_o}{2r_o}$$

where r_o is the occupant's radius and d is the 2-d distance from the vertex to the occupant's center.

4. The occupant count is then divided by the area at the vertex to produce the density value.

Level of Service

Level of Service is implemented the same as density but with three special color maps. Each color map represents a different type of terrain, including Walkway, Stairway, and Queuing [Fruin, 1987].

Speed

For each occupant, their speed is applied to the vertices of the contour mesh within a distance of r_i (see **Influence Radius** in the User Manual). The speed value at each vertex is calculated as follows:

$$s_v = \max_{1 \leq x \leq n} \left(s_x \frac{r_i - d_x}{r_i} \right)$$

where n is the number of occupants within a 2-d radius of r_i surrounding the vertex's position, s_x is the current speed of occupant x , and d_x is the 2-d distance from the vertex to the position of occupant x .

Normalized Speed

The normalized speed contour is calculated similarly to the speed contour, except that the speed value per vertex is the following:

$$s_v = \max_{1 \leq x \leq n} \left(\frac{s_x}{s_{xmax}} * \frac{r_i - d_x}{r_i} \right)$$

where s_{xmax} is the maximum speed of occupant x .

Usage [Instantaneous]

The usage value at each vertex is calculated as follows:

$$u_v = \max_{1 \leq x \leq n} \frac{r_i - d_x}{r_i}$$

Usage [Accumulated]

The accumulated usage contour is implemented as a time-integrated filter on top of an instantaneous usage contour. See Integrate on page 80 for more information about the integration filter.

Time to Exit

The Time to Exit contour value at each vertex is calculated as follows:

$$tte_v = \max_{1 \leq x \leq n} \left((t_{exit} - t_{curr}) \frac{r_i - d_x}{r_i} \right)$$

where t_{exit} is the occupant's exit time and t_{curr} is the current contour calculation time.

Average

Average contours are applied on top of other contours. They are calculated over a user-specified interval that is always trailing a particular time step. So if the averaging interval is 30 s long, then at $t=120$ s, the averaging interval is [90,120] s. The interval cannot go back in time, however, so at $t=10$ s, the interval is [0,10] s.

For each vertex, the vertex contour values of the base contour are averaged over this interval to produce the final vertex value as follows:

$$v_x = \frac{1}{n} \sum_{f=x-n+1}^x v_f$$

where x is the index of the contour frame for the current time step, n is the number of frames that intersect the calculation interval (see **Calculation Interval** in the Pathfinder User Manual), v_f is the vertex value of the base contour at frame f .

Maximum

Maximum contours are applied similarly to average contours. For each vertex, the final vertex value is the maximum of the vertex values of the base contour over the time interval as follows:

$$v_x = \max_{x-n+1 \leq f \leq x} v_f$$

Integrate

Currently, the integration filter is not exposed as a general contour filter – it is only available incorporated into the accumulated usage contour. The integration filter is implemented on a per-vertex basis as follows:

$$v_x = \sum_{f=x-n+1}^x v_f (t_f - t_{f-1})$$

where t_f is the time of frame f , and t_{f-1} is the time at frame $f - 1$.

10. Troubleshooting

Floating License Server Problems

The floating license server provides an administration web service that can assist with diagnosing many floating license problems. If the license server is installed on the local machine, the license administration server can be accessed at the following web address:

<http://localhost:5054>

If the license server is installed on a remote machine, replace "localhost" with the name of the remote machine (i.e. the host name of the floating license server).

Once you have opened the Reprise License Server Administration page, you can use the Status commands to attempt to debug the license server problem yourself, or you can send a diagnostic report to support@thunderheadeng.com for assistance. To generate a diagnostic report:

1. Under RLM Administration Commands, click **Diagnostics**
2. Click **Run Diagnostics**
3. Email the generated file (rlmdiag.txt) to support@thunderheadeng.com

Video Display Problems

The Results application utilizes several advanced graphics card features in order to provide accelerated display of results. If you experience crashes or problems with display, you should first make sure your video card drivers are up-to-date. If they are, there may be some rendering options that can be turned off in the preferences at the expense of decreased fidelity or rendering speed.

If you encounter these problems, please let us know the make/model of your video card and what video driver you are using. This will help us improve our software and increase compatibility.

Results crashes on startup

If the Results application immediately closes when first started, this is usually the result of bugs in the graphics driver. First make sure you are using the latest drivers for your graphics hardware. Some computers may have two graphics adapters, especially laptops. If you are not sure which graphics are being used for the Results application, update the drivers for both GPUs. If this still doesn't fix the problem, try running the Results in **safe-mode**. This can be done as follows:

1. Create a shortcut to the Results application, either *PyroSimResults.exe* for PyroSim or *PathfinderResults.exe* for Pathfinder. This executable is found in the install folder for PyroSim or Pathfinder.
2. Right-click the shortcut and select **Properties**.
3. At the end of the **Target** field, add a space and "-safemode".
4. Click **OK**.
5. Double-click the shortcut to run the results in safe-mode.

Running the results in safe-mode will disable several advanced graphics features. It will also show a console window with some logging information.

If the Results application starts successfully in safe-mode, the graphics features can be re-enabled one at a time in the **Preferences** dialog to see which one caused the problem. See Controlling drawing detail/speed on page 15.

NOTE: The advanced graphics settings are disabled every time the safe-mode shortcut is used. After re-enabling the non-problematic settings, delete the shortcut and run the Results application normally.

Results application crashes when results are loaded

If the Results application crashes when specific results are loaded, such as FDS smoke and fire, the graphics card drivers are usually at fault. First try updating the graphics drivers. If that does not fix the problem, try using a compatibility setting for the result type being loaded. For more information, see Controlling drawing detail/speed on page 15.

Results don't display correctly

If particular results do not display correctly, such as a 2D slice that never changes color, this is usually an issue with the graphics card drivers. Older Intel HD Graphics drivers are especially problematic. First try updating the graphics drivers. If that does not fix the problem, try using a compatibility setting for the result type being displayed. For more information, see Controlling drawing detail/speed on page 15.

Stuttering FDS Results

If FDS results, such as fire and smoke, stutter or pause frequently during playback, this may be because the results files are too large to stream. See File Streaming on page 15 for more information on combatting this problem.

Unresponsive Results

When results are streaming from disk (see File Streaming on page 15), they require a connection to the file on disk. If the result files are on a remote machine, there is a possibility that the files can become disconnected from the Results application. In this case, the application may become frozen as it tries to read more data from the disconnected disk. This is usually caused by stability issues with the network.

The current remedy for this problem is to copy the results to the local machine to ensure a more reliable connection with the Results application.

Known Issues

There are some known issues with the Results. Some of these issues may be fixed in future releases, while others may be deemed non-critical with no plan for a fix.

- FDS volumetric renderings, such as fire and smoke and Data3d, may not have the correct transparency if there are solution meshes embedded inside other meshes. FDS does not prevent embedded meshes, but it does discourage them.

- Smoke and fire do not appear correctly when viewed behind translucent surfaces, such as glass. We hope to improve this in future versions.
- Translucent objects do not blend correctly when overlapping each other. This is related to the problem with smoke and fire and will hopefully be fixed in future revisions.
- For the smoke_test problem presented in Chapter 3 of the Smokeview Verification Guide (Forney, 2017), only the **Use Hardware Shaders** render method for smoke produces the expected greyscale values when viewing the scene from the front camera. See Controlling drawing detail/speed on page 15 for more information on the render method. This is because the other modes draw one smoke slice too many, making the smoke appear slightly too opaque. The extra slice is drawn because the obstructions line up exactly with some of the slice planes, and the scene geometry is drawn slightly offset away from the camera using polygon offset to allow some objects to appear on top of the obstructions without interference artifacts. When viewed from any perspective other than the front, the greyscale values should be correct in these modes.
- Volumetric Data3d and Data3d Slices do not work over remote desktop or on older graphics hardware.

Contacting Technical Support

If you need assistance, or wish to provide feedback about any of our products, contact support@thunderheadeng.com

11. References

Forney, Glenn P. 2017. *Smokeview, A Tool for Visualizing Fire Dynamics Simulation Data Volume II: Technical Reference Guide*. Gaithersburg, Maryland, USA : NIST, 2017. NIST Special Publication 1017-2 Sixth Edition.

—. **2017.** *Smokeview, A Tool for Visualizing Fire Dynamics Simulation Data Volume III: Verification Guide*. Gaithersburg, Maryland, USA : NIST, 2017. NIST Special Publication 1017-3 Sixth Edition.

Fruin, John J. 1987. pedestrian planning and design. Revised Edition. s.l. : Elevator World, Inc., 1987, 4, pp. 71-87.

SFPE. 2003. Movement. *Engineering Guide to Human Behavior in Fire*. s.l. : Society of Fire Protection Engineers, 2003.

Thunderhead Engineering. Pathfinder Resources. *Pathfinder Technical Reference Manual*. [Online] <http://www.thunderheadeng.com/pathfinder/pathfinder-resources/>.