

Using the Fire and Smoke Wizard

This section describes the fire and smoke wizard feature of Simcenter STAR-CCM+.

This section explains the following:

- [What is the fire and smoke wizard?](#)
- [Other capabilities](#) of Simcenter STAR-CCM+ that can be used with the fire and smoke wizard for problems of this type
- [Setting up the fire properties](#)

Contents:

[What is the Fire and Smoke Wizard?](#)

[Simcenter STAR-CCM+ Capabilities Related to Fire and Smoke](#)

[Setting Up the Fire Properties](#)

What is the Fire and Smoke Wizard?

The fire and smoke wizard is an automatic simulation tool for simulating a building fire.

Its capabilities for this application include :

- Fire regions and sources
- Soot sources
- Velocity inlet boundaries for ventilation
- Fire doors
- Heat and smoke detectors

Simcenter STAR-CCM+ Capabilities Related to Fire and Smoke

This section lists some of the other Simcenter STAR-CCM+ features that can help model fire and smoke.

The fire and smoke wizard is the primary tool for these types of problems, but there are several other Simcenter STAR-CCM+ features that can help with the CFD solution:

- Modeling Options:
 - Stationary
 - Three-Dimensional
- Radiation Models:
 - Discrete Ordinates Method (DOM)
- Turbulence Models:
 - Spalart-Allmaras
 - K-Epsilon, K-Omega...
- Boundary Conditions:
 - Velocity Inlet
 - Stagnation

- Pressure
- Solvers:
 - Second-Order
 - Segregated
 - Algebraic Multigrid Acceleration
- Reports, monitors, and plots
- Derived parts

Setting Up the Fire Properties

This section outlines the process for setting up the fire properties.

This feature would be used after you generate a mesh. The process involves the following:

- [Activating the wizard](#)
- [Creating fire volumes](#)
- [Setting up fire dynamics](#)
- [Specifying fire doors](#)
- [Setting up ventilation](#)
- [Preparing the soot source](#)
- [Adding heat and smoke detectors](#)
- [Specifying optical properties](#)
- [Using post-processing planes](#)
- [Setting graph parameters](#)

Contents:

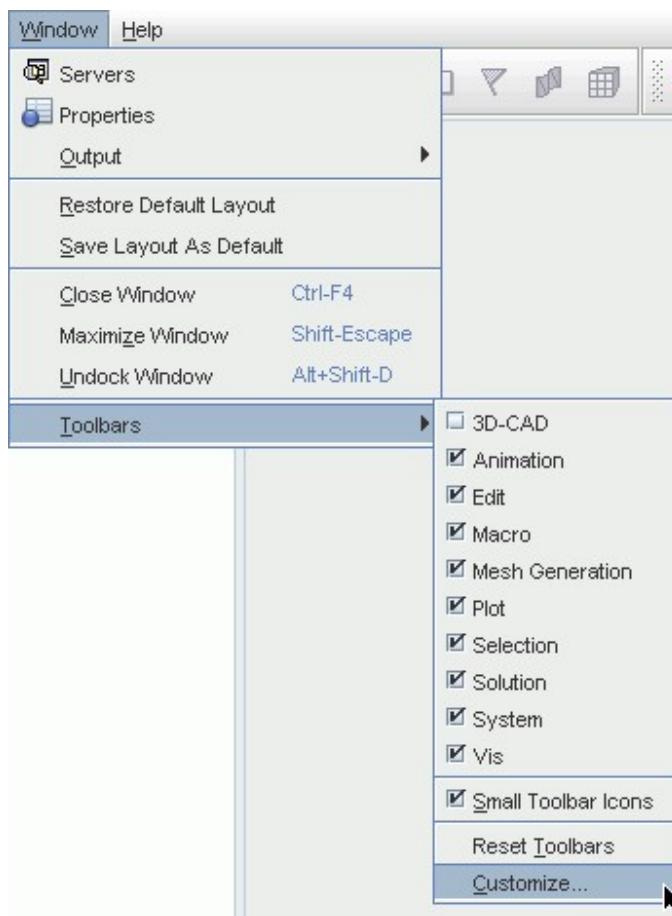
- [Activating the Fire and Smoke Wizard](#)
- [Creating Fire Volumes](#)
- [Setting Up Fire Dynamics](#)
- [Specifying Fire Barriers](#)
- [Setting Up Ventilation](#)
- [Preparing the Soot Source](#)
- [Adding Heat and Smoke Detectors](#)
- [Specifying Optical Properties](#)
- [Using Post Processing Planes](#)
- [Setting Graph Parameters](#)

Activating the Fire and Smoke Wizard

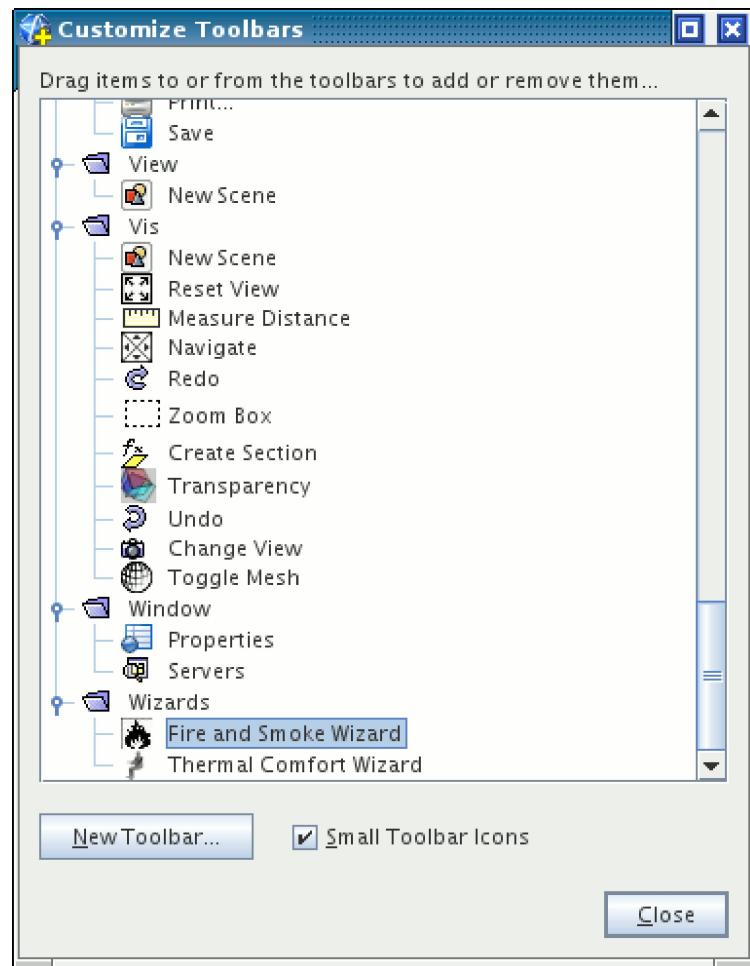
The properties of the fire are set using the **Fire and Smoke Wizard**.

To begin accessing this feature:

1. Select **Windows > Toolbars > Customize** from the menu bar.

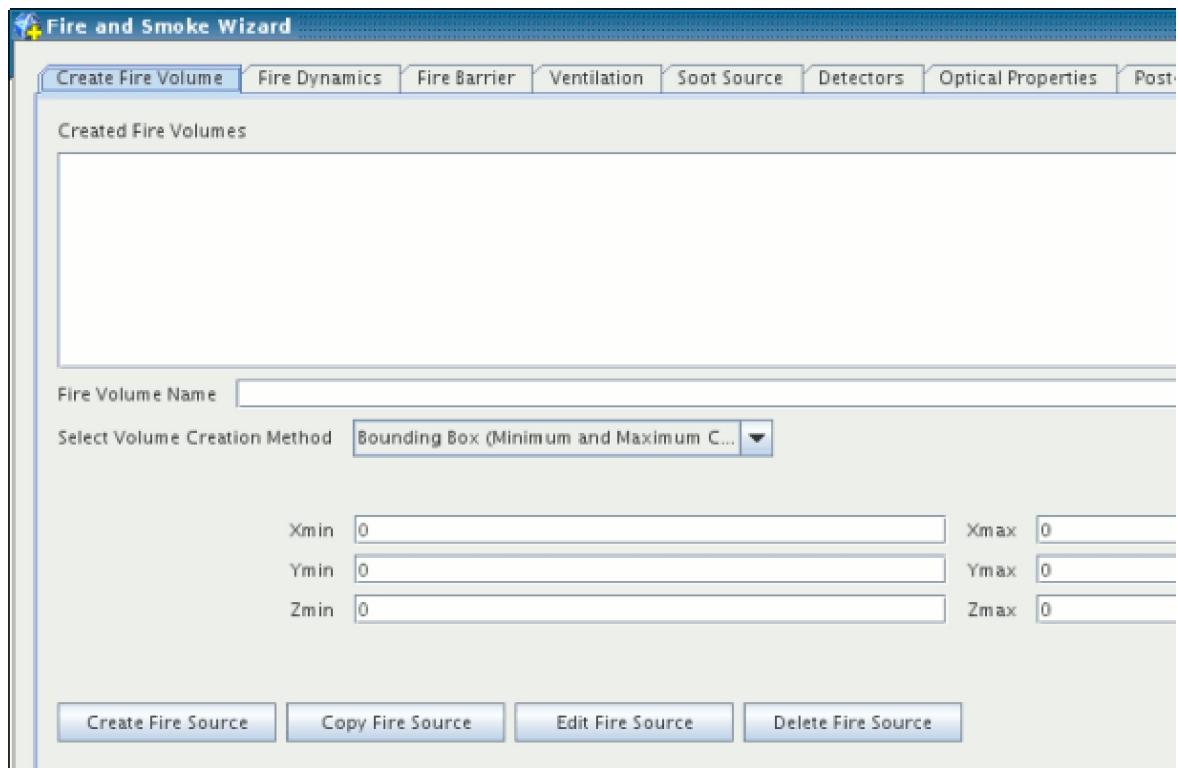


This opens the *Customize Toolbars* dialog as shown in the following screenshot.



2. Drag and drop the **Fire and Smoke Wizard** icon onto the end of one of the toolbars near the top of the workspace.
3. Click **Close** on the *Customize Toolbars* dialog.
4. Click the  (**Fire and Smoke Wizard**) button.

This opens the *Fire and Smoke Wizard* dialog.



There are two broad categories of detail that this dialog provides:

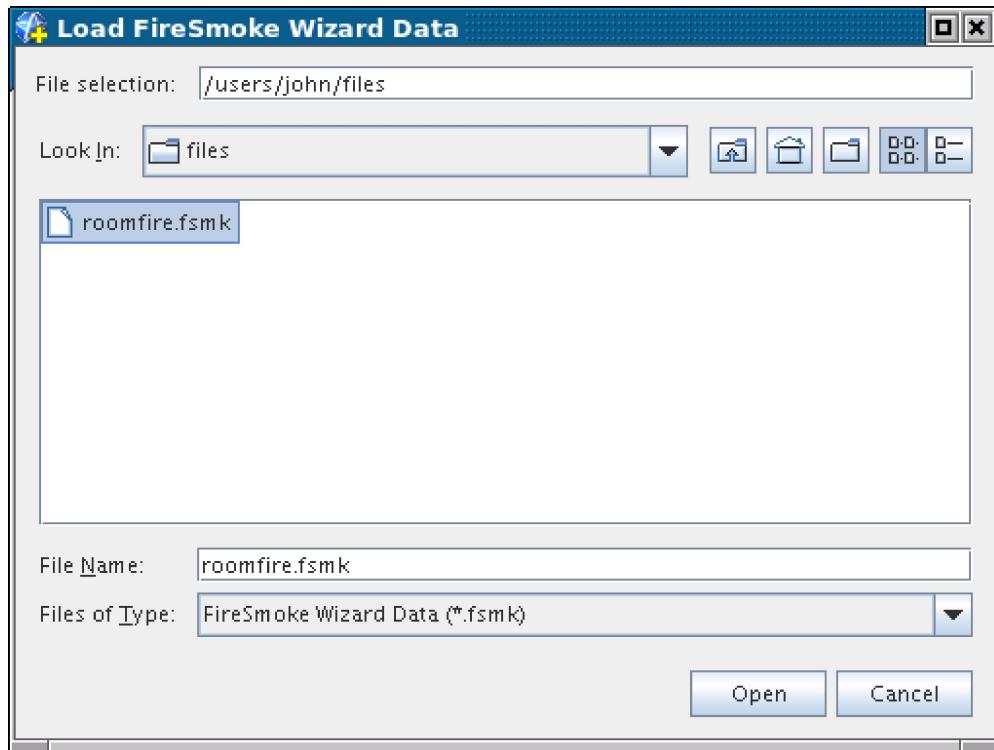
- Required fire characteristics are provided in the following tabs:
 - *Fire Dynamics*
 - *Soot Source*
 - *Optical Properties*
- Optional characteristics of the fire and building are provided in the following tabs:
 - *Create Fire Volume*
 - *Fire Barrier*
 - *Ventilation*
 - *Detectors*

Post-Processing Planes let you display data in precisely defined sections and output that data to images.

5. Finally, after you click **Apply**, the wizard automatically saves your settings to an external .fsmk file. This allows you to import those settings into another fire and smoke simulation.
6. To import the file, click the **Load FireSmoke** button near the bottom of the dialog.

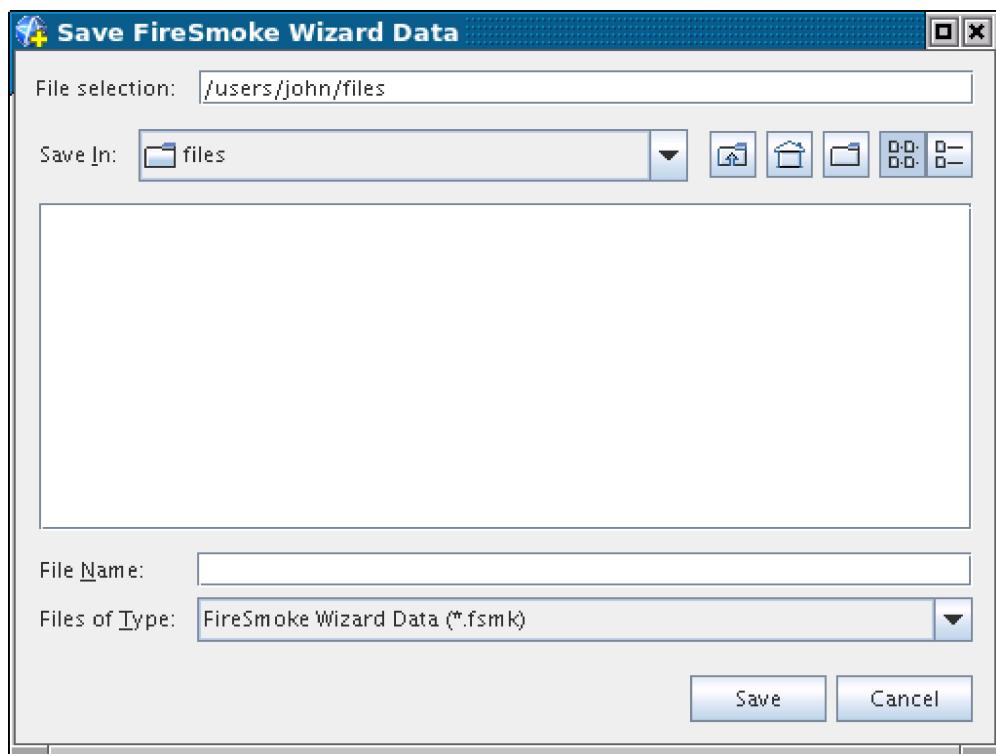


This activates a standard file-opening dialog with the Files of Type filter set to the .fsmk format.



7. Navigate to the file and click **Open**.
8. Additionally, before you click **Close** in the wizard, you can save the file with a name and path of your choice by clicking the **Save FireSmoke** button.

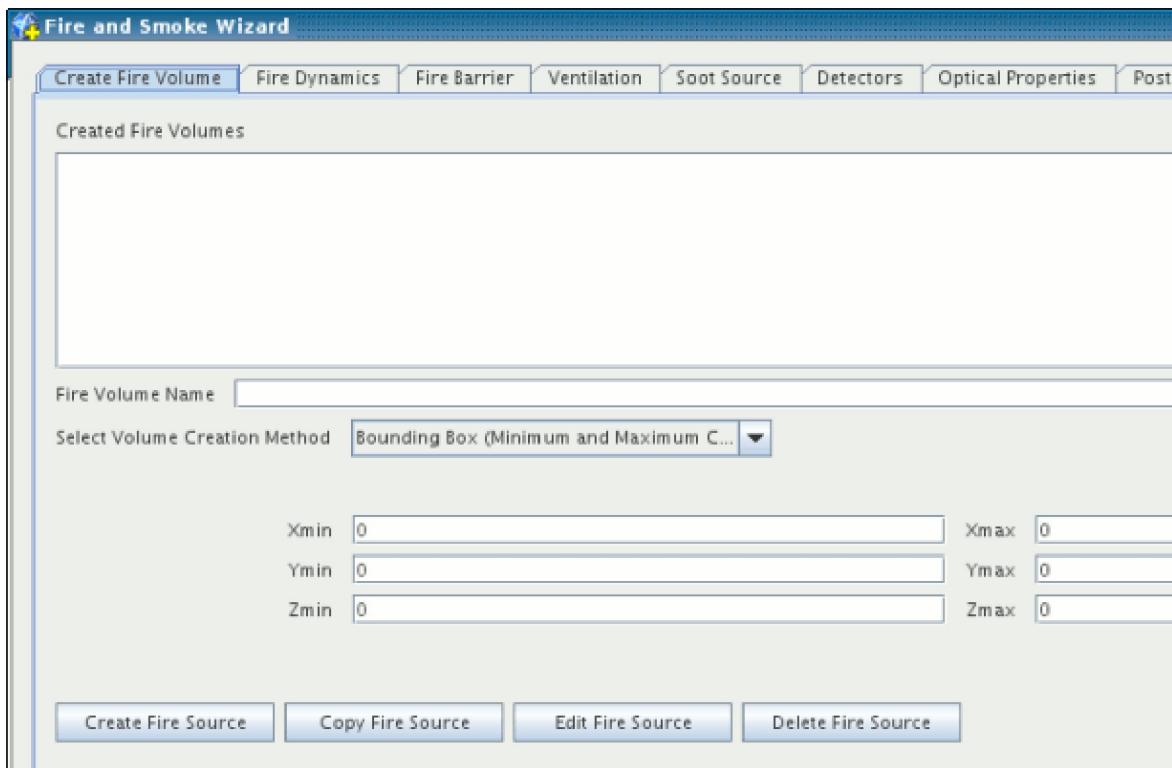
This opens a *Save* dialog.



9. Navigate to the preferred location, enter a file name, and click **Save**.

Creating Fire Volumes

The *Create Fire Volume* tab allows you to define a space for the fire that is an alternative to the regions created in the mesh generation process.



Begin by entering a name for the fire volume in the **Fire Volume Name** text box.

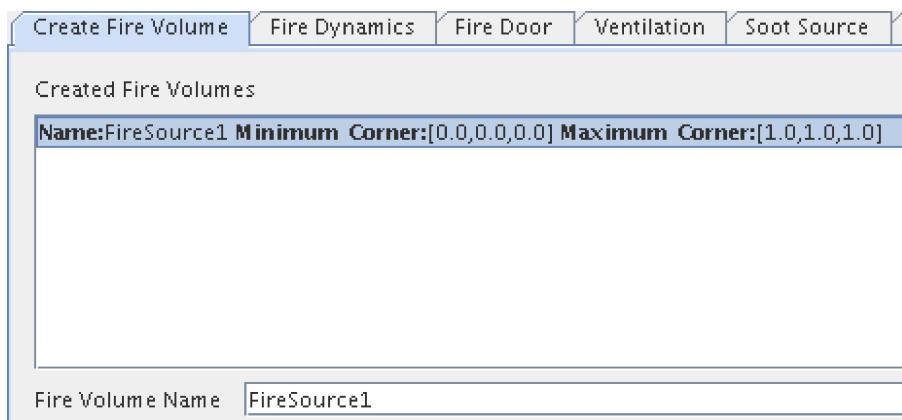
Next select the volume creation method from the drop-down list:

- A bounding box with minimum and maximum coordinates
- The absolute size with a midpoint and size bounds

The fields below the drop-down list change with your selection of method. Enter the appropriate coordinates.

Once you enter values, you can create the volume by clicking the **Create Fire Source** button.

The fire volume that you create appears with its pertinent data in the **Created Fire Volumes** list.



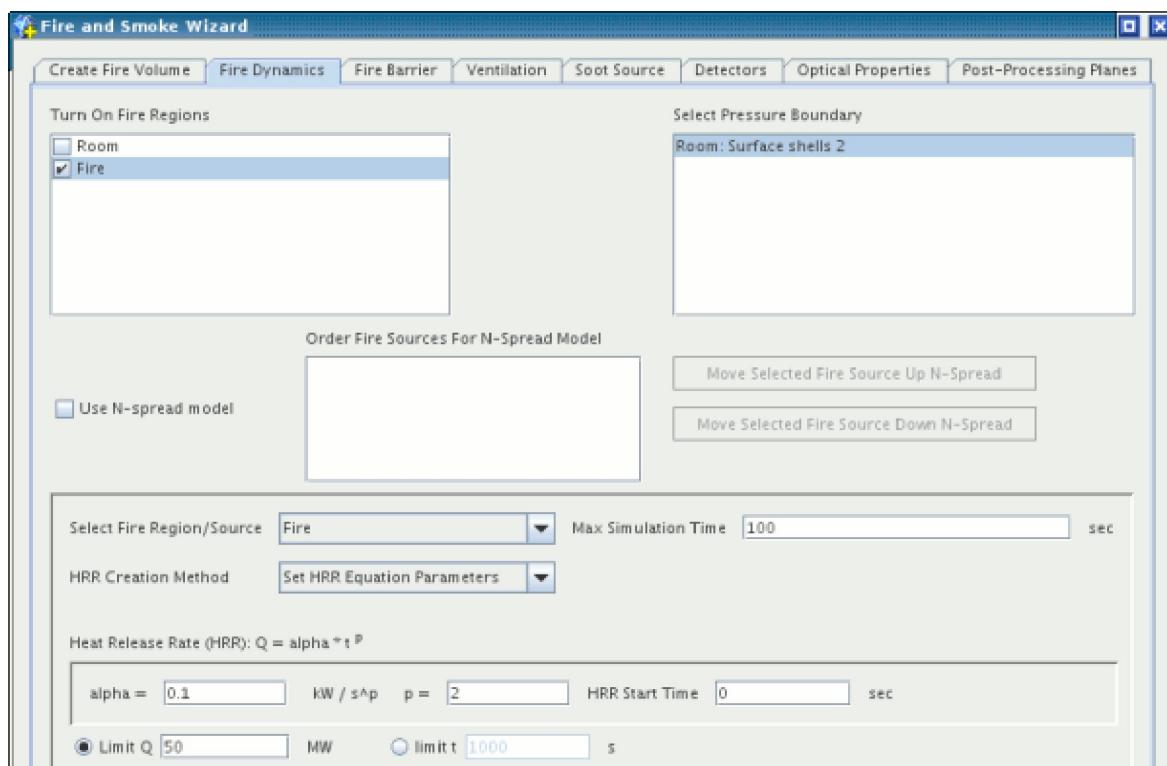
Once you finish with the wizard by clicking **Apply**, the fire volume also appears in the object tree within the **Regions** node.



It is also possible to manage the fire volumes in the list. For example, if you wish to create a second fire volume similar to the first, you need not repeat the entire creation process. Simply select the name of the fire volume in the list and click **Copy Fire Source**. An additional fire volume appears in the list. Select that copy, make any changes you wish, and then click **Edit Fire Source** to apply your changes. You can also delete fire volumes by selecting them in the list and clicking **Delete Fire Source**.

Setting Up Fire Dynamics

Set up the dynamic properties of the fire by selecting the *Fire Dynamics* tab.

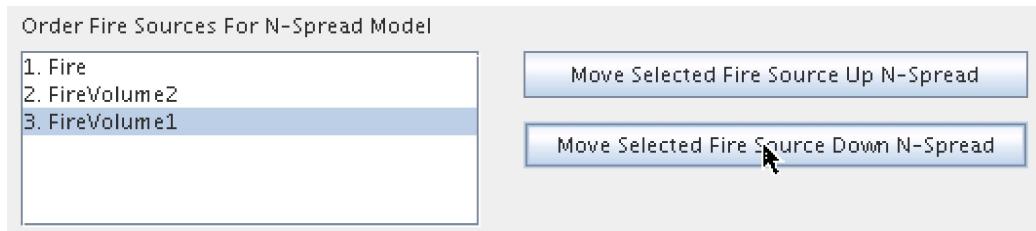


The *Turn On Fire Regions* list in the upper left lets you tick the checkboxes of regions that represent the fire, including any fire volumes that you may have created in the previous tab.

Select the pressure boundary in the list in the upper right.

If you wish to use an N-spread model, then tick the checkbox.

This makes the fire sources appear in the list, and you can use the buttons to the right of it to change the sequence. Simply select a fire source in the list and click the button to move it up or down within the list.

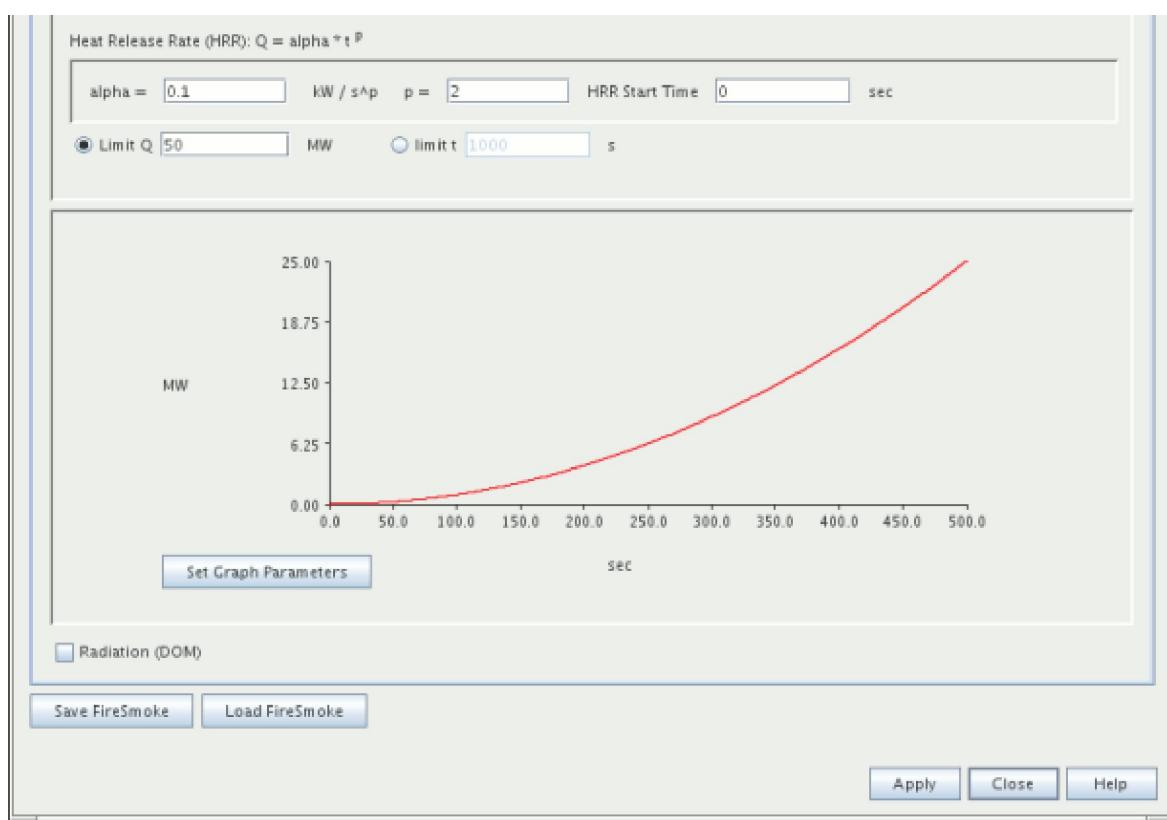


Select a fire volume from the drop-down list, and do as follows:

- Specify the maximum simulation time for it.
- Specify the values for alpha, p, and the HRR start time.
- Use the radio buttons to select whether you want to specify the limit for Q or for t, and then enter the value.
- Select another fire volume from the list, if applicable, and repeat the above steps.

The fire and smoke wizard retains your entries for each of the fire volumes.

This wizard also sets up a monitor plot of the heat release rate, which is previewed in the panel.



To control the parameters of this plot, click the **Set Graph Parameters** button.

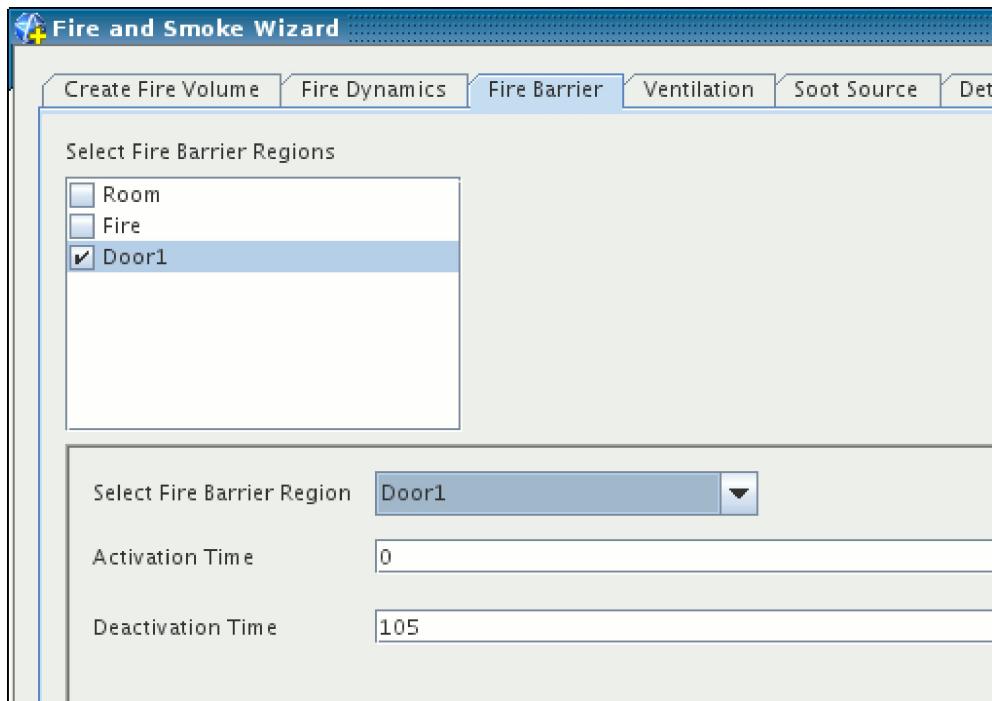
You can also tick a checkbox to use the DOM (Discrete Ordinates Method) radiation model. When you activate the radiation model, Simcenter STAR-CCM+ calculates the absorption coefficient of the air and gas mixture [1] as:

$$\begin{aligned} a &= 0.01, \text{ if } T < 323\text{K} \\ a &= 0.01 + (3.49/377)(T - 323), \text{ if } 323\text{K} \leq T < 700\text{K} \\ a &= 3.5 + (3.5/700)(T - 700), \text{ if } T > 700\text{K} \end{aligned} \quad (19)$$

- [1] A.J. Grandison, E.R. Galea, and M.K. Patel. May 2001. "Fire Modeling Standards/Benchmark—Report on SMARTFIRE Phase 2 Simulations", Fire Safety Engineering Group, University of Greenwich, May 2001.

Specifying Fire Barriers

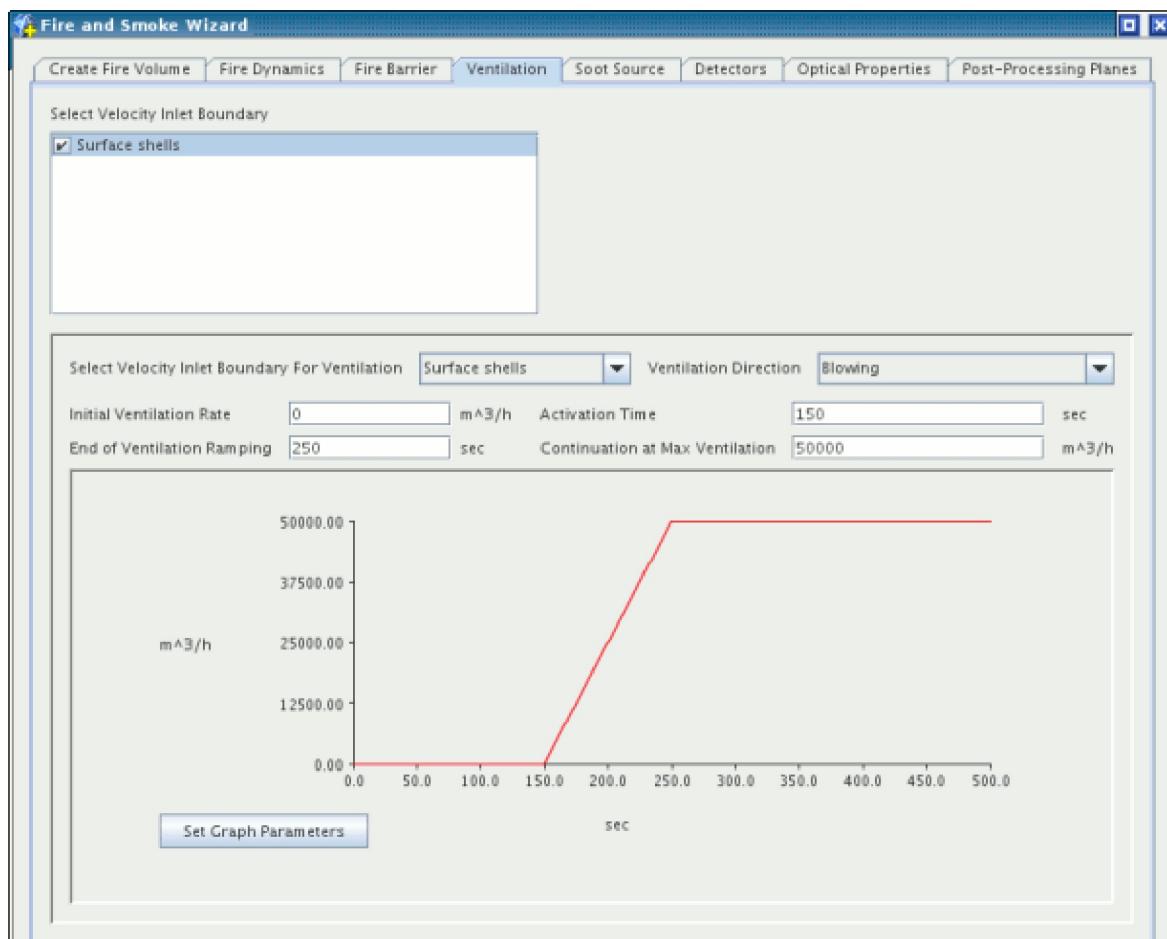
If a fire barrier, such as a fire door, is one of the regions created in your mesh, you can select it in this panel.



You can specify the activation and deactivation times of each fire barrier that you select in the drop-down list.

Setting Up Ventilation

You can select a velocity inlet boundary for ventilation, and then specify ventilation properties.



Begin by selecting a velocity inlet boundary from the list in the upper left.

If you are using multiple velocity inlet boundaries, use the drop-down list **Select Velocity Inlet Boundary For Ventilation** to select each one, and do as follows:

- Decide whether the ventilation direction is blowing or extraction, and select that option from the drop-down list.
- Enter values for initial ventilation rate, activation time, end-of-ventilation ramping, and continuation at maximum ventilation.
- Select another velocity inlet boundary from the list, if applicable, and repeat the above steps.

The fire and smoke wizard retains your entries for each of the velocity inlet boundaries.

This wizard also sets up a monitor plot of the heat release rate, which is previewed in the panel. To control the parameters of this plot, click the **Set Graph Parameters** button.

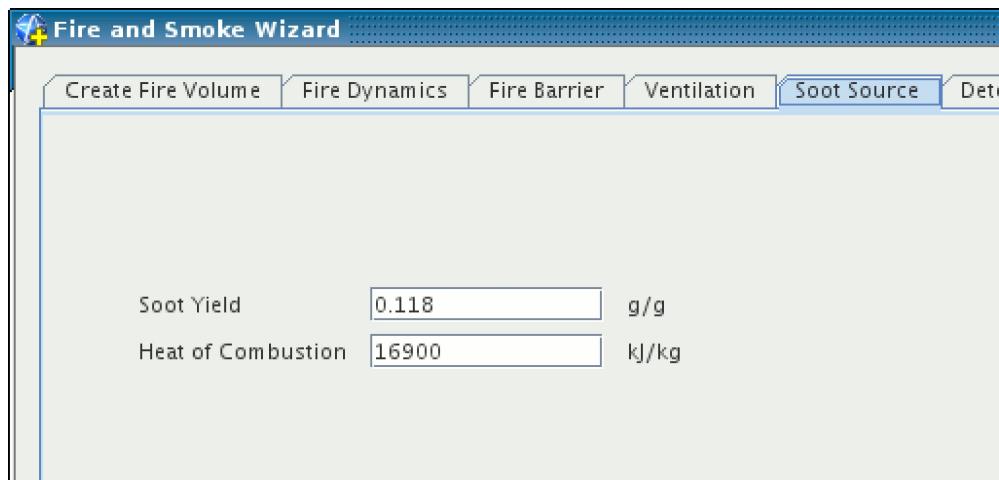
See Also:

[Setting Graph Parameters](#)

Preparing the Soot Source

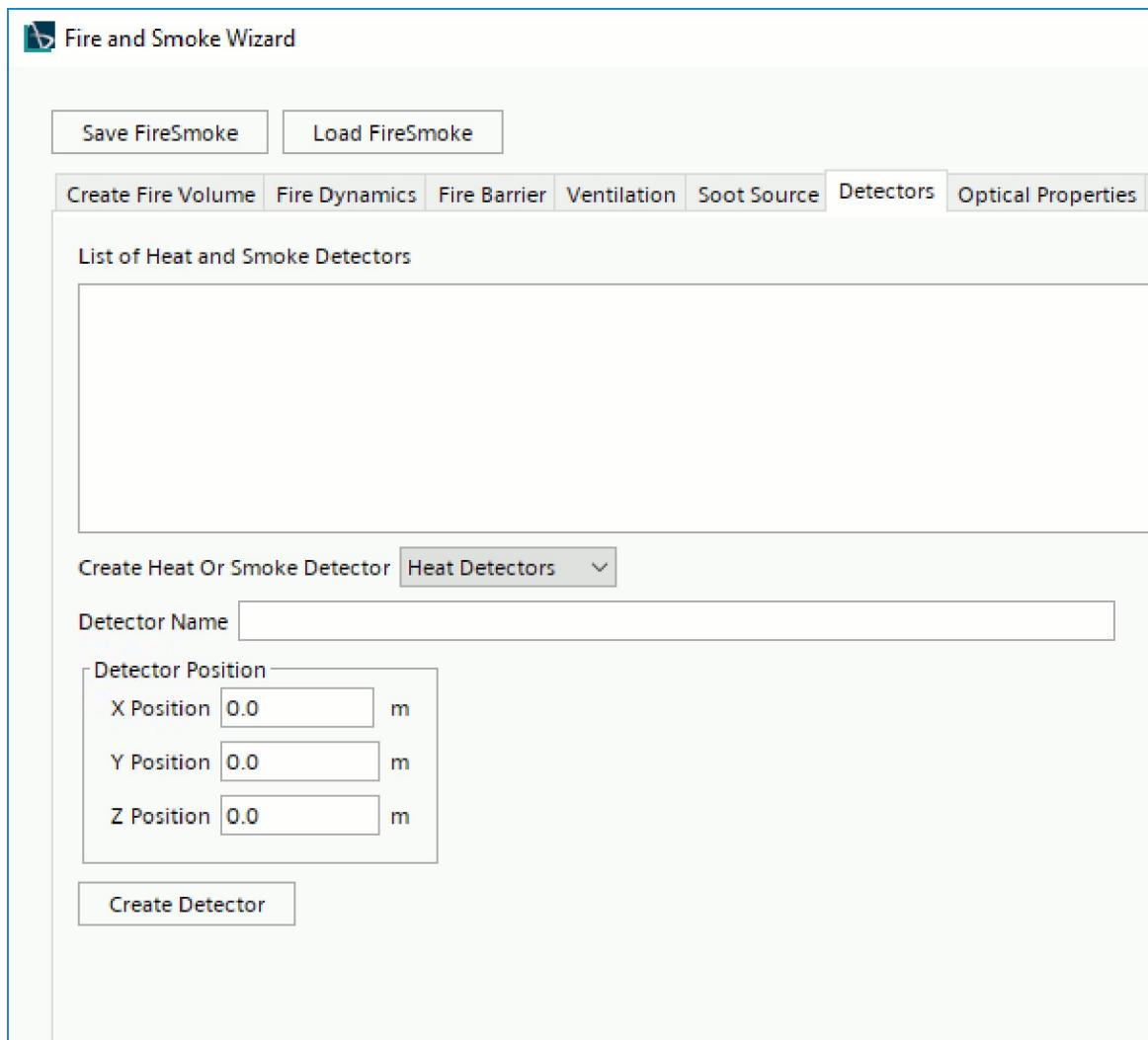
The Soot Source tab lets you specify the soot yield and heat of combustion.

Enter values, or accept the defaults, for the soot yield and heat of combustion.



Adding Heat and Smoke Detectors

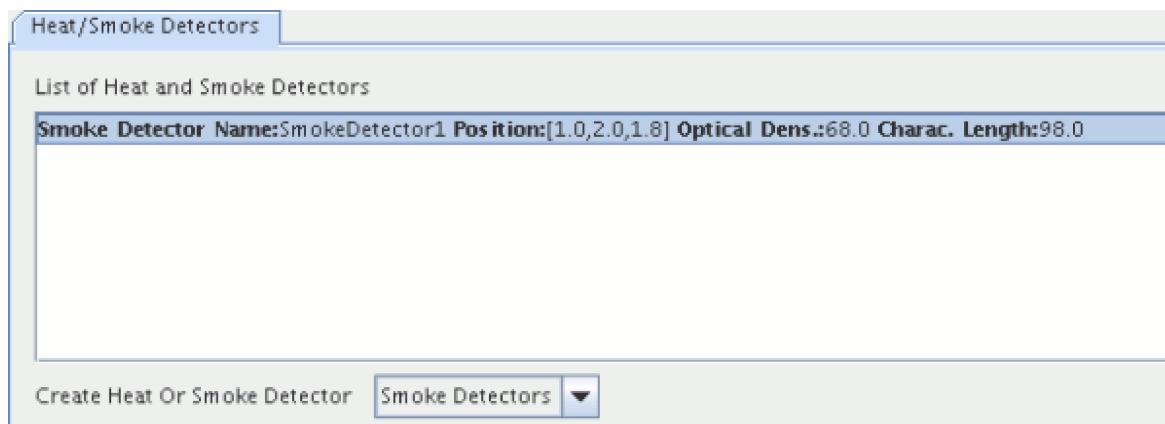
The *Detectors* tab lets you incorporate heat detectors or smoke detectors into your fire and smoke simulation.



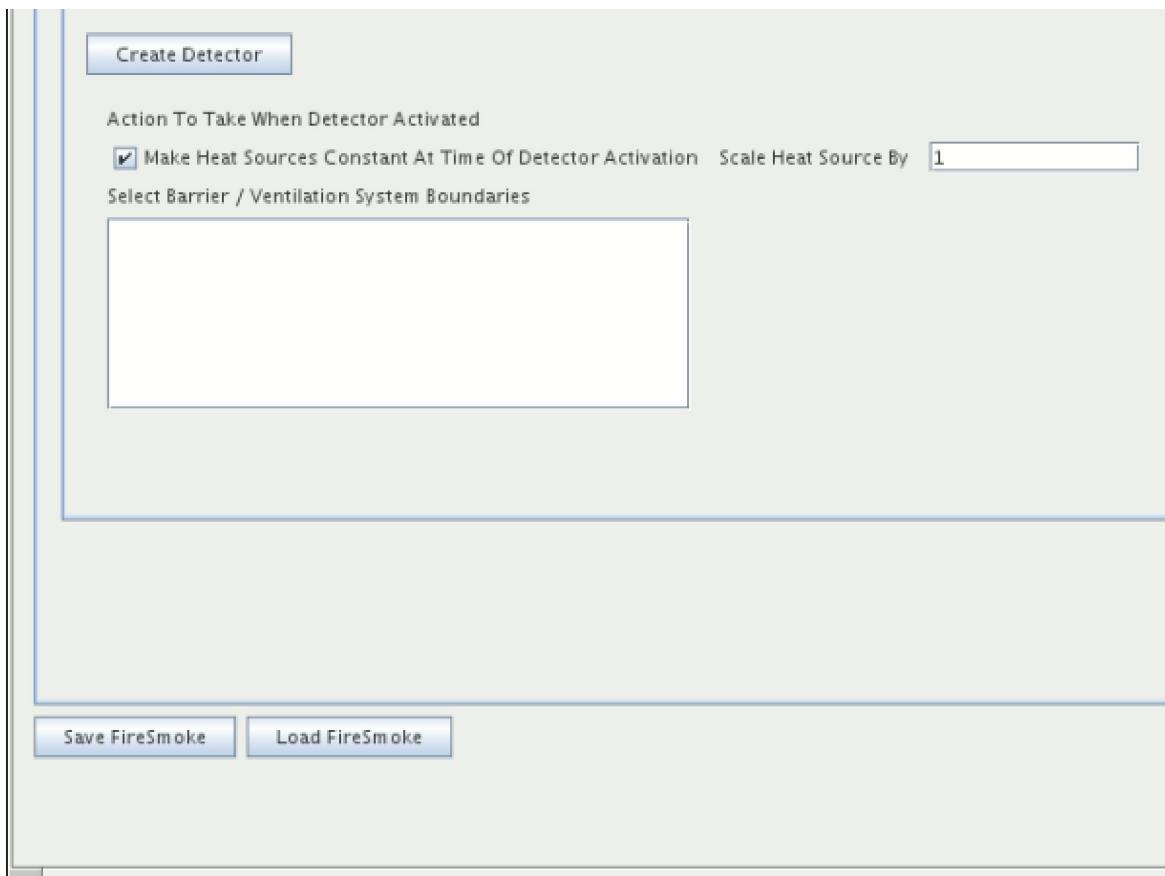
To incorporate heat detectors or smoke detectors into your fire and smoke simulation:

1. Begin by selecting the type of detector from the **Create Heat Or Smoke Detector** drop-down list. The settings for a particular type of detector change with your selection.
2. Enter a name for the detector in the **Detector Name** text box.
3. Enter the coordinates of the detector in the **Detector Position** group box.
4. Create the detector by clicking the **Create Detector** button.

The new detector appears with its pertinent data in the List of Heat and Smoke Detectors.

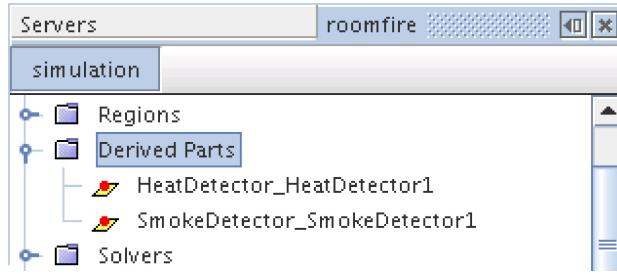


When the list contains one or more detectors, an additional set of options becomes available.

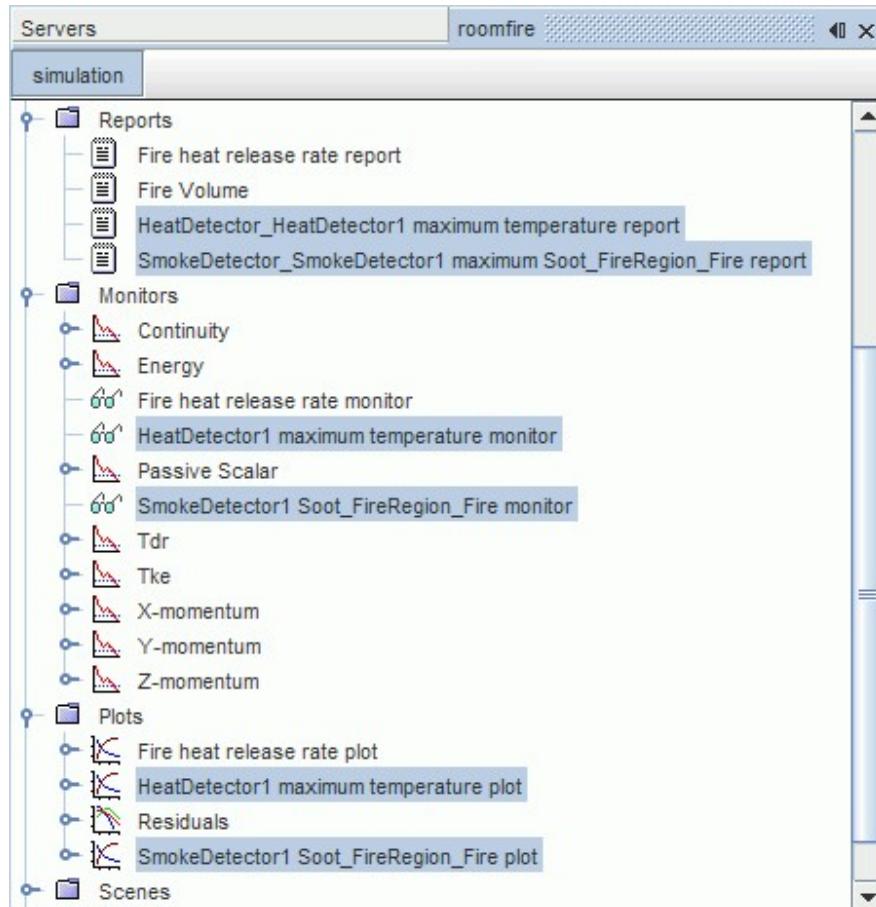


The settings are specific to each detector that you highlight in the detector list.

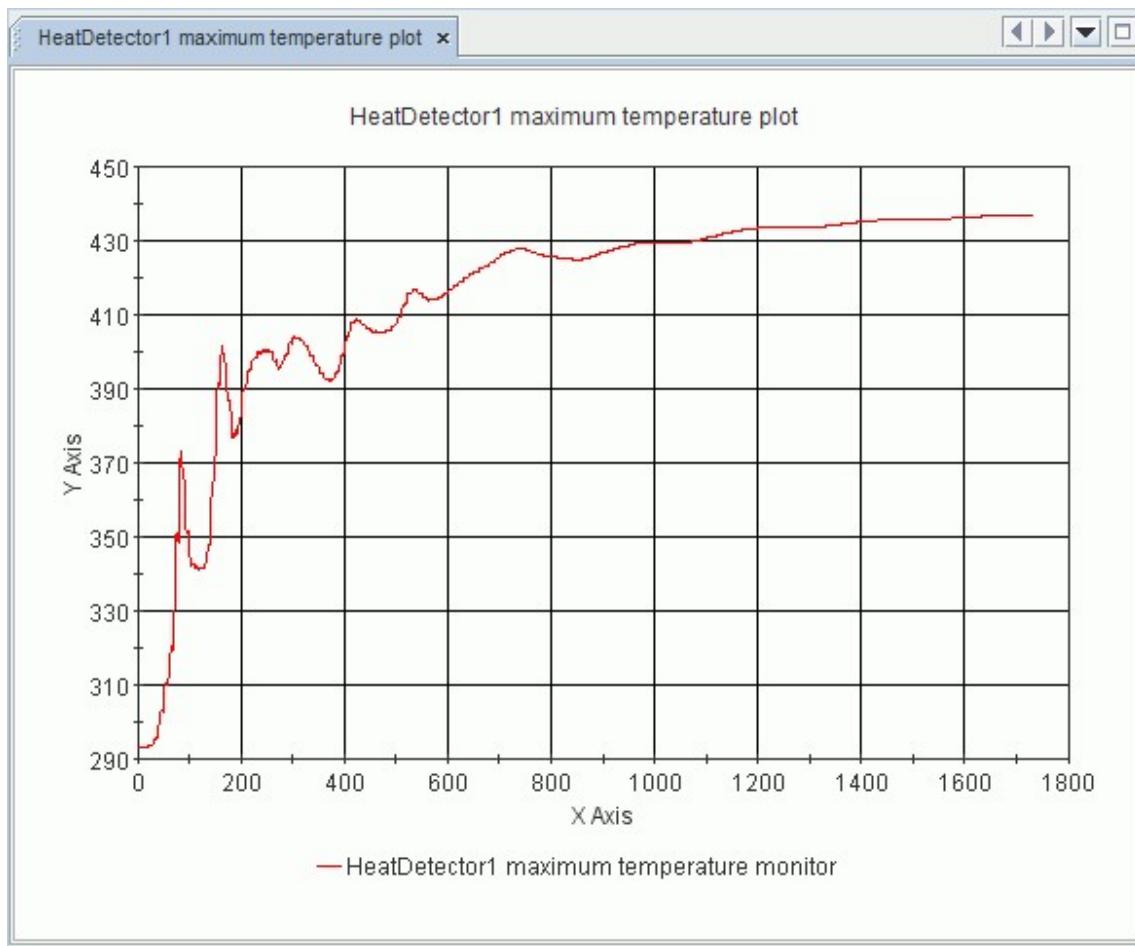
5. After you finish with the wizard by clicking **Apply**, a point derived part is created for each detector and appears in the object tree in the **Derived Parts** node.



Additionally, a report, monitor, and plot is created for each detector. These appear in the object tree in the **Reports**, **Monitors**, and **Plots** nodes respectively.



For heat detectors, the report monitors the maximum temperature at the probe location. The corresponding plot displays the temperature against iteration number.



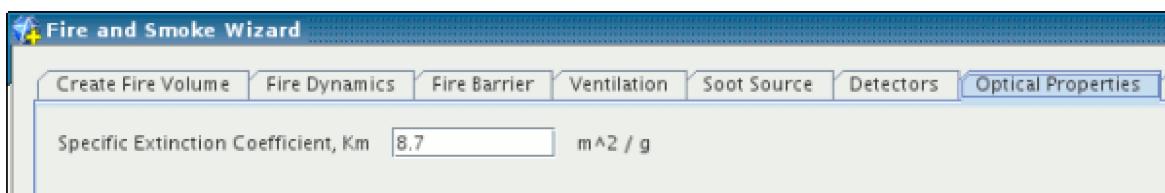
The report created for smoke detectors monitors the soot mass fraction at the probe location.

Specifying Optical Properties

The Optical Properties tab lets you set the specific extinction coefficient.

To specify optical properties:

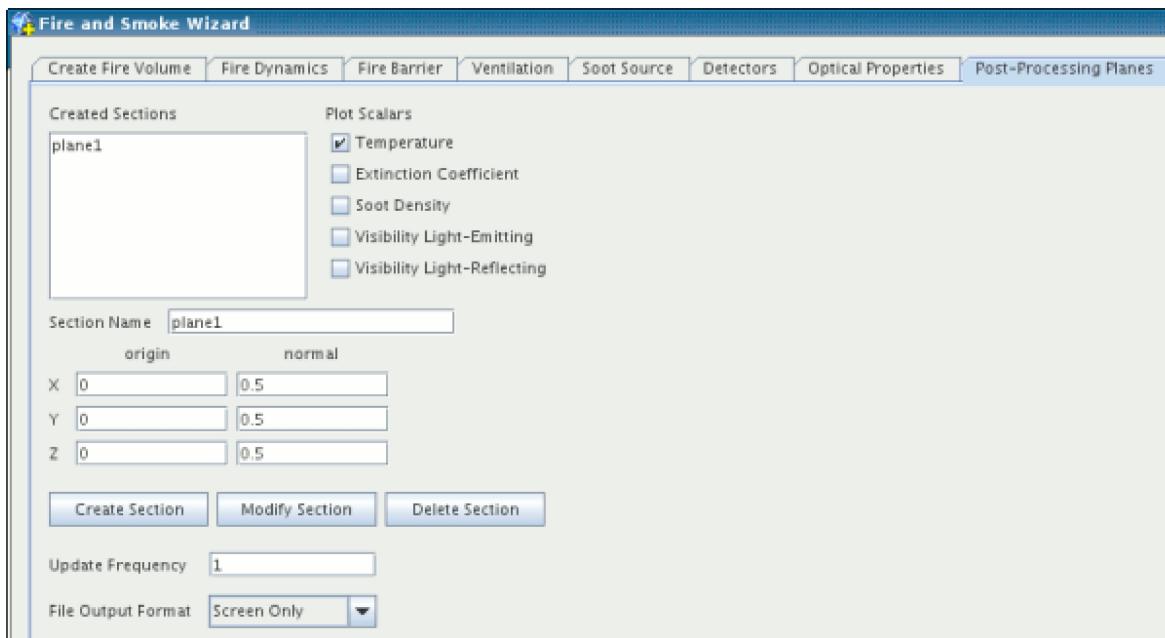
Enter a value, or accept the default, for the specific extinction coefficient.



Using Post Processing Planes

The Post-Processing Planes tab lets you create plane sections.

You can create plane sections with this tab of the wizard.



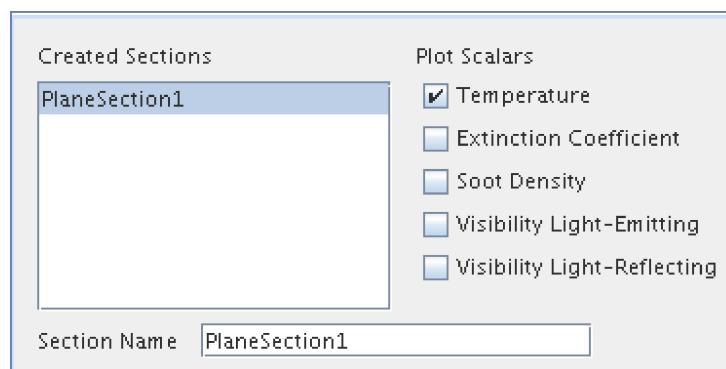
Begin by entering a name for the plane section in the **Section Name** text box.

Make your selection of scalar values to display by ticking the appropriate checkbox:

- **Temperature**
- **Extinction Coefficient**
- **Soot Density**
- **Visibility Light-Emitting**
- **Visibility Light-Reflecting**

Enter the appropriate coordinates for the origin and normal.

Once you have finished, you can create the plane section by clicking the **Create Section** button. The name of the section appears in the Created Sections list.



The plane section also appears in the object tree within the **Derived Parts** node.

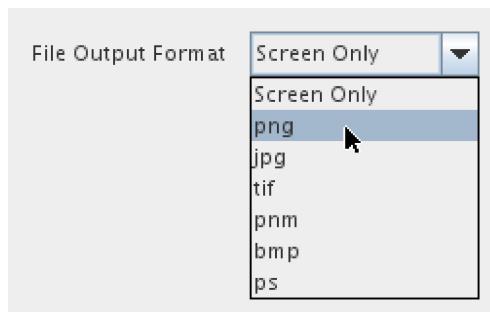


It is also possible to manage the plane sections in the list:

- To change a section, select its name in the list, make changes, and click **Modify Section**.
- To create another section similar to an existing one, select the name of the section in the list, make changes to it, give it a new name and click **Create Section**. An additional section appears in the list. If you did not name it, a number 2 is added to the original section's name by default.
- To delete a section, select it in the list and click **Delete Section**.

The **Update Frequency** setting lets you control how often the scene containing the data should be updated during the solver run. The default setting is 1, or each iteration.

Finally, you can select a file format for output of the displayed data to an image file.



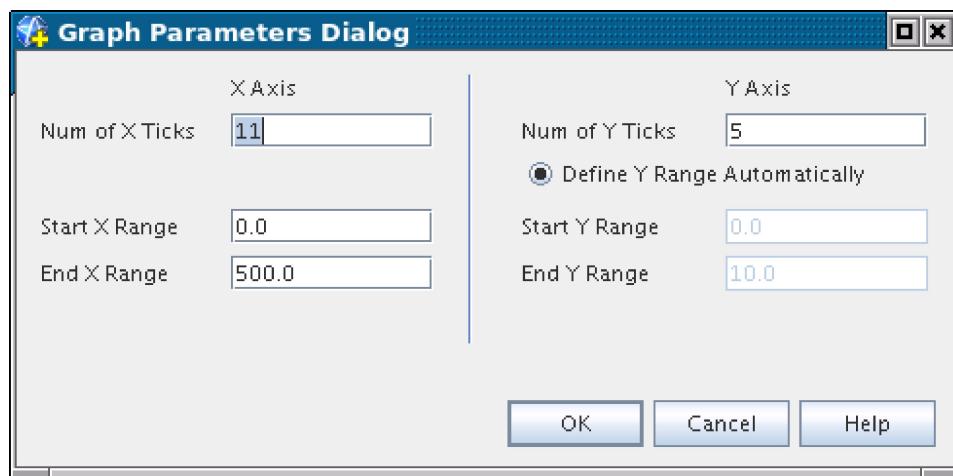
Setting Graph Parameters

This feature in the fire and smoke wizard lets you customize the monitor plots that it automatically generates.

To control the parameters of a plot:

1. Click the **Set Graph Parameters** button in the *Fire Dynamics* or *Ventilation* tabs of the wizard.

This opens the *Graph Parameters* dialog.



2. Enter your specifications and click **OK**.