Volume Rendering: Steckler Room

This tutorial demonstrates the advanced post-processing capabilities of Simcenter STAR-CCM+ using the simulation file that was created in the tutorial, Fire and Smoke Wizard: Steckler Room. If your graphics card does not support volume rendering, activate advanced rendering for the relevant scene.

The procedure includes adjusting scalar values, creating a resampled volume derived part, setting up a custom color map, adjusting lighting, rerunning the transient solution, and capturing transient results using the solution recording module.

Contents:

Prerequisites

The instructions in the Volume Rendering tutorial assume that you are already familiar with certain techniques in Simcenter STAR-CCM+.

Before attempting this tutorial, make sure that you are familiar with the following techniques:

Technique	Tutorial
The Simcenter STAR-CCM+ workflow	Introduction to STAR-CCM+
Using visualization tools, scenes, and plots	Introduction to STAR-CCM+

Loading an Existing Simulation

For this tutorial, use an existing simulation file that contains objects and solution data from the tutorial, Fire and Smoke Wizard: Steckler Room.

To start this tutorial, you can either:

- use your own version of the simulation file from the Steckler Room tutorial, if you have already worked through it; or,
- use the version from the Tutorials Files package located on the Support Center portal, https://support.sw.siemens.com/en-US/product/226870983/downloads. To obtain the package:
  1. Follow the instructions in the section, Downloading the Tutorial Files.
  2. After extracting the package to disk, copy the file, `tutorials/reactingFlow/FireAndSmokeWizardSteckerRoom_final.sim`, into the working directory.
  3. Rename the file to `roomfire.sim`.

To open the simulation file:

1. Launch Simcenter STAR-CCM+.
2. Select **File** > **Load...**
3. In the *Load a File* dialog, click **Browse...**
4. In the *Open* dialog, navigate to the directory that contains `roomfire.sim`.
5. Select `roomfire.sim` then click **Open**.
6. In the *Load a File* dialog, click **OK**.
7. Save the simulation as `FireAndSmokeResampled.sim`.

Adjusting the Scalar

One application of volume rendering is for visualizing the distribution of scalar quantities that represent combustion products.

In the reference tutorial, the Soot Density function provides an estimate of the amount of soot that a room fire generates. The function is defined with a built-in conversion to mg/m^3 as:

```
${Passive Scalar}*$Density*1000*1000
```

For this tutorial, the function is modified to compute soot density in kg/m^3, so that a custom unit can be applied afterward in the scalar scene:

1. Select the **Automation** > **Field Functions** > **Soot Density** node.
2. Set the *Dimensions* property to **Mass/Volume**.
3. Set *Definition* to `${Passive Scalar}*$Density`.

Create a custom unit:

4. Right-click the **Tools** > **Units** node and select **New** > **Units**.
5. Rename the **Units_1** node `mg/m^3`.
6. With the **mg/m^3** node selected, set *Conversion* to `1.0E-6`.
7. Set *Description* to `mg/m^3`.

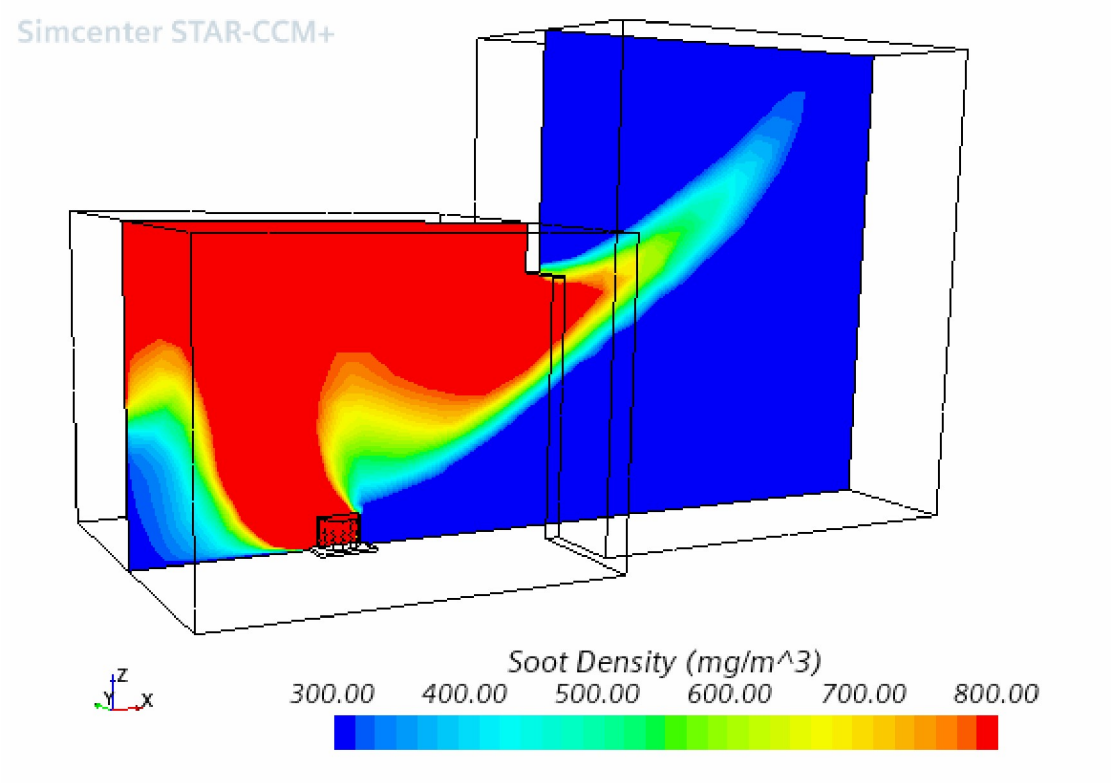
8. Set *Dimensions* to **Mass/Volume**.

To get a sense of the range of the scalar, use the scalar displayer.

9. Open **Scenes** > **Scalar Scene 1**.
10. Edit the **Scalar Scene 1** > **Scalar 1** > **Scalar Field** node and set the following properties:

Property	Setting
Function	**Soot Density**
Clip	**Off**
Units	**mg/m^3**
Min	300.0
Max	800.0

The scalar scene now shows an approximation of the scalar variation in the volume.



Setting Up the Resampled Volume

To activate volume rendering, create a resampled volume derived part.

Volume resampling is a technique for rendering volume at a relatively low computational cost. A key element of this technique is the voxel, a pixel that represents volume in three-dimensional space.

In the dialog that you use for setting up a resampled volume derived part, the Voxel Count preview lets you decide how to balance quality and computing costs. A larger number of voxels produces better volume rendering, but takes more time. A smaller number saves time but reduces quality. The recommended starting voxel count is between 1 million and 10 million.

1. To create a resampled volume, right-click the **Derived Parts** node and select **New Part** > **Resampled Volume...**
2. In the dialog, set the following properties:

Property	Setting
Input Parts	**Room**, **Fire**
Cell Size to Voxel Ratio	`1.0`
Display	**New Scalar Displayer**

   Accept the other default settings.
3. To see what the resampled volume looks like, and to get the voxel count, click **Preview**.
   Beneath the **Preview** button, the voxel count appears: 3.5 million.
4. Click **Create**, then click **Close**.
   A new derived part named **Resampled Volume** is created in the **Derived Parts** manager node.

Setting Up a Color Map

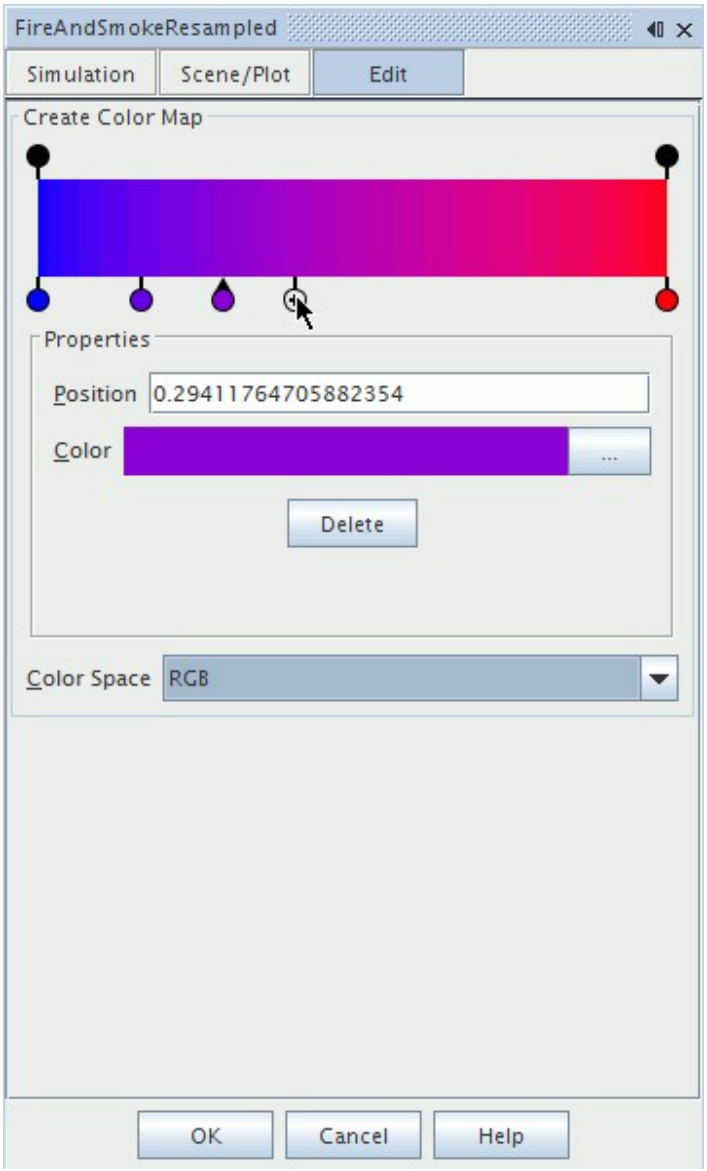
Prepare a custom color map for the resampled volume.

The color map maps scalar values to a set of colors for interpolating the results. This example includes a color map to make volumes of higher soot density more obvious in two ways:

- Highlighting the scalars of interest by customizing the color choices -- colors that are isoluminant, or nearly so, make it easier to perceive change in the scalar.
- Using transparency to emphasize some changes in the scalar range

Color settings for control points are specified as RGB (Red, Green, Blue) in this example. RGB specification is the most common.

1. Expand the **Tools** node and right-click the **Color Maps** node. Select **New** from the pop-up menu.
2. In the *Create Color Map* dialog:
   a) Set **Color Space** to **RGB**.
   b) Add three color control points by clicking beneath the color spectrum for each one.



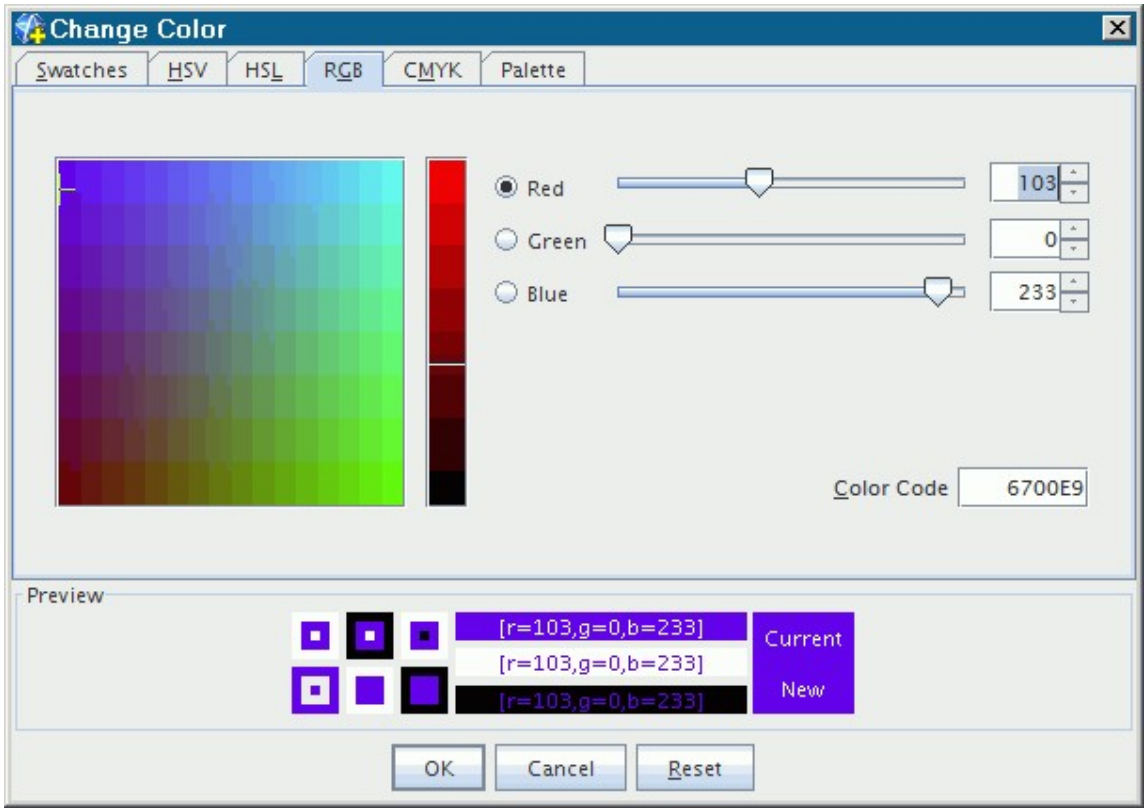
A total of five color control points are beneath the spectrum.

c) Select the new control points one at a time and set the *Position* property to `0.25`, `0.5`, and `0.75`, respectively.

You are ready to edit the color of each control point.

d) To edit a color control point, double-click it.

e) In the *Change Color* dialog, select the *RGB* tab and enter values for *Red*, *Green*, and *Blue* for the respective control point as listed in the table below. After you finish the color settings for each control point, click **OK**.

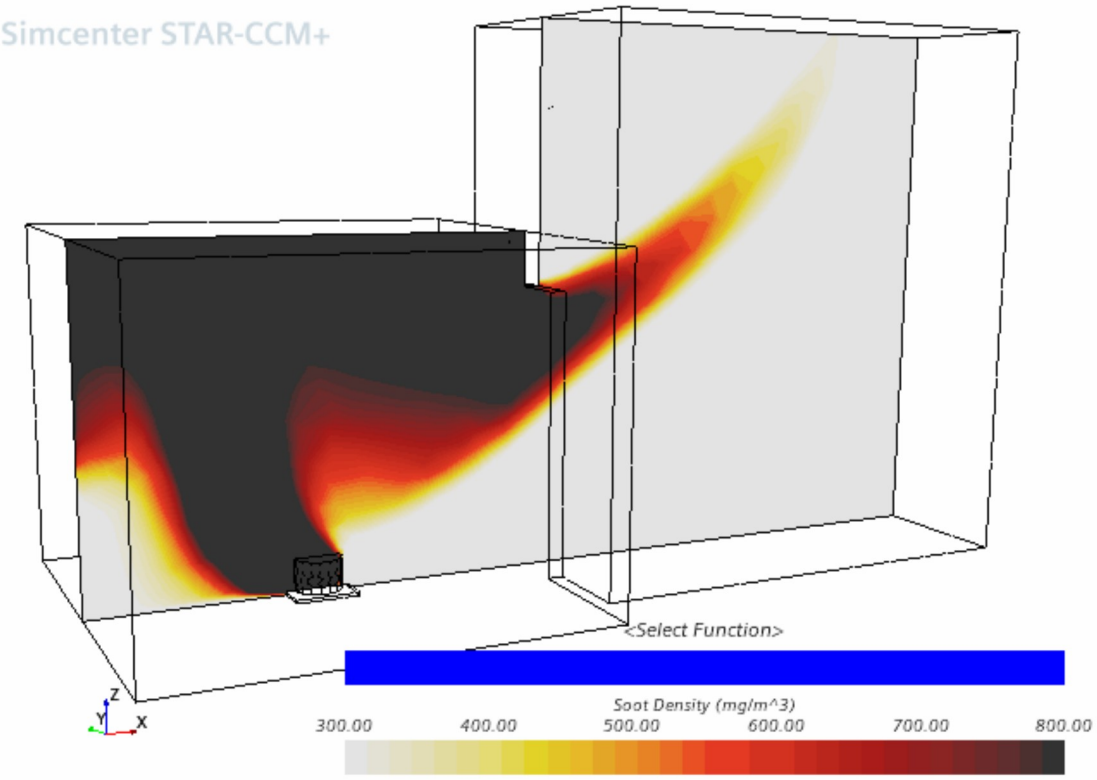


Position	Red	Green	Blue
Left Limit	231	231	231
0.25	229	216	41
0.5	236	61	37
0.75	167	26	25
Right Limit	51	51	51

f) Click **OK**.

3. Rename the new **user 1** color map node, `soot_ramp`.
4. To preview the soot density color map in the plane section, select the **Scalar Scene 1** > **Scalar 1** > **Color Bar** node and set *Color Map* to **soot_ramp**.

   The soot density colors appear in the scalar scene.



5. Save the simulation.

Viewing the Volume-Rendered Soot Density

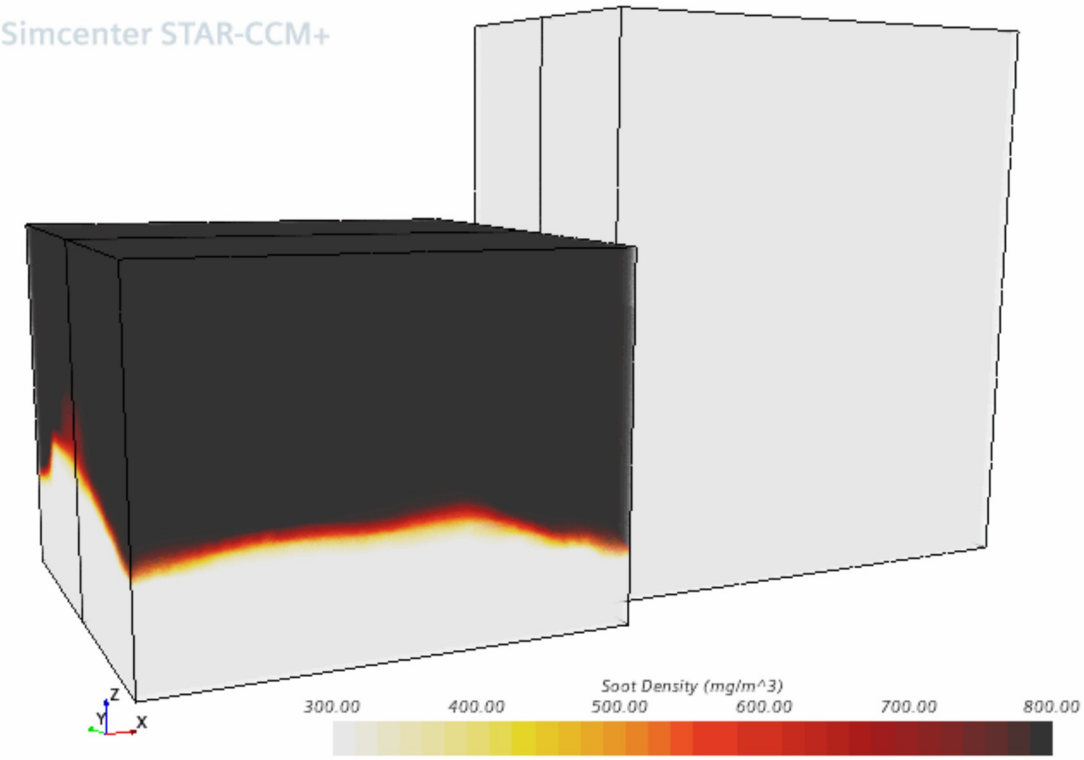
Use the resampled volume derived part for rendering the volume of the soot density.

Switch scalar displayers to begin working with the resampled volume.

1. Right-click the **Scalar Scene 1** > **Scalar 1** node and select **Toggle Visibility**.
2. Edit the **Resampled Volume Scalar 1** node and set the following properties:

Node	Property	Setting
Resampled Volume Scalar 1	*Representation*	**Volume Mesh**
└ **Scalar Field**	*Function*	**Soot Density**
	Clip	**Off**
	Units	**mg/m^3**
	Min	300.0
	Max	800.0

3. Select the **Color Bar** node and set *Color Map* to **soot_ramp**.
   At this point, the room is filled with the volume rendered resampled part.

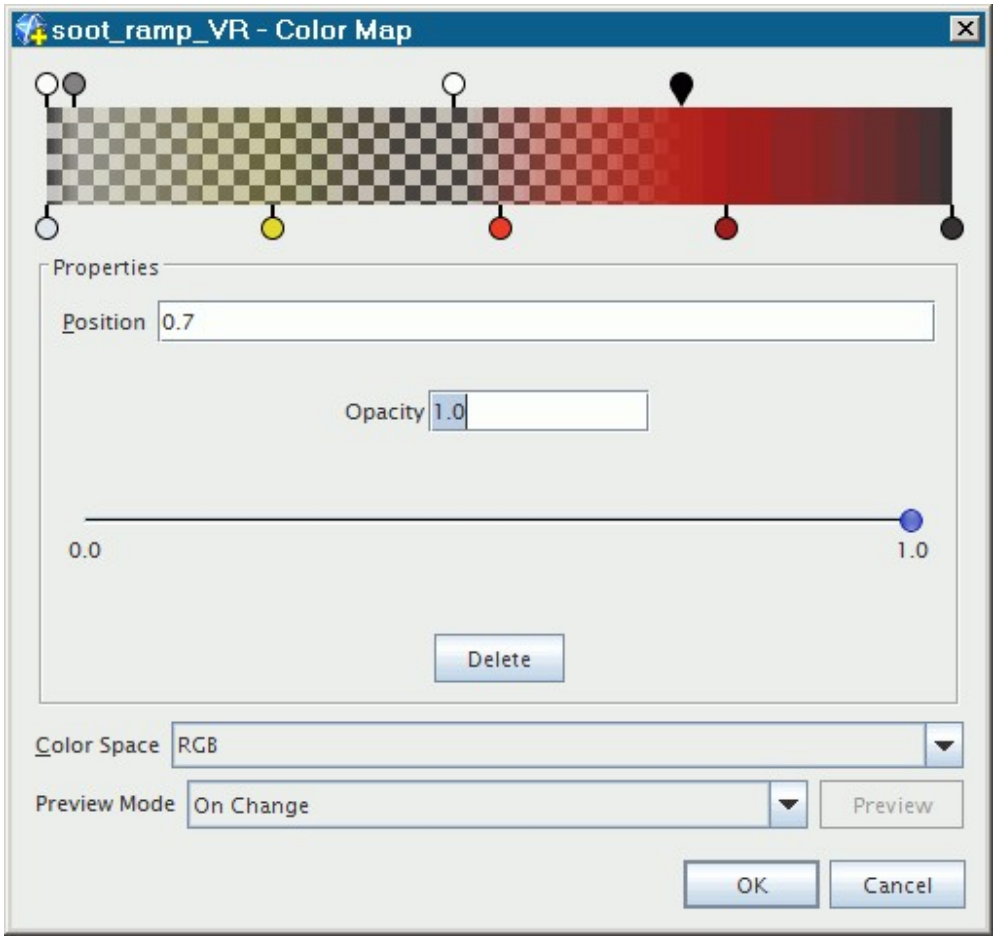


Modifying the Color Map Settings

To determine exactly where in the room the soot density is highest, change the opacity of the color map.

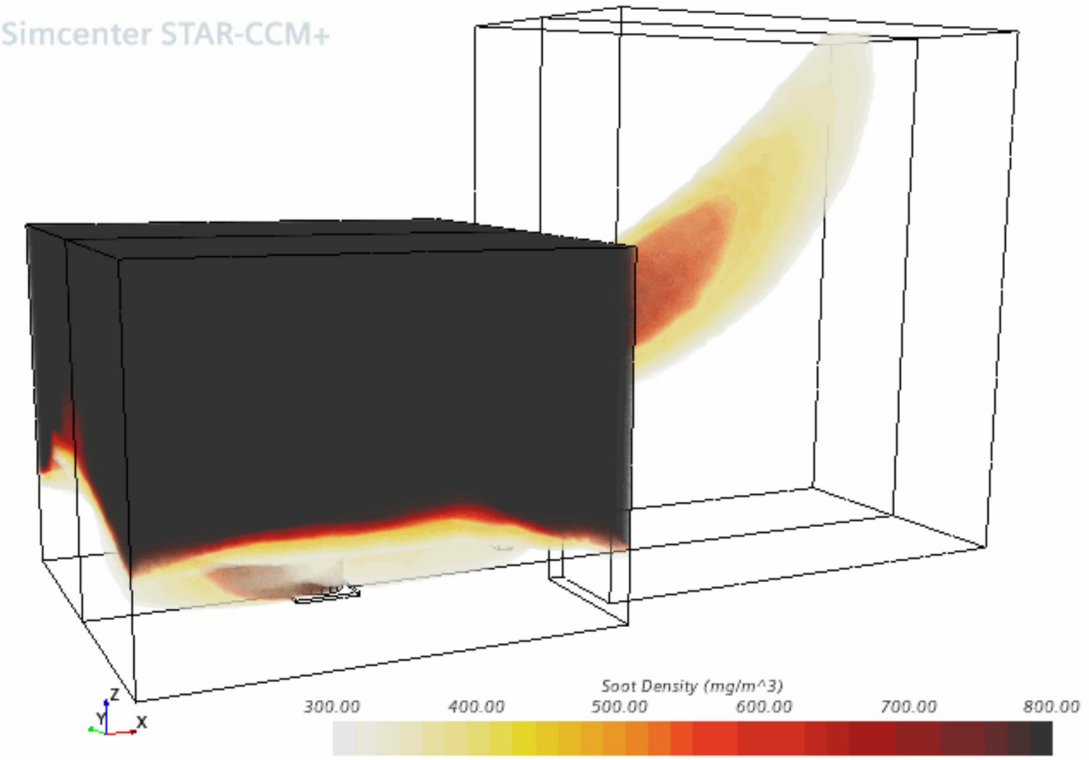
1. In **Tools** > **Color Maps**, create a copy of the **soot_ramp** color map node.
2. Rename the new color map `soot_ramp_VR`.
3. To edit the **soot_ramp_VR** node, click ... (**Custom Editor**) for the *Color Map* property.
4. In the *soot_ramp_VR - Color Map* dialog:
   a) Add two opacity control points by clicking above the color spectrum for each one.
   A total of four opacity control points are above the spectrum.
   b) Select each control point, beginning with the one at the far left, and set its *Position* and *Opacity*:

Opacity Control Point	Position	Opacity
First	0.0	0.0
Second	0.03	0.5
Third	0.45	0.0
Fourth	0.7	1.0



c) Click **OK**.

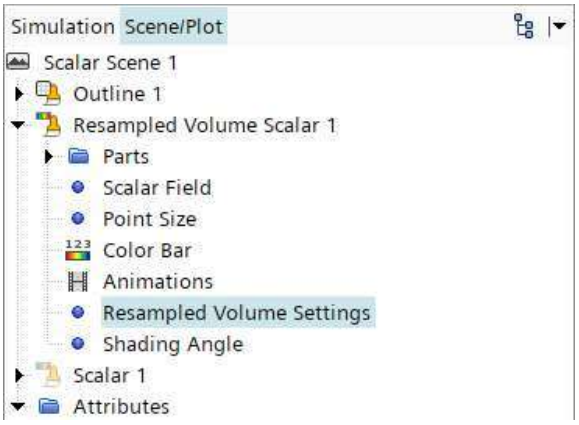
5. To preview the soot density color map in the plane section, select the **Scalar Scene 1** > **Resampled Volume Scalar 1** > **Color Bar** node and set *Color Map* to **soot_ramp_VR**.
   The scene uses the new color set.



Adjusting the Lighting

To improve the appearance of the volume-rendered object, change lighting in the volume rendering settings and the displayer.

1. Select the **Resampled Volume Scalar 1** > **Resampled Volume Settings** node.



2. Set the following properties:

Property	Setting
Lighting Mode	**Local Lighting**
Quality	`0.8`

3. Select the **Resampled Volume Scalar 1** node and click ... (**Custom Editor**) for the *Lighting* property.
4. In the *Resampled Volume Scalar 1 - Lighting* dialog, set the following:

Property	Setting
Specular	`0.7`
Specular Power	`25.0`

5. Click **OK**.

Setting Solver Parameters and Stopping Criteria

Prepare the simulation for a new analysis run in order to see how the soot cloud changes over time.

Change the time-step and run time as follows:

1. Select the **Solvers** > **Implicit Unsteady** node and set *Time-Step* to `0.1` s.
2. Select the **Stopping Criteria** > **Maximum Physical Time** node and set *Maximum Physical Time* to `25.0` s.

Setting Up the Solution History File

From the new transient analysis, you want to obtain an animation of the soot cloud developing over time. In order to generate the input frames for the animation, create a solution history file and use it to store selected solution data at specified time intervals.

1. Right-click the **Solution Histories** node and select **New...**.
2. In the *Save* dialog, enter `FireAndSmokeResampledData.simh` as the name of the solution history file and click **Save**.

   Within the **Solution Histories** node, Simcenter STAR-CCM+ adds a new node containing the name of the solution history file.

   The red asterisk next to this node means that data is actively written to the file when the simulation runs.

   Another new node, **FireAndSmokeResampledData**, is added to the **Solution Views** node.

Choose what data to save to the solution history file. As this is a fire and smoke case it is appropriate to store results for soot density:

3. Select the **Solution Histories** > **FireAndSmokeResampledData** node and set *Functions* to **Density**, **Passive Scalar**, and **Soot Density**.
4. Set *Regions* by selecting **Fire** and **Room**.

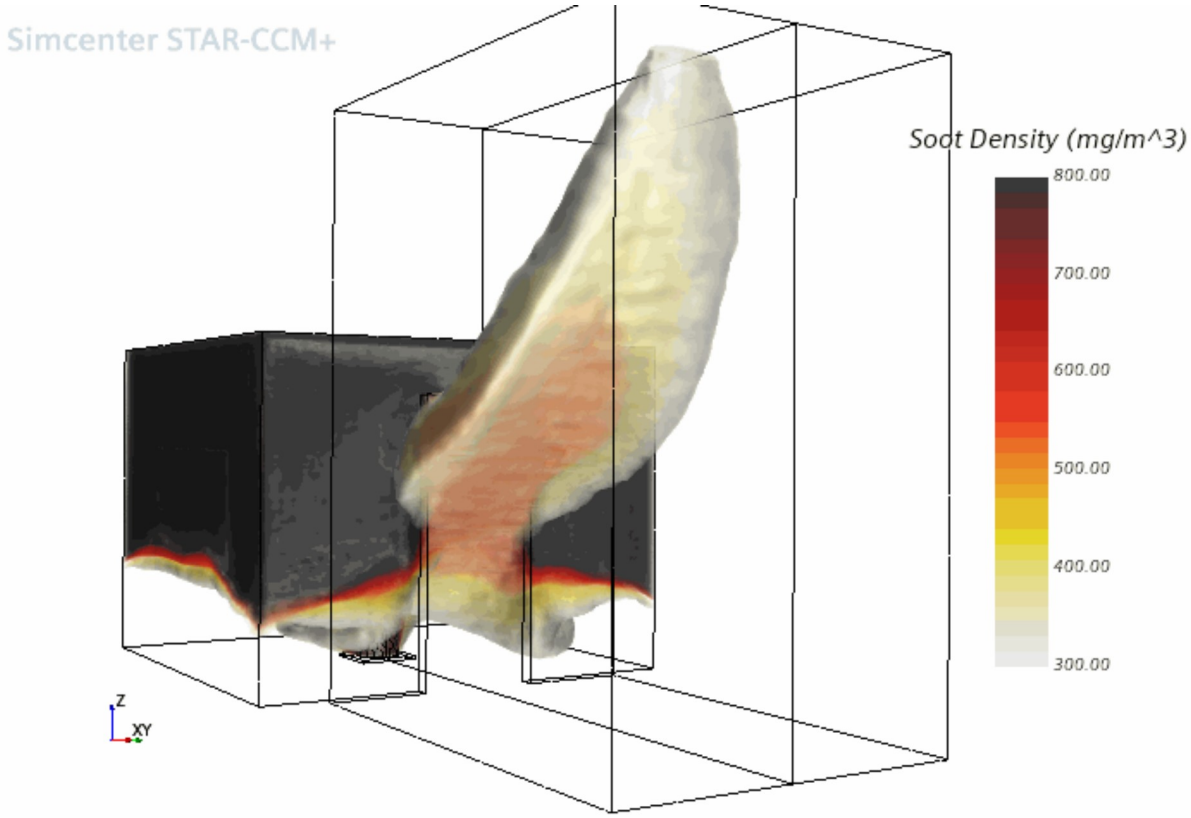
Set the frequency with which the selected data is written to the solution history file:

5. Select the **Solution Histories** > **FireAndSmokeResampledData** > **Update** node and set *Trigger* to **Time Step**.
6. Select the **Update** > **Time-Step Frequency** node and make sure that *Frequency* is set to `1`.
7. Select the **Solution Histories** > **FireAndSmokeResampledData** node.
   The *Auto-record* property records the data to the solution history file at the required intervals. The *States* property displays the number of saved states stored in the selected solution history file; currently this is displaying 0 as the solver has not yet run.
8. Save the simulation.

Visualizing the Solution

Rearrange the scalar scene for an effective view angle and add a time annotation.

1. Rearrange the color bar and display to provide the following view.



Add an annotation displaying solution time to the scene:

2. Expand the **Tools** > **Annotations** node and drag the **Solution Time** node into the scene.
   The Solution Time annotation appears at the bottom left of the scene.
3. Drag the annotation to a more visible position in the scene.

Running the Simulation

Clear the results of the previous simulation and run the simulation again.

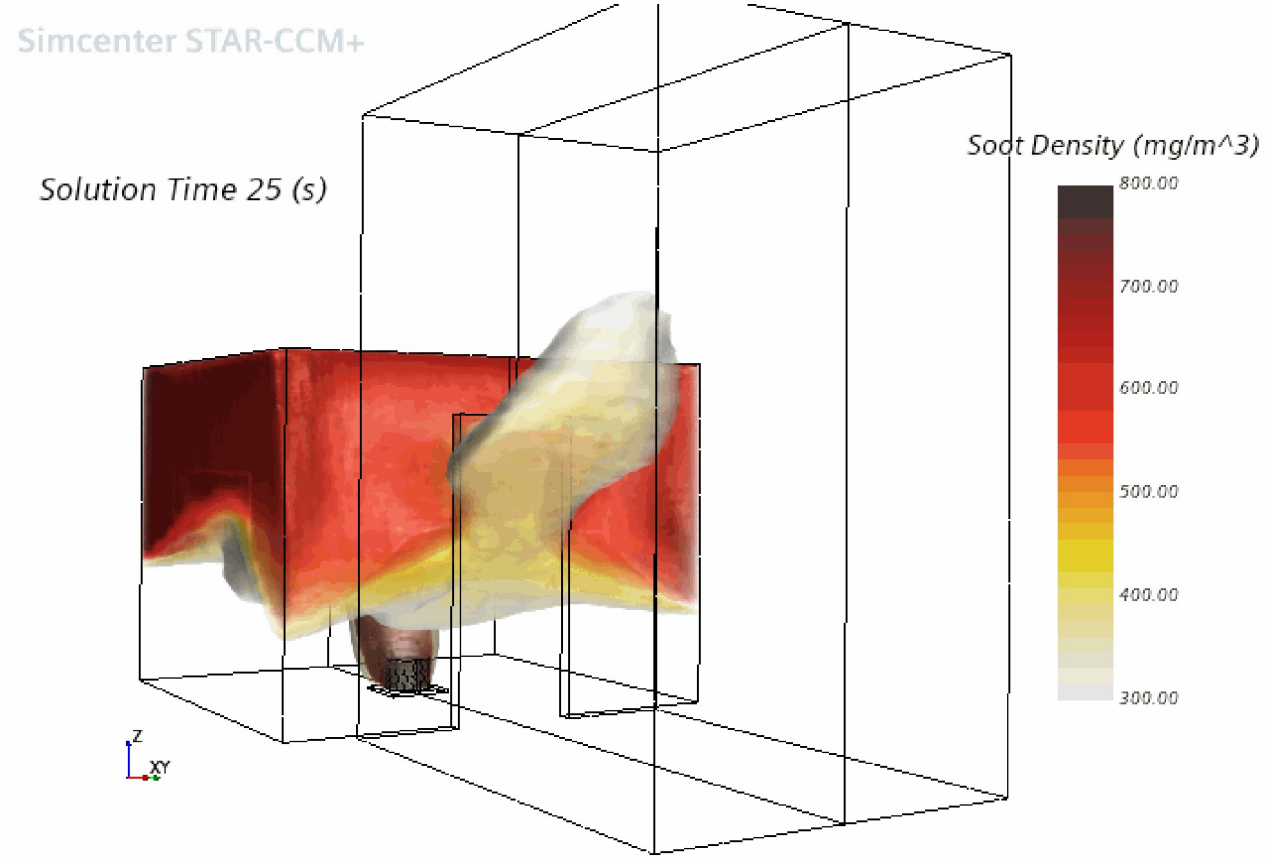
To clear the results and run the simulation again:

1. In the menu, select **Solution** > **Clear Solution**.
2. In the *Clear Solution* dialog, click **OK**.
3. Click (**Run**) in the *Solution* toolbar.
4. When the simulation has finished running, save it.

Visualizing the Results

When the simulation run is finished, the scalar scene shows the complete volume rendering of the soot density.

Take a look at the scalar scene.



Creating a Recorded Solution View

Solution views are used to interrogate the solution history data and make it available for post-processing.

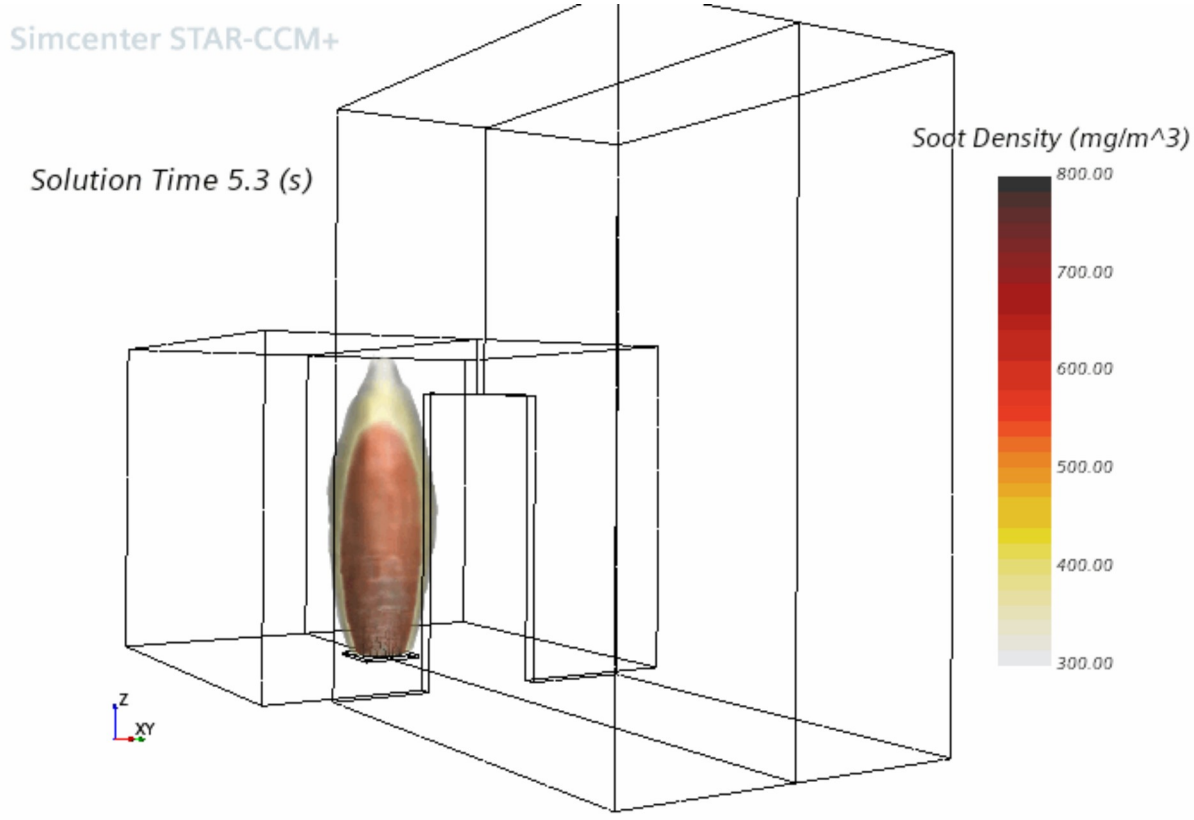
Properties of the solution view set the point in the solution history at which data is read. Data is read into a separate representation linked to the solution view. The solution history file contains all the data that was specified in the previous part of the tutorial.

1. Click the **Scalar Scene 1** tab in the **Graphics** window.
2. Drag and drop the **Solution Views** > **FireAndSmokeResampledData** node onto a blank area in the scene window.

   This action sets the *Representation* property for all displayers (**Outline 1**, **Resampled Volume Scalar 1**, and **Scalar 1**) and solution history annotation (**Solution Time**) in the scene to **FireAndSmokeResampledData**.

   This scene corresponds to the data stored in the solution history file for the first time-step. Adjust the solution time to display the solution data at 5.3 seconds.
3. Select the **Solution Views** > **FireAndSmokeResampledData** node and set the *Solution Time* property to approximately 5.3 seconds.

   The scene updates and the annotation indicates the time that you selected.



4. Save the simulation.

Creating an Animation from the Solution View

Create an animation that shows the spread of soot in the room.

1. Select the **Solution Views** > **FireAndSmokeResampledData** > **Animation** node and set the *Animation Mode* to **Solution Time**.

A new node, **Solution Time Animation**, appears below the **Animation** node.

You have 250 states and a physical time of 25 seconds. Knowing this, you can calculate that 10 frames make up one second of simulation time.
Adjust the framerate for the animation.

2. Click the **Scene/Plot** tab in the explorer pane.
3. Select the **Scalar Scene 1** > **Attributes** > **Scene animation** node and set the *Target Frame Rate (fps)* to `10`.

Record the animation to a video file.

4. Click  (**Write a movie of the animation in the current scene**)
5. In the *Write Animation* dialog, set the following properties:

Property	Value
Animation Length	`25`
Size	A resolution of your choice
File Name	`sootAnimation.mp4`

6. Click **Save** to write the animation to disk.
7. Play back the animation using a player of your choice.

Summary

This tutorial demonstrated volume rendering and other advanced post-processing features in Simcenter STAR-CCM+.

The features that were introduced include:

- Refining the settings of a scalar in a scene.
- Setting up a resampled volume derived part.
- Preparing a custom color map.
- Viewing volume-rendered data.
- Adjusting display lighting.
- Creating a Solution History file.
- Creating a Recorded Solution View.
- Viewing recorded solution data in a scene.
- Recording an animation from the Solution History file.