

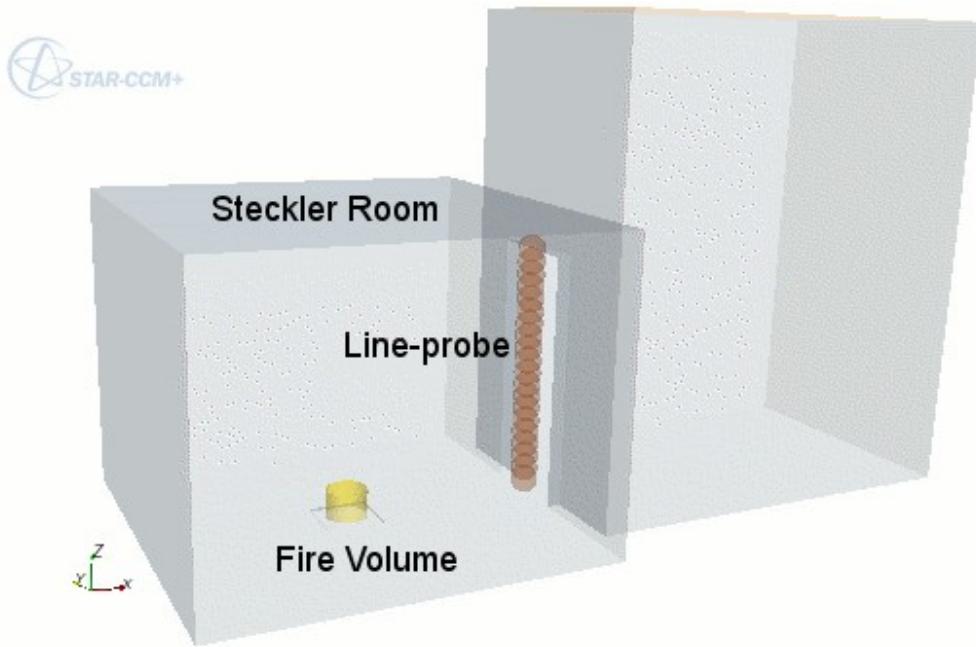
## Fire and Smoke Wizard: Steckler Room

This tutorial demonstrates the Fire and Smoke Wizard. The geometry and fire settings closely resemble those Steckler reported, including radiation and wall heat absorption. This setup allows a preliminary comparison with experimental data.

Before you begin, take note of the following important conventions in the **Fire and Smoke Wizard**:

- Gravity is in negative z-direction (z pointing up).
- Reference Density is 1.2 Kg/m<sup>3</sup>.
- Reference Altitude is 0, 0, 0 m.

The problem geometry is shown below.



In this scene, the location of the line-probe corresponds to the experimental measurement stack.

### Contents:

- [Prerequisites](#)
- [Importing the Geometry](#)
- [Assigning Parts to Regions](#)
- [Generating the Volume Mesh](#)
- [Setting up Fire Properties](#)
- [Reviewing Model Settings](#)
- [Setting up Solver Parameters and Stopping Criteria](#)
- [Visualizing the Solution](#)
- [Running the Simulation](#)
- [Analyzing Results](#)

[Settings for Steckler Room Fire Validation](#)  
[Comparison with Measurements](#)  
[Adjusting Stopping Criteria and Continuing](#)  
[Analyzing Validation Results](#)  
[Summary](#)

## Prerequisites

The instructions in the Fire and Smoke Wizard: Steckler Room tutorial assume that you are already familiar with certain techniques in Simcenter STAR-CCM+.

Technique	Tutorial
The Simcenter STAR-CCM+ workflow	<a href="#">Introduction to Simcenter STAR-CCM+</a>
Using visualization tools, scenes and plots	<a href="#">Introduction to Simcenter STAR-CCM+</a>

If you have not already done so, download the tutorial files bundle. See [Downloading the Tutorial Files from the Support Center Portal](#).

## Importing the Geometry

To set up the Simcenter STAR-CCM+ simulation, launch a simulation and import the supplied geometry.

1. Create a directory for the tutorial called `stecklerFire`.
2. Navigate to the `reactingFlow` folder of the downloaded tutorial files and copy the following files to your working directory:
  - `stecklerRoomFire.x_t`
  - `DoorUvelExpt.csv`
  - `DoorTempExpt.csv`
3. Launch Simcenter STAR-CCM+.
4. Start a simulation.
5. Select **File > Import > Import Surface Mesh....**
6. In the *Open* dialog, navigate to your working directory, select `stecklerRoomFire.x_t`, and click **Open**.
7. In the *Import Surface Options* dialog, click **OK** to accept the default options.  
 The *Geometry Scene 1* display appears in the *Graphics* window.
8. Save the simulation as `roomfire.sim`.

## Assigning Parts to Regions

To define the computational domain, assign the parts to regions. In this tutorial, there are two regions, one for each part.

1. Expand the **Geometry > Parts** node and multi-select the **Fire** and **Room** nodes.

2. Right-click on one of the selections and choose **Assign Parts to Regions....**

3. In the *Assign Parts to Regions* dialog, select:

- a) **Create a Region for Each Part**
- b) **Create a Boundary for Each Part Surface**

4. Click **Apply** then **Close**.

Two new regions are created with boundaries that define the outlet and the interface between the two regions.

Set the outlet boundary as a Pressure Outlet:

5. Expand the **Regions > Room > Boundaries** node.

6. Select the **Outlet** node, and set *Type* to **Pressure Outlet**.

In the *Graphics* window, the surface that corresponds to this boundary changes to an orange color.

## Generating the Volume Mesh

Set up and generate the volume mesh.

1. Right-click the **Geometry > Operations** node and select **New > Mesh > Automated Mesh**.

2. In the *Create Automated Mesh Operation* dialog:

- a) Select **Fire** and **Room** from the *Parts* list.
- b) Select the following meshers, in order:

Group	Mesher
Surface Meshers	<b>Surface Remesher</b>
Core Volume Meshers	<b>Polyhedral Mesher</b>
Optional Boundary Layer Meshers	<b>Prism Layer Mesher</b>

c) Click **OK**.

An automated mesh operation is added to the **Operations** node.

3. Select the **Automated Mesh > Default Controls > Base Size** node and set *Base Size* to  $0.1\text{ m}$ .

4. Select the **Default Controls > Minimum Surface Size** node and set *Percentage of Base* to  $50.0\%$ .

This change prevents the mesher from creating small cells within the fire part.

5. Deactivate prism layers on the interface between the fire and the room:

- a) Right-click the **Automated Mesh > Custom Controls** node and select **New > Surface Control**.
- b) Select the **Custom Controls > Surface Control** node and set *Part Surfaces* to **Fire.Interface** and **Room.Interface**.

c) Select the **Surface Control > Controls > Prism Layers** node and set *Prism Layers* to **Disable**.

6. Force the volume mesher to create cells within the fire part that are similar in size to cells in the rest of the mesh:

- a) Right-click the **Automated Mesh > Custom Controls** node and select **New > Part Control**.
- b) Select the **Custom Controls > Part Control** node and set *Parts* to **Fire**.
- c) Select the **Part Control > Controls > Volume Growth Rate** node and set *Volume Growth Rate* to **Custom**.

- d) Select the **Part Control > Values > Volume Growth Rate** node and set *Volume Growth Rate* to 1.0.

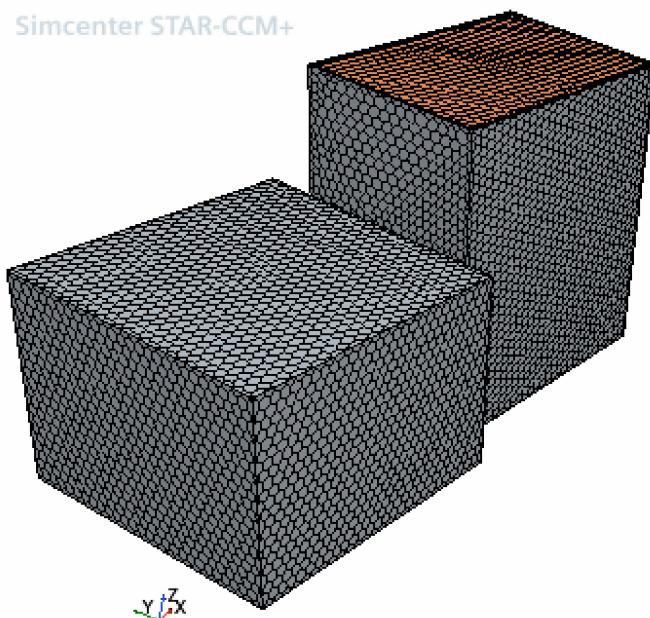
Generate the volume mesh:

7. Click  **(Generate Volume Mesh)**.

When the volume mesh is generated, the message **Volume Meshing Pipeline Completed** is displayed in the *Output* window. The numbers of Cells, Faces, and Vertices are also reported.

8. **Create a mesh scene.**

The volume mesh is displayed below.

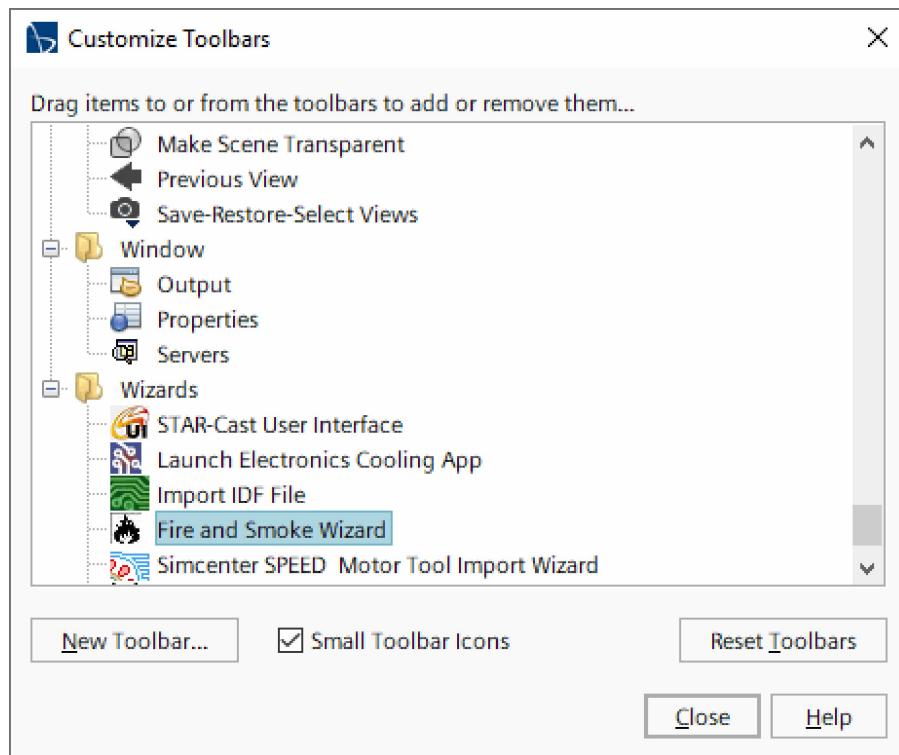


## Setting up Fire Properties

Set up the properties of the fire using the **Fire and Smoke Wizard**.

To access the wizard:

1. In the menu bar, select **Window > Toolbars > Customize**.
2. In the *Customize Toolbars* dialog, scroll down to locate the **Fire and Smoke Wizard**.

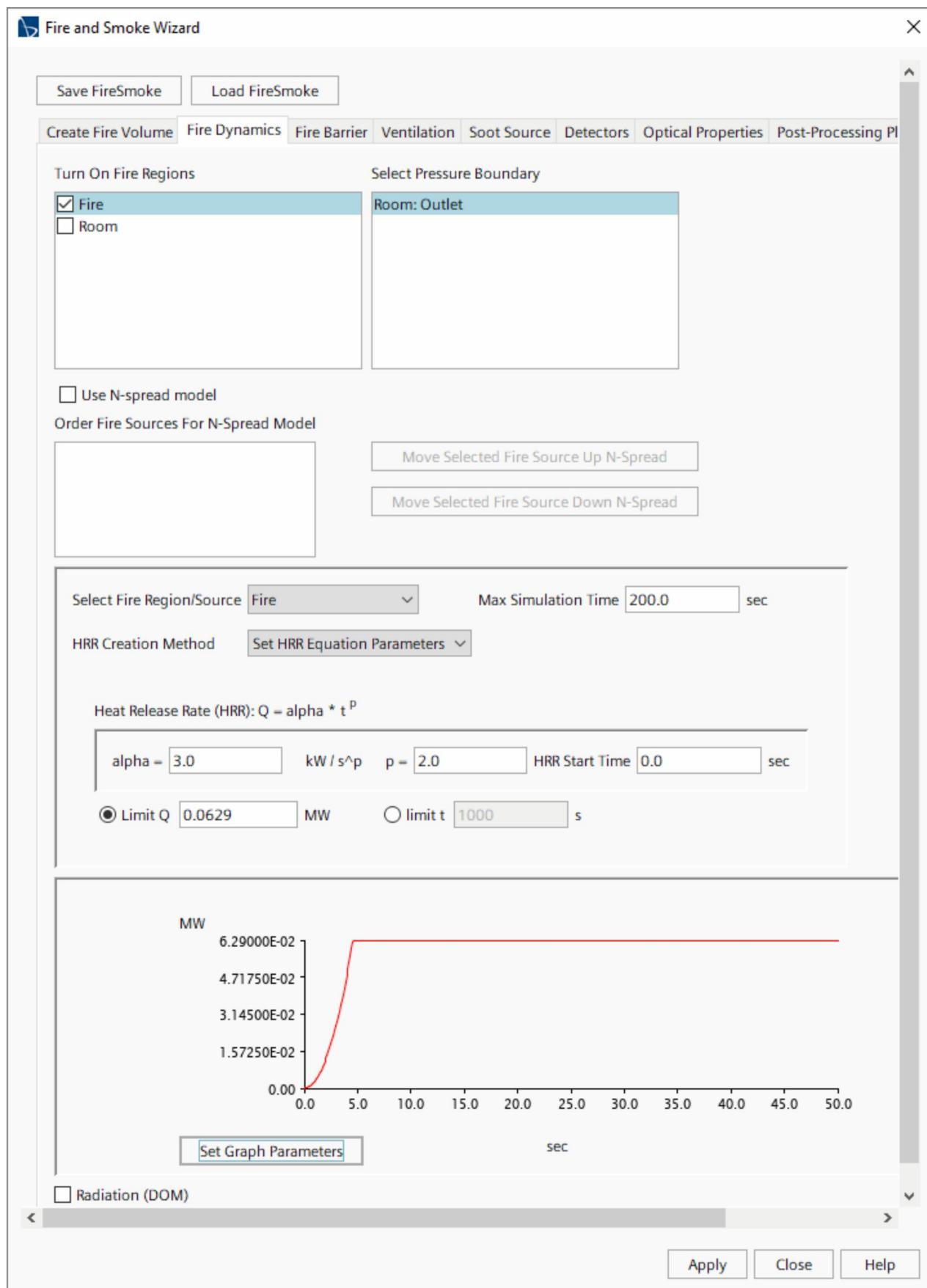


3. Drag and drop the **Fire and Smoke Wizard** icon onto the end of one of the toolbars near the top of the workspace.
4. In the *Customize Toolbars* dialog, click **Close**.
5. To open the *Fire and Smoke Wizard* dialog, click  (**Fire and Smoke Wizard**).
6. Click the *Fire Dynamics* tab and set the following properties:

Property	Setting
<i>Turn On Fire Regions</i>	<b>Fire</b>
<i>Select Pressure Boundary</i>	<b>Room: Outlet</b>
<i>Max Simulation Time</i>	200 sec
<i>alpha =</i>	3
<i>p =</i>	2
<i>Limit Q</i>	0.0629 MW

7. Click **Set Graph Parameters**.
8. In the *Graph Parameters Dialog*, set *End X Range* to 50 and click **OK**.

When these steps are complete, the dialog appears as follows:



9. Click **Apply**.

In this process, a fire source, height-varying pressure boundary setting, and scalar source are all set-up automatically.

10. Save the simulation.

## Reviewing Model Settings

---

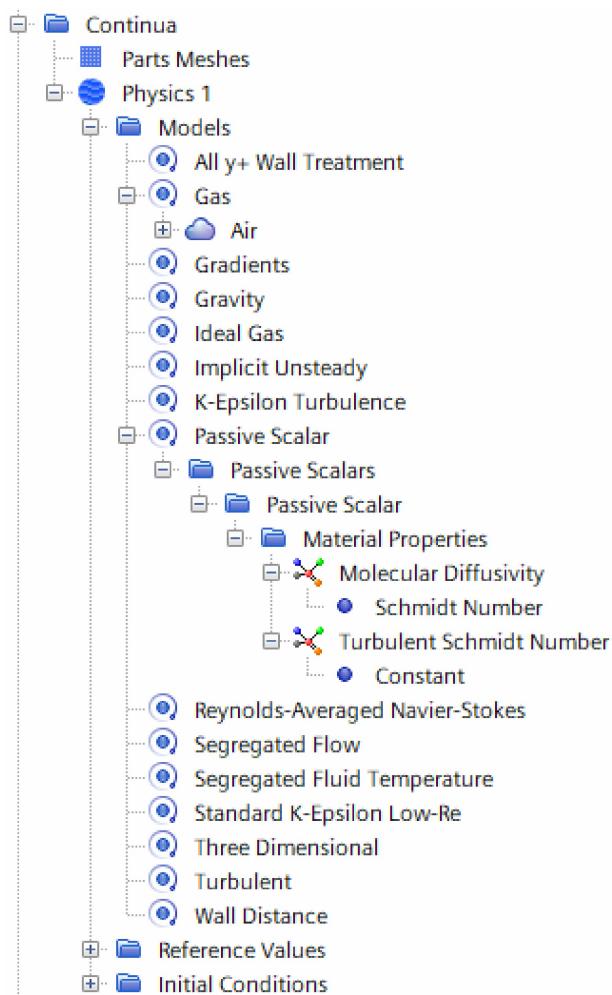
This tutorial describes an unsteady problem in which:

- A three-dimensional grid is employed.
- The fluid is a slightly compressible gas (air) that obeys the ideal gas equation of state.
- The flow is turbulent and non-isothermal.
- The  $k - \varepsilon$  low Re model is used for representing turbulence effects.
- The segregated flow solver is used.
- Gravity effects are taken into account.

All of the above conditions are automatically set up as properties of the fire.

To make sure that the model setup is accurate, you can inspect the selected models and edit them as necessary:

Open the **Continua > Physics 1 > Models** node.



In this case, the model setup is correct and no change is required.

## Setting up Solver Parameters and Stopping Criteria

Before running the analysis, specify the maximum physical time over which the simulation is run.

Set the time step and the temporal (and spatial) discretization schemes. Also reduce the maximum number of iterations that are carried out for each time step.

1. Select the **Solvers > Implicit Unsteady** node and set *Time-Step* to  $1.0 \text{ s}$ .
2. Select the **Stopping Criteria > Maximum Inner Iterations** node and set *Maximum Inner Iterations* to 5.

## Visualizing the Solution

To view the progress of the simulation as it runs, set up scalar scenes using a section plane.

Create a scalar scene to display the temperature on the plane section.

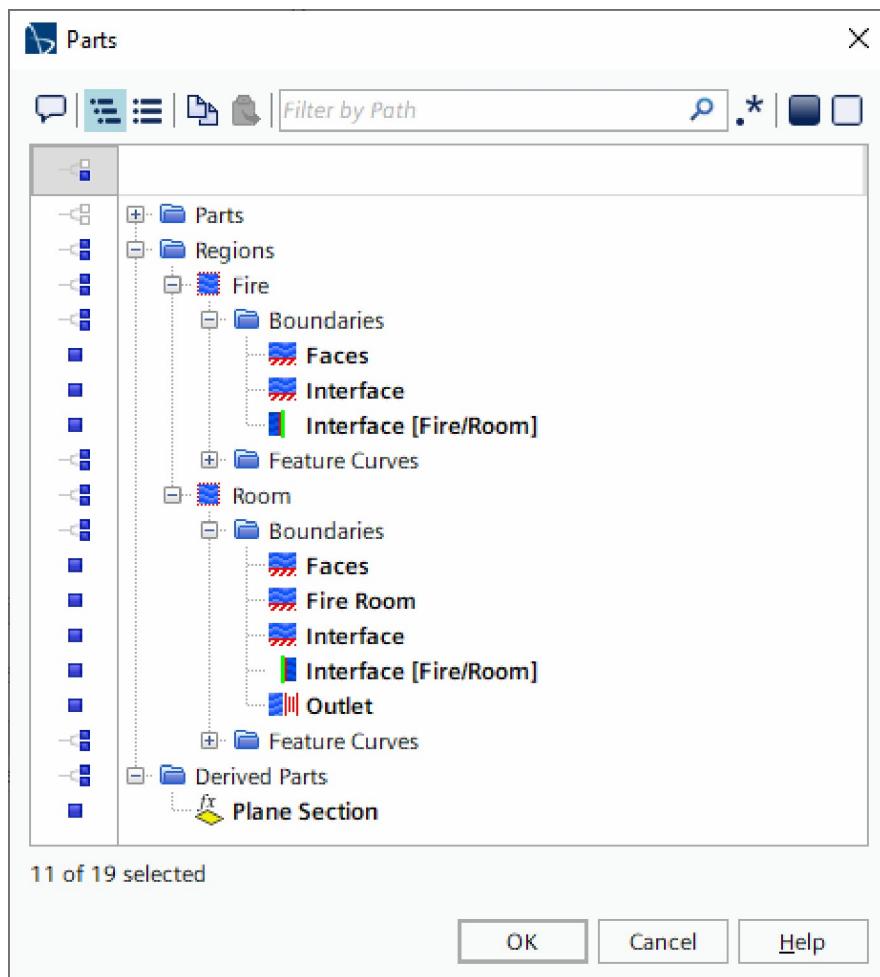
1. Create a scalar scene.

Create the plane section:

2. Right-click the **Derived Parts** node and select **New > Section > Plane Section**, and set the following properties:

Property	Setting
<i>Input Parts</i>	<b>Fire</b>
	<b>Room</b>
<i>Origin</i>	[0.0, 0.0, 0.0] m
<i>Normal</i>	[0.0, 1.0, 0.0] m
<i>Display</i>	<b>Existing Displayer &gt; Scalar 1</b>

3. Click **Create** then click **Close**.
4. Right-click the **Scalar Scene 1 > Outline 1 > Parts** node and select **Edit**.
5. In the *Parts* dialog, activate the  (**Display as a Tree**) option near the top, and select all regions and derived parts.

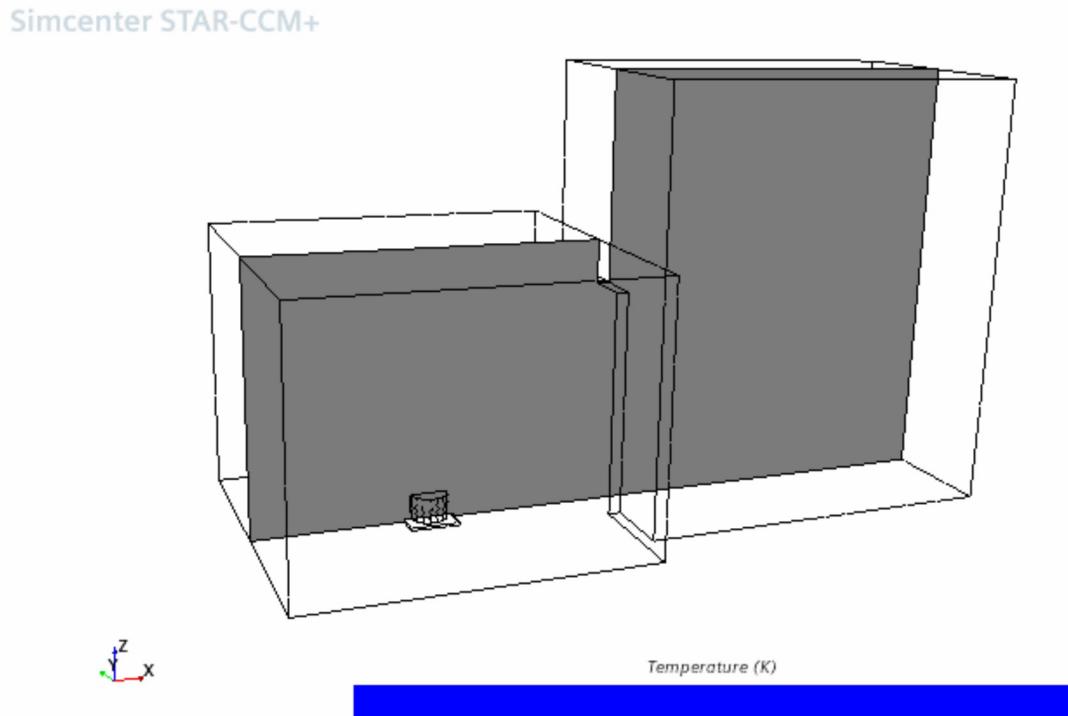


6. Click **OK**.

7. Expand the **Scalar Scene 1** node and set the following properties:

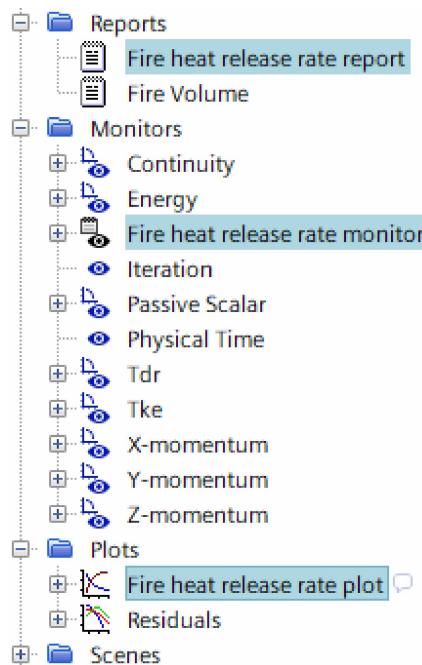
Node	Property	Setting
Outline 1	<i>Feature Lines</i>	activated
Scalar 1	<i>Contour Style</i>	<b>Smooth Filled</b>
└ Scalar Field	<i>Function</i>	<b>Temperature</b>

8. Use the mouse buttons to position the geometry as shown below:



During the Fire properties set up, reports, monitors, and plots for the fire heat release rate are all set up automatically.

9. Open the **Reports**, **Monitors**, and **Plots** nodes.



10. Save the simulation.

## Running the Simulation

Run the simulation.

To start the solver, click  (Run).

The analysis now starts and the results, in the form of a graph of residuals vs. iteration number, is displayed automatically in the *Graphics* window. An example of a residual plot is shown in a separate part of the User Guide.

The solution stops after 200 time-steps.

## Analyzing Results

Analyze the contours of temperature for the converged solution.

The contours of temperature on the plane section that is displayed in *Scalar Scene 1* are dynamically updated with each time-step.

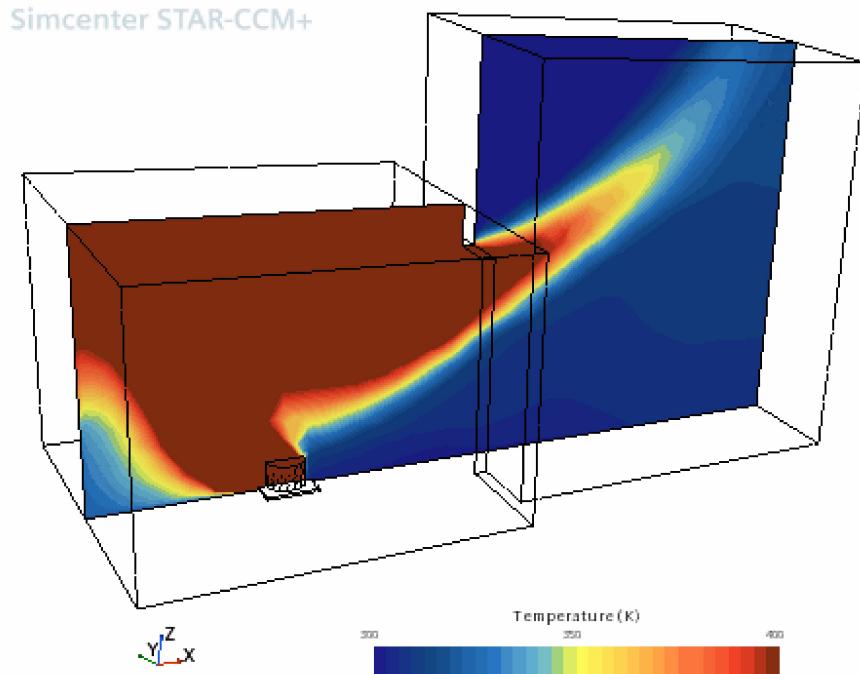
Adjust the temperature range that is displayed in the scalar scene to help you visualize the results.

1. Expand the **Scenes > Scalar Scene 1 > Scalar 1** node.
2. Select the **Scalar Field** node and set the following properties:

Property	Value
Min	300

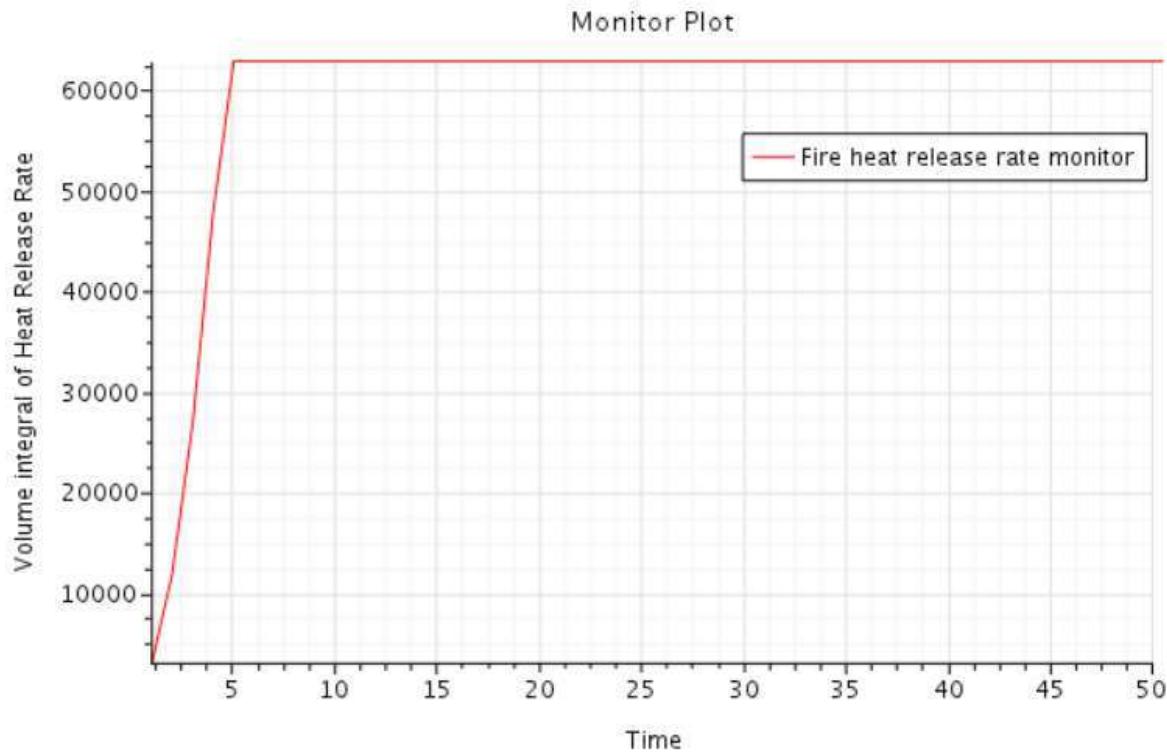
Property	Value
Max	400
Clip	Off

A scene similar to the one shown below is displayed.



Examine the Heat Release Rate in the Fire region:

3. Expand the **Plots** node.
  4. Right-click the **Fire heat release rate plot** node and select **Open**.
  5. Select the **Fire heat release rate plot > Axes > Bottom Axis** node and set *Maximum* to 50.
- A plot similar to the one shown below is now displayed.



- Save the simulation.

## Settings for Steckler Room Fire Validation

Set heat transfer coefficients for the surfaces that represent the walls and ceiling.

In the Steckler room fire experiment, the walls and ceiling are composed of conduction brick of thickness 0.1m and conductivity of 0.69W/m-K. Set an effective heat transfer coefficient of 6.9 W/m<sup>2</sup>-K for each of the five shell surfaces representing the walls and ceiling of the room.

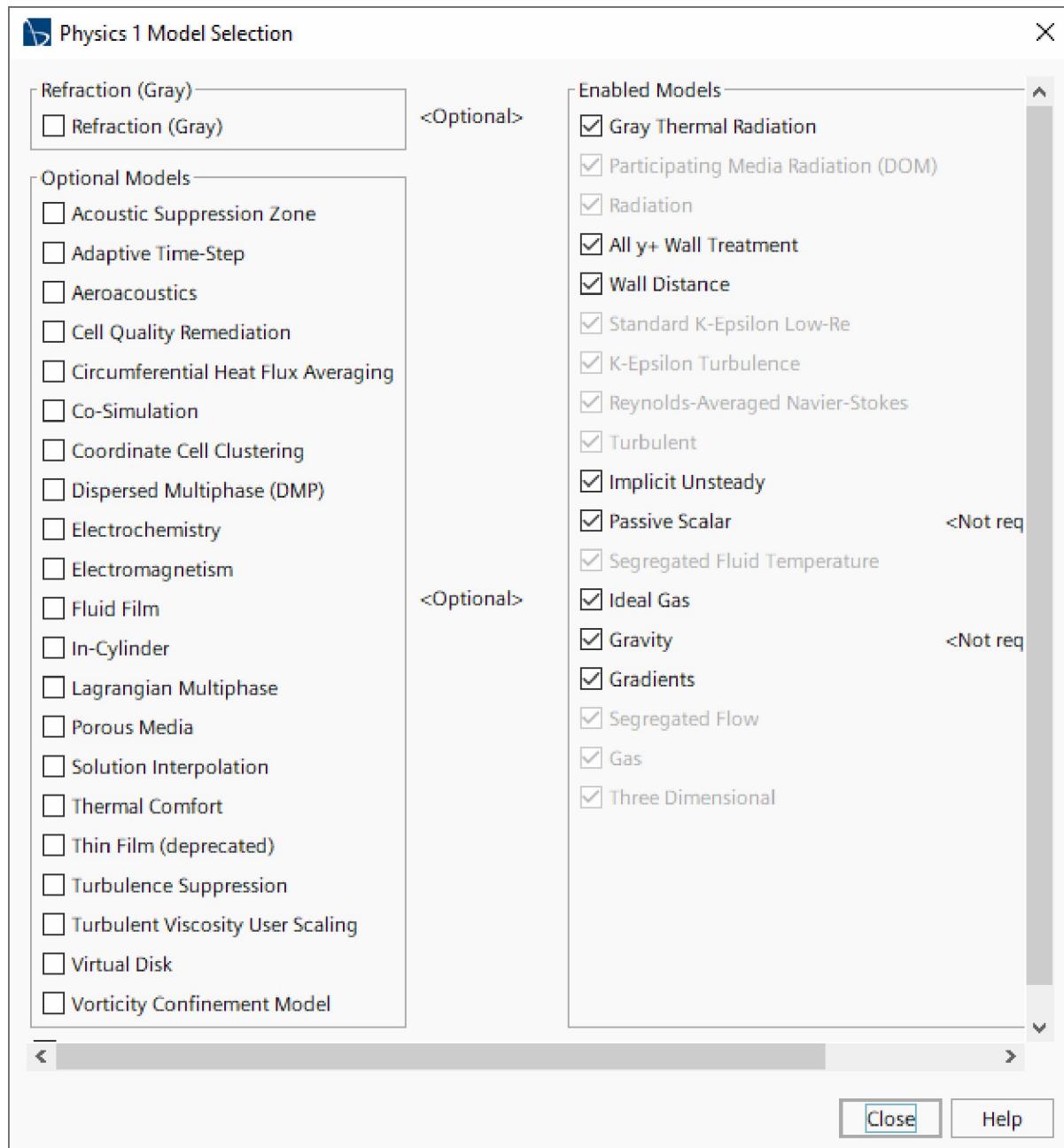
- Expand the **Regions > Room > Boundaries > Fire Room** node.
- Select the **Physics Conditions > Thermal Specification** node and set *Condition* to **Convection**.
- Select the **Physics Values > Heat Transfer Coefficient** node and set *Value* to 6.9 W/m<sup>2</sup>-K.

Switch on the radiation model and set the gray-gas absorptivity to a field function based on temperature:

- Open the **Continua > Physics 1** node.
- Right-click the **Models** node and select **Select Models**.
- In the *Physics Model Selection* dialog, select the following models in order:

Group Box	Model
<i>Optional Models</i>	<b>Radiation</b>
<i>Radiation</i>	<b>Participating Media Radiation (DOM)</b>
<i>Radiation Spectrum (Participating)</i>	<b>Gray Thermal Radiation</b>

The *Physics Model Selection* dialog has the following physics models enabled:



7. Click **Close**
8. Select the **Models > Gas > Air > Material Properties > Absorption Coefficient** node and set the following properties:

Node	Property	Setting
Absorption Coefficient	Method	Field Function
└ Field Function	Scalar Function	AbsorTempFunction

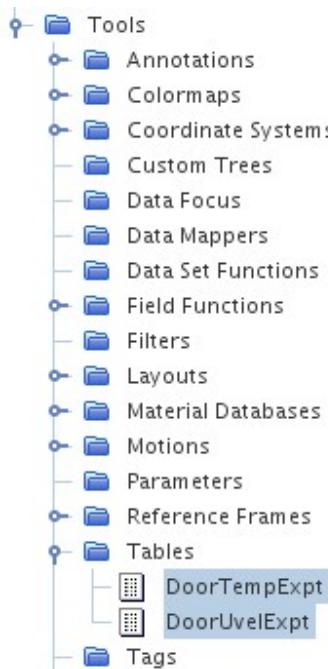
- Save the simulation.

## Comparison with Measurements

To compare the predictions with the experimental data, first load the files containing the vertical temperatures and horizontal velocities at the door.

- Expand the **Tools** node.
- Right-click the **Tables** node and select **New Table > File Table**.
- In the *Open* dialog, navigate to your `stecklerFire` directory and select the `DoorTempExpt.csv` and `DoorUvelExpt.csv` files.
- To read the data, click **Open**.

The `DoorTempExpt.csv` and `DoorUvelExpt.csv` files are now listed under the **Tables** node.



Create a line probe against which to plot the predicted profiles:

- Click the *Geometry Scene 1* tab at the top of the *Graphics* window to activate the scene.
- Right-click the **Derived Parts** node and select **New > Probe > Line Probe**.
- Set the following properties for the line probe:

Property	Setting
<i>Input Parts</i>	<b>Regions &gt; Room</b>
<i>Point 1</i>	[1.4, 0.0, 0.0] m
<i>Point 2</i>	[1.4, 0.0, 1.83] m
<i>Resolution</i>	20
<i>Display</i>	<b>New Surface Displayer</b>

8. Click **Create** then **Close**.

A **Derived Parts > Line Probe** node appears in the simulation tree.

Create a plot for temperature comparison with measurement using the following settings:

9. Right-click the **Plots** node and select **New Plot > XY Plot**.

10. Rename the **XY Plot 1** node to **Door Temperature**.

11. Edit the **Plots > Door Temperature** node and set the following properties:

Node	Property	Setting
<b>Door Temperature</b>	<i>Title</i>	Door Temperature
	<i>Parts</i>	<b>Derived Parts &gt; Line Probe</b>
<b>L X Type &gt; Vector Quantity</b>	<i>Value</i>	[0, 0, 1] m
<b>L Y Types &gt; Y Type 1</b>	<i>Smooth Values</i>	Activated
<b>L Scalar Function</b>	<i>Field Function</i>	<b>Temperature</b>
<b>L Line Probe</b>	<i>Sort Plot Data</i>	Activated
<b>L Line Style</b>	<i>Style</i>	<b>Solid</b>
<b>L Axes</b>		
<b>L Left Axis &gt; Title</b>	<i>Title</i>	Temperature [K]
	<i>Lock Title Name</i>	Activated
<b>L Bottom Axis &gt; Title</b>	<i>Title</i>	Position (m)
	<i>Lock Title Name</i>	Activated
<b>L Legend</b>	<i>x-Position</i>	0.8
	<i>y-Position</i>	0.1

12. To re-orient the plot axes:

a) Right-click the **Axes** node and select **Orient Axes**.

b) In the *Orient Axes* dialog, click then .

The Temperature axis appears on the bottom side of the plot and the Position axis appears on the left side of the plot.

13. Right-click the **Plots > Door Temperature > Data Series** node and select **Add Data**.

14. In the *Add Data Providers to Plot* dialog, select **DoorTempExpt** and click **OK**.

15. Edit the **Data Series > DoorTempExpt** node and set the following properties:

Node	Property	Setting
<b>DoorTempExpt</b>	<i>Legend Name</i>	Experimental Temp.
	<i>Lock Legend Name</i>	Activated
	<i>Table</i>	<b>DoorTempExpt</b>
	<i>X Column</i>	<b>Y</b>
	<i>Y Column</i>	<b>Temperature [K]</b>
	<i>Sort Plot Data</i>	Activated
<b>Symbol Style</b>	<i>Shape</i>	<b>Empty Square</b>
	<i>Color</i>	<b>Blue</b>

The temperature comparison graph setup is now complete. Create a similar plot for the door velocity.

16. Copy and paste the **Door Temperature** plot and rename as **Door Velocity**.
17. Edit the **Door Velocity** node and set the following properties:

Node	Property	Setting
<b>Door Velocity</b>	<i>Title</i>	<b>Door Velocity</b>
<b>Y Types</b>		
<b>Y Type 1 &gt; Scalar Function</b>	<i>Field Function</i>	<b>Velocity[i]</b>
<b>Axes</b>		
<b>Bottom Axis &gt; Title</b>	<i>Title</i>	<b>Velocity (m/s)</b>

18. Expand the **Door Velocity > Data Series > DoorTempExpt** node and set the following properties:

Property	Setting
<i>Legend Name</i>	Experimental Velocity
<i>Table</i>	<b>DoorUvelExpt</b>
<i>X Column</i>	<b>Y</b>
<i>Y Column</i>	<b>V [m/s]</b>

The velocity comparison graph setup is now complete.

## Adjusting Stopping Criteria and Continuing

Increase the maximum physical time of the simulation.

To allow you to solve for a further 200 time-steps, increase the maximum physical time to 400 seconds.

1. Select the **Stopping Criteria > Maximum Physical Time** node and set *Maximum Physical Time* to 400.0 s.

2. Save the simulation as `roomFireValidation.sim`.

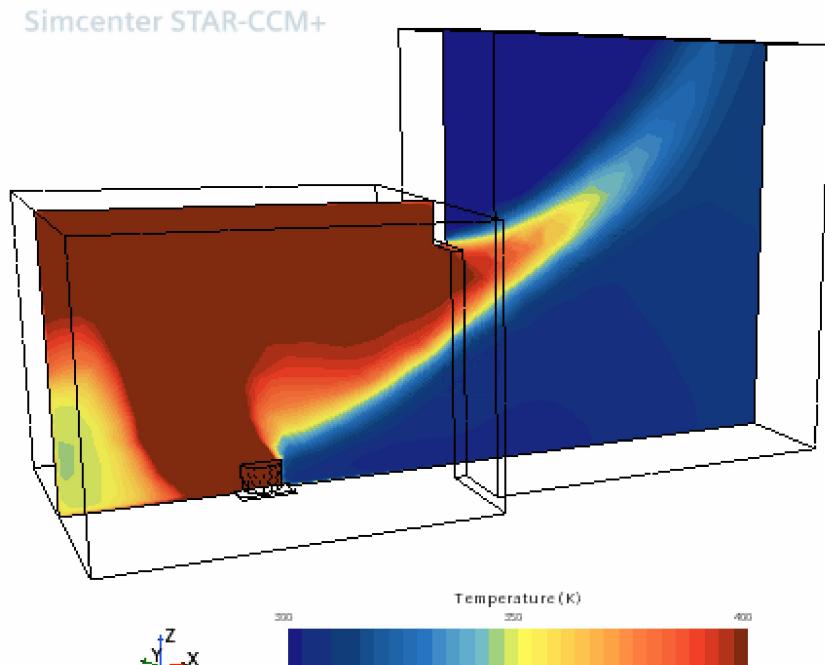
3. To start the solver, click  (Run).

The solution stops after 400 time-steps.

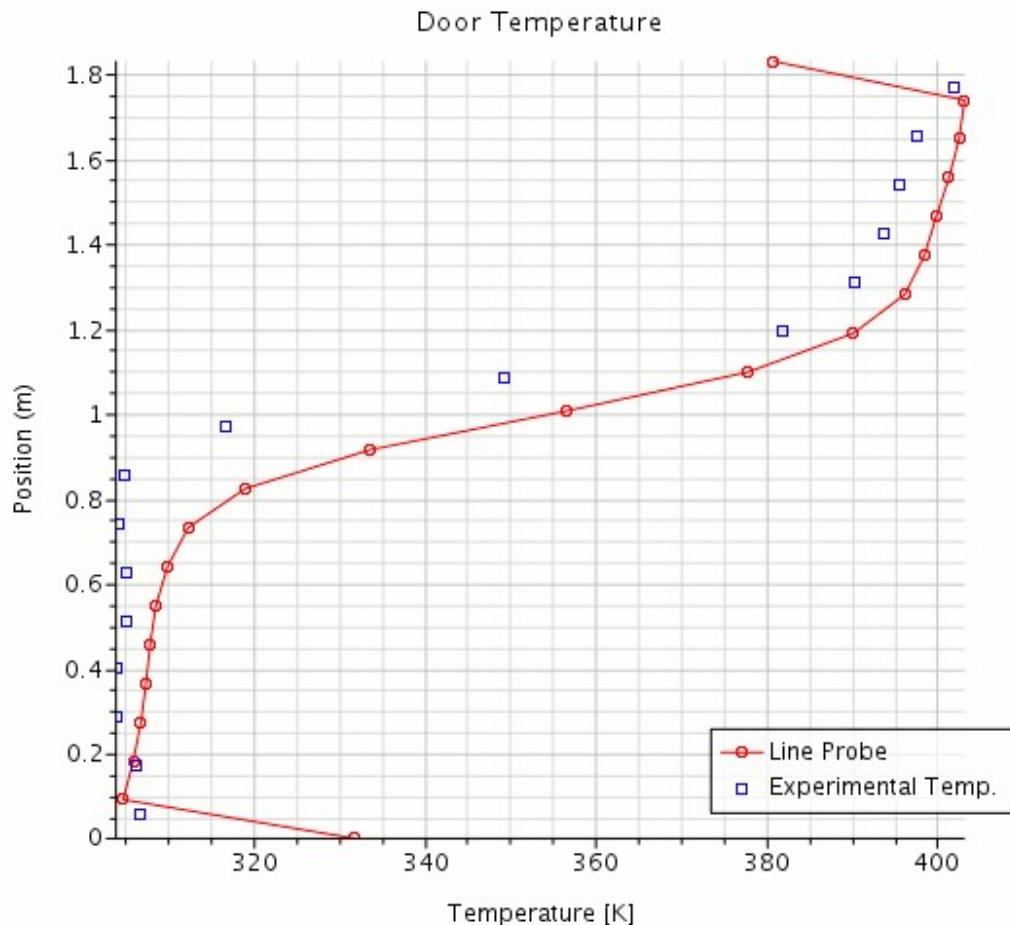
## Analyzing Validation Results

This section of the tutorial covers the analysis of the validation results.

The following scene shows the solution after 400 time-steps, plotted in the temperature range of 300 K to 400 K.

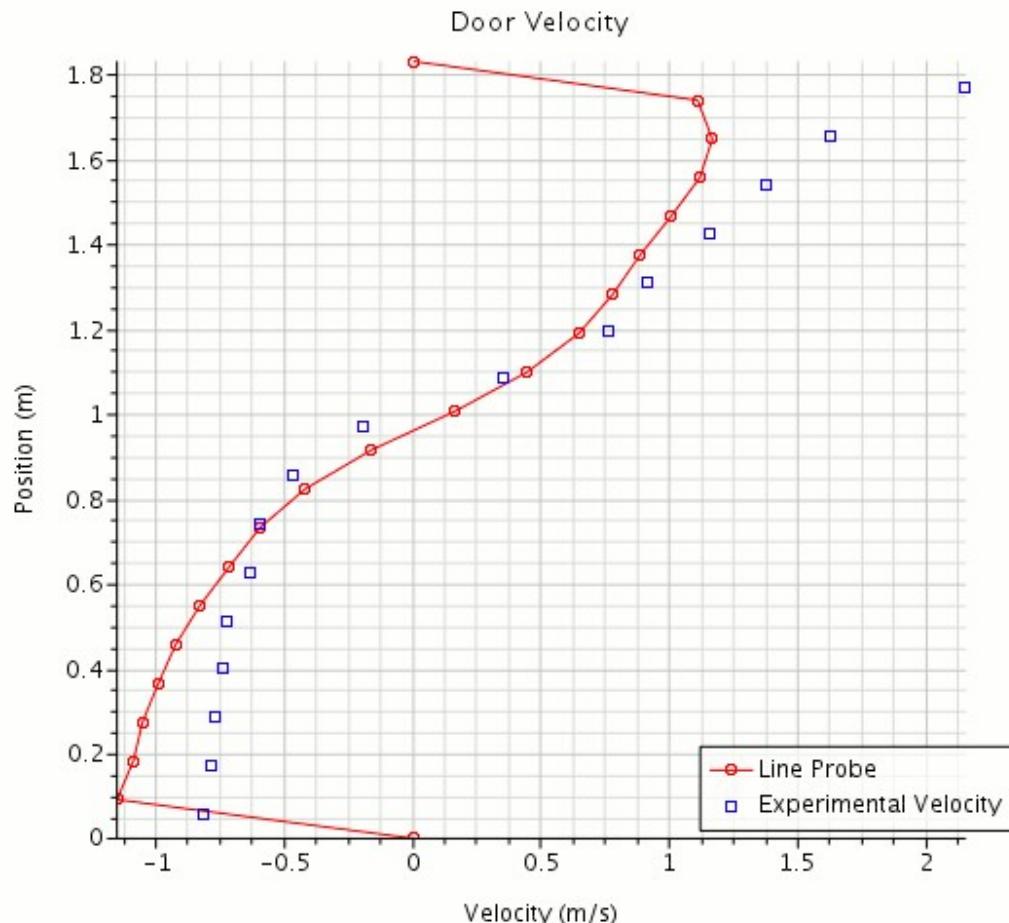


1. Open the *Door Temperature* plot which shows comparison at the defined line-probe location.



The temperature profile from the simulation is in good agreement with the profile based on experimental data.

2. Open the *Door Velocity* plot which shows comparison at the defined line-probe.



The simulated velocity profile shows good agreement with the experimental data up to a height of 1.5 m. After this point, the experimental data indicates that the maximum velocity in the boundary layer is higher than the simulation predicts. One cause for this difference could be an insufficient density of cells in the upper region of the doorway. Good simulation practice would be to reduce the volume mesh **Base Size** to 0.05 m, remesh the domain, and rerun the simulation using the finer mesh. Using a finer mesh size gives closer agreement between the simulated and experimental results.

## Summary

This tutorial demonstrated how to use the Fire and Smoke Wizard.

The steps covered were:

- Invoking the Fire and Smoke Wizard.
- Defining the simulation models.
- Setting up radiation properties.
- Analyzing the results using the built-in plotting and visualization facilities, including the XY Plot, comparing the experimental data with the predictions.