



Smoke Layer Height and Heat Flow through a Door

Smoke Layer Height and Heat Flow through a Door

In this tutorial you will simulate a growing fire in the corner of a 5m x 5m room. The room has a 1m doorway. You will learn how to measure smoke layer height in the compartment and heat flow through the doorway. The door will be initially closed, but will open after the smoke detector is activated.

In this example we also introduce some more practical considerations that must be considered in a simulation:

- We discuss how the heat release rate for a fire may vary with time. We show one approach to calculating this data.
- We demonstrate how to add a control, so that smoke detector activation will open a closed door.
- We change the reaction to match that described in the New Zealand building code, C/VM2 - Verification Method: Framework for Fire Safety Design.

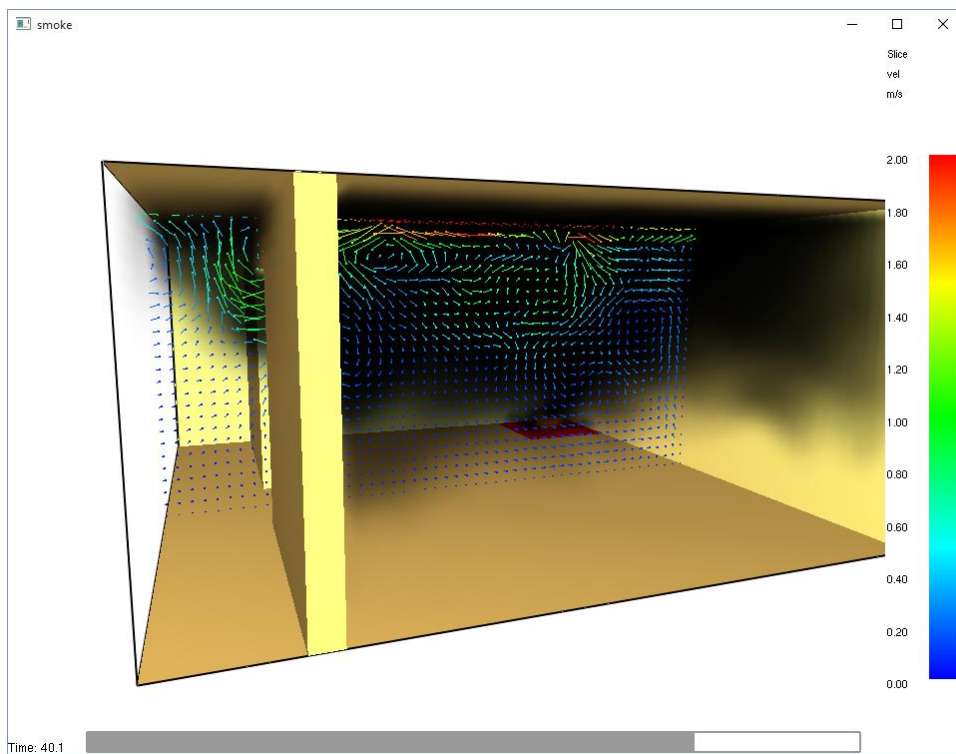


Figure 1. Smoke in the model with air flow in the bottom of the door and out the top.

In this tutorial you will:

- Create a fire with a time-varying heat release rate (HRR).
- Define a new reaction that corresponds to the New Zealand framework for fire safety design.
- Create a doorway using a hole.

- Add a smoke detector that opens the door when it activates.
- Add a flow measurement device to measure heat flow through the door.
- Add a layer zoning device (to measure layer height).

Throughout this example, the instructions will describe data input using menu dialogs. This is done for clarity and consistency. However, PyroSim provides both drawing tools and shortcut toolbars that can speed many of these tasks. The user is encouraged to experiment with these alternate approaches to model creation.

Select SI Units

To select SI units:

1. On the **View** menu, click **Units** and select **SI** to display values using the metric system.

Name and Save the Model

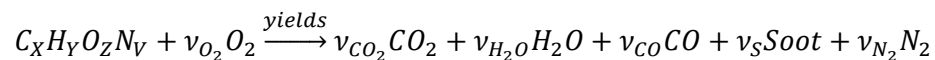
It is a good idea to save your model at the start. This will make it possible for you to save it at any time in the future and return to continue your work.

1. On the **File** menu, click **Save**.
2. Choose a location to save the model. Because FDS simulations generate many files and a large amount of data, it is a good idea to use a new folder for each simulation. For this example, we will create a **Smoke Layer** folder and name the file **smoke layer.psm**.
3. Click **OK** to save the model.

Reaction based on New Zealand Verification Method

In this example we will demonstrate how to create a reaction that corresponds to that defined in the New Zealand building code, C/VM2 - Verification Method: Framework for Fire Safety Design. The New Zealand code defines the fire products, it does not specify the input fuel composition, see Figure 2.

We will assume the stoichiometric simple chemistry reaction as described in the FDS User Manual (Chapter 12). This general relationship is given by:



Using this relationship and the specified product species, we can solve for the fuel composition $C_1 H_{2.20209} O_{0.622494}$. A fuel composition calculator is provided on the PyroSim web site (<https://www.thunderheadeng.com/pyrosim/resources/>).

Table 2.1 Pre-flashover design fire characteristics				
Building use	Fire growth rate (kW)	Species	Radiative fraction	Peak HRR/ HRR/m ²
All buildings including storage with a stack height of less than 3.0 m	0.0469 t ²	Y _{soot} = 0.07 kg/kg Y _{CO} = 0.04 kg/kg ΔH _C = 20 MJ/kg Y _{CO2} = 1.5 kg/kg ⁽¹⁾ Y _{H2O} = 0.82 kg/kg ⁽¹⁾	0.35	20 MW 500 – 1000 kW/m ²⁽²⁾ 250 kW/m ²⁽³⁾
Carparks (no stacking)	0.0117 t ²		0.35	
Capable of storage to a stack height of between 3.0 m and 5.0 m above the floor	0.188 t ²		0.35	50 MW
Capable of storage to a stack height of more than 5.0 m above the floor and car parks with stacking systems	0.00068 t ³ H		0.35	1000–2500 kW/m ²⁽²⁾ 250 kW/m ²⁽³⁾
NOTE: t = time in seconds H = height to which storage is capable of in metres Y = yield kg/kg ΔH _C = heat of combustion (1) As an alternative to CO ₂ + H ₂ O yields use generic fuel as CH ₂ O _{0.5} and calculate yields. (2) In a CFD model the fire is intended to be modelled as a plan area where the size is determined from the peak HRR/m ² . A range is provided for HRR/m ² to accommodate different HRR and mesh sizes. (3) Use in a zone model.				

Figure 2: Table from the New Zealand C/VM2 Verification Method: Framework for Fire Safety Design. The species yields are defined, not the fuel input.

To define the reaction (see Figure 3 and Figure 4):

1. On the **Model** menu, click **Edit Reactions**.
2. Click **New**. In the **Reaction Name** box, type **NZ Reaction**.
3. Click **OK** to create the new reaction.
4. The **Fuel Type** is the **Simple Chemistry Model**.
5. In the **Carbon atoms** box, type **1**.
6. In the **Hydrogen atoms** box, type **2.20209**.
7. In the **Oxygen atoms** box, type **0.622494**.
8. In the **Nitrogen atoms** box, type **0**.
9. Click the **Byproducts** tab.
10. Select **Specify heat of combustion** and in the box, type **2.0E4**.
11. In the **CO Yield** box, type **0.04**.
12. In the **Soot Yield** box, type **0.07**.
13. In the **Hydrogen Fraction** box, type **0**.
14. Click **OK** to close the **Edit Reactions** dialog.

Edit Reactions

Description:

Fuel | Fire Suppression | Byproducts | Advanced

Fuel Type:

Fuel assumed to contain only C, O, H, and N.

Composition

Carbon atoms:

Hydrogen atoms:

Oxygen atoms:

Nitrogen atoms:

&REAC ID='NZ Reaction', FUEL='REAC_FUEL', C=1.0, H=2.20209, O=0.622494/

Apply OK Cancel

Figure 3: Fuel composition.

Edit Reactions

Description:

Fuel | Fire Suppression | Byproducts | Advanced

Energy Released:

☐ Specify release per unit mass oxygen:

☒ Specify heat of combustion:

☐ Radiative Fraction:

☐ Energy is Ideal (does not account for yields of CO, H₂, or Soot)

CO Yield (Y_{CO}):

Soot Yield (Y_s):

Hydrogen Fraction:

&REAC ID='NZ Reaction', FUEL='REAC_FUEL', C=1.0, H=2.20209, O=0.622494, SOOT_H_FRACTION=0.0, CO_YIELD=0.04, SOOT_YIELD=0.07, HEAT_OF_COMBUSTION=2.0E4/

Apply OK Cancel

Figure 4: The reaction byproducts.

When FDS runs, it writes a *.out file (in this case smoke.out) that includes the details of the simulation. You can verify that the FDS reaction coefficients match the expected values based on our fuel component input.

Create the Fire (Burner Surface)

It is the responsibility of the fire safety engineer to identify the appropriate fire heat release rate and products. As background information, Kim and Lilley (Heat Release Rates of Burning Items in Fires, AIAA 2000-0722, January 2000) summarized several different experiments, representing the results using t^2 fire for the growth and decay periods, with a constant maximum heat release rate between these two periods. This paper is especially useful in demonstrating the broad range of fire time histories and heat release rates that can be encountered. A calculator based on this paper can be found on the PyroSim support website (<https://www.thunderheadeng.com/pyrosim/resources/>).

For this example, we will use the heat release rate shown in Figure 5. This curve was calculated for the Chair 2 data set in the Kim and Lilley paper. The detailed results of the calculation for the first 100 seconds are shown in Figure 6. In FDS the curve is defined by a nominal Heat Release Rate per Unit Area (HRRPUA) value that is multiplied by a factor (RAMP) to obtain the curve. The total heat release rate is then obtained by multiplying the HRRPUA by the total surface area of the fire. So, an FDS user must be aware of the surface area of the fire, the nominal value of HRRPUA, and the modifying factor. In this example, the surface area of the fire will be 1 m².

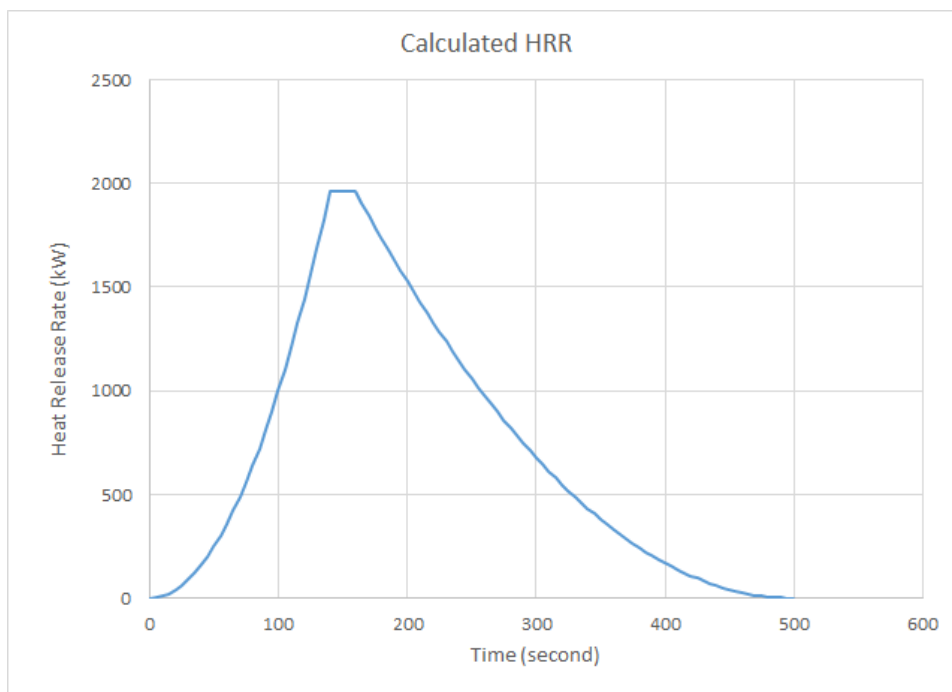


Figure 5: HRR history that represents measured data for a chair with a polypropylene foam frame, urethane foam, and polyolefin fabric. This is called Chair 2 in Kim and Lilley (2000).

HRR Max (kW) =		1960
Time (s)	HRR (kW)	Normalized
0	0.0	0.000
10	10.0	0.005
20	40.0	0.020
30	90.0	0.046
40	160.0	0.082
50	250.0	0.128
60	360.0	0.184
70	490.0	0.250
80	640.0	0.327
90	810.0	0.413
100	1000.0	0.510

Figure 6: First 100 seconds of the HRR curve used in this example

Surfaces are used to define the properties of objects in your FDS model. In this example, we define a burner surface that releases heat at the rate shown above in Figure 5.

1. On the **Model** menu, click **Edit Surfaces**.
2. Click **New**.
3. In the **Surface Name** box, type **Fire**.
4. In the **Surface Type** list, select **Burner**.
5. Click **OK** to create the new default burner surface.
6. In the **Heat Release Rate (HRR)** box, type **1960 kW/m²**. Note that this corresponds to the peak value of HRRPUA.
7. Select **Custom** for the **Ramp-Up Time**.
8. Click **Edit Values** enter the time and normalized values given in Figure 6 (also see Figure 7). If you prepare the data in two columns, you can paste it directly into the table in one operation. Click **OK**.
9. Click **OK** to save changes and close the **Edit Surfaces** dialog.

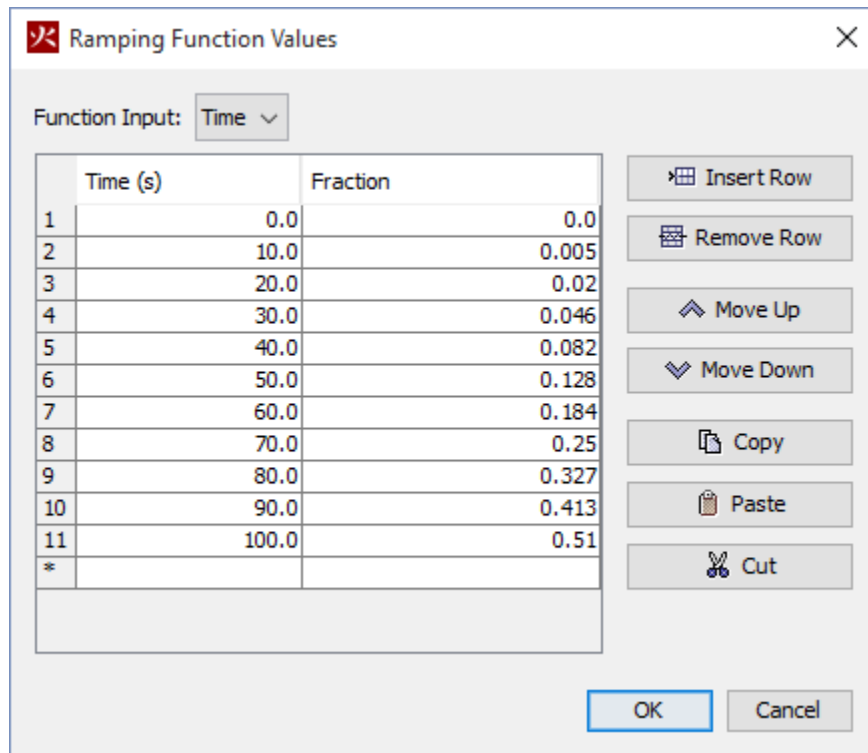


Figure 7: The ramp values multiply the specified HRRPUA to define a function in time.

Create the Mesh

The *mesh* defines the solution grid. In this example we will use mesh cells that are 0.1 m across. This value is slightly larger than 1/10 of the characteristic diameter (D^*) for a 800 kW fire ($D^*/10=0.088$ m). As a rule of thumb, this is a reasonable size for initial calculations and should provide a moderate level of accuracy in modeling the plume (McGrattan, et al., 2007). Using mesh cells that are smaller by a factor of 2 should decrease error by a factor of 4, but will increase the simulation run time by a factor of 16.

1. On the **Model** menu, click **Edit Meshes**.
2. Click **New**.
3. Click **OK** to create the new mesh.
4. In the **Min X** box, type **0** and in the **Max X** box, type **5**.
5. In the **Min Y** box, type **0** and in the **Max Y** box, type **5**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **2.4**.
7. In the **X Cells** box, type **50**.
8. In the **Y Cells** box, type **50**.
9. In the **Z Cells** box, type **24**.
10. Click **OK** to save changes and close the **Edit Meshes** dialog.

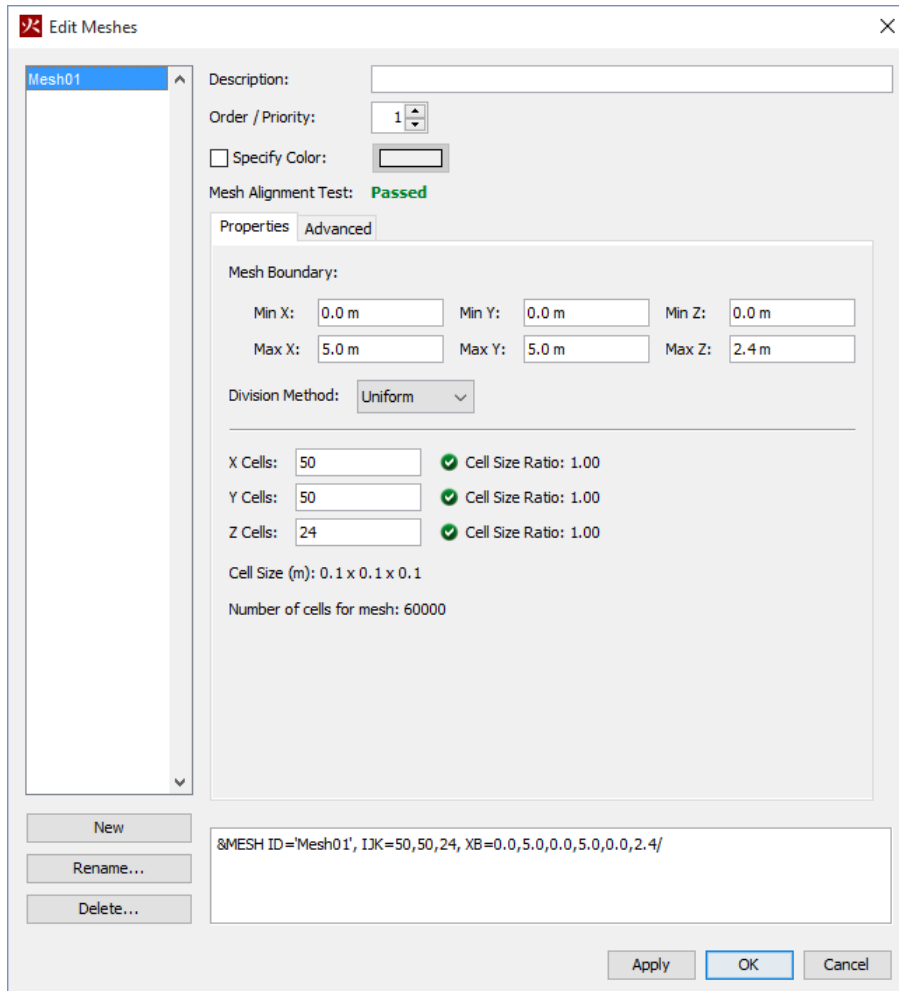


Figure 8. Creating the mesh

Create the Fire using a Vent

Vents have general usage in FDS to describe 2D planar objects. Taken literally, a vent can be used to model components of the ventilation system in a building, like a diffuser or a return. In these cases, the vent coordinates define a plane forming the boundary of the duct. No holes need to be created; air is supplied or exhausted by the vent.

You can also use vents as a means of applying a boundary condition to a rectangular patch on a surface. A fire, for example, can be created by specifying a vent on either a mesh boundary or solid surface. The vent surface defines the desired characteristics of fire. This is the approach used in this example.

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Fire vent**, Figure 9.
3. In the **Surface** list, select **Fire**. This specifies that the previously created burner surface will define the properties of the vent.
4. Click on the **Geometry** tab. In the **Plane** list, select **Z**. Set the value to **0.0**.

5. In the **Min X** box, type **4.0** and in the **Max X** box, type **5.0**.
6. In the **Min Y** box, type **0.0** and in the **Max Y** box, type **1.0**.
7. Click **OK** to create the new burner vent.

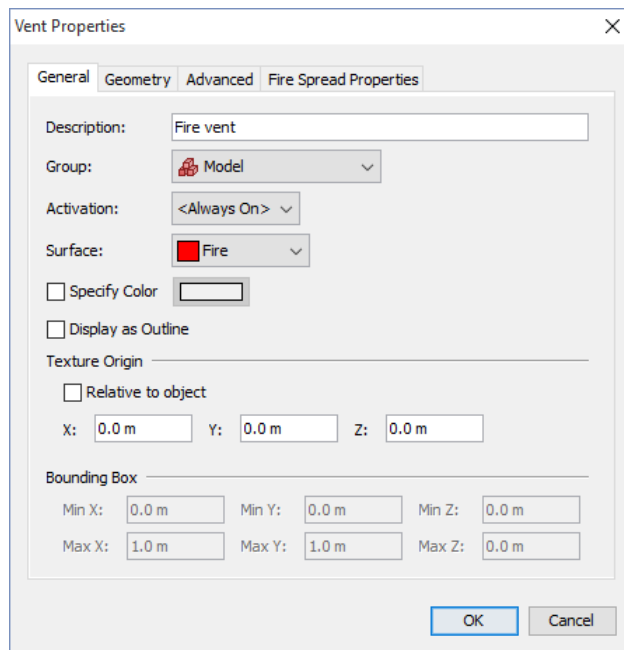


Figure 9. Creating the fire vent

Create the Open Vent outside the Door

We will place an open vent outside the door. This means that the default ambient conditions will exist outside and that air can flow in and smoke flow out through this open vent. This is a nice way to simulate the flow out of a door, both because it lets you visualize the smoke outside the door and because the air flow through the door is affected by the outside ceiling and floor.

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Open side**.
3. In the **Surface** list, select **OPEN**. This is a default surface that means this will be an open boundary.
4. Click on the **Geometry** tab. In the **Plane** list, select **Y** and type **5**.
5. In the **Min X** box, type **0** and in the **Max X** box, type **5**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **2.4**.
7. Click **OK** to create the open vent.

Add the Wall

In FDS obstructions are used to define solid objects in the model. In this example, we will use an obstruction to define a wall.

1. On the **Model** menu, click **New Obstruction**.

2. In the ID box, type **Wall**.
3. Click on the **Geometry** tab.
4. In the **Min X** box, type **0** and in the **Max X** box, type **5**.
5. In the **Min Y** box, type **4** and in the **Max Y** box, type **4.2**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **2.4**.
7. Click **OK** to create the wall obstruction.

Add a Door

In FDS *holes* are used to define openings through solid objects. In this example, we will use a hole to represent a door. The alternate approach is to represent the wall with three obstructions (left, right, and top) and then have a fourth obstruction for the door. The hole is simpler, but either approach can work.

To make the hole:

1. On the **Model** menu, click **New Hole**.
2. In the ID box, type **Door**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **2** and in the **Max X** box, type **3**. In the **Min Y** box, type **3.9** and in the **Max Y** box, type **4.3**. In the **Min Z** box, type **0.0** and in the **Max Z** box, type **2.0**.
5. Click **OK** to create the doorway hole.

Add a Smoke Detector

We will make a smoke detector that will be used in a control that opens the door when the detector activates. To make the smoke detector:

1. On the **Devices** menu, click **New Smoke Detector**.
2. In the **Name** box, type **Smoke detector**. We will use the default Cleary Ionization I1 detector.
3. We position the detector near the ceiling in the center of the room. For **Location**, in the **X** box, type **2.5**, in the **Y** box, type **2**, and in the **Z** box, type **2.3** (just below the ceiling).
4. The default **Orientation** points the detector in a -Z direction and does not need to be changed.
5. Click **OK** to create the smoke detector.

Add a Control to Open the Door when Smoke is Detected

Now we have a smoke detector and a door. We want to keep the door closed until smoke is detected, then simulate someone leaving the room with an open door. To do this, we must tie the detector and the door together with a *control*.

1. On the **Devices** menu, click **Edit Activation Controls**.
2. Click **New** and in the **Name** box, type **Smoke detector control**. Click **OK**.
3. For the **Input Type** select **Detector**. This means that the control will rely on input from the smoke detector to perform its assigned tasks.
4. Since the Door is a hole, we want to activate the Door (hole) when smoke is detected. So the **Action to Perform** is to **Activate**.

5. Now click on the first **<nothing>** and select **Door** from the list, so the Door will be created. Click **OK**.
6. Now click on the second **<nothing>** and select **Smoke detector** from the list. Now we can see the logic associated with the control. The **Door** will activate (be created) when the **Smoke detector** activates. Click **OK**.
7. Click **OK** to create the control and close the **Activation Controls** dialog.

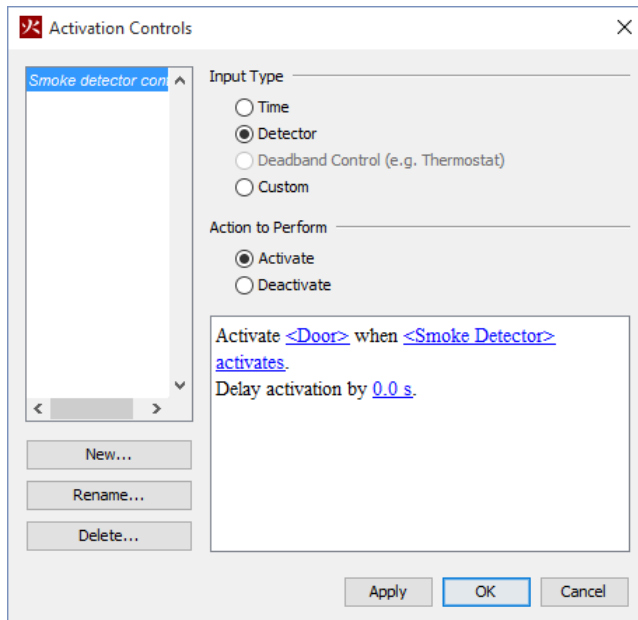


Figure 10: Creating the control that opens the door when the smoke detector activates.

Spin the Model for a Better View

1. To reset the zoom and properly center the model, press **CTRL + R**. PyroSim will now be looking straight down at the model along the Z axis.
2. Press the **left mouse button** in the **3D View** and drag to spin the model. Note that the door hole is displayed, but in the final results view will only appear when activated.

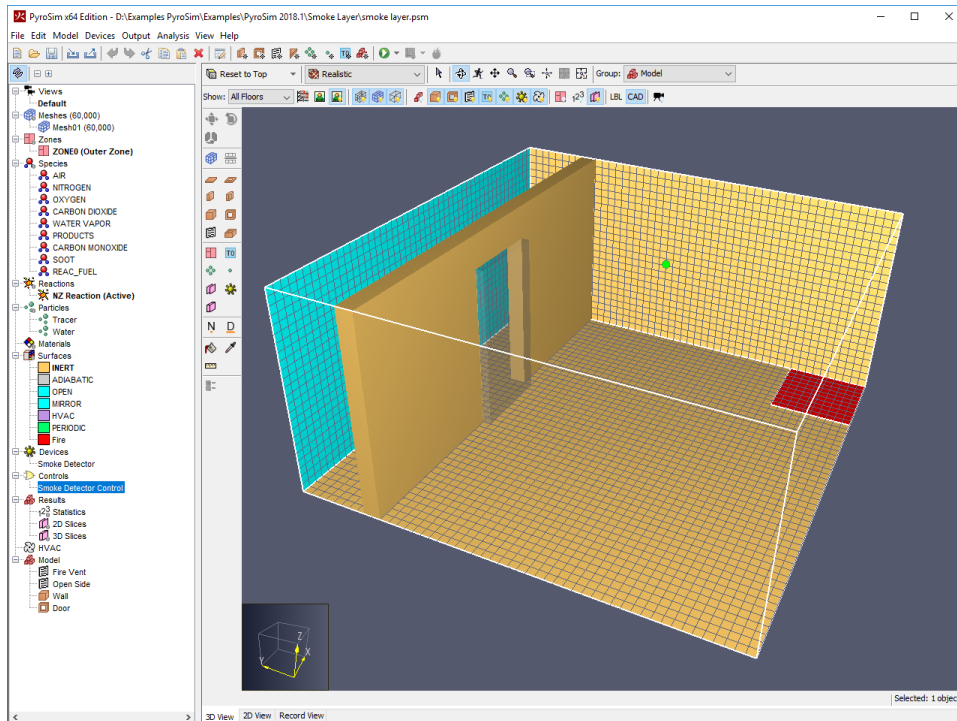


Figure 11. The model after rotating. The burner is shown in red and the open vent in blue

Add a Layer Zoning Device

FDS estimates the location of the interface between the hot, smoke-laden upper layer and the cooler lower layer in a burning compartment (see the FDS manual for details). This is calculated using a special *Layer Zoning Device*. To create the device:

1. On the **Devices** menu, click **New Layer Zoning Device**.
2. In the **Name** box, type **Layer Zone**.
3. Click the options to **Measure Layer Height**, **Measure Upper Temperature**, and **Measure Lower Temperature**. We will not use setpoints, but enabling setpoints would make it possible to use the layer data to activate a control.
4. For the **End Point 1** coordinates, in the **X** box, type **2.5**, in the **Y** box, type **2.5**, and in the **Z** box, type **0**.
5. For the **End Point 2** coordinates, in the **X** box, type **2.5**, in the **Y** box, type **2.5**, and in the **Z** box, type **2.4**.
6. Click **OK** to create the layer zoning device. It will be displayed as a line in the model.

Add a Flow Measuring Device

In addition to point measurements (such as thermocouples or smoke detectors), FDS supports more general options to evaluate parameters on planes or in volumes. We will use a planar device to measure heat flow through the door.

1. On the **Devices** menu, click **New Flow Measuring Device**.

2. In the **Device Name** box, type **Door Heat Flow**.
3. In the **Quantity** options, select **Heat Flow**.
4. In the **Plane** list, select **Y** and type **4**.
5. In the **Min X** box, type **2** and in the **Max X** box, type **3**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **2**.
7. Click **OK** to create the flow measuring device. It will appear as a yellow plane in the model.

Add a Slice Planes to Display Velocity Vectors

We will use a slice plane to display the velocity vectors of flow through the door.

1. On the **Output** menu, click **2D Slices**.
2. In the **XYZ Plane** column, select **X**.
3. In the **Plane Value** column, type **2.5**.
4. In the **Gas Phase Quantity** column, click **Velocity**.
5. In the **Use Vector?** column, select **YES**.
6. In the **Cell Centered?** column, select **NO**.
7. Add a second row to plot **Temperature**. Do not use vectors.
8. Click **OK** to create the slice planes.

On the toolbar, you can click the **Show Slices** button to toggle the slice planes on and off.

Set the Simulation Time

1. On the **Analysis** menu, click **Simulation Parameters**.
2. On the **Time** panel, in the **End Time** box, type **50**. Although the fire is still growing at 50 seconds, it is sufficient to activate the smoke detector, open the door, and illustrate smoke flowing out the door.
3. Click **OK** to save the simulation parameters.

Save the model

1. On the **File** menu, click **Save**.

Run the Simulation

1. On the **FDS** menu, click **Run FDS**.
2. The **FDS Simulation** dialog will appear and display the progress of the simulation.
3. When the simulation is complete, PyroSim Results should launch automatically and display a 3D still image of the model.

View Smoke in 3D

We will first view the smoke and reaction volumes.

1. In the PyroSim Results window, double click **3D Smoke** to load the **HRRPUV** and **Soot Mass Fraction** datasets.

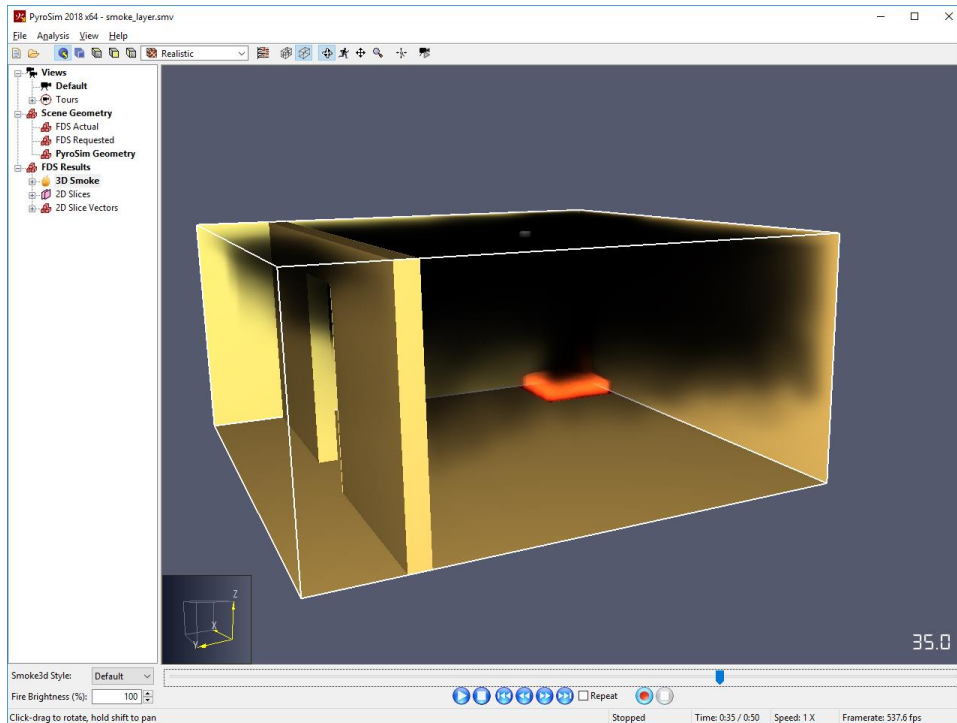


Figure 12. 3D Smoke and fire in the model

View the Velocity and Temperature Slices

We now display the velocity profile.

1. In the tree, expand **Slice Vectors** and double click **VELOCITY**. This will load the vector slice data.
2. Double-click the **Temperature** 2D Slice to see a contour of temperatures. At this time, we can see the development of hot and cool zones, the smoke in the room, and the flow of hot air out the top of the door and cold air in the bottom of the door.

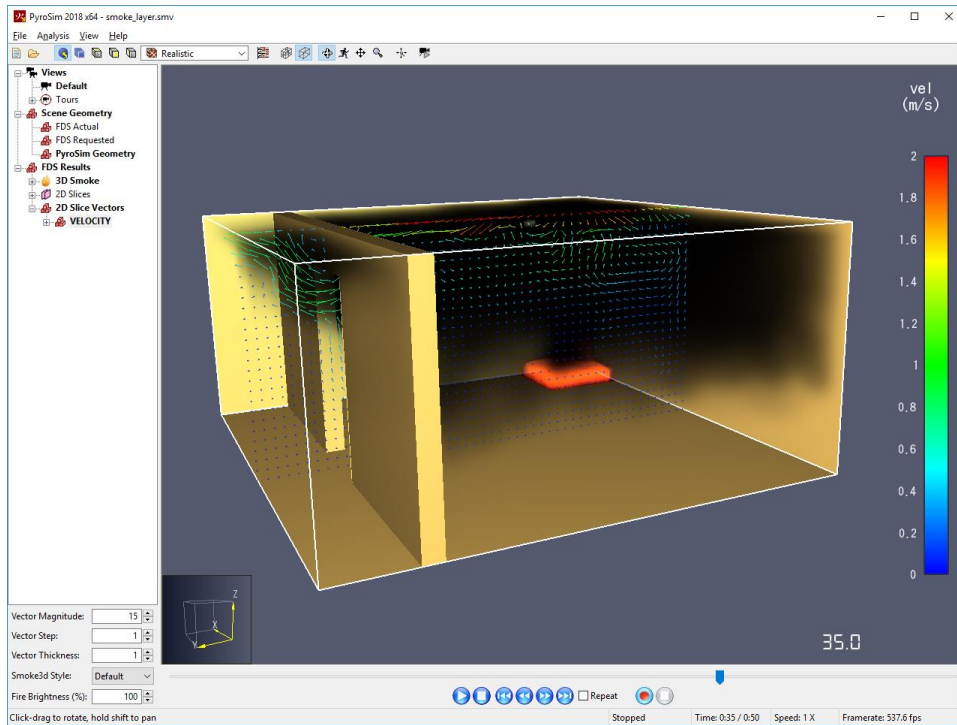


Figure 13: Velocity vectors in the model.

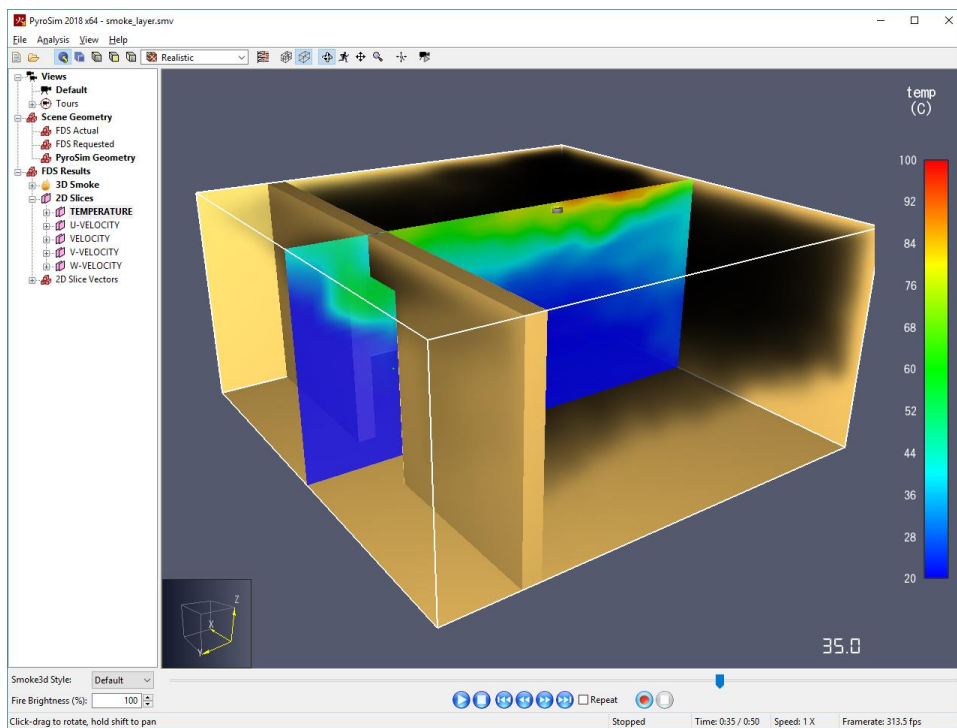


Figure 14: Temperature slice

View Time History Data

We will explore just some of the time history data.

1. In the PyroSim window, on the **Analysis** menu, click **Plot Time History Results**.
2. A dialog will appear showing the different types of 2D results that are available. Select **smoke_hrr.csv** and click **Open** to view the device output. The first display will be the heat release rate, Figure 15. At 50 seconds, the expected value is the ramp fraction (0.128) multiplied by the HRRPUA (1960 kW/m^2), multiplied by the surface area (1 m^2), to obtain a value of 251 kW. This matches the calculated result.
3. Now click **Plot Time History Results**, select **smoke_ctrl.csv** and click **Open** to view the smoke detector control output, Figure 16. As can be seen, the smoke detector control activated at about 15 seconds and opened the door.

Note: You can access these plots faster using the graph button on the toolbar.

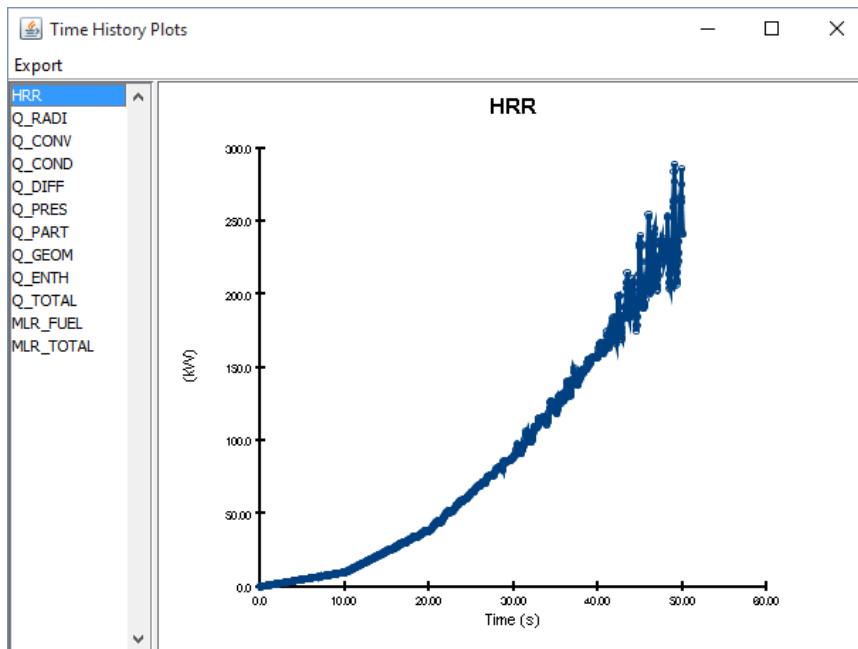


Figure 15. Time history plot of heat release rate.

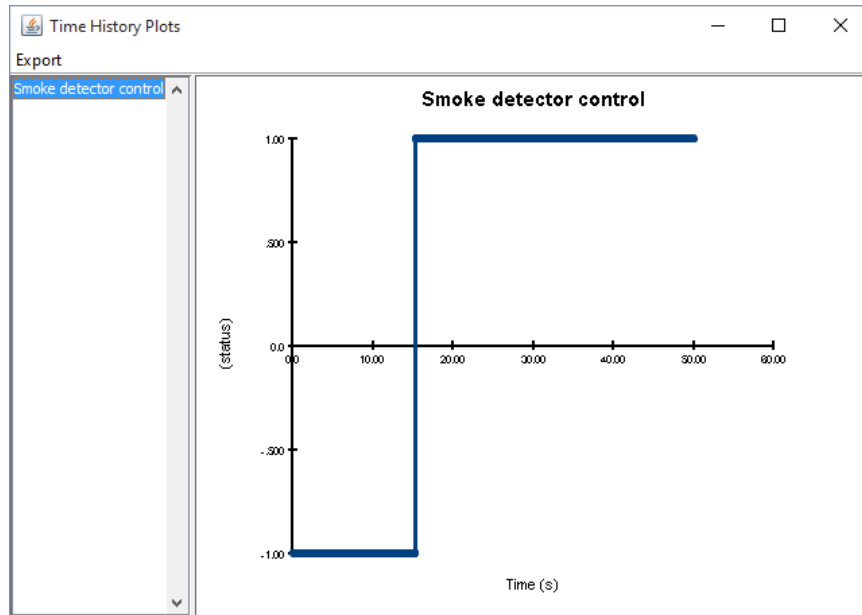


Figure 16: Smoke detector control output.

Summary

We have completed a fairly complex fire simulation that illustrates what, for most fire protection engineers, will be a sufficient approach to modeling a fire by defining a reaction and specifying a heat release rate that varies with time. Of course, the engineer will be required to determine the exact fire composition for the reaction and the heat release rate time history.

Other important points in this model include the user of a smoke detector device to activate a control that opened a door and the display of results.

References

C/VM2, Verification Method: Framework for Fire Safety Design, Amendment 4. 2014. The Ministry of Business, Innovation and Employment, PO Box 10-729, Wellington.

FDS-SMV Official Website. <https://pages.nist.gov/fds-smv/>.

McGrattan, Kevin, et al. 2017. *Fire Dynamics Simulator Technical Reference Guide (Sixth Edition)*. Gaithersburg, Maryland, USA, November 2017. NIST Special Publication 1018-1.

McGrattan, Kevin, et al. 2017. *Fire Dynamics Simulator User's Guide (Sixth Edition)*. Gaithersburg, Maryland, USA, November 2017. NIST Special Publication 1019.