Smoke Layer Height and Heat Flow Through a Door
Smoke Layer Height and Heat Flow Through a Door

In this tutorial you will simulate an 800 kW 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 though the doorway.

In this tutorial you will:

- Create an 800 kW burner fire.
- Create a doorway using a hole.
- Add a flow measurement device.
- Add a layer zoning device (to measure layer height).
- View 3D results using Smokeview.
- View 2D results using PyroSim.

Before you begin, ensure that you are using SI units.

Create the Burner Surface

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 a rate of 800 kW/m2.

1. On the Model menu, click Edit Surfaces....
2. Click New....
3. In the Surface Name box, type burner, Figure 2.
4. In the Surface Type list, select Burner.
5. Click OK to create the new default burner surface.
1. In the Description box, type 800 kW/m² burner, Figure 3.
2. In the Heat Release Rate (HRR) box, type 800.
3. Click OK to save changes and close the Edit Surfaces dialog.

Create the Burner 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 particular 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 **Description** box, type `burner vent`, Figure 4.
3. In the **Surface** list, select `burner`. 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.

![Vent Properties](image)

Figure 4. Creating the burner vent

**Create the Open Side Vent**

One side of the model is an open boundary.

1. On the **Model** menu, click **New Vent**....
2. In the **Description** 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 5.0.
5. In the **Min X** box, type 0.0 and in the **Max X** box, type 5.0.
6. In the **Min Z** box, type 0.0 and in the **Max Z** box, type 2.4.
7. Click **OK** to create the open vent.
Create the Mesh
In this example we will use mesh cells that are 0.17 m across. This value is approximately 1/5 of the characteristic diameter (D*) for a 800 kW fire. As a rule of thumb, this is as large as the mesh cells can be while still maintaining 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. The boundary dimensions will automatically be set to the correct size based on the two vents (Figure 5).
4. In the X Cells box, type 30.
5. In the Y Cells box, type 30.
6. In the Z Cells box, type 15.
7. Click OK to save changes and close the Edit Meshes dialog.

![Edit Meshes dialog](image)

Figure 5. Creating the mesh

Add the Wall
In FDS obstructions are used to define solid object 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 Description box, type wall.
3. Click on the Geometry tab, Figure 6.
4. In the Min X box, type 0.0 and in the Max X box, type 5.0.
5. In the Min Y box, type 4.0 and in the Max Y box, type 4.2.
6. In the Min Z box, type 0.0 and in the Max Z box, type 2.4.
7. Click OK to create the wall obstruction.

![Obstruction Properties](image)

**Figure 6. Creating the wall**

**Add the Door**
1. In FDS **holes** are used to define openings through solid objects. In this example, we will use a hole to define a door.
2. On the **Model** menu, click **New Hole**....
3. In the **Description** box, type **door**.
4. Click on the **Geometry** tab. In the **Min X** box, type 2.0 and in the **Max X** box, type 3.0.
5. In the **Min Y** box, type 3.9 and in the **Max Y** box, type 4.3.
6. In the **Min Z** box, type 0.0 and in the **Max Z** box, type 2.0.
7. Click **OK** to create the doorway hole.

**Orbit 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 orbit the model. You can also unselect the **Show Holes** button so that the hole object will not be displayed and you will just see the opening through the wall (Figure 7).

![Figure 7](image)

*Figure 7. The model after rotating. The burner is shown in red and the top vent in blue*

**Add a Layer Zoning Device**
1. On the **Devices** menu, click **New Layer Zoning Device**....
2. In the **Device Name** box, type *layer zone 01*.
3. 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 0.0.
4. 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.
5. Click **OK** to create the layer zoning device. It will be displayed as a line in the model.

**Add a Flow Measuring Device**
1. On the **Devices** menu, click **New Flow Measuring Device**....
2. In the **Device Name** box, type *door flow*.
3. In the **Quantity** options, select **Heat Flow**.
4. In the **Plane** list, select **Y** and type 4.0.
5. In the **Min X** box, type 2.0 and in the **Max X** box, type 3.0.
6. In the **Min Z** box, type 0.0 and in the **Max Z** box, type 2.0.
7. Click **OK** to create the flow measuring device. It will appear as a yellow plane in the model.
Set the Simulation Time
1. On the FDS menu, click Simulation Parameters....
2. On the Time panel, in the End Time box, type 45.0.
3. Click OK to save the simulation parameters.

Save the model
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 folder and name the file smoke.psm.
3. Click OK to save the model.

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. By default, PyroSim specifies a 10 second simulation. This should take approximately 2-3 minutes to run depending on computing hardware.
3. When the simulation is complete, Smokeview should launch automatically and display a 3D still image of the model, Figure 8.

View Smoke in 3D
1. In the Smokeview window, right-click to activate the menu.
2. In the menu, click Load/Unload > 3D Smoke > soot mass fraction (RLE). This will start an animation of the smoke in this model.
3. In the menu, click Load/Unload > 3D Smoke > HRRPUV (RLE). This will start add an animation of fire to the model in addition to the smoke (Figure 9).
4. To view a specific time in the animation, click the **timeline bar** in the bottom of the Smokeview window. To return to animation mode, press **t**.
5. To reset Smokeview, right-click to activate the menu, then click **Load/Unload > Unload All**.

![Figure 9. 3D smoke in the model](image)

**View Time History Data**

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results**....
2. A dialog will appear showing the different types of 2D results that are available. Select **smoke_devc.csv** and click **Open** to view the device output. The first display will be the heat flow through the door, Figure 10.
3. To select a data set from a different sensor, select that data in the left panel. To view smoke layer height data, click **layer zone 01->HEIGHT**, Figure 11.

![Figure 10. Time history plot of heat flow through the door](image)
Figure 11. Time history plot of smoke layer height
References

FDS-SMV Official Website. *Fire Dynamics Simulator and Smokeview*. Gaithersburg, Maryland, USA: National Institute of Standards and Technology.
