PyroSim

A MODEL CONSTRUCTION TOOL FOR FIRE DYNAMICS SIMULATOR (FDS)

THUNDERHEAD
ENGINEERING

PyroSim Example Guide

2010.2

Example Guide

Table of Contents

List of Figures

List of Tables

Chapter 1. Before Starting

Install PyroSim

In order to work through this tutorial you must be able to run PyroSim. You can download PyroSim from the Internet by going to http://www.pyrosim.com/ to obtain the free trial.

Units

Except where noted, the instructions given in this tutorial will assume that PyroSim's current unit system is **SI**. If PyroSim is using a different unit system, the simulation will not produce the expected results. To ensure that you are using **SI** units:

1. In the **View** menu, click **Units**.

2. In the **Units** sub-menu, verify that **SI** is selected.

At any time, you can switch between SI and English units. The data is stored once in the original system, so there is no loss of accuracy when you switch units.

Manipulating the 3D Image

- To **spin** the 3D model, select ⊕ then **left-click** on the model and move the mouse. The model will spin as though you have selected a point on a sphere.

- To **zoom**, select 🔍 (or hold the **ALT** key) and drag the mouse vertically. Select 🔍 then click and drag to define a zoom box.

- To **move** the model, select ⊕ (or hold the **SHIFT** key) and drag to reposition the model in the window.

- To change the focus of the view, select an object(s) and then select 🔲 to define a smaller viewing sphere around the selected objects. Selecting ⁎ will reset the view to include the entire model.

- At any time, selecting ⁎ (or pressing **CTRL + R**) will **reset** the model.

You can also use Smokeview and person-oriented controls. See the PyroSim User Manual for instructions.

FDS Concepts and Nomenclature

Material

Materials are used to define thermal properties and pyrolysis behavior.

Surface

Surfaces are used to define the properties of solid objects and vents in your FDS model. The surface can use previously defined materials in mixtures or layers. By default, all solid objects and vents are inert, with a temperature that is fixed at the initial temperature.

Obstruction

Obstructions are the fundamental geometric representation in the Fire Dynamics Simulator (FDS) [FDS-SMV Official Website]. Obstructions are rectangular solids defined by two points in 3D space. Surface properties are assigned to each face of the obstruction. Devices and control logic can be defined to create or remove an obstruction during a simulation.

When creating a model, the geometry of an obstruction does not need to match the geometry of the mesh used for the solution. However, the FDS solution will align all geometry with the solution mesh. In the FDS analysis, all faces of an obstruction are shifted to correspond to the nearest mesh cell. Thus, some obstructions may become thicker in the analysis; others may become thin and correspond to a single cell face, which has the potential to introduce unwanted gaps into a model. These ambiguities can be avoided by making all geometry correspond to the mesh spacing.

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.

Computational Mesh

FDS calculations are performed within a domain made of rectilinear volumes called *meshes*. Each mesh is divided into rectangular *cells*. Two factors that must be considered when choosing the cell size are the required resolution to define objects in the model (obstructions) and the desired resolution for the flow dynamics solution (including local fire induced effects). Although geometric objects (obstructions) in an FDS analysis can be specified using dimensions that do not fall on cell coordinates, during the FDS solution, all faces of an obstruction are shifted to the closest cell. If an obstruction is very thin, the two faces may be approximated on the same cell face. The FDS Users Guide [McGrattan, Klein, Hostikka, and Floyd, 2009] recommends that, for full functionality, obstructions should be specified to be at least one cell thick. As a result, the cell size must be selected small enough to reasonably represent the problem geometry. In addition, cells should be as close to cubes as possible.

Whether the cell size is sufficient to resolve the flow dynamics solution can only be determined by a **grid sensitivity study**. A discussion of model sensitivity to mesh size is given in Chapter 5 of Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications [U.S. NRC, 2007]. It is the responsibility of the analyst to perform a sensitivity study as part of any simulation.

Chapter 2. Example Problems Provided with FDS 5

If you want to have fun and quickly run some example analyses, you can import FDS 5 input files that are included with the NIST installation of FDS 5. These examples are provided with the PyroSim distribution in the PyroSim 2009\samples\FDS5 folder. In this chapter, we illustrate this using a few examples; you can import additional examples. Studying these examples is a good way to learn about input for different types of analyses.

To open any of these examples:

1. Obtain the desired FDS 5 input file.

2. Start PyroSim.

3. On the **File** menu, click **Import** and select **FDS File**.

4. In the **Open File** dialog, click the FDS input file. PyroSim will import the file. PyroSim will warn if any records on the FDS file cannot be imported.

5. On the **File** menu, click **Save**. Save the model in a new directory.

6. On the **FDS** menu, click **Run FDS...**. This will start the analysis. At the end of the analysis, SmokeView will be launched to view the results.

Important

If PyroSim doesn't fully support a FDS input file, it will issue a warning that includes information about how it handled the unfamiliar records. In some cases, PyroSim is able to add records to the **Additional Records** section and the simulation is unaffected. However, if records are "dropped" (i.e. omitted from the simulation) the simulation results will no longer represent the original intent of the example problem.

Ethanol Pan Fire

The **ethanol_pan.fds** example illustrates an ethanol pan fire. The model is shown in Figure 2.1. A typical result is shown in Figure 2.2. The FDS input file can be downloaded at: http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Validation/VU_Ethanol_Pan_Fire/FDS_Input_Files/VU_Ethanol_Pan_Fire.fds.

Figure 2.1. Ethanol pan model
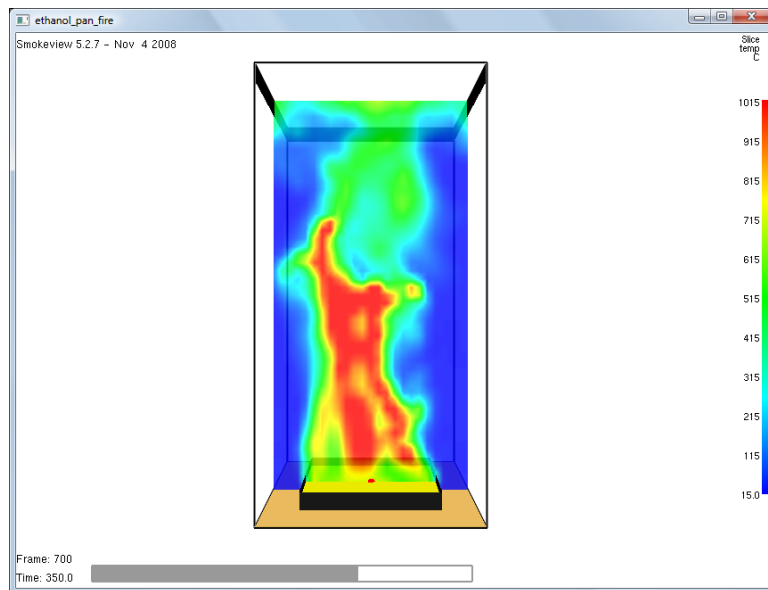
Figure 2.2. Ethanol pan results
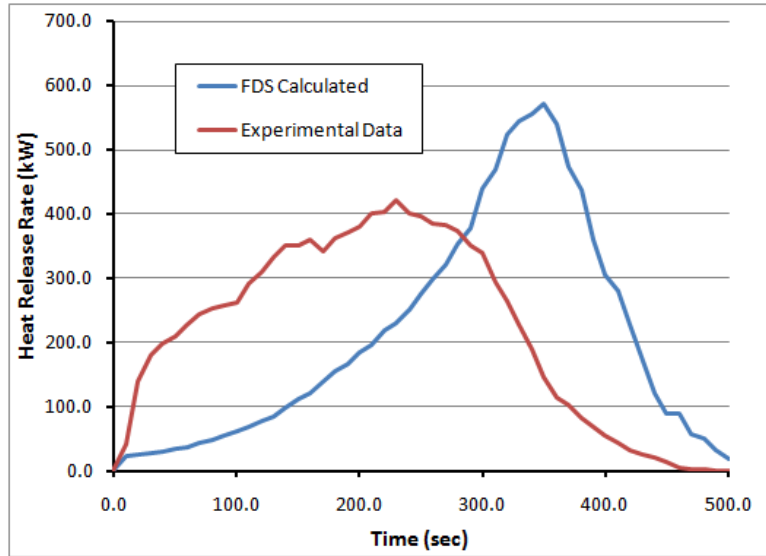
Figure 2.3. Comparison of calculated and measured heat release rates

Box Burn Away

The **box_burn_away.fds** example illustrates burning of a foam box. The model is shown in Figure 2.4. A typical result is shown in Figure 2.5. The FDS input file can be downloaded at: http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Verification/Fires/box_burn_away.fds.
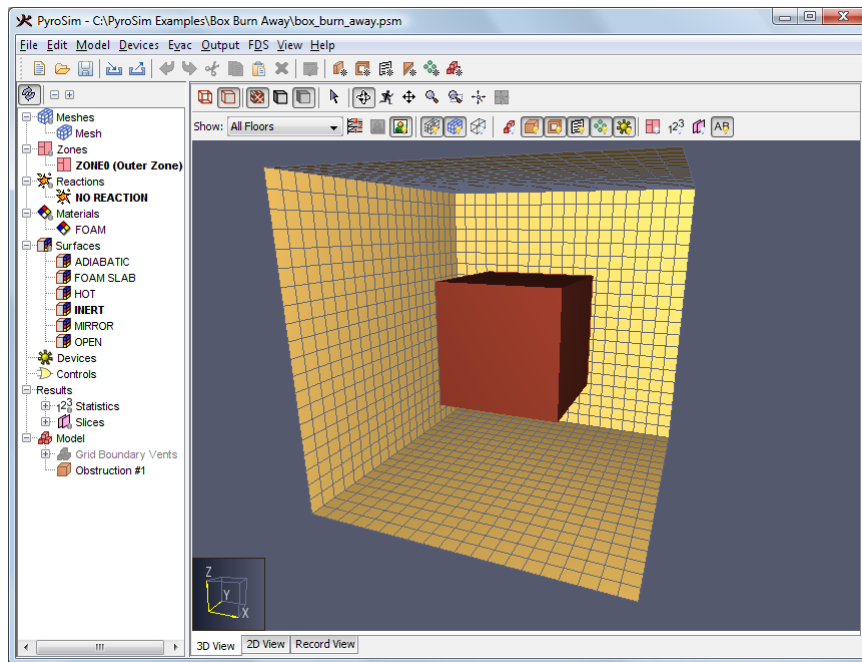
Figure 2.4. Foam box burn away model
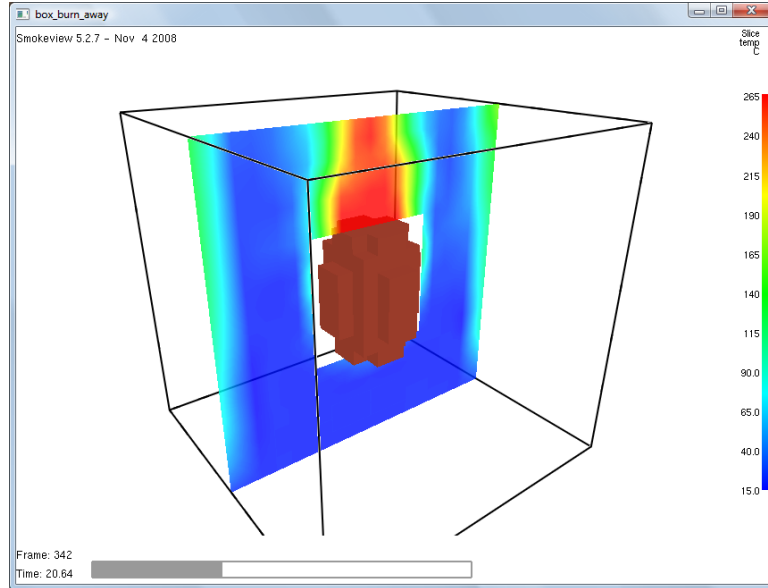
Figure 2.5. Foam box burn away results

Insulated Steel Column

The **insulated_steel_column.fds** example illustrates heat conduction into a column. The model is shown in Figure 2.6. A typical result is shown in Figure 2.7. The FDS input file can be downloaded at: http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Verification/Heat_Transfer/insulated_steel_column.fds.
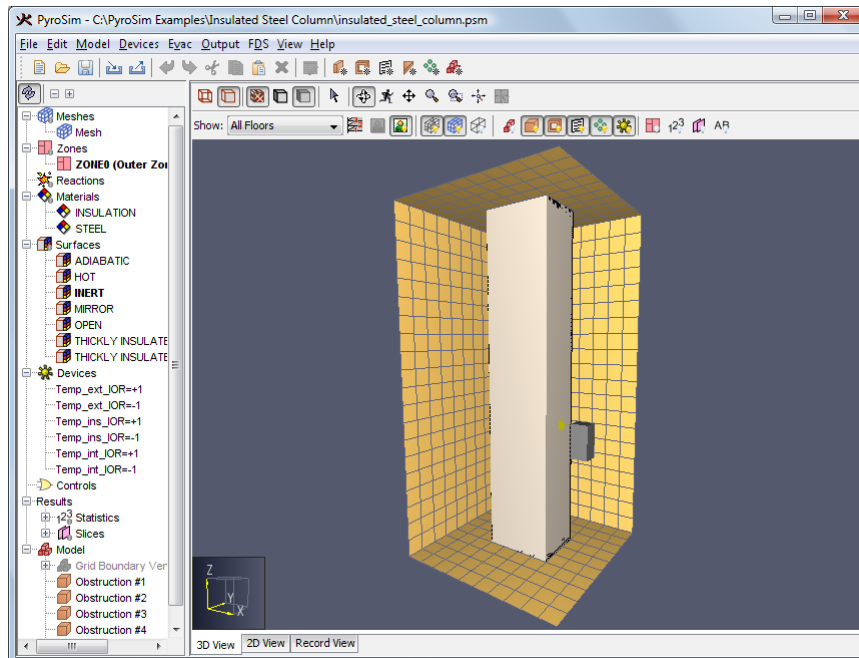
Figure 2.6. Insulated Steel Column model
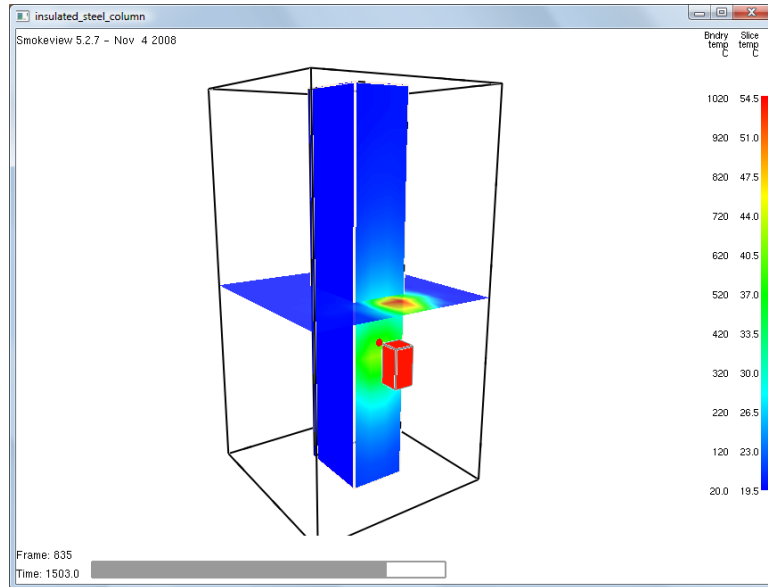
Figure 2.7. Insulated Steel Column results

Water Cooling

The **water_cooling.fds** example illustrates water cooling spray onto a wall. The model is shown in Figure 2.8. A typical result is shown in Figure 2.9 The FDS input file can be downloaded at: http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Verification/Sprinklers_and_Sprays/water_cooling.fds
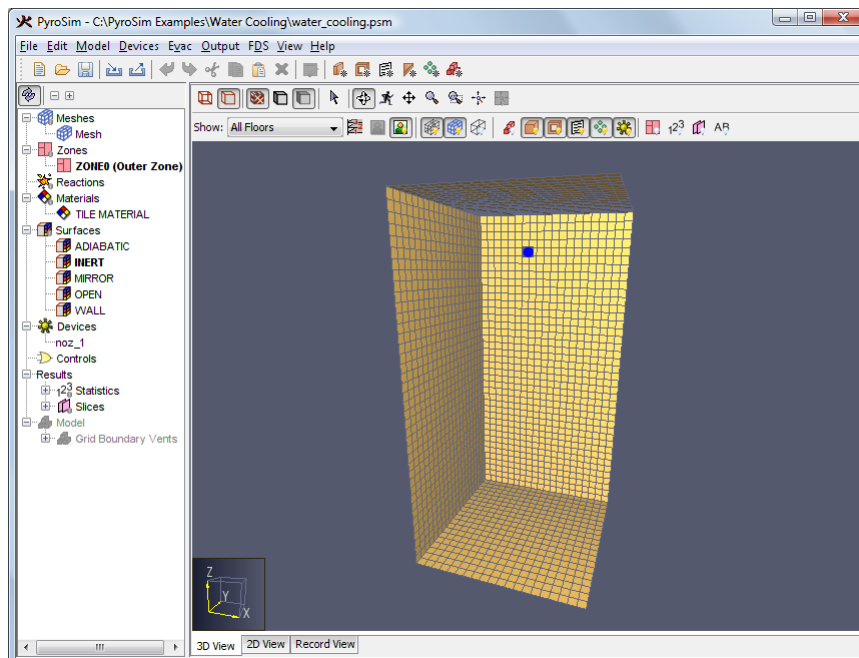
Figure 2.8. Water cooling model
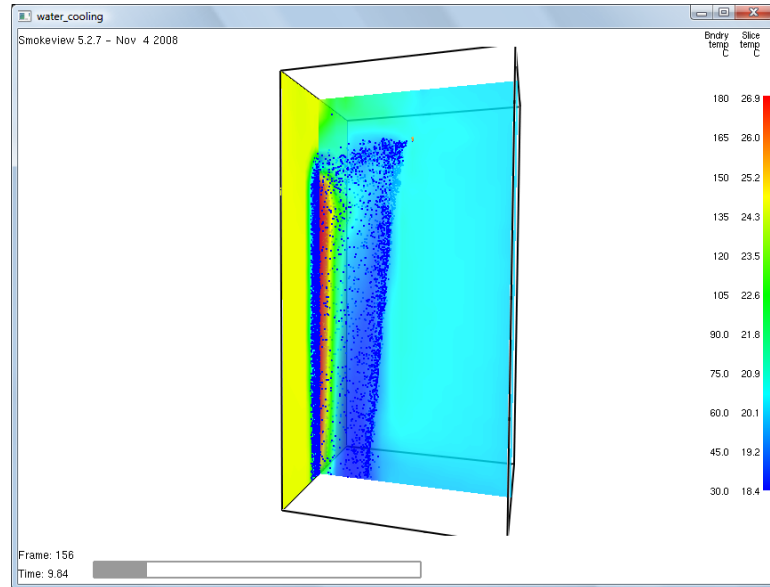
Figure 2.9. Water cooling results

Evacuation

PyroSim supports input for the FDS+Evac evacuation model. The **evac_example1a.fds** example illustrates evacuation from a simple room. The model is shown in Figure 2.10. A typical result is shown in Figure 2.11 The FDS input file can be downloaded at: http://virtual.vtt.fi/virtual/proj6/fdsevac/fds5/examples/evac_example1a.fds. For more information on FDS+Evac, visit: http://www.vtt.fi/proj/fdsevac/index.jsp.

Please note that Thunderhead Engineering provides an agent-based evacuation model in the Pathfinder code (http://www.thunderheadeng.com/pathfinder/index.html), which can easily import existing FDS model geometry. The FDS+Evac and Pathfinder models can be used to provide alternate approaches to evacuation modeling.

Figure 2.10. Evacuation modeling example
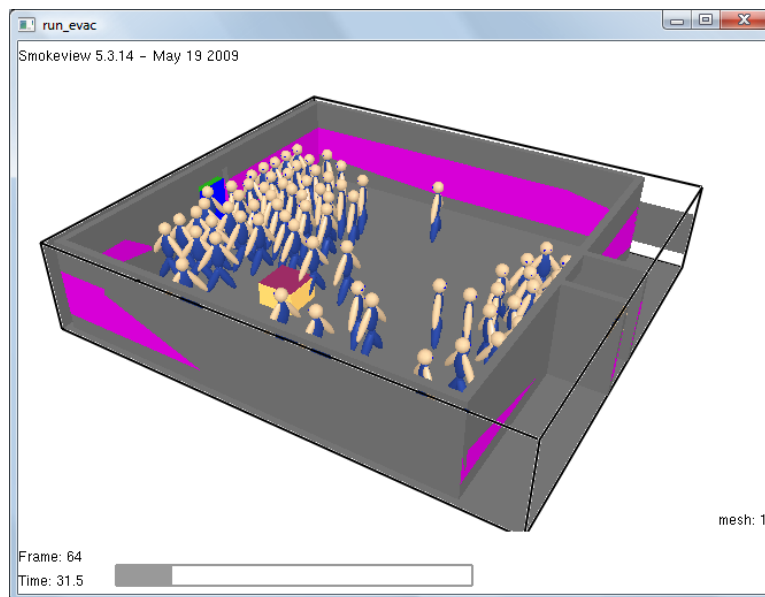
Figure 2.11. Evacuation modeling results

Chapter 3. Burner Fire

In this tutorial you will create a 500 kW burner fire and measure the temperature in the center of the plume at a height of 1.5 m.

This tutorial demonstrates how to:

• Create a burner fire.

• Add a thermocouple.

• Add a slice plane for temperature visualization.

• View 3D results using Smokeview.

• View 2D results using PyroSim.

Figure 3.1. Burner fire in this example

Before you begin, ensure that you are using **SI** units (see Chapter 1).

Create the Mesh

In this example we will use mesh cells that are 0.13 m across. This value is approximately 1/5 of the characteristic diameter (D*) for a 500 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, [U.S. NRC, 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. Accept the default name **MESH**. Click **OK**.

4. In the **Min X** box, type `-1.0` and in the **Max X** box, type `1.0`.

5. In the **Min Y** box, type `-1.0` and in the **Max Y** box, type `1.0`.

6. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `3.0`.

7. In the **X Cells** box, type `15`.

8. In the **Y Cells** box, type `15`.

9. In the **Z Cells** box, type `24`.

10. Click **OK** to save changes and close the **Edit Meshes** dialog.

Figure 3.2. Creating the mesh

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 500 kW/m2.

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New...**.

3. In the **Surface Name** box, type `burner`, Figure 3.3.

4. In the **Surface Type** list, select **Burner**.

5. Click **OK** to create the new default burner surface.
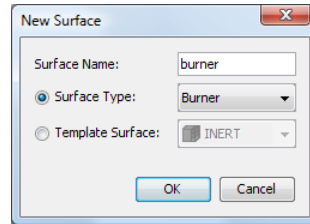
Figure 3.3. Creating a new burner surface

1. In the **Description** box, type `500 kW/m2 burner`, Figure 3.4

2. Click **OK** to save changes and close the **Edit Surfaces** dialog.
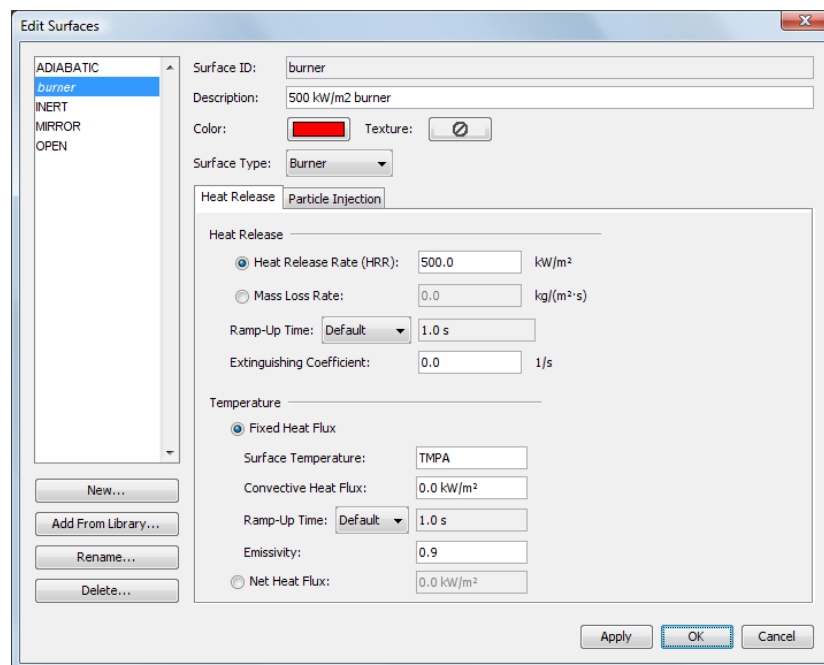
Figure 3.4. Defining parameters for the burner surface

Create the Burner Vent

In this example, we use a vent and the previously created **burner** surface to define the fire. (Recall that, in FDS, a "vent" can be a 2D surface used to apply boundary conditions on a rectangular patch.)

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

2. In the **Description** box, type `burner vent`, Figure 3.5.

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**.

5. In the **Min X** box, type `-0.5` and in the **Max X** box, type `0.5`.

6. In the **Min Y** box, type `-0.5` and in the **Max Y** box, type `0.5`.

7. Click **OK** to create the new burner vent.
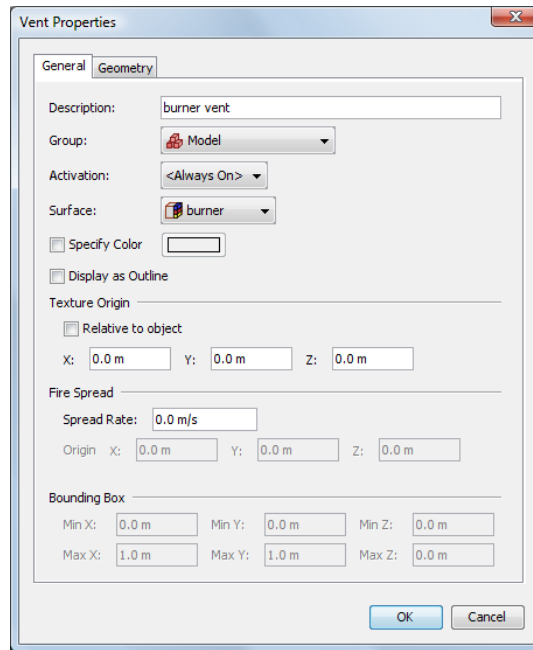
Figure 3.5. Creating the burner vent

Create the Top Vent

The top of the mesh is an open boundary.

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

2. In the **Description** box, type `open top`.

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 **Z** and type `3.0`.

5. In the **Min X** box, type `-1.0` and in the **Max X** box, type `1.0`.

6. In the **Min Y** box, type `-1.0` and in the **Max Y** box, type `1.0`.

7. Click **OK** to create the open vent.

Add a Thermocouple

1. On the **Devices** menu, click **New Thermocouple...**.

2. In the **Device Name** box, type `thermocouple at 1.5 m`.

3. On the **Location** row, in the **Z** box, type `1.5`.

4. Click **OK** to create the thermocouple. It will appear as a yellow dot in the center of the model. Click the **Show Labels** button to toggle the labels on and off.

Add a Temperature Slice Plane

1. On the **Output** menu, click **Slices...**.

2. In the **XYZ Plane** column, click the cell and select **Y**.

3. In the **Plane Value** column, click the cell and type `0.0`.

4. In the **Gas Phase Quantity** column, click the cell and select **Temperature**.

5. In the **Use Vector?** column, click the cell and select **NO**.

6. Click **OK** to create the slice plane. Click the **Show Slices** button to toggle the slice planes on and off.

Rotate 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** button in the **3D View** and drag to rotate the model.
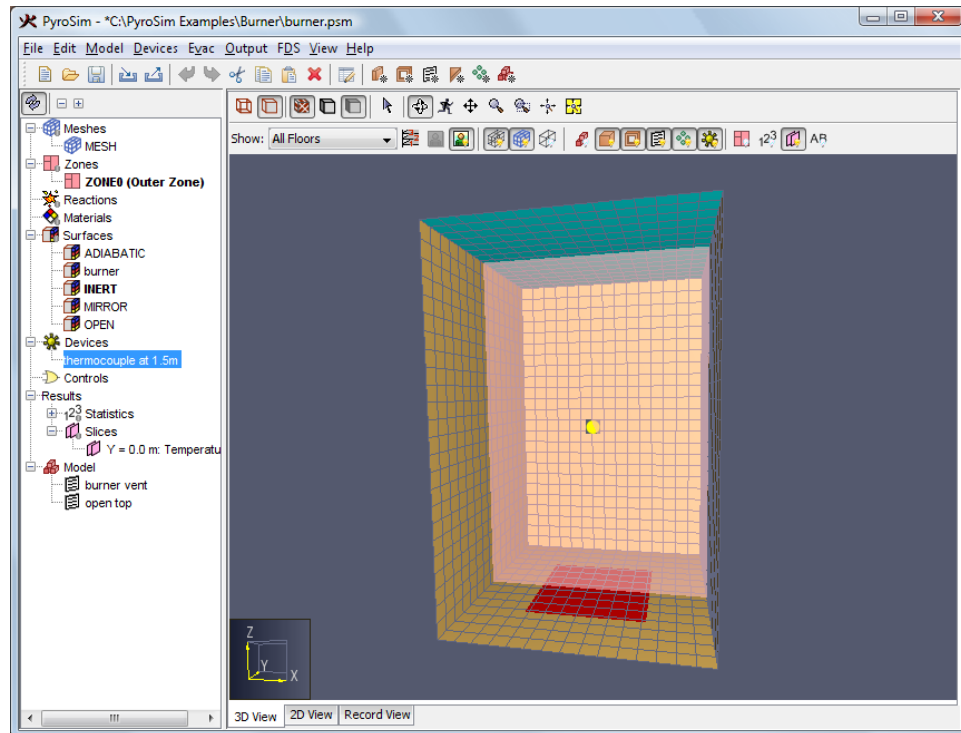
Figure 3.6. The model after rotating. The burner is shown in red and the thermocouple as a yellow dot. The slice plane is semi-transparent and the open vent is blue.

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 `Burner` folder and name the file `burner.psm`.

3. Click **OK** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS...**.

2. Save the FDS input file with the name `burner.fds.`.

3. Click **Save** to save the FDS input file and begin the simulation.

4. 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 1 minute to run depending on computing hardware, Figure 3.7.

5. When the simulation is complete, Smokeview will start and display a 3D still image of the model, Figure 3.8..
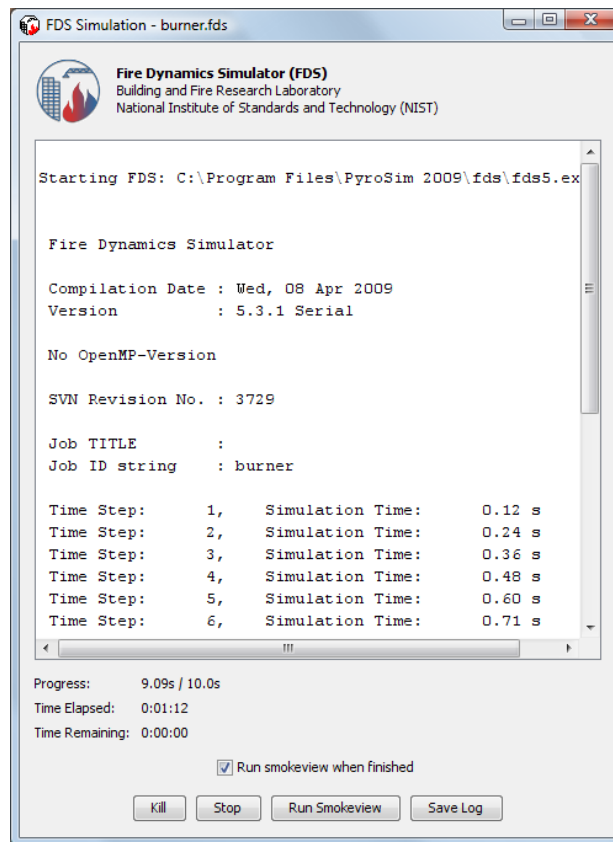
Figure 3.7. The simulation dialog during the analysis.
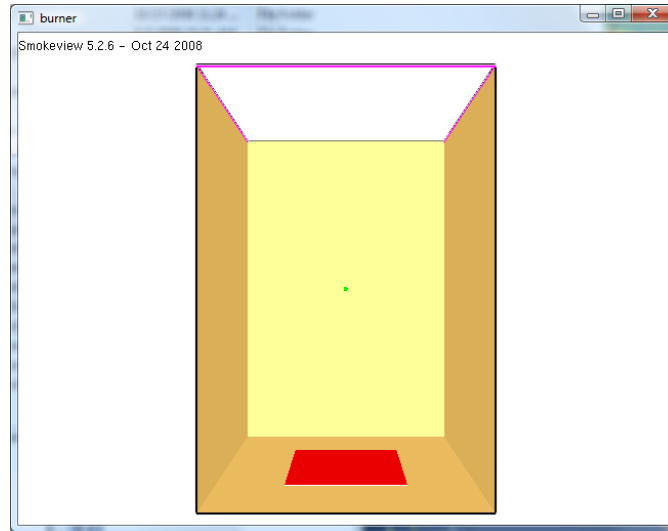
Figure 3.8. The initial Smokeview display.

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. 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**.

4. To reset Smokeview, right-click to activate the menu, then click **Load/Unload > Unload All**.
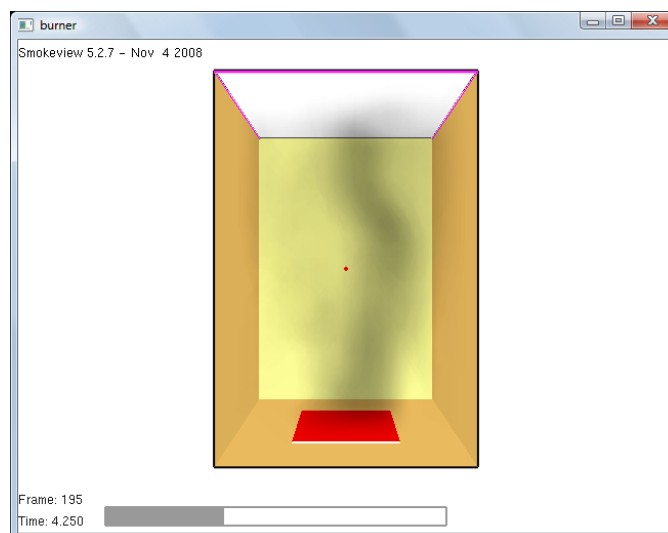
Figure 3.9. 3D smoke in the model.

View Temperature Slice Plane

1. In the Smokeview window, right-click to activate the menu.

2. In the menu, click **Load/Unload > Slice File > TEMPERATURE > Y=0.06667**. This will start an animation of the temperature slice plane. Note that the Y coordinate of the plane was shifted by FDS to correspond to the center of a cell.
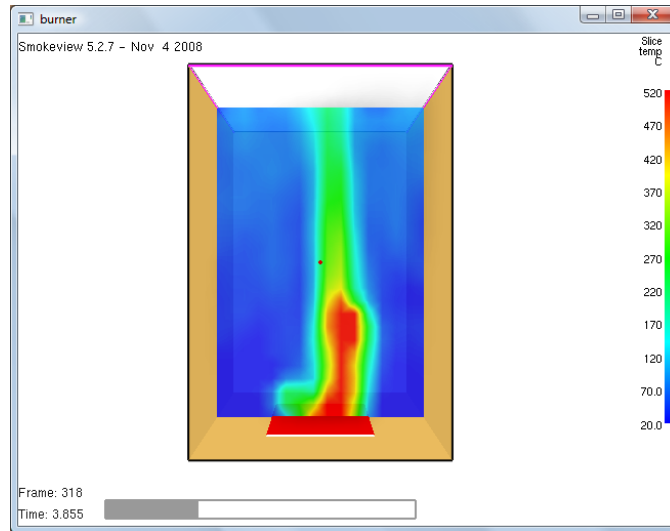
Figure 3.10. Temperature contours on the slice plane.

View Temperature Measurements

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 **burner_devc.csv** and click **Open** to view the temperature device output.
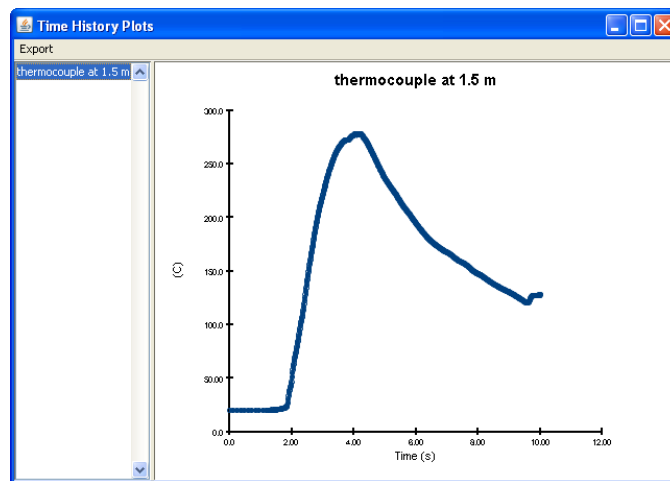
Figure 3.11. Temperature time history plot.

Chapter 4. Air Movement

In this tutorial you will create a simple air flow using a supply vent and an "open" vent.

This tutorial demonstrates how to:

• Create vents.

• Add slice planes for velocity visualization.

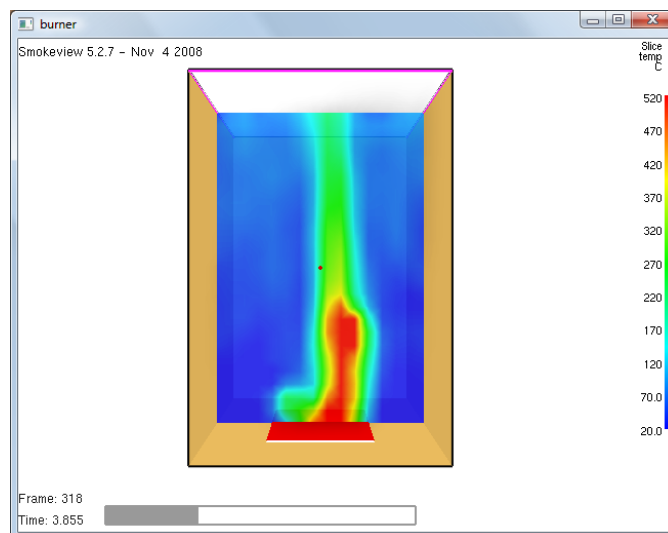• View 3D results using Smokeview.
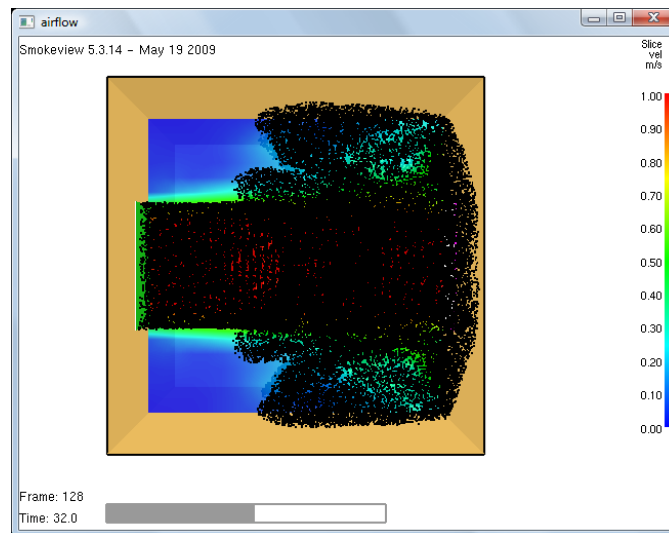
Figure 4.1. 3D visualization of air flow in this example

Before you begin, ensure that you are using **SI** units (see Chapter 1).

Create Mesh

In this example we will use a 10 m x 10 m x 10 m mesh with 0.5 m cells.

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 `0.0` and in the **Max X** box, type `10.0`.

5. In the **Min Y** box, type `0.0` and in the **Max Y** box, type `10.0`.

6. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `10.0`.

7. In the **X Cells** box, type `20`.

8. In the **Y Cells** box, type `20`.

9. In the **Z Cells** box, type 20.

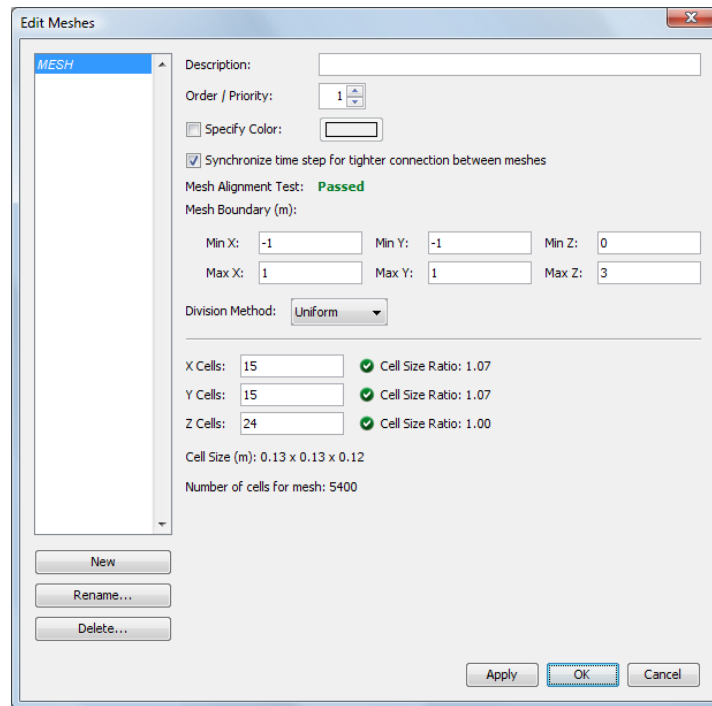10. Click **OK** to save changes and close the **Edit Meshes** dialog.
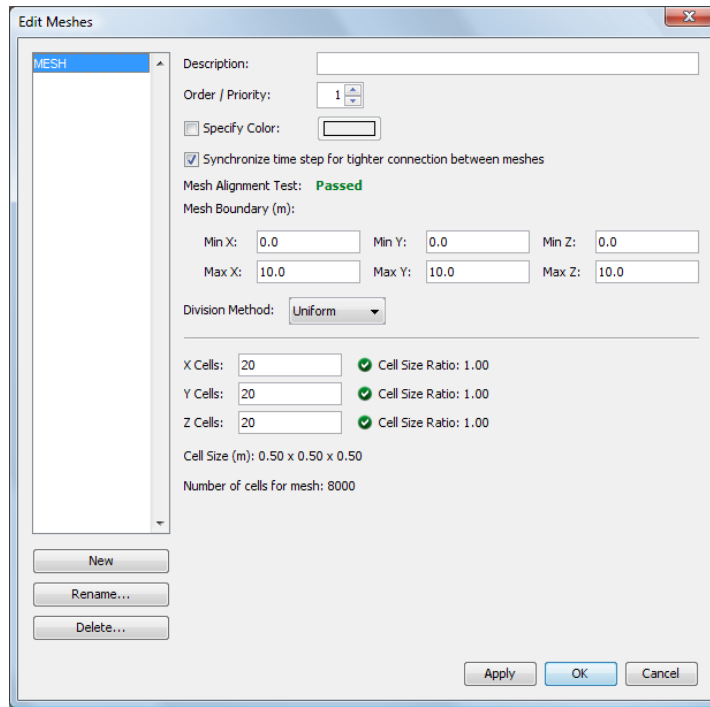
Figure 4.2. Creating the mesh

Create the Supply Surface

Surfaces are used to define the properties of objects in your FDS model. Supply surfaces are used to blow air into the domain. In this example, we will define a supply surface with a velocity of 1.0 m/s.

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New...**.

3. In the **Surface Name** box, type Blow.

4. In the **Surface Type** list, select **Supply**.

5. Click **OK** to create the new supply surface.
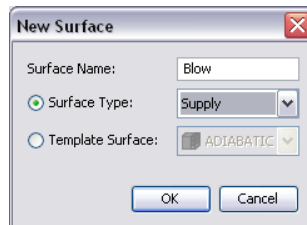
Figure 4.3. Naming the new supply surface

1. In the **Description** box, type `1.0 m/s supply`.

2. In the **Specify Velocity** box, type `1.0`.
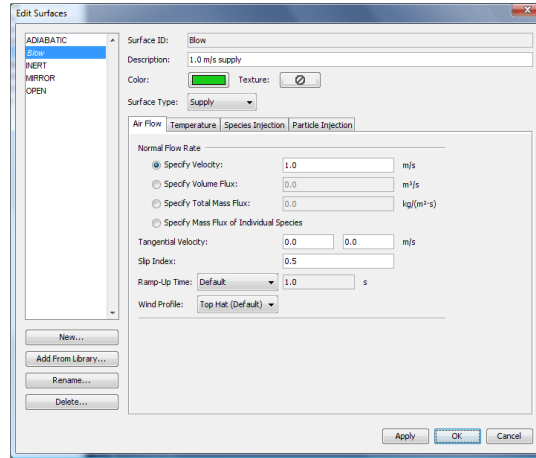
Figure 4.4. Creating a new supply surface

To emit particles

1. Click the **Particle Injection** tab.

2. Select the **Emit Particles** checkbox.

3. In the **Particle Type** list, select **Tracer**.

4. Click **OK**.

Create Vents

Vents are used to define flow conditions in a model. Vents are 2D objects and must be aligned with one of the model planes. In this example, we will use a vent and the previously created **Blow** surface to create the wind source.

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

2. In the **Description** box, type `Vent blow`.

3. In the **Surface** list, select **Blow**. This specifies that the previously created surface will define the properties of the vent.

4. Click on the **Geometry** tab. In the **Plane** list, select **X** and set the value to `0.0`.

5. In the **Min Y** box, type `3.0` and in the **Max Y** box, type `7.0`.

6. In the **Min Z** box, type `3.0` and in the **Max Z** box, type `7.0`.
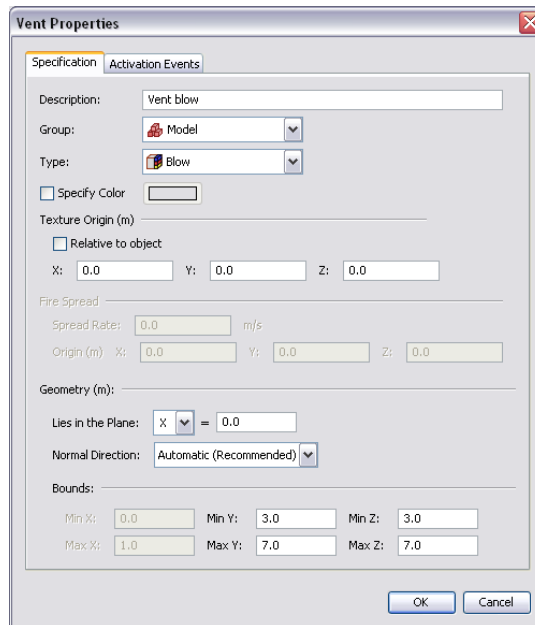
7. Click **OK**.

Figure 4.5. Creating the new blow vent.

To create the open (exhaust) vent:

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

2. In the **Description** box, type `Vent open`.

3. In the **Surface** list, select **Open**.

4. Click on the **Geometry** tab. In the **Plane** list, select **X** and type `10.0`.

5. In the **Min Y** box, type `3.0` and in the **Max Y** box, type `7.0`.

6. In the **Min Z** box, type `3.0` and in the **Max Z** box, type `7.0`.
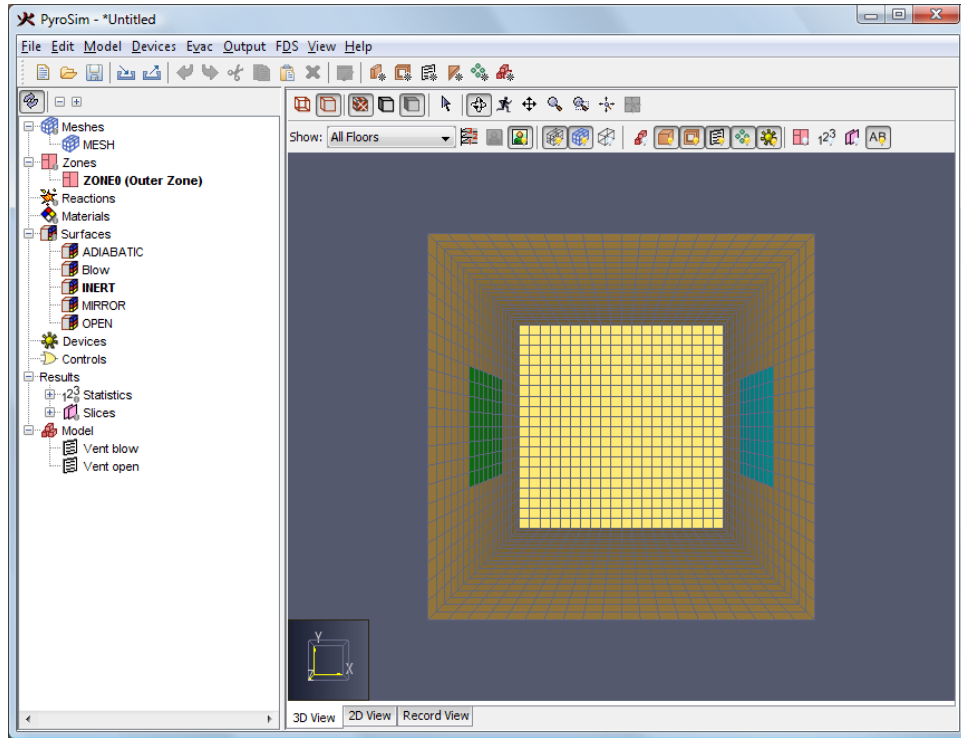
7. Click **OK**.

Figure 4.6. The resulting view

Create Slice Records

1. On the **Output** menu, click **Slices...**.

2. In the **XYZ** column, select **Y**.

3. In the **Plane Value** column, type 5.0.

4. In the **Gas Phase Quantity** column, select **Velocity**.

5. In the **Use Vector?** column, select **NO**.

6. Click **OK** to create the slice plane.
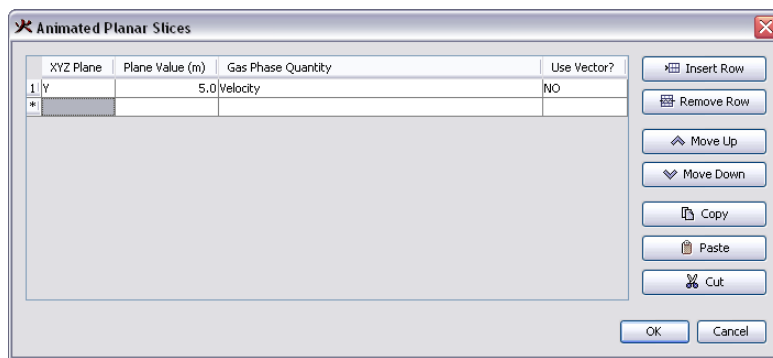
Figure 4.7. Slice record data

Specify Simulation Properties

To define the end time:

1. On the **FDS** menu, click **Simulation Parameters...**.

2. In the **Simulation Title** box, type `Air flow`.

3. In the **End Time** box, type `60.0`.

4. Click **OK**.
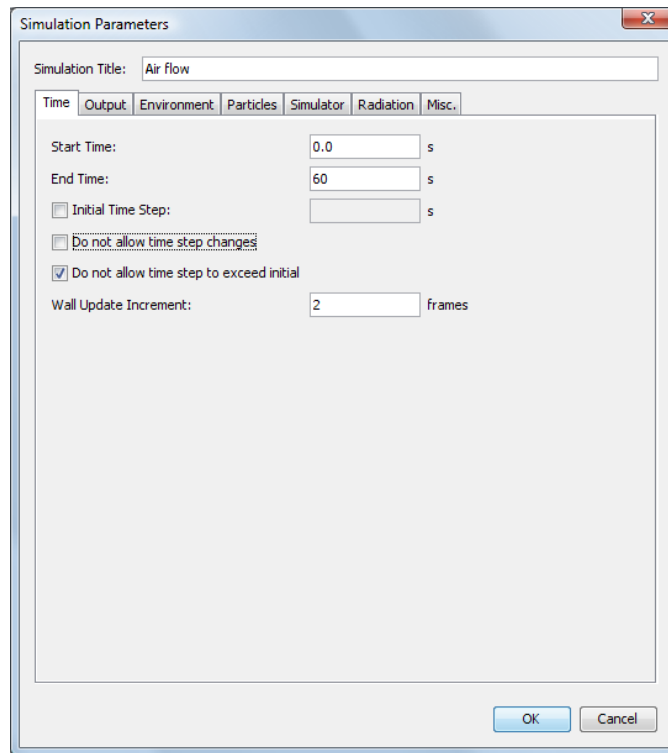
Figure 4.8. Defining the simulation parameters

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 folder names `Airflow` and name the file `airflow.psm`.

3. Click **Save** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS...**.

2. Choose a location to save FDS input file. For this example, we will name the file `airflow.fds`.

3. Click **OK** to save the FDS input file and begin the simulation.

4. The **FDS Simulation** dialog will appear and display the progress of the simulation. This should take approximately 30 seconds to run depending on computing hardware.

5. When the simulation is complete, Smokeview will launch automatically and display a 3D image of the model.

View Particles

1. In the Smokeview window, right-click to activate the menu.

2. In the menu, click **Load/Unload > Particle File > particles** to load the particle data.

View Slice Data

1. In the Smokeview window, right-click to activate the menu.

2. In the menu, click **Load/Unload > Slice File > Velocity > Y=5.0**.

Unload the particle data to view only the velocity contours.
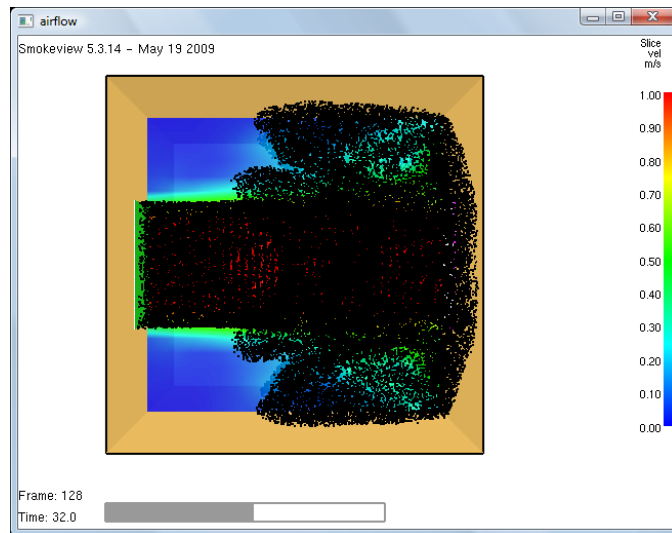
Figure 4.9. View in Smokeview
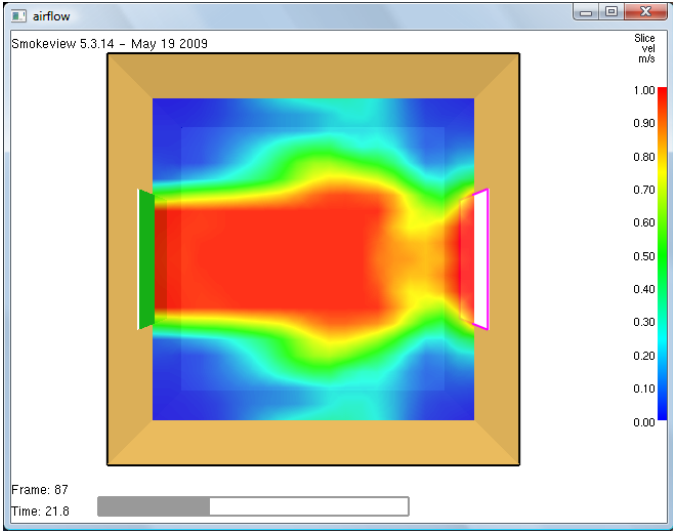
Figure 4.10. View in Smokeview of velocity contours

Chapter 5. 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.
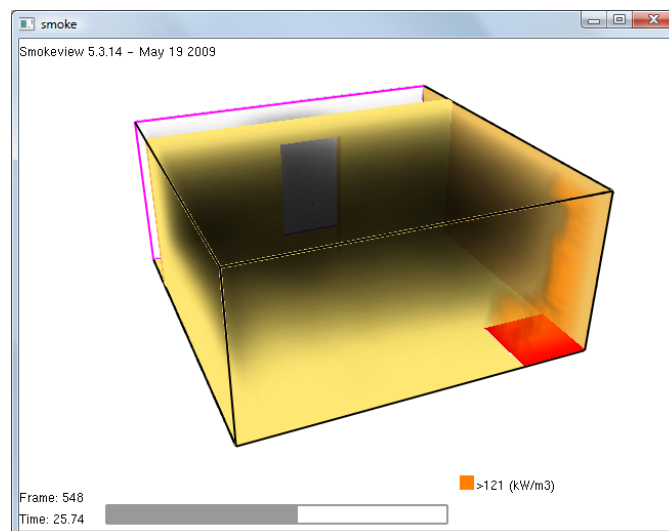
Figure 5.1. Smoke in the model

Before you begin, ensure that you are using **SI** units (see Chapter 1).

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 5.2.

4. In the **Surface Type** list, select **Burner**.

5. Click **OK** to create the new default burner surface.

Figure 5.2. Creating a new burner surface

1. In the **Description** box, type `800 kW/m2 burner`, Figure 5.3

2. In the **Heat Release Rate (HRR)** box, type `800`.

3. Click **OK** to save changes and close the **Edit Surfaces** dialog.
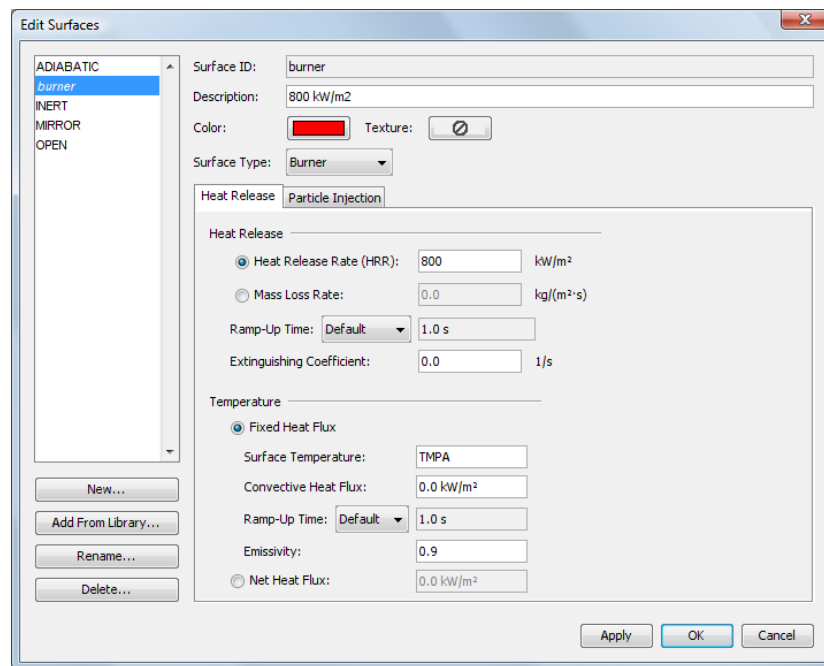
Figure 5.3. Defining parameters for the burner surface

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 5.4.

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.
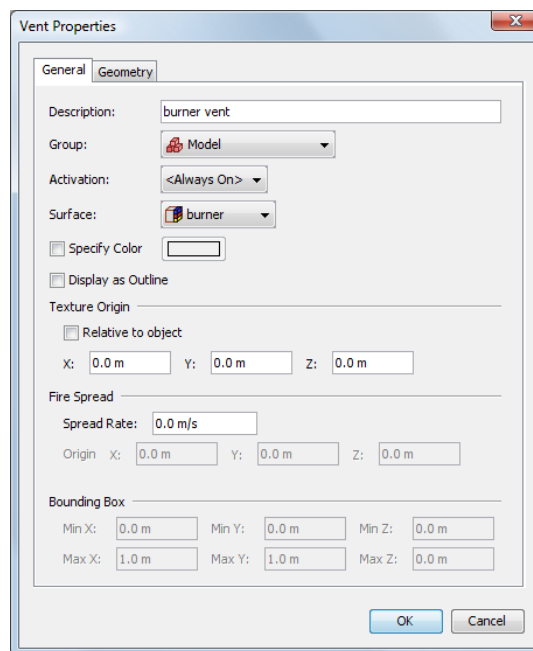
Figure 5.4. Creating the burner vent

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, [U.S. NRC, 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 5.5).

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.
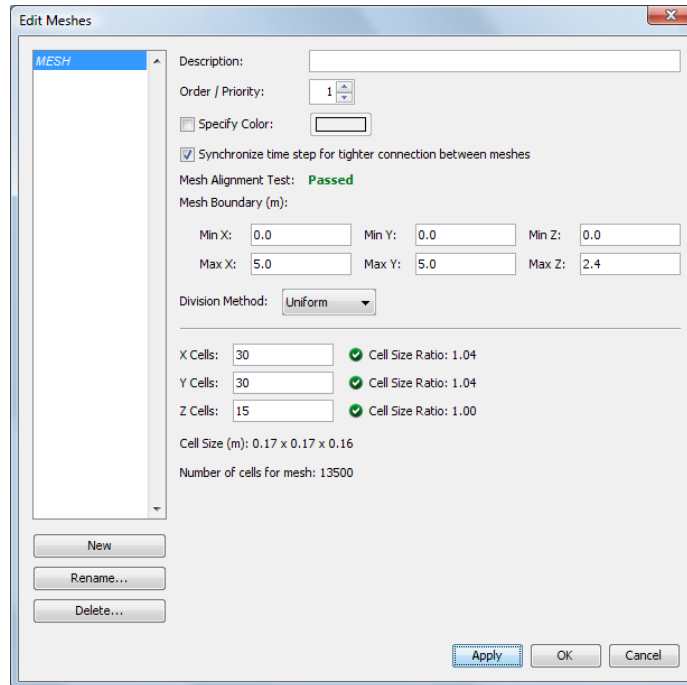
Figure 5.5. Creating the mesh

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 5.6.

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.
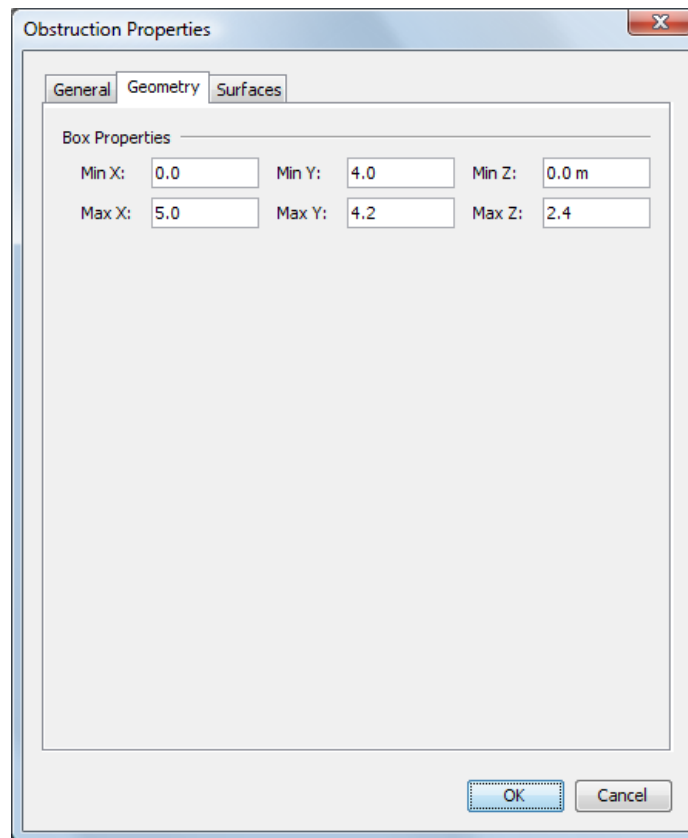
Figure 5.6. Creating the wall

Add the Door

In FDS **holes** are used to define openings through solid objects. In this example, we will use a hole to define a door.

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

2. In the **Description** box, type `door`.

3. Click on the **Geometry** tab. In the **Min X** box, type `2.0` and in the **Max X** box, type `3.0`.

4. In the **Min Y** box, type `3.9` and in the **Max Y** box, type `4.3`.

5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.

6. Click **OK** to create the doorway hole.

Rotate 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** button in the **3D View** and drag to rotate 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.
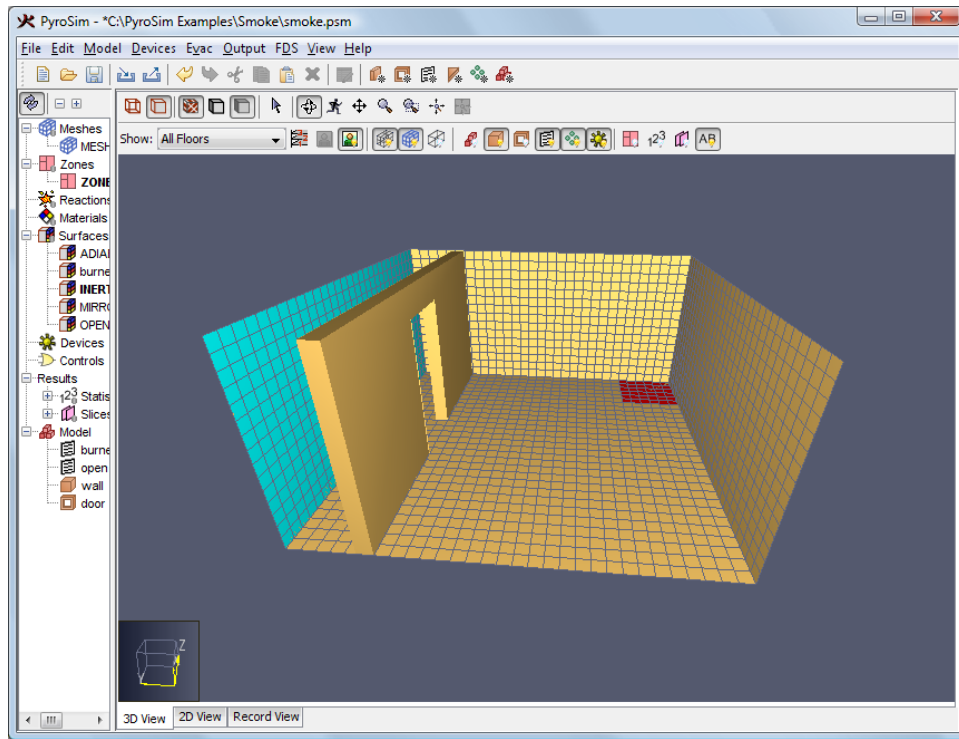
Figure 5.7. The model after rotating. The burner is shown in red and the top vent in blue.

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. Choose a location to save simulation data. For this example, we will name the file `c:\Smoke\smoke.fds`..

3. Click **OK** to save the FDS input file and begin the simulation.

4. 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 1 minute to run depending on computing hardware, Figure 5.8.

5. When the simulation is complete, Smokeview should launch automatically and display a 3D still image of the model, Figure 5.9..
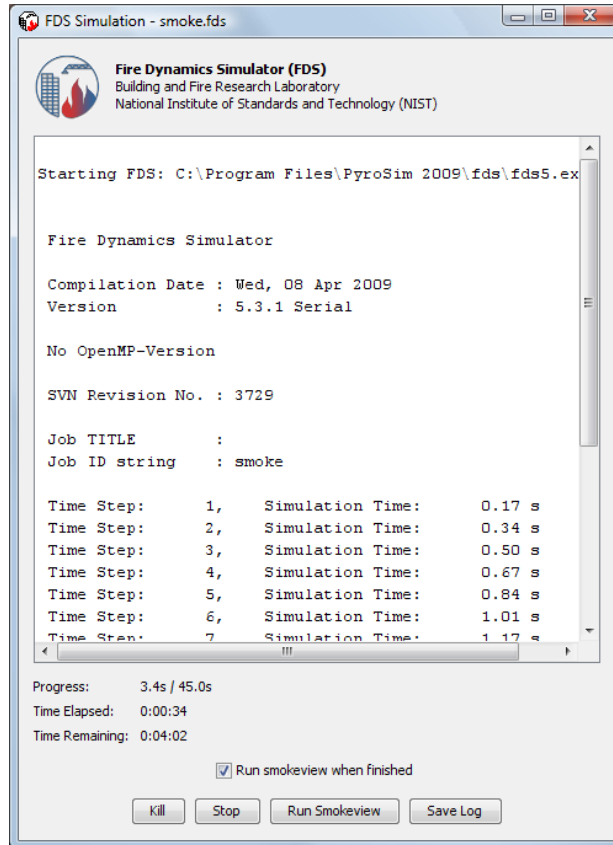
Figure 5.8. The simulation dialog during the analysis.
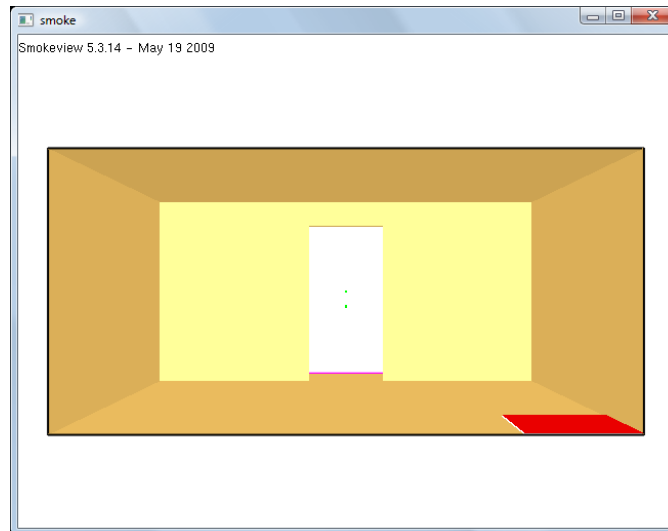
Figure 5.9. The initial Smokeview display.

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.

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**.
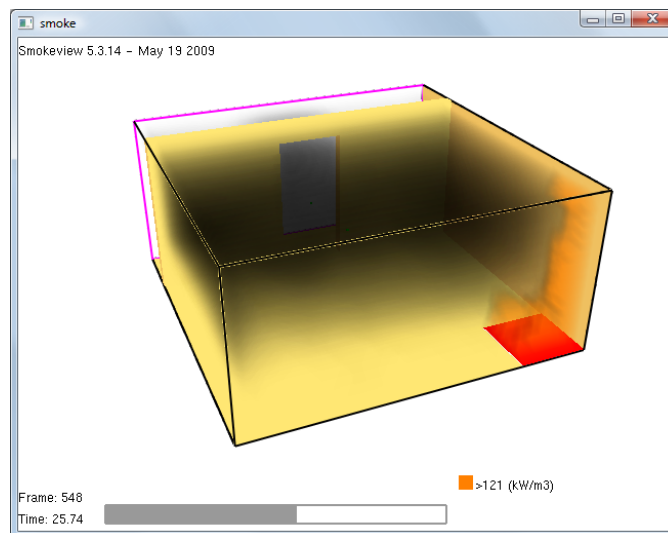
Figure 5.10. 3D smoke in the model.

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 5.11.

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 5.12.
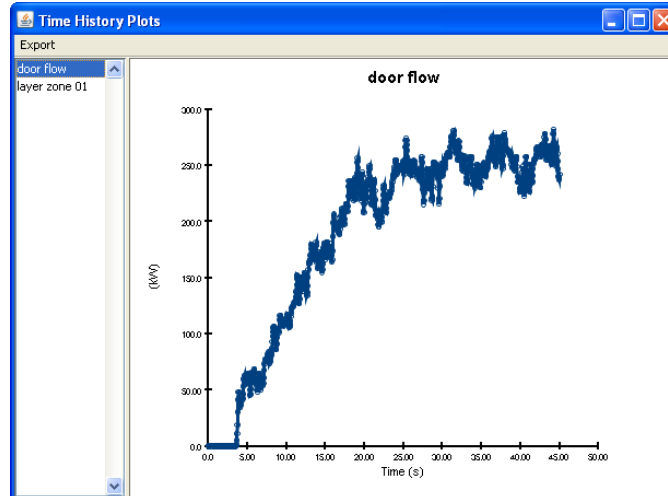
Figure 5.11. Time history plot of heat flow through the door.
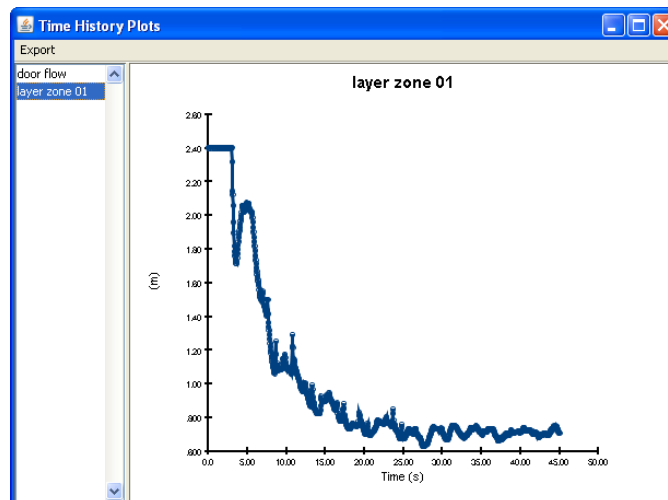
Figure 5.12. Time history plot of smoke layer height.

Chapter 6. Room Fire

This tutorial demonstrates how to:

- Import properties from a database.

- Define a combustion reaction.

- Replicate and rotate obstructions (furniture).

- Use a hole to represent an open door.

- Define an open surface on the exterior of the model.

- View 3D results using Smokeview.

- View 2D results using PyroSim.
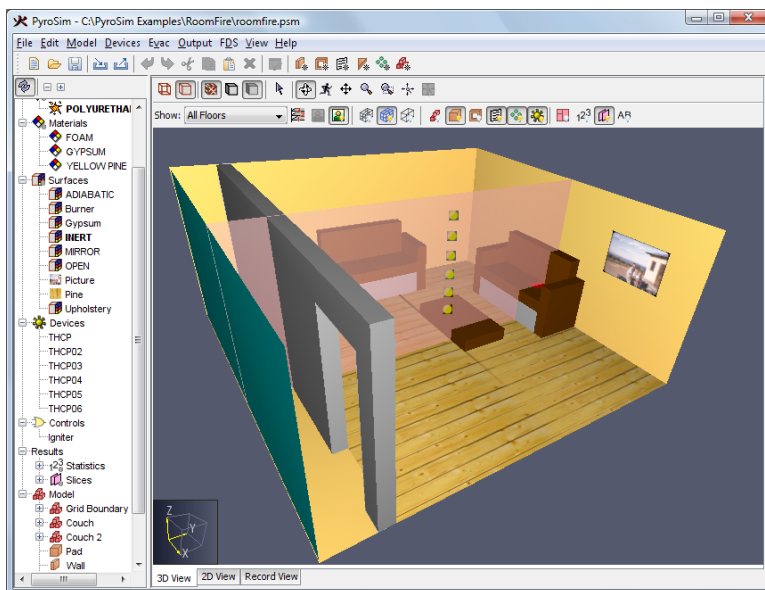
Figure 6.1. Room fire in this example

This example is a simplification of the Roomfire problem provided as an FDS verification problem. You can download the complete FDS verification file at: http://fds-smv.googlecode.com/svn/trunk/FDS/trunk/Verification/Fires/room_fire.fds, then import this file into PyroSim.

Import Reaction and Material Data

PyroSim includes a database file that includes references for the source of the data. We will import selected properties from this file.

1. On the **Model** menu, click **Edit Libraries...**.

2. In the **Category** box, select **Gas-phase Reactions**.

3. Copy the **POLYURETHANE** reaction from the library into the **Current Model**.

4. In the **Category** box, select **Materials**.

5. Copy the **FOAM**, **GYPSUM**, and **YELLOW PINE** materials from the library into the **Current Model**.

6. Close the **PyroSim Libraries** dialog.
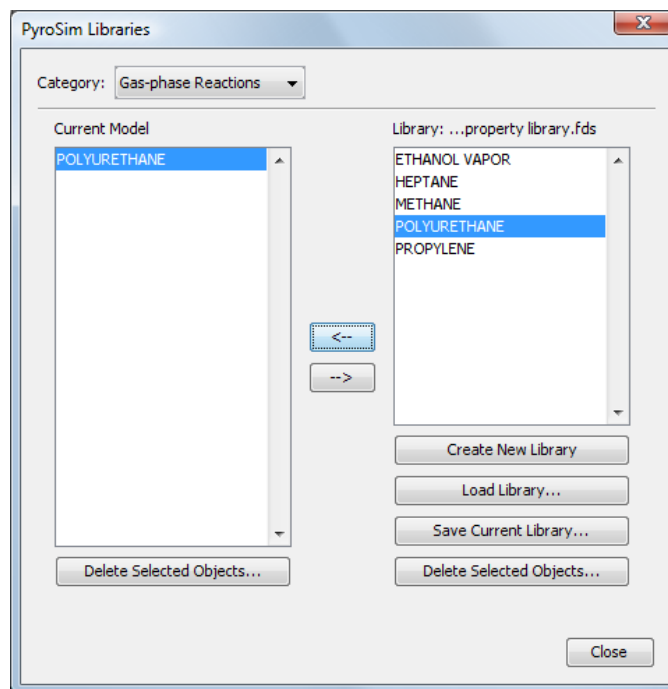
Figure 6.2. Copy the reaction from the library

Save the Model

This is a good time to 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. Name the file `roomfire.psm`.

3. Click **OK** to save the model.

Create the Mesh

In this example we will use mesh cells with a size of 0.10 m. This is geometrically convenient and is fine enough relative to the burner HRR to give moderate numerical accuracy.

1. On the **Model** menu, click **Edit Meshes...**.

2. Click **New** and then **OK** to create a new mesh, see Figure 6.3.

3. In the **Min X** box, type `0.0` and in the **Max X** box, type `5.2`.

4. In the **Min Y** box, type `-0.8` and in the **Max Y** box, type `4.6`.

5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.4`.

6. In the **X Cells** box, type `52`.

7. In the **Y Cells** box, type `54`.

8. In the **Z Cells** box, type `24`.

9. Click **OK** to save changes and close the **Edit Meshes** dialog.
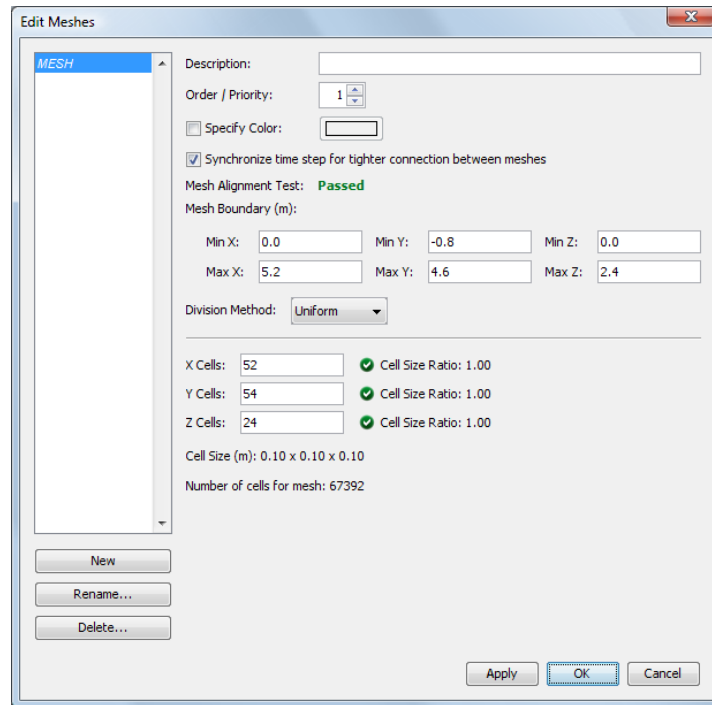
Figure 6.3. Creating the mesh

Specify Combustion Parameters

Since there is only one reaction in the model, by default that will be the reaction used for the analysis. No other action is necessary.

You can double-click on POLYURETHANE to display the properties, Figure 6.4.
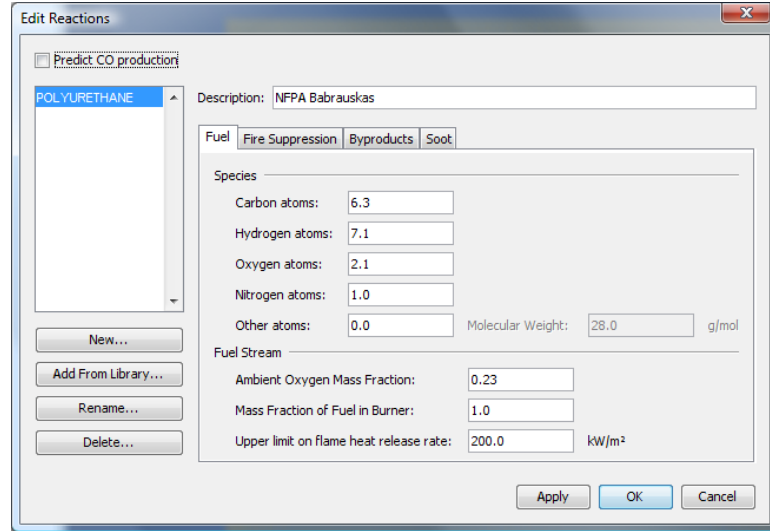
Figure 6.4. The POLYURETHANE reaction parameters

Click the **Cancel** button to close the **Edit Reactions** dialog.

Create Surfaces

Materials, which we have already imported, define physical properties. Surfaces that represent solid objects in the model use the material properties. Vent and burner surfaces are defined directly, without reference to materials.

The floor will be made of yellow pine. To create the surface:

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New**, give the **Surface Name** as Pine, select the **Surface Type** as Layered, and click **OK**.

3. Click on the **Texture** box and select **psm_spruce.jpg**. Click **OK** to close the **Texture** dialog.

4. In the **Material Layers** panel, in the **Thickness** column, type 0.01.

5. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction** column, type 1.0. In the **Material** column, select YELLOW PINE. Click **OK** to close the **Composition** dialog.

6. In the **Edit Surfaces** dialog, click **Apply** to save the changes.
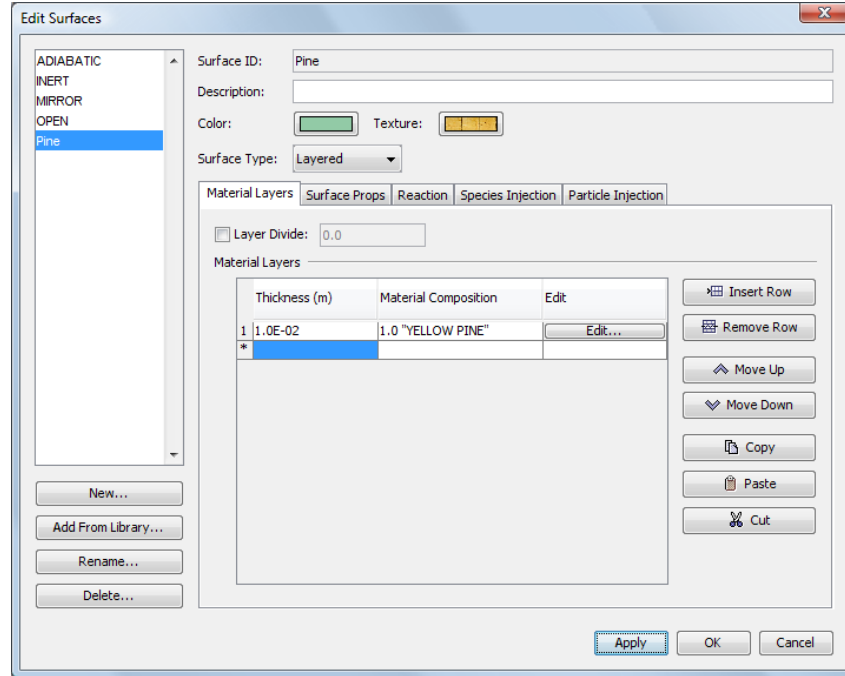
Figure 6.5. Creating the floor surface

We will use gypsum for the walls:

1. In the **Edit Surfaces** dialog, click **New...** Give the **Surface Name** as Gypsum, select the **Surface Type** as Layered, and click **OK**.

2. Click on the **Color** box and select a gray color (e.g. RGB of 0.7, 0.7, 0.7). Click **OK** to close the **Surface Color** dialog.

3. In the **Material Layers** panel, in the **Thickness** column, type 0.013.

4. Click the **Edit** button. In the **Mass Fraction** column, type 1.0. In the **Material** column, select GYPSUM. Click **OK** to close the **Composition** dialog.

5. In the **Edit Surfaces** dialog, click **Apply** to save the changes.

For the upholstery:

1. In the **Edit Surfaces** dialog, click **New**. Give the **Surface Name** as Upholstery, select the **Surface Type** as Layered, and click **OK**.

2. Click on the **Color** box and select a color (e.g. RGB of 0.4, 0.2, 0.0). Click **OK** to close the **Surface Color** dialog.

3. In the **Material Layers** panel, in the **Thickness** column, type 0.1.

4. Click the **Edit** button. In the **Mass Fraction** column, type 1.0. In the **Material** column, select FOAM. Click **OK** to close the **Composition** dialog.

5. Click on the **Surface Props** tab. In the **Backing** box, select **Insulated**.

6. Click on the **Reaction** tab. Select **Allow the Obstruction to Burn Away**. When this option is selected, the solid object disappears from the calculation cell by cell, as the mass contained by each mesh cells is consumed either by the pyrolysis reactions or by the prescribed HRR.

7. In the **Edit Surfaces** dialog, click **Apply** to save the changes.

We will place an initial *burner* surface on the sofa. The burner will release heat at a constant rate that will ignite the upholstery. To create this burner surface:

1. In the **Edit Surfaces** dialog, click **New**. Give the **Surface Name** as `Burner`, select the **Surface Type** as `Burner`, and click **OK**.

2. In the **Heat Release** panel, in the **Heat Release Rate (HRR)** box, type `1000`.

3. In the **Edit Surfaces** dialog, click **OK** to save the changes and close the dialog.

Create Furniture (Obstructions)

We will now create some furniture to place in the model.

Couch

The first will be a couch. Create a **Couch** group that will help us organize the input.

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

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Couch`.

4. Click **OK** to close the **Create Group** dialog.

To create the couch base:

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

2. In the **Description** box, type `Base`.

3. In the **Group** list, select `Couch`.

4. Click on the **Geometry** tab. In the **Box Properties** boxes, enter the values in Table 6.1, see Figure 6.6.

5. Click on the **Surfaces** tab. Select **Single** and select **Gypsum** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.1. Couch base dimensions

	X	**Y**	**Z**
Min	1.5	3.8	0.0
Max	3.1	4.6	0.4
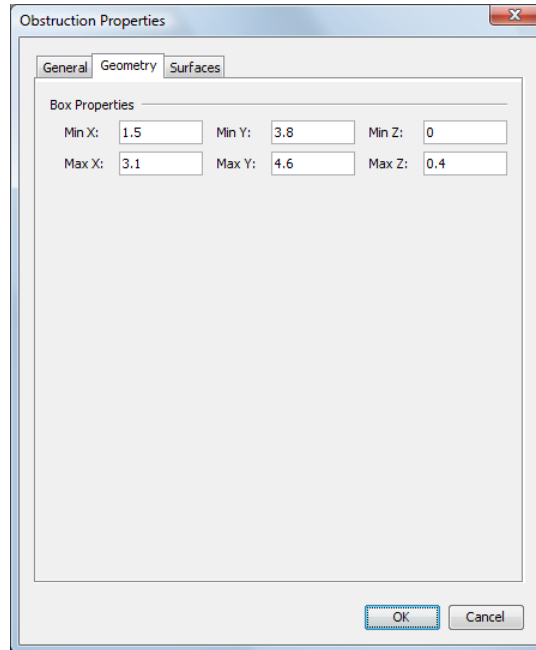
Figure 6.6. Input for the couch base

To create the seat:

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

2. In the **Description** box, type Seat cushion.

3. In the **Group** list, select Couch.

4. On the **Geometry** tab, enter the values in Table 6.2.

5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.2. Couch seat dimensions

	X	**Y**	**Z**
Min	1.5	3.8	0.4
Max	3.1	4.6	0.6

To create an armrest:

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

2. In the **Description** box, type Right armrest.

3. In the **Group** list, select Couch.

4. On the **Geometry** tab, enter the values in Table 6.3.

5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.3. Right armrest dimensions

	X	Y	Z
Min	1.3	3.8	0.0
Max	1.5	4.6	0.9

We will use the copy function to create the other armrest.

1. Right-click the **Right armrest** either in the Tree View or the 3D view.

2. Click **Copy/Move**.

3. In the **Mode** options, select **Copy** with 1 copy.

4. In the **Offset** boxes, enter X = 1.8, Y = 0.0, and Z = 0.0.

5. Click **OK** to close the **Translate** dialog.

By default, the name given to the armrest copy will be Right armrest[1], where the [1] indicates the first copy. To rename, double-click on the **Right armrest[1]** in the Tree view and change the **Description** to Left armrest. Click **OK**.

To create the back:

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

2. In the **Description** box, type Back cushion.

3. In the **Group** list, select Couch.

4. On the **Geometry** tab, enter the values in Table 6.4.

5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.4. Couch back dimensions

	X	Y	Z
Min	1.5	4.4	0.6
Max	3.1	4.6	1.2

The display will appear as shown in
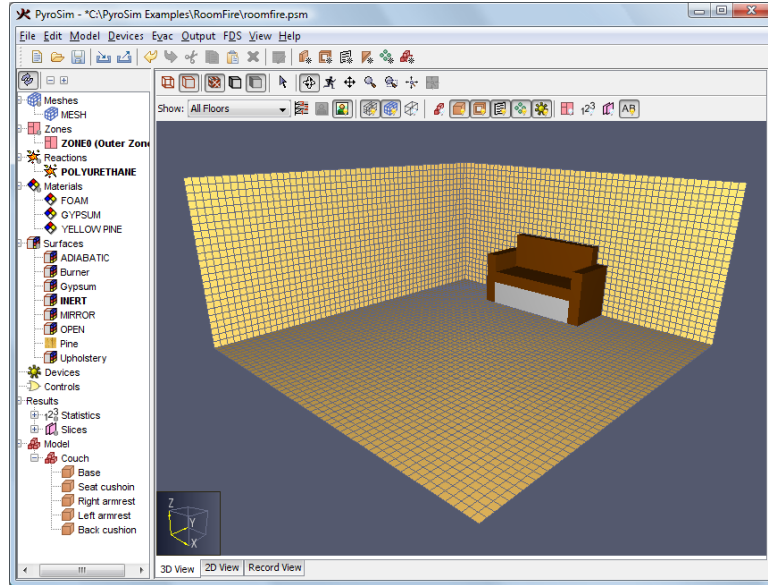
Figure 6.7. The room after the couch is added'

Second Couch

We will now create a second couch using the copy function.

1. In the Tree View, right-click the **Couch** group.

2. Click **Copy/Move**.

3. In the **Mode** options, select **Copy** with 1 copy.

4. In the **Offset** boxes, enter X = -1.3, Y = -3.6, and Z = 0.0.

5. Click **OK** to close the **Translate** dialog.

Rename Couch[1] to Couch 2.

Rotate the second couch to lie against the well.

1. In the Tree View, right-click **Couch 2** group.

2. Click **Rotate...**.

3. In the **Mode** options, select **Move**.

4. In the **Angle** box, type 90.

5. In the **Base Point** boxes, enter X = 0.0 and Y = 1.0.

6. Click **OK** to close the **Rotate Objects** dialog.

Additional Furniture

Add a pad:

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

2. In the **Description** box, type `Pad`.

3. In the **Group** list, select `Model`.

4. On the **Geometry** tab, enter the values in Table 6.5.

5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.5. Table dimensions

	X	**Y**	**Z**
Min	1.6	2.4	0.0
Max	3.0	3.2	0.2

The resulting room display is shown in Figure 6.8.
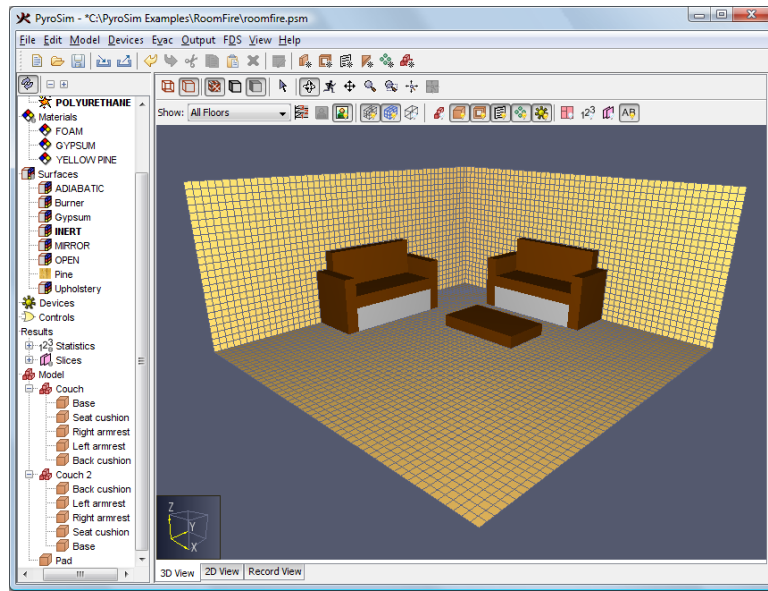
Figure 6.8. The resulting room display

Constucting complex objects can be time consuming. If your geometry is available in DXF format, PyroSim supports import. For walls, PyroSim provides sketching on a background image. Alternately, if you use the same geometry in many models, you can create the geometry and save it. You can then copy any object from one model to another. You can even copy just the text from an FDS input file and paste it into a PyroSim model.

Walls

We will add a wall using the 2D View. Since we will be adding only one wall, this wall could also be added quickly as a single obstruction. However, we will use the 2D view in order to demonstrate its use.

1. Select the **2D View**.

2. Select the **Wall** (▯) Tool.

3. Select the **Tool Properties** (▤) icon. Set the **Max Z** to 2.4 and change the **Surface Prop** to Gypsum. Click **OK**.

4. With the wall tool draw the wall from left to right along the Y=0 line. Hold down the Shift key to position the wall in the lower part of the model, Figure 6.9.
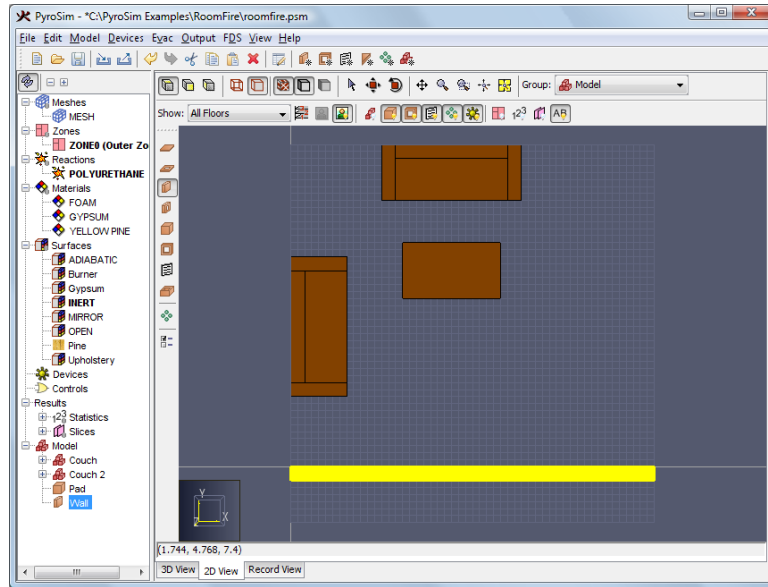
Figure 6.9. Drawing the wall

Create Door (Hole in Wall)

To add a door by creating a hole in the wall:

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

2. In the **Description** box, type Door.

3. In the **Group** list, select Model.

4. On the **Geometry** tab, enter the values in Table 6.6. Note that we extend the hole beyond the bounds of the wall it intersects. This ensures the hole will take priority over the wall.

5. Click **OK** to close the **Obstruction Properties** dialog.

Table 6.6. Door dimensions

	X	**Y**	**Z**
Min	4.0	−0.3	0.0
Max	4.9	0.1	2.0

The model now looks like:
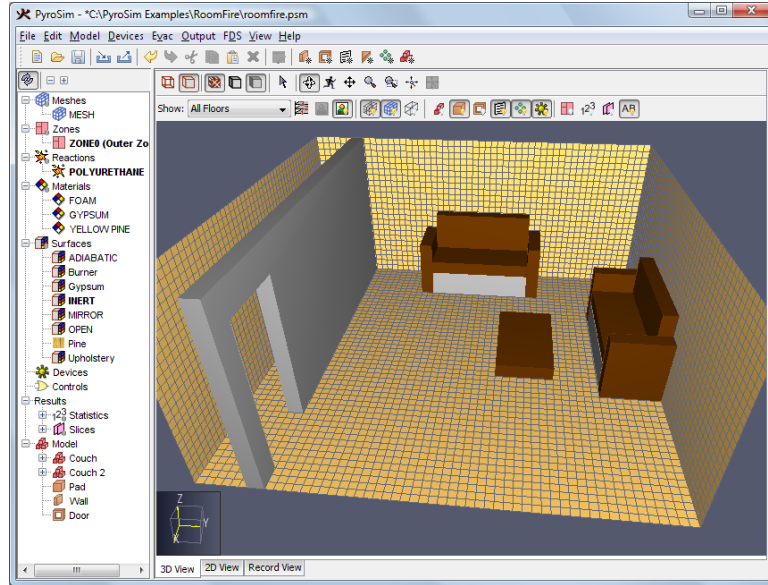
Figure 6.10. The model after adding the door

Use Vents to Define the Burner Fire and Floor

In FDS, *Vents* are used to describe 2D planar objects. In this example, we use vents to define the burner fire and the carpet on the floor.

Create the Fire

The fire is ignited by a burner that releases heat at a fixed rate for 120 seconds. The adjacent material eventually reaches ignition temperature and begins to burn. Here, we use a vent for the burner fire on the couch.

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

2. In the **Description** box, type `Burner`.

3. In the **Group** list, select `Model`.

4. In the **Surface** list, select `Burner`.

5. Click the **Geometry** tab. In the **Plane** list, select `Z` and give the value as `0.601` (The small value greater than 0.6 ensures the vent is displayed above the couch.).

6. In the **Bounds** boxes, enter the values in Table 6.7.

7. Click **OK** to close the **Vent Properties** dialog.

Table 6.7. Burner fire dimensions

	X	**Y**
Min	2.5	4.1
Max	2.7	4.4

Floor

The floor is also represented as a vent.

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

2. In the **Description** box, type `Floor`.

3. In the **Group** list, select `Model`.

4. In the **Surface** list, select `Pine`.

5. Click on the **Geometry** tab. In the **Plane** list, select `Z` and give the value as `0.001`.

6. In the **Bounds** boxes, enter the values in Table 6.8.

7. Click **OK** to close the **Vent Properties** dialog.

Table 6.8. Open boundary dimensions

	X	Y
Min	0.0	0.0
Max	5.2	4.6

Add an Open Boundary

We will add an open boundary on the model outside the door. PyroSim provides a shortcut that can create open vents on mesh boundaries.

1. In the navigation view, right-click on the **MESH** and click **Open Mesh Boundaries**. This will add a group named **Vents for MESH** that includes vents on each grid boundary.

2. Holding the CNTRL key, click on all Grid Boundary Vents **except** the `Vent Min Y for MESH`.

3. Right-click and delete the selected vents.

4. Right-click on the Model and select **Show All Objects**.

Hang a Picture to the Wall

Let us hang a picture on the wall. First decide what picture you want to hang.

1. On the **Model** menu, click **Edit Surfaces...**.

2. In the **Edit Surfaces** dialog, click **New**. Give the **Surface Name** as `Picture`, select the **Surface Type** as `Adiabatic`, and click **OK**.

3. Click on the **Texture** box.

4. Click the **Import...** button and select the image you want as a picture. I used the image call motorcycle.jpg that is included in the PyroSim installation in the samples folder (C:\Program Files\PyroSim 2009\samples).

5. The image you selected will be displayed. Under the image, click the **Details** tab. Set the **Width** to `1.0` and the **Height** to `0.675` (or whatever values are appropriate for your image.

6. Click **OK** to close the **Textures** dialog.

7. Click **OK** to close the **Edit Surfaces** dialog.

We now create a vent that uses the texture.

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

2. In the **Description** box, type `Picture`.

3. In the **Group** list, select `Model`.

4. In the **Surface** list, select `Picture`.

5. Under **Texture Origin**, click to select `Relative to Object`.

6. Click the **Geometry** tab. In the **Plane** list, select `Y` and give the value as `4.599`.

7. In the **Bounds** boxes, enter the values in Table 6.9.

8. Click **OK** to close the **Vent Properties** dialog.

Table 6.9. Picture dimensions

	X	Z
Min	3.6	1.2
Max	4.6	1.875

Create Thermocouple Records

We will first create one thermocouple near the ceiling and then replicate it to create a stack of thermocouples.

1. On the **Devices** menu, click **New Thermocouple...**.

2. In the **Location** boxes, enter `X = 2.6`, `Y = 2.3`, and `Z = 2.1`.

3. Click **OK** to close the **Thermocouple** dialog.

To copy the thermocouple:

1. Right-click on **THCP** and click **Copy/Move...**.

2. In the **Mode**, select **Copy** with the **Number of copies** 5.

3. In the **Offset** boxes, enter X = 0.0, Y =0.0, and Z = -0.3.

4. Click **OK** to close the **Translate** dialog.

Create Slice Records for 3D Results Plotting

Slice planes can be used to display 2D contours in the Smokeview display of the results. In this analysis, we will save temperature data for future plotting. To define the slice plane:

1. On the **Output** menu, click **Slices...**.

2. Fill the table by entering the values in Table 6.10. You can click on the row number to select entire rows to copy and paste, speeding the entry.

3. Click **OK** to close the **Animated Planar Slices** dialog.

Table 6.10. Slice plane data

XYZ Plane	Plane Value (m)	Gas Phase Quantity	Use Vector?
X	2.6	Temperature	NO

Create Boundary Records

Temperatures (and other quantities) on the surface of objects can be plotted by selecting *Surface Plots*. To define the surface plots

1. On the **Output** menu, click **Boundary Quantities...**.

2. Click the **Wall Temperature** checkbox.

3. Click **OK** to close the **Animated Boundary Quantities** dialog.

Specify Simulation Properties

To define the end time:

1. On the **FDS** menu, click **Simulation Parameters...**.

2. In the **Simulation Title** box, type Room fire.

3. In the **End Time** box, type 600 s.

4. Click **OK**.

The Model is Completed

Your model should now look like Figure 6.11. Save it.
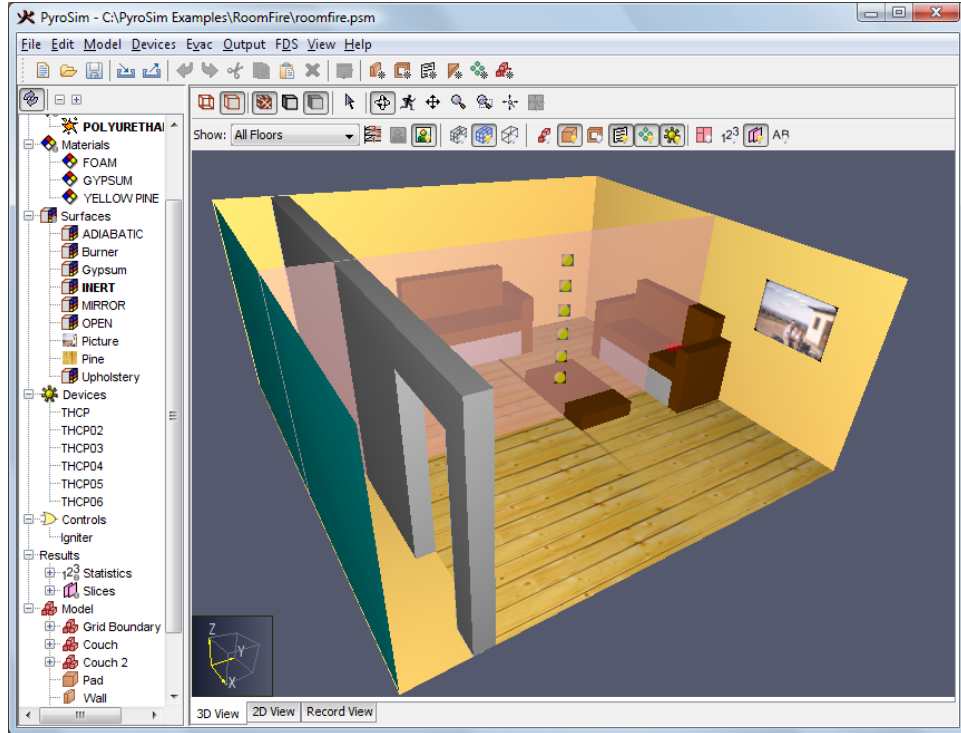
Figure 6.11. Completed model

Run the Analysis

To run the analysis:

1. On the **FDS** menu, click **Run FDS...**. The analysis will take about four hours to run on a 2.0 GHz computer.

View the Results

You can use the Smokeview software to make plots of the results. In Smokeview, on the **Shoe/Hide** menu click **Textures** and then select **Show All** to display all the textures, An image showing the heat release rate isosurface and temperature contours on the slice plane is shown in Figure 6.12. Notice that the couch is burning away.
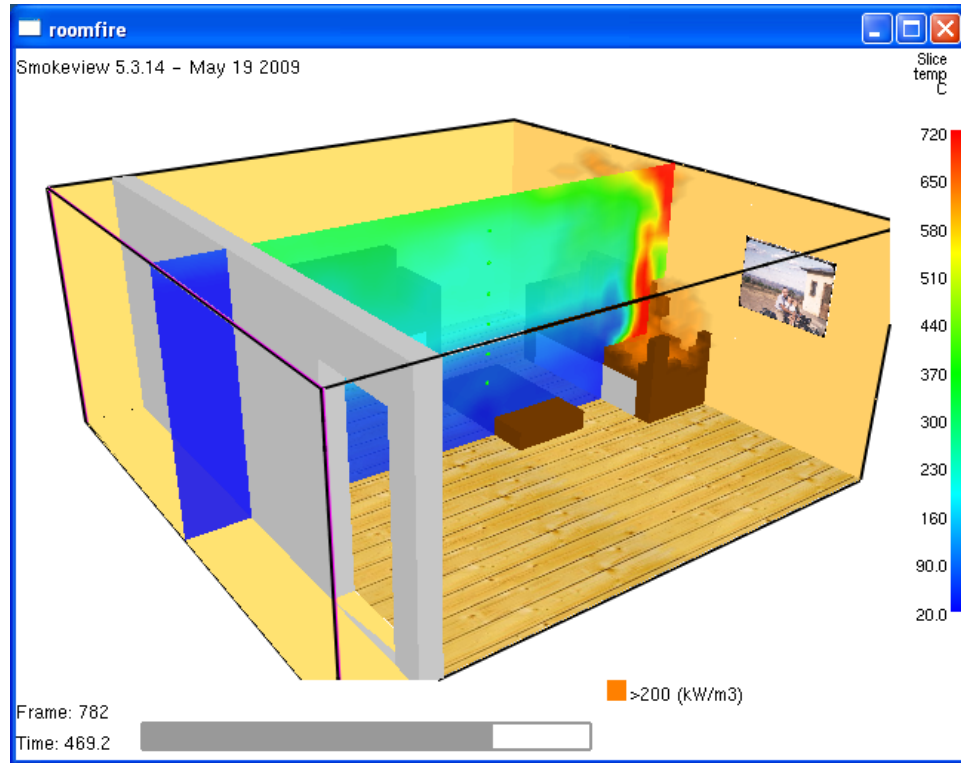
Figure 6.12. Heat release rate isosurface and temperature contours

To view time history results

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results...**.

2. A dialog will appear showing a list of 2D result files. Select **roomfire_hrr.csv** and click **Open** to view the heat release rate as a function of time, Figure 6.13.
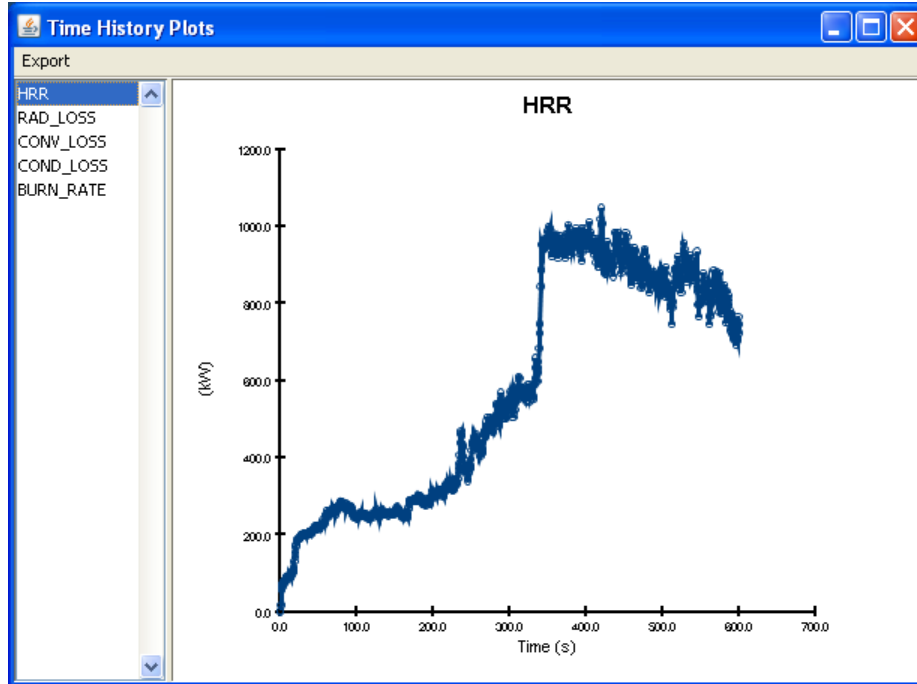
Figure 6.13. Heat release rate

Chapter 7. Switchgear Fire Example

This example evaluates fire conditions in two adjacent switchgear rooms connected by a double fire door, Figure 7.1. In the fire scenario, a fire starts in a switchgear cabinet in room 3A. The fire modeling results will used to estimate the time available for operators to conduct manual actions in one of the switchgear rooms. This example was provided by Bryan Klein [Klein, 2007].
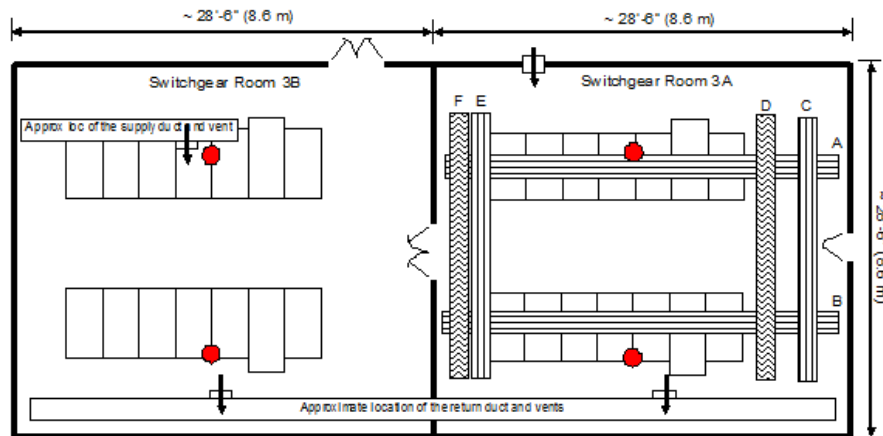
Figure 7.1. Pictorial representation of the switchgear room complex, showing switchgear cabinets, cable trays, supply ducts and vents, and smoke detectors. Drawing not to scale.

This tutorial demonstrates how to:

• Define materials.

• Create and replicate geometry.

• Open doors after a specified time.

• Create a burner fire.

• Add a smoke layer device.

• Add a slice plane for temperature visualization.

• View 3D results using Smokeview.

• View 2D results using PyroSim.

Figure 7.2. Completed model

Model parameters are given below.

Table 7.1. Room size (interior dimensions)

Dimension	English	Metric
Length	28'-6"	8.6 m
Width	28'-6"	8.6 m
Height	20'	6.0 m
Wall Thickness	2'	0.6096 m

Table 7.2. Door size

Dimension	English	Metric
Width	3'	0.9 m
Height	8'	2.4 m

Table 7.3. Concrete properties (NBSIR 88-3752)

Property	Value
Density	2280 kg/m^3
Specific Heat	1.04 kJ/kg-K
Conductivity	1.8 W/m-K

Table 7.4. Sheet metal properties (Drysdale, Intro to Fire Dynamics)

Property	Value
Density	7850 kg/m^3
Specific Heat	0.46 kJ/kg-K
Conductivity	45.8 W/m-K

Table 7.5. Cable properties (NUREG/CR-6850)

Property	Value
Density	1380 kg/m^3
Specific Heat	1.289 kJ/kg-K
Conductivity	0.192 W/m-K

Computational Mesh

In this example, we will use two meshes. We will use relatively coarse meshes that should be refined for a final analysis. In Room 3A (the room on the right) the cell size will be approximately 0.5 ft (0.1524 m) and in Room 3B, approximately 1.0 ft (0..3048 m). We have selected a finer resolution in Room 3A to more accurately represent the geometry of the cable trays and to provide a finer resolution for the flow solution near the fire. The two meshes much touch in order to transfer information between them. We will position the common plane inside Room 3B, so that the finer mesh includes all of Room 3A and the door between the rooms.

There is always a compromise between number of cells and acceptable solution time. As described, this model will have 162 000 cells and run in approximately 8 hours on a single CPU computer.

This problem uses English units as the primary values for the geometry. Switch to English units:

1. On the **View** menu, click **Units**.

2. Select **English**.

To create the first solution mesh for Room 3A:

1. On the **Model** menu, click **Edit Meshes...**.

2. Click **New** to create a mesh.

3. In the **Name** box, type Room 3A. Click **OK** to close the **New mesh** dialog.

4. In the **Order/Priority** list, select 1. This ensures that the finer mesh is the primary mesh for the solution.

5. In the **Mesh Boundary** boxes, enter the values in Table 7.6.

6. In the **X, Y, and Z** cell boxes, enter 60, enter 60, and enter 40 respectively, as shown in Figure 7.3. The FDS solution is optimized when the mesh cell division is defined by a number that can be formed using multiples of powers of 2, 3 and 5. These divisions give a cell size of approximately 0.5 ft (0.1524 m).

7. Click **Apply** to create the mesh.

Table 7.6. Dimensions for the mesh in Room 3A (including 2' thick walls)

	X (ft)	Y (ft)	Z (ft)
Min	27.5	0.0	0.0
Max	59.0	28.5	20.0
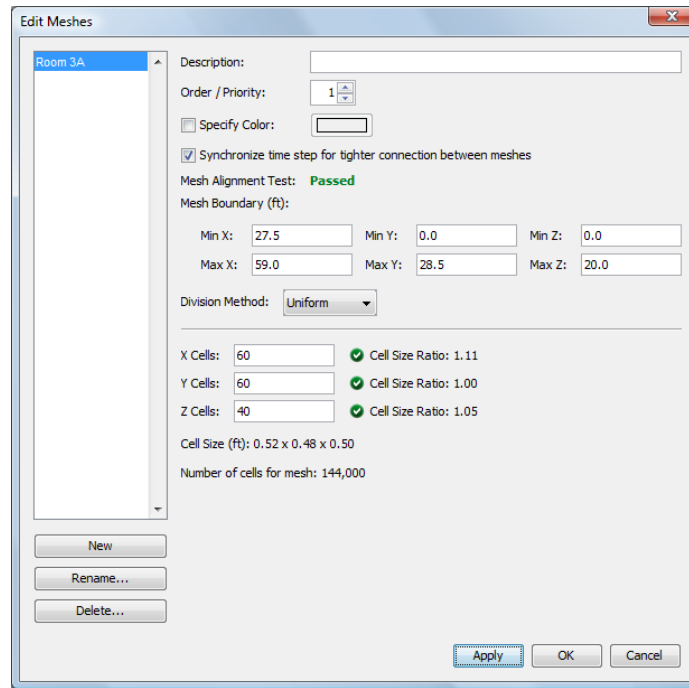
Figure 7.3. Input to create the mesh

To create the second solution mesh for Room 3B:

1. Click **New** to create a mesh.

2. In the **Name** box, type Room 3B. Click **OK** to close the **New mesh** dialog.

3. In the **Order/Priority** list, select 2.

4. In the **Mesh Boundary** boxes, enter the values in Table 7.7.

5. In the **X, Y, and Z** cell boxes, enter 30, enter 30, and enter 20 respectively. These divisions give a cell size of approximately 1.0 ft (0.3048 m).

6. Click **OK** to save the data close the **Edit Meshes** dialog.

Table 7.7. Dimensions for the mesh in Room 3B (including 2' thick walls)

	X (ft)	Y (ft)	Z (ft)
Min	0.0	0.0	0.0
Max	27.5	28.5	20.0

The meshes are shown in Figure 7.4. On the toolbar, click ⚓ to reset the image. Click ✛. You can rotate, pan, and zoom the model using the mouse and the **Shift** and **Alt** keys.
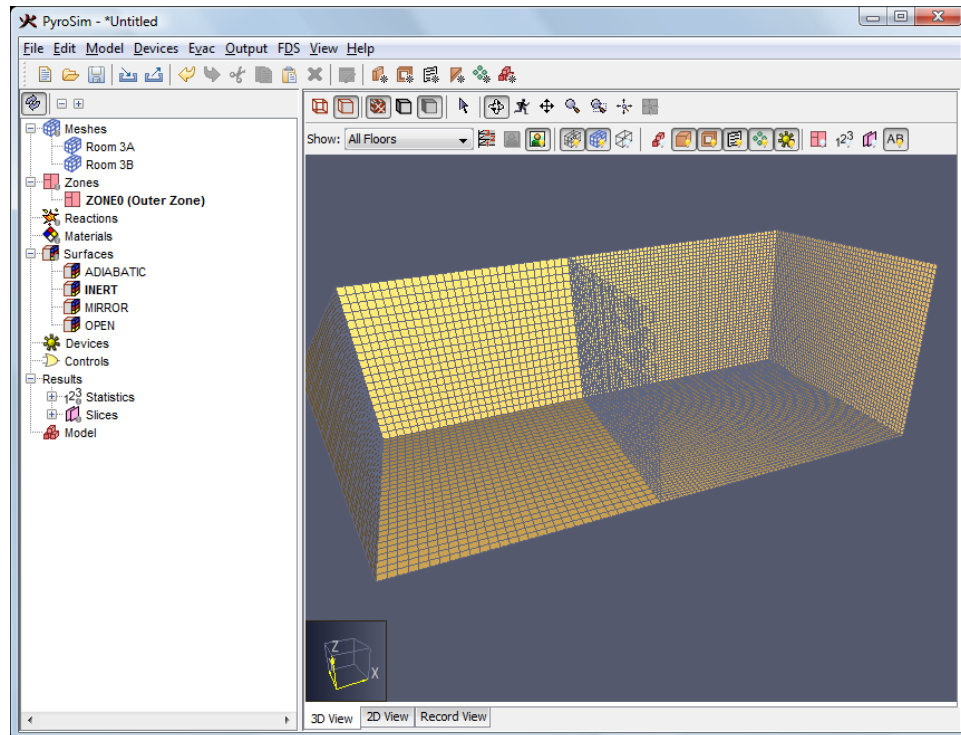
Figure 7.4. Display of the meshes

Material Properties

FDS uses materials to define physical properties. In this model, we will include the following material types: concrete, steel, and thermo-plastic cable. PyroSim includes a database file with material data and the references from which that data was obtained. We will import the concrete and steel material properties from this file.

1. On the **Model** menu, click **Edit Libraries...**.

2. In the **Category** box, select **Materials**.

3. Use the arrow to copy the **CONCRETE** and **STEEL** materials from the library into the **Current Model**, Figure 7.5.

4. **Close** the **PyroSim Libraries** dialog.
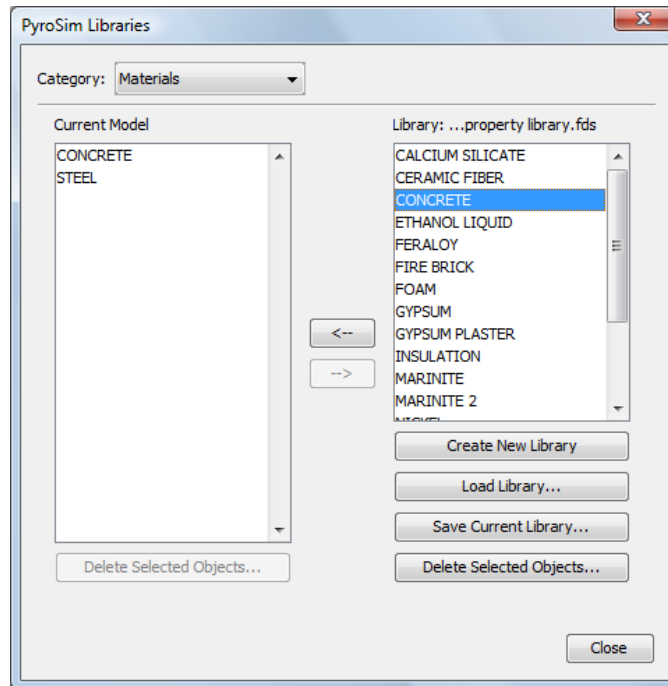
Figure 7.5. Copy the material data from the library to the model

We will enter the material properties for the cable manually. We note that the material properties in the problem description have been provided in metric units, so we will temporarily switch to metric units:

1. On the **View** menu, click **Units**.

2. Select **SI**.

The cables will be represented as a thermo-plastic material:

1. On the **Model** menu, click **Edit Materials...**.

2. In the **Edit Materials...**dialog, click **New...**.

3. In the **Material Name** box, type `Cable`, in the **Material Type** list, select `Solid`. Click **OK** to close the **New Material** dialog.

4. In the **Density** box, type `1380` kg/m^3, Figure 7.6

5. In the **Specific Heat** box, type `1.289` kJ/kg-K,

6. In the **Conductivity** box, type `0.192` W/m-K,

7. Click **OK** to save the changes and close the **Edit Materials** dialog.
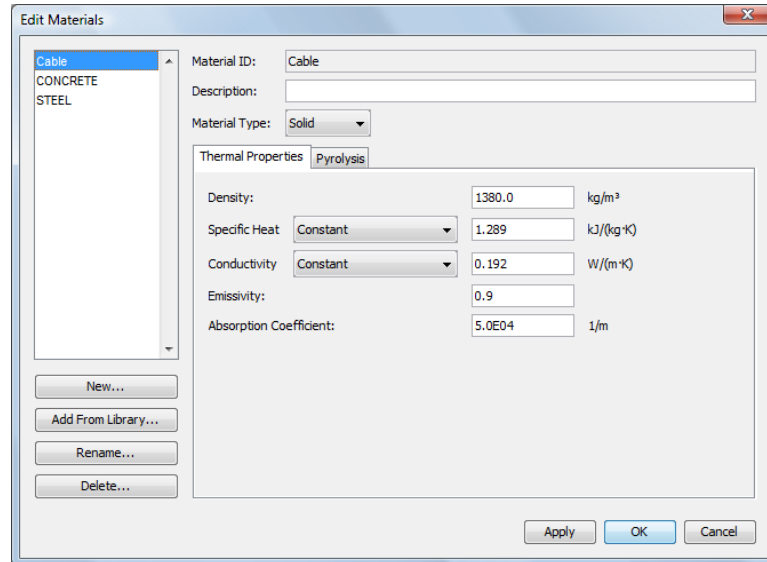
Figure 7.6. Thermo-plastic properties

Save the Model

This is a good time to 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 name the file `C:\Switchgear\switchgear.psm`.

3. Click **OK** to save the model.

Surface Properties

Surfaces are used to define the properties of objects in your model. Solid surfaces will use the material properties previously defined.

This problem uses English units as the primary values for the geometry. Switch to English units.

1. On the **View** menu, click **Units**.

2. Select **English**.

The concrete wall is 2 ft (0.6096 m) thick. This will be a **Layered** surface, although we will define only one layer. To create this surface

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New...**.

3. In the **Surface Name** box, type `Concrete Wall`, in the **Surface Type** list, select `Layered`. Click **OK** to create the surface and close the **New Surface** dialog.

4. Click the **Color** button to open the **Surface Color** dialog, then select a dark gray and click **OK**.

5. In the **Material Layers** panel, in the **Thickness** column, type `2.0` ft.

6. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction** column, type `1.0`. In the **Material** column, select `Concrete`, Figure 7.7. Click **OK** to close the **Composition** dialog.

7. In the **Edit Surfaces** dialog, click **Apply** to save the changes, Figure 7.8. We have just stated that this surface is 2 feet thick and composed entirely of concrete.
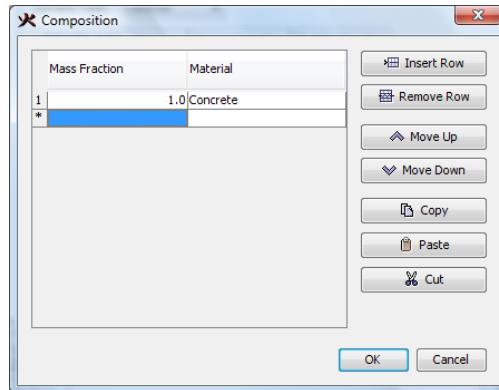
Figure 7.7. Composition of the concrete surface
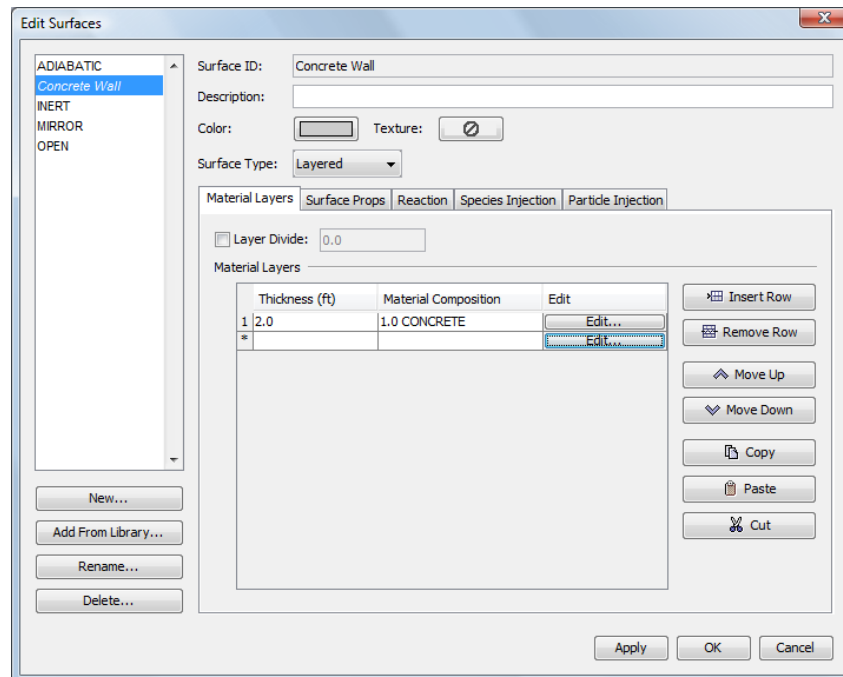
Figure 7.8. The concrete surface

The sheet metal is 0.05 in (0.0042 ft, 0.0013 m) thick. To create this surface

1. In the **Edit Surfaces** dialog, click **New...**.

2. In the **Surface Name** box, type `Sheet Metal`, in the **Surface Type** list, select `Layered`. Click **OK** to close the **New Surface** dialog.

3. Click the **Color** button to open the **Surface Color** dialog, then select blue and click **OK**.

4. In the **Material Layers** panel, in the **Thickness** column, type `0.0042` ft.

5. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction** column, type `1.0`. In the **Material** column, select `Steel`. Click **OK** to close the **Composition** dialog.

6. In the **Edit Surfaces** dialog, click **Apply** to save the changes.

The cable is 1.5 in (0.125 ft, 0.0381 m) thick. To create this surface

1. In the **Edit Surfaces** dialog, click **New...**.

2. In the **Surface Name** box, type `Cable`, in the **Surface Type** list, select `Layered`. Click **OK** to close the **New Surface** dialog.

3. Click the **Color** button to open the **Surface Color** dialog, then select a black and click **OK**.

4. In the **Material Layers** panel, in the **Thickness** column, type `0.125` ft.

5. The material composition can be defined as a mixture. Click the **Edit** button. In the **Mass Fraction** column, type `1.0`. In the **Material** column, select `Cable`. Click **OK** to close the **Composition** dialog.

6. Click on the **Surface Props** tab. In the **Backing** list, select `Insulated`. This will approximate a solid being heated from all sides.

7. Click **OK** to close the **Edit Surfaces** dialog.

Switch to **SI** units to input additional properties given in SI units.

1. On the **View** menu, click **Units**.

2. Select **SI**.

Edit the heat release rate and ignition temperature for the **Cable** surface. These values should only be considered as examples.

1. Right-click on **Cable** in the **Surfaces** list. Click **Properties**.

2. Click the **Reaction** tab.

3. Click to select **Governed Manually**.

4. In the **Heat Release Rate** box, type `265` kW/m^2. In the **Ignition** panel select **Ignite at** and type `250` C.

5. Click **OK** to save the changes and close the **Edit Surfaces** dialog.

Model Geometry

We will now create the geometry using a few dialogs and by drawing a few walls. If you were working on a more complex building, you could display the floor plan in the 2D View and then sketch walls directly on the background image or import the data from a DXF file.

The model will only include the separating wall between the two rooms. There is no need to define the perimeter walls, since the perimeter will be assigned the surface properties of the concrete wall.

During the creation of a PyroSim model, it is recommended that you group the various parts of the model for purposes of editing and later viewing. For example, all walls will be placed in a sub-group called **Walls**. This allows us to quickly select this group to modify it or to hide it for viewing of other interior details.

The problem dimensions use English units as the primary values. To switch to English units

1. On the **View** menu, click **Units**.

2. Select **English**.

Walls and Doors

To make a new group:

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

2. In the **Parent Group** list, select Model.

3. In the **Group Name** box, type Walls.

4. Click **OK** to close the **Create Group** dialog.

To define the dividing wall

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

2. In the **Description** box, type Dividing Wall.

3. In the **Group** list, select Walls.

4. Click on the **Geometry** tab. In the **Bounding Box** boxes, enter the values listed in Table 7.8 and shown in Figure 7.9.

5. Click on the **Surfaces** tab, select **Single** and select **Concrete Wall** from the list.

6. Click **OK** to close the **Obstruction Properties** dialog.

7. In the toolbar, click to turn off the **Show Mesh Divisions** option. The display should now look like Figure 7.10.

Table 7.8. Dividing wall dimensions

	X (ft)	Y (ft)	Z (ft)
Min	28.5	0.0	0.0
Max	30.5	28.5	20.0
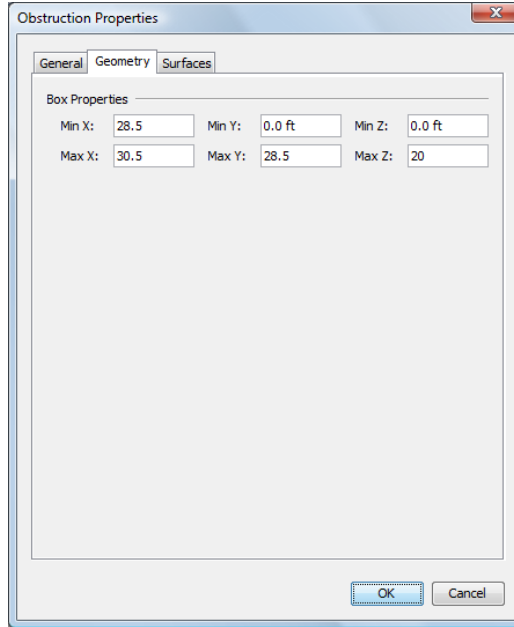
Figure 7.9. Input for the wall dividing the two rooms
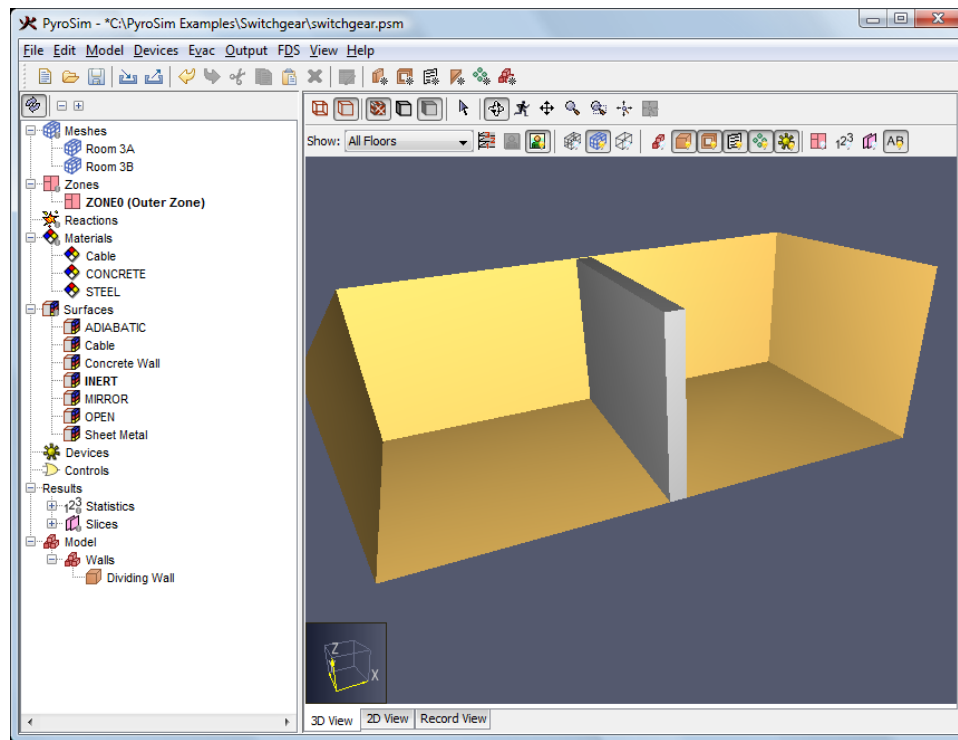
Figure 7.10. Display of the dividing wall

We will use a **Hole** obstruction to represent the door between the rooms. A hole can defined in the model, but not activate until a specified time or event. In this case, we will define a hole (door) between the rooms that will open 5 minutes (300 seconds) after the start of the fire. To add the dividing door to the model:

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

2. In the **Description** box, type `Dividing Door`.

3. In the **Group** list, select `Walls`.

4. Click on the **Geometry** tab. In the **Bounding Box** boxes, enter the values in Table 7.9. The values are chosen thicker than the wall to ensure that the door takes precedence. In the toolbar, click to turn off the **Show Holes** option. Now we just see the door through the wall.

5. Return to the **General** tab. *Devices* are used to link a response (i.e. open a door) to an event in the model (i.e. a temperature measurement reaching a set value). In this model, we will open the door after 300 seconds.

6. Click the **Activation** button. In the list, select **New...** to create a new device. For the **Name** type `Open Door`. Click **OK**.

7. For the **Input Type**, select **Time**. For the **Action to Perform**, select **Create/Activate**. In the text window, click on **TBEGIN** to change the time. In the pop-up box, type `300.0 s` and press ENTER on your keyboard. The control logic will now be displayed, Figure 7.11. Click **OK** to create the control.

8. Click **OK** to close the **Vent Properties** dialog. You will notice that an **Open Door** control has been added to **Controls** in the Navigation View.

Table 7.9. Dividing door dimensions

	X (ft)	**Y (ft)**	**Z (ft)**
Min	28.25	11.25	0.0
Max	30.75	17.25	8.0
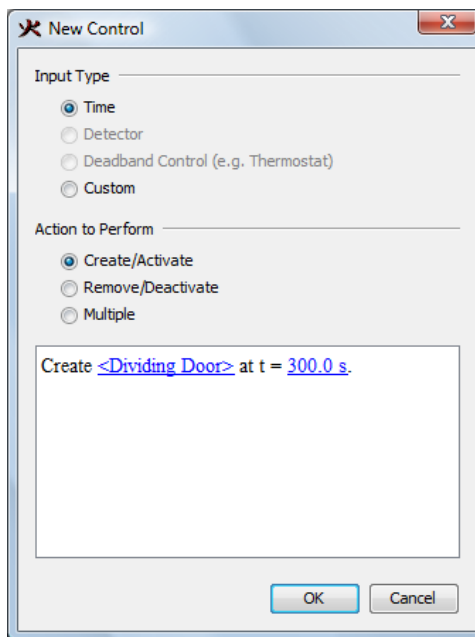
Figure 7.11. The control logic that opens the dividing door

We also will add the Room 3B entrance door. We want this door to stay closed for 300 sec. By default, FDS assumes the boundary of a mesh is closed. To define a door on the boundary, we will first create an open vent on the mesh boundary, then plug it with an obstruction that will be removed at 300 sec. To add the entrance door to the model:

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

2. In the **Description** box, type `Entrance Door`.

3. In the **Group** list, select `Walls`.

4. In the **Surface** list, select `OPEN`.

5. Click on the **Geometry** tab. In the **Lies in the Plane** list, select **Y** and type `28.5`.

6. In the **Min X** box, type `20.0` and in the **Max X** box, type `26.0`.

7. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `8.0`.

8. Click **OK** to save the changes and close the **Vent Properties** dialog.

To define the blockage in the entrance door:

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

2. In the **Description** box, type `Entrance Door Blockage`.

3. In the **Group** list, select `Walls`.

4. Click on the **Geometry** tab. In the **Bounding Box** boxes, enter the values in Table 7.10. The values are chosen thicker than the wall to ensure that the door takes precedence.

5. Click on **Surfaces** tab. Select **Single** and select **Concrete Wall** from the list.

6. Click the **Activation** button. In the list, select **New...** to create a new device. For the **Name** type `Entrance Blockage`. Click **OK**.

7. For the **Input Type**, select **Time**. For the **Action to Perform**, select **Remove/Deactivate**. In the text window, click on **TBEGIN** to change the time. In the pop-up box, type `300.0 s` and press ENTER on your keyboard. Click **OK** to create the control.

8. Click **OK** to close the **Obstruction Properties** dialog. You will notice that an **Entrance Blockage** control has been added to **Controls** in the Navigation View. This control will re-move the obstruction (open the door) at 300.0 seconds.

Table 7.10. Entrance door blockage dimensions

	X (ft)	Y (ft)	Z (ft)
Min	19.5	28.0	0.0
Max	26.5	29.0	8.0

Switchgear Cabinets

First, make a switchgear group:

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

2. In the **Parent Group** list, select Model.

3. In the **Group Name** box, type Switchgear.

4. Click **OK** to close the **Create Group** dialog.

We could define the cabinets using the **New Obstruction...** menu and dialog as described above. Instead we will sketch the cabinets. To prepare to draw the switchgear cabinets:

1. Click **2D View**.

2. On the **View** menu, click Snap to Sketch Grid. The **Sketch Grid** allows you to specify a grid spacing that is useful for creating geometry independent of the solution meshes.

3. On the **View** menu, click **Set Sketch Grid Spacing...**. In the **Snap Spacing** box, type 0.5. Click **OK** to close the **Snap-to Grid** dialog.

4. Use the **Pan View** tool to drag the model so that the dividing wall is to the right of the window.

To draw the cabinet:

1. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

2. To set the tool properties, click **Tool Properties** ( ).

3. In the **Min Z** box, type 0.0 and in the **Max Z** box, type 8.0. In the **Surface Prop** list, select **Sheet Metal**. Click **OK** to close the **New Obstruction Properties** dialog.

4. In the **2D View** action toolbar (top of window), select **Switchgear** in the **Group** list.

5. Press and drag the mouse, approximating the dimensions of the lower left cabinet shown in Figure 7.1.

6. Your sketch will approximate that shown in Figure 7.12.
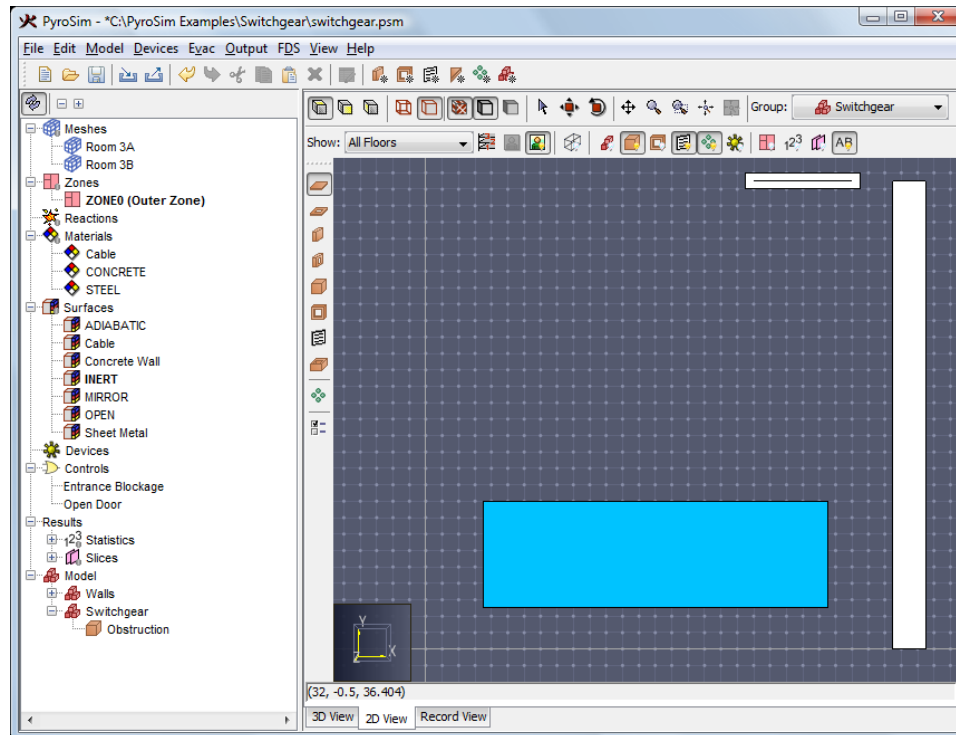
Figure 7.12. Sketch of the lower left cabinet

By default, the cabinet sketch snapped to the sketch grid points. To precisely define the dimensions of the cabinet

1. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool ( ).

2. Double-click on the cabinet to edit its properties.

3. In the **Description** box, type Cabinet 1.

4. Click the **Geometry**tab. In the **Bounding Box** boxes, enter the values in Table 7.11.

5. Click **OK** to close the **Obstruction Properties** dialog.

Table 7.11. Cabinet Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	4.0	4.0	0.0
Max	20.0	10.0	8.0

We will now copy the first cabinet using the **Translate Objects** tool in the **2D View**.

1. Using the **Select and Manipulate Objects** tool ( ), click on **Cabinet 1**.

2. Click the **Translate Objects** tool ( ).

3. Press the **CTRL** key (to create a copy) and drag upward to create a second cabinet, Figure 7.13.

4. Click the **Select and Manipulate Objects** tool ( ).

5. Double click on the new cabinet and change the name to **Cabinet 2**. On the **Geometry**tab, set the dimensions for Cabinet 2 given in Table 7.12.
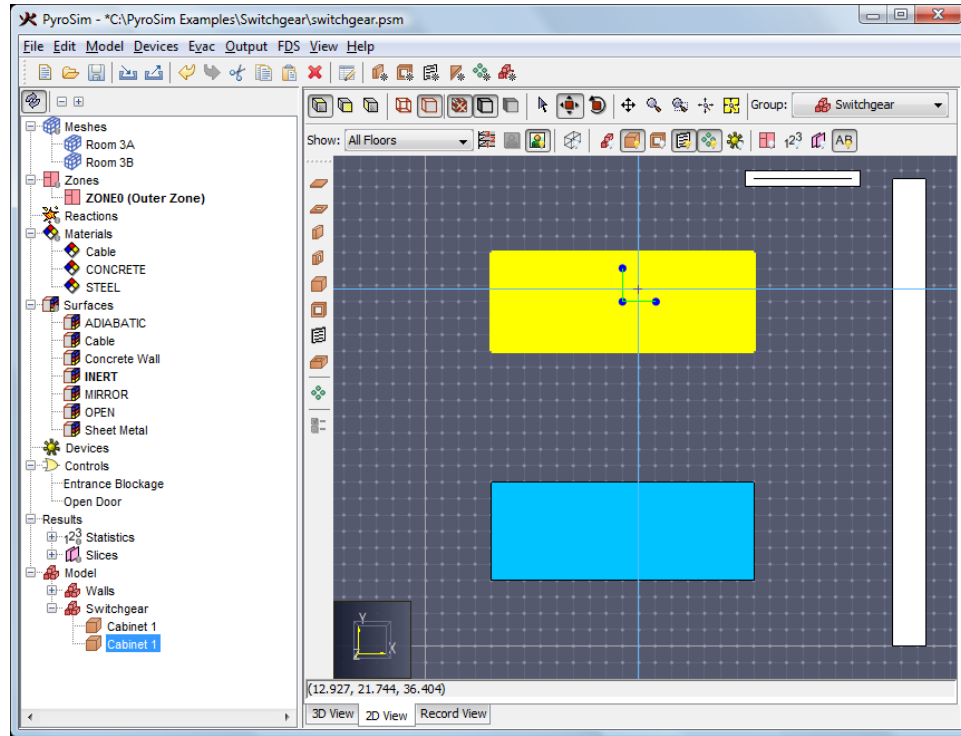
Figure 7.13. Making a copy of Cabinet 1 by dragging. The final position will be 4 feet from the left and top boundaries.

Table 7.12. Dimensions for Cabinet 2

	X (ft)	Y (ft)	Z (ft)
Min	4.0	18.5	0.0
Max	20.0	24.5	8.0

We now copy both cabinets using the **Copy/Move...** menu. You could alternately use the **Translate Objects** tool as before.

1. Using the **Select and Manipulate Objects** tool ( ), press **CTRL** and click on both **Cabinet 1** and **Cabinet 2**.

2. Right-click on either cabinet and click **Copy/Move...**.

3. Click **Copy** and type 1 in the **Number of Copies** box.

4. In the **Offset** box for **X**, type 30.5 .

5. Click **OK**.

6. Change the cabinet names to **Cabinet 3** and **Cabinet 4**.

The cabinets are shown in Figure 7.14.
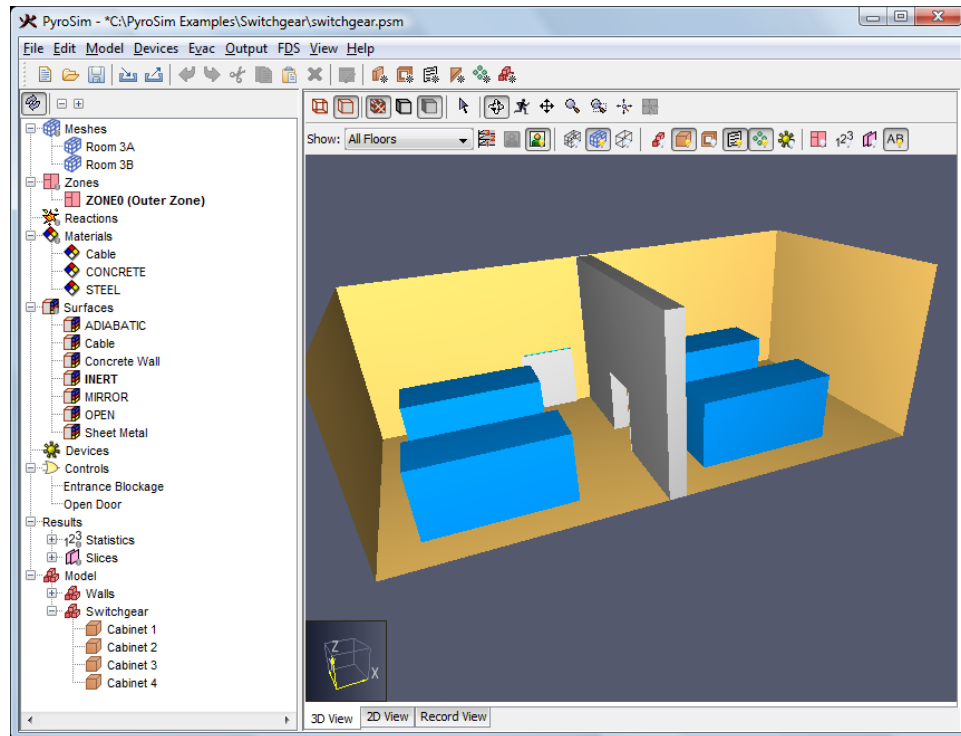
Figure 7.14. The rooms showing the switchgear cabinets

Cable Trays

The cables are supported by ladder trays. These trays are relatively open to air flow and have relatively little thermal mass compared to the cables. Therefore, we will not model the details of the trays, but will just model the cables. First, make a cables group:

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

2. In the **Parent Group** list, select Model.

3. In the **Group Name** box, type Cables.

4. Click **OK** to close the **Create Group** dialog.

To prepare to draw the cables:

1. Click **2D View**. This will display the sketch grid.

2. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

3. To set the tool properties, click **Tool Properties** (![icon]). In the **Min Z** box, type 8.5. In the **Max Z** box, type 9.0. In the **Surface Prop** list, select **Cable**. Click **OK** to close the **New Obstruction Properties** dialog.

4. In the **2D View** action toolbar (top of window), select **Cables** in the **Group** list.

To draw cable A:

1. Press and drag the mouse, approximating the dimensions of cable A shown in Figure 7.1. After sketching, the cable will be as shown in Figure 7.15.

2. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool.

3. Double-click on the cable to edit its properties. In the **Description** box, type Cable A. On the **Geometry** tab, enter the values in Table 7.13 in the **Bounding Box** boxes.

4. Click **OK** to close the **Obstruction Properties** dialog.
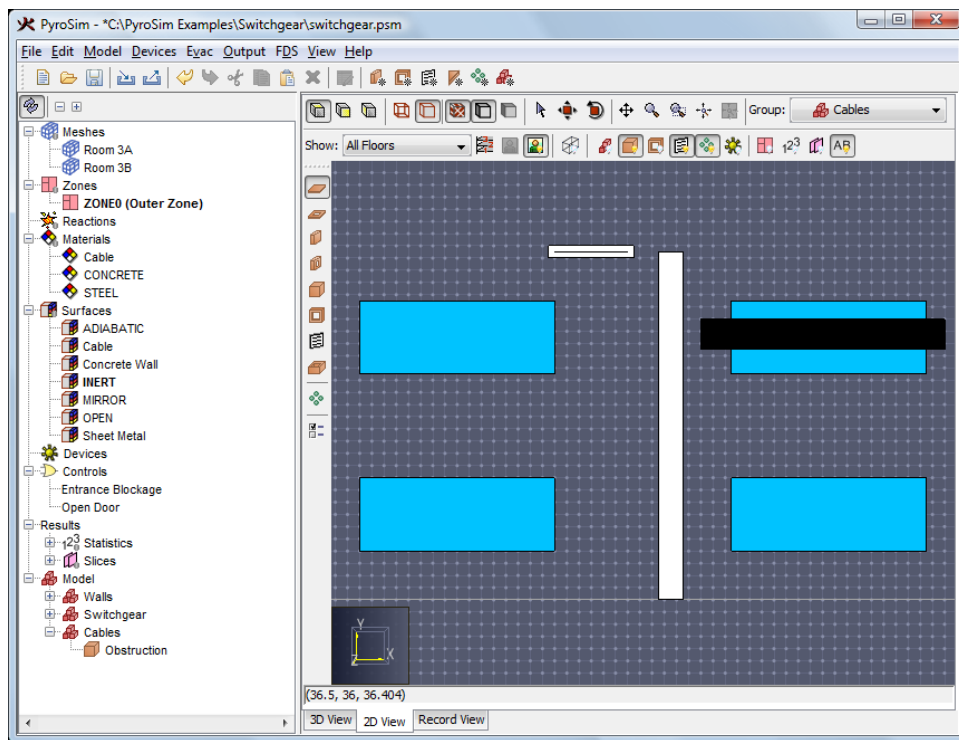
Figure 7.15. The sketch of Cable A

Table 7.13. Cable A Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	31.0	20.0	8.5
Max	58.0	23.0	9.0

Copy to create cable B:

1. Right-click on **Cable A** and click **Copy/Move...**.

2. Click **Copy** and type `1` in the **Number of Copies** box. In the **Offset** box for **Y**, type `-14.5`. Click **OK** to create the copy.

3. Double click on the new cable (Cable A[1]) and change the name to **Cable B**.

4. Click **OK** to close the **Obstruction Properties** dialog.

Drawing cables C, D is similar. First:

1. Click **2D View**.

2. In the **2D View** drawing tools toolbar (left of window), click the **Draw an Obstruction** tool ( ).

3. To set the tool properties, click **Tool Properties** ( ). In the **Min Z** box, type `9.5`. In the **Max Z** box, type `10.0`. In the **Surface Prop** list, select **Cable**. Click **OK** to close the **New Obstruction Properties** dialog.

4. In the **2D View** action toolbar (top of window), select **Cables** in the **Group** list.

To draw cable C:

1. Press and drag the mouse, approximating the dimensions of cable C shown in Figure 7.1.

2. In the **2D View** action toolbar (top of window), click the **Select and Manipulate Objects** tool.

3. Double-click on the new cable to edit its properties. In the **Description** box, type `Cable C`. On the **Geometry** tab, in the **Bounding Box** boxes, enter the values in Table 7.14.

4. Click **OK** to close the **Obstruction Properties** dialog.

Table 7.14. Cable C Dimensions

	X (ft)	Y (ft)	Z (ft)
Min	55.0	3.0	9.5
Max	58.0	25.5	10.0

Copy to create cable D:

1. Right-click on **Cable C** and click **Copy/Move...**.

2. Click **Copy** and type `1` in the **Number of Copies** box.

3. In the **Offset** box for **X**, type `-4.0`.

4. Click **OK**. This will create **Cable C[1]**. Double click and change the name to **Cable D**.

5. Click **OK** to close the **Obstruction Properties** dialog.

Copy to create cable E:

1. Right-click on **Cable D** and click **Copy/Move...**.

2. Click **Copy** and type `1` in the **Number of Copies** box.

3. In the **Offset** box for **X**, type `-20.0`.

4. Click **OK**. This will create **Cable D[1]**. Double click and change the name to **Cable E**.

5. Click **OK** to close the **Obstruction Properties** dialog.

If wanted, you could add additional cables. We will stop here. The cables are shown in Figure 7.16.
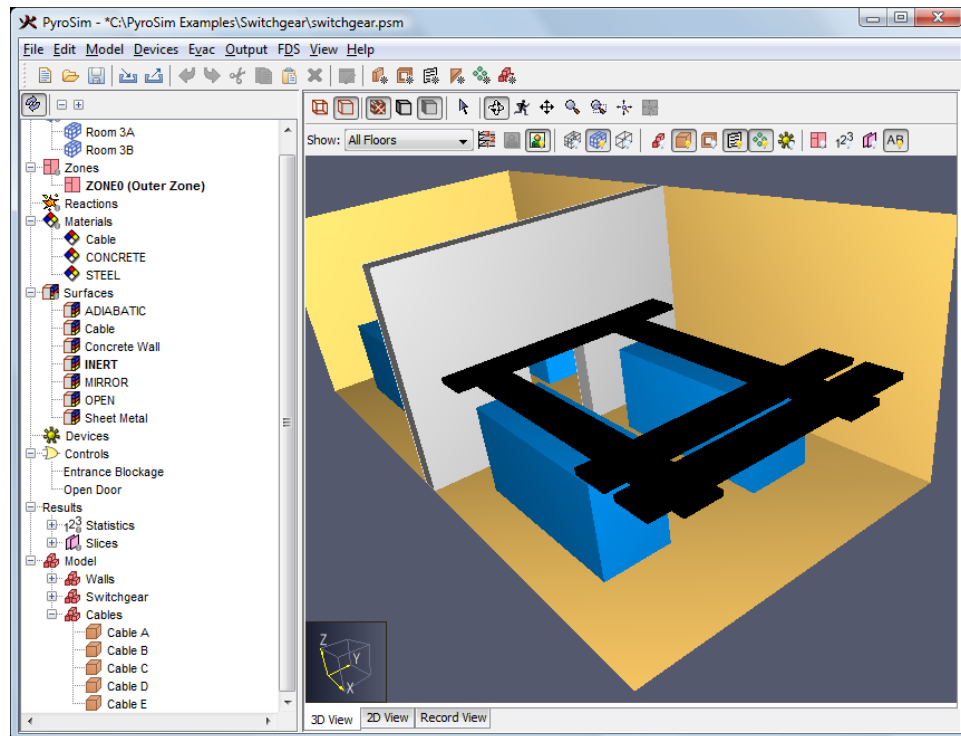
Figure 7.16. The room showing the cables

Ventilation

Each room has supply and return ventilation. Vents have general usage in FDS to describe 2D planar objects. In this case, we will use vents to model components of the ventilation system. No holes need to be created; it is assumed that air is supplied or returned into duct work within the wall.

We will first create a vent group:

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

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Vents`.

4. Click **OK** to close the **Create Group** dialog.

There are three steps to defining a vent. First, a surface will be created that defines the flow boundary conditions. Next an obstruction is created. Finally, the vent (using the vent surface), is attached to the obstruction. We will assume all vents are 24"x24" (2'x2'). For the supply vents, we will define a flow rate. For the return vents, we will just leave the vents open to the atmosphere. This ensures that the air pressure in the room is not affected by any difference between supply and return flow rates.

To create the surface that represents air supply:

1. On the **Model** menu, click **Edit Surfaces...**.

2. In the **Edit Surfaces** dialog, click **New...**.

3. In the **Surface Name** box, type `Supply Vent`. Select **Surface Type** and in the list, select `Supply`.

4. Click **OK** to close the **New Surface** dialog.

To change the properties of the **Supply Vent**:

1. In the **Edit Surfaces** dialog, click to select **Supply Vent**.

2. Click the **Color** button to open the **Surface Color** dialog, then select a green and click **OK**.

3. On the **Air Flow** tab, click **Specify Velocity** and type 8.33 ft/s, Figure 7.17. This corresponds to 2000 CFM flow through a 2'x2' duct.

4. Click the **Particle Injection** tab. Click the **Emit Particles** checkbox.

5. Click **OK** to save the changes and close the **Edit Surfaces** dialog.
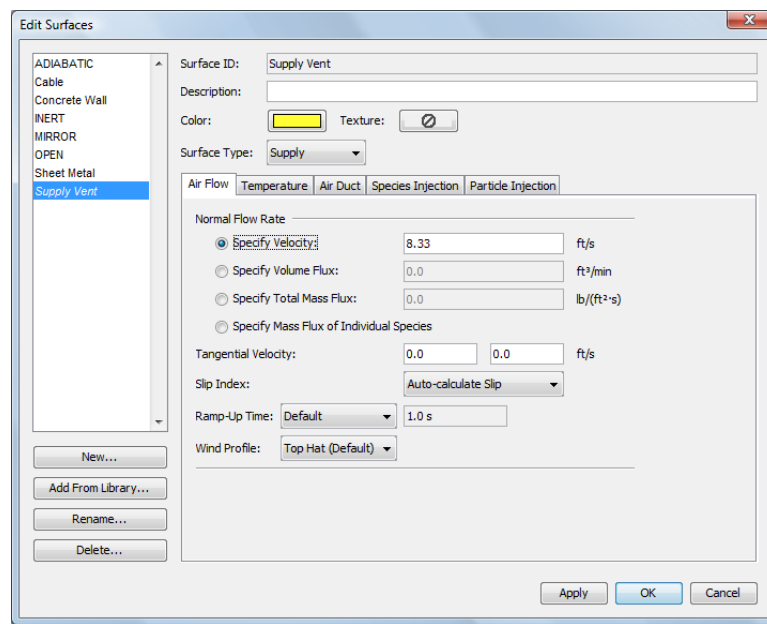
Figure 7.17. Creating the supply vent surface

In this model, we will not explicitly represent the return duct. Instead, we will position the return vents on the outer boundary. This has the advantage of simplicity and allows us to define these vents as open to the atmosphere. To add the return vent to the model:

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

2. In the **Description** box, type `Return Vent 3B`.

3. In the **Group** list, select `Vents`.

4. In the **Surface** list, select `OPEN`.

5. Click the **Geometry**tab. In the **Lies in the plane** list, select `Y` and type `0.0`. In the **Bounds** boxes, enter the values in Table 7.15.

6. Click **OK** to close the **Vent Properties** dialog.

Table 7.15. Return vent dimensions

	X (ft)	Z (ft)
Min	13.0	11.0
Max	15.0	13.0

Copy to create the second return vent:

1. In the **Navigation View** (tree on left), right-click on **Return Vent 3B** and click **Copy/Move...**.

2. Click **Copy** and type `1` in the **Number of Copies** box.

3. In the **Offset** box for **X**, type `30.5`.

4. Click **OK**. This will create **Return Vent 3B[1]**.

5. Double click and change the name to **Return Vent 3A**.

6. Click **OK** to close the **Vent Properties** dialog.

To create the supply duct for Room 3B:

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

2. In the **Description** box, type `Supply Duct`.

3. In the **Group** list, select `Vents`.

4. Click **Specify Color**. The default gray color does not need to be changed.

5. Click the **Geometry**tab. In the **Bounding Box** boxes, enter the values in Table 7.16.

6. Click **OK** to close the **Obstruction Properties** dialog.

Table 7.16. Supply duct dimensions

	X (ft)	Y (ft)	Z (ft)
Min	1.0	23.5	11.0
Max	18.0	25.5	13.0

Add the supply vent to the supply duct in Room 3B:

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

2. In the **Description** box, type `Supply Vent 3B`.

3. In the **Group** list, select `Vents`.

4. In the **Surface** list, select `Supply Vent`.

5. Click the **Geometry**tab. In the **Lies in the plane** list, select `Y` and type `23.49`. This places it just barely outside the duct for visualization, but it will be returned to the exact mesh coordinates during solution. In the **Bounds** boxes, enter the values in Table 7.17.

6. Click **OK** to close the **Vent Properties** dialog.

Table 7.17. Supply vent dimensions

	X (ft)	Z (ft)
Min	11.0	11.0
Max	13.0	13.0

Add the supply vent to the wall in Room 3A:

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

2. In the **Description** box, type `Supply Vent 3A`.

3. In the **Group** list, select `Vents`.

4. In the **Surface** list, select `Supply Vent`.

5. Click the **Geometry**tab. In the **Lies in the plane** list, select `Y` and type `28.5`. In the **Bounds** boxes, enter the values in Table 7.18.

6. Click **OK** to close the **Vent Properties** dialog.

Table 7.18. Return vent dimensions

	X (ft)	Z (ft)
Min	34.0	11.0
Max	36.0	13.0

An image showing the ducts and vents is given in Figure 7.18. The duct color has been changed to cyan.
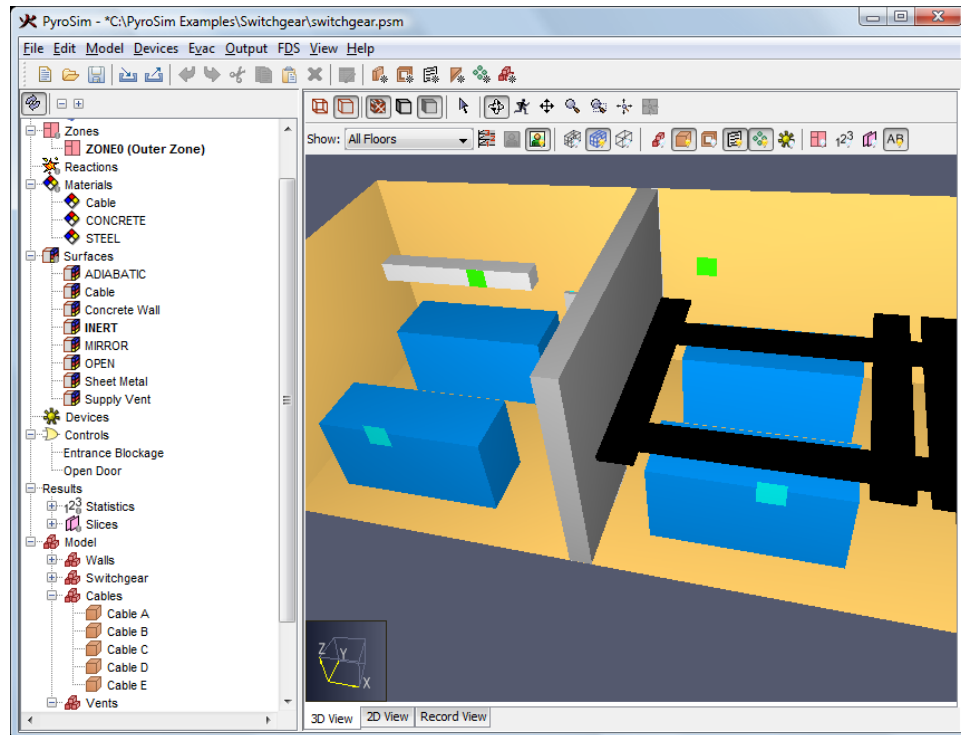
Figure 7.18. The room showing the vents

Fire

The fire is postulated to start in Room 3A in a switchgear cabinet. Heat is released to the room through a vent in the side of the cabinet. The size of the vent is 3.0'x1.0' and located 0.5 ft below the top of the cabinet. We will model the fire as a surface with a specified heat release rate. A surface type is used to specify the fire.

We will first create a fire group:

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

2. In the **Parent Group** list, select `Model`.

3. In the **Group Name** box, type `Fire`.

4. Click **OK** to close the **Create Group** dialog.

There are two additional steps to defining a fire. First, a surface will be created that defines the heat release rate. Next, the fire will be attached to the top of the switchgear cabinet. Since the fire properties are specified in SI units

1. On the **View** menu, click **Units**.

2. Select **SI**.

To create the surface that represents fire:

1. On the **Model** menu, click **Edit Surfaces...**.

2. In the **Edit Surfaces** dialog, click **New...**.

3. In the **Surface Name** box, type `Fire`.

4. Select **Surface Type** and in the list, select `Burner`.

5. Click **OK** to close the **New Surface** dialog.

To change the properties of the **Fire**:

1. In the **Edit Surfaces** dialog, click to select **Fire**.

2. Select **Heat Release Rate** and type `3000.0` kW/m^2. Note that this value is the HRR per unit area. This corresponds to a peak HRR of 834 kW total distributed over an area of 3.0 square feet (0.278 square meters).

3. In the **Ramp-up Time** list, select `T2` and type a value of `60` s. This is a time-squared fire that will ramp up over 60 seconds to the peak value.

4. Click **OK** to close the **Edit Surfaces** dialog.

Switch back to English units to define the geometry.

1. On the **View** menu, click **Units**.

2. Select **English**.

To add the fire to the cabinet:

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

2. In the **Description** box, type `Fire`.

3. In the **Group** list, select `Fire`.

4. In the **Surface** list, select `Fire`.

5. Click the **Geometry** tab. In the **Lies in the plane** list, select `X` and type `50.51`. This places it just barely outside the cabinet for visualization, but it will be returned to the exact mesh coordinates during solution. To repeat a word of caution. In most cases this slight offset will not cause any problem, since both the vent (fire) and cabinet will be moved to the same mesh coordinates during the solution. However, it can happen that the small offset will mean that the vent is moved to a different mesh coordinate than the cabinet. An unattached vent will be ignored in the solution, so either: position the vent directly on the cabinet or look at the mesh in Smokeview to make sure you see the vent (fire) displayed.

6. In the **Bounds** boxes, enter the values in Table 7.19.

7. Click **OK** to close the **Vent Properties** dialog.

Table 7.19. Fire dimensions

	Y (ft)	Z (ft)
Min	20.0	6.5
Max	23.0	7.5

Smoke Detectors

Each room has a smoke detectors. In this analysis, they will be used to output data, but will not activate any features in the model.

To create the smoke detectors:

1. On the **Devices** menu, click **New Smoke Detector...**.

2. In the **Detector Name** box, type `Room 3B`. In the **Location** boxes, type `14.0`, `10.0`, and `19.75`.

3. Click **OK** to create the detector.

4. On the **Devices** menu, click **New Smoke Detector...**.

5. In the **Detector Name** box, type `Room 3A`. In the **Location** boxes, type `44.5`, `10.0`, and `19.75`.

6. Click **OK** to create the detector.

Post-Processing Controls

There are a number of tools provided in PyroSim to instrument the building and determine what is going on at each time step. Slice files, thermocouples, and boundary files are some of the most common instruments. We will define a layer device.T.

Layer Zone Device

To add a layer zone device:

1. On the **Devices** menu, click **New Layer Zoning Device...**.

2. In the **Device Name** box, type `Layer Room 3A`.

3. To add the device to Room 3A, fill the **Path** table by entering the values in Table 7.20.

4. Click **OK** to close the **Layer Zoning Device** dialog.

Repeat these steps, but this time adding a device to Room 3B, using the data in Table 7.21.

Table 7.20. Layer data for Room 3A

	X (ft)	Y (ft)	Z (ft)
End Point 1 (ft)	44.25	14.25	1.0
End Point 2 (ft)	44.25	14.25	19.0

Table 7.21. Layer data for Room 3B

	X (ft)	Y (ft)	Z (ft)
End Point 1 (ft)	14.25	14.25	1.0
End Point 2 (ft)	14.25	14.25	19.0

Click the **Show Devices** tool to enable display of the layer and temperature measurement devices.

Slice Planes

Slice planes can be used to display 2D contours in the Smokeview display of the results. In this analysis, we will save temperature data for future plotting. We will define a slice plane through the center of the room and the fire. To define the slice planes:

Click the **Show Slices** tool to enable display of the slices.

1. On the **Output** menu, click **Slices...**.

2. Fill the table by entering the values in Table 7.22. You can click on the row number to select entire rows to copy and paste, speeding the entry.

3. Click **OK** to close the **Animated Planar Slices** dialog.

Table 7.22. Slice plane data

XYZ Plane	Plane Value (ft)	Gas Phase Quantity	Use Vector?
Y	14.25	Temperature	NO
Y	21.5	Temperature	NO
Z	19.0	Temperature	NO

Boundary (Surface) Plots

Temperatures (and other quantities) on the surface of objects can be plotted by selecting *Surface Plots*. To define the surface plots:

1. On the **Output** menu, click **Boundary Quantities...**.

2. Click the **Wall Temperature** checkbox.

3. Click **OK** to close the **Animated Boundary Quantities** dialog.

Isosurfaces

Isosurfaces are 3D plots that show the surface on which a quantity has a specified value. We will save isosurface data for temperatures, which is one way to define the location of a hot gas layer. To define the isosurfaces:

1. On the **Output** menu, click **Isosurfaces...**.

2. Click the **Temperature** checkbox.

3. In the **Contour Values** column, enter `50 100 150 200 250`, where the contour values are separated by a space, Figure 7.19

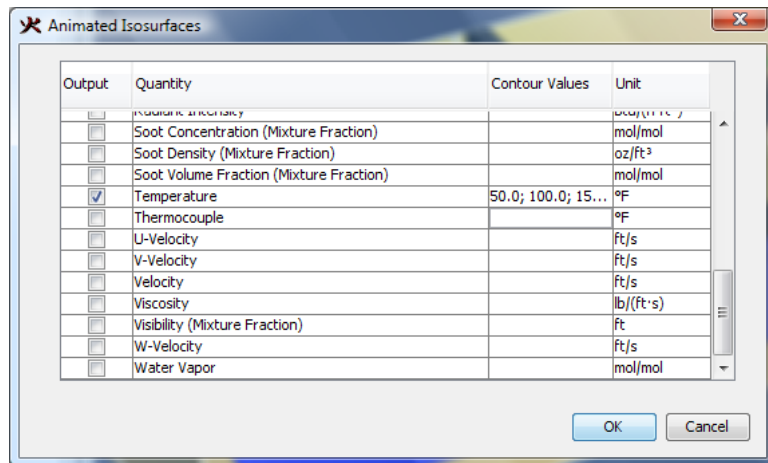4. Click **OK** to close the **Animated Isosurfaces** dialog.

Figure 7.19. Defining the temperature isosurfaces

Simulation Parameters

To define the end time:

1. On the **FDS** menu, click **Simulation Parameters...**.

2. In the **Simulation Title** box, type `Switchgear Fire`.

3. In the **End Time** box, type `600` s.

To define the surface to be used on the mesh boundary:

1. On the **Simulation Parameters** dialog, click the **Misc.** tab.

2. In the **Default Surface Type** list, select `Concrete Wall`.

3. Click **OK** to close the **Simulation Parameters** dialog.

Run the Analysis

Your model should now look like Figure 7.20. Save the model. To run the analysis
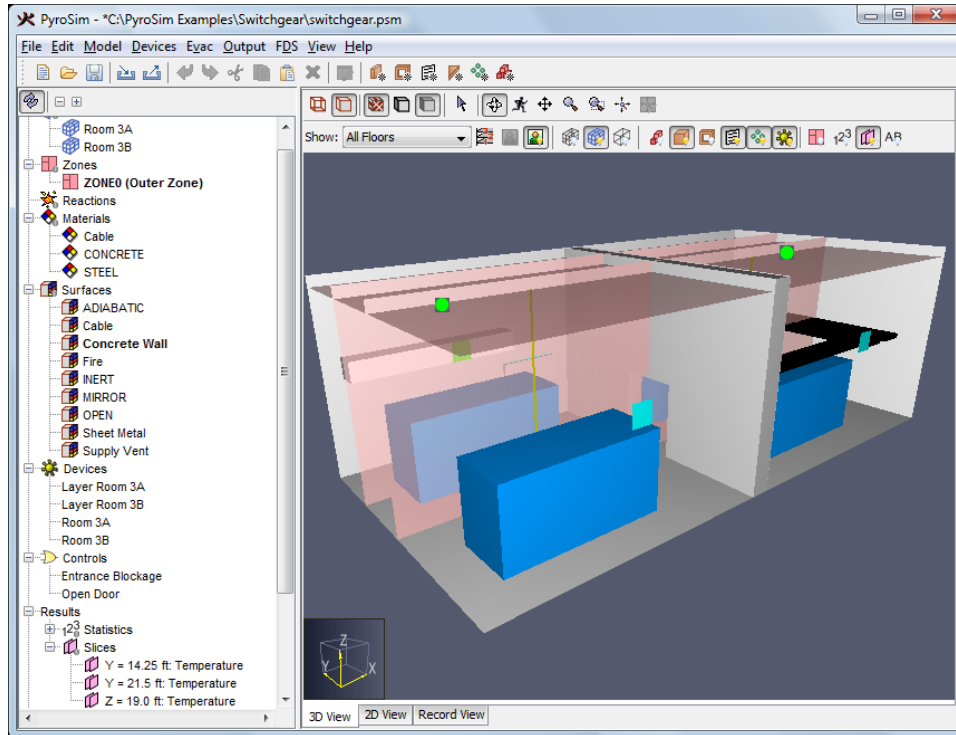
1. On the **FDS** menu, click **Run FDS...**.

Figure 7.20. Completed model

View the Results

You can use the SmokeView software to make plots of the results. An image showing the smoke density is shown in Figure 7.21. An image showing the temperature contours on the slice planes is shown in Figure 7.22. An image showing the temperature contours on the surfaces is shown in Figure 7.23.
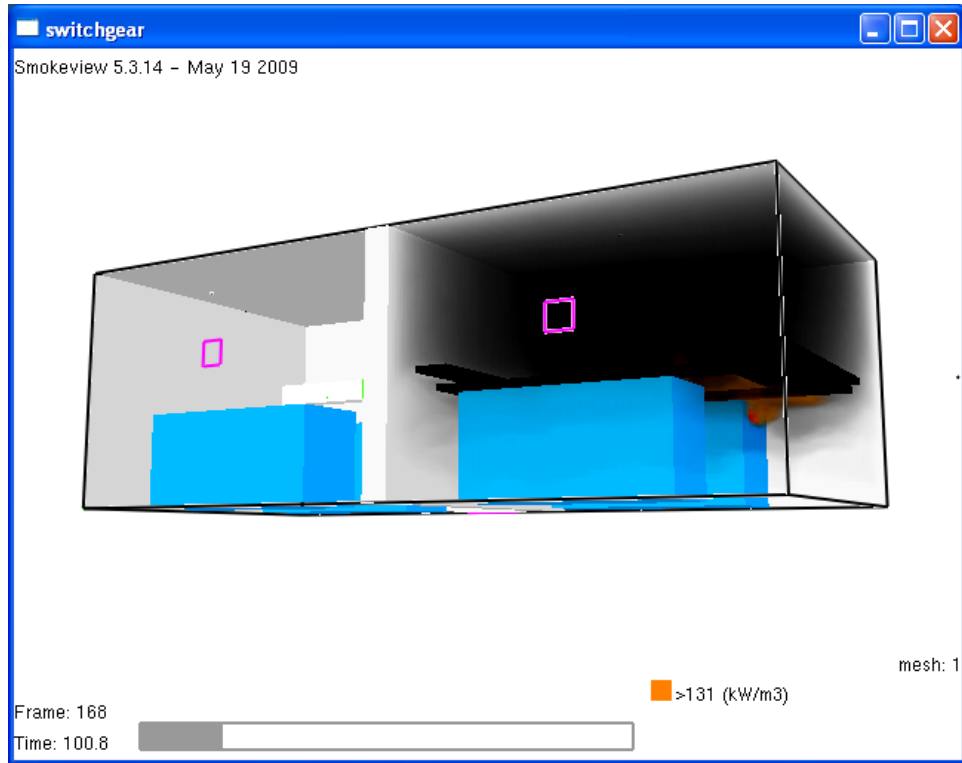
Figure 7.21. Smoke in rooms at 100 seconds
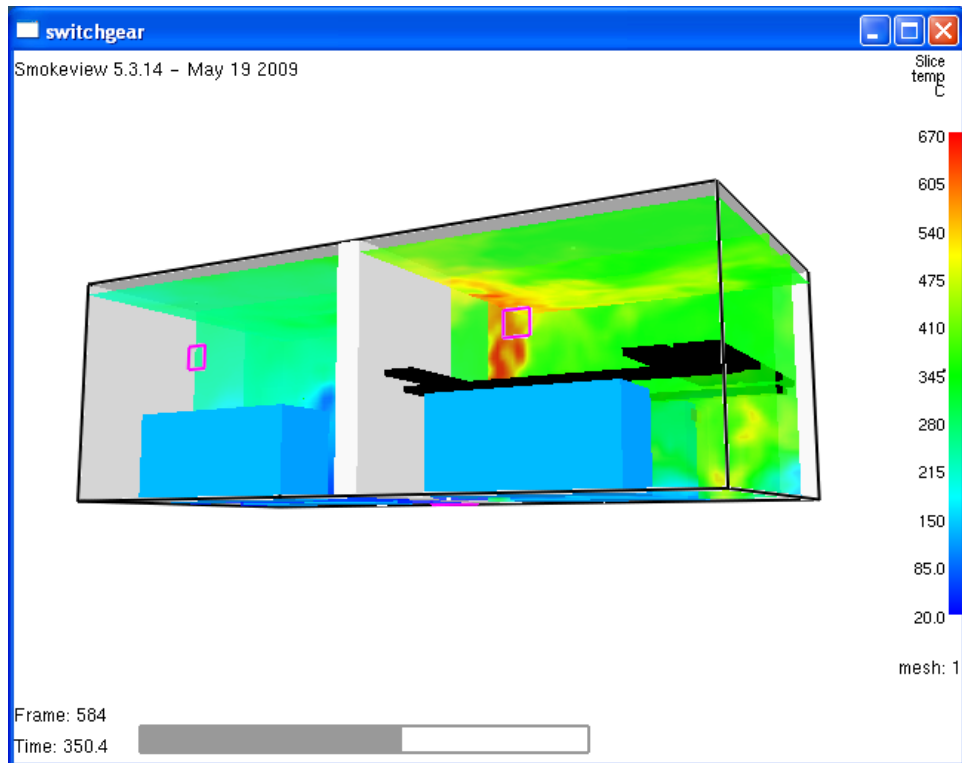
Figure 7.22. Temperature contours on slice planes at 350 seconds
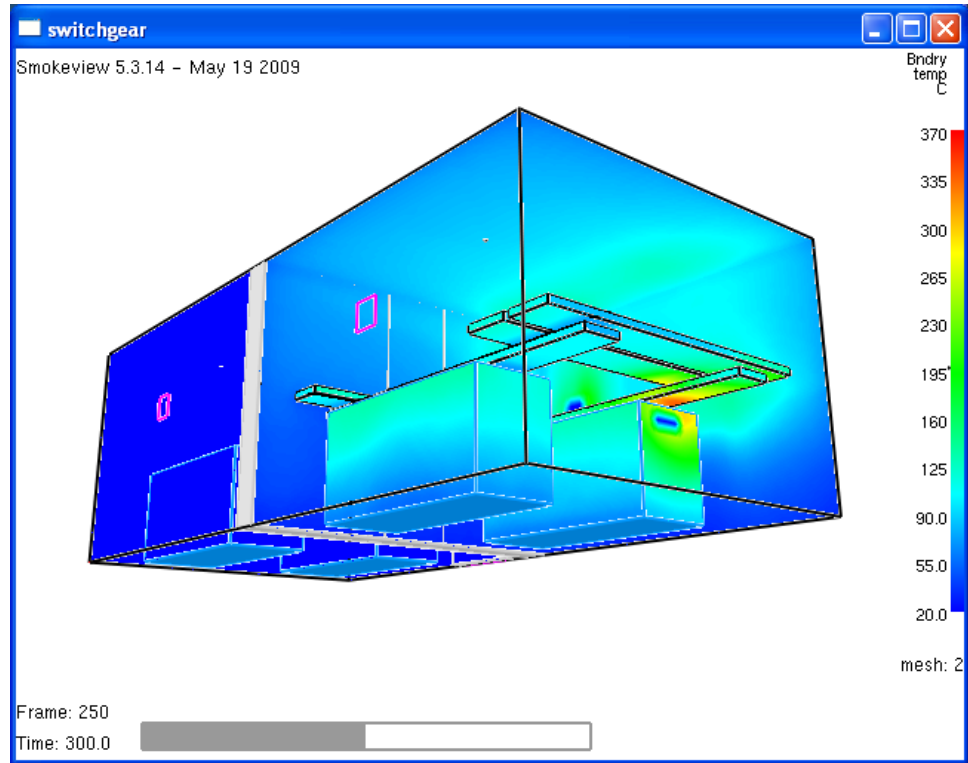
Figure 7.23. Temperature contours on the solid surfaces at 300 seconds

To view time history results

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 **switchgear_devc.csv** and click **Open** to view the device output. The first display will be the layer height in Room 3A, Figure 7.24.

Figure 7.24. Layer height in room 3A

Chapter 8. Single Room Evacuation

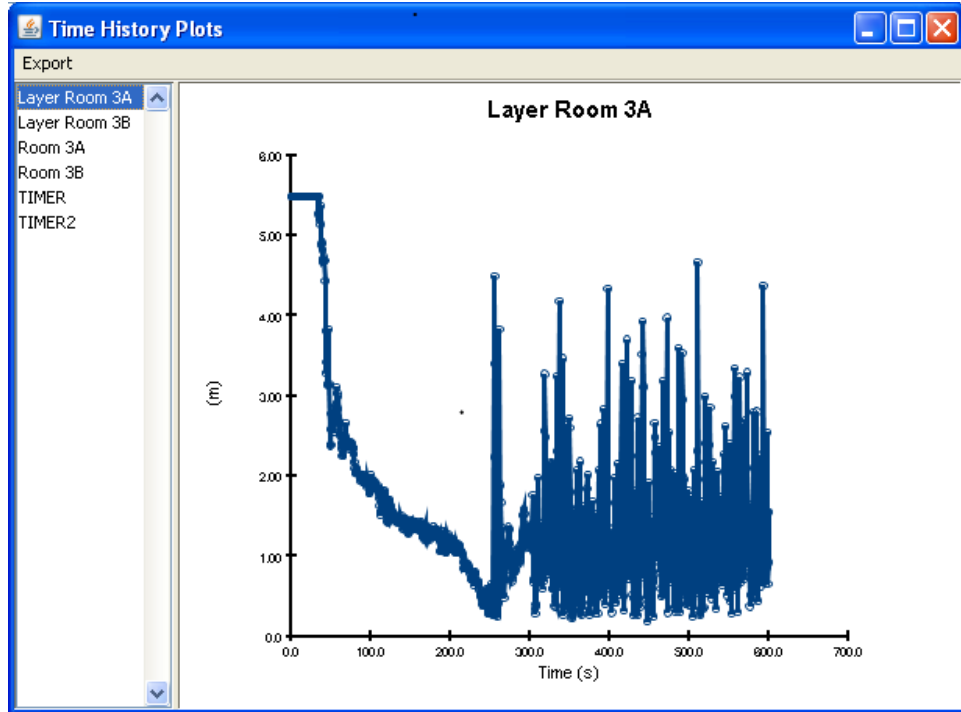This example walks you though a minimal FDS+EVAC example. The scenario is based on the 4th test case in the IMO evacuation simulator verification problem list.
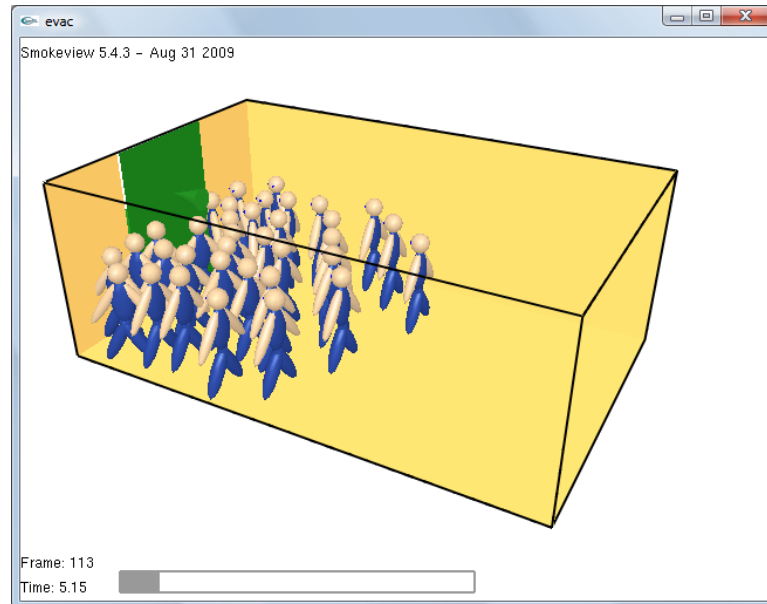
Figure 8.1. The EVAC example solution

Before you begin, ensure that you are using **SI** units (see Chapter 1).

Enable FDS+EVAC

To use PyroSim's FDS+EVAC features, these features must be manually activated. To activate PyroSim's FDS+EVAC features: on the **Evac** menu, click **Enable FDS+EVAC**.

Create Mesh

EVAC uses a flow solution to define the movement of the occupants. This flow calculation is independent of any meshes and flows used for the fire simulation. In this example we will use an 8 m x 5 m x 3 m EVAC mesh.

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 0.0 and in the **Max X** box, type 8.0.

5. In the **Min Y** box, type 0.0 and in the **Max Y** box, type 5.0.

6. In the **Min Z** box, type 0.0 and in the **Max Z** box, type 3.0.

7. In the **X Cells** box, type `16`.

8. In the **Y Cells** box, type `10`.

9. In the **Z Cells** box, type `1`.

You can safely ignore the cell size ratio warning. All evacuation MESH records must have only 1 cell in the Z direction.

At this point, the MESH is still a fire mesh. To specify that this mesh will be used for evacuation:

• Click to select the **Evacuation Grid** option.

The current setting is sufficient for MESH entries that will be used only to calculate flow fields for doors and exits, but since this MESH will be used directly by the agents it is necessary to enable the **Evac Humans** option as well:

• Click to select the **Evac Humans** option.

See Chapter 8 of [Korhonen and Hostikka, 2009] for a detailed discussion of the meshes.

Click **OK** to save changes and close the **Edit Meshes** dialog.

Create an Exit

Each DOOR and EXIT in FDS+EVAC requires an exhaust vent to generate the correct flow field to be used with the movement simulation. This vent must have a very weak outflow to prevent numerical instability. We first create a surface with an exhaust flow:

1. On the **Model** menu, click **Edit Surfaces...**.

2. Click **New**.

3. In the **Surface Name** type `Outflow`.

4. In the **Surface Type** list, select `Exhaust`.

5. Click **OK** to create the new surface.

6. In the **Specify Velocity** box, type `1.0E-6`.

7. In the **Ramp-Up Time** list, select `Tanh` and in the box, type `0.1 s` for the ramp-up time.

8. Click **OK** to close the **Edit Surfaces** dialog.

We now define the vent that will use the **Outflow** surface. To create the outflow vent:

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

2. In the **Description** type `Exit Vent`.

3. In the **Surface** list, select `Outflow` surface we just created.

4. Click the **Geometry** tab.

5. In the **Plane** list, select `X` and in the box, type `0.0 m`.

6. In the **Min Y** box, type `1.5` and in the **Max Y** box, type `3.5`.

7. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.

8. Click the **Evac** tab.

9. In the **Use In** list, select `Evac Only`.

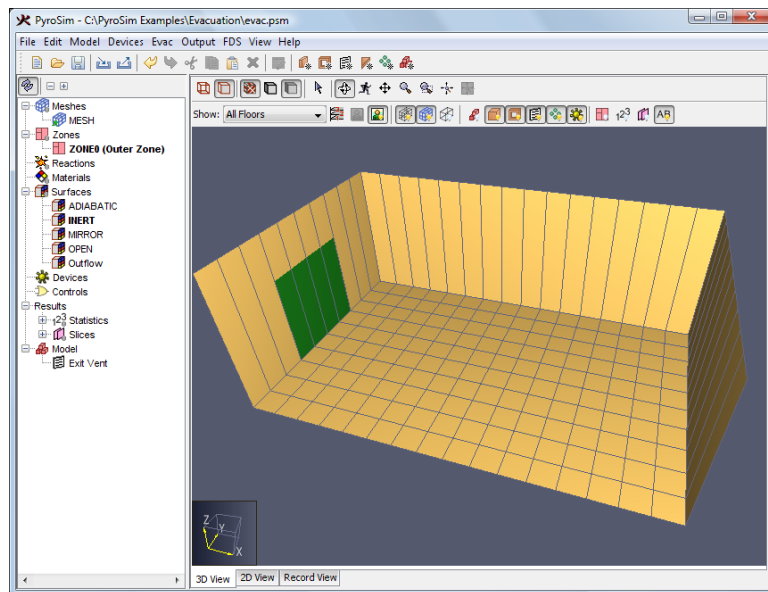10. Click **OK** to close the **Vent Properties** dialog.

Figure 8.2. Model after adding vent

Create an EVAC exit at the same location as the Exit Vent:

1. On the **Evac** menu, click **Exits...**.

2. Click **New** and then click **OK**.

3. In the **Min X** box, type `0.0` and in the **Max X** box, type `0.0`.

4. In the **Min Y** box, type `1.5` and in the **Max Y** box, type `3.5`.

5. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.

6. In the **Orientation** list, select `-X`. This defines the direction in which the occupants will pass through the exit.

7. Click **OK** to close the **Edit Exits** dialog.

Add Occupants

Each DOOR and EXIT in FDS+EVAC requires an exhaust vent to generate the correct flow field to be used with the movement simulation. This vent must have a very weak outflow to prevent numerical instability. We first create a surface with an exhaust flow:

1. On the **Evac** menu, click **Person Types...**.

2. Click **New** and click **OK**.

3. For the **Reaction Time** click the **Edit** button and in the **x** box, type `0.0`. Click **OK**.

4. Click **OK** again to close the **Edit PERS** dialog.

To place the occupants:

1. On the **Evac** menu, click **Initial Positions...**.

2. Click **New** and click **OK**.

3. In the **Persons** box type `50`.

4. In the **Min X** box, type `0.0` and in the **Max X** box, type `8.0`.

5. In the **Min Y** box, type `0.0` and in the **Max Y** box, type `5.0`.

6. In the **Min Z** box, type `0.0` and in the **Max Z** box, type `2.0`.

7. Click **OK** to close the **Initial Positions** dialog.

Specify Simulation Properties

To define the end time:

1. On the **FDS** menu, click **Simulation Parameters...**.

2. In the **Simulation Title** box, type `Evacuation`.

3. In the **End Time** box, type `50 s`.

4. Click **OK**.

Save the Model

This is a good time to 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. Name the file `evac.psm`.

3. Click **OK** to save the model.

Run the Analysis

To run the analysis:

1. On the **FDS** menu, click **Run FDS...**. Click **Save** to save the FDS+EVAC input file in the same location as the model.

View the Results

The Smokeview program will launch at the end of the analysis.

1. Right-click the Smokeview window and in the **Load/Unload** menu select **Evacuation** and then select **humans**.

2. Right-click the Smokeview window and in the **Show/Hide** menu select **Use Avatar** and then select **human_fixed**.
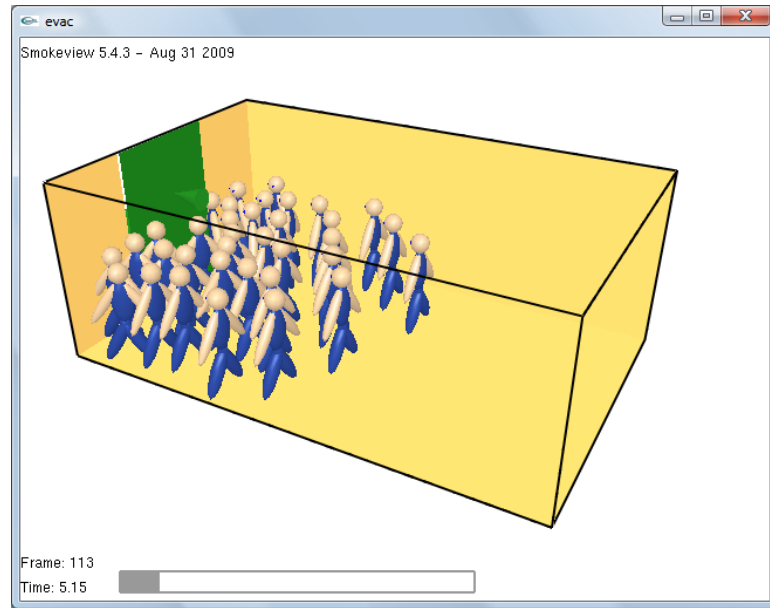

Figure 8.3. Display of movement to exit

To view time history results

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results...**.

2. A dialog will appear showing a list of 2D result files. Select **evac_evac.csv** and click **Open** to view the occupants in the room as a function of time, Figure 8.4.
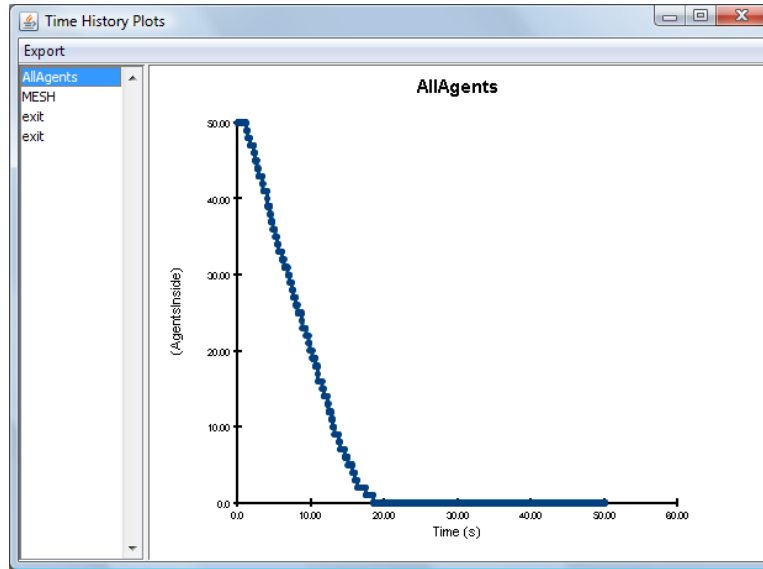
Figure 8.4. Occupants as a function of time

Using Pathfinder to Solve the Same Problem

Pathfinder, software developed by Thunderhead Engineering, can also perform an egress analysis. In addition to realistic 3D images, Pathfinder can extract floors from dxf data and offers an "SFPE Mode" that follows the SFPE *Engineering Guide - Human Behavior in Fire* guidelines, [SFPE, 2003]. We now repeat the same example using Pathfinder.
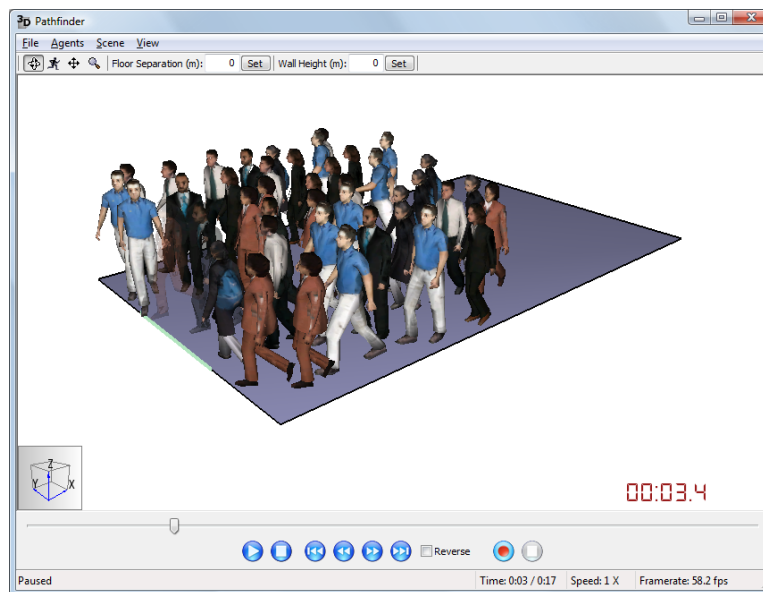

Figure 8.5. Snapshot of 3D movement in Pathfinder

Define the Room

In Pathfinder, rooms define the geometry:

1. In the **Top** view, select the **Add Rectangular Room** tool.

2. Either click and drag or use the dimension boxes to define a room starting at **X**= `0.0`, **Y**= `0.0` and extending to **X**= `8.0`, **Y**= `5.0`.

Create an Exit

In Pathfinder, doors that do not connect rooms are exit doors. To define the door:

1. In the **Top** view, select the **Add a New Door** tool.

2. Click and drag to define a door starting at **X**= `0.0`, **Y**= `1.5` and extending to **X**= `0.0`, **Y**= `3.5`.

Add Occupants

In Pathfinder, occupants are assigned to rooms:

1. Right-click on the room and select **Add Occupants...**.

2. By default, 50 occupants will be added. Click **OK** to add the occupants and close the **Add Occupants** dialog.
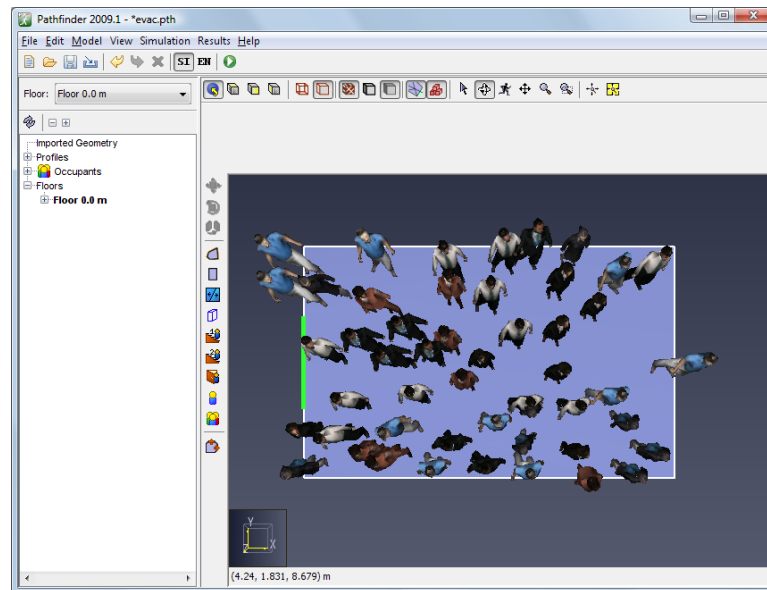

Figure 8.6. The Pathfinder model after occupants are added

Specify Simulation Properties

To define the end time:

1. On the **Simulation** menu, click **Simulation Parameters...**.

2. In the **Time Limit** box, type `50`.

3. Click **OK** to close the **Simulation Parameters** dialog.

Save the Model

This is a good time to save the model:

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

2. Choose a location to save the model. Name the file evac.pth.

3. Click **OK** to save the model.

Run the Analysis

To run the analysis:

1. On the **Simulation** menu, click **Run Simulation...**. Click **Save** to save the FDS+EVAC input file in the same location as the model.

View the Results

The 3D Pathfinder results window will launch at the end of the analysis.

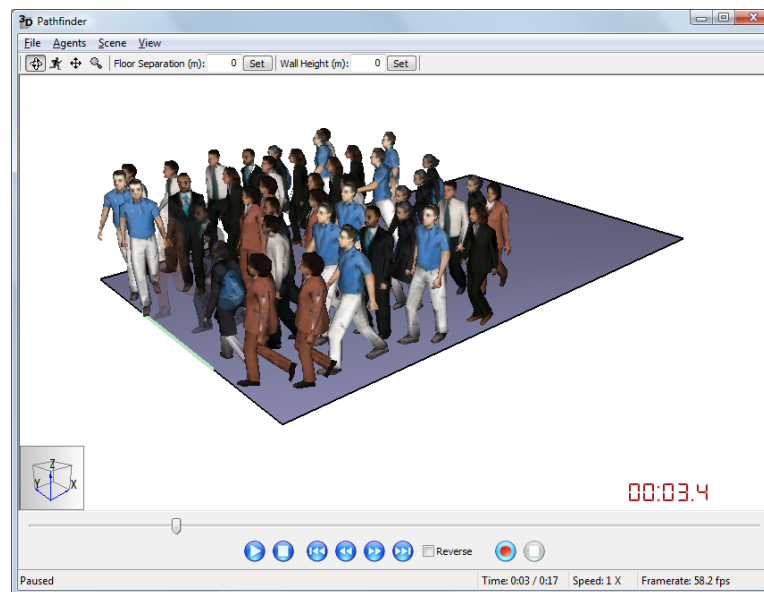1. Click the **Run** button to display the results..

Figure 8.7. Snapshot of 3D movement

To view time history results:

1. In the PyroSim window, on the **Results** menu, click **Room Usage...**. Note that for this analysis, the exit times predicted by Pathfinder and FDS+EVAC are essentially identical.
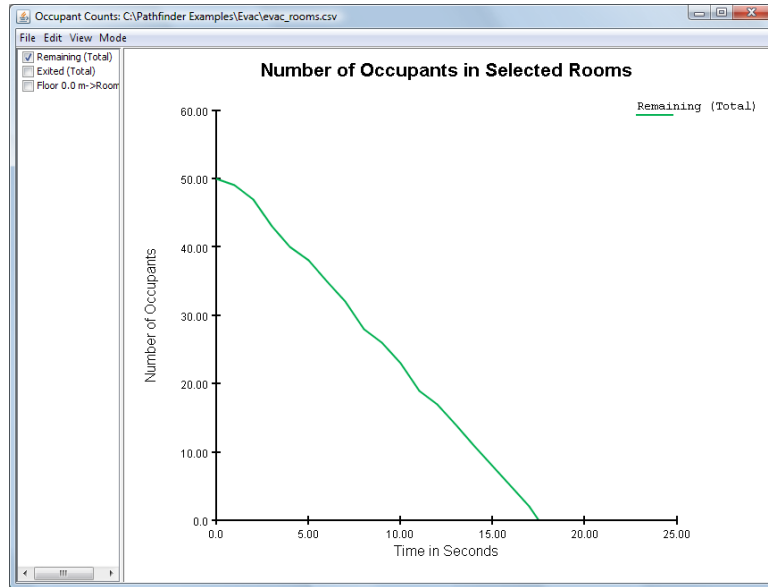
Figure 8.8. Room occupants as a function of time

References

[FDS-SMV Official Website] *Fire Dynamics Simulator and Smokeview*. Hosted at the National Institute of Standards and Technology. Gaithersburg Maryland USA .

[Klein, 2007] Bryan Klein. *Switchgear example problem, personal communication.*. 2007. National Institute of Standards and Technology. Gaithersburg MD USA .

[Korhonen and Hostikka, 2009] Timo Korhonen and Simo Hostikka. *Fire Dynamics Simulator with Evacuation: FDS+Evac*. April, 2009. ISBN: 978-951-38-7180-2 (http://www.vtt.fi/publications/index.jsp). VTT Technical Research Centre of Finland. P.O. Box 1000 Finland . VTT Working Papers 119.

[McGrattan, Klein, Hostikka, and Floyd, 2009] Kevin McGrattan, Bryan Klein, Simo Hastikka, and Jason Floyd. *Fire Dynamics Simulator (Version 5) User's Guide*. April, 2009. NIST Building and Fire Research Laboratory. Gaithersburg Maryland USA . NIST Special Publication 1019-5.

[SFPE, 2003] *Engineering Guide - Human Behavior in Fire*. June, 2003. Society of Fire Protection Engineers. 7315 Wisconsin Avenue, Suite 620E Bethesda Maryland USA 1-301-718-2910 .

[U.S. NRC, 2007] Kevin McGrattan, Bryan Klein, Simo Hastikka, and Jason Floyd. *Verification and Validation of Selected Fire Models for Nuclear Power Plant Applications*. May, 2007. U.S. Nuclear Regulatory Commission, Office of Regulatory Research. Washington DC USA . NUREG-1824, EPRI 1011999.