PyroSim

Atrium Smoke Movement

THUNDERHEAD ENGINEERING

2019

Smoke Movement in Atrium Buildings

In this example you will simulate smoke movement in an atrium with and without extraction fans. Some new concepts in this example include:

- More complex geometry.
- Using multiple meshes.
- Defining a fan activation control that uses input from multiple smoke detectors.
- Using extraction fans created with HVAC components.

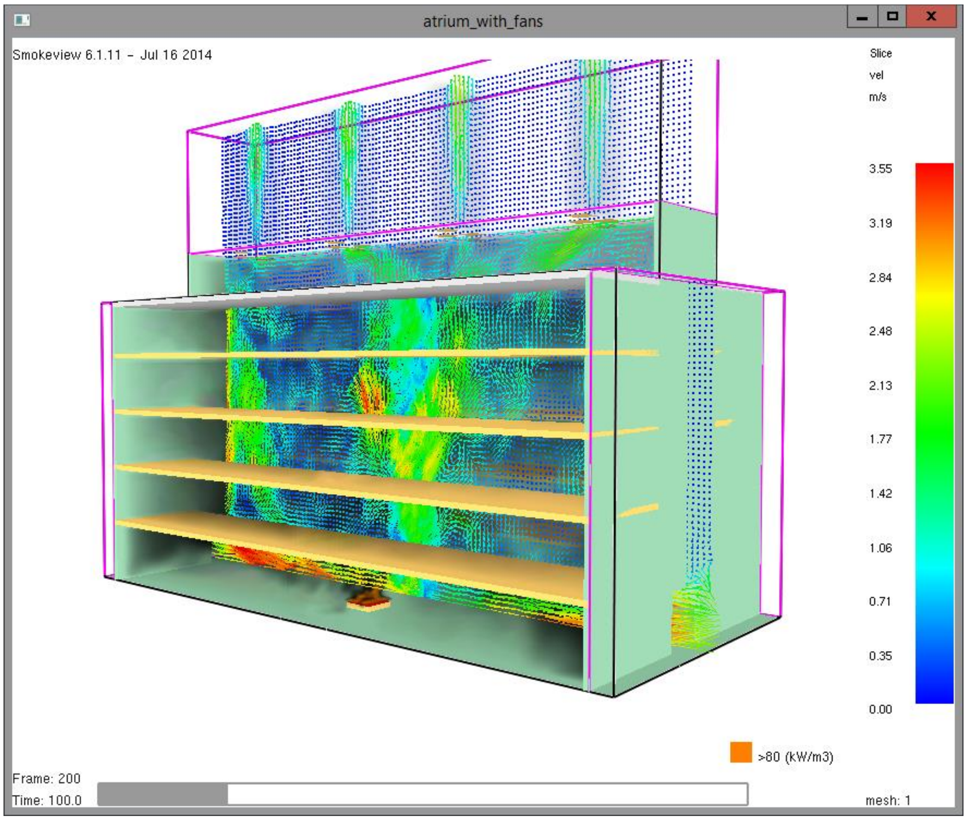

Figure 1: Velocity contours with fans.

This tutorial demonstrates how to:

- Create the atrium geometry.
- Use multiple meshes with open boundaries.
- Define a Heat Release Rate fire.
- Create smoke detectors and a control.
- Define HVAC fans and vents that are activated by the control.

Our simulation will replicate the model described by Chew and Liew, 2000. The geometry of the problem is shown in Figure 2.

There are two openings of 5 m by 5 m at the first level for air inlet, and eight powered extraction fans with discharge capacity of 20 m^3/s, each located at the smoke reservoir. Each level has a 10 m by 60 m floor plate on both sides with the central opening forming the atrium and the smoke reservoir. The numerical experiment NA is a 5-storey atrium of 30 m high and NB is an 11-storey atrium of 60 m high.

The fire source was modelled by defining heat flux at one designated location of the fire outbreak. Mobile stores were located in the atrium. The fire size was assumed to be 5 MW

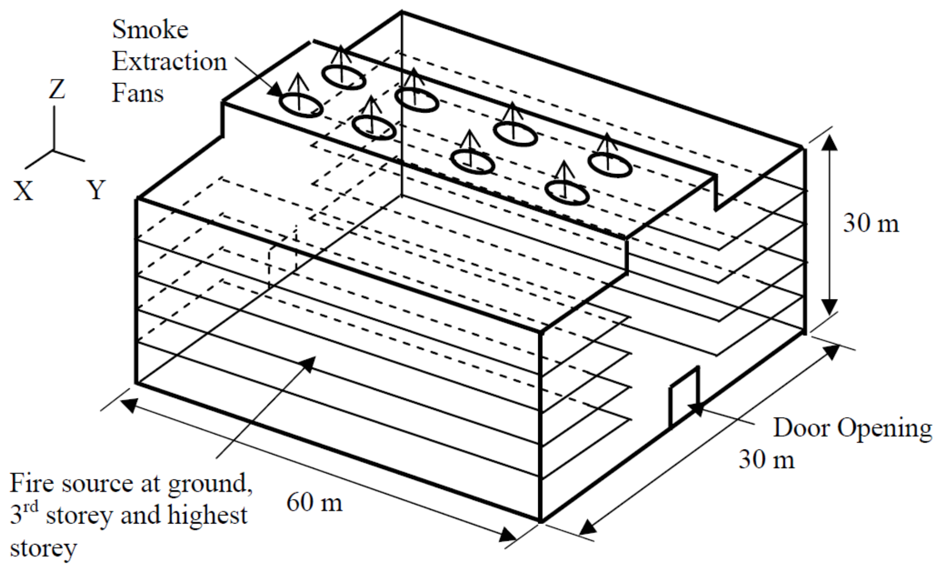

Figure 2: Geometry of atrium NA (image from Chew and Liew, 2000)

We will make the following assumptions:

- The floor height is 6 m.
- The floor, walls, and floor plates are concrete 0.3 m thick.
- The roof is insulated (adiabatic).

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

1. Select SI Units

To select SI units:

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

2. Name and Save the Model

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

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

3. Concrete Material

To create the concrete walls and floor, we first create a concrete material. This defines the conductivity and other thermal properties of the concrete.

1. On the **Model** menu, click **Edit Materials**.
2. Click **Add From Library**.
3. From the list of materials in the Library, click **CONCRETE** and then click the left arrow (**<--**) to move concrete into the **Current Model**. Click **Close** to close the library.
4. Click **OK** to close the **Edit Materials** dialog.

4. Concrete Surface

The concrete surface will be used when we make the floor, walls, and floor plates. We will use a *Layered* surface so that thermal conduction into the surface will be included in the calculation. We will use a 0.3 m thick slab of concrete and has a back face boundary condition to ambient air (PyroSim default). To define the surface:

1. On the **Model** menu, click **Edit Surfaces**.
2. Under the surface list on the left side of the dialog, click **New**.
3. In the **Surface Name** box type **Concrete**. In the **Surface Type** list, click **Layered**. Click **OK**.
4. In the **Material Layers** tab, click on the first row and in the **Thickness** column, type **.3**.
5. In the row, click **Edit**.
6. In the **Mass Fraction** box type **1** and in the **Material** list, click **CONCRETE**. Click **OK**.
7. Click **OK** to close the **Edit Surfaces** dialog.

5. Create the Model Geometry

In building this model, we will include all exterior walls and simulate flow out of the building ends and above the building. We do this both because it makes the final visualization more realistic and because the ground and open space above the building can affect the flow.

In parts of the model we will use a solution grid of 0.5 m, therefore the geometry is described at that level of resolution. This prevents thin walls or floors from snapping down to the mesh in regions where we use a finer (0.25 m) mesh.

Floor

First add the floor. The floor will be a vent with a concrete surface. As discussed in previous examples, a *vent* in FDS has a general meaning of a boundary condition on a surface. In this case, the boundary condition is the floor.

1. On the **Model** menu, click **New Vent.**
2. In the **ID** box, type **Floor**.
3. In the **Surface** list, click **Concrete**.
4. Click the **Geometry** tab.
5. In the **Min X** box, type **0** and in the **Max X** box, type **30.**
6. In the **Min Y** box, type **-2** and in the **Max Y** box, type **62.**
7. Click **OK** to create the vent.
8. On the **View** menu, click **Reset View to All Objects** (or press CTRL+R**)**.

Floor Plates

Next create the floor plates. These will be dimensioned to meet the walls at the interior surfaces.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Floor Plate**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **0** and in the **Max X** box, type **10.**
5. In the **Min Y** box, type **0** and in the **Max Y** box, type **60.**
6. In the **Min Z** box, type **6** and in the **Max Z** box, type **6.5**.
7. Click the **Surfaces** tab.
8. In the **Single** list, click **Concrete**.
9. Click **OK** to create the obstruction.

Now we will copy this floor plate vertically.

1. Right-click on the **Floor Plate** and click **Copy/Move**.
2. For **Mode**, select **Copy** and in the **Number of Copies** box, type **3.**
3. In the **Offset Z** box, type **6.**
4. Click **OK** to copy.

Now we will copy the four floor plates horizontally.

1. Holding the **CNTRL** key, click to select all four floor plates.
2. Right-click on the selection and click **Copy/Move**.
3. For **Mode**, select **Copy** and in the **Number of Copies** box, type **1.**
4. In the **Offset X** box, type **20.**
5. Click **OK** to copy.

Exterior Walls

Now add the exterior walls. These will be obstructions with concrete surfaces. First the front wall.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **End Wall**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **0** and in the **Max X** box, type **30.**
5. In the **Min Y** box, type **59.5** and in the **Max Y** box, type **60.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **30.**
7. Click the **Surfaces** tab.
8. In the **Single** list, click **Concrete**.
9. Click **OK** to create the obstruction.

Now the front smoke reservoir at the top of the wall.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **End Reservoir**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type 1**0** and in the **Max X** box, type **20.**
5. In the **Min Y** box, type **59.5** and in the **Max Y** box, type **60.**
6. In the **Min Z** box, type 3**0** and in the **Max Z** box, type **36.**
7. Click the **Surfaces** tab.
8. In the **Single** list, click **Concrete**.
9. Click **OK** to create the obstruction.

Now the end door. Note that the boundaries of the hole extend past the wall to ensure the hole is properly cut in the FDS mesh.

1. On the **Model** menu, click **New Hole**.
2. In the **ID** box, type **End Door**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type 1**2.5** and in the **Max X** box, type **17.5.**
5. In the **Min Y** box, type **59** and in the **Max Y** box, type **60.5.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **5**.
7. Click **OK** to create the hole.

Copy the front wall, reservoir, and door to the back.

1. Press the CTRL key and click to select the End Wall, End Reservoir, and End Door.
2. Right-click and click **Copy/Move.**
3. For **Mode**, select **Copy** and in the **Number of Copies** box, type **1.**
4. In the **Offset Y** box, type **-59.5**.
5. Click **OK** to copy.

In the upper toolbar, select the display option **Realistic with Outlines** and your model should look like Figure 3.

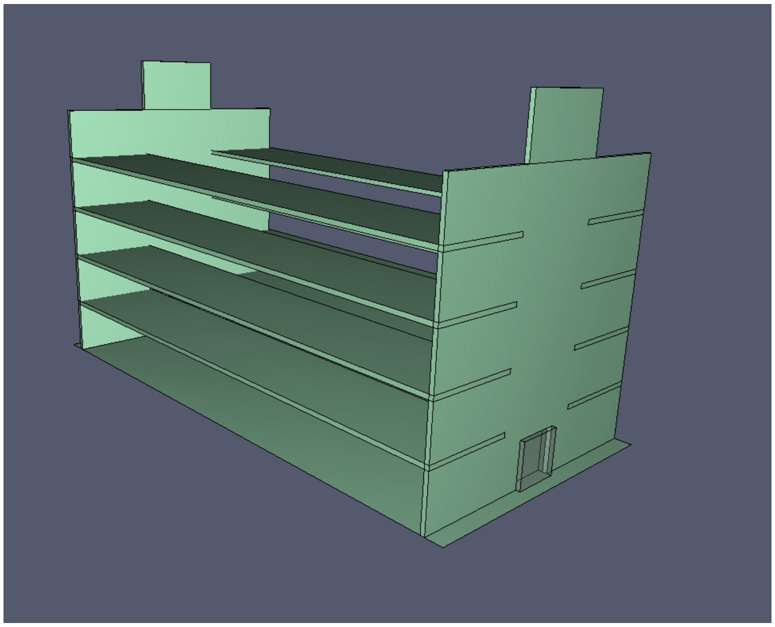

Figure 3: Model after adding floor, floor plates, and front and back walls.

Now the right side wall.

1. On the **Model** menu, click **New Obstruction.**
2. In the **ID** box, type **Side Wall**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **29.5** and in the **Max X** box, type **30.**
5. In the **Min Y** box, type **0** and in the **Max Y** box, type **60.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **30.**
7. Click the **Surfaces** tab.
8. In the **Single** list, click **Concrete**.
9. Click **OK** to create the obstruction.

Copy the right side wall.

1. Right-click on the Right Side wall and click **Copy/Move.**
2. For **Mode**, select **Copy** and in the **Number of Copies** box, type **1.**
3. In the **Offset X** box, type **-29.5**.
4. Click **OK** to copy.

Roof

We will use copy and edit functions to create the roof.

1. Press the CTRL key and click to select the two upper Floor Plates, **Error! Reference source not found.**.
2. Right-click on a selected floor plate and click **Copy/Move.**
3. For **Mode**, select **Copy** and in the **Number of Copies** box, type **1.**

4. In the **Offset Z** box, type **5.5**.
5. Click **OK** to copy.

Edit the two roof panels.

1. Press the CTRL key and select the two newly copied obstructions, right-click and click **Properties.**
2. In the **ID** box, type **Roof**.
3. Click the **Surfaces** tab.
4. In the **Single** list, click **ADIABATIC**.
5. Click **OK** to change the properties.

Now add the roof panels that cover the smoke reservoir. Make three new obstructions with the dimensions shown in Table 1 and with adiabatic surfaces.

Table 1: Geometry of smoke reservoir panels

Name	Min X	Max X	Min Y	Max Y	Min Z	Max Z
Reservoir 1	10.0	10.5	0.0	60.0	29.5	36.0
Reservoir 2	19.5	20.0	0.0	60.0	29.5	36.0
Reservoir 3	10.0	20.0	0.0	60.0	35.5	36.0

Press the CTRL key and select the +X Side Wall and Reservoir 2 wall, then right-click and select **Hide Objects**. At this point, your model should look like Figure 4.

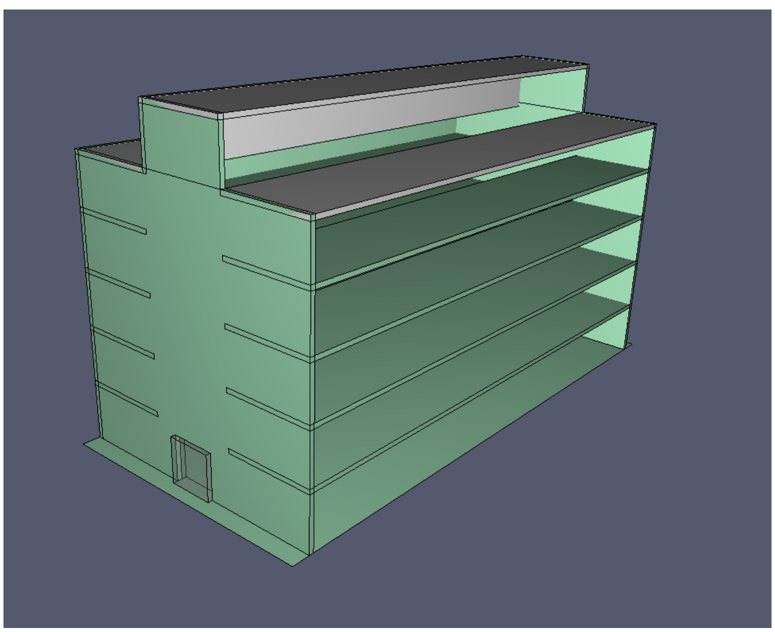

Figure 4: After walls and roof have been added. In this figure, the right wall and right reservoir have been hidden.

6. Define Fire

Reaction

We will first change the reaction to Propylene. This fire will produce less smoke than the default Polyurethane reaction.

1. On the **Model** menu, click **Edit Reactions.**
2. Under the reaction list on the left of the dialog, click **Add From Library.**
3. In the Library list, click **PROPYLENE** and then click the left arrow (**<--**) to move reaction into the Current Model. Click **Close** to close the library.
4. Click **OK** to close the **Edit Reactions** dialog.

Burner Surface

The burner surface will define heat release rate to the simulation. We want a total heat release rate of 5 MW, which will be distributed over a 9 m^2 area. The fire will ramp to the full value and extinguish at 50 seconds. To define the surface:

1. On the **Model** menu, click **Edit Surfaces.**
2. Under the surface list on the left of the dialog, click **New.**
3. In the **Surface Name** box type **Burner**. In the **Surface Type** list, click **Burner**. Click **OK**.
4. In the **Heat Release Rate Per Area** box type **555.6** (5000 kW HRR divided by 9 m^2).
5. In the **Ramp-Up Time**, select **Custom**. Click **Edit Value** and type in the values given below, then click **OK**.

Time	Fraction
0	0
1	1
50	1
51	0
100	0

6. Click the **Thermal** tab, for the **Boundary Condition Model** select **Fixed Temperature**. For **Surface Temperature**, type **500**. For **Emissivity**, type **0.9**.
7. Click **OK** to close the **Edit Surfaces** dialog.

The fire will be defined on the top surface of a 3x3x0.5 m obstruction located under the lower floor plate.

1. On the **Model** menu, click **New Obstruction.**
2. In the **ID** box, type **Burner**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **20** and in the **Max X** box, type **23.**
5. In the **Min Y** box, type **28.5** and in the **Max Y** box, type **31.5.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **.5.**

7. Click the **Surfaces** tab.
8. Select **Multiple** and in the **Surface** list for the **Max Z** face, click **Burner** (**Error! Reference source not found.**).
9. Click **OK** to create the obstruction.

7. Computational Mesh

For this fire, D* is 1.825 m. We will use a grid size of 0.25 at the fire location (~D*/7) and 0.50 m away from the fire. The easiest way to do this is to define two meshes that cover the entire model and then split the mesh around the fire. To create the mesh in the main building:

1. On the **Model** menu, click **Edit Meshes**
2. Click **New** and then click **OK** to create the new mesh.
3. In the **Min X** box, type **0** and in the **Max X** box, type **30.**
4. In the **Min Y** box, type **-2** and in the **Max Y** box, type **62.**
5. In the **Min Z** box, type **0** and in the **Max Z** box, type **30**
6. In the **X Cells** box, type **60.**
7. In the **Y Cells** box, type **128.**
8. In the **Z Cells** box, type **60.**
9. Click **Apply** to save changes to that mesh.

Now we add the mesh that will represent the smoke reservoir and the exterior of the building above the reservoir. That will allow us to visualize the smoke leaving the building.

1. In the **Edit Meshes** dialog, click **New** and then click **OK** to create the new mesh.
2. Click **OK** to create the new mesh.
3. In the **Min X** box, type 1**0** and in the **Max X** box, type **20.**
4. In the **Min Y** box, type **0** and in the **Max Y** box, type **60.**
5. In the **Min Z** box, type **30** and in the **Max Z** box, type **50**
6. In the **X Cells** box, type **20.**
7. In the **Y Cells** box, type **120.**
8. In the **Z Cells** box, type **40.**
9. Click **OK** to save changes and close the **Edit Meshes** dialog.

We now have two meshes that could be used to solve the problem. The mesh that represents the building has 460 800 cells and the mesh that represents the smoke reservoir has 96 000. We will split the building mesh for two reasons: to more evenly distribute calculations between processors and to refine the mesh at the fire.

To split the mesh, we will use the **Mesh Split** tool:

1. In the upper toolbar, click **Show Mesh Divisions**.
2. In the **Navigation View**, select **Mesh01** and then select the **Split Mesh** tool, Figure 5.
3. Drag the cursor along the lower edge of the mesh until it displays Y=27.0 m, then click, Figure 6.

4. Continue dragging the cursor along the lower edge of the mesh until it displays Y=33.0 m, then click.
5. Now, right-click and select **Finish**. The original mesh is now split into three meshes.

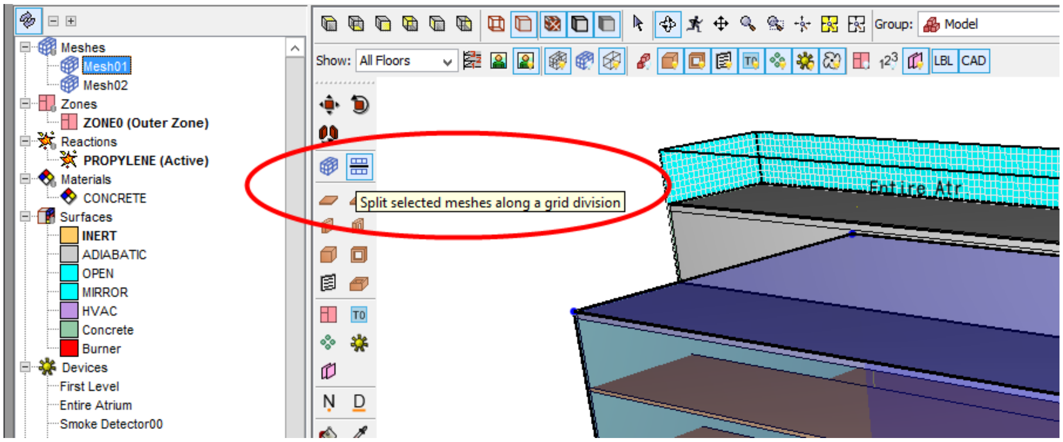

Figure 5: Select the Split Mesh tool

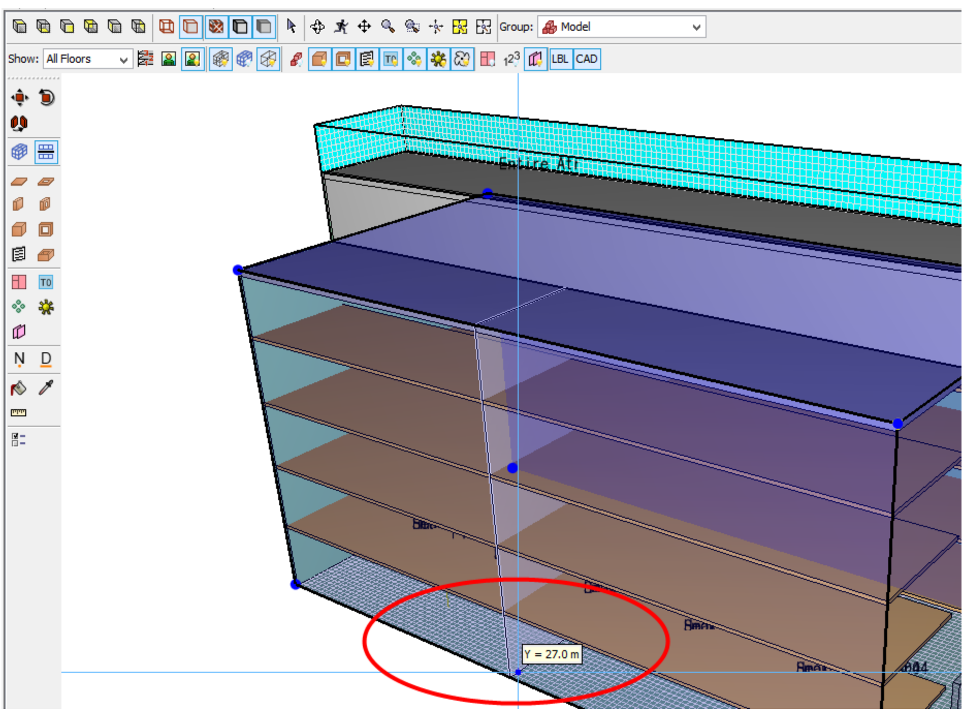

Figure 6: By moving cursor along edge, set the split mesh location, then click.

The new middle mesh now includes the fire. To refine this mesh:

1. In the Navigation View, select Mesh01-b (the middle mesh), right-click and then select **Properties**.
2. In the **X Cells** box, type **120.**
3. In the **Y Cells** box, type **24.**

4. In the **Z Cells** box, type **120.**
5. Click **OK**.

8. Add Open Vents on the Mesh Boundaries

The default FDS assumption is that mesh boundaries are closed and with INERT surfaces. We will add OPEN vents to the mesh boundaries on the front, back, and top of the model.

Front of Atrium

To add OPEN vents to the front boundary.

1. In the Navigation View, right-click Mesh01-c (the front mesh) and click **Open Mesh Boundaries**, Figure 7. This will add seven OPEN vents to the model.
2. Edit the XMAX and XMIN vents and on the Geometry tab, in the **MIN Y** box type **60**.
3. Delete the two OPEN vents over the roof.
4. Delete the ZMIN vent.

There should now be four OPEN vents on the front of the building, Figure 8.

Back of Atrium

Repeat these steps to add OPEN vents to the back of the atrium (MESH01-a). Edit the XMAX and XMIN vents and on the Geometry tab, in the **MAX Y** box type **0**. There should be four open vents at the back of the building.

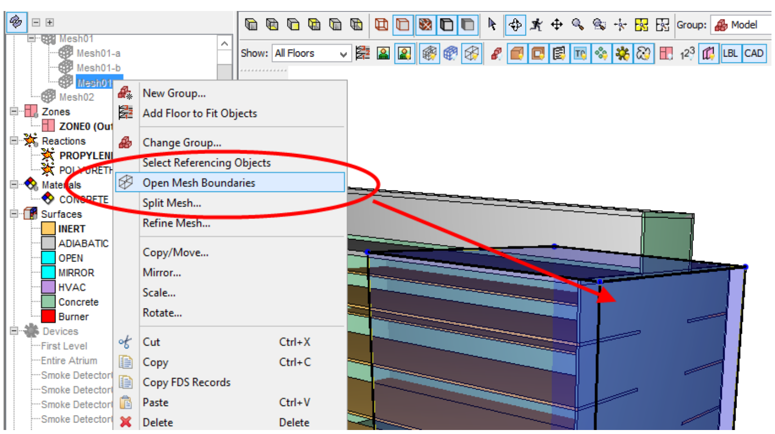

Figure 7: Open mesh boundaries

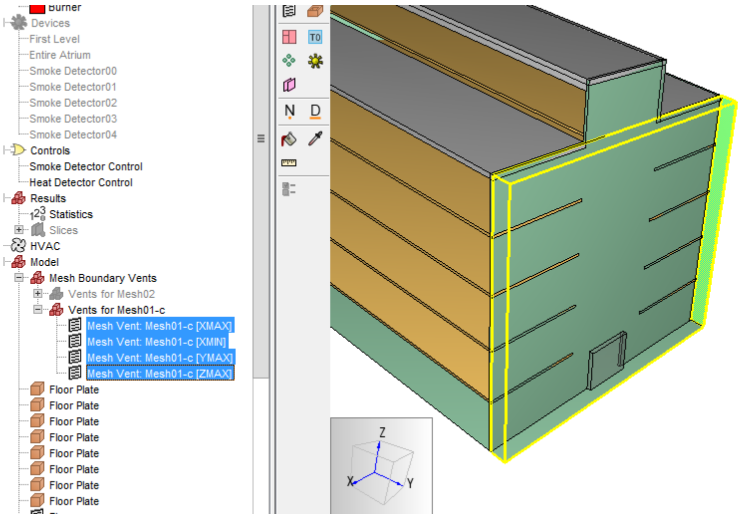

Figure 8: Open vents on front of atrium

Top of Smoke Reservoir

To add OPEN vents to the top of the smoke reservoir.

1. In the Tree View, right-click Mesh02 (the smoke reservoir mesh) and click **Open Mesh Boundaries**. This will add six OPEN vents to the model.
2. Edit the side (XMAX and XMIN), back (YMIN), and front (YMAX) vents and on the **Geometry** tab, in the **MIN Z** box type **36**.

As shown in Figure 9, there should now be open vents on the front, back, and top of the model.

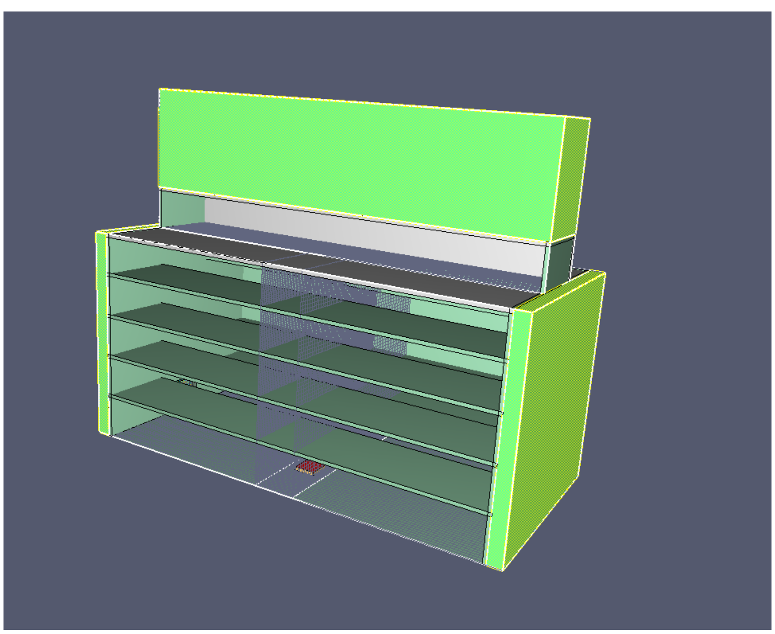

Figure 9: Final OPEN vents on mesh boundaries

9. Add Detectors

Detectors are used to detect the fire and can then be used as part of a control that initiates an action such as activating extraction fans. To define the detectors:

1. On the **Devices** menu, click **New Smoke Detector.**
2. In the **Name** box, type **Smoke Detector**. We will use the default detector.
3. In the **Location** boxes, type the **X, Y,** and **Z** coordinates as **25**, **10**, and **5.5**. The detectors must not be on a surface, but free-standing in the air.
4. Click **OK** to close the **New Smoke Detector** dialog.
5. Now right-click the **Smoke Detecto**r and click **Copy/Move.**
6. Make 4 copies, each offset by **10** m in the **Y** direction.

10. 2D Slice Planes

Slice planes are used to display 2D contours. In this analysis, we will save temperature, velocity, and visibility data for future plotting. To define the slice planes:

1. On the **Output** menu, click **2D Slices.**
2. Fill the table by entering the values in Table 2. 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.
4. Click the **Show Slices** tool to enable/disable display of the slices.

Table 2. Slice plane data

XYZ Plane	Plane Value (m)	Gas Phase Quantity	Use Vector?	Cell Centered?
X	**15**	Velocity	YES	NO
X	**15**	Temperature	NO	NO
X	**15**	Visibility	NO	NO
Y	**6**	Velocity	YES	NO
Y	**21**	Velocity	YES	NO
Y	**36**	Velocity	YES	NO
Y	**51**	Velocity	YES	NO
Y	**30**	Velocity	YES	NO
Y	**30**	Temperature	NO	NO
Y	**30**	Visibility	NO	NO

11. 3D Slices

FDS has two methods to output 3D data: 3D slices and Plot3D data. 3D slices (a 3D region for plotting data) is usually recommended, since it uses a more compact storage. We will plot temperature data for illustration.

1. On the **Output** menu, click **3D Slices.**
2. In the **Name** box, type **Temperature.**

3. In the **Min X** box, type **0** and in the **Max X** box, type **30.**
4. In the **Min Y** box, type **-2** and in the **Max Y** box, type **62.**
5. In the **Min Z** box, type **0** and in the **Max Z** box, type **50**
6. Select **NO** for **Use Vector?** and **Cell Centered?.**
7. Repeat for **Velocity** and **Visibility.**
8. Click **OK** to close the **Animated 3D Slices** dialog.

12. Specify Simulation Properties

We will run the simulation for 100 seconds.

1. On the **Analysis** menu, click **Simulation Parameters.**
2. In the **End Time** box, type **100.**
3. Click **OK**.

13. Optional - Run the Simulation

If desired, you can skip to section 15 and run the simulation without extraction fans.

14. Add Smoke Extraction Fans

There are two approaches we could take in defining the smoke extraction fans. The simplest would be to create what PyroSim calls Exhaust vents and position them at the top of the smoke reservoir. Then, the user can just specify the flow rate and connect the activation of the fans to the detectors, using controls. This is simple, but the smoke will just disappear from the model.

It is much more visually interesting if we use HVAC fans that will display the smoke being extracted from the building. The following instructions describe HVAC fans. We must define a vent on the top and bottom of the smoke reservoir obstruction. Then, we will connect the two vents with two ducts in series: the first duct will have a fan and the second duct a damper. The damper is used to set flow through the duct to zero and will open when connected to the detectors through the controls.

Add Vents

To add the 2x2 m vents:

1. On the **Model** menu, click **New Vent.**
2. In the **ID** box, type **Extraction Bottom**.
3. In the **Surface** list, click **HVAC.**
4. Click the **Geometry** tab.
5. In the **Plane** list, click **Z** and in the box, type **35.49** (the small offset avoids flashing when rotating and the vent will snap to the grid during the solution).
6. In the **Min X** box, type **12** and in the **Max X** box, type **14.**
7. In the **Min Y** box, type **5** and in the **Max Y** box, type **7.**
8. Click **OK** then click **OK** again to accept the HVAC warning and create the vent.

We repeat for the vent on the top surface of the obstruction:

1. On the **Model** menu, click **New Vent.**
2. In the **ID** box, type **Extraction Top**.
3. In the **Surface** list, click **HVAC.**
4. Click the **Geometry** tab.
5. In the **Plane** list, click **Z** and in the box, type **36.01** (the small offset avoids flashing when rotating and the vent will snap to the grid during the solution).
6. In the **Min X** box, type **12** and in the **Max X** box, type **14.**
7. In the **Min Y** box, type **5** and in the **Max Y** box, type **7.**
8. Click **OK** then click **OK** again to accept the HVAC warning and create the vent.

HVAC Fan

This creates a fan that we can use in the HVAC ducts.

1. On the **Model** menu, click **Edit HVAC**
2. Click **New** and in the **Type** list, click **Fan**. In the **Name** box, type **Smoke Fan**.
3. Select **Constant Flow** and in the **Volume Flow Rate** box, type **16**.
4. Click **OK** to close the **Edit HVAC** dialog.

HVAC Nodes and Ducts

Ducts connect the two vents. Ducts need two end nodes that we will create at the vents. The ducts will contain fans.

Add the bottom and top HVAC nodes that correspond to the HVAC vents:

1. In the navigation view, right-click the **Extraction Bottom** vent.
2. Click **Add HVAC Nodes.** This will make a new HVAC node named Node01.
3. In the tree view, right-click on Node01.
4. Click **Rename.**
5. Type in the name **Extraction Bottom**, and hit the Enter key.

Repeat for the **Extraction Top** vent:

1. In the tree view, right-click the **Extraction Top** vent.
2. Click **Add HVAC Nodes.**
3. In the tree view, right-click on Node02.
4. Click **Rename.**
5. Type in the name **Extraction Top**, and hit the Enter key.

Now create the HVAC fan duct. A control will be added to turn on the fan when a smoke detector activates. Note that even with the fan off, a small amount of flow through the duct can occur. To completely block flow, add a damper duct in series with the fan duct.

1. On the **Model** menu, click **New HVAC Duct.**
2. In the **ID** box, type **Fan Duct**.
3. In the **Node 1** list, click **Extraction Bottom**.

4. In the **Node 2** list, click **Extraction Top**.
5. Select **Non-circular** and in the **Area** box, type **4**, in the **Perimeter** box, type **8**.
5. Click the **Flow Model** tab.
6. In the **Flow Device** list, click **Fan** (not **Basic Fan).**
7. In the **Fan** list, click **Smoke Fan.** This selects the HVAC fan we created earlier.
8. In the **Activation** list, click **New**.
9. In the **Name** box, type **Fire Detector Control** and click **OK** to create the control.
10. For **Input Type**, select **Detector**.
11. In the text description, click **<nothing>** and select all five smoke detectors, then click **OK**.
12. Click **OK** again to close the **New Control** dialog.
13. Click **OK** to close the **HVAC Duct Properties** dialog.

Now we will copy these smoke extraction vents to make a row.

1. Holding the CTRL key, click on the **Extraction Bottom** vent, the **Extraction Top** vent, the **Extraction Bottom** HVAC node, the **Extraction Top** HVAC node, and the **Fan Duct**.
2. Right-click the selected objects and click **Copy/Move.**
3. For **Mode**, select **Copy** and in the **Number of Copies** box, type **3.**
4. In the **Offset Y** box, type **16**.
5. Click **OK** to copy.

Now we will copy the four extraction vents to a new row.

1. All the copied objects will be selected (alternately, select all copies of the vents, nodes, and ducts we just created).
2. Right-click on the selection and click **Copy/Move.**
3. For **Mode**, select **Copy** and in the **Number of Copies** box, type **1.**
4. In the **Offset X** box, type **4**.
5. Click **OK** to copy.

At this point, your model should look like Figure 10, with vents and fans located in the top of the roof.

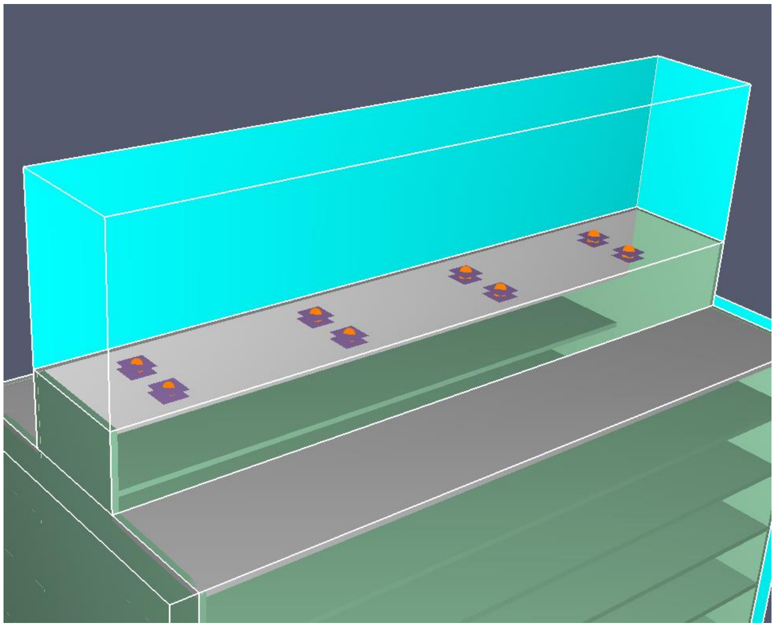

Figure 10: Model after HVAC ducts and fans are added (labels have been toggled off and the Reservoir 3 obstruction has been hidden)

15. Save the Model

1. On the **File** menu, click **Save**.
2. Click **Save** to save the model.

16. Run the Simulation

1. On the **FDS** menu, click **Run FDS Parallel.** Note that we can use parallel processing because we defined four meshes.
2. The **FDS Simulation** dialog will appear and display the progress of the simulation.
3. During the analysis, periodically view Results just to ensure that the solution is proceeding satisfactorily.
4. When the simulation is complete, PyroSim Results will launch automatically and display a 3D image of the model.

17. View Smoke and HRRPUA

We use PyroSim Results to view the smoke and heat release rate per unit volume.

1. In the PyroSim Results window, double click the **3D Smoke** menu in the tree. This will load both the **HRRPUV** and Soot **Mass Fraction** data sets for all meshes. Select **Wireframe Render**.

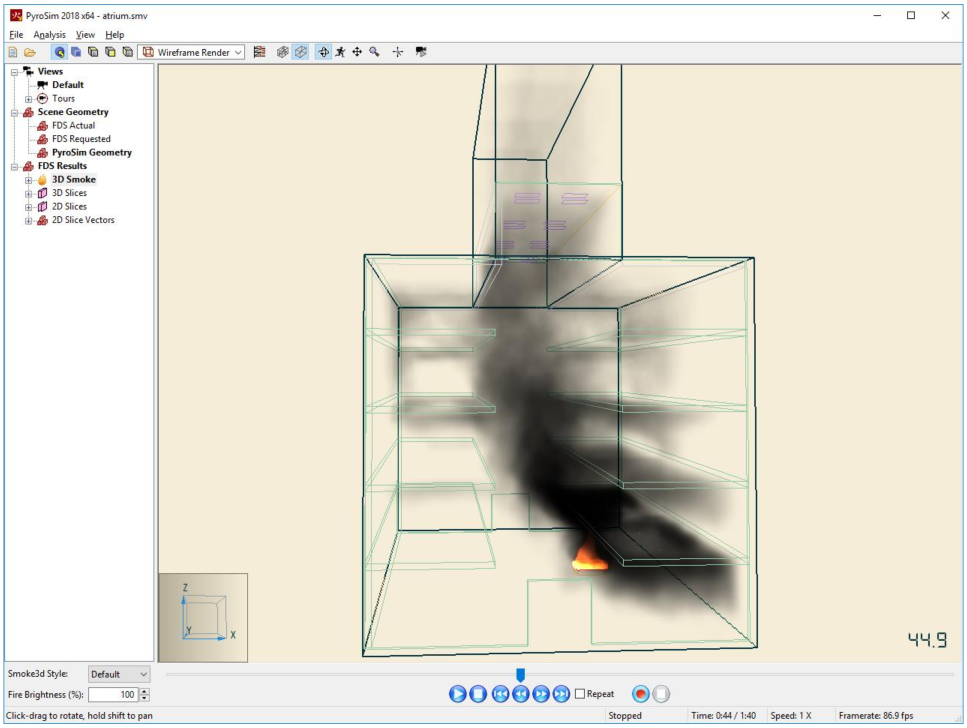

Figure 11: Dense smoke in the atrium at 45 seconds. Extraction fans activated.

In **3D Slices**, double-click **Temperature** to display temperature isosurfaces. Click the colorbar to highlight a value, right-click to turn off highlight. Double-click the colorbar to change plot properties.

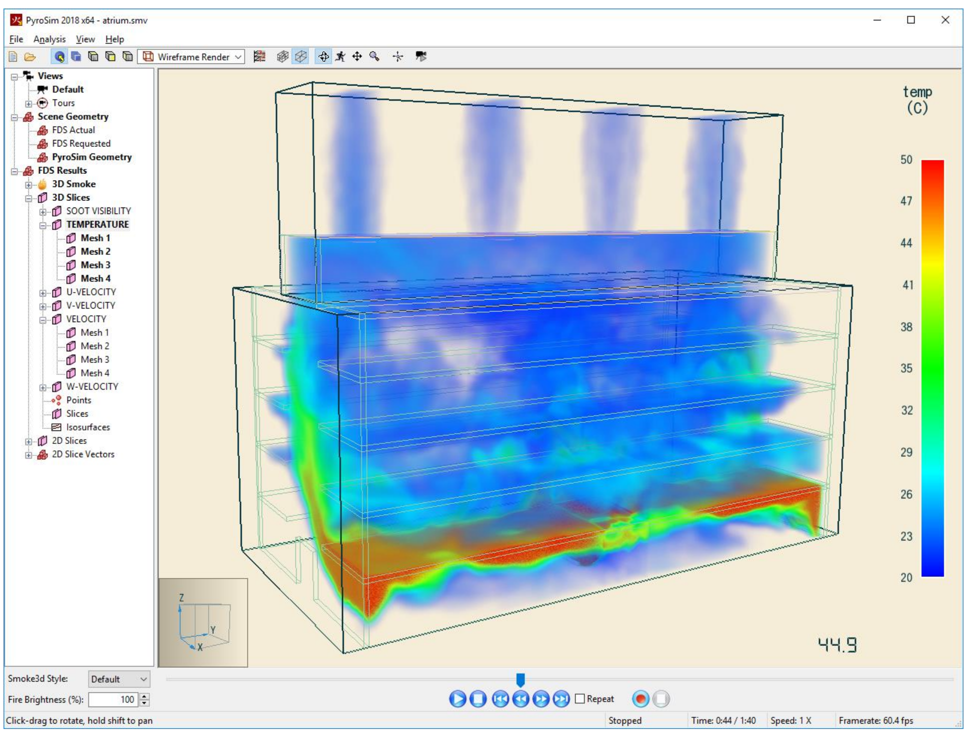

Figure 12: Temperature isosurfaces at 45 seconds.

18. View Slice Data

We can use 2D slice data to view output on planes (you can also define slices through the 3D slice data). We will view the velocity vectors.

1. In Results window, double-click **2D Slice Vectors > VELOCITY**.

At 45 seconds, the evacuation fans are operating and air is entering through the open doors, Figure 13.

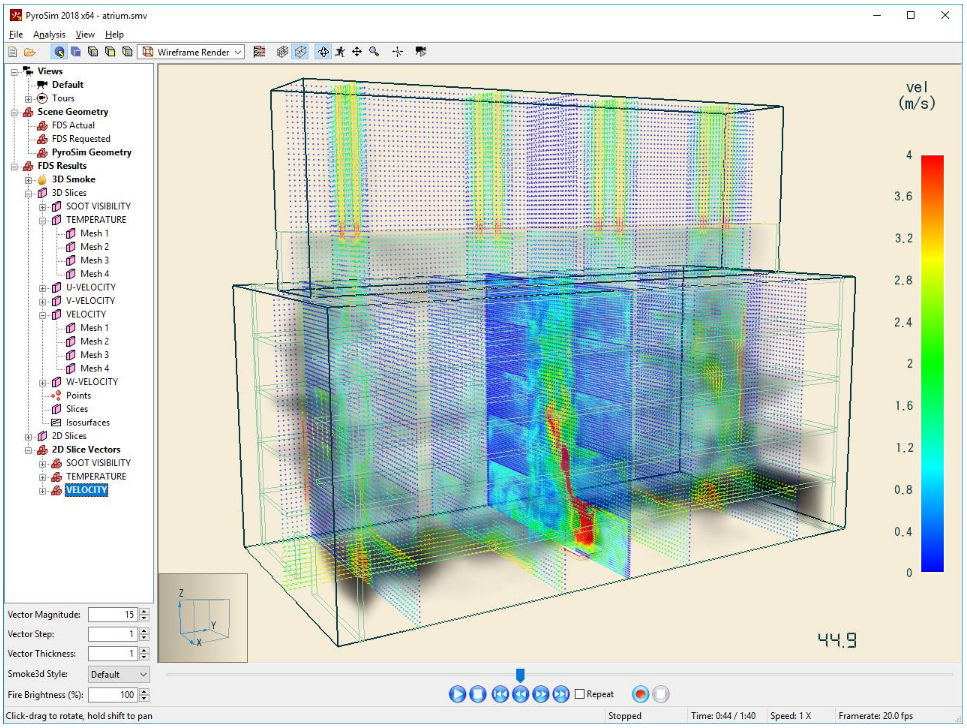

Figure 13: Velocity slice display showing fans extracting air at 45 s.

References

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

McGrattan, Kevin, et al. 2015. *Fire Dynamics Simulator Technical Reference Guide (Sixth Edition).* Gaithersburg, Maryland, USA, October 2015. NIST Special Publication 1018-1.

McGrattan, Kevin, et al. 2015. *Fire Dynamics Simulator User's Guide (Sixth Edition).* Gaithersburg, Maryland, USA, October 2015. NIST Special Publication 1019.