PyroSim

HVAC Room

THUNDERHEAD ENGINEERING

2021

HVAC Room

In this tutorial you will use the Heating Ventilation and Air Conditioning (HVAC) features to model a duct that heats and circulates air in two rooms. We will define a fire in one room to demonstrate how the HVAC network will distribute the smoke throughout the network.

An HVAC system is described as a network of duct segments and nodes. Nodes can either connect two or more ducts (for example, a T connection of three ducts) or be an endpoint where an HVAC duct connects to the FDS computational domain or the infinite ambient void outside of the domain. The HVAC network consists of lines (ducts) and points (nodes). It is only necessary to draw the true geometry at vents where the HVAC system connects to the FDS domain.

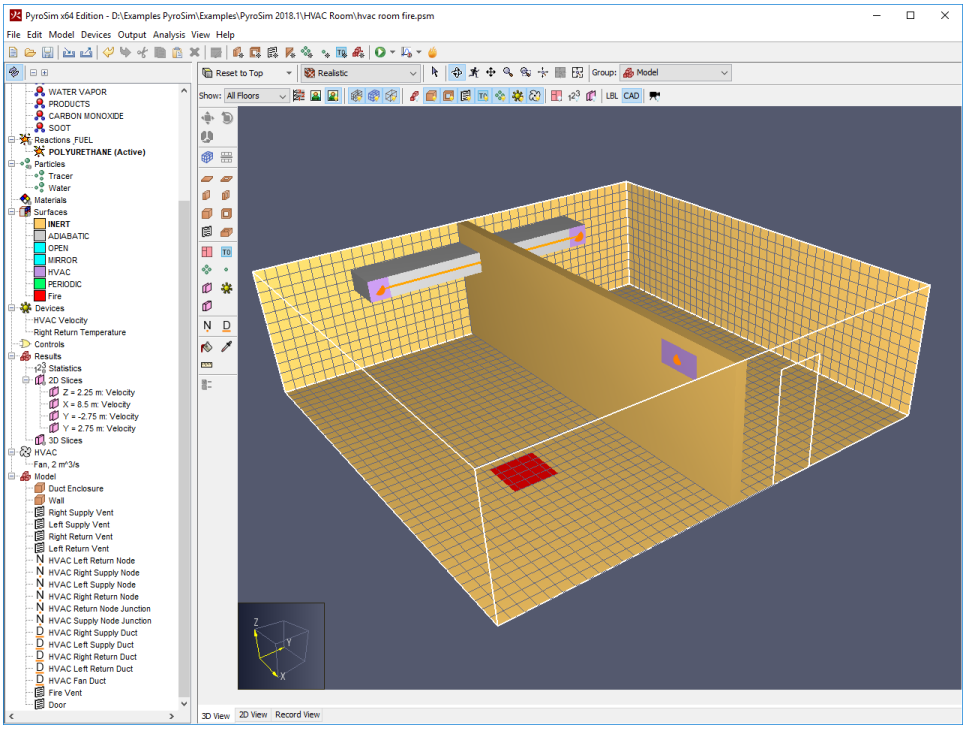

Figure 1. HVAC model.

This tutorial demonstrates how to:

- Create HVAC nodes and ducts.
- Create an HVAC fan.
- Define an HVAC surface on a vent.
- Use devices to plot HVAC parameters.

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.

Start a New Model

To start a new model.

1. On the **File** menu, click **New**.
2. On the **View** menu, click **Units** and select **SI** to display values using the metric system.
3. Save the model. On the **File** menu, click **Save** and choose a folder. Name the file **hvac_room.psm**.
4. Click **OK** to save the model.

Computational Mesh

In this example we will use a 10 m x 10.25 m x 3 m mesh with 0.25 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** and in the **Max X** box, type **10.**
5. In the **Min Y** box, type **-5.125** and in the **Max Y** box, type **5.125.**
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 **40.**
8. In the **Y Cells** box, type **41.**
9. In the **Z Cells** box, type **12.**
10. Click **OK** to save changes and close the **Edit Meshes** dialog.
11. On the toolbar, click **Reset View to All Visible Objects**.

You will notice that in the Y direction, I arranged the mesh so that there are an odd number of cell divisions and made the mesh symmetric about the Y=0 plane. The reason for this is that we will use a wall to divide the mesh into two rooms. FDS "snaps" all geometry to the grid boundaries. I wanted the wall to be 0.25 m thick, so that is the reason for the extra Y cell.

Create the Duct Enclosure

Strictly speaking, we only need to define obstructions at the vent locations where the HVAC network connects to the FDS computational domain. However, in this model, we will represent the entire duct geometry, partly because it makes the final visualization more realistic.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Duct Enclosure**.
3. Click **Specify Color** and then select the color of your choice (I used boring grey).
4. Click the **Geometry** tab.
5. In the **Min X** box, type **0** and in the **Max X** box, type **1.**
6. In the **Min Y** box, type **-3** and in the **Max Y** box, type **3.**
7. In the **Min Z** box, type **2** and in the **Max Z** box, type **2.5.**
8. Click **OK** to create the obstruction.

Create the Wall

A wall will split the mesh into two rooms.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Wall**.
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 **-0.125** and in the **Max Y** box, type **0.125.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **3.**
7. Click **OK** to create the obstruction.

Create Vents

In this model, vents are used at the end of HVAC duct segments to connect to the FDS computational domain. We will place the supply vents on the supply duct and the inlet vents on the wall. To create the supply vents:

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Right Supply Vent**.
3. In the **Surface** list, select **HVAC**.
4. Click on the **Geometry** tab.
5. Select the **X Plane** and type 1**.0** in the box.
6. In the **Min Y** box, type **-3** and in the **Max Y** box, type **-2.5.**
7. In the **Min Z** box, type **2.0** and in the **Max Z** box, type **2.5.**
8. Click **OK** and then click **OK** again to accept the conversion to an HVAC vent.

Now we will copy the **Right Supply Vent** to make the **Left Supply Vent**.

1. In the Navigation View, right-click the **Right Supply Vent** and click **Copy/Move**.
2. Select **Copy**.
3. In the **Offset**, in the **Y** box type **5.5.**
4. Click **OK**.
5. In the **Navigation View**, right-click the copied vent and rename it **Left Supply Vent.**

The return vents will be positioned on the wall. To create the **Right Return Vent**:

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Right Return Vent**.
3. In the **Surface** list, select **HVAC**.
4. Click on the **Geometry** tab.
5. Select the **Y Plane** and type **-0.125** in the box.
6. In the **Min X** box, type **8** and in the **Max X** box, type **9.**
7. In the **Min Z** box, type **2.25** and in the **Max Z** box, type **2.75.**
8. Click **OK** and then click **OK** again to accept the conversion to an HVAC vent.

Now we will copy the **Right Return Vent** to make the **Left Return Vent**.

1. In the Tree View, right-click the **Right Return Vent** and click **Copy/Move**.
2. Select **Copy**.
3. In the **Offset**, in the **Y** box type **0.25**.

4. Click **OK**.
5. In the **Navigation View**, right-click the copied vent and rename it **Left Return Vent.**

The last step is to add HVAC nodes at the vent locations. To do this:

1. In the **Navigation View**, press the **CTRL** key and select the four vents we just made (**Right Supply Vent, Left Supply Vent, Right Return Vent, Left Return Vent)**.
2. Right-click on the selected vents, and click **Add HVAC Nodes**.
3. Rename the numbered HVAC nodes with names that match the vents (for example, **HVAC Right Supply Node**).

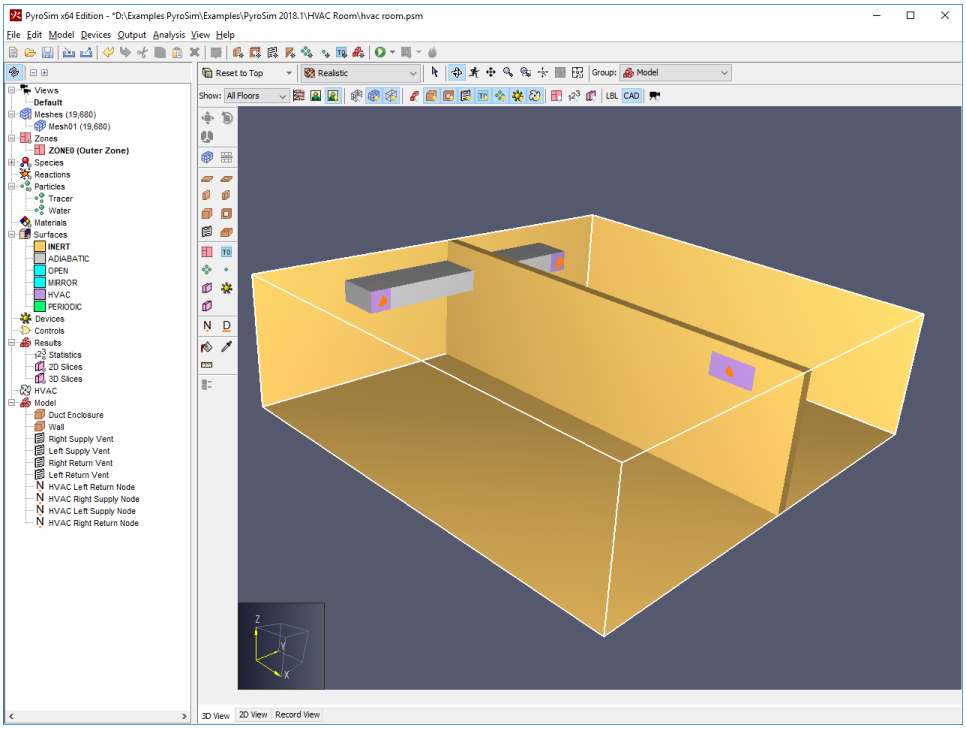

Figure 2. The duct obstruction and supply and return vents

Figure 2 shows the duct obstruction and the supply and return vents. It is worth repeating that vents are required at the ends of the HVAC network to connect the HVAC calculations with the FDS computational domain.

Create HVAC Network

We now create the HVAC network. This is the network of ducts, nodes, and fan that are calculated by a separate module with the results then coupled to the FDS solution. As stated above, this coupling occurs at special HVAC vents.

We now create the HVAC nodes. First, we will add an HVAC node at the junction of the two supply ducts.

1. On the **Model** menu, click **New HVAC Node**.
2. In the **ID**, type **HVAC Supply Node Junction**.
3. Click the **Geometry** tab.
4. In **X** box type **1**, in **Y** box type **0**, and in **Z** box type **2.25**.
5. Click **OK**.

Now add an HVAC node at the junction of the two return ducts.

1. On the **Model** menu, click **New HVAC Node**.
2. In the **ID**, type **HVAC Return Node Junction**.
3. Click the **Geometry** tab.
4. In **X** box type **8.5**, in **Y** box type **0**, and in **Z** box type **2.5**.
5. Click **OK**.

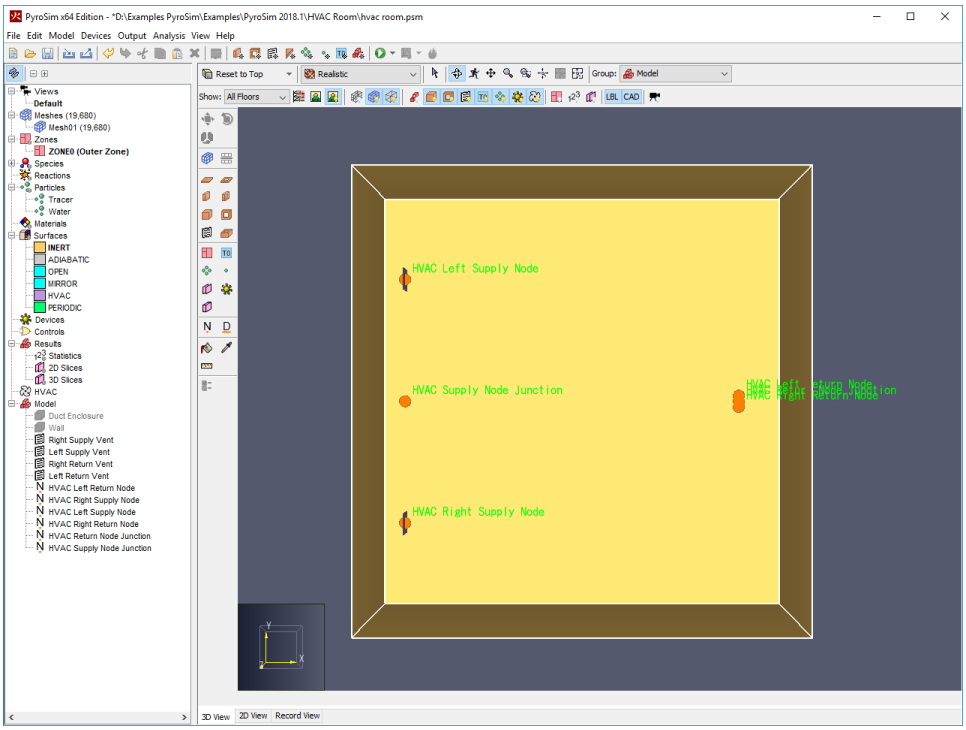

Figure 3: Top view showing the locations of the HVAC nodes.

Now we create the HVAC fan. In this example, we will create one fan and only use it once, but this could be extended to using the same fan in many locations. To create the fan:

1. On the **Model** menu, click **Edit HVAC.**
2. Click **New**, select **Type** as **FAN** and in the **Name** box type **Fan, 2 m^3/s**. Click **OK**.
3. For **Fan Model**, select **Constant Flow** and in the **Volume Flow Rate** box type **2.**
4. Click **OK** to close the **Edit HVAC** dialog.

Now we define the HVAC ducts. These must connect HVAC nodes. First, we make the right supply duct.

1. On the **Model** menu, click **New HVAC Duct**.
2. In the **ID**, type **HVAC Right Supply Duct**.
3. For **Node 1** select **HVAC Supply Node Junction** and for **Node 2** select **HVAC Right Supply Node**.
4. Click **OK** to create the duct.

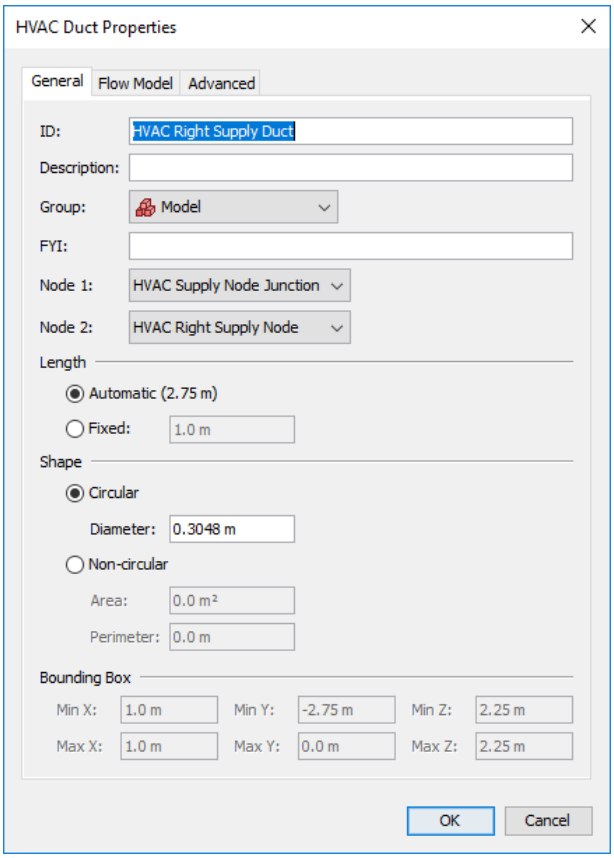

Figure 4: Creating the first HVAC duct.

Repeat for the left supply duct.

1. On the **Model** menu, click **New HVAC Duct**.
2. In the **ID**, type **HVAC Left Supply Duct**.
3. For **Node 1** select **HVAC Supply Node Junction** and for **Node 2** select **HVAC Left Supply Node**.
4. Click **OK** to create the duct.

Repeat this process for both return ducts.

Make the right return duct.

1. On the **Model** menu, click **New HVAC Duct**.
2. In the **ID**, type **HVAC Right Return Duct**.
3. For **Node 1** select **HVAC Return Node Junction** and for **Node 2** select **HVAC Right Return Node**.
4. Click **OK** to create the duct.

Repeat for the left return duct.

1. On the **Model** menu, click **New HVAC Duct**.
2. In the **ID**, type **HVAC Left Return Duct**.
3. For **Node 1** select **HVAC Return Node Junction** and for **Node 2** select **HVAC Left Return Node**.
4. Click **OK** to create the duct.

There are now four HVAC ducts in the model for both Supply and Return sides of the network.

Finally, we define the duct that connects the supply and return center nodes.

1. On the **Model** menu, click **New HVAC Duct**.
2. In the **ID**, type **HVAC Fan Duct**.
3. For **Node 1** select **HVAC Return Node Junction** and for **Node 2** select **HVAC Supply Node Junction**.
4. Click the **Flow Model** tab and select the **Flow Device** as **Fan** (not **Basic Fan**). In the **Fan** pulldown menu, select the previously created **Fan, 2 m^3/s**.
5. The **Flow Direction** is from **Node 1 to Node 2**.
6. Click **OK.**

Note that for this fan that we created, there is the option to use controls to activate and deactivate.

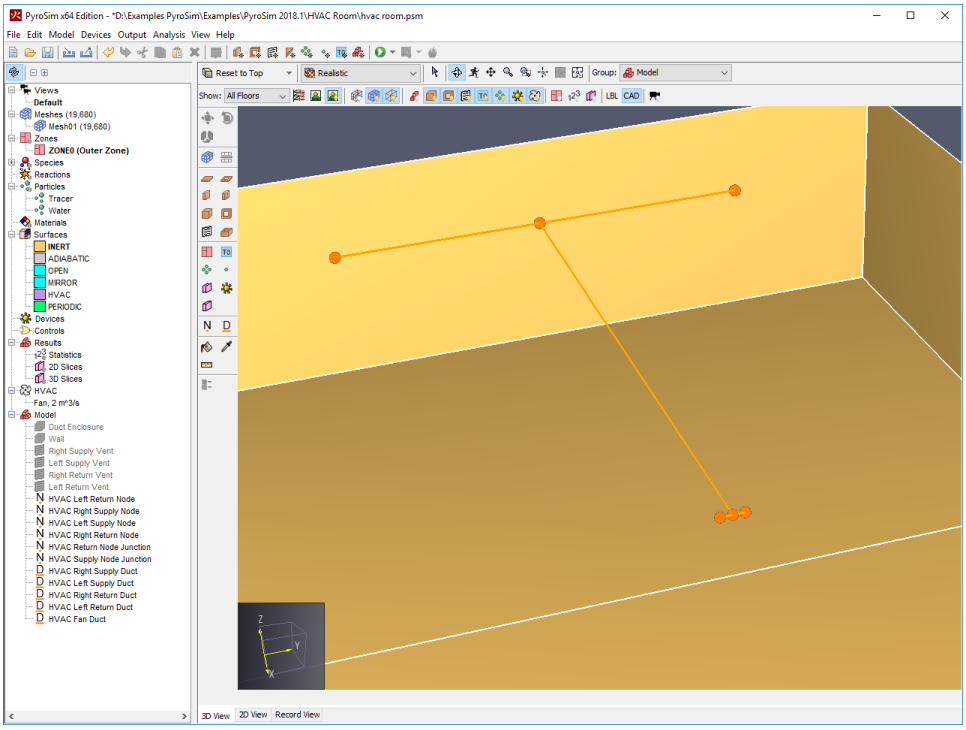

Figure 5: The HVAC network (labels have been turned off).

HVAC Devices

We use devices to plot HVAC quantities. Both duct and node quantities can be plotted, we will illustrate with one of each. First a device to plot the duct velocity.

1. On the **Devices** menu, click **New HVAC Duct Device**.
2. In the **Name** box type **HVAC Velocity** and in the **Quantity** menu select **[HVAC Quantity]**. This will open a **Choose Quantity** dialog.
3. For the **Quantity** select **Velocity of a Duct** and in the **HVAC Duct** select **HVAC Fan Duct**.
4. Click **OK** to close the **Choose Quantity** dialog and then OK again to close the **HVAC Device** dialog.

Now a device to plot the return air temperature.

1. On the **Devices** menu, click **New HVAC Node Device**.
2. In the **Name** box type **Right Return Temperature** and in the **Quantity** menu select **[HVAC Quantity]**. This will open a **Choose Quantity** dialog.
3. For the **Quantity** select **Temperature of the Flow through a Node** and in the **HVAC Node** select **HVAC Right Return Node**.
4. Click **OK** to close the **Choose Quantity** dialog and then **OK** again to close the **HVAC Device** dialog.

Slice Planes

Slice planes can be used to display 2D contours in the PyroSim Results display. In this analysis, we will save velocity 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 1. 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.

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

Table 1: Slice plane data

XYZ Plane	Plane Value (m)	Gas Phase Quantity	Use Vector?	Cell Centered?
Z	2.25	**Velocity**	**YES**	**NO**
X	8.5	**Velocity**	**YES**	**NO**
Y	-2.75	**Velocity**	**YES**	**NO**
Y	2.75	**Velocity**	**YES**	**NO**

Specify Simulation Properties

1. On the **Analysis** menu, click **Simulation Parameters**.
2. In the **End Time** box, type **20.**
3. Click **OK**.

Save and Run the Simulation

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

View Slice Data

1. In the tree click to expand **2D Slices > VELOCITY**.
2. Double click **Y = -2.625** to show the velocity slice at that location.
3. Double click **Y = 2.875** to show the velocity slice at that location.
4. In the tree click to expand **2D Slice Vectors > VELOCITY**.
5. Double click **Z = 2.250** to show the vector slice at that location.

The plot of the velocity contours is shown in Figure 6. Notice the flow into the return vents.

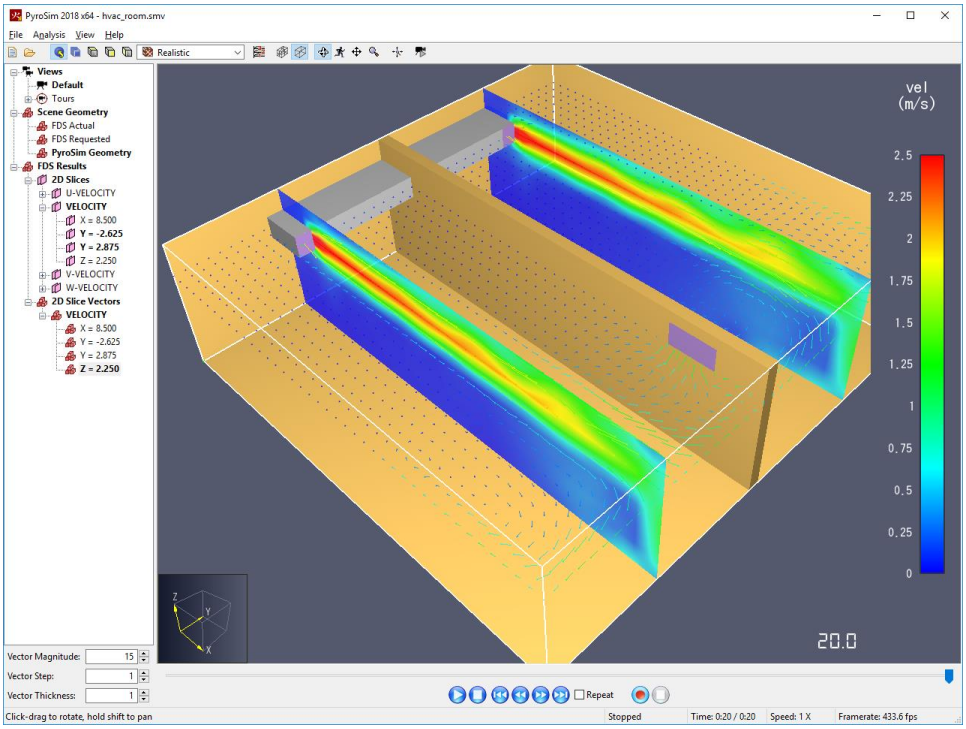

Figure 6: Velocities in model, plotted using a combination of slice and vector slice planes.

On the toolbar, select **Plot Device Results.** The duct diameter is 0.3048 m, so the expected velocity of flow in the duct for a volumetric flow rate of 2 m^3/s is 27.4 m/s.

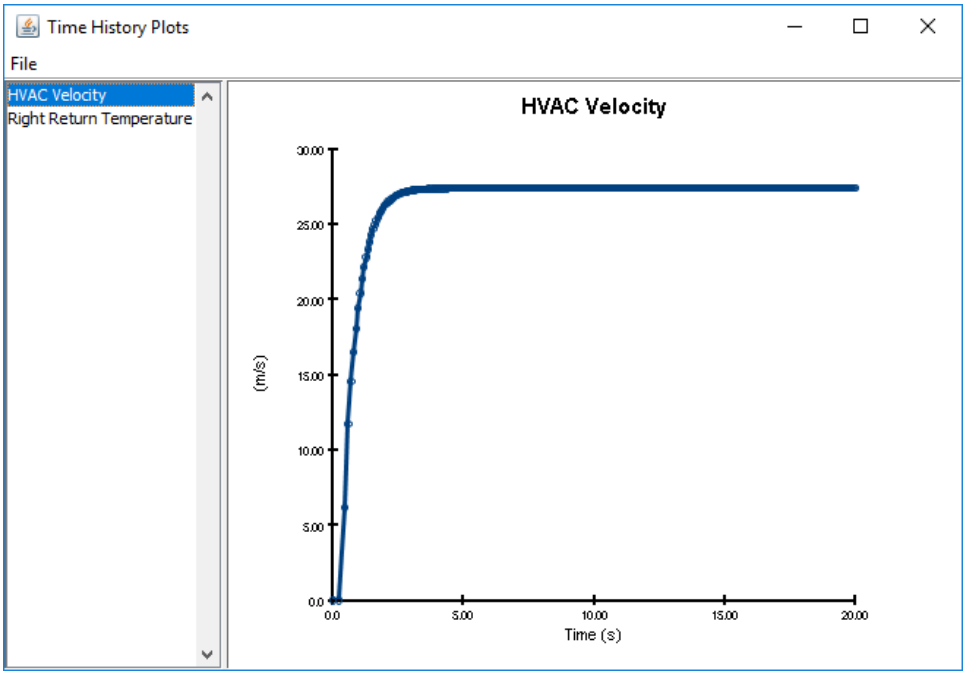

Figure 7: Velocity of flow in duct.

Add Fire and Smoke to the Simulation

To demonstrate how the HVAC network can distribute smoke throughout the model, we will add a fire in one room and watch it appear in the second room.

Save the Fire Model

To save the fire model:

1. On the **File** menu, click **Save As**.
2. Name the file **hvac_room_fire.psm**.
3. Click **OK** to save the model.

Reaction

Define the reaction for this simulation.

1. On the **Model** menu, click **Edit Reactions**.
2. Click the **Add From Library** button, select the **POLYURETHANE** reaction, and move it to the **Current Model**. Click **Close**.
3. Click **OK** to close the dialog.

Create the Fire Surface

In this example, we define a burner (fire) surface that releases heat at a rate of 500 kW/m^2.

1. On the **Model** menu, click **Edit Surfaces.**
2. Click **New.**
3. In the **Surface Name** box, type **Fire**.
4. In the **Surface Type** list, select **Burner**.
5. Click **OK** to create the burner surface.
6. In the **Heat Release** tab of the **Fire** surface, select **Heat Release Rate Per Area (HRRPUA)** and in the box type **500**.
7. Click **OK** to save changes and close the **Edit Surfaces** dialog.

Create the Fire

Now we define the fire location in our model.

1. On the **Model** menu, click **New Vent.**
2. In the **ID** box, type **Fire Vent**.
3. In the **Surface** list, select **Fire**.
4. Click on the **Geometry** tab. In the **Plane** list, select **Z** and in the location box type **0.0**.
5. In the **Min X** box, type **6** and in the **Max X** box, type **7**.
6. In the **Min Y** box, type **–3** and in the **Max Y** box, type **-2.**
7. Click **OK** to create the new burner vent.

Add a Vent to Represent an Open Door

Now we will define an open vent that will represent an open door. This will provide both a fresh air supply for the fire and an opening that will allow the heated air a place to vent to atmosphere. If we did not do this, the average air pressure in the rooms would increase as the air is heated by the fire.

1. On the **Model** menu, click **New Vent.**
2. In the **ID** box, type **Door**.
3. In the **Surface** list, select **OPEN**. This is a special surface always available in FDS.
4. Click on the **Geometry** tab. In the **Plane** list, select **X** and in the location box type **10**.
5. In the **Min Y** box, type **1** and in the **Max Y** box, type **2**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **2.5.**
7. Click **OK** to create the open door.

Save and Run the Simulation

1. On the **File** menu, click **Save.**
2. On the **FDS** menu, click **Run FDS**.
3. Click **Ignore** to dismiss the warning about the model not being connected to an ambient pressure boundary condition. Sealed compartments typically indicate a modeling oversight, but for this demonstration it is safe to ignore the warning.
4. The **FDS Simulation** dialog will appear and display the progress of the simulation.
5. When the simulation is complete, PyroSim Results will launch automatically and display a 3D image of the model.

View Smoke and Fire

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

The plot of the smoke is shown in Figure 8.

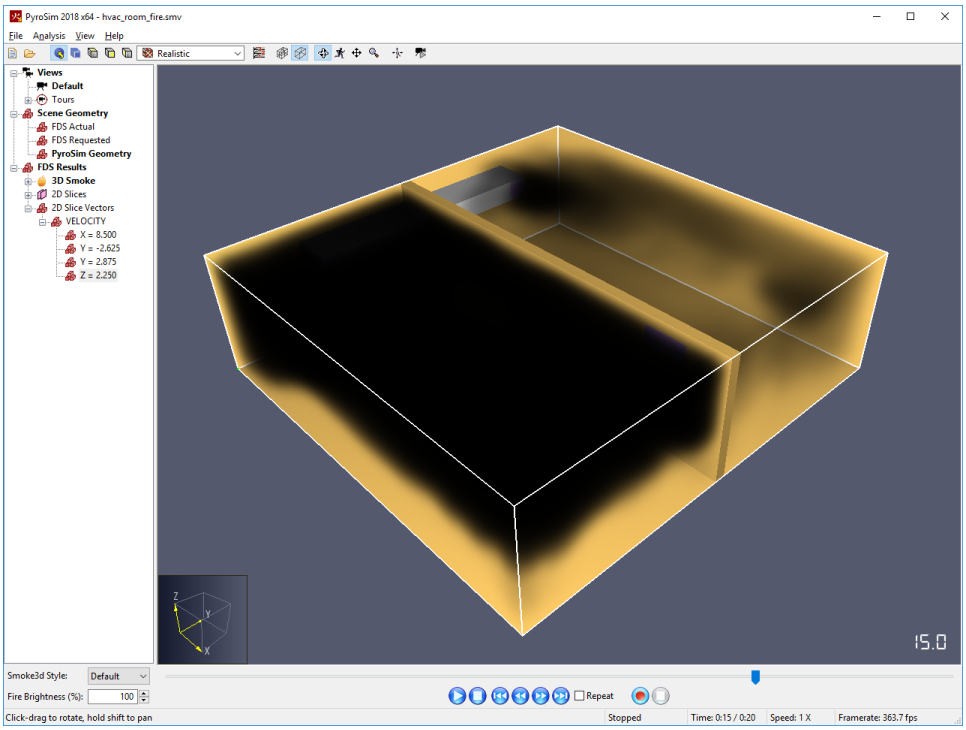

Figure 8: Smoke distribution in the model. The smoke in the room with a fire is drawn into the return vent and then mixes and distributed to the second room.

On the toolbar, select **Plot Device Results** and then select **Right Return T**, Figure 9.

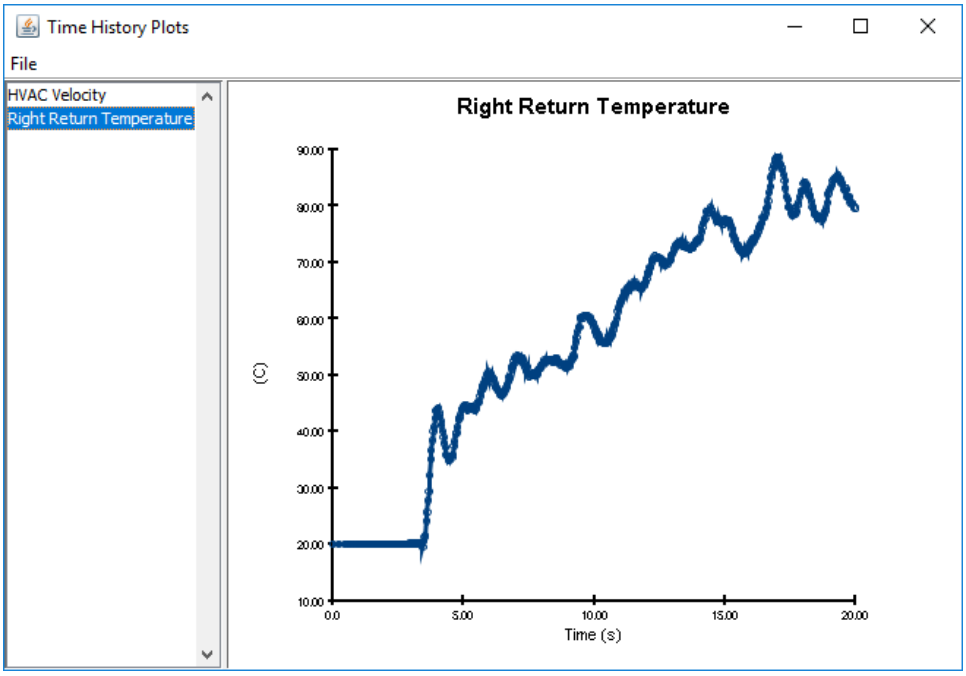

Figure 9: Temperature of air in return duct in room with fire.

References

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

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

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