PyroSim

Air Movement

Air Movement

In this tutorial you will create an air flow using a supply vent on one side of a room and an open vent on the opposite side. This is a very simple PyroSim/FDS simulation, but illustrates several important points:

- By default, the boundaries of a model are **closed** to flow, with an **inert** surface for thermal and radiation effects. An inert surface remains at a constant temperature (usually the ambient temperature), so an inert surface can heat or cool the air by convection and radiation.
- A **vent** as used in FDS is more than just a supply or exhaust vent, instead, a vent is a general way to apply 2D boundary conditions to a surfaces. In this example, we use one vent to supply air and a second vent to be an **open** boundary to ambient conditions.
- Since the model boundaries are closed by default, if we supply air into the model there must be a way for the air to exhaust. The recommended approach is to use an **open** boundary for the exhaust that provides an ambient boundary condition. FDS provides a special vent type **OPEN** to accomplish this.

  An open boundary is recommended because the density of the air can change due to heating/cooling, with the consequence that specifying velocities on both the supply and exhaust will cause mass to accumulate (or decrease) in the model. Similarly, specifying mass flow rate boundary conditions will not work if a fire is introducing fuel into the model.
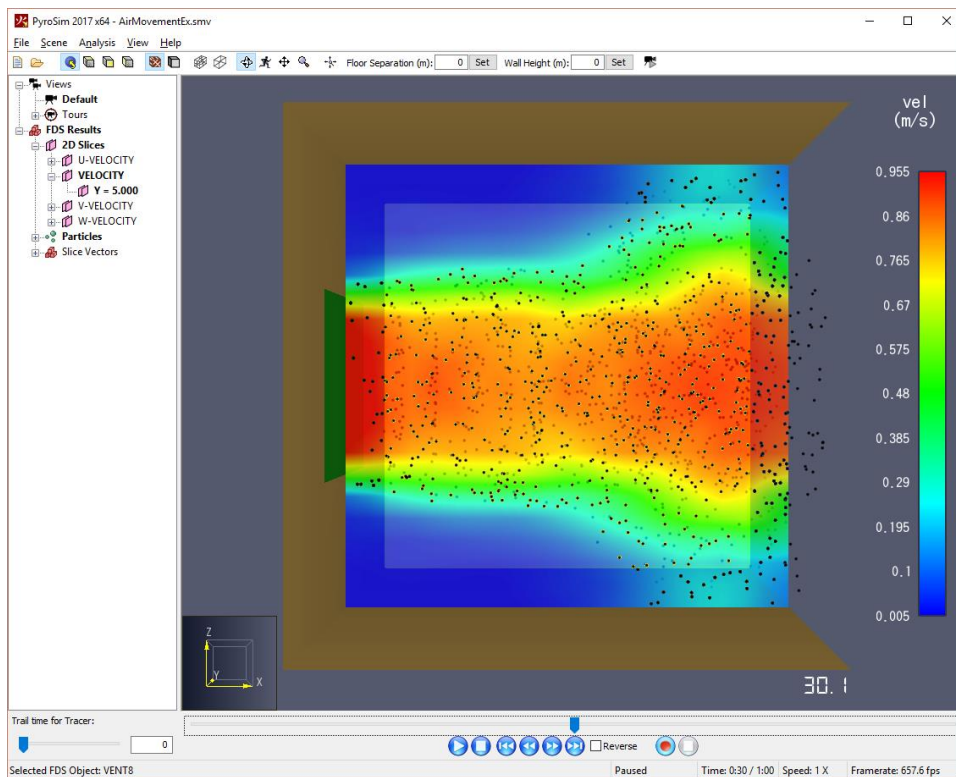

Figure 1. 3D visualization of air flow in this example

This tutorial demonstrates how to:

- Create vents.
- Add slice planes and particles for velocity visualization.
- View 3D results.

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

Select SI Units

To select SI units:

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

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** and in the **Max X** box, type **10.**
5. In the **Min Y** box, type **0** and in the **Max Y** box, type **10.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **10.**
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.
11. In the upper toolbar, click the **Reset View to All Visible Objects** button.
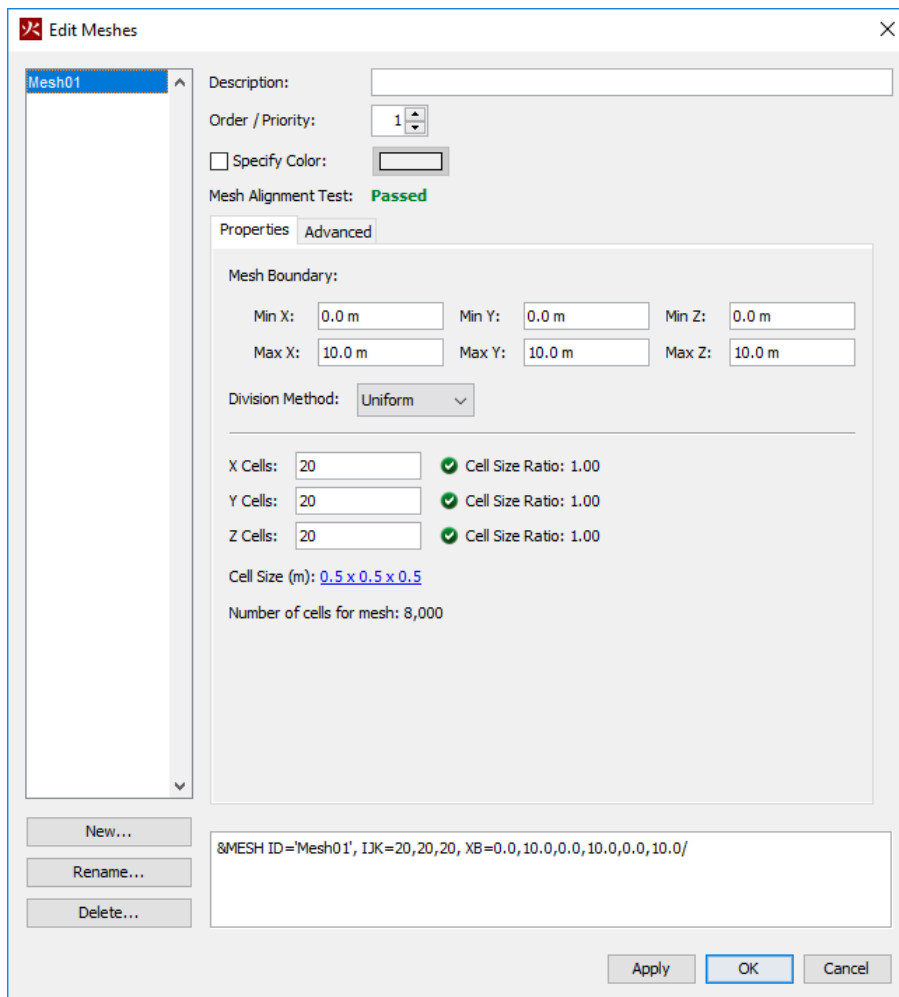
Figure 2. Creating the mesh

Create the Supply Surface

Surfaces are used to define the properties of objects in your FDS model. Surfaces can be assigned to both obstructions and to vents in the 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 **Air Supply**.
4. In the **Surface Type** list, select **Supply**.
5. Click **OK** to create the new supply surface. This will open the **Edit Surfaces** dialog showing the new surface.
6. Select the **Specify Velocity** option and in the box, type **1**. This means that air will flow from this surface with a velocity of 1 m/s.
7. To emit massless particles to show the flow, click the **Particle Injection** tab.
8. Click **Emit Particles**.

9. The default **Particle Type** will be **Tracer**.
10. In the **Insertion Interval** box, type **1**.
11. Click **OK**.

Create Vents

The surface we just created defines a set of properties, but is not yet attached it to any feature of the model. One way to define flow conditions in a model is using a "vent." Vents are 2D objects and must be aligned with one of the model planes. To create an air supply vent:

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Inlet**.
3. In the **Surface** list, select **Air Supply**. 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.**
5. In the **Min Y** box, type **3** and in the **Max Y** box, type **7.**
6. In the **Min Z** box, type **3** and in the **Max Z** box, type **7.**
7. Click **OK**.

By default, FDS assumes that the exterior boundary of the mesh is a solid wall with the temperature fixed at ambient. This is referred to as 'INERT'. So we must provide a way for the supply air to exit the model. To do this, we will create a vent that is open to atmospheric conditions. In general, it is better to have an open vent where the flow will be calculated, rather than specifically defining the exhaust flow rate using a surface as we did for the air supply. One example of why an open vent works best is that if a fire heats the air in the room, to maintain the inlet mass flow equal to the exhaust, the velocity of the less dense exit flow will be larger for the same flow areas.

To create an open vent that covers the entire right face of the model:

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Exhaust**.
3. In the **Surface** list, select **OPEN**.
4. Click on the **Geometry** tab. In the **Plane** list, select **X** and type **10.**
5. In the **Min Y** box, type **0** and in the **Max Y** box, type **10.**
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **10.**
7. Click **OK**.
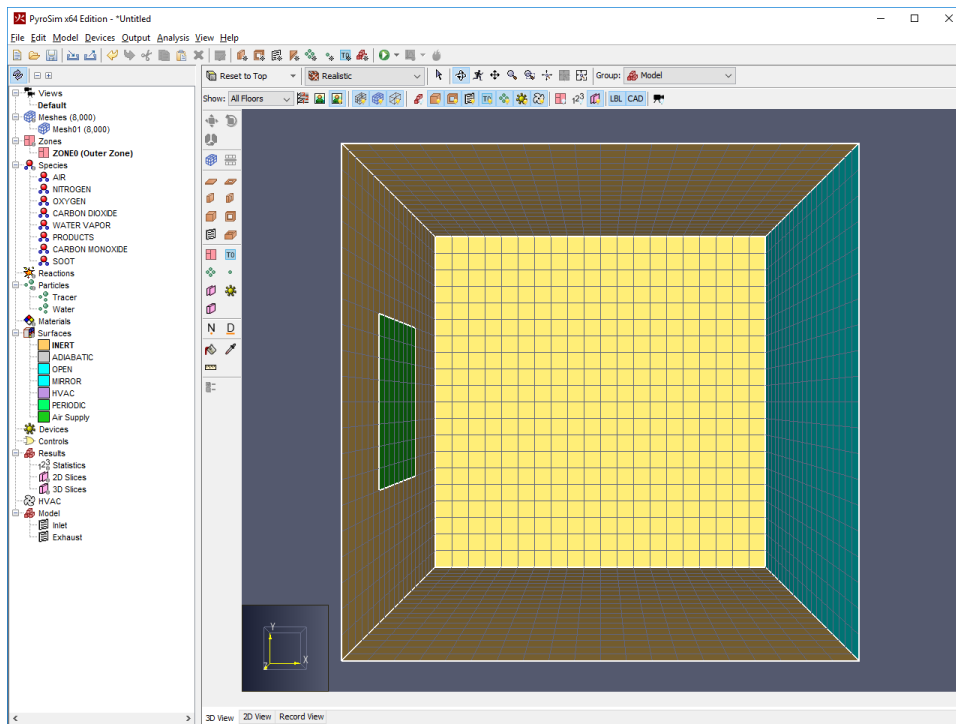
Figure 3. The resulting model. Here we have selected the Show Mesh Boundaries tool and clicked the Reset to Top View button.

Create Slice Records

Slices define planes used to display contours or vectors of the results. A new slice must be defined for each quantity that will be plotted. To make a velocity vector slice:

1. On the **Output** menu, click **2D Slices**.
2. In the **XYZ** column, select **Y** (the slice will be normal to the Y coordinate).
3. In the **Plane Value** column, type **5.**
4. In the **Gas Phase Quantity** column, select **Velocity**.
5. In the **Use Vector?** column, select **YES**.
6. In the **Cell Centered?** column, select **NO**.
7. Click **OK** to create the slice plane.

You can toggle the display of slices on or off using the **Show Slices** button.

Specify Simulation Properties

To define the end time:

1. On the **Analysis** menu, click **Simulation Parameters.**
2. In the **End Time** box, type **60.**
3. Click **OK**.

Save the Model

1. On the **File** menu, click **Save**.
2. Choose a location to save the model. Use folders to organize your models. When you run the analysis, PyroSim will create a new sub-folder with the model name and write all the input and output files to this folder. For this example, make a folder named **Air Movement** and name the file **air movement.psm**.
3. Click **Save** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS**.
2. The **FDS Simulation** dialog will appear and display the progress of the simulation. This should take approximately 90 seconds to run, depending on computing hardware.
3. When the simulation is complete, PyroSim Results will launch automatically and display a 3D image of the model.

View Velocity Slice Data

We will first display the velocity contours:

1. In the PyroSim Results window, expand the 2D Slices in the tree.
2. Double click **VELOCITY** in the tree to display the velocity slice at Y=5.0.
3. Spin the model to view the slice (or click the **Front View** button).
4. Click the **Play** button.

View Particles

We can also display the particles:

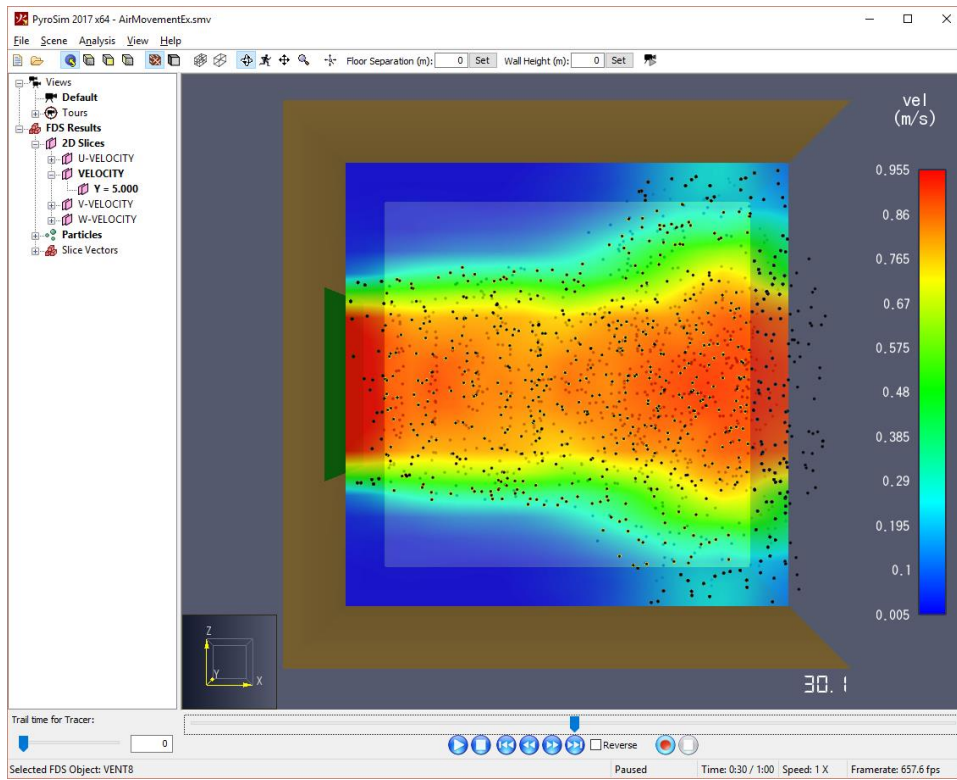1. In the PyroSim Results window, double click **Particles** to display the particle data.

Figure 4. Velocity contours and particles displayed in 3D Results

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.