



Import a CAD Model

Import CAD Model

In this tutorial you will import a CAD file, then add a *500 kW* burner fire.

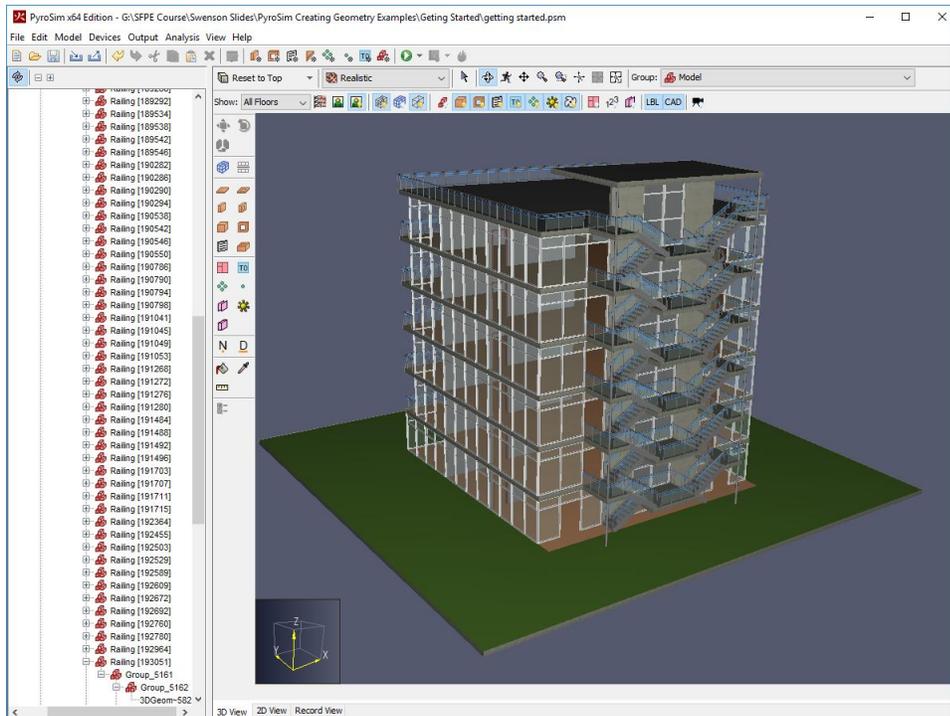


Figure 1. Burner fire in this example

This tutorial demonstrates how to:

- Import a CAD file.
- Add a fire.
- Mesh only part of the model.
- View 3D results using PyroSim Results.
- View 2D results using PyroSim.

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.

Import CAD Model

To import the CAD model:

1. First, download the pyrosim_examples.zip and extract the fbx folder. This contains the CAD model.
2. On the PyroSim **File** menu, click **Import FDS/CAD File** and select the **GettingStartedModel V2 - SimLab.fbx** file.
3. All default selections are on the import dialog are valid, so click **Next** on each pane and then click **Finished**.
4. Spin the model to view and explore.

Correct Thin Geometry

To prevent FDS errors caused by thin geometry in the CAD model:

1. On the **Analysis** menu, click **Simulation Parameters**.
2. Click the **Angled Geometry** tab.
3. Check the **Rasterize axis-aligned** box.
4. Change **Thickening** to **Force all obstructions to be thickened**.
5. Click **OK** to save changes and close the **Simulation Parameters** dialog.

Create a New View that Clips to Display Only the Ground Floor

Let us add a new view that clips to the ground floor:

1. On the **View** menu, click **New View**. This will create a new view named **View01**.
2. In the **Navigation View**, right-click **View01** and **Rename** it to **Ground Floor**.
3. In the **Navigation View**, right-click **Ground Floor** and click **Add Section Box**.
4. In the **Navigation View**, double-click the **Section Box** and in the **Max Z** box, type **2.7**. Click **OK**.
5. Click **Reset**.
6. Right-click **Ground Floor** view and click **Save Viewpoint** to update the viewing direction.

We will place a 0.4x0.4 m fire at the location shown in Figure 2.

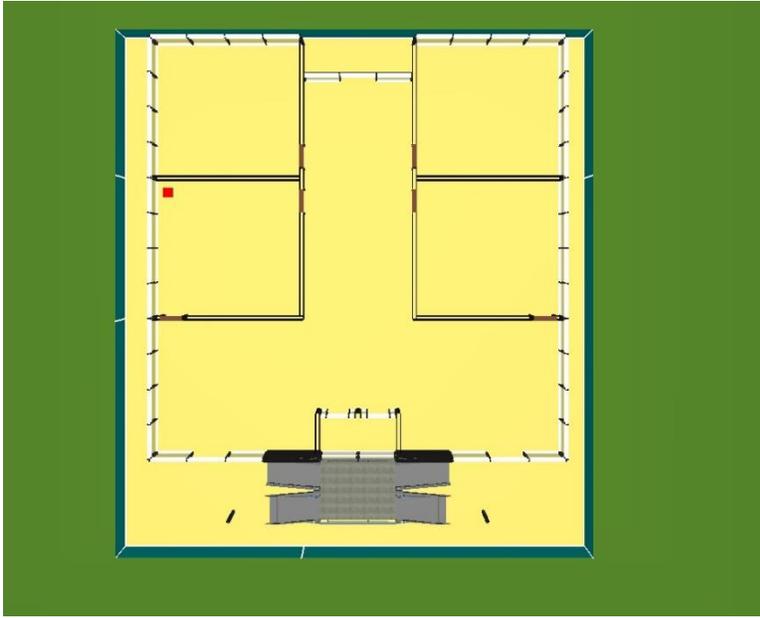


Figure 2: Location of fire on ground floor.

Create the Mesh

In this example we will use mesh cells that are 0.2 m across. This value is too large to properly simulate the fire, so we will locally refine the mesh around the fire.

1. On the **Model** menu, click **Edit Meshes**.
2. Click **New**.
3. Click **OK** to create the mesh.
4. In the **Min X** box, type **-20** and in the **Max X** box, type **-1.0**.
5. In the **Min Y** box, type **-6.0** and in the **Max Y** box, type **15.0**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **3**.
7. In the **X Cells** box, type **95**.
8. In the **Y Cells** box, type **105**.
9. In the **Z Cells** box, type **15**.
10. Click **OK** to save changes and close the **Edit Meshes** dialog.

In the **Navigation View**, double-click the **Default** view. Click **Show Mesh Boundaries** and **Show Mesh Outlines** to turn on the display of the mesh. The mesh we created bounds the ground floor extends past the exterior stairs, Figure 3.

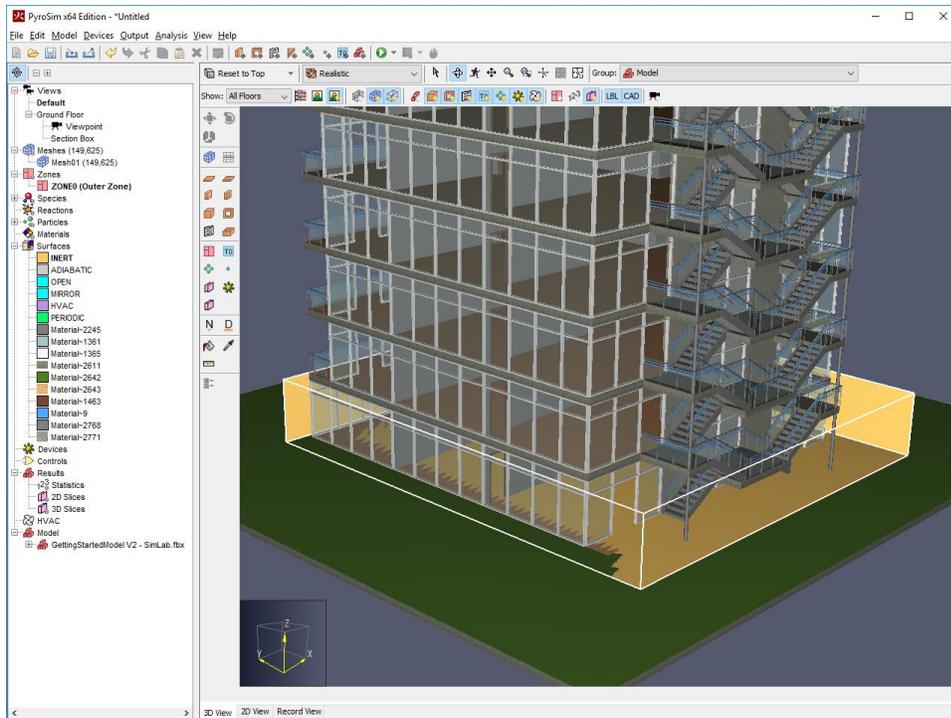


Figure 3: Display of mesh.

Split and Refine the Mesh

We will now split the mesh into multiple meshes and refine the mesh in the room with the fire.

1. In the **Navigation View**, under **Model** right-click **GettingStartedModel V2 - SimLab.fbx** and then click **Hide Objects**. This will hide the display of the geometry.
2. In the **Navigation View**, right-click **Mesh01** and click **Split Mesh**.
3. In the **Split Near** options select **X** and in the box, type **-12.6**. Click **OK**.
4. In the **Navigation View**, right-click **Mesh01** and click **Split Mesh**.
5. In the **Split Near** options select **Y** and in the box, type **3.4**. Click **OK**.
6. In the **Navigation View**, right-click **Mesh01** and click **Split Mesh**.
7. In the **Split Near** options select **Y** and in the box, type **9.2**. Click **OK**.

Splitting the meshes forms a new hierarchy of meshes as sub-groups of the original **Mesh01**. We will reorganize this, but first we will refine the mesh near the fire and merge the other meshes.

1. Click **Reset**.
2. On the upper toolbar, click **Show Mesh Divisions**.
3. Hold the **g** key and right-click the left, center mesh (Mesh01-a-b-a). Click **Refine Mesh**.
4. In the **Refine Factor** box, type **2**. Click **OK**.
5. Holding the **g** key, right-click the mesh you just refined. Click **Split Mesh**.

6. In the **Split Near** options select **X** and in the box, type **-16.2**. Click **OK**. Note that we split the refined mesh on a X coordinate that exactly matches the mesh divisions in the coarse meshes adjacent to the refined mesh.
7. Hold the **g** key and the **CTRL** key and click the top two meshes. Still holding the **g** and **CTRL** keys, right-click and click **Merge Meshes**. This will merge the two meshes.
8. Hold the **g** key and the **CTRL** key and click the two remaining meshes on the right. Still holding the **g** and **CTRL** keys, right-click and click **Merge Meshes**. This will merge the two meshes.

There are now five meshes in the model. Within the constraints of the geometry, we have made the number of cells in each mesh approximately equal so that when we solve the problem, each core will have a similar load. Figure 4 shows the meshes and the renamed and rearranged tree locations.

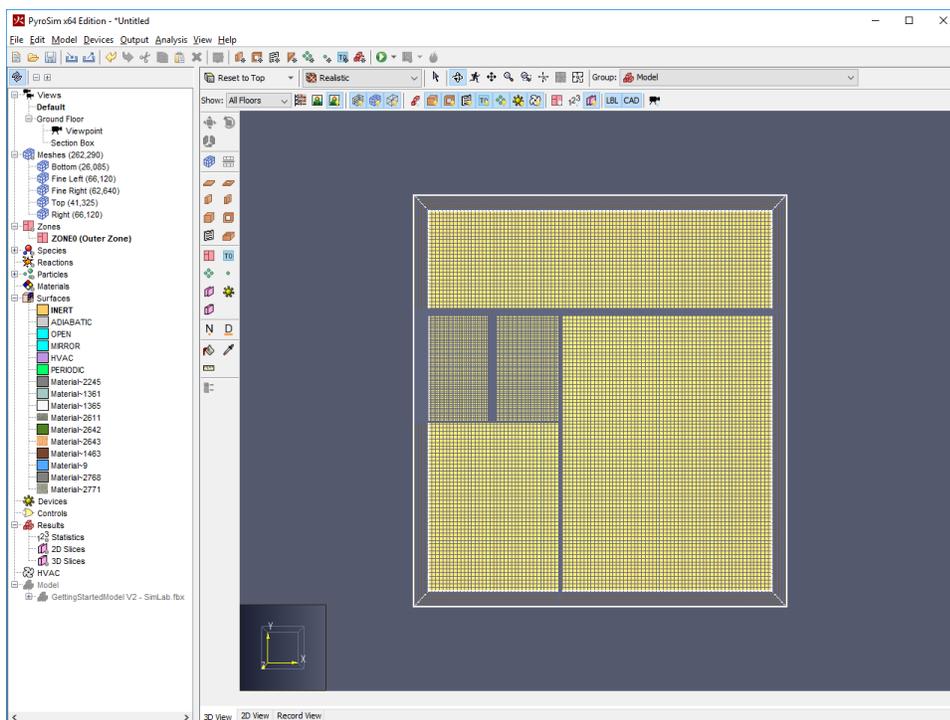


Figure 4: Meshes after splitting, refinement, and merging.

Define the Reaction

For simulations that include combustion, the user must define a reaction.

1. On the **Model** menu, click **Edit Reactions**.
2. Click **Add From Library**.
3. Select the **WOOD_OAK** reaction and add it to the current model.
4. Click **Close**.
5. Click **OK** to close the **Edit Reactions** dialog.

Create the Fire Surface

Surfaces are used to define the properties of objects in your FDS model. In this example, we define a burner (fire) surface that releases heat at a rate of 500 kW.

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 new burner surface.
6. In the **Heat Release Rate Per Area** box, type **3125.0 kW/m²**. The area of the fire will be 0.16 m², so the total Heat Release Rate will be 500 kW.
7. On the **Thermal** tab, change the **Surface Temperature** to **500**.
8. Click **OK** to save changes and close the **Edit Surfaces** dialog.

Create the Fire

Now that we define the fire location in our model. We create an obstruction and assign the fire surface to the top of the obstruction.

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Fire Obstruction**.
3. Click the **Geometry** tab.
4. In the **Min X** box, type **-18.4** and in the **Max X** box, type **-18**.
5. In the **Min Y** box, type **8.4** and in the **Max Y** box, type **8.8**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **.4**.
7. Click the **Surfaces** tab.
8. Click **Multiple**.
9. For the **Max Z** surface, select **Fire**.
10. Click **OK** to create the new obstruction.

Note that we have made sure that the geometry of the obstruction corresponds to the cell size of 0.1 m. This removes any uncertainty as to what dimensions the obstruction will snap to. The top surface area is 0.16 m², so the total heat release rate will be 500 kW.

Open Sides and Top of the Mesh

We want the sides and top of the mesh to be open boundaries. This is accomplished by creating open surfaces on these boundaries. PyroSim can automatically do this for all the boundaries of a mesh.

1. In the **Navigation View**, right-click **Meshes** and click **Open Mesh Boundaries**. This will create open vents on the boundary sides of each mesh.
2. Since we want the bottom to be closed, expand the vents for each mesh and delete the **[ZMIN]** vent.

Add 2D Slice Planes (Contour planes)

1. On the **Output** menu, click **2D Slices**.
2. Enter slice planes as shown in Figure 5.
3. Click **OK**.
4. You can click the **Show Slices** button to toggle the slice display on and off.

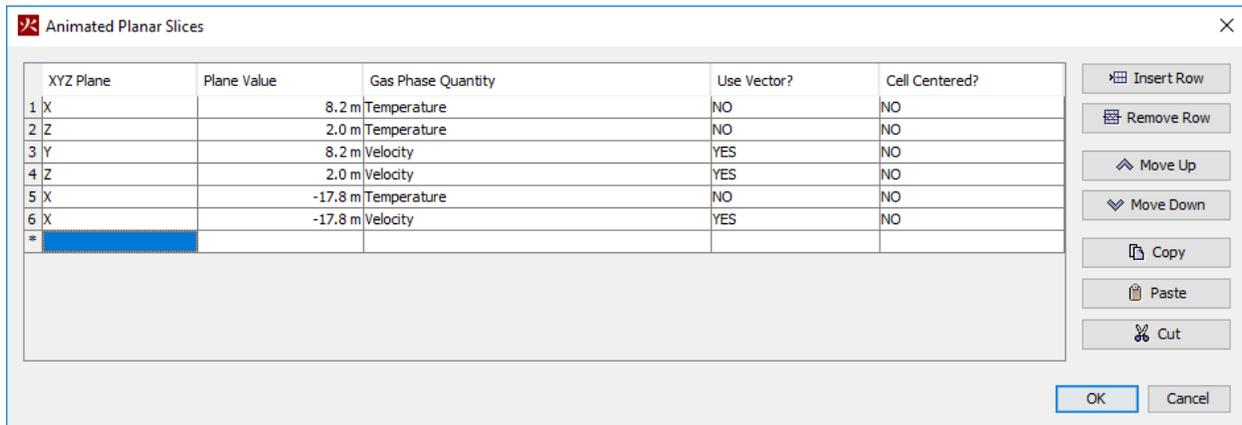


Figure 5: Defining the slice plane plots.

Add 3D Slices (Isosurfaces)

1. On the **Output** menu, click **3D Slices**.
2. Enter slice planes as shown in Figure 6.
3. Click **OK**.
4. You can click the **Show Slices** button to toggle the slice display on and off.

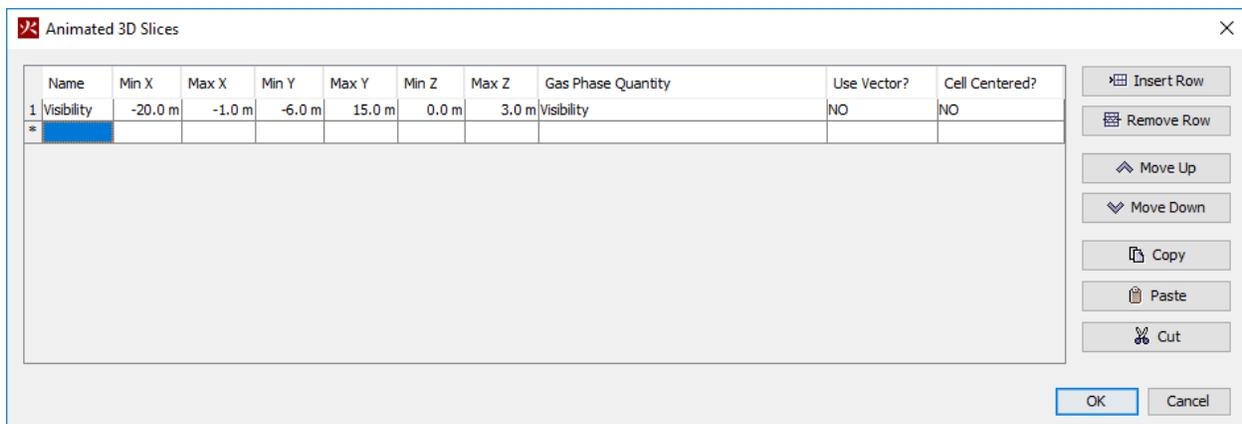


Figure 6: 3D visibility slice

Solution Time

Set the solution end time.

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

2. In the **End Time** box, type **100**.
3. Click **OK**.

Make Additional Views for Results Viewing

The best way to define interior views is using the **Roam** tool. To practice:

1. In the Navigation View, right-click **Model** and click **Show All**.
2. Click **Reset**.
3. Click **Roam**.
4. Hold the **ALT** key while dragging a mouse button to move the camera up and down along the Z axis.
5. Hold the **CTRL** key while dragging a mouse button to move the camera forward, backward, and side-to-side in the camera's XY plane.
6. Release all keys and click and rotate with the mouse to change viewing direction. Now the **CTRL** key movements will be relative to the new viewing direction. Also, after you have moved the camera to a new viewpoint, you save that viewpoint by right-clicking on the view and the click **Save Viewpoint**.

After you have practiced with the **Roam** tool, use it to define four views: **Exterior** that looks at the building from a distance, **Entry** that is just outside the building at the entry door, **Hallway** that has moved down the hallway to the door of the room with the fire, and **Fire** that is inside the room with the fire, looking at the fire.

Save the Model

1. On the **File** menu, click **Save**.
2. Choose a location to save the model (usually a new folder). For this example, we will create a **Import CAD** folder and name the file **import cad**.
3. Click **OK** to save the model.

Run the Simulation

1. On the **FDS** menu, click **Run FDS Parallel**.
2. The **FDS Simulation** dialog will appear and display the progress of the simulation.

View Smoke in 3D

When the simulation is complete, PyroSim Results will display a 3D still image of the model.

1. In the toolbar, select **Realistic**.
2. In **Viewpoints**, double-click **Ground Floor**.
3. In **Scene Geometry**, select **PyroSim Geometry**.
4. In **FDS Results**, double-click **3D Smoke**.

Figure 7 shows the fire and smoke at 50 seconds.

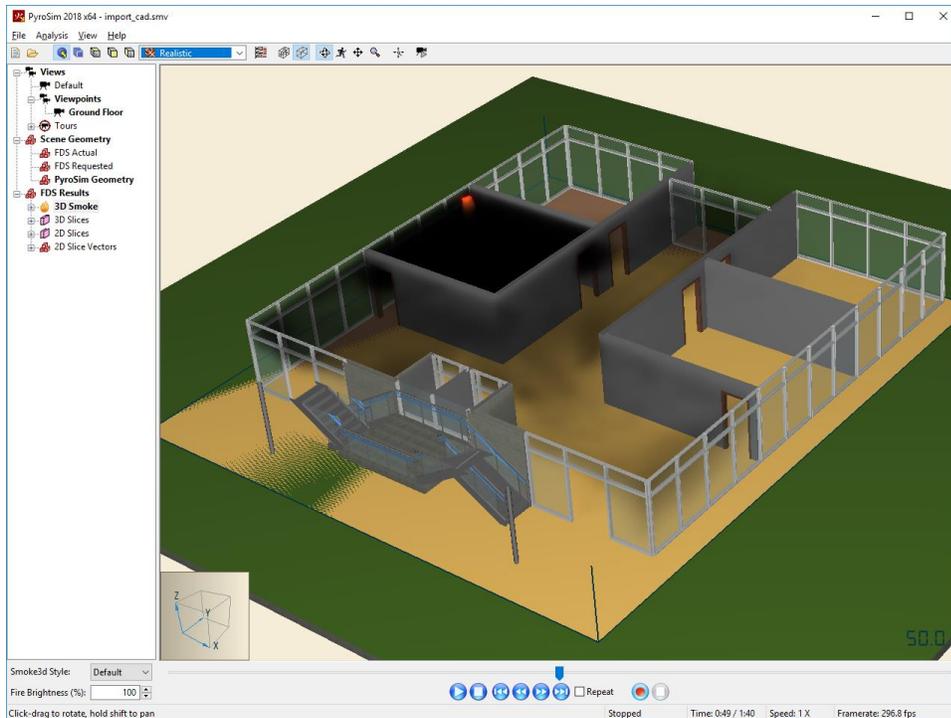


Figure 7: Fire and smoke at 50 seconds.

We can also view from any of the other views we saved. Here is the view from the room with the fire.

1. In the toolbar, select **Realistic**.
2. In **Viewports**, double-click **Fire**.
3. In **Scene Geometry**, select **PyroSim Geometry**.
4. In **FDS Results**, double-click **3D Smoke**.

Figure 8 shows the fire and smoke at 10 seconds.

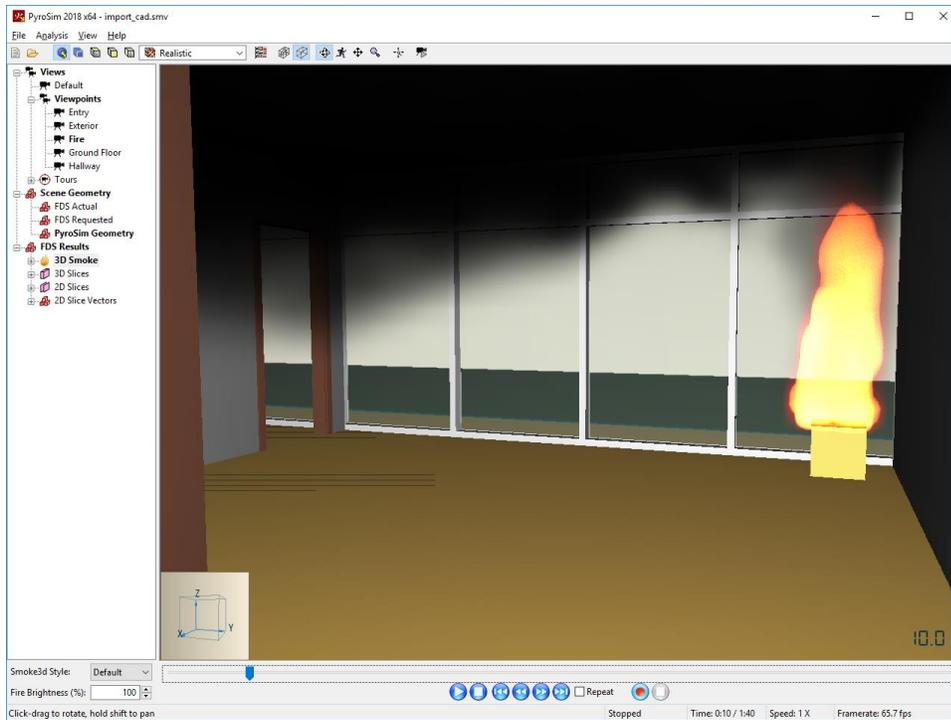


Figure 8: Fire viewed from inside the room.

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.