PyroSim

Room Fire

Room Fire

This is a more complex example. Although the geometry of the model is relatively simple, the fire is ignited by glowing particles that heat the sofa upholstery. The upholstery releases fuel and the fire results. As fuel is released, the sofa burns away.

This approach to modeling a fire pushes the limits of FDS capabilities and is beyond the usual heat release rate approach used by fire safety engineers. This detailed approach may be used by researchers or forensic investigators. There are large uncertainties of ignition and propagation in this detailed approach to modeling.

This example is a simplification of the room fire problem provided by NIST as an FDS verification problem. The simplifications are in the geometry; the material properties and approach to modeling fire is the same as used by NIST. We add optional sections that describe using a texture map for a picture and sprinklers to extinguish the fire.
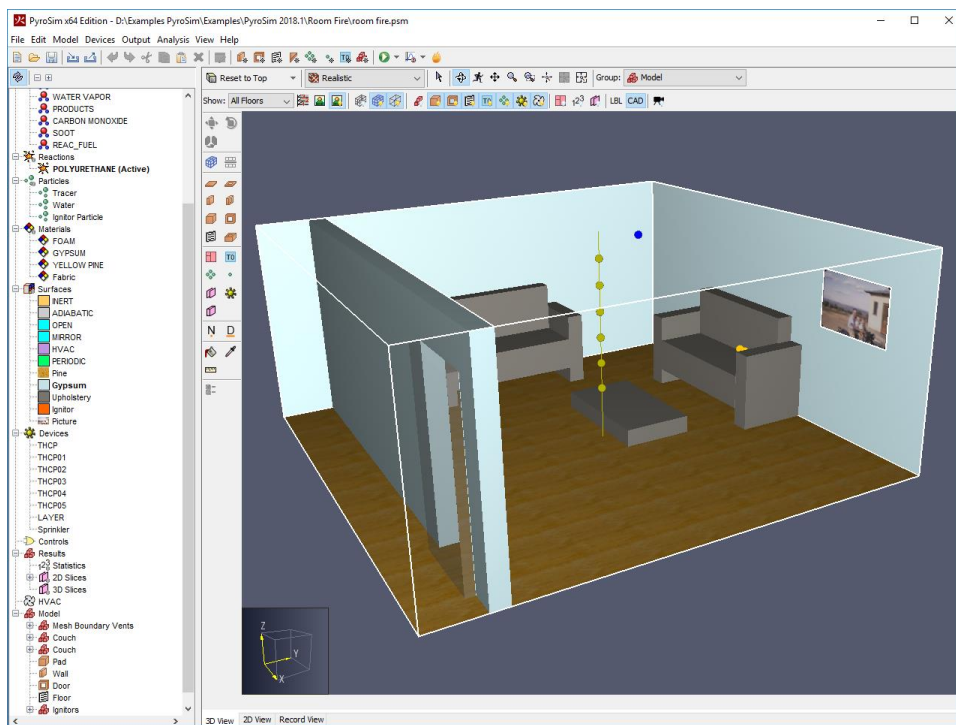

Figure 1. Room fire in this example

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.
- 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.

1. Brief Overview of Fire Modeling

In PyroSim/FDS, a fire is defined by pyrolysis and combustion:

1. Pyrolysis – This is the rate at which a solid or liquid surface generates fuel vapor. For example, wood is heated and releases combustible gas.
2. Combustion – This is the reaction of fuel vapor and oxygen. The products are defined by the chemical reaction. The fuel vapor results from pyrolysis, oxygen is provided by air. The combustion process mixes fuel and oxygen and controls the reaction rate.

A more complete discussion is given in the FDS User Manual (Pyrolysis in Chapter 8 and Combustion in Chapter 12). On the PyroSim web site, the Resources also provide links to a set of videos that further explain the details (see PyroSim: Modeling Fire, Part 1 – Reactions, Part 2 - Heat Release Rate, and Part 3 - Combustion using HRR).

Standard Approach to Fire Modeling

The most frequently used approach to modeling a reaction in PyroSim/FDS assumes a single **fuel** species composed of C, H, O, and N that reacts with **air** to form **products** H_2O, CO_2, soot, and CO. The user specifies the chemical formula of the fuel (and the yields for soot and CO and hydrogen fraction in soot), and PyroSim/FDS calculates the stoichiometric coefficients and heat release rate.

The most common way to define pyrolysis in PyroSim/FDS is to specify a Heat Release Rate (HRR) on a surface. When HRR is specified, FDS uses the heat of combustion defined by the reaction to calculate the fuel vapor mass release rate that will result that will result in the desired energy release.

The most commonly used combustion model in PyroSim/FDS is the "simple chemistry" model, in which the reaction of fuel and oxygen is infinitely fast and controlled only by mixing.

More Complex Solid Phase Pyrolysis used in this Example

While this example uses the standard approach to define the reaction and combustion, it uses a much more complicated approach for the calculation of the solid phase pyrolysis (the conversion of the solid to fuel vapor).

In this example, fuel vapor mass release rate is calculated by simulating solid phase pyrolysis using a **layered** surface and a **pyrolysis rate** defined by the material properties. Layered surfaces are used so that a heat transfer calculation can be performed. The temperature is then used to calculate the solid

phase pyrolysis rate. See the FDS User Manual (Pyrolysis in Chapter 8 and Combustion in Chapter 12) for more information.

2. 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 **room fire.psm**.
4. Click **OK** to save the model.

3. Reaction

First, we define the reaction for this simulation.

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

4. Create the Mesh

The mesh (or meshes) define the computational domain. It is convenient to define the mesh early and use it to help create geometry. You can always modify or refine the mesh later.

In this example we will use mesh cells with a size of 0.10 m. This is geometrically convenient and is somewhat smaller than 1/5 of the characteristic diameter (D*) for a 500 kW fire. In this simulation, the fire is ignited and grows, so there is not a single value of heat release rate with which to calculate D*.

It is the responsibility of the analyst to run verification calculations that show the mesh resolution is sufficiently small so that the results have converged. This is a compromise. 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** and then **OK** to create a new mesh.
3. In the **Min X** box, type **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** 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.
10. On the **View** menu, click **Reset View to All Objects**.

5. Import Material Data

Materials are the building blocks for *Surfaces* that are used to define the geometry of the model. PyroSim includes a database file that provides data and references for the source of the data. We will import selected properties from this file.

1. On the **Model** menu, click **Edit Materials.**
2. Click **Add From Library**.
3. Copy the **FOAM, GYPSUM, and YELLOW PINE** materials from the library into the **Current Model.**
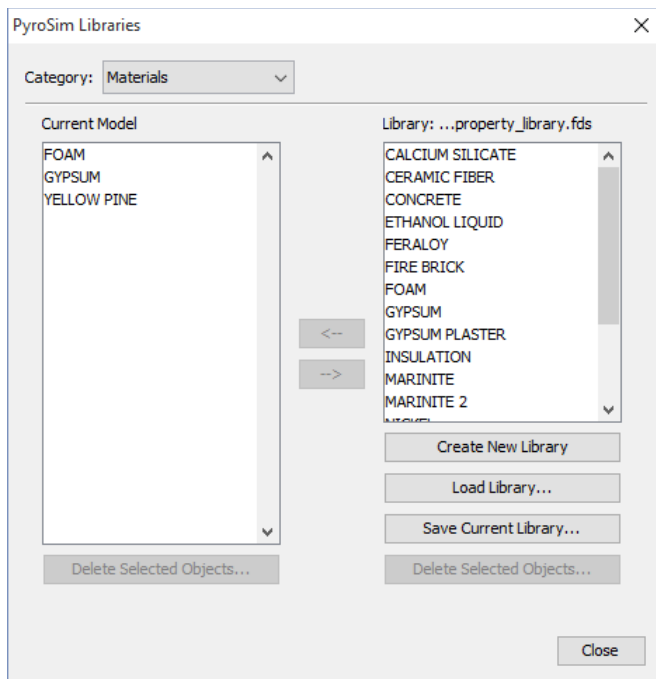4. Close the **PyroSim Libraries** dialog.

Figure 2. Copy the materials from the library into the current model

Because this ignition and burn-away simulation is sensitive to material properties, we will edit the imported FOAM material to have the same values as used in the NIST room fire example (the GYPSUM and YELLOW PINE do not affect the simulation, so we will leave them with the imported properties). To edit FOAM:

1. In the **Edit Materials** dialog list, click **FOAM**.
2. In the **Description** box, type **NIST – Completely Made Up**.
3. In the **Density** box, type **40**.
4. In the **Specific Heat** box, type **1**.
5. Click the **Pyrolysis** tab.
6. In the **Heat of Combustion** box, type **30000**.
7. Under **Reaction**, click **Edit**.
8. In the **Reference Temperature** box, type **300**.
9. Click the **Byproducts** tab.

10. In the **Heat of Reaction** box, type **800**.
11. In the **Composition** column, type **1**.
12. In the **Residue** column select **[Species: SPEC] REAC_FUEL**. This means that the pyrolysis byproducts will be the fuel.
13. Click **OK** to close the **Edit Pyrolysis Reaction** dialog.
14. Click **Apply** to save the changes to **FOAM**.

The sofa cushions will be made of a thin layer of fabric that covers a foam pad. No default fabric is provided in the PyroSim libraries, so we will create one. The values used for this material are the same as used in the NIST model. Note – NIST clearly states the fabric properties are not realistic, so should not be used for other models without justification.

To add a new material:

1. In the **Edit Materials** click **New**.
2. In the **Name** box, type **Fabric**. Click **OK**.
3. Click the **Thermal Properties** tab. In the **Density** box, type **100**. Do not change the default values for the other Thermal Properties.
4. Click the **Pyrolysis** tab.
5. Select the **Heat of Combustion** checkbox and in the value box type **15000**.
6. Under **Reaction**, click **Add**.
7. In the **Reference Temperature** box, type **280**.
8. Click the **Byproducts** tab.
9. Select the **Heat of Reaction** option and in the box, type **1000**. The reaction is Endothermic, that is, pyrolysis requires energy input to occur.
10. In the **Composition** column, type **1**.
11. In the **Residue** column select **[Species: SPEC] REAC_FUEL**. This means that the pyrolysis byproducts will be the fuel.
12. Click **OK** to close the **Edit Pyrolysis Reaction** dialog.
13. Click **OK** to close the **Edit Materials** dialog.

6. Create Surfaces

Materials define physical properties. Solid surfaces can use the material properties to calculate the thermal response.

The floor will be made of yellow pine material. We use a layered surface. Heat transfer through the surface uses a back face boundary condition of air at ambient temperature. No pyrolysis reaction is defined for the **YELLOW PINE** material, so the **Pine** surface will not burn.

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**. A layered surface will calculate heat conduction into the surface.

3. Click on the **Appearance** box and select **psm_wood2.jpg**. Click **OK** to close the **Texture** dialog.
4. In the **Material Layers** tab, 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 3. Creating the Pine 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 define a neutral blue color (e.g. RGB of 198, 225, 230 and Alpha 255). 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.

The upholstery will have a fabric surface and a foam core:

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 grey color (e.g. RGB of 120, 116, 110 and Alpha 255). Click **OK** to close the **Surface Color** dialog.

3. In the **Material Layers** we define the two materials for the upholstery. First the fabric. In the **Thickness** column, type **0.002**. Click the **Edit** button. In the **Mass Fraction** column, type **1.0**. In the **Material** column, select **Fabric**. Click **OK** to close the **Composition** dialog.

4. Now we add the foam. Click **Insert Row**. In the **Thickness** column, type **0.1**. 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. You should now have two material layers in the upholstery surface.

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 cell is consumed either by the pyrolysis reaction. This option can also be used when HRR is prescribed.

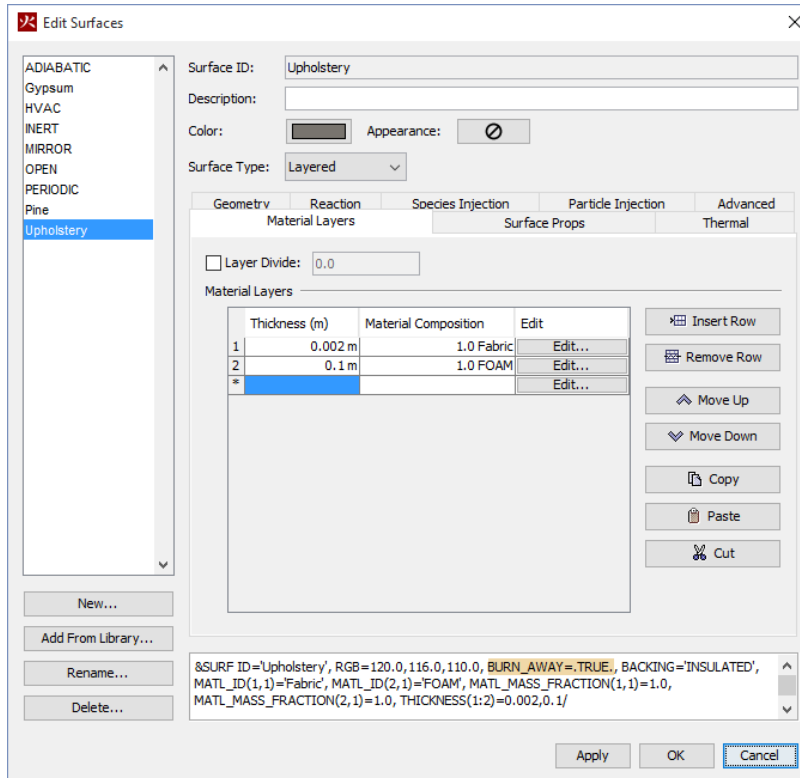7. In the **Edit Surfaces** dialog, click **Apply** to save the changes.

Figure 4: The upholstery has two material layers.

We will place an initial *ignitor* object on the sofa. This ignitor will be at a constant temperature of 1000 °C and will radiate heat that will ignite the upholstery. To create this burner surface:

1. In the **Edit Surfaces** dialog, click **New**. Give the **Surface Name** as **Ignitor**, select the **Surface Type** as **Heater/Cooler**, and click **OK**.

2. In the **Thermal** tab, in the **Boundary Condition Model**, select **Fixed Temperature**. This means the surface will stay at a constant temperature.
3. In the **Thermal** tab, in the **Surface Temperature** box, type **1000**. Leave the **Emissivity** at **0.9**.
4. Click the **Geometry** tab, in the **Geometry** list, select **Cylindrical**. In the **Radius** box, type **0.01**; select **Length** and in the box, type **0.15**.
5. In the **Edit Surfaces** dialog, click **OK** to save the changes and close the dialog.

Note that since the ignitor is just a constant surface temperature, we do not need to define any material properties for the surface.

7. Create Furniture (Obstructions)

We will now add furniture to the model.

Couch

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

1. Right-click the **Model** category in the tree view.
2. On the **Model** menu, click **New Group**. The **Parent Group** will be **Model**.
3. In the **Group Name** box, type **Couch**.
4. Click **OK** to close the **Create Group** dialog.

To create the seat:

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Seat cushion**.
3. In the **Group** list, select **Couch**.
4. On the **Geometry** tab, enter the values in Table 1.
5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 1. Couch seat dimensions

	X	Y	Z
Min	1.5	3.8	0.2
Max	3.1	4.6	0.6

To create the right armrest:

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Right armrest**.
3. In the **Group** list, select **Couch**.
4. On the **Geometry** tab, enter the values in Table 2.
5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 2. Right armrest dimensions

	X	Y	Z
Min	1.3	3.8	0.0
Max	1.5	4.6	0.9

Use the copy function to create the left 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.

Double-click on the new **Right armrest** in the Tree view and change the **ID** to **Left armrest**. Click **OK**.

To create the back:

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Back cushion**.
3. In the **Group** list, select **Couch**.
4. On the **Geometry** tab, enter the values in Table 3.
5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 3. Couch back dimensions

	X	Y	Z
Min	1.5	4.4	0.6
Max	3.1	4.6	1.2

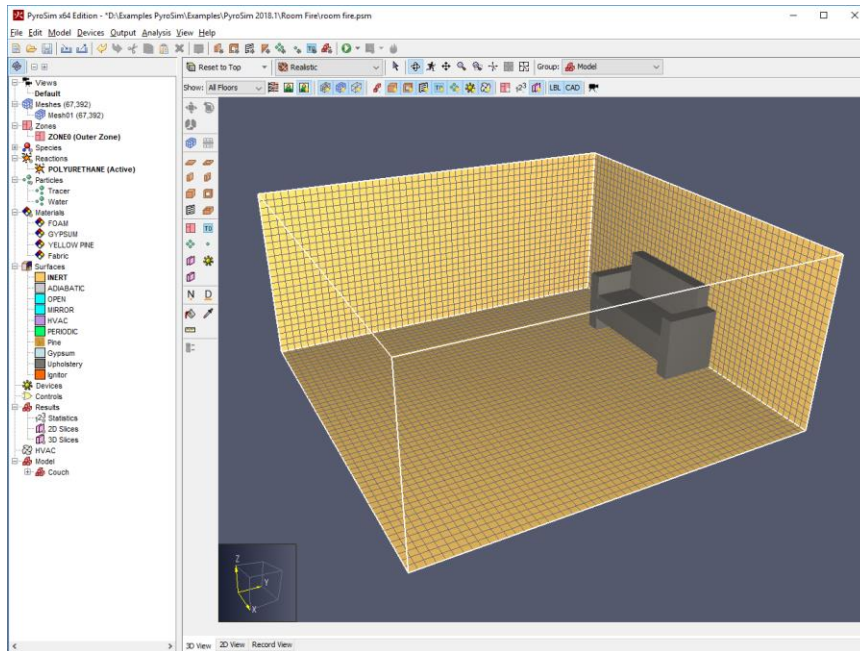The display will appear as shown in Figure 5.

Figure 5. 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.

Rotate the second couch to lie against the well.

1. In the Tree View, right-click the new **Couch** 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 **ID** box, type **Pad**.
3. In the **Group** list, select **Model**.
4. On the **Geometry** tab, enter the values in Table 4.

5. On the **Surfaces** tab, select **Single** and select **Upholstery** from the list.
6. Click **OK** to close the **Obstruction Properties** dialog.

Table 4. 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.
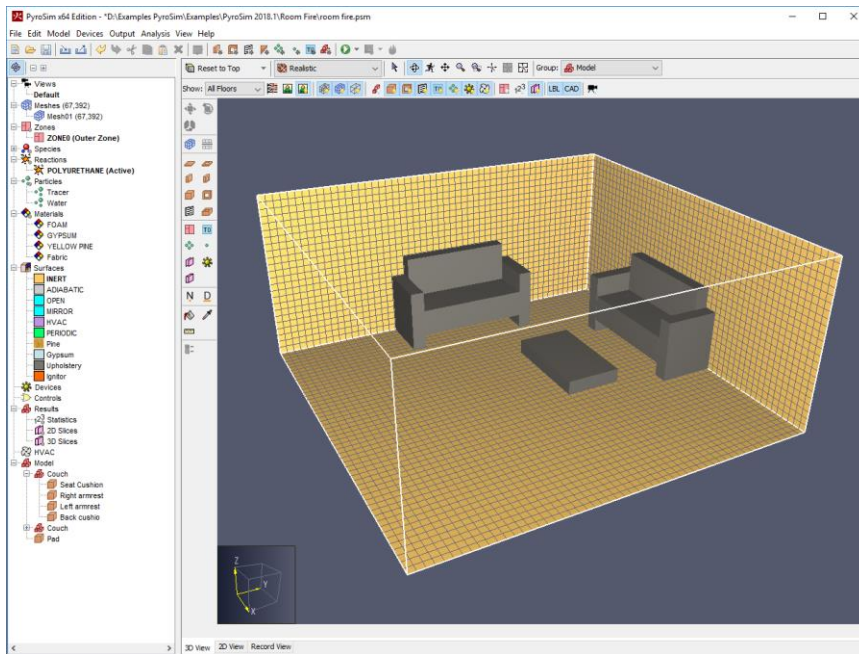
Figure 6. The resulting room display

Constructing complex objects can be time consuming and PyroSim supports several options. If your geometry is available in DXF, DWG, or STL format, PyroSim supports import. You can also import existing FDS models. 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 the text from an FDS input file and paste it into a PyroSim model.

8. Walls

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

1. Select the **2D View**. Click the **Reset** tool.
2. Select the **Wall** (🗂) Tool.

3. Click **Tool Properties** ( ). In the **Name** box, type **Wall**. Set the **Z Location** to **0.0**, the **Height** to **2.4**, the **Thickness** to **0.2**, 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. Press the CTRL key to flip the wall to the lower part of the model, Figure 7.
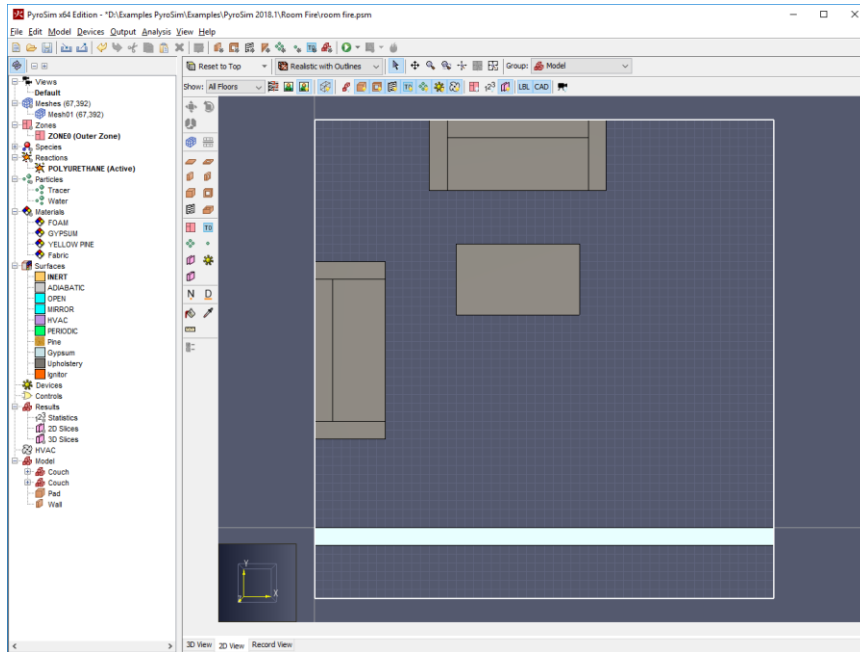

Figure 7. Drawing the wall

9. 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 **ID** box, type **Door**.
3. In the **Group** list, select **Model**.
4. On the **Geometry** tab, enter the values in Table 5. 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 5. Door dimensions

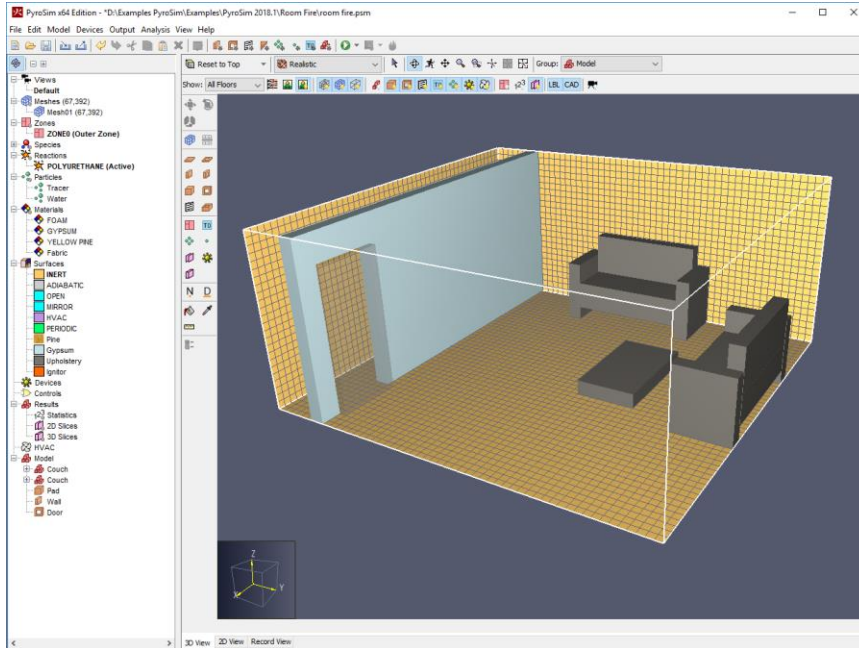|	X	Y	Z
Min	4.0	-0.3	0.0
Max	4.9	0.1	2.0

The model now looks like:

Figure 8. The model after adding the door

10. Use Vents to Define the Floor, Walls, and Ceiling

In FDS, *Vents* are used to describe 2D planar objects. In this example, we use vents to define the wood floor and the gypsum walls and ceiling. We do this because we want the fire to heat the walls and ceiling. Recall that the Pine and Gypsum surfaces were layered with the air gap boundary condition.

To create the floor:

1. On the **Model** menu, click **New Vent**.
2. In the **ID** 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 **0.0.**
6. In the **Bounds** boxes, enter the values in Table 6.
7. Click **OK** to close the **Vent Properties** dialog.

Table 6. Floor dimensions

	X	Y
Min	0.0	-0.8
Max	5.2	4.6

We want the walls and ceiling to use the Gypsum surface. As we did for the floor, we could define vents on the walls and ceiling. However, since these are on the boundary of the model, it is easier to just set the **Default Surface Type** to Gypsum.

To define the **Default Surface Type**:

1. On the **Analysis** menu, click **Simulation Parameters**.
2. Click the **Misc** tab.
3. For the **Default Surface Type**, select **Gypsum**.
4. Click **OK** to close the **Simulation Parameters** dialog.

11. Add 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 **MESH01** and click **Open Mesh Boundaries**. This will add a group named **Vents for MESH** that includes vents on each grid boundary.
2. Delete four vents, the XMAX, XMIN, YMAX, and ZMIN vents.
3. Now double-click the **Mesh Vent: Mesh [ZMAX]** vent. Click the **Geometry** tab and change **Max Y** to **0**. Click **OK**.

Now there are just two open vents. One opposite the door and one above the door. This will let smoke leave the door and flow up. It will also allow cold air to flow in the lower half of the door.

12. Create Ignitors that will Light the Fire

In this model, we will define ignitors that will glow at a fixed temperature, heat the sofa, and ignite the sofa. Creating the ignitors is a two-step process: first we define particles, then we position them in the model.

To define the particles:

1. On the **Model** menu, click **Edit Particles**.
2. Click **New**. In the **Particle Name** box, type **Ignitor Particle**.
3. In the **Particle Type** list, select **Solid**.
4. Click **OK** to create the particle.
5. On the **Solid** tab, for **Surface**, select **Ignitor** (this is the fixed temperature surface we previously created.
6. In the **Movement** option, select **Particles are Stationary**.
7. Click the **Injection** tab. In the **Duration** box, type **1E5**. This means that the particle will remain in the model for the entire simulation.
8. Click **OK** to close the **Edit Particles** dialog.

Now we will create the actual ignitors. First create an Ignitor group:

1. Click the **Model** category in the tree view.
2. On the **Model** menu, click **New Group**. The **Parent Group** will be **Model**.
3. In the **Group Name** box, type **Ignitors**.
4. Click **OK** to close the **Create Group** dialog.

Now create the first ignitor:

1. On the **Model** menu, click **New Particle Location**.
2. In the **ID** box, type **Ignitor**.
3. In the **Group** list, select **Ignitors**.
4. In the **Particle** list, select **Ignitor Particle**.
5. In the **Droplet Count** select **Constant** and change the number to **1**.
6. Click the **Geometry** tab. In the **Location Properties** boxes, enter the values in Table 7.
7. Click **OK** to close the **Particle Cloud Properties** dialog.

Table 7. Ignitor location

X	Y	Z
2.45	4.35	0.65

Copy to create two more ignitors:

1. Right-click on the Ignitor and select **Copy/Move**.
2. In the **Model** options, select **Copy** and in the **Number of Copies** box, type **2**.
3. In the **Offset** boxes, enter the values in Table 8.
4. Click **OK** to close the **Translate** dialog.

Table 8. Ignitor Offsets

X	Y	Z
0.1	0.0	0.0

13. 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.
5. Click the **LBL** icon to toggle labels off. Click the **Devices** icon to toggle devices off.

14.   Create Layer Zoning Device

The upper and lower layers in the room can be calculated using the Layer Zone Device. To create this:

1. On the **Devices** menu, click **New Layer Zoning Device.**
2. Select the three checkboxes to enable **Measure Layer Height**, **Upper Temperature**, and **Lower Temperature**.
3. In the **End Point** boxes enter the data in Table 9.
4. Click **OK** to close the **Layer Zoning Device** dialog.

Table 9. Layer Zone Geometry

	X	Y	Z
End Point 1	2.6	2.3	0
End Point 2	2.6	2.3	2.4

15.   Create Slice Records for Results Plotting

Slice planes can be used to display 2D contours in the 3D Results display. In this analysis, we will save temperature and velocity data for future plotting. To define the slice plane:

1. On the **Output** menu, click **2D Slices.**
2. Fill the table by entering the values in Table 10.
3. Click **OK** to close the **Animated Planar Slices** dialog.
4. In the Toolbar, click **Show Slices** to toggle display on and off.

Table 10. Slice plane data

XYZ Plane	Plane Value (m)	Gas Phase Quantity	Use Vector?	Cell Centered?
X	2.6	Temperature	NO	NO
X	2.6	Visibility	NO	NO
X	2.6	Velocity	YES	NO
X	4.45	Temperature	NO	NO
X	4.45	Visibility	NO	NO
X	4.45	Velocity	YES	NO

16.   Create Surface Plots

Temperatures (and other quantities) on the surface of objects can be plotted. 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.

17.   Specify Simulation Properties

To define the end time:

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

18.　Hang a Picture on the Wall (Optional)

Let us fancy things up by hanging 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 **Basic** and click **OK**. (**Basic** is just an **INERT** surface that allows you to change the appearance. You could also use a layered surface with Gypsum material as we did for the wall.)
3. Click on the **Appearance** box.
4. Click the **Import** button and select the image you want as a picture. In homage to my Dad and Brother, I used the image called motorcycle.jpg. That is me in the middle. Right-click the link and click **Save Link As** to download the image if you would like to use it.
5. The image you selected will be displayed. 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 **ID** 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 **4.6.**
7. In the **Bounds** boxes, enter the values in Table 11.
8. Click **OK** to close the **Vent Properties** dialog.

Table 11. Picture dimensions

	X	Z
Min	3.6	1.2
Max	4.6	1.875

19.　Add Sprinklers to Extinguish Fire (Optional)

We can add a sprinkler that will spray water to extinguish the fire.

1. On the **Devices** menu, click **New Sprinkler**.
2. Change the **Name** to **Sprinkler**.

3. For **Temperature Link**, select **Generic Residential Link**.
4. For **Spray Model**, select **Generic Residential Spray**.
5. In the **Location** boxes, enter **X = 2.6**, **Y = 2.8**, and **Z = 2.3**.
6. Click **OK** to close the **Sprinkler** dialog.

20.   The Model is Complete

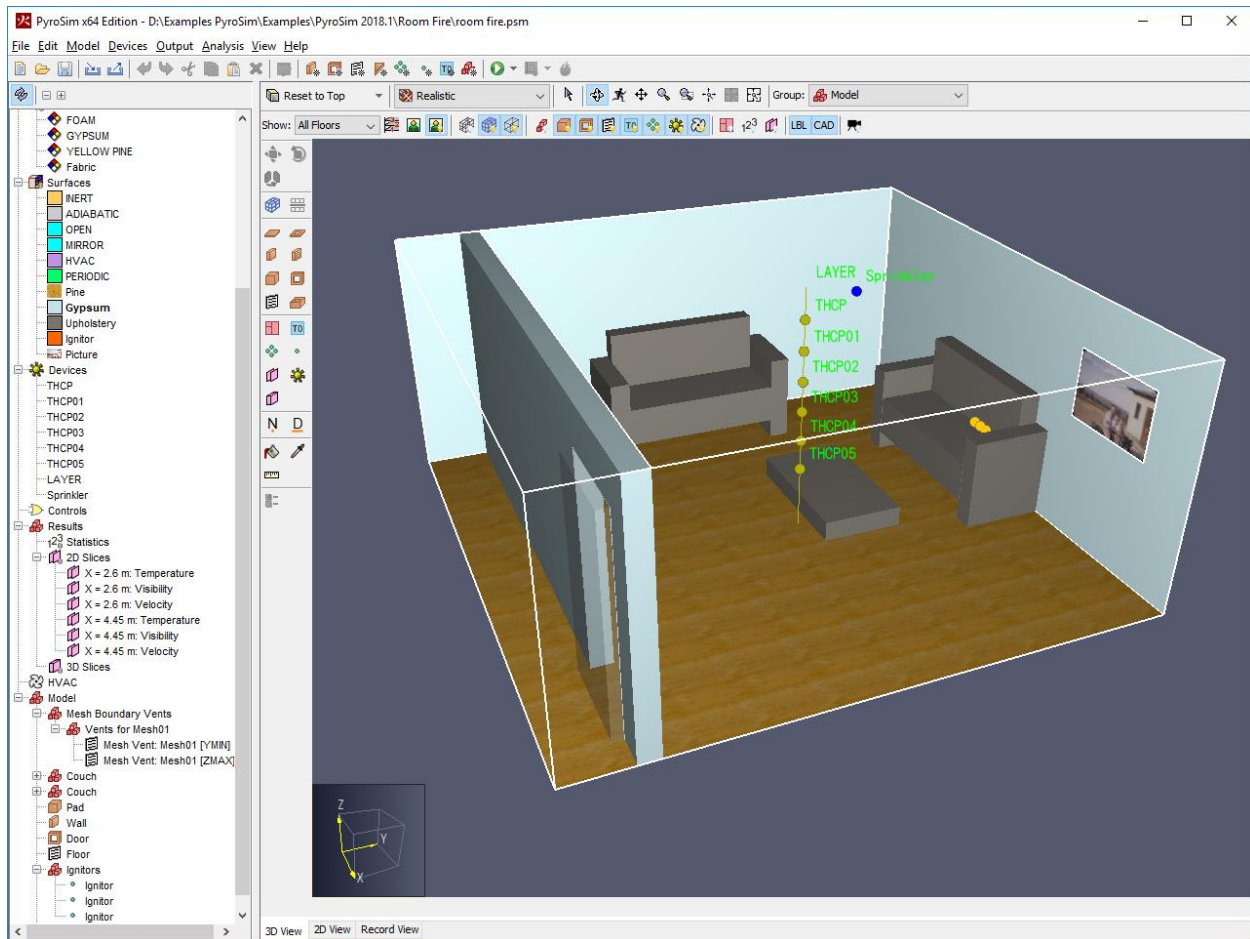Your model should now look like Figure 9. Save it.

Figure 9. Completed model

21.   Run the Analysis

To run the analysis:

1. On the **FDS** menu, click **Run FDS**. If asked to expand holes, click **Yes**. You will receive a warning that two vents overlap.

22.   View the Results

An image showing the smoke and heat release rate per unit volue is shown in Figure 10. This is a time when the sofa has begun to burn, but the fire is still relatively small. You can clearly see the air

circulation caused by the fire, with hot air flowing out the top half of the door and cold air flowing in the bottom half of the door. The smoke has formed a layer.
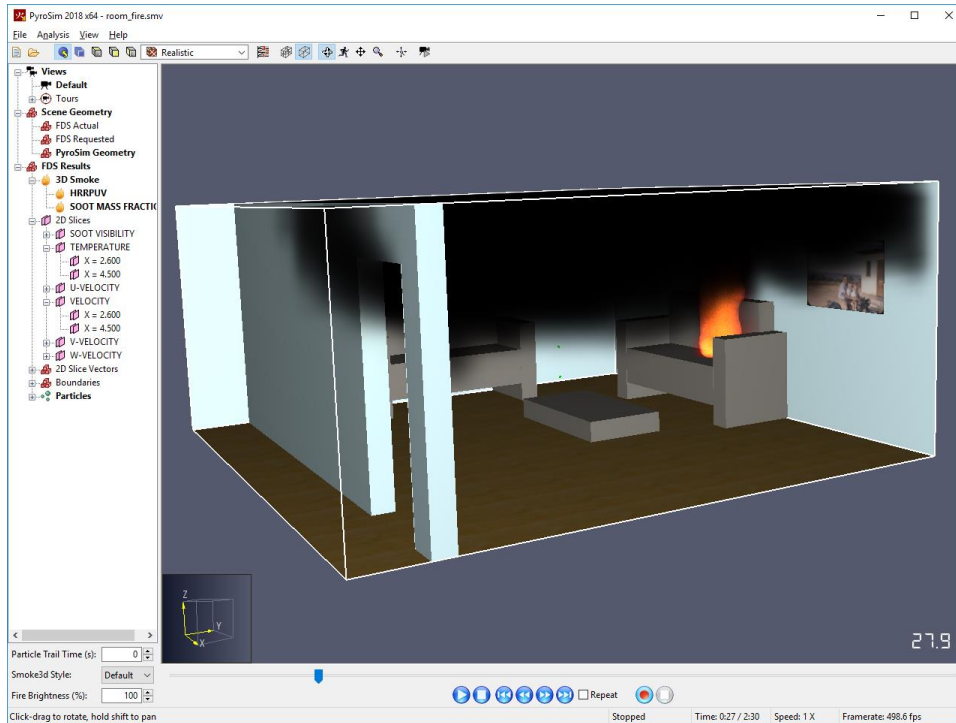

Figure 10. Plot showing smoke and fire. You can see the formation of layers, the hot air and products flowing out the top of the door, and fresh air flowing in the bottom of the door.

A plot of surface temperature is given in Figure 11. In this simulation, the sprinkler activated and extinguished the fire. However, if you remove the sprinkler and run the simulation for 200 seconds, you will see the parts of the sofa burn away.

Other plot options include the sprinkler particles and slice vectors. When displaying the smoke, right-click the **SOOT MASS FRACTION** and click **Properties** to edit the display. Note that the default options are the most realistic rendering of the smoke, but there are times where it helps to reduce the smoke opacity to simultaneously display smoke and slices, for example.
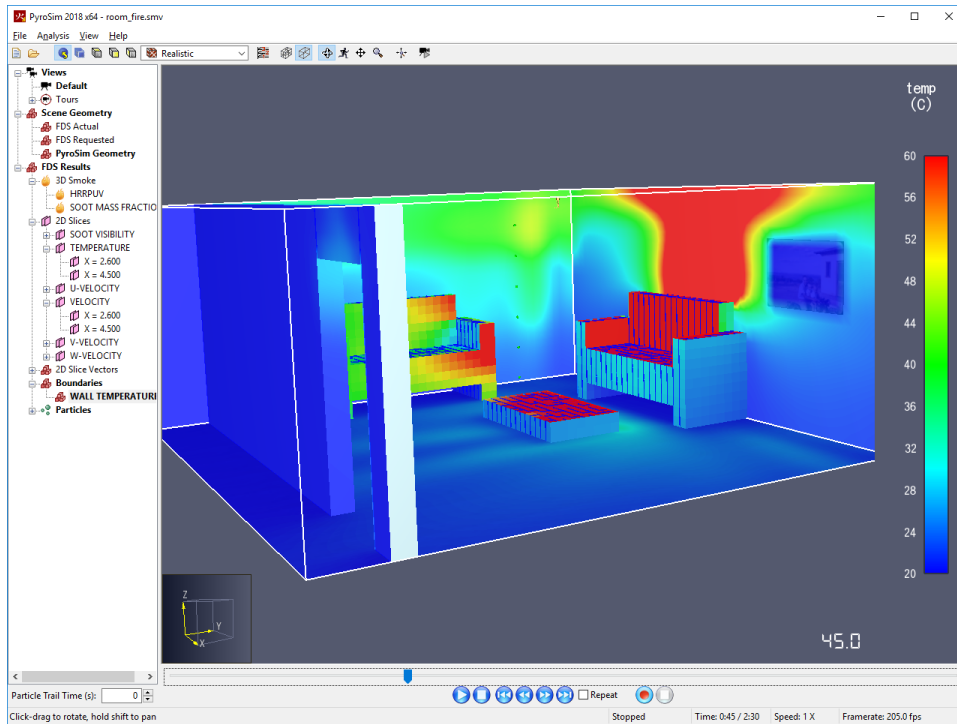
Figure 11: Plot of surface temperature.

To view time history results

1. In the PyroSim toolbar, click **Plot Thermal Results**.
2. Plot the heat release rate as a function of time, Figure 12. This simulation included sprinkler activation that extinguished the fire.
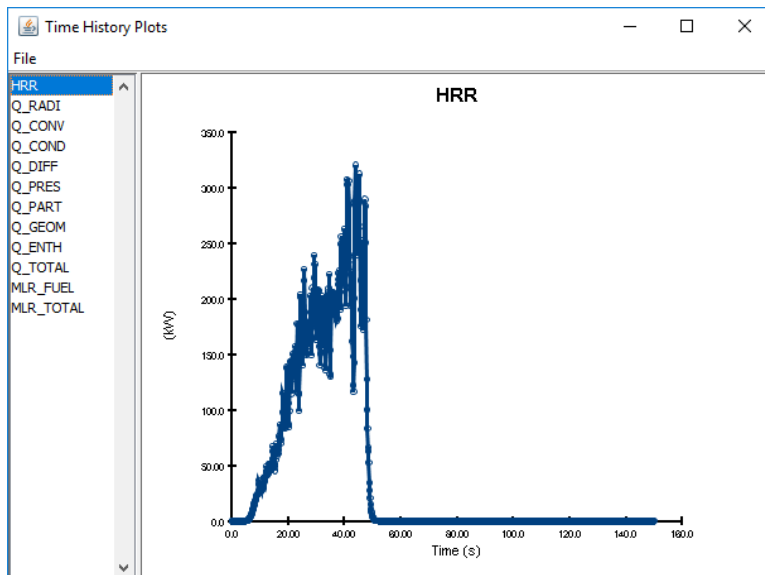3. You can also make plots of the thermocouple readings and the layer height.

Figure 12. Heat release rate.

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.

Original NIST FDS Verification Input File

This example we created is a simplification of the room fire problem provided by NIST as an FDS verification problem. You can download the complete original FDS verification file at:

https://github.com/firemodels/fds/blob/master/Verification/Fires/room_fire.fds

Save the **Raw** FDS file. To import the original FDS file into PyroSim, on the **File** menu, click **Import FDS/CAD File**. During import, specify that the NIST model was created using FDS 6.

The simplifications we made in the example are mostly related to geometry.