



Burner Fire

Burner Fire

In this tutorial you will create a 500 kW burner fire and measure the temperature in the center of the plume at a height of 1.5 m . This example defines a fire by specifying a Heat Release Rate (HRR). This is both the simplest and most commonly used approach for fire safety engineers to represent fire.

For this example, it is sufficient to understand that modeling a fire using the heat release rate requires the user to specify two pieces of input data:

- A **reaction** that defines the products and energy release during combustion and,
- A **heat release rate** that defines size of the fire. When this is specified, FDS actually releases fuel at a mass flow rate based on the reaction energy for complete combustion. This is usually specified as heat release rate per unit area and is assigned to a surface.

Once the reaction and heat release rate are defined, fuel is released from the surface, mixes with air, and reacts to form combustion products (including heat). A longer (but still brief) introduction to fire modeling is given in the Room Fire example.

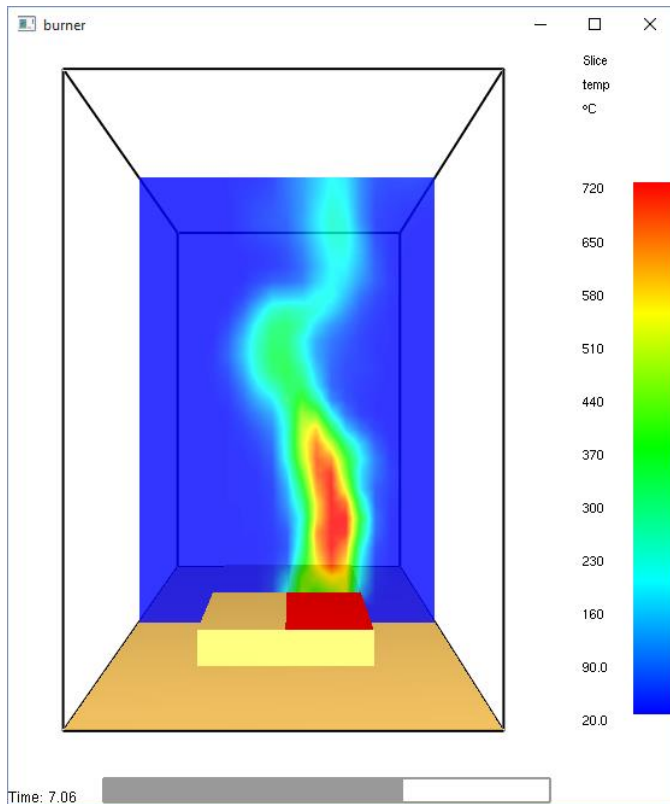


Figure 1. Burner fire in this example

This tutorial demonstrates how to:

- Create a burner fire.
- Add a thermocouple.
- Add a slice plane for temperature visualization.
- View 3D results using Smokeview.
- View 2D results using PyroSim.

The FDS input file listing is provided at the end of this document. You can import this directly into PyroSim.

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 the Mesh

In this example we will use mesh cells that are 0.1 m across. This value is somewhat smaller than 1/5 of the characteristic diameter (D^*) for a 500 kW fire. As a rule of thumb, recommended cell sizes range from 1/5 to 1/20 of D^* to ensure at least a moderate level of accuracy in modeling the plume (McGrattan, Kevin, et al. 2014). 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**.
3. Click **OK** to create the mesh.
4. In the **Min X** box, type **-1** and in the **Max X** box, type **1**.
5. In the **Min Y** box, type **-1** and in the **Max Y** box, type **1**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **3**.
7. In the **X Cells** box, type **20**.
8. In the **Y Cells** box, type **20**.
9. In the **Z Cells** box, type **30**.
10. Click **OK** to save changes and close the **Edit Meshes** dialog.
11. On the **View** menu, click **Fill View** to resize the image.

Burner Fire

2. Click **New**.
3. In the **Surface Name** box, type **Fire**, Figure 3.
4. In the **Surface Type** list, select **Burner**.
5. Click **OK** to create the new burner surface.
6. Click **OK** to save changes and close the **Edit Surfaces** dialog.

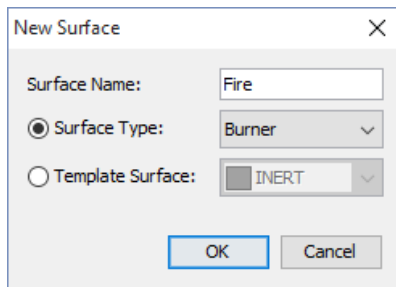


Figure 3. Inserting a new burner surface

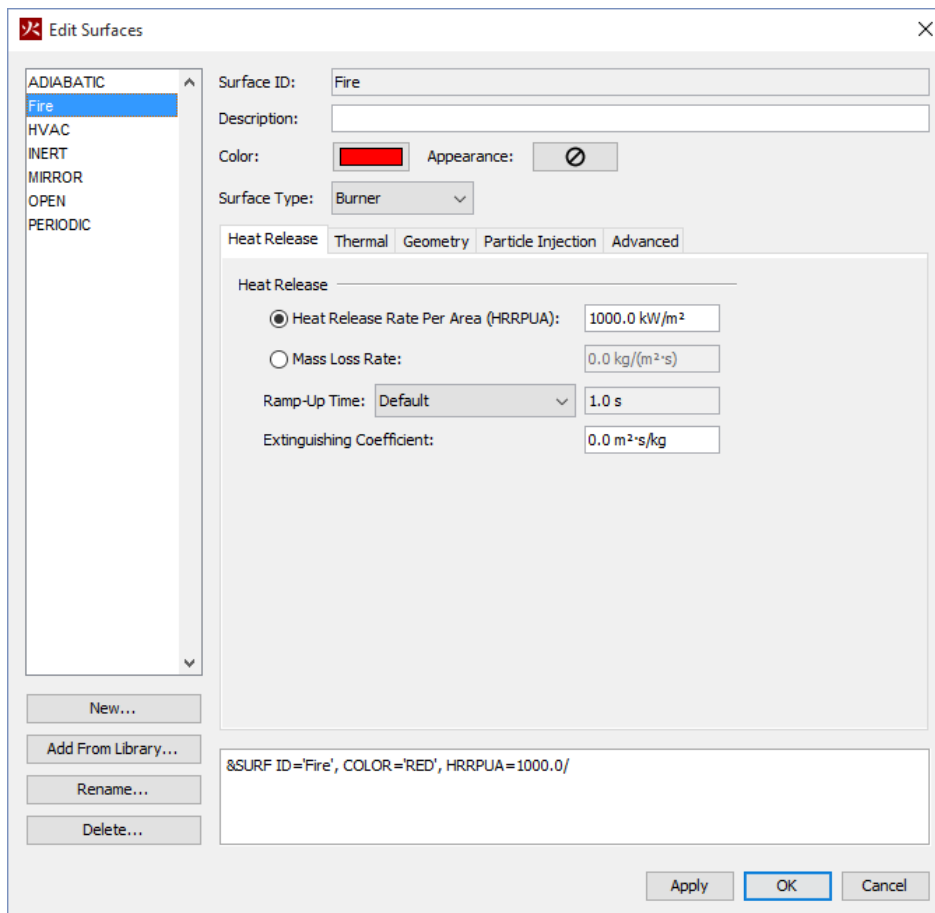


Figure 4. Default parameters for the burner surface

Create the Fire

Now that we define the fire location in our model. In this example, we will first create an obstruction and then place the fire (defined by a vent) on the obstruction. If the fire covered an entire side of the obstruction, we could just assign the fire surface to the obstruction and not use a vent. If the fire was on a model boundary, we could just use a vent and not include an obstruction.

First we create the obstruction:

1. On the **Model** menu, click **New Obstruction**.
2. In the **ID** box, type **Fire Obstruction**.
3. Click on the **Geometry** tab.
4. In the **Min X** box, type **-.5** and in the **Max X** box, type **.5**.
5. In the **Min Y** box, type **-.5** and in the **Max Y** box, type **.5**.
6. In the **Min Z** box, type **0** and in the **Max Z** box, type **.2**.
7. 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. Also, all surfaces of the obstruction are the default inert boundary condition.

Now we will define the fire on one half of the top surface of the obstruction. Recall that FDS broadly uses the term “vent” to reference any 2D surface used to apply boundary conditions on a rectangular patch. Our vent will have a surface area of 0.5 m², so the total heat release rate will be 500 kW.

1. On the **Model** menu, click **New Vent**.
2. In the **ID** box, type **Fire Vent**, Figure 5.
3. In the **Surface** list, select **Fire**. This specifies that the previously created burner surface will define the properties of the vent.
4. Click on the **Geometry** tab. In the **Plane** list, select **Z** and in the location box type **.2**.
5. In the **Min X** box, type **0** and in the **Max X** box, type **.5** (recall that the vent covers only half the top surface).
6. In the **Min Y** box, type **-.5** and in the **Max Y** box, type **.5**.
7. Click **OK** to create the new burner vent.

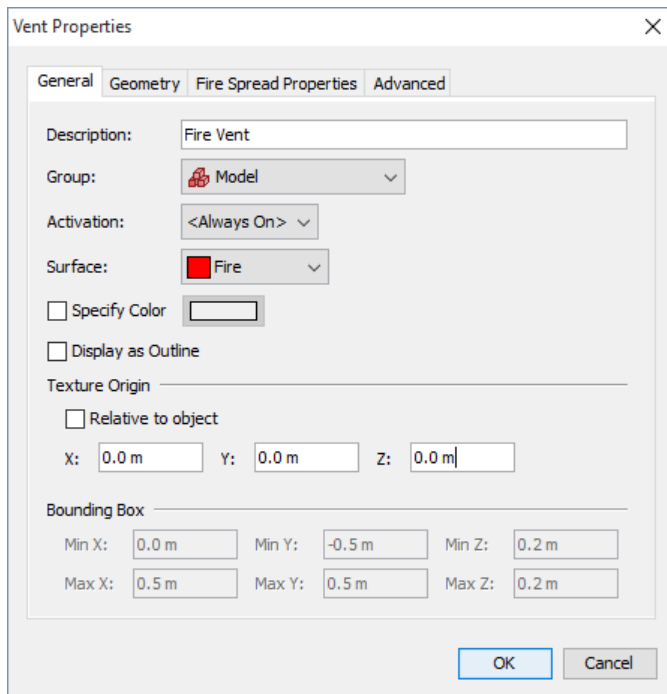


Figure 5. Creating the burner vent

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 **Tree** view, right-click **Mesh01** and click **Open Mesh Boundaries**. This will create six new open vents on all sides of the mesh.
2. Since we want the bottom to be closed, select the vent **Mesh Vent: Mesh01 [ZMIN]** and delete it.

Burner Fire

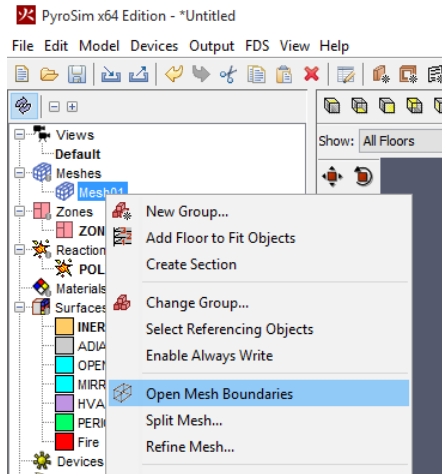


Figure 6: Create open mesh boundaries

Add a Thermocouple

1. On the **Devices** menu, click **New Thermocouple**.
2. In the **Device Name** box, type **Thermocouple at 1.5 m**.
3. On the **Location** row, in the **Z** box, type 1.5.
4. Click **OK** to create the thermocouple. It will appear as a yellow dot in the center of the model.
5. On the toolbar, click the **Show Labels** button to toggle the labels on and off.

Add Slice Planes (Contour planes)

1. On the **Output** menu, click **Slices**.
2. In the **XYZ Plane** column, click the input cell and select **Y**.
3. In the **Plane Value** column, click the cell and type **0**.
4. In the **Gas Phase Quantity** column, click the cell and select **Temperature**.
5. In the **Use Vector?** column, click the cell and select **NO**.
6. Click **OK** to create the slice plane.
7. Repeat, but this time plot **Velocity** and select **YES** for vectors.
8. Repeat, but this time in the **Gas Phase Quantity** column, click and scroll to the top of the list. Select **[Species Quantity]**, then for **Quantity** select **Volume Fraction** and for **Species** select **Air**. Click **OK**. Select **NO** for vectors. Press the **ENTER** key for a new line.
9. Repeat so that you are plotting the Volume Fraction of **AIR**, **PRODUCTS**, and **REAC_FUEL**. This will show us where the reaction is occurring. Air is supplied from the open boundaries, fuel is supplied by the fire surface, and products are the result of combustion. See Figure 7.
10. On the toolbar, click the **Show Slices** button to toggle the slice planes on and off.

Burner Fire

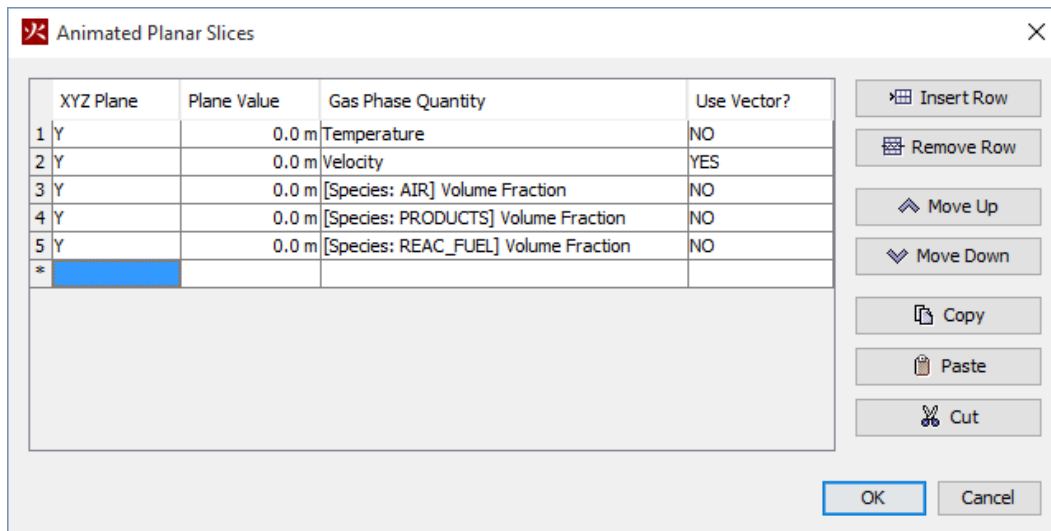


Figure 7: Defining the slice plane plots.

Rotate the Model for a Better View

1. To reset the zoom and properly center the model, press CTRL+R. PyroSim will now be looking straight down at the model along the Z axis.
2. Press the **left mouse button** in the **3D View** and drag to rotate the model. In Figure 8, the burner is shown in red and the thermocouple as a yellow dot. The slice plane is semi-transparent and the open vents are blue.

Burner Fire

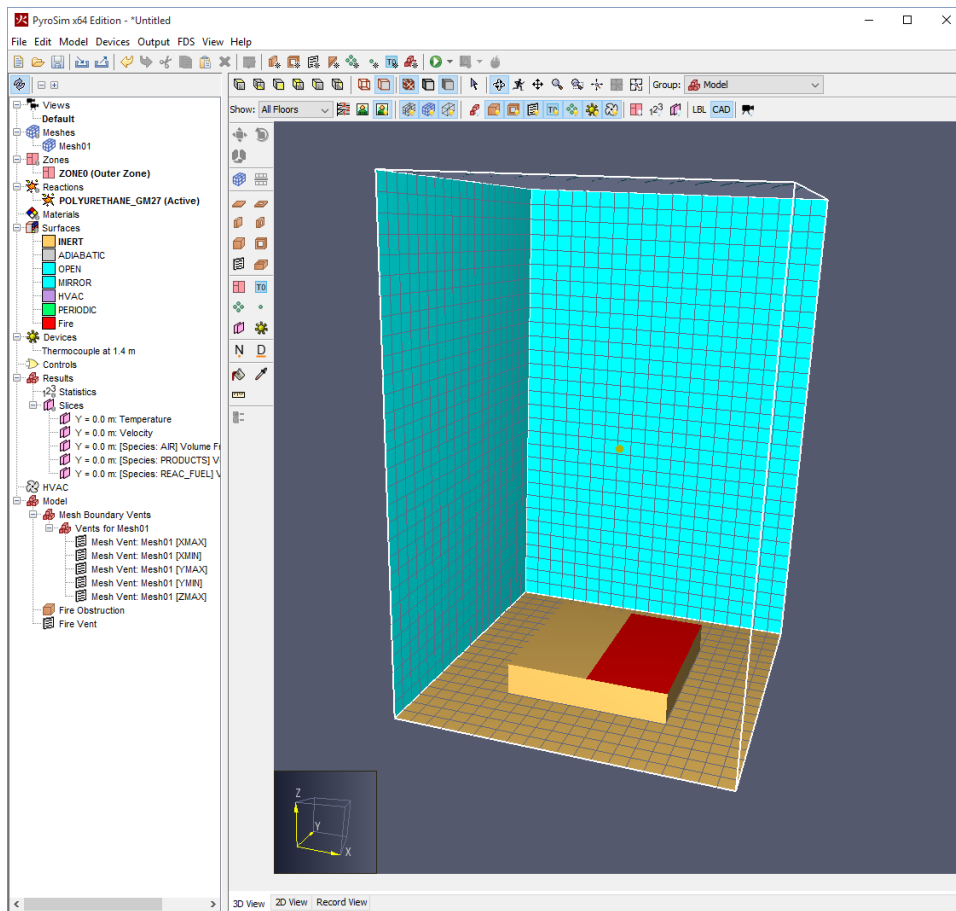


Figure 8. The Burner Fire model

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 **Burner** folder and name the file **burner**.
3. Click **OK** 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. By default, PyroSim specifies an end time of 10 seconds. When the simulation is complete, Smokeview will start and display a 3D still image of the model.

View Smoke in 3D

1. In the Smokeview window, right-click to activate the menu.
2. In the menu, click **Load/Unload > 3D Smoke > SOOT MASS FRACTION**.

Burner Fire

3. In the menu, click **Load/Unload > 3D Smoke > HRRPUV**. This is the heat release rate per unit volume and is an indication of the location of the combustion reaction.
4. In the menu, click **Load/Unload > Vector Slices > Velocity > Y=0**.
5. This will start an animation of the smoke, HRRPUV, and velocity vectors. This is a busy plot, but it does show many details of the fire. The polyurethane reaction results in very dense smoke.

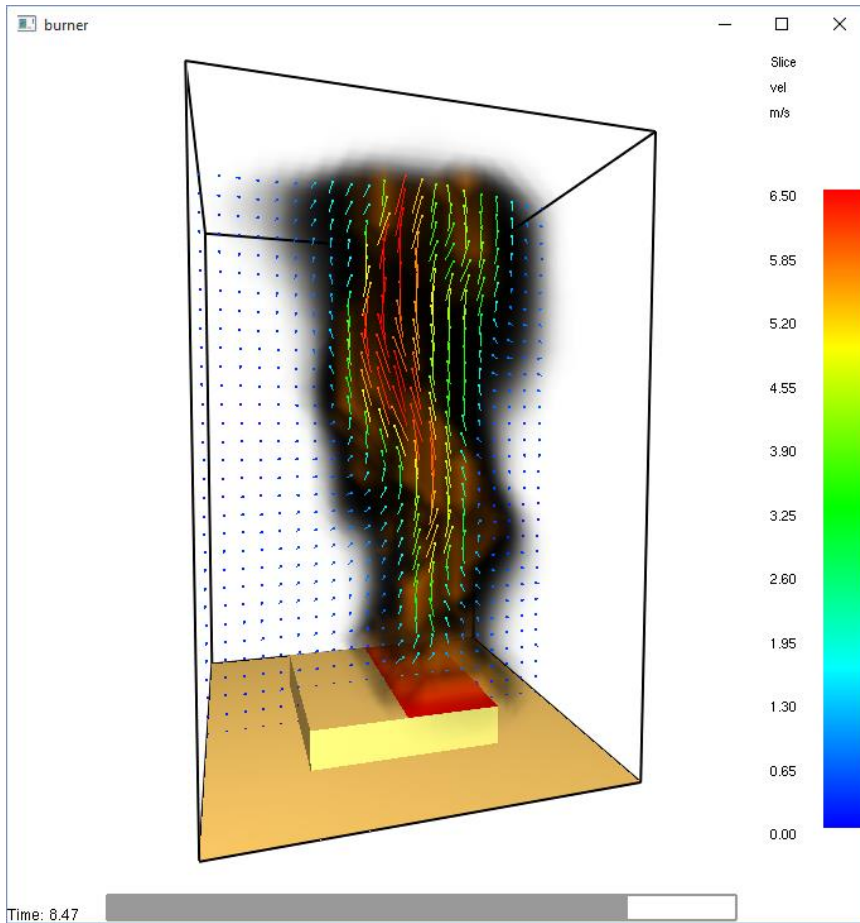


Figure 9: A plot of smoke, the heat release rate per unit volume, and velocity vectors.

Quick Aside Smokeview Controls

Smokeview is developed by NIST, so some of the controls are different than PyroSim.

1. To rotate in Smokeview, click and drag the mouse.
2. To zoom and pan horizontally in Smokeview, press the CTRL key and drag up/down (for zoom) or left/right (for horizontal pan).
3. To pan vertically in Smokeview, press the ALT key, click the mouse, and drag up/down.
4. To view a specific time in the animation, click the **timeline bar** in the bottom of the Smokeview window. To return to animation mode, press **t**.

Burner Fire

5. To clear the display in Smokeview, right-click to activate the menu, then click **Load/Unload > Unload All**.
6. Right-click and select **Help** for more hints.

Air, Fuel, and Products Contours

At any location, the volume fractions of air, fuel, and products will sum to 1. Contours of these quantities are shown in Figure 10. Air is most of the volume fraction for most of the model. The fuel is released from the fire surface and the products result from mixing of the fuel and air.

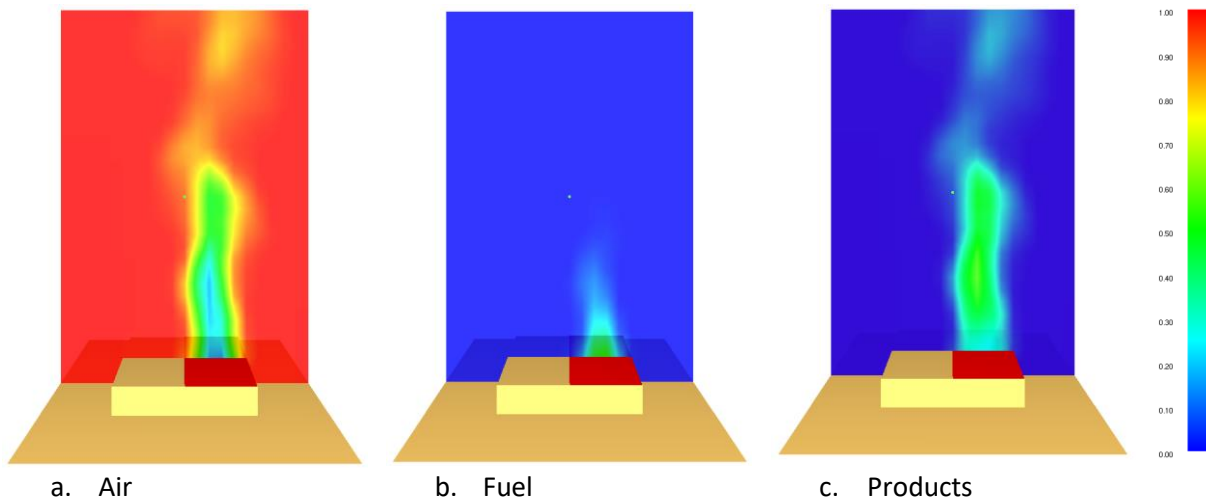


Figure 10: Contours of the air, fuel, and products volume fractions. Blue corresponds to zero and red to 1.

As a mechanical engineer, I initially assumed (incorrectly) that the heat release rate was just a simple heat source, not a fuel vapor. I did not understand that if we specify heat release rate and combustion, the simulated fire will vary with time, depending on the mixing and ventilation. In fact, if the mesh boundaries are not large enough, the unburned fuel vapor will be carried out of the model bounds and the model heat release rate will never reach the specified value.

Burner Fire

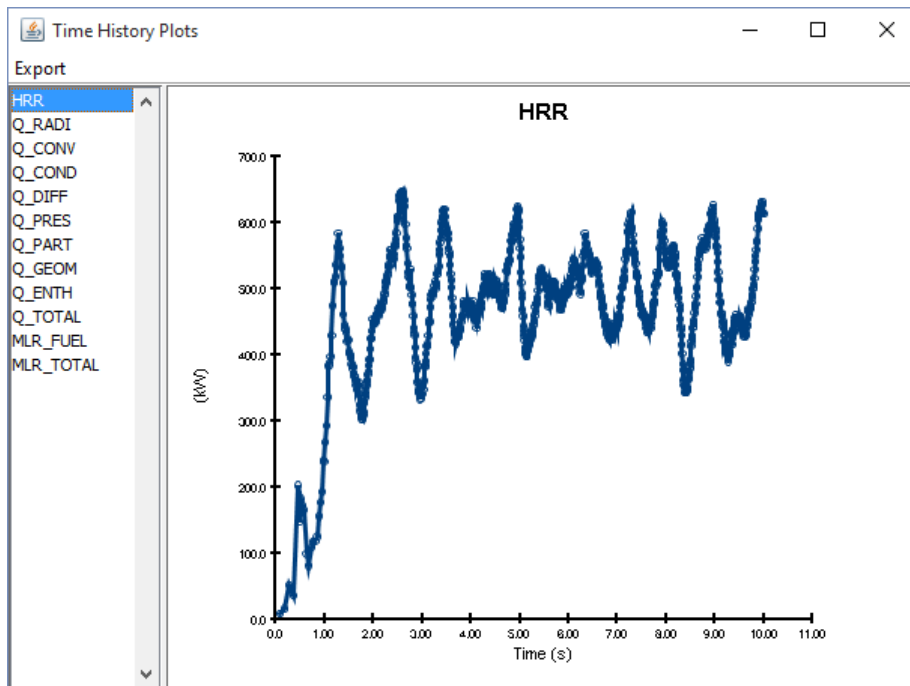


Figure 11: Calculated heat release rate. This is obtained by integrating the heat release rate per unit volume over the model domain. The specified HRR was 500 kW ($1000 \text{ kW/m}^2 \cdot 0.5 \text{ m}^2$). The calculated HRR varies with time as the fuel mixes with the air and results in combustion.

View Temperature Measurements

The output of the devices such as thermocouples can be plotted:

1. In the PyroSim window, on the **FDS** menu, click **Plot Time History Results**.
2. A dialog will appear showing the different types of 2D results that are available. Select **burner_devic.csv** and click **Open** to view the temperature device output.

Burner Fire

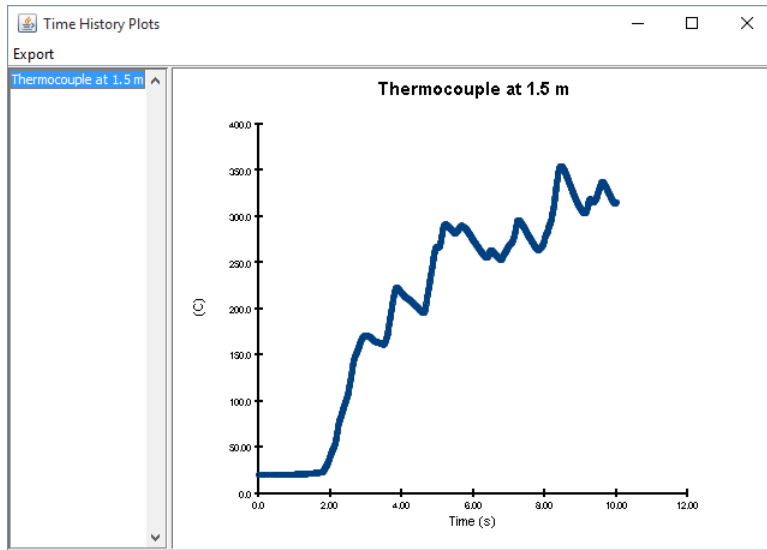


Figure 12. Temperature time history plot

References

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

McGrattan, Kevin, et al. 2014. *Fire Dynamics Simulator Technical Reference Guide (Sixth Edition)*. Gaithersburg, Maryland, USA, July 2014. NIST Special Publication 1018-6.

McGrattan, Kevin, et al. 2014. *Fire Dynamics Simulator User's Guide (Sixth Edition)*. Gaithersburg, Maryland, USA, July 2014. NIST Special Publication 1019-6.

FDS Input File

Following is a listing of the FDS input file. You can either save this as a text file and import into PyroSim, or you can:

1. Open PyroSim and on the **File** menu, click **New**.
2. Select and copy the entire listing below.
3. On the PyroSim **Edit** menu, click **Paste**. You will receive a warning that Simulation Parameters already exists in the model, click **Yes to All** to replace all existing data with the FDS file data.
4. Click the **Reset** tool.
5. Save and run.

Paste is great, but please work through the example.

```
burner.fds
Generated by PyroSim - Version 2015.4.1117
Nov 17, 2015 2:21:33 PM

&HEAD CHID='burner'/
&TIME T_END=10.0/
&DUMP RENDER_FILE='burner.gel', COLUMN_DUMP_LIMIT=.TRUE., DT_RESTART=300.0/

&MESH ID='Mesh01', IJK=20,20,30, XB=-1.0,1.0,-1.0,1.0,0.0,3.0/

&REAC ID='POLYURETHANE_GM27',
      FYI='SFPE Handbook, GM27',
      FUEL='REAC_FUEL',
      C=1.0,
      H=1.7,
      O=0.3,
      N=0.08,
      CO_YIELD=0.042,
      SOOT_YIELD=0.198/

&DEVC ID='Thermocouple at 1.4 m', QUANTITY='THERMOCOUPLE', XYZ=0.0,0.0,1.5/
&SURF ID='Fire',
      COLOR='RED',
      HRRPUA=1000.0/

&OBST XB=-0.5,0.5,-0.5,0.5,0.0,0.2, SURF_ID='INERT'/ Fire Obstruction

&VENT SURF_ID='OPEN', XB=1.0,1.0,-1.0,1.0,0.0,3.0/ Mesh Vent: Mesh01 [XMAX]
&VENT SURF_ID='OPEN', XB=-1.0,-1.0,-1.0,1.0,0.0,3.0/ Mesh Vent: Mesh01 [XMIN]
&VENT SURF_ID='OPEN', XB=-1.0,1.0,1.0,1.0,0.0,3.0/ Mesh Vent: Mesh01 [YMAX]
&VENT SURF_ID='OPEN', XB=-1.0,1.0,-1.0,-1.0,0.0,3.0/ Mesh Vent: Mesh01 [YMIN]
&VENT SURF_ID='OPEN', XB=-1.0,1.0,-1.0,1.0,3.0,3.0/ Mesh Vent: Mesh01 [ZMAX]
&VENT SURF_ID='Fire', XB=0.0,0.5,-0.5,0.5,0.2,0.2/ Fire Vent

&SLCF QUANTITY='TEMPERATURE', PBY=0.0/
&SLCF QUANTITY='VELOCITY', VECTOR=.TRUE., PBY=0.0/
&SLCF QUANTITY='VOLUME FRACTION', SPEC_ID='AIR', PBY=0.0/
```



```
&SLCF QUANTITY='VOLUME FRACTION', SPEC_ID='PRODUCTS', PBY=0.0/  
&SLCF QUANTITY='VOLUME FRACTION', SPEC_ID='REAC_FUEL', PBY=0.0/  
  
&TAIL /
```