PyroSim

# Burner Fire

THUNDERHEAD ENGINEERING

2019

# 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*** (HRR) that defines the size of the fire. When HRR is specified, FDS uses the reaction energy to calculate the corresponding fuel mass flow rate from the surface.

Once the reaction and heat release rate are defined, gaseous fuel flows 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.

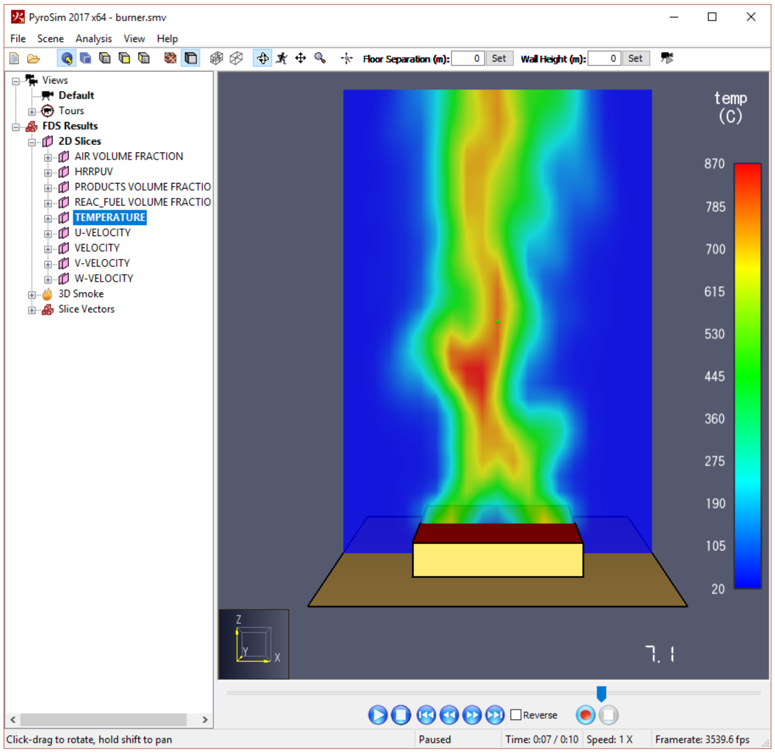



**Figure 1. Burner fire in this example**

This tutorial demonstrates how to:

- Create a burner fire.

- Add a thermocouple.
- Add a slice plane for temperature visualization.
- View 3D results using 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.

## 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.

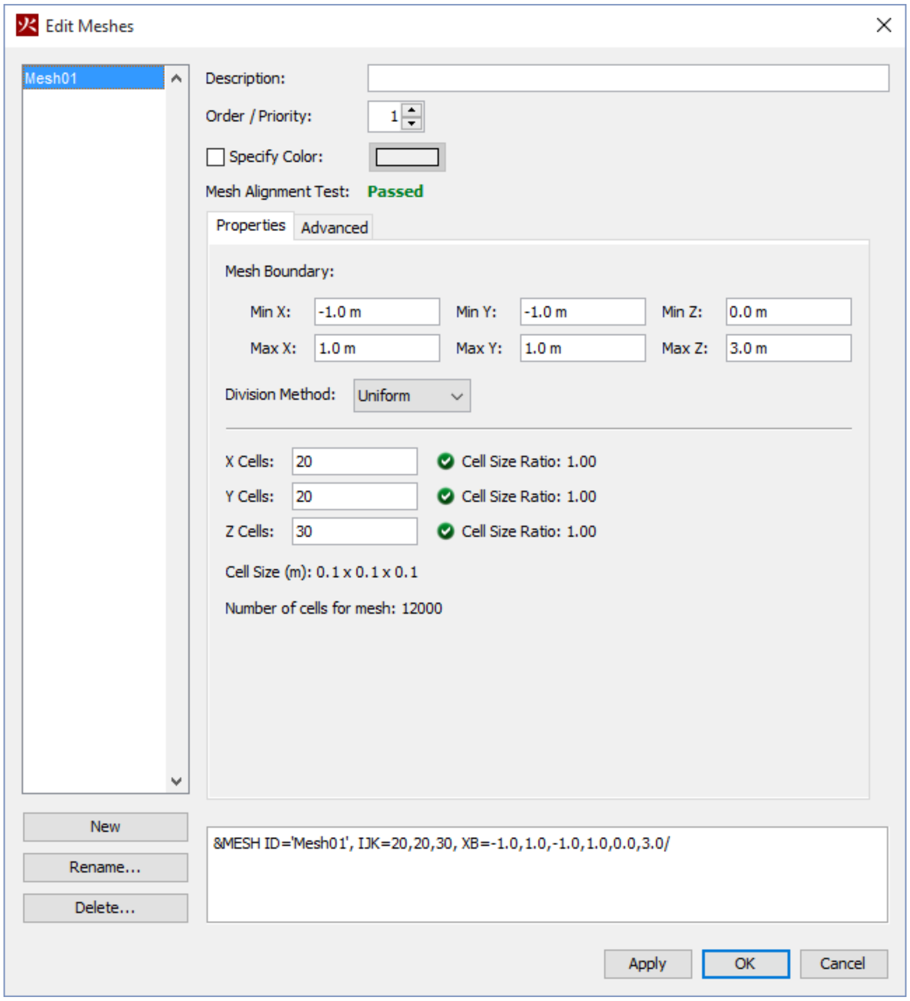



**Figure 2. Creating the mesh**

## 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 **POLYURETHANE_GM27** 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 fuel at a rate corresponding to 1000 kW/m$^2$. By default, all surfaces in FDS are INERT and remain at a fixed temperature (usually ambient). We want to specify the temperature of the burner surface, but the possible range is large. A pool fire may remain near the boiling point of the liquid (50-100 °C), ignition temperatures for wood range from 200-700 °C (Babrauskas, 2001), and a porous burner with natural gas as fuel can range from 530-750 °C (FDS Validation Guide based on McCaffre, 1979). We will assume 500 °C.

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, for **Surface Type** select **Burner**.
5. Click **OK** to create the new burner surface. The default heat release values are satisfactory (Figure 3).
6. On the **Thermal** tab, select **Fixed Temperature** for **Boundary Condition Model**. In the **Surface Temperature** box type **500**, and in the **Emissivity** box type **0.9**.
7. Click **OK** to save changes and close the **Edit Surfaces** dialog.

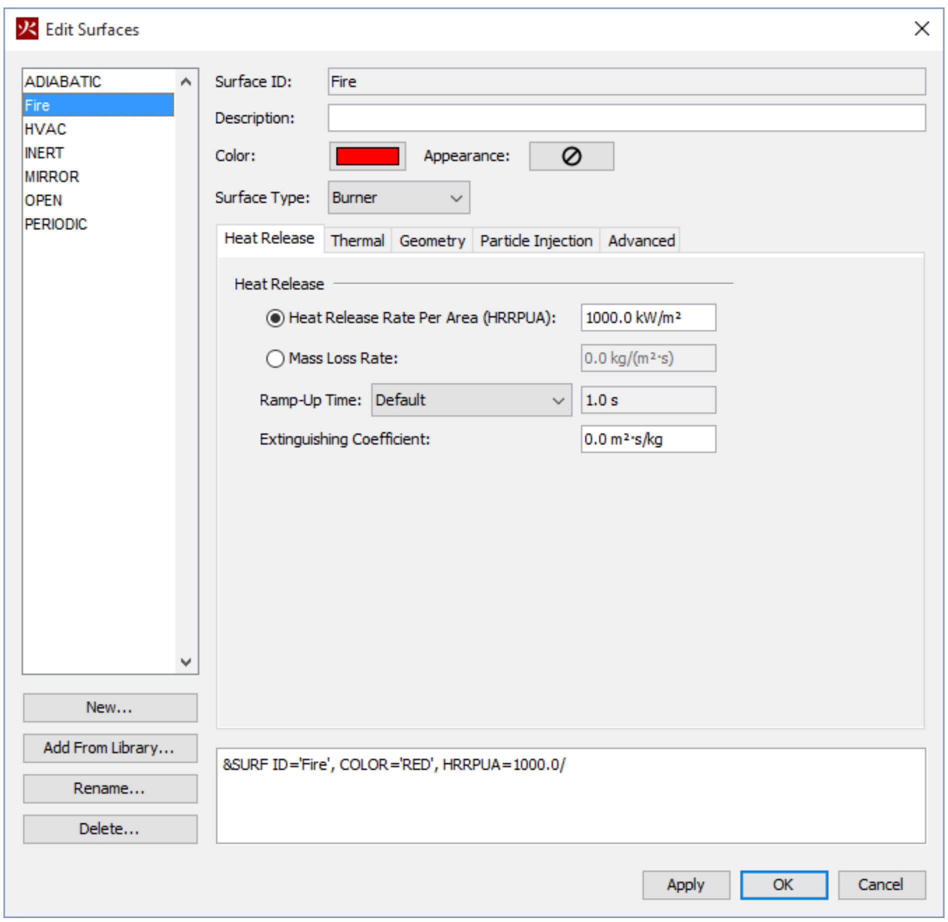



**Figure 3. Default parameters for the burner surface**

## Create the Fire

To locate the fire (fuel source) in our model, we create an obstruction and assign the fire surface to the top of the obstruction. 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 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 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 1.0 $m^2$, so the total heat release rate will be 1000 kW.

## Open Sides and Top of the Mesh

We want the sides and top of the mesh to be open to air flow. 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.

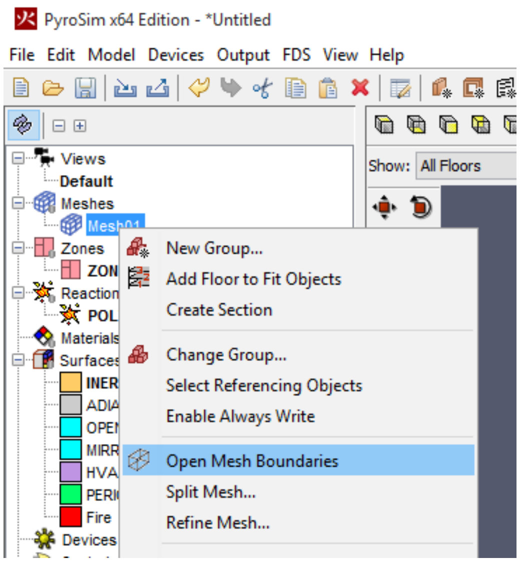



**Figure 4: Create open mesh boundaries**

## 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 **2D 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, select **NO**.
6. In the **Cell Centered?** column, select **NO**.
7. Click **OK** to create the slice plane.
8. Repeat, but this time plot **Heat Release Rate per Unit Volume**.
9. Repeat, but this time plot **Velocity** and select **YES** for vectors.
10. Now add a plot for air volume fraction. 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 and cell centered. Press the **ENTER** key for a new line.
11. Repeat so that you are plotting the Volume Fraction of AIR, PRODUCTS, and REAC_FUEL. See Figure 5. These plots 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.

12. Click **OK** to close the **Animated Planar Slices** dialog.
13. On the toolbar, click the **Show Slices** button to toggle the slice planes on and off.

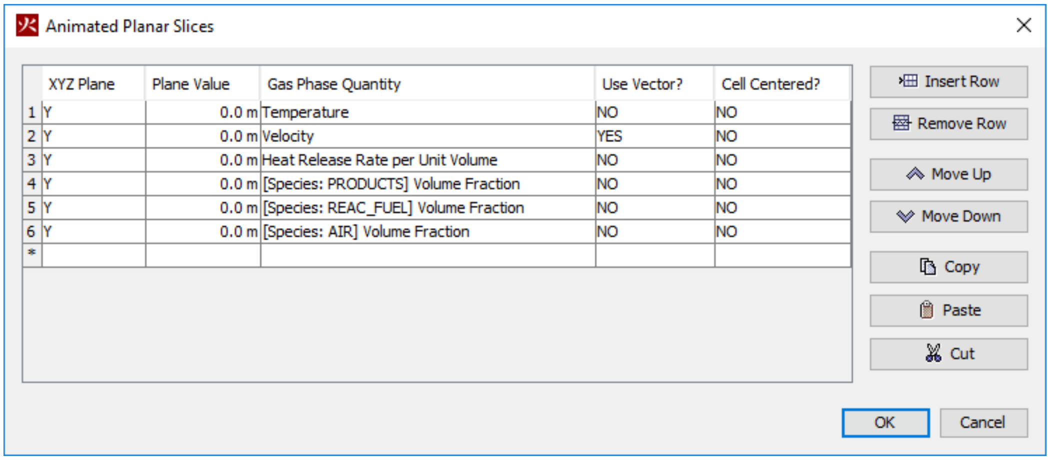



**Figure 5: Defining the slice plane plots.**

## Add a Steady State Line Temperature Measurement

Steady state line measurements simplify the output of a quantity on a line. By default, this device averages the output over the second half of a simulation.

1. On the **Output** menu, click **Statistics.**
2. Click **New**. For the **Quantity**, select **Temperature**. Click **OK**.
3. In the **Statistic Type** box, select **Temporal**.
4. Click **Line Statistics**.
5. Select **Steady State Profile**.
6. For **Num Points**, type **50**.
7. Change **Point 1 Z** to **0.2**.
8. Change **Point 2 Z** to **3.0**.
9. Click **OK** to create the device.
10. On the toolbar, click the **Statistics Regions** button to toggle the line on and off.

## 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 6, the burner is shown in red and the thermocouple as a yellow dot. The open vents are blue.

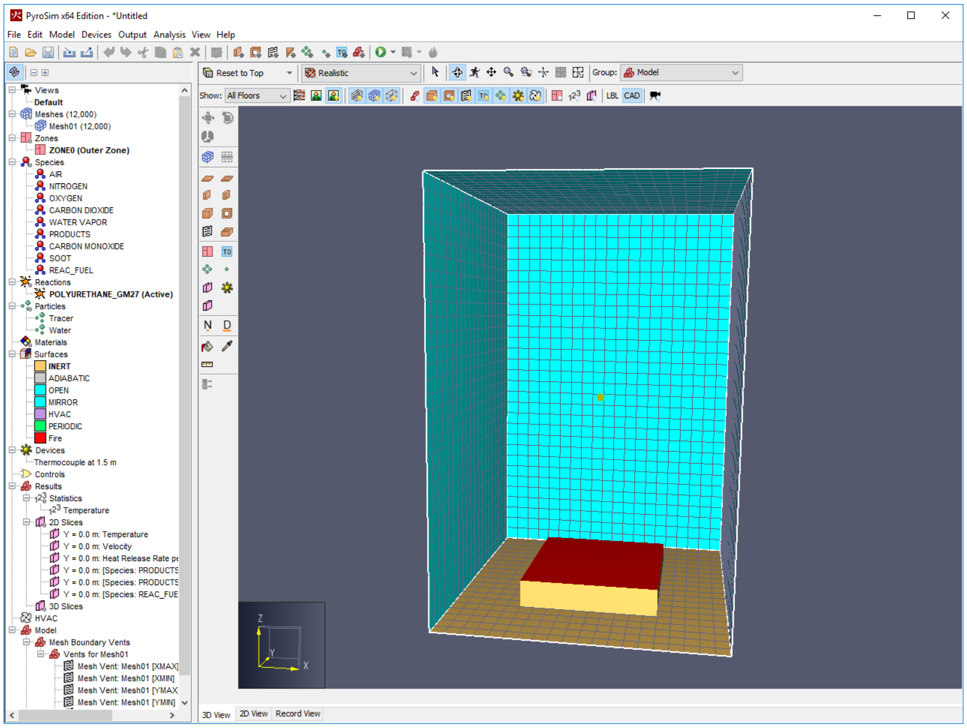


**Figure 6. The Burner Fire model**

## 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 Fire** folder and name the file **burner fire**.
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, PyroSim Results will start and display a 3D still image of the model.

## View Smoke in 3D

1. In the PyroSim Results window, double click **3D Smoke** to load the **HRRPUV** and **Soot Mass Fraction** datasets.
2. To clear the display in PyroSim Results, double click **3D Smoke** once more.

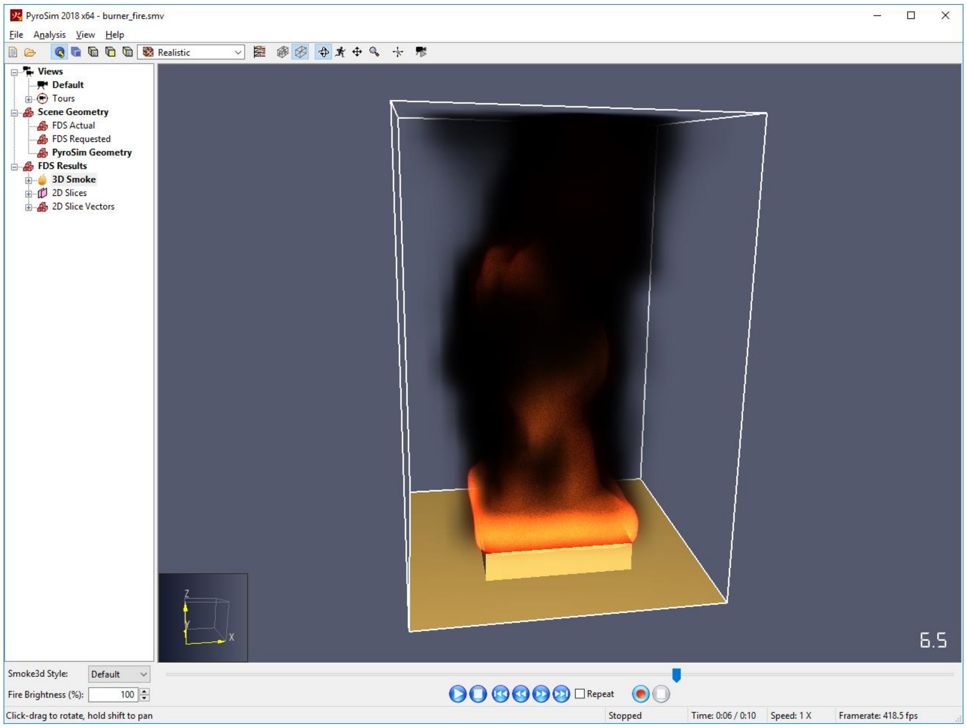
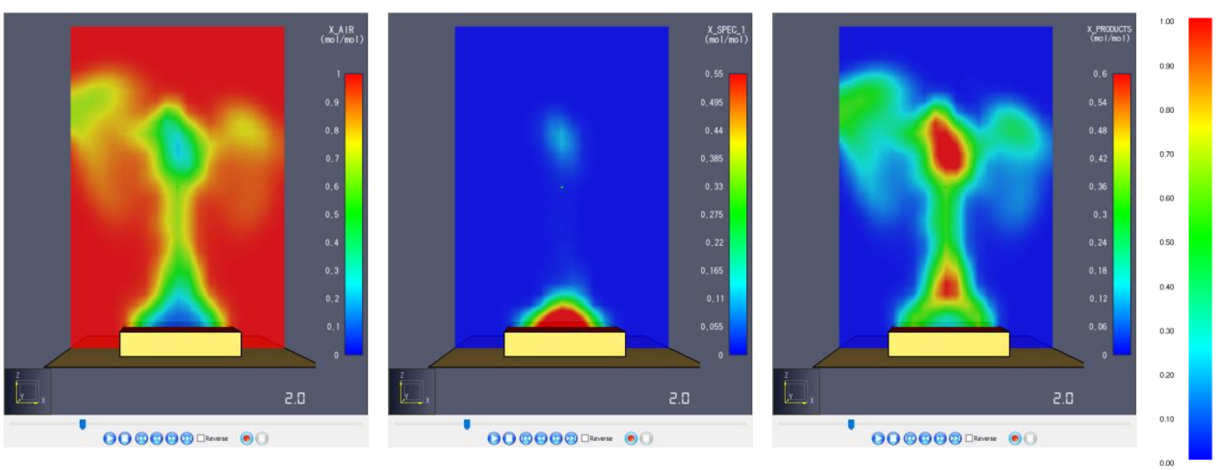



**Figure 7: A plot of smoke and heat release rate per unit volume.**

## Air, Fuel, and Products Contours

At any location, the volume fractions of air, fuel, and products will sum to 1. Typical contours of these quantities are shown in Figure 8. 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. A plot of Heat Release Rate per Unit Volume (HRRPUV) shows the locations of combustion.




a. Air b. Fuel c. Products

**Figure 8: 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 that burns. 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.

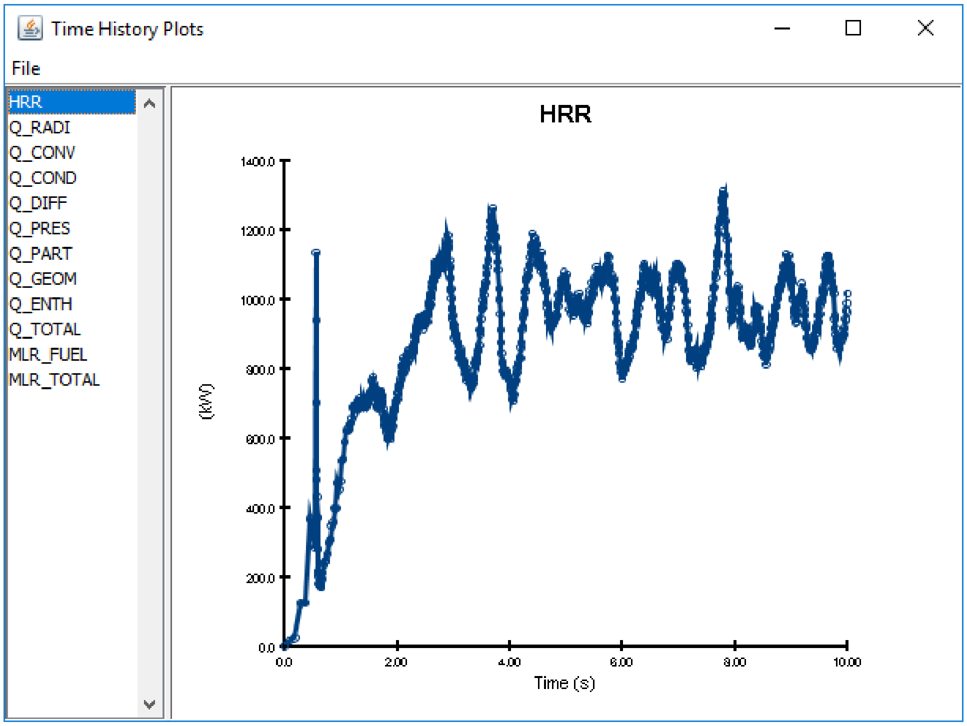



**Figure 9: Calculated heat release rate. This is obtained by integrating the heat release rate per unit volume over the model domain. The specified HRR was 1000 kW. The calculated HRR varies with time as the fuel mixes with the air and results in combustion.**

## View Temperature Measurements

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

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

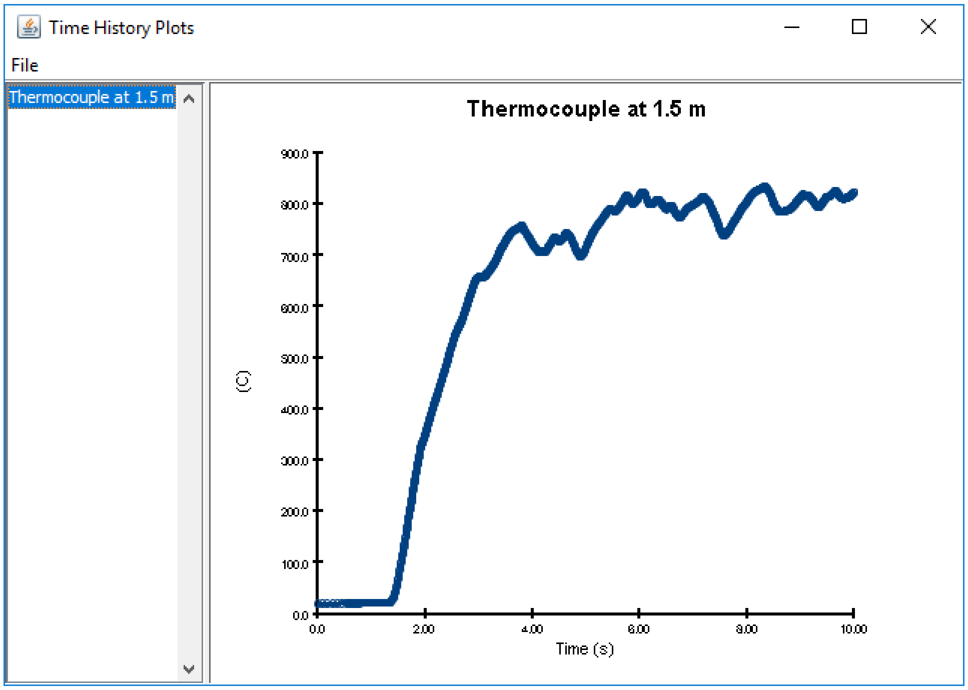



**Figure 10. Temperature time history plot**

## View Steady State Temperature Line Measurement

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

1. In the **burner_fire** folder and open the **burner_fire_line.csv** file.
2. Plot the output data (Figure 11).

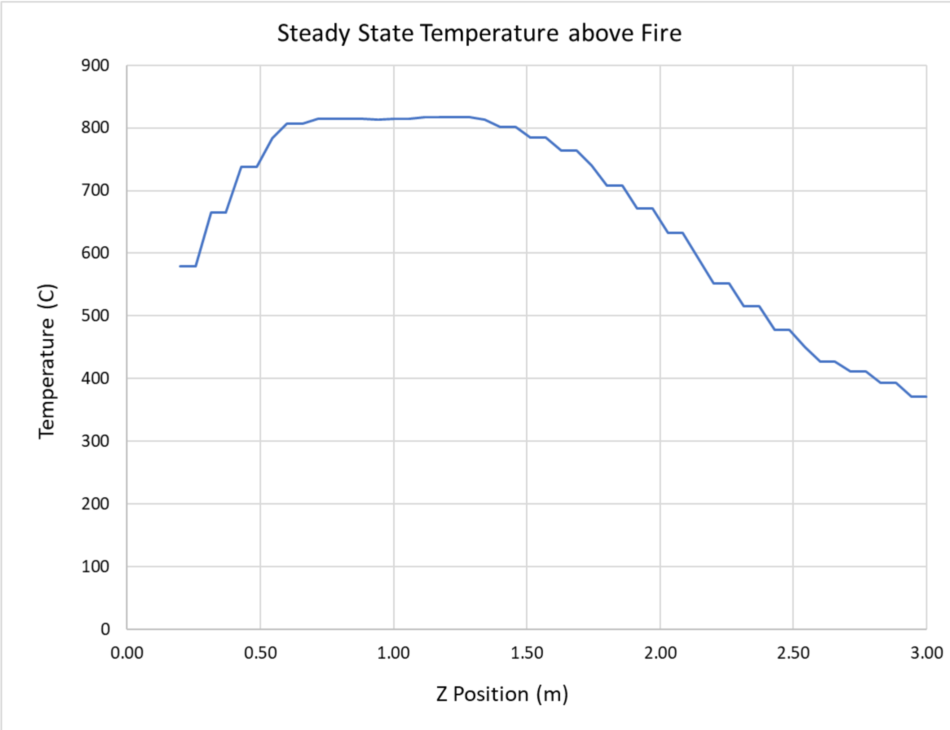



**Figure 11. Line plot of steady state temperature above fire**

# References

**Babrauskas, V., 2001.** *Ignition of Wood: A Review of the State of the Art*, pp. 71-88 in Interflam 2001, Interscience Communications Ltd., London.

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

**McCaffrey, Bernard, 1979.** *Purely Bouyant Diffusion Flames: Some Experimental Results.* National Bureau of Standards. NBSIR 79-1910.

**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.