Radiative Heat Transfer in a Utility Boiler

Introduction

In recent years, many studies have been conducted in the field of performance optimization of large power plant boilers. The main aims have been to extend the lifetime, increase the thermal efficiency, and reduce the pollutant emissions of the boilers. A good furnace design is the most important part in the energy conversion process in the boilers. A furnace is where the fuel is burnt and the chemical energy is converted into heat to be transferred into the water walls of steam boilers. The temperatures in fuel furnaces are high enough that the radiation becomes the most important mechanism in heat transfer. Due to complexity of the radiation mechanism and its dependence on the enclosure's geometry, there is no analytical solution except for very simple problems. This fact along with expensive experimental modeling leads researchers to develop numerical models for analyzing these enclosures. Three of the most attractive methods, as far as accuracy and computational requirement are concerned, are the discrete transfer, the discrete ordinates and the finite-volume methods.

Note: Solving this model requires approximately 22 GB of memory.

Model Definition

Of practical relevance is the radiative heat transfer in furnaces containing obstacles, such as protrusions and obstructions. In some applications, the thickness of the obstacles is very small as it occurs in utility boilers, where panels are often hanged in the radiation chamber.

In order to reduce the mesh (and then the computational cost), these obstructions are modeled as baffles (zero thickness). This study handles zero thickness obstacles containing an emitting-absorbing medium.

A three-dimensional enclosure resembling the combustion chamber of a utility boiler is modeled. The enclosure contains five baffles, as shown in Figure 1, which simulates the panels of superheaters suspended at the top of the combustion chamber.

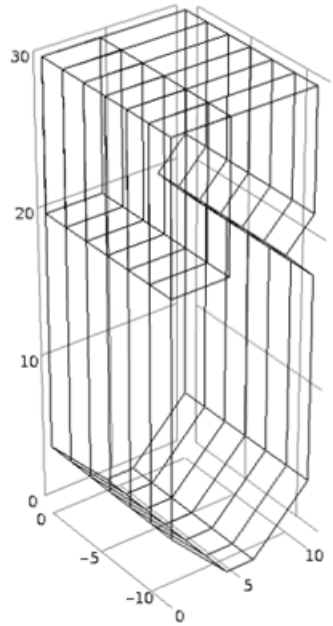

Figure 1: Utility Boiler with obstructions

In this example, the S8 discrete ordinate method was employed for predicting the heat flux on the side walls of enclosures and incident radiation distribution within the furnace. It results in a set of 80 discrete directions to represent radiative intensity transport.

The main assumption is using an existing uniform temperature and properties within the volume and surface zones, as proposed in Ref. 1. The temperature and emissivity of the boundaries, including the surface of the baffles, are taken as 800 K and 0.65, respectively, except at x = 10 m and for $22 \leq z \leq 30$ m, where the temperature was set

 ©2012 COMSOL

equal to 1200 K and a black-body surface is assumed. An emitting-absorbing medium is assumed, with the following distribution of temperature and absorption coefficient:

TABLE 1: DISTRIBUTION OF TEMPERATURE AND ADSORPTION COEFFICIENT

COORDINATE [M]	ABSORPTION COEFFICIENT[1/M]	TEMPERATURE [K]
$z \leq 5$	0.20	1600
$5 < z \leq 10$	0.25	2000
$10 < z \leq 20$	0.20	1600
$20 < z \leq 30$	0.18	1200

THERMAL ANALYSIS

The discrete-ordinates method (DOM) relies on the discrete representation of the directional dependence of the radiation intensity. The radiative transfer equation (RTE) is solved for a set of directions which span he total solid angle range of 4π around a point in space.

The RTE for this type of configuration can be written as:

$$\Omega \cdot \nabla I(\Omega, s) = \kappa I_b(T) - \beta I(\Omega, s) + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\Omega, \Omega')\phi(\Omega, \Omega')\partial\Omega'$$

where

- $I(\Omega,\sigma)$ is the radiative intensity at a given position s, following Ω direction
- T is the temperature
- κ, β, σ_s are absorption, extinction, and scattering coefficients, respectively
- $I_b(T)$ is the blackbody radiative intensity
- $\phi(\Omega, \Omega') = 1 + a_1\mu_0$ and $\mu_0 = \Omega..\Omega'$ is the cosine of the scattering angle.

The boundary intensities in the furnace walls are given physically by the effective emitted intensity plus reflected incident intensities into a respective direction.

$$I_{\text{bnd}}(\Omega) = \varepsilon_w I_b(T) + \frac{\rho_d}{\pi} q_{\text{out}} \qquad \text{for all } \Omega \text{ such that } \mathbf{n} \cdot \Omega < 0$$

where

- ε_w is the surface emissivity, which is in the range $[0, 1]$
- $\rho_d = 1 - \varepsilon_w$ is the diffusive reflectivity
- $\mathbf{n}$ is the outward normal vector

- q_{out} is the heat flux striking the wall:

$$q_{\text{out}} = \int_{\mathbf{n}\cdot\mathbf{\Omega}_j > 0} (\mathbf{n}\cdot\Omega) I(\Omega, s)\partial\Omega'$$

The above equations can be discretized in Cartesian coordinates for monochromatic or gray radiation as:

$$\Omega_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\Omega, \Omega')\phi(\Omega, \Omega')\partial\Omega'$$

The Sn approximation of the RTE in the m direction can be expressed as:

$$\Omega_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi}\sum_{j=1}^{n} w_j I_j \phi(\Omega_j, \Omega_i)$$

For a discrete direction, Ω_i, the values of $\Omega_{i,1}$, $\Omega_{i,2}$, and $\Omega_{i,3}$ define the direction cosines of Ω_i obeying the condition ${\Omega_{i,1}}^2 + {\Omega_{i,2}}^2 + {\Omega_{i,3}}^2 = 1$. The j index in the above equation denotes the direction of incoming radiation contributing to the direction Ω_i.

For a diffuse reflecting surface on a wall boundary, the boundary condition equation is transformed as:

$$I_{i,\,\text{bnd}} = \varepsilon_w I_b(T) + \frac{\rho_d}{\pi}\sum_{\mathbf{n}\cdot\mathbf{\Omega}_j > 0} w_j I_j \mathbf{n}\cdot\Omega_j \qquad \text{for all } \Omega_i \text{ such that } \mathbf{n}\cdot\Omega_i < 0$$

Results and Discussion

Figure 2 and Figure 3 show the predicted incident radiation surface plots. The maximum incident radiation occurs at the level where the temperature and absorbing coefficient of the medium are the highest (that is, at the boiler's burner level).

 ©2012 COMSOL

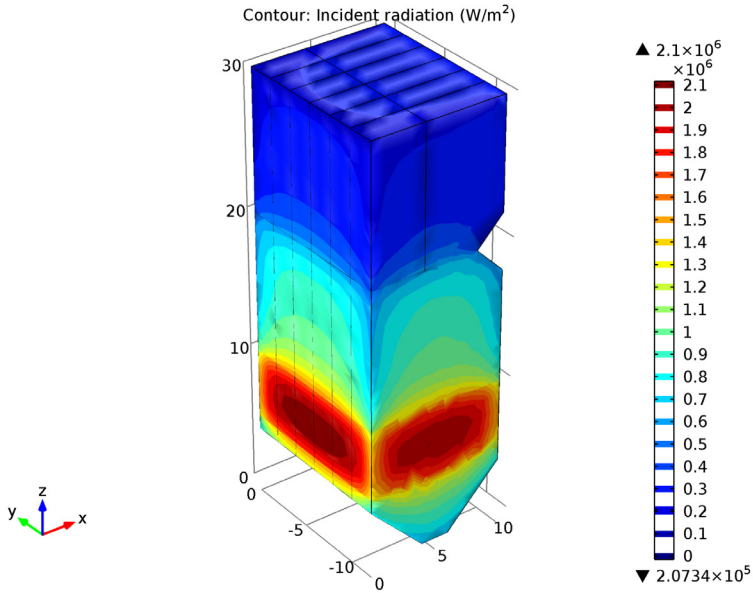

Figure 2: Incident radiation.

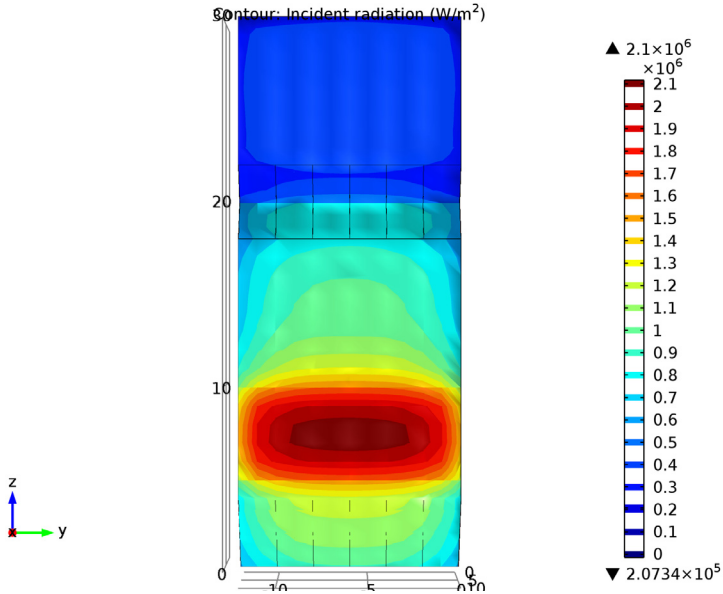

Figure 3: Incident radiation on the front of the boiler (W/m^2).

Predicted outgoing heat flux is shown on and in good agreement with published data (Ref. 1).

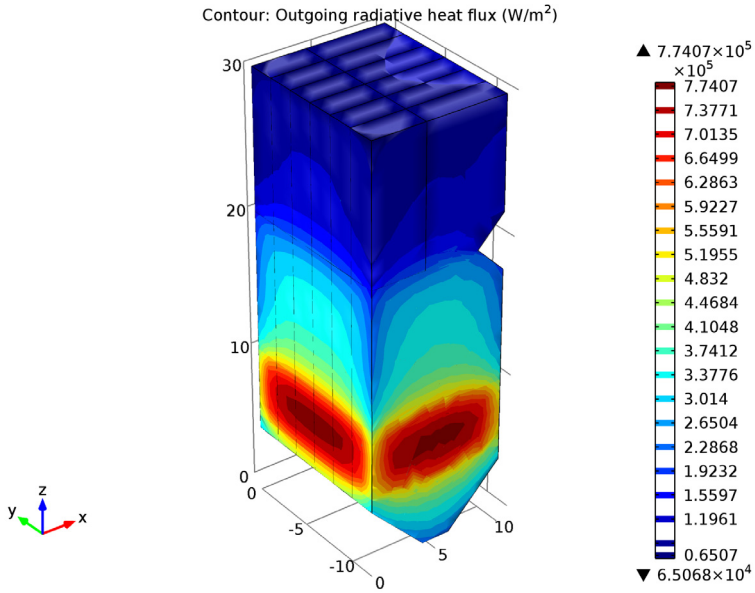

Figure 4: Outgoing heat flux on walls of the boiler (W/m^2)

Reference

1. P.J. Coelho, J.M. Goncalves, and M.G. Carvalho, "Modelling of Radiative Heat Transfer in Enclosures with Obstacles," *International Journal of Heat and Mass Transfer*, vol. 41, no. 4–5, pp. 745–756, 1998.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/boiler`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

 ©2012 COMSOL

3 In the **Add physics** tree, select **Heat Transfer>Radiation>Radiation in Participating Media (rpm)**.

4 Click **Add Selected**.

By default, the **Discrete ordinates method** setting is **S4**, which corresponds to 24 discrete angular directions. To obtain the maximum resolution of 80 directions, select **S8** from the list. Note, however, that this requires approximately 20 GB of memory.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
Thot	1200[K]	Temperature, hot surface zone
Tlow	800[K]	Temperature, cool surfaces
em	0.65	Emissivity, cool surfaces
scattC	0[1/m]	Scattering coefficient

Piecewise 1

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Functions>Piecewise**.

2 In the **Piecewise** settings window, locate the **Function Name** section.

3 In the **Function name** edit field, type T.

4 Locate the **Parameters** section. In the **Argument** edit field, type z.

5 Find the **Intervals** subsection. In the table, enter the following settings:

START	END	FUNCTION
0	5	1600
5	10	2000
10	20	1600
20	30	1200

6 Locate the **Units** section. In the **Arguments** edit field, type `m`.

7 In the **Function** edit field, type `K`.

Piecewise 2

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Functions>Piecewise**.

2 In the **Piecewise** settings window, locate the **Function Name** section.

3 In the **Function name** edit field, type `absC`.

4 Locate the **Parameters** section. In the **Argument** edit field, type `z`.

5 Find the **Intervals** subsection. In the table, enter the following settings:

START	END	FUNCTION
0	5	0.20
5	10	0.25
10	20	0.20
20	30	0.18

6 Locate the **Units** section. In the **Arguments** edit field, type `m`.

7 In the **Function** edit field, type `1/m`.

GEOMETRY 1

Work Plane 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 From the **Plane** list, choose **xz-plane**.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **yw** to 4.

5 In row **2**, set **yw** to 30.

6 Find the **Added segments** subsection. Click the **Add Linear** button.

7 Find the **Control points** subsection. In row **2**, set **xw** to 10.

 ©2012 COMSOL

8 Find the **Added segments** subsection. Click the **Add Linear** button.

9 Find the **Control points** subsection. In row **2**, set **yw** to 22.

10 Find the **Added segments** subsection. Click the **Add Linear** button.

11 Find the **Control points** subsection. In row **2**, set **xw** to 8 and **yw** to 20.

12 Find the **Added segments** subsection. Click the **Add Linear** button.

13 Find the **Control points** subsection. In row **2**, set **xw** to 10 and **yw** to 18.

14 Find the **Added segments** subsection. Click the **Add Linear** button.

15 Find the **Control points** subsection. In row **2**, set **yw** to 4.

16 Find the **Added segments** subsection. Click the **Add Linear** button.

17 Find the **Control points** subsection. In row **2**, set **xw** to 6 and **yw** to 0.

18 Find the **Added segments** subsection. Click the **Add Linear** button.

19 Find the **Control points** subsection. In row **2**, set **xw** to 4.

20 Find the **Added segments** subsection. Click the **Add Linear** button.

21 Find the **Control points** subsection. Click the **Close Curve** button.

Rectangle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 4.

4 In the **Height** edit field, type 10.

5 Locate the **Position** section. In the **yw** edit field, type 20.

Extrude 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

DISTANCES (M)
2
4
6
8
10
12

MATERIALS

Add a material to specify the absorption and scattering coefficients inside the boiler.

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Chamber` in the **New name** edit field.

4 Click **OK**.

5 In the **Material** settings window, locate the **Material Contents** section.

6 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Absorption coefficient	kappaR	`absC(z)`
Scattering coefficient	sigmaS	`scattC`

Analogously, specify the emissivity of the boiler walls using a material.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Walls` in the **New name** edit field.

4 Click **OK**.

5 In the **Material** settings window, locate the **Geometric Entity Selection** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 From the **Selection** list, choose **All boundaries**.

8 Locate the **Material Contents** section. In the table, enter the following settings:

PROPERTY	NAME	VALUE
Surface emissivity	epsilon_rad	`em`

RADIATION IN PARTICIPATING MEDIA

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

2 In the **Radiation in Participating Media** settings window, click to expand the **Advanced Settings** section.

3 From the P_{index} list, choose **1**.

 ©2012 COMSOL

Radiation in Participating Media 1

1 In the **Model Builder** window, under **Model 1>Radiation in Participating Media** click **Radiation in Participating Media 1**.

2 In the **Radiation in Participating Media** settings window, locate the **Model Inputs** section.

3 In the T edit field, type `T(z)`.

Opaque Surface 1

1 In the **Model Builder** window, under **Model 1>Radiation in Participating Media** right-click **Opaque Surface 1** and choose **Rename**.

2 Go to the **Rename Opaque Surface** dialog box and type `Low Temp` in the **New name** edit field.

3 Click **OK**.

4 In the **Opaque Surface** settings window, locate the **Model Inputs** section.

5 In the T edit field, type `Tlow`.

Opaque Surface 2

1 In the **Model Builder** window, right-click **Radiation in Participating Media** and choose the boundary condition **Opaque Surface**.

2 Right-click **Opaque Surface 2** and choose **Rename**.

3 Go to the **Rename Opaque Surface** dialog box and type `Hot Temp` in the **New name** edit field.

4 Click **OK**.

5 In the **Opaque Surface** settings window, locate the **Model Inputs** section.

6 In the T edit field, type `Thot`.

7 Locate the **Wall Settings** section. From the **Wall type** list, choose **Black wall**.

8 Select Boundaries 82, 84, 86, 88, 90, and 92 only.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 5 only.

Free Quad 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Quad**.

2 Select Boundary 2 only.

3 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The default plot groups show the incident radiation as a slice plot and the net radiative heat flux as a surface plot. Follow the instructions below to replace the slice plot for the incident radiation by a contour plot.

Incident Radiation (rpm)

1 In the **Model Builder** window, expand the **Results>Incident Radiation (rpm)** node.

2 Right-click **Slice 1** and choose **Delete**. Click **Yes** to confirm.

3 Right-click **Incident Radiation (rpm)** and choose **Contour**.

4 In the **Contour** settings window, locate the **Levels** section.

5 From the **Entry method** list, choose **Levels**.

6 In the **Levels** edit field, type `range(0e5,1e5,21e5)`.

7 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

8 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

9 Click the **Plot** button.

Rescale the zoom to get results shown in Figure 2.

10 Click the **Zoom Extents** button on the Graphics toolbar.

Switch the *yz* perspective to display results from Figure 3.

11 Click the **Go to YZ View** button on the Graphics toolbar.

 ©2012 COMSOL

Net Radiative Heat Flux (rpm)

Now, replace the net radiative heat flux by the outgoing radiative heat flux in the second graph.

1. In the **Model Builder** window, under **Results** right-click **Net Radiative Heat Flux (rpm)** and choose **Rename**.
2. Go to the **Rename 3D Plot Group** dialog box and type `Outgoing Radiative Heat Flux (rpm)` in the **New name** edit field.
3. Click **OK**.

Outgoing Radiative Heat Flux (rpm)

1. In the **Model Builder** window, expand the **Outgoing Radiative Heat Flux (rpm)** node.
2. Right-click **Surface 1** and choose **Delete**. Click **Yes** to confirm.
3. Right-click **Outgoing Radiative Heat Flux (rpm)** and choose **Contour**.
4. In the **Contour** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Radiation in Participating Media>Outgoing radiative heat flux (rpm.qout)**.
5. Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.
6. Locate the **Quality** section. From the **Smoothing** list, choose **None**.
7. Click the **Go to Default 3D View** button on the Graphics toolbar.

   Compare with Figure 4.

©2012 COMSOL