Radiation in ANSYS FLUENT
Overview

- ANSYS Fluent provides five radiation models that allow us to include radiation, with or without a participating medium, in heat transfer simulations.
- Heating or cooling of surfaces due to radiation and/or heat sources or sinks due to radiation within the fluid phase can be included in your model using one of the following radiation models.

  - Discrete Transfer Radiation Model (DTRM)
  - P1 Radiation Model
  - Rosseland Model
  - Surface-to-Surface (S2S)
  - Discrete Ordinates Model (DOM)

In addition to these radiation models, ANSYS Fluent also provides a solar load model that allows us to include the effects of solar radiation in the simulations.
We should include radiative heat transfer in our simulations when the radiant heat flux is large compared to the heat transfer rate due to convection or conduction.

\[ Q_{rad} = \sigma \left( T_{max}^4 - T_{min}^4 \right) \]
Typical applications

- Radiative heat transfer from flames
- Surface-to-surface radiant heating or cooling
- Coupled radiation, convection, and/or conduction heat transfer
- Radiation through windows in HVAC (Heating, ventilation, and air conditioning) applications, and cabin heat transfer analysis in automotive and aircraft applications
- Radiation in glass processing, glass fiber drawing, and ceramic processing (materials processing)
how radiant ceiling heating and cooling work
The radiative transfer equation (RTE) for an absorbing, emitting, and scattering medium at position $\vec{r}$ in the direction $\vec{s}$ is

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = a n^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s'}) \Phi(\vec{s} \cdot \vec{s'}) d\Omega'$$

- $\vec{r}$ = position vector
- $\vec{s}$ = direction vector
- $\vec{s'}$ = scattering direction vector
- $s$ = path length
- $a$ = absorption coefficient
- $n$ = refractive index
- $\sigma_s$ = scattering coefficient
- $\sigma$ = Stefan-Boltzmann constant ($5.669 \times 10^{-8} \text{ W/m}^2\text{K}^4$)
- $I$ = radiation intensity, which depends on position ($\vec{r}$) and direction ($\vec{s}$)
- $T$ = local temperature
- $\Phi$ = phase function
- $\Omega'$ = solid angle

$a + \sigma_s$ is extinction coefficient of the medium. The refractive index $n$ is important when considering radiation in semi-transparent media.
1. Discrete Transfer Radiation Model

Main assumption – Radiation leaving a surface element within a specified range of solid angles can be approximated by a single ray.

Uses a ray-tracing technique to integrate radiant intensity along each ray:

\[
\frac{dI}{ds} + a I = \frac{a \sigma T^4}{\pi}
\]

Advantages:
- Relatively simple model.
- Can increase accuracy by increasing number of rays.
- Applicable to a wide range of optical thicknesses.

Limitations:
- Assumes all surfaces are diffuse.
- Effect of scattering not included.
- Solving a problem with a large number of rays is CPU-intensive.
Main assumption – The directional dependence in RTE is integrated out, resulting in a diffusion equation for incident radiation.

✓ Advantages:
  ▪ Radiative transfer equation easy to solve with **small CPU demand**.
  ▪ Includes effect of scattering. Thus effects of particles, droplets, and soot can be included.
  ▪ Works reasonably well for applications where the optical thickness is large (e.g., combustion).

✓ Limitations:
  ▪ Assumes all surfaces are diffuse.
  ▪ May result in loss of accuracy (depending on the complexity of the geometry) if the optical thickness is small.
  ▪ Tends to overpredict radiative fluxes from localized heat sources or sinks.
  ▪ Boundary conditions might be tricky.
3. Rosseland Model

✓ Advantages:
   The Rosseland model has two advantages over the P-1 model. Since it does not solve an extra transport equation for the incident radiation (as the P-1 model does), the Rosseland model is faster than the P-1 model and requires less memory.

✓ Limitations:
   The Rosseland model can be used only for optically thick media. Note also that the Rosseland model is not available when the density-based solver is being used; it is available with the pressure-based solver, only.
The S2S radiation model can be used for modeling radiation in situations where there is no participating media.

- For example, spacecraft heat rejection system, solar collector systems, radiative space heaters, and automotive underhood cooling.
- S2S is a view-factor based model.
- Non-participating media is assumed.

Limitations:

- The S2S model assumes that all surfaces are diffuse.
- The implementation assumes gray radiation.
- Storage and memory requirements increase very rapidly as the number of surface faces increases. Memory requirements can be reduced by using clusters of surface faces.
- Clustering does not work with sliding meshes or hanging nodes.
- Not to be used with periodic or symmetry boundary conditions.
5. Discrete Ordinates Model

The radiative transfer equation is solved for a discrete number of finite solid angles, $\sigma_s$:

$$\frac{\partial I}{\partial x_i} + (a + \sigma_s) I(r,s) = a n^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(r,s') \Phi(s \cdot s') d\Omega'$$

**Advantages:**
- Conservative method leads to heat balance for coarse discretization. Accuracy can be increased by using a finer discretization.
- Most comprehensive radiation model: Accounts for scattering, semi-transparent media, specular surfaces, and wavelength-dependent transmission using banded-gray option.

**Limitations**
- Solving a problem with a large number of ordinates is CPU-intensive.
Gray radiation and non-gray radiation

A gray surface has a spectral emissivity and absorptivity that is not dependent on the wavelength of the radiation. Gray radiation refers to the type of radiation one would expect to see from a gray surface.

The gray model offers a huge simplification to real problems. The gray model should always be used as a first option wherever possible. If more accurate solutions are desired, then it may be appropriate to use a non-gray model.
Solar Load Model

✓ Solar load model
  ▪ Ray tracing algorithm for solar radiant energy transport: Compatible with all radiation models
  ▪ Available with parallel solver (but ray tracing algorithm is not parallelized)
  ▪ 3D only

✓ Specifications
  ▪ Sun direction vector
  ▪ Solar intensity (direct, diffuse)
  ▪ Solar calculator for calculating direction and direct intensity using theoretical maximum or “fair weather conditions”
  ▪ Transient cases
Choosing a radiation model

For certain problems, one radiation model may be more appropriate than others.

- **Computational effort** – P1 gives reasonable accuracy with less effort.
- **Accuracy** – DTRM and DOM more accurate.
- **Optical thickness** – DTRM/DOM for optically thin media \((\alpha L \ll \text{small})\); P1 better for optically thick media.
- **Scattering** – P1 and DOM account for scattering.
- **Particulate effects** – P1 and DOM account for radiation exchange between gas and particulates.
- **Localized heat sources** – DTRM/DOM with sufficiently large number of rays/ordinates is more appropriate.
Main steps

The procedure for setting up and solving a radiation problem is outlined below, and described in detail in referenced sections. Steps that are relevant only for a particular radiation model are noted as such. The steps that are pertinent to radiation modeling, are shown here.
Related settings

- Set the appropriate radiation parameters.
  - If we are modeling non-gray radiation using the P-1 model, define the non-gray radiation parameters as described in *Setting Up the P-1 Model with Non-Gray Radiation*.
  - If we are using the DTRM, define the ray tracing as described in *Setting Up the DTRM*.
  - If we are using the S2S model, define the surface clusters and view factors settings and compute or read the view factors as described in *Setting Up the S2S Model*.
  - If you are using the DO model, choose DO/Energy Coupling if desired, define the angular discretization as described in *Setting Up the DO Model* and, if relevant, define the non-gray radiation parameters as described in *Defining Non-Gray Radiation for the DO Model*.
Radiation in ANSYS Fluent

Related settings

- Define the material properties as described in *Defining Material Properties for Radiation*.
- Define the boundary conditions as described in *Defining Boundary Conditions for Radiation*. If the model contains a semi-transparent medium, see the information below on setting up semi-transparent media.
- Set the parameters that control the solution (DTRM, DO, S2S, and P-1 only) as described in *Solution Strategies for Radiation Modeling*.
- Run the solution as described in *Running the Calculation*.
- Postprocess the results as described in *Postprocessing Radiation Quantities*. 
In this example, combined radiation and natural convection are solved in a three-dimensional square box on a mesh consisting of hexahedral elements. This example demonstrates how to do the following:

- Use the surface-to-surface (S2S) radiation model in ANSYS FLUENT.
- Set the boundary conditions for a heat transfer problem involving natural convection and radiation.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of wall temperature, surface cluster ID, and radiation heat flux.
Problem Description

A three-dimensional box $0.25\,\text{m} \times 0.25\,\text{m} \times 0.25\,\text{m}$ has a hot wall of aluminum at $473.15\,\text{K}$. All other walls are made of an insulation material and are subject to radiative and convective heat transfer to the surrounding environment, which is at $293.15\,\text{K}$. Gravity acts downwards. The medium contained in the box is assumed not to emit, absorb, or scatter radiation. All walls are gray. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model available in ANSYS FLUENT.

The working fluid has a Prandtl number of approximately $0.71$, and the Rayleigh number based on $L$ ($0.25\,\text{m}$) is $10^8$. This means the flow is most likely laminar. The Planck number is $0.006$, and measures the relative importance of conduction to radiation.
Main steps

Step 1: Mesh
Step 2: General Settings
Step 3: Models
Step 4: Materials
Step 5: Boundary Conditions
Step 6: Solution
Step 7: Postprocessing
Step 8: Compare the Contour Plots after Varying Radiating Surfaces
Step 9: S2S Definition, Solution, and Postprocessing with Partial Enclosure
Step 1: Mesh
Step 1: Mesh-geometry

Geometry > surface > standard_shapes
Step 1: Mesh-creating part
● Step 1: Mesh-creating block

Structured mesh
Step 1: Mesh-setting nodes
Step 1: Mesh-check mesh quality
- **Step 1: Mesh**

(40×40×40)  (80×80×80)
Step 1: Mesh-output mesh
Step 1: Mesh

The mesh size will be reported as 64,000 cells ($40 \times 40 \times 40$).
Generated by Software ICEM. Using Fluent 17.0 read the mesh.

Check the mesh. ANSYS FLUENT will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.
Step 2: General Settings

a. Retain the default settings in the Solver group box.

b. Enable the Gravity option.

c. Enter -9.81 m/s² for Y in the Gravitational Acceleration group box.
1. Enable the energy equation.

2. Set up the Surface to Surface (S2S) radiation model.
The surface-to-surface (S2S) radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by the geometric function called “view factor”.

The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, emission, or scattering of radiation by the medium can be ignored. Therefore only “surface-to-surface” radiation is considered for analysis.
Step 3: Models

Click the **Settings...** button to open the **View Factors and Clustering** dialog box. You will define the view factor and cluster parameters.

The S2S radiation model is computationally very expensive when there are a large number of radiating surfaces. The number of radiating surfaces is reduced by clustering surfaces into surface “clusters”. The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.

**For a small problem, the default value of 1 for Faces per Surface Cluster for Flow Boundary Zones is acceptable.** For a large problem you can increase this number to reduce the memory requirement for the view factor file that is saved in a later step. This may also lead to some reduction in the computational expense. However, this is at the cost of some accuracy.
Step 4: Materials

Gas properties

- Select incompressible-ideal-gas from the Density drop-down list.
- Enter 1021 J/kgK for Cp (Specific Heat).
- Enter 0.0371 W/mK for Thermal Conductivity.
- Enter 2.485e-05 kg/ms for Viscosity.
- Retain the default value of 28.966 kg/kmol for Molecular Weight.
- Click Change/Create and close the Create/Edit Materials dialog box.
Solid properties

Materials → Solid → Create/Edit...

a. Enter **insulation** for **Name**.
b. Delete the entry in the **Chemical Formula** field.
c. Enter **50 kg/m³** for **Density**.
d. Enter **800 J/kgK** for **Cp (Specific Heat)**.
e. Enter **0.09 W/mK** for **Thermal Conductivity**.
Set the boundary conditions for the front wall (\textbf{w-high-x}).
Step 5: Boundary Conditions

Copy boundary conditions to define the side walls \textbf{w-high-z} and \textbf{w-low-z}.
Set the boundary conditions for the front wall \( w\text{-low-x} \).

a. Click the Thermal tab and select Temperature from the Thermal Conditions list.
b. Retain the default selection of \textit{aluminum} from the Material Name drop-down list.
c. Enter 473.15 K for Temperature.
d. Enter 0.95 for Internal Emissivity.
Step 5: Boundary Conditions

Set the boundary conditions for the top wall (w-high-y).

Copy boundary conditions to define the bottom wall (w-low-y).
1. Set the solution parameters.

- a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.
- b. Select Body Force Weighted from the Pressure drop-down list in the Spatial Discretization group box.
- c. Retain the default selection of Second Order Upwind from the Momentum and Energy drop-down lists.
**Step 6: Solution**

2. Initialize the solution.

   - Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** list.
   - Click **Initialize**.
Step 6: Solution

3. Define a surface monitor to aid in judging convergence.

Define a surface monitor to aid in judging convergence. It is good practice to use monitors of physical solution quantities together with residual monitors when determining whether a solution is converged. In this step you will set up a surface monitor of the average temperature on the z=0 plane.

a. Creating a surface

b. Create the surface monitor.
Step 6: Solution

- a. Select User Specified from the Time Step Method list.
- b. Retain the default value of 1 for Pseudo Time Step.
- c. Enter 300 for Number of Iterations.
- d. Click Calculate.
Step 6: Solution

Temperature Surface Monitor

The surface monitor history shows that the average temperature on zz_center_z has stabilized, thus confirming that the solution has indeed reached convergence. You can view the behavior of the residuals by selecting Scaled Residuals from the graphics window drop-down list.
Step 7: Postprocessing

1. Create a new surface, `zz_x_side`, which will be used later to plot wall temperature.

   **Surface → Line/Rake...**

   ![Line/Rake Surface dialog box]

   - **a.** Enter (-0.125, 0, 0.125) for \((x_0, y_0, z_0)\), respectively.
   - **b.** Enter (0.125, 0, 0.125) for \((x_1, y_1, z_1)\), respectively.
   - **c.** Enter `zz_x_side` for New Surface Name.
   - **d.** Click Create and close the Line/Rake Surface dialog box.
Step 7: Postprocessing

2. Display contours of static temperature.

- Graphics and Animations → Contours → Set Up...

a. Enable the Filled option in the Options group box.
b. Select Temperature... and Static Temperature from the Contours of drop-down lists.
c. Select zz_center_z from the Surfaces selection list.
d. Enable the Draw Mesh option in the Options group box to open the Mesh Display dialog box.
i. Select Outline from the Edge Type list.
ii. Click Display and close the Mesh Display dialog box.
e. Disable the Auto Range option.
f. Enter 421 K for Min and 473.15 K for Max.
A regular check for most buoyant cases is to look for evidence of stratification in the temperature field. This is observed as nearly horizontal bands of similar temperature. These may be broken or disturbed by buoyant plumes. For this case you can expect reasonable stratification with some disturbance at the vertical walls where the air is driven around.
### Step 7: Postprocessing

3. Display contours of wall temperature (outer surface).

#### Graphics and Animations → Contours → Set Up...

![Contours settings](image)

- **a.** Ensure that the Filled option is enabled in the Options group box.
- **b.** Disable the Node Values option.
- **c.** Select Temperature... and Wall Temperature (Outer Surface) from the Contours of drop-down lists.
- **d.** Select all surfaces except default-interior and zz_x_side in the Surfaces selection list.
- **e.** Disable the Auto Range and Draw Mesh options.
- **f.** Enter 413 K for Min and 473.15 K for Max.
In the context of the Wall Temperature field variables, Outer Surface refers to whichever surface of the wall is in contact with the fluid domain. This may or may not correspond to an “outer” wall of the problem geometry.
• Step 7: Postprocessing


Graphics and Animations → Contours → Set Up...

a. Ensure that the **Filled** option is enabled in the **Options** group box.
b. Select **Wall Fluxes...** and **Radiation Heat Flux** from the **Contours of** drop-down list.
c. Make sure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.
d. Click **Display**.
e. Close the **Contours** dialog box.
Step 7: Postprocessing

Contours of Radiation Heat Flux

Contour levels:
- 8.82e+02
- 8.25e+02
- 7.68e+02
- 7.10e+02
- 6.53e+02
- 5.96e+02
- 5.39e+02
- 4.82e+02
- 4.25e+02
- 3.68e+02
- 3.10e+02
- 2.53e+02
- 1.96e+02
- 1.39e+02
- 8.18e+01
- 2.47e+01
- 3.25e+01
- 8.96e+01
- 1.47e+02
- 2.04e+02
- 2.61e+02

Contours of Radiation Heat Flux (W/m²)

ANSYS FLUENT (3d, pons, lam)
5. Display vectors of velocity magnitude.

Graphics and Animations → Vectors → Set Up...

- Options:
  - Global Range
  - Auto Range
  - Clip to Range
  - Auto Scale
  - Draw Mesh

- Style:
  - arrow

- Scale:
  - Min: 0
  - Max: 0

- Surfaces:
  - w-high-y
  - w-high-z
  - w-low-x
  - w-low-y
  - w-low-z

- Surface Name Pattern:
  - Match

- Surface Types:
  - axis
  - clip-surf
  - exhaust-fan
  - fan
Step 7: Postprocessing

6. Compute view factors and radiation emitted from the front wall (w-high-x) to all other walls.

   - Ensure that the View Factors option is enabled in the Report Options group box.
   - Enable the Incident Radiation option.
   - Select w-high-x from the From selection list.
   - Select all zones except w-high-x from the To selection list.
   - Click Compute and close the S2S Information dialog box.
7. Compute the heat transfer rate.

Total heat transfer rate

Radiation heat transfer rate
8. Display the temperature profile for the side wall.

- Select Plots → XY Plot → Set Up...

![Temperature Profile Along Outer Surface](image-url)
Step 8: Changing radiation settings

Step 8: Compare the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.
   - **Models → Radiation → Edit...**
     a. Click the **Settings...** button to open the View Factors and Clustering dialog box.
     b. Click the **Compute/Write/Read...** button to open the Select File dialog box and to compute the view factors.

2. Initialize the solution.

3. Start the calculation by requesting 300 iterations.

Repeat the procedure outlined in Step 8: for 100, 400, 800, and 1600 faces per surface cluster.
Step 8: Changing radiation settings

Contours of Wall Temperature (Outer Surface): 1 Face per Surface Cluster

Contours of Wall Temperature (Outer Surface): 100 Face per Surface Cluster
Step 8: Changing radiation settings

Contours of Wall Temperature (Outer Surface): 400 Face per Surface Cluster

Contours of Wall Temperature (Outer Surface): 800 Face per Surface Cluster

Contours of Wall Temperature (Outer Surface): 1600 Face per Surface Cluster
Step 8: Changing radiation settings

Contours of Surface Cluster ID—1600 Faces per Surface Cluster (FPSC)

Contours of Surface Cluster ID—400 FPSC
As mentioned previously, when the S2S model is used, one also has the option to define a “partial enclosure”; that means, one can disable the view factor calculation for walls with negligible emission/absorption, or walls that have a uniform temperature.

Even though the view factor will not be computed for these walls, they will still emit radiation at a fixed temperature called the “partial enclosure temperature”. The main advantage of this is to speed up the view factor calculation.

In the steps that follow, one has to specify the radiating wall (w-low-x) as a boundary zone that is not participating in the S2S radiation model. Consequently, one has to specify the partial enclosure temperature for the wall.
● **Step 9: Partial enclosure settings**

a. Click the Radiation tab.

b. Disable the Participates in View Factor Calculation option in the S2S Parameters group box.

c. Click OK to close the Wall dialog box.

---

a. Click the **Settings...** button to open the **View Factors and Clustering** dialog box.

b. Click the **Select...** button to open the **Participating Boundary Zones** dialog box.
Step 9: Partial enclosure settings

Temperature Profile Comparison on Outer Surface

Temperature Profile Comparison on Outer Surface further confirms that the use of a partial enclosure did not significantly affect the results.
In this example, we combined natural convection and radiation in a three-dimensional square box and compared how varying the settings of the surface-to-surface (S2S) radiation model affected the results. The S2S radiation model is appropriate for modeling the enclosure radiative transfer without participating media, whereas the methods for participating radiation may not always be efficient.
Hopefully we are ready for the computer session!