TIPS & TRICKS

View-Factoring Radiation into ANSYS Workbench Simulation

The ANSYS Radiosity Solution Method accounts for heat exchange between surfaces using Named Selections and a Command object.

By Sheldon Imaoka, ANSYS, Inc.

Radiation can play an important role in heat transfer analyses. In ANSYS Workbench Simulation, a “Radiation” load is available that allows users to account for losses to the surroundings, although this does not include radiation exchange between surfaces. To utilize the ANSYS surface-to-surface radiation capabilities, an easy method is available to include these effects within ANSYS Workbench Simulation via Named Selections and a Command object.

How Heat Radiates

Radiation is a highly nonlinear mode of heat transfer. A simplified form of the equation describing radiation from one surface to another is:

\[ Q = A \varepsilon F \sigma (T_i^4 - T_j^4) \]

in which \( A \) is the surface area, \( \varepsilon \) is the emissivity of the surface, \( F \) is the view factor between surfaces, \( \sigma \) is the Stefan–Boltzmann constant, and \( T_i \) and \( T_j \) are the two surface temperatures in absolute temperature units.

When using the Radiation load in ANSYS Workbench Simulation, the user enters the emissivity and ambient temperature, while the form factor \( F \), is assumed to be 1.0. In this situation, \( T \) reflects the absolute temperature of a node on the surface while \( T_e \) represents the ambient temperature. Temperature values are specified in °C or °F, and ANSYS Workbench Simulation automatically converts these values to Kelvin or Rankine, respectively.

Performing Radiosity Solutions

The Radiosity Solution Method differentiates each independent region of radiation as an enclosure. Within each enclosure, view factors are determined, and the radiated heat flux is computed between each element face as well as to space, if the enclosure is open.

It is worthwhile to note that conduction and radiation calculations are performed in a segregated fashion. Depending on the degree of radiation heat flow in the model, the solution may require more iterations than regular ANSYS conduction-based solutions.

To use the Radiosity Solution Method, the following two steps are performed:

1. Designate surfaces that will radiate to each other via Named Selections.

To create Named Selections, surfaces (or edges in 2-D) are selected and the “Create Selection Group” icon on the Named Selection toolbar is chosen. Surfaces that radiate between each other in an enclosure can be defined in one or several Named Selections. The only point the user needs to keep in mind is that all surfaces in a given Named Selection will be assumed to have the same emissivity values. With multiple Named Selections, however, each set of surfaces can have different values of emissivity.

As an example, consider one hollow block positioned inside another. Two Named Selections (REGION_A and REGION_B — one for each block) are needed to represent all surfaces that can radiate between each other. By making use of two Named Selections, the emissivity values of the REGION_A surfaces can be specified independent of those of the REGION_B surfaces.

In addition to specifying the surfaces of the blocks that can radiate between each other, a Command object also needs to be inserted under the “Environment” branch in ANSYS Workbench Simulation. Only a few APDL commands are required to specify the parameters needed for the Radiosity Solution Method:

\[
\text{sf,REGION}_A,\text{rdsf,0.9,1} \\
\text{sf,REGION}_B,\text{rdsf,0.8,1} \\
\text{stef,5.67e-8} \\
\text{toffst,273.15} \\
\text{hemiopt,10} \\
\text{tunif,20}
\]
The meaning of these commands is explained below:

- The first two commands (SF) group Named Selections (in this case “REGION_A” and “REGION_B”) to a given enclosure (1) with emissivity values of “0.9” and “0.8,” respectively. If the enclosure is open, the SPCTEMP command also should be used to designate the space (ambient) temperature.

- The next two commands, STEF and TOFFST, specify the Stefan–Boltzmann constant and the temperature offset to absolute zero. These are unit-dependent, so users should make sure that the active unit system is set accordingly from the “Units” menu.

- The HEMIOPT command is optional, but it is used to set the resolution of the viewfactor calculations \( F_{ij} \). The default value is 10, but it may be increased for more accurate viewfactor calculations at the expense of additional CPU time. (For 2-D analyses, the analogous command is V2DOPT instead of HEMIOPT.)

- The last command, TUNIF, specifies the initial temperature in °C or °F. For a nonlinear steady-state thermal analysis, specifying a reasonable initial temperature will help convergence.

Handling Multiple Enclosures

If multiple enclosures (i.e., independent radiating regions) are present, SF commands may be repeated for the other Named Selections but referencing a different enclosure ID.

Users also are advised to turn on “Auto Time Stepping” and to specify the number of initial, minimum and maximum substeps under the “Solution” branch since radiation problems can be highly nonlinear.

A number of advanced Radiosity Solution Method options also are available:

- RADOPT to specify solver controls
- VFOPT to write or read the viewfactor file
- RSYMM and RSURF to take advantage of planar or cyclic symmetry
- RDEC and RSURF to coarsen the element faces for radiation calculations only
- Temperature-dependent emissivity specification is also possible

All of the APDL commands are documented in the ANSYS Commands Reference, and additional details of Radiosity Solution Method options can be found in Sections 4.6 and 4.7 of the ANSYS Thermal Analysis Guide.