Date	November 1, 2006	Memo Number	STI:06/03
Author	Sheldon Imaoka	ANSYS Revision	10.0
Subject	Sheldon's ANSYS Tips and Tricks: Radiosity Solver in WB Simulation		
Keywords	Radiosity Solver, Workbench Simulation		

1. Introduction:

Radiation can play an important role in heat transfer analyses. In Workbench Simulation 10.0, a "Radiation" load has been added to allow users to account for losses to the surroundings, although this does not include radiation exchange between surfaces.

For users wishing to utilize the ANSYS surface-to-surface radiation capabilities, this memo hopes to introduce an easy method to include these effects within Workbench Simulation via Named Selections and Command objects.

This type of methodology can be extended to include any other advanced solution feature of ANSYS which the user may wish to incorporate inside of Workbench Simulation.

2. Background:

Radiation is a highly nonlinear mode of heat transfer, a simplified form of the equation shown below:

$$Q_{ij} = A_i \varepsilon_i F_{ij} \sigma \left(T_i^4 - T_j^4 \right)$$

where A is the surface area, ε is the emissivity, F is the view factor, σ 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 Workbench Simulation, the user inputs the emissivity and ambient temperature, while the form factor F_{ij} is assumed to be 1.0. In this case, T_i reflects the absolute temperature of a node on the surface while T_j represents the ambient temperature. The user specifies temperature values in °C or °F, and Workbench Simulation automatically converts these values to Kelvin or Rankine, respectively.

This type of radiation is suitable when radiation occurs to 'space' and when no surfaces are assumed to radiate to each other.

However, in cases where radiation occurs between surfaces of the model, this built-in Workbench Simulation "Radiation" load is not sufficient – surface-to-surface radiation may need to be defined. Surface-to-surface radiation utilizing the *ANSYS Radiosity Solution Method* can account for this phenomenon.

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

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

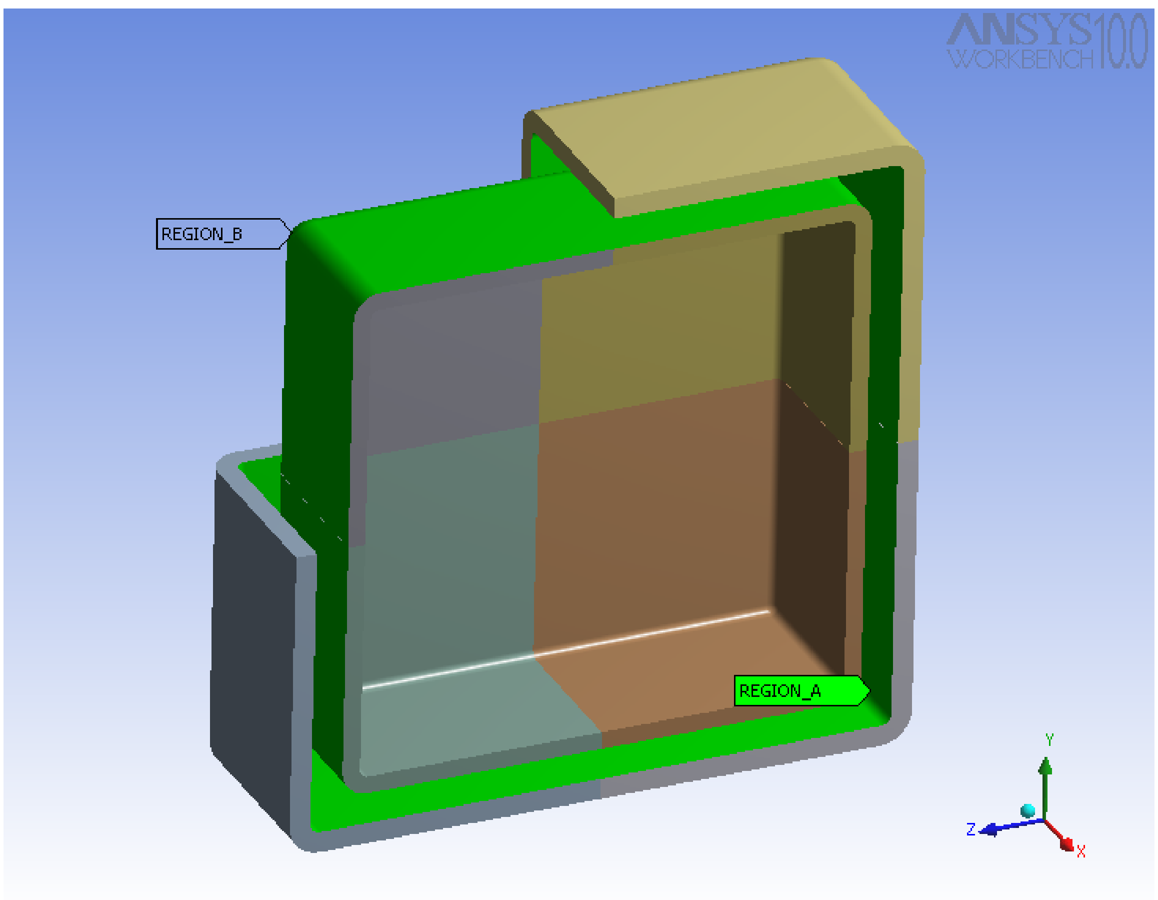
3. **Incorporating the Radiosity Solution Method in Workbench Simulation:**

In order to utilize the Radiosity Solution Method, the user only needs to perform the following two steps:

1. Designate surfaces which will radiate to each other via *Named Selections*
2. Insert necessary APDL commands via *Commands* object.

To create Named Selections, the user just needs to select the surfaces (or edges for 2D analyses), then use the "Create Selection Group" icon on the Named Selection toolbar. The surfaces which will be in an enclosure (those surfaces which can radiate to each other) can be defined in one or several Named Selections – the only point that 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, each set of surfaces can have different values of emissivity, however.

An example of this is shown in Figure 1 below. In this example, two Named Selections – REGION_A and REGION_B – represent all of the surfaces that can radiate to one other (some parts of the model are hidden for clarity). The emissivity values of REGION_A surfaces can be specified independent of those of REGION_B surfaces.



In addition to specifying the surfaces of an enclosure that can radiate towards each other, a "Commands" object also needs to be inserted under the "Environment" branch. The contents are as follows:

```
sf,REGION_A,rdsf,0.9,1
sf,REGION_B,rdsf,0.8,1

stef,5.67e-8
toffst,273.15

hemiopt,10

tunif,20
```

Only a few APDL commands are required to specify the use of the Radiosity Solution Method:

- 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 should also 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 ensure 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 2D analyses, the analogous command is `V2DOPT` instead of `HEMIOPT`.)
- The last command, `TUNIF`, specifies the initial temperature in ºC or ºF. For a nonlinear steady-state thermal analysis, specifying a reasonable initial temperature will help convergence.

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

The user is also 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.

Note that the user can also include advanced Radiosity Solution Method options:

- `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*.

The temperature distribution of three blocks radiating to each other is shown in Figure 2. In this case, the center, small block (transparent) has a fixed temperature and is radiating to the middle block (displayed). This block, in turn, radiates to another block (transparent) that encloses it, and the third block radiates to space. This model illustrates the use of three radiation enclosures – two are closed while one is open. Radiation is the only mode of heat transfer between the three blocks, yet the Radiosity Solution Method can easily incorporated inside of Workbench Simulation to solve these type of problems.