Collaborative Solutions Inc

An ANSYS Support Distributor

Date March 3, 2000 Memo Number STI65:000311B
Subject **ANSYS Tips & Tricks: Radiation with the Radiosity Solver Part 1**
Keywords Advanced Thermal: Radiation: Radiosity Solver

1. Introduction:

Starting with ANSYS 5.6, the Radiosity solution method has been introduced to allow for flexibility and easier definition of surface-to-surface radiation in 3D. This memo will cover some details of the Radiosity method. This memo assumes that the user is already familiar with the Radiosity solution method and will not cover procedural aspects.

2. Background Discussion:

In ANSYS, radiation can be treated in a handful of methods, depending on the user's needs. In ANSYS/Mechanical[1], LINK31 provides node-to-node radiation capabilities. SURF151/152 allow for node-to-surface radiation in 2D or 3D. The /AUX12 radiation matrix enables the user to define surface-to-surface radiation in 2D or 3D. On the other hand, in ANSYS/Flotran, radiation is specified via a surface boundary condition with a fixed ambient temperature.

At 5.6, the Radiation solution method has been added for 3D problems, giving the user an alternative to the /AUX12 approach for surface-to-surface radiation. *It is worthwhile to note that at ANSYS 5.7, the Radiosity solution method will be 'extended' to 2D planar and axisymmetric analyses. ANSYS/Flotran will also support the Radiosity solver in both 2D and 3D analyses.*

For 3D surface-to-surface radiation, the user has two choices at 5.6: the /AUX12 radiation matrix or the new Radiosity solution method. It may be useful to discuss the background of both approaches.

In the simple case of two surfaces radiating to each other, radiation can be defined as:

$$Q_{12} = A_1 F_{12} \varepsilon_1 \sigma \left(T_1^4 - T_2^4\right)$$

where

Q_{12} = Radiating heat flow rate between surface 1 to 2

A_1 = Area of surface 1

ε_1 = Emissivity of surface 1

F_{12} = Form factor representing fraction of total radiant energy going from surface 1 to 2

σ = Stefan - Boltzmann constant

T_1 = Absolute temperature of surface 1

T_2 = Absolute temperature of surface 2

2.1 /AUX12 Radiation Matrix Technique:

In the case of the /AUX12 radiation matrix method, radiation is accounted for by including a superelement in the conductivity matrix. To understand this better, the equation for steady-state heat transfer (conduction-based) can be examined:

$$[K]\{T\} = \{Q\}$$

where

K = Conductivity Matrix

T = Temperature (DOF) vector

Q = Heat flow rate vector

One may recall from the above discussion that radiation provides a nonlinear response ($\propto T^4$). In the case of arbitrary surface-to-surface radiation, the form factors often result in a full matrix

[1] ANSYS/Mechanical, ANSYS/Multiphysics, and other related packages support general radiation capabilities, including the Radiosity solver, for conduction-based analyses. ANSYS/Flotran and ANSYS/Multiphysics support Flotran-related radiation options. ANSYS/Professional supports some radiation features but not the Radiosity solver. Please see the online help for more details of which options are supported for your particular ANSYS license.

relating each of the radiation surfaces to each other. (F_{ij} term for exchange between surface i to surface j) Moreover, the temperature terms need to be linearized so as to be compatible with our basic heat transfer equation. As a result, we can rearrange the simplified equation of radiation between any two surfaces i and j:[2]

$$Q_{ij} = A_i F_{ij} \varepsilon_i \sigma \left(T_i^4 - T_j^4\right)$$
$$Q_{ij} = \left[A_i F_{ij} \varepsilon_i \sigma\right]\left[T_i^4 - T_j^4\right]$$
$$Q_{ij} = \left[K^{ts}\right]\left[\left(T_i^2 + T_j^2\right)\left(T_i^2 - T_j^2\right)\right]$$
$$Q_{ij} = \left[K^{ts}\right]\left[\left(T_i^2 + T_j^2\right)\left(T_i + T_j\right)\right]\left(T_i - T_j\right)$$
$$Q_{ij} = \left[K^{ts}\right]\left[K^{nl}\right]\left(T_i - T_j\right)$$
$$Q_{ij} = \left[K'\right]\left(T_i - T_j\right)$$

One can see that the $\left[K^{ts}\right]$ matrix contains the contributions of the form factors, emissivities, etc. This is a linear matrix (computed by ANSYS as a superelement matrix); in other words, this part of the effective conductivity matrix $\left[K'\right]$need not be recomputed at each substep. However, this means that the /AUX12 method only allows for constant values of emissivity (i.e., no temperature-dependent emissivity is permitted).

The other matrix, $\left[K^{nl}\right]$, contains the T^3 terms. This means that $\left[K'\right]$ needs to be recomputed at each equilibrium iteration, but only the portion $\left[K^{nl}\right]$ needs to be recalculated, not $\left[K^{ts}\right]$.

2.2 Radiosity Solution Method:

For the Radiosity solution method, radiation is treated differently than in /AUX12. Instead of adding radiation with an 'effective conductivity matrix', radiation is calculated as a heat flow rate vector. Recall that, as a simplified viewpoint, radiation heat flow rate between two surfaces i and j is defined as:

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

Assuming that the temperatures at the radiation surfaces are known, the radiation heat flux (radiosity) can be calculated. The radiation heat flux can then be applied to the thermal system as a load vector:

$$[K]\{T\} = \{Q\} + \{Q_{rad}\}$$

This is the basics of the Radiosity solution method. Since the surface temperatures and the radiative fluxes are not necessarily known in advance, the radiosity equation and the conduction equation above are solved in a segregated, iterative fashion until convergence is achieved.

The benefits of such an approach lie in the fact that temperature-dependent emissivities can be taken into account since the radiation heat flux is calculated independent from the conduction equation. Possibly the greatest benefit for the user is the ease by which the user can specify radiation using the radiosity solver method. Since generating and using a radiation superelement is not required, preprocessing is much easier.

However, the author has found that the Radiosity solution method is less stable when the problem is radiation- or heat-flux dominant – the most common error one may encounter is a "DOF limit exceeded" message. If radiation is the only heat sink in the model, special steps need to be taken to ensure that a solution can be obtained. Before discussing solution techniques, it may be instructive to understand why this problem occurs in the first place.

Since radiation is added to the conduction equation via a heat load vector, one can see that $[K]$ will be a singular matrix if there are insufficient temperature constraints (i.e., no temperature constraint or convection boundary condition present in the model). This usually occurs, as noted above, when radiation is the only heat sink present in the model. Hence, the next section will discuss some tips to help solve any problem using the Radiosity solution method.

[2] Please note that this is a very simplified way of viewing the /AUX12 radiation matrix, but an instructive one. For a better treatment of the equations, please refer to Eq. 6.4-3 or 6.5-1 in the ANSYS 5.6 Theory Manual.

3. Tips on Using the Radiosity Solution Method:

As noted above, the preprocessing features of the Radiosity solver make it very attractive to use.[3] Moreover, the solution times achieved with the Radiosity solution method is much less than that of the /AUX12 method.[4]

However, for *steady-state* radiation- or heat flux-dominant problems, there may be the possibility that convergence may be difficult to obtain due to the fact that radiation is incorporated into the conduction equation via a heat load vector. Recall that the /AUX12 method constructs an element to relate the temperature of one surface to another.[5] Hence, if one has a radiation-dominant problem, for example, a part in outer space, the effective conductivity relates the space node temperature with the rest of the model. In the case of the Radiosity solution method, there is no such direct relationship between a "space node" and the rest of the model, which would result in a singular [K] matrix if radiation were the only heat sink present.

If this occurs, one may need to run the problem as a quasi-steady-state one using the QSOPT command. This makes the problem stable by adding thermal capacitance effects (heat storage terms) via a specific heat matrix:

$$[C]\dot{T} + [K]T = Q + Q_{rad}$$

One can see that the equations are no longer singular and can be solved for without much difficulty. This is analogous to adding inertial terms in structural dynamics problems where rigid-body motion is present. If the Radiosity solution method were used in transient problems with radiation, no special treatment is needed since the specific heat matrix is automatically included.

The choice of the definition of specific heat and density when running quasi-steady-state radiation problems is based on the diffusivity of the system. The lower the diffusivity, the more stable the problem but the longer it will take to solve (it takes longer to dissipate the stored energy). The higher the diffusivity, the faster convergence will be achieved (transient term will be dissipated faster), but the less stable the solution. The author finds that, by approximating how much heat is put into the system (heat fluxes or heat generation), one can approximate the required values of specific heat and density. On the other hand, one can always start off with small values of C and DENS (values of 1, for example). If the "DOF limit exceeded" error appears (i.e., indication of an unstable solution where thermal runaway occurs due to reasoning explained above), one can increase C or DENS by a factor of 10 and try again (the author refers to this second approach as a brute-force, trial-and-error way of selecting DENS and C values).

When the analysis is specified as transient (ANTYPE,TRANS) with quasi-steady-state turned on (QSOPT,ON), ANSYS automatically increases the 'time' value and continues to run until a steady-state solution is achieved.[6] Convergence is obtained if the maximum temperature change does not exceed 0.1 units in three substeps. This behavior can be changed with open control (OPNCON), if a tighter (or looser) measure of steady-state is required.

Lastly, to reiterate, the above discussion is assuming that the user wants to run a *radiation- and/or heat flux-dominant steady-state* problem.

[3] Please see Ch. 4.7 "Using the Radiosity Solver Method" in the ANSYS 5.6 Thermal Analysis Guide on procedures of using the Radiosity solution method.
[4] Comparison of /AUX12 and Radiosity Methods will not be covered in this present memo but is planned to be addressed in a future memo.
[5] LINK34 node-to-node radiation and SURF151/152 node-to-surface radiation also employ similar techniques as /AUX12 by relating radiation between nodes/surfaces as an element (effective conductivity)
[6] If DENS and C are selected based on guidelines as mentioned above, the final time for the solution is not 'true' time. On the other hand, if actual values of DENS and C are used, then the final time is representative of actual time. Of course, as mentioned above, for quasi-steady-state problems, DENS and C should be selected to define an appropriate diffusivity since the values of DENS and C do not affect the final results.

On the other hand, for problems which are not radiation-dominant, the user can change radiation options (RADOPT) by increasing the flux relaxation value. Because of the fact that the conduction and radiosity equations are nonlinear and solved in a segregated fashion, under-relaxation is used for the radiative heat flux vector as follows:

$$q_{rad}^{i+1} = \phi q_{rad\_calc}^{i+1} + (1-\phi) q_{rad\_calc}^{i}$$

where

q_{rad}^{i+1} = Radiative heat flux used at current iteration i + 1

$q_{rad\_calc}^{i+1}$ = Radiative heat flux calculated at current iteration i + 1

$q_{rad\_calc}^{i}$ = Radiative heat flux calculated at previous iteration i

ϕ = Under - relaxation value (default is 0.1)

The default value of 0.1 for under-relaxation provides stability in this segregated solution approach. However, for problems not radiation dominant, a higher value of under-relaxation (FLUXRELX argument of RADOPT command) will provide faster convergence.

While the user may want to consider leaving the default values of the radiosity options, after an initial run, there are some controls which the author has found useful to achieve better accuracy, especially for coarsely-meshed models. Please note that this should be done *after* an initial run if better accuracy is needed.

Tightening up the convergence criteria on the FLUXTOL argument of the RADOPT command provides increased accuracy. This specifies a convergence criterion during the radiosity calculations. The default is 0.1, but the author has found that tightening this tolerance to 0.01 has sometimes provided more accurate results. To determine if this is necessary, one can view the output in the output window/file during the radiosity iterations:

```
RAD FLUX CONVERGENCE VALUE=  0.502753E-15 CRITERION=  0.100000E-01
```

If the radiative heat flux convergence value is close to the criterion, then tightening the criterion may help. In the above sample output, tightening the criterion would have no effect since the radiative heat flux convergence value is so small.

Increasing the hemicube viewfactor calculation resolution (HEMIOPT) from 10 (default) to 20, even up to 100, can provide better answers as well. Note that increasing this value will lead to longer viewfactor calculation times, but since this is needed for coarser models, the computational cost usually is not too bad. Also, by reusing the stored viewfactor calculations (VFOPT), the user will not have to redo this part of the analysis if the user is running “what-if” studies on a parametric model.

To reiterate, the above accuracy controls should be done *after* an initial run is completed with the default settings. The tightening of the convergence criteria for the radiosity solver iterations and the increase of the hemicube resolution can help provide better accuracy, especially for coarse meshes.

For models with SHELL57 elements, please note the face 1 & 2 notation (as indicated in the online help for SHELL57) for top and bottom surfaces. To see which is the “top” face, use /PSYM,ESYS,1 to see the element coordinate system. The element z-axis (blue) will be pointing outward, indicating the “top” surface of the shell. One may want to consider using selection logic to select the appropriate shells, then use SFE with the appropriate load face to ensure that radiation will act upon the proper face (i.e., proper direction).

Included are some examples taken from J.P. Holman’s “Heat Transfer” text, 8th ed., Chapter 8, example problems #6-10, excluding #9. These examples provide some comparison with lumped-parameter models in Holman’s book. Some involve radiation dominant situations, so, while the examples themselves are quite simple, they may aid the user in understanding how to run quasi-steady-state options for radiation-dominant steady-state problems.

4. Conclusions/Recommendations:

ANSYS provides node-to-node, node-to-surface, and surface-to-surface radiation capabilities. In this memo, the new Radiosity solution method has been discussed. There are benefits and limitations for using /AUX12 and Radiosity methods in solving surface-to-surface radiation problems, some of which are listed below:

/AUX12 Radiation Matrix Technique:

+ Radiation terms in conductivity matrix, which provides a stable approach
+ Postprocessing of "reaction" heat flow rate is easy
- Requires significantly more preprocessing effort to generate radiation matrix
- Does not allow temperature-dependent emissivity values

Radiosity Solution Method:

+ Easy to define radiating surfaces
+ Allows for temperature-dependent emissivities
+ Querying of view factors is available
- Limited to 3D problems without symmetry at 5.6. [Note that at 5.7, Radiosity solution method will be extended to 2D planar and axisymmetric problems as well as Flotran]
- Postprocessing is limited at 5.6, such as lack of ability to list "reaction" heat flow.
- Radiation is treated as a heat flux vector, so it may be less stable for steady-state, radiation-dominant problems. As a result, a quasi-steady-state option is available.

Because the Radiosity solution method treats radiation as a heat flux vector instead of as an element (effective conductivity matrix), as with other ANSYS radiation capabilities, there are different concerns the user needs to be aware of. However, the author hopes that this memo may provide some useful insight into the Radiosity method, such that the user can more easily solve problems using this technique. At 5.7, the capabilities of the Radiosity method have been extended to make it an even more attractive way of dealing with radiation effects in a system.

5. References:

Kohnke, Peter, Ph.D, ed., ANSYS Theory Reference, Release 5.6, 1999
ANSYS 5.7 Update Training, 8/31/00
Private conversation, D. Looman, 10/5/2000

Sheldon Imaoka
Collaborative Solutions, Inc. (LA Office)
Engineering Consultant

ANSYS Tip of the Week

"ANSYS Tip of the Week" (TOTW) is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe or to view older TOTW archives can visit

`http://www.csi-ansys.com/tip_of_the_week.htm`

Corrections, comments, and suggestions are welcome and can be sent to operator@csi-ansys.com [they will be distributed to the appropriate person(s)]. While CSI engineers base their TOTW on technical support calls and user questions, ideas on future topics are appreciated. Users who wish to submit their own TOTW are encouraged to do so by emailing the above address for more information.

XANSYS Mailing List

The XANSYS mailing list is a forum for questions and discussions of the use of ANSYS. As of 04/00, there are more than 1000 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the "digest" option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

```
This list is for *ALL* users of the ANSYS finite element analysis program from around
the world. The list allows rapid communication among users concerning program
bugs/ideas/modeling techniques. This list is NOT affiliated with ANSYS, Inc. even
though several members of the ANSYS, Inc. staff are subscribers and regular
contributors.

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com
To unsubscribe send blank email to xansys-unsubscribe@onelist.com
Archived on http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html
ANOTHER archive on http://www.eScribe.com/software/xansys/
(A poor archive is also at http://www.onelist.com/archives.cgi/xansys)
```

CSI ANSYS Technical Support, Training, & Mentoring

Collaborative Solutions, Inc. is committed to providing the best customer support in our industry. Three people will be devoted to technical support from 8:00 a.m. to 5:00 p.m. PST every working day. CSI customers with active TECS (maintenance) agreements may contact CSI by any of the following ways:

```
Phone:  760-431-4815                            WWW: http://www.csi-ansys.com
Fax:    760-431-4824                            FTP:  ftp://ftp.csi-ansys.com
E-mail: firstname.lastname@csi-ansys.com
```

CSI Engineers:

Karen Dhuyvetter	Greg Miller	Sean Harvey	Alfred Saad
Bill Bulat	Sheldon Imaoka	David Haberman	Mike Rife

CSI believes strongly in the value of training and mentoring to help make customers successful using ANSYS. Training classes are usually 2-3 days in duration and provide instruction on various topics, including structural nonlinearities, heat transfer, and dynamics. Mentoring sessions involve working with a CSI engineer one-on-one on specific projects. These sessions help reinforce applicable subject matter covered in training classes or help ensure that the customer is using ANSYS most efficiently and effectively.

Training class schedules are posted at: `http://www.csi-ansys.com/training.htm`

Please contact your account manager for more details on training, mentoring, consulting, and other CSI services.