Automating Thermal Analysis with Thermal Desktop™

Mark J. Welch
Tim Panczak
C&R Technologies, Inc.

ABSTRACT

Thermal analysis is typically executed with multiple tools in a series of separate steps for performing radiation analysis, generating conduction and capacitance data, and for solving temperatures. This multitude of programs often leads to many user files that become unmanageable with their multitude, and the user often looses track as to which files go with which cases. In addition to combining the output from multiple programs, current processes often involve the user inputting various hand calculations into the math model to account for MLI/Insulation and contact conductance between entities. These calculations are not only tedious to make, but users often forget to update them when the geometry is changed.

Several new features of Thermal Desktop are designed to automate some of the tedious tasks that thermal engineers now practice. To start with, Thermal Desktop is a single program that does radiation analysis, generates conduction/capacitance data and automates the building of a SINDA/FLUINT model to solve for temperatures. Some of these new features of Thermal Desktop are Radiation Analysis Groups, Property Aliases, MLI/Insulation Objects, Contact Conductance Objects, Model Browser, and the Case Set Manager.

This paper describes the application and benefits of Thermal Desktop along with other unique features used to automate the thermal analysis process.

INTRODUCTION: THERMAL DESKTOP™

Thermal Desktop™ is a PC/Cad based thermal analyzer. This system, while fully integrated into a neutral, low cost CAD system, and which utilizes both FEM and FD methods, does not compromise the needs of the thermal engineer. Rather, the features needed for concurrent thermal analysis are specifically addressed by combining traditional parametric surface based radiation and FD based conduction modeling with CAD and FEM methods. The use of flexible and familiar temperature solvers such as SINDA/ FLUINT is retained.

The Thermal Desktop is implemented as a single application that:

1) Integrates CAD, FEM, FD, radiation (RadCAD®), and procedural modeling into a single low-cost environment. The environment simultaneously supports both "design geometry" used for the exact specification of hardware and "analysis geometry" which may (or may not) be a simplified abstraction of the design geometry used for thermal analysis.
2) Allows analysis geometry to be constructed using CAD operations: booleans, sweeping, blending, ruling, revolving, etc.
3) Allows design geometry to be imported from other CAD systems using IGES and/or DXF formats.
4) Permits design geometry to be used "as is" for analysis geometry, or used as "scaffolding" on which to construct suitable analysis geometry using interactive graphics operations.
5) Provides familiar types of thermal modeling surfaces such as cones, paraboloids, discs, rectangles, and cylinders using true mathematically precise representations (rather than as a collection of facets). These surfaces provide all of the functionality associated with TRASYS type surfaces but are directly integrated within the CAD environment.
6) Imports radiation models from TRASYS, Nevada, and IDEAS.
7) Integrates CAD methods for generating, resizing, and positioning surface types.
8) Integrates conduction/capacitance generation, surface insulation, radiation analysis, and contact conductance calculations.
9) Provides graphical construction of arbitrary nodes and conductors for abstract thermal network modeling.
10) Allows FE models to be created natively, or imported from popular FE programs such as NASTRAN, IDEAS, or FEMAP.
11) Provides efficient radiation analysis for common types of finite elements plus implements new types of curved

finite elements for even faster radiation analysis.

12) Implements a new thermal sub node that simplifies a collection of complex nodes/conductors/finite elements into one or more SINDA/FLUINT nodes.
13) Provides graphical construction of procedural thermal model entities such heat loads and thermostats.
14) Outputs data in both SINDA/FLUINT and SINDA/G type formats.

RADIATION ANALYSIS GROUPS

Consider a model that has enclosures of some kind within it. This model could be a simple satellite that has electronics on the interior of some type of body, such as the satellite shown in Figure 1. Radiation plays an important part in the thermal analysis of this interior body. Radiation also plays an important part on the exterior part of this body, not only for radiation conductors, but also for heating rates from the sun and the planet. For a small model, the user could place all the surfaces in the same model and do radiation analyses. This overloads the analysis by calculating zero values for the heating rates on the interior of the satellite. A better way is to create two separate radiation models. Prior methods using programs such as TRASYS[1] and TSS[2] would require the user to build two completely separate models to analyze the radiation, one for the interior and one for the exterior. Thermal Desktop allows the user to build a single model and to place different sides of surfaces in different radiation analysis groups. This input is shown in Figure 2. The benefits of radiation analysis groups are:
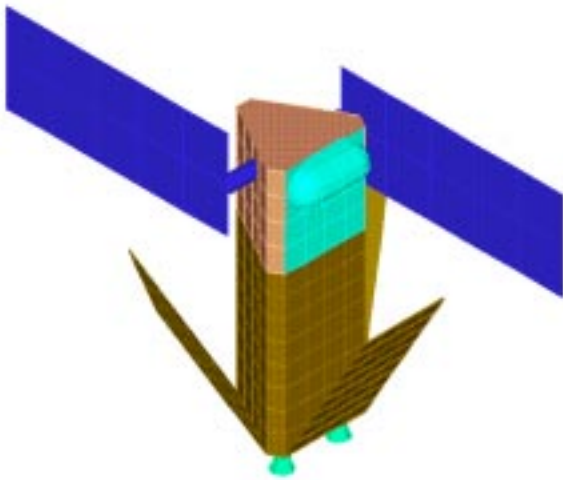

Figure 1: Simple Satellite Model

• Changes to the geometry such as scaling the body size are only performed once, thus eliminating the need to edit two models which lowers the risk of making an error.
• For surfaces with the same node numbers on each side, the management is much simpler for the user because the node id is referenced from only one place.
• Allows the program to make calculations more efficiently, thus doing them faster and also using less disk space to store the results.
• All of the geometric data is stored in a single place.

This not only creates an environment for simpler file management, but it also helps to eliminate modeling errors.
• There is no limit to the number of radiation analysis groups in a model. Thus, models with multiple enclosures may easily be analyzed.
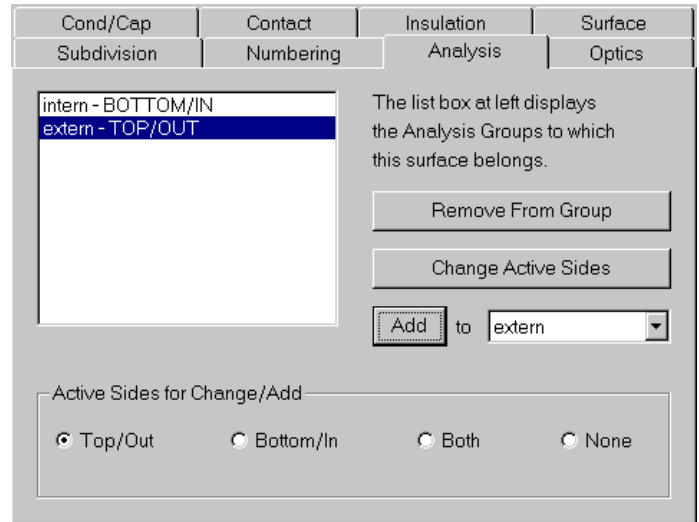

Figure 2: Radiation Analysis Input Page

PROPERTY ALIASES

Property aliases are a unique feature of Thermal Desktop. Using aliases, optical and thermophysical properties for many surfaces may be changed by changing the value of a single alias. Property aliases consist of an alias name, and a name of a property definition contained in a property database file.

Thermal Desktop surfaces reference optical properties by name, rather than entering values directly for each surface. When surfaces are processed for calculations, actual property values associated with the name are retrieved from the current optical property database. Surfaces may also use a property alias. In this case, properties defined for the physical property name associated with the alias are used.

An example of using property aliases could be that you have decided to place multi layered insulation (MLI) blankets around some surfaces in your model. Your next task will be to determine the proper number of layers of the blankets, so you will have to do your analysis several times. With property aliases, the user can create the alias blanket and specify that for each surface requiring the blanket. Blanket can then point to 8 layer blanket in the material database for the first run. The second run can point to 12 layer blanket, and so on.

MLI/INSULATION NODES

Spacecraft thermal analysis almost always involves the modeling of Multi Layered Insulation (MLI). Prior geometric based programs such as TRASYS and TSS do not provide simple methods for the modeling of MLI. A typical past process of modeling the MLI is for the user to look up the area

of a node from the radiation output, and then to hand type a radiation conductor based on that area and an effective emissivity. The effective emissivity is a function of the age of the blanket and the number of layers in the MLI. This prior process is not only tedious, it is conducive to modeling errors when the original geometry is changed because the user may forget to update the area.

The process of modeling MLI using Thermal Desktop is really quite simple. The user must first define the material property of the MLI, this is shown on the Thermophysical Property From (Figure 3). There are several items of interest on the form.

- Input for the effective emissivity is input at the bottom of the form.
- The user may specify to also output a linear conductor by inputting a non-zero value for the conductivity (the conductivity may also be temperature varying). This is especially useful when the MLI is at extremely cold temperatures where the linear conductance is more dominant than the radiative conductance across the MLI.
- Inputting the specific heat or density as zero will cause Thermal Desktop to output arithmetic nodes for the MLI/Insulation material.

Figure 3: Thermophysical Properties Input Form

Once the MLI material has been defined the user may specify to apply the MLI on the Insulation Page of the Surface Properties Form (Figure 4). Items to note on this form are:

- Application may be made to the top/out or bottom/in side of a surface.
- A thickness may be input. This thickness is used to calculate the linear conductor, if applicable, and also the capacitance of the insulation node.

- The user may select to offset the node id's by an input value, or to place the MLI nodes in a different sub-model.

Figure 4: Insulation Page

While the prior discussion involves details MLI modeling, the same process can be used to model any type of insulating material. For other insulating materials, the effective emissivity would be input as zero, and the user would concentrate on inputting the proper values for the linear conductivity and the thickness of the insulation.

CONTACT CONDUCTANCE

Thermal analysis almost always involves creating conductors between objects that have some kind of contact conductance between them. Whether it is the mounting of a chip to a circuit board (area to area contact) or attaching a circuit board to a box frame via a card guide (edge to area contact), thermal engineers have been doing these calculations by hand for a long time.(See Figure 5)
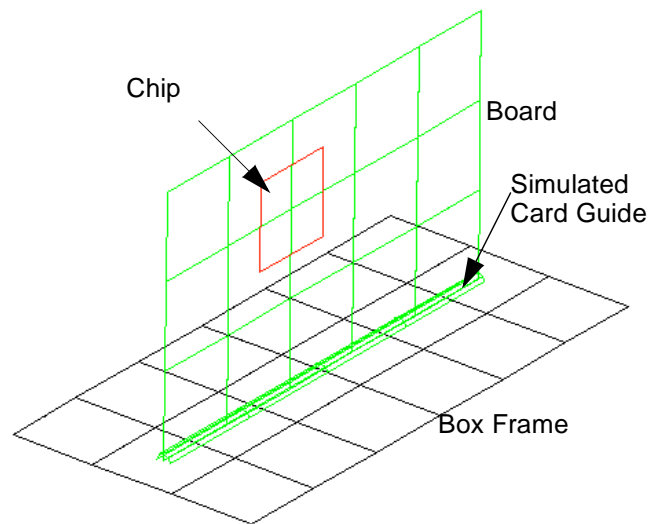

Figure 5: Contact Conductance Examples

The user input to specify the contact conductance is extremely simple. The user simply selects to apply the contact conductance from the contact page of the surface

properties form. Figure 6 shows the input for applying the contact conductance from the chip. When Thermal Desktop performs the SINDA/FLUINT conductance/capacitance calculations, the program will find the closest nodes to the contact area and hook up the conductors. Some significant advantages to this method are:

- The nodes do not have to line up. For the above example, there would be four conductors from the chip to the board that would be evenly weighted.
- The user can input the contact conductance in their desired units. Thermal Desktop does all the conversions to create a proper SINDA/FLUINT model.
- Since the calculation is performed when the SINDA/FLUINT output is calculated, the user can easily change submodels, nodes ids, and the breakdown of not only the surface with contact applied, but also the breakdown of the surface/s to where the contact is made and the conductors will be properly made at output time.
- The contact calculation is extremely fast as Thermal Desktop uses the same type of Oct Cell acceleration that is present in the radiation calculations (RadCAD).
- The user has control over the number of integration levels used to determine the contact. Higher integration levels can lead to more accurate area contact distribution at the expense of more CPU time.

Figure 6: Area Contact Specification

MODEL BROWSER

Thermal Desktop's Model Browser, shown in Figure 7, is a unique approach that allows the user to debug their model from a graphical standpoint. Items are listed in the tree by Node ID, with the user being able to edit, delete, turn visibility on/off, display node ids on the model, and to isolate the specific node by displaying only it. The tree can be listed by submodel name and node id, radiation analysis group, user conductors, heatloads, assemblys, or trackers.

In addition to isolation of nodes, users can select to isolate an entire submodel or analysis group. The Model Browser is quickly becoming one of the most popular features of
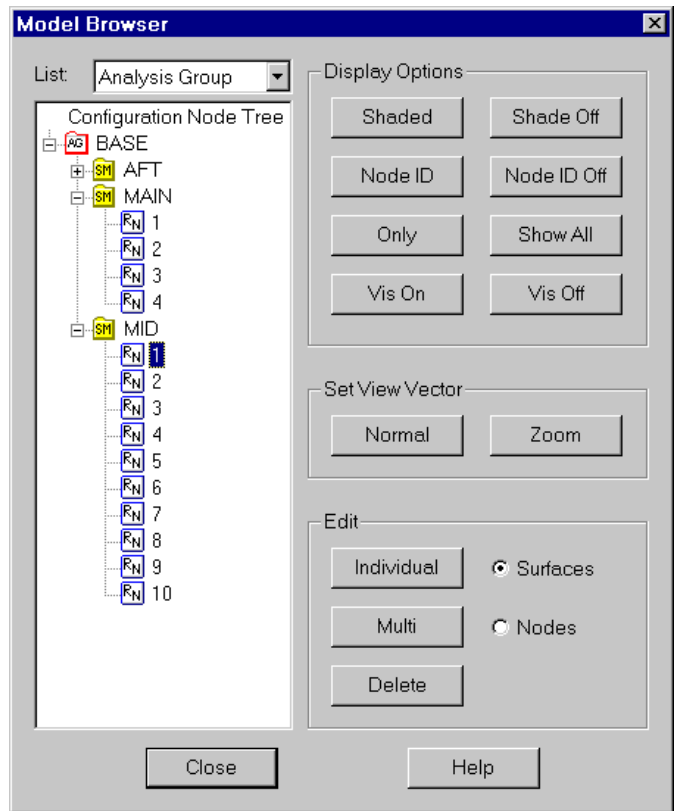

Figure 7: Model Browser

Thermal Desktop.

CASE SET MANAGER

A new feature implemented in Thermal Desktop allows a "one button" solution to integrating all thermal analysis activities.

The "Case Set" feature allows an analysis case to be created from one or more "jobs". Radiation conductor jobs, orbital heating rate jobs, conduction/capacitance jobs, and SINDA/FLUINT jobs may all be defined as part of a Thermal Desktop case. When a case is executed, radiation jobs are executed as needed, SINDA/FLUINT data is generated and executed, and postprocessing results are automatically returned to Thermal Desktop for color temperature mapping.

The Case Set Manager form, shown in Figure 8, allows the user to set up various cases for analysis. Each case has various properties to define the radiation analysis jobs (both for radiation conductors and orbital heating rates), and steady state or transient temperature calculations. Each case is referred to by a user input name. Once the properties are associated with a case, the user may select the run case button to solve that case.

The tasks associated with a case are to calculate radiation, conductance/capacitance, build a SINDA model, run the SINDA model, and then to postprocess the results. For simple SINDA models that have limited user logic, the user never has to enter a text editor. Of course, the user may

edit the SINDA model to include any non-geometric entities or input any specialized user logic.
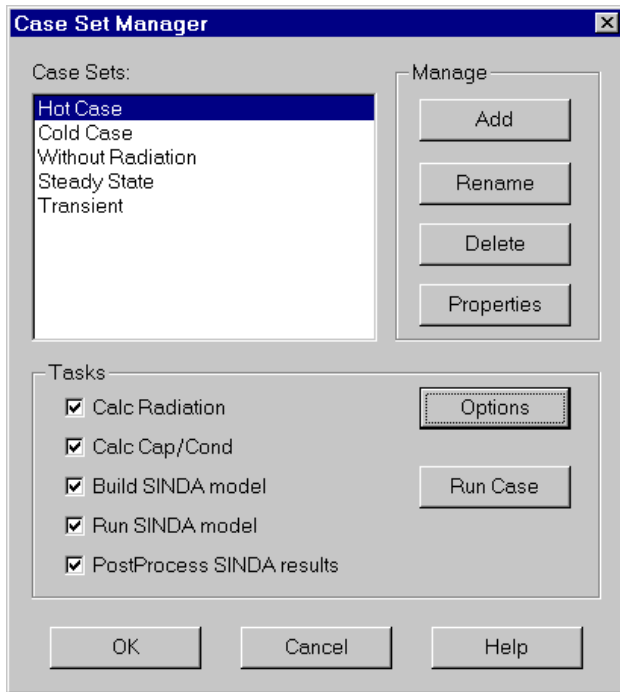

Figure 8: Case Set Manager

Various properties are associated with each case set. These properties allow the user to specify the radiation analysis jobs, the type of temperature calculations, and the type of output of the temperature calculations. The radiation properties page is shown Figure 9. The user inputs the radiation job by specifying the analysis group, the orbit, and the type of calculations, whether it is full Monte Carlo Ray Tracing or uses the ultra fast Progressive Radiosity routine that is unique to RadCAD.
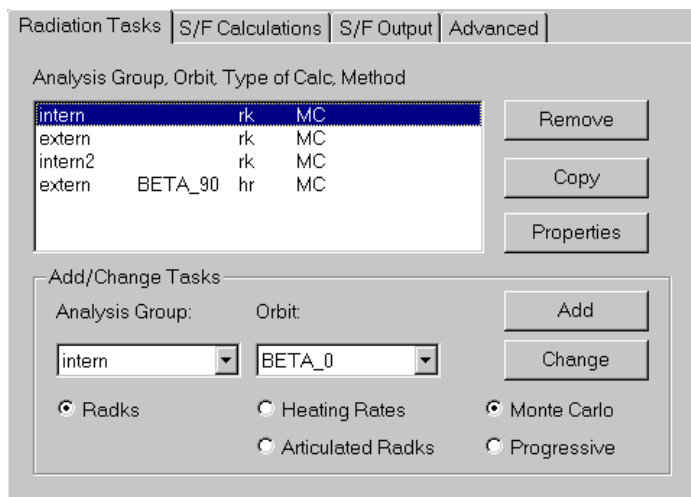

Figure 9: Case Set Radiation Page

Each radiation job has properties associated with it, as shown in Figure 10. The control page is shown and it allows the user to specify the number of rays, and various oct cell control parameters, as well as a few other items. Other pages allow the user to specify the output file names, as well as specify to shoot rays only for specific nodes, or

specific positions.

One of the benefits of setting up the radiation jobs in this manner is that the parameters used for the analysis and the output file names are stored with the model on the drawing file. When the model is transferred to another person, that person knows exactly where to look to see how the data was calculated.
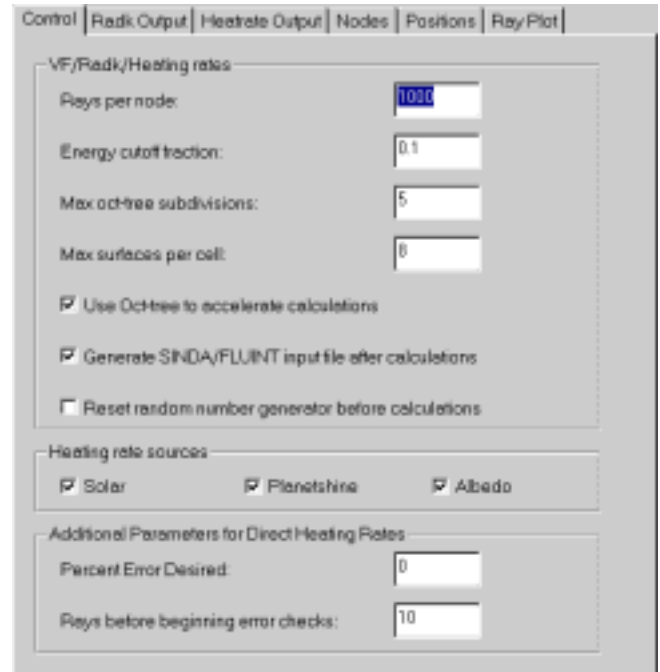

Figure 10: Radiation Properties

The user may specify a case set to be a steady state calculation or a transient as shown in Figure 11. If both steady state and transient are selected, then a steady state calculations will be performed before the transient is started. For transient cases, the user may input the start and stop time, as well as the timestep. Of course, inputting a time step of zero will use the automatic time step feature of SINDA/FLUINT, which is the highly recommended default. The user may specify the control criteria based on the maximum temperature change between iterations and for steady state cases, both nodal and system energy balances may be converged upon.

The case set output page, shown in Figure 12, allows the user to control the output of the SINDA/FLUINT job. The output may be in two forms, one for ASCII which may be read in a text editor, and one that is binary and can be mapped directly on the model. The user can control the output file names, the interval between output calls, and the type of data output.

The advanced page, shown in Figure 13, provides the user with three capabilities. First, it allows the user to include a file into the SINDA deck when it is built. This file may contain specialized user logic or non geometric entities that the model should consider. The second option on the advanced form allows the user to use the matrix inversion
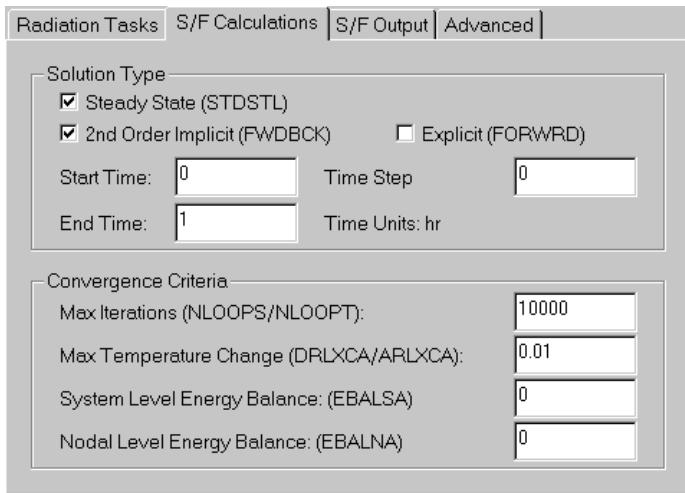
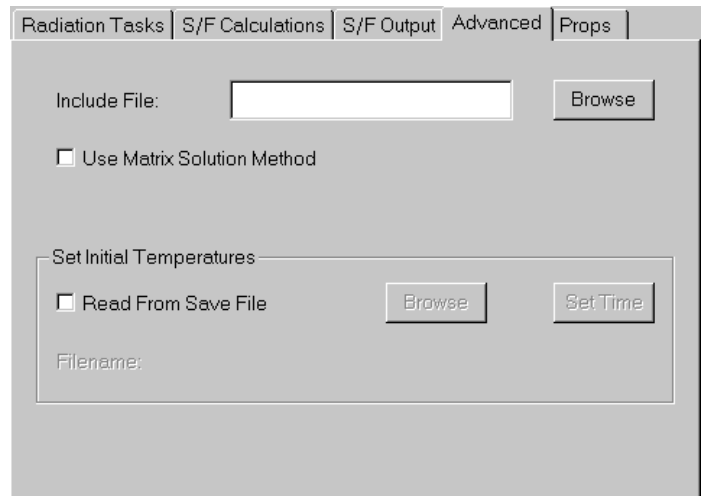Figure 11: Case Set SINDA/FLUINT Calculations Page

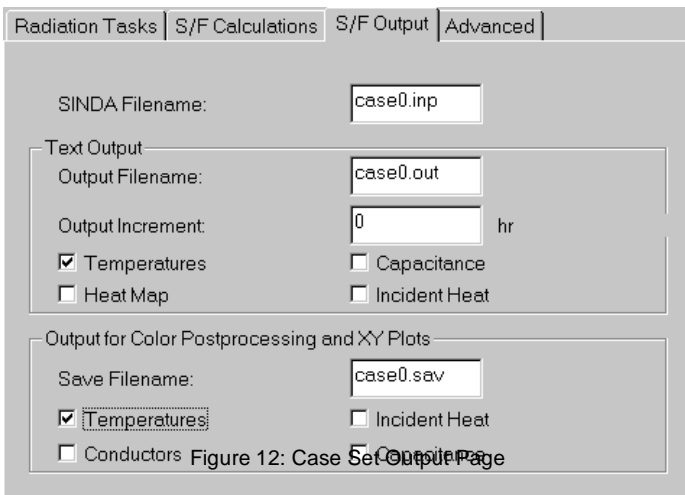Figure 13: Case Set Advanced Page
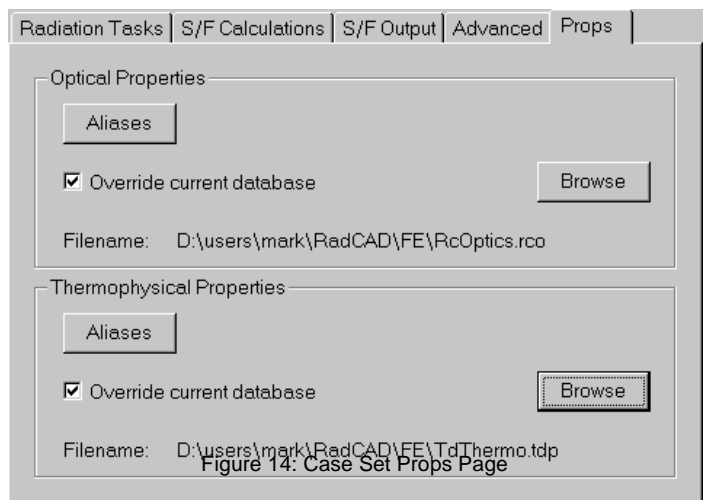
Figure 12: Case Set Output Page

Figure 14: Case Set Props Page

method for temperature calculations as opposed to the iterative method that is the default. This new matrix inversion method is new to SINDA/FLUINT Version 4.1. The third option allows the user to start the SINDA run by reading a time point from a previously generated SAVE file. Once the user has selected the restart file, the Set Time button can be selected and the user can select the restart time from a list of times on the file. This method of restart is much simpler that previous methods.

The case set props page, shown in Figure 14, allows the user to specify property databases and property aliases that will be used for this case set. The default properties for a case set are the global properties. When the override is selected, that database will be loaded into the program and used for the analysis of that case set.

An example of where this would be applied is for beginning of life (BOL) properties and end of life (EOL) properties. The user could set up two different case sets that only differ by the properties to be used for the analysis. By setting the model up in this fashion, each case set can be solved with the click of a button.

In addition to allowing different databases for each case

set, the user could use different Property Aliases for each case set. Thus, in the MLI Blanket example described above, two case sets could be set up, one with the blanket alias set to 8 layer, while the second could be set up for 12 layer.

ACKNOWLEDGMENTS

CONTACT

C&R Technologies, Inc.
303 971 0292
303 971 0035 (FAX)
www.crtech.com

User's manuals, tutorials, and training notes for software discussed are freely available in PDF format at www.crtech.com.

REFERENCES

1. Lockheed Engineer and Management Services Company, "Thermal Radiation Analysis System Users Manual," JSC 22964, Contract NAS 917900, October 1991.

2. Panczak, T., Welch, M.J., "Thermal Synthesizer System: An Integrated Approach to Spacecraft Thermal Analysis," SAE paper 911582, 22nd ICES Conference, July 1992.