A CAD-based Tool for FDM and FEM Radiation and Conduction Modeling

Tim Panczak, Steve Ring, and Mark Welch
Cullimore and Ring Technologies

ABSTRACT

Thermal engineering has long been left out of the concurrent engineering environment dominated by CAD (computer aided design) and FEM (finite element method) software. Current tools attempt to force the thermal design process into an environment primarily created to support structural analysis, which results in inappropriate thermal models. As a result, many thermal engineers either build models "by hand" or use geometric user interfaces that are separate from and have little useful connection, if any, to CAD and FEM systems.

This paper describes the development of a new thermal design environment called the Thermal Desktop. This system, while fully integrated into a neutral, low cost CAD system, and which utilitizes 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.

INTRODUCTION

Thermal engineering (especially in aerospace) has been performed with tools that have had a long heritage, before CAD systems and graphical FEM systems even existed. Finite difference based conduction and capacitance models were built by hand, and solved with programs such as CINDA (Chrysler Improved Numeric Difference Analyzer). Radiation analysis was performed by constructing geometric models consisting of conic primitives (cones, spheres, discs, etc.) using a text editor and solved using programs such as TRASYS[1]. Early radiation tools did not have the ability to graphically verify model geometry.

Improvements continued over the years, with CINDA ultimately being superceded by SINDA/FLUINT[2]. Progress was made in improving radiation analysis tools from TRASYS with the development the graphical based TSS[3]. However, the basic analysis process remained the same, independent of advances being made in CAD and FEM systems.

Part of the reason that thermal engineering has been outside of the concurrent fold is the entrenchment of current FD based analysis processes, but also because alternative FEM based systems have failed to address important thermal analysis requirements. Current FEM based systems facilitate only one aspect of thermal modeling: generating conduction and capacitance data directly from a geometric description.

Other, equally important aspects are addressed poorly or not at all. Not all thermal modeling is derived from detailed geometric specifications. Often thermal models are represented more abstractly with a conduction capacitance network generated using basic first law principles (for example, to simulate an interface for a vendor or customer).

Many geometrically complex components can be reduced to a simple and efficient network representation using an engineering estimate of the heat flow paths and basic finite difference methods. These techniques are used to develop fast executing thermal models, which have a significant advantage over brute-forced FEM models when many thermal simulations must be performed under different scenarios, or a number of design trades must be considered.

Many thermal control components are not represented by a network at all, but must be simulated procedurally. Examples are heaters, thermostats, and variable conductance heatpipes. This type of modeling is not found in FEM based thermal tools, which have usually

been based on a program originally designed for structural analysis.

More specifically, support for radiation analysis in FEM based systems has been traditionally weak. In addition, the standard types of finite elements result in model nodal densities greatly exceeding thermal requirements. Coupled with inefficient radiation analyzers, models quickly become intractable.

For example, with radiation analyzers such as TRASYS, a cylinder may be modeled with one node around the circumference. With current FEM based tools, a cylinder must be represented by a sufficient number of flat facets (each of which contributes one or more nodes) to approximate the geometric shape. Model size is governed by geometric fidelity rather than by thermal requirements. In addition, ray-surface intersection tests employed by radiation codes must be performed on a large collection of facets, rather than a single mathematical cylinder, which greatly increases run times.

Complex radiation phenomena such as angular dependent optical properties, transparency, refraction, and specularity are not well supported in the FEM world. More specific aerospace needs such as articulating geometry that tracks the sun, planet, or other satellites are not well supported, if at all.

Until now, the thermal engineer has had to choose between two undesirable solutions: (1) use current analysis tools that efficiently satisfy thermal modeling requirements but have no connectivity to the CAD design database or other engineering disciplines, or (2) use tools that support concurrency but sacrifice thermal modeling functionality.

The Thermal Desktop was designed from the ground up with both the requirements for concurrent engineering and for thermal analysis as key design drivers. This paper discusses the design approach and features of the Thermal Desktop that enable it to successfully integrate CAD and FEM with traditional analysis approaches.

OVERVIEW OF THERMAL DESKTOP

The Thermal Desktop is implemented as a single application that:

1) Integrates CAD, FEM, FD, radiation, 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) Integrates CAD methods for generating, resizing, and positioning surface types.

7) Integrates conduction/capacitance generation, surface insulation, radiation analysis, and contact conductance calculations.

8) Provides graphical construction of arbitrary nodes and conductors for abstract thermal network modeling.

9) Allows FE models to be created natively, or imported from popular FE programs.

10) Provides efficient radiation analysis for common types of finite elements plus implements new types of curved finite elements for even faster radiation analysis.

11) Implements a new thermal super element that simplifies a collection of complex finite elements into one or more SINDA/FLUINT nodes.

12) Provides graphical construction of procedural thermal model entities such heatpipes, heaters, and thermostats.

The main screen of the Thermal Desktop is shown in Figure 1. The figure shows the design geometry for an optical assembly. Commands may be typed at the command line, selected from pull-down menus or from user customizable tool bars.

The Thermal Desktop is implemented as an embedded application integrated with Autodesk's AutoCAD™ or Mechanical Desktop™ product. Integrating the application directly with the CAD environment provides the necessary link with the CAD world and provides viewing and model building functions at a very low cost.

An example of thermal analysis geometry constructed from design geometry is shown in Figure 2. The four views show the analysis geometry created from the solid model design geometry shown in Figure 1. These modeling entities were easily constructed by picking on key points in the design model. Sample results for the

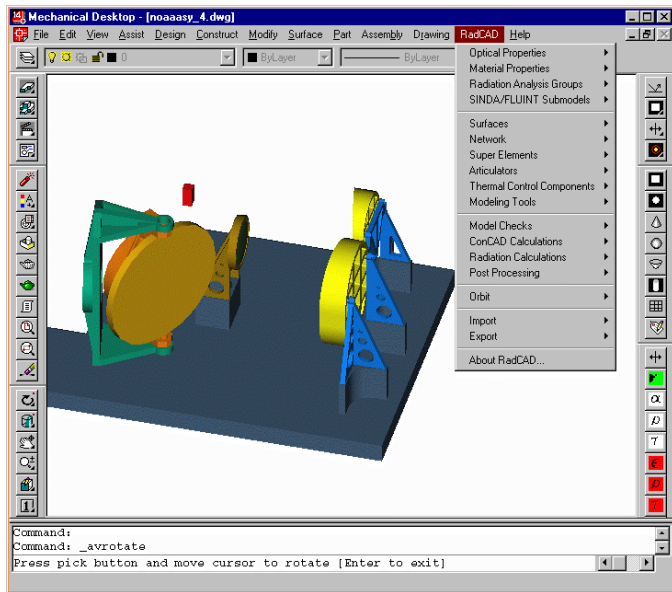calculation of radiation exchange factors are shown in Figure 3.

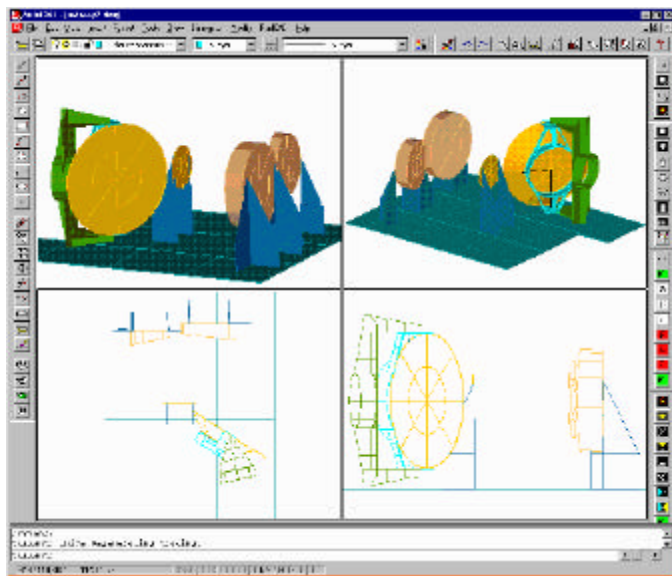Figure 1. Thermal Desktop main screen

Figure 2. Analysis geometry for example model

Thermal Desktop's layering feature allows design and analysis geometry to be viewed independently. Geometry may be placed on as many different layers as desired. The visibility of each layer may be controlled to aid in model building and postprocessing.

Thermal analysis geometry may be constructed and modified using two complementary methods. The first technique is to use dialogs invoked by the pull-down menu or tool bar icons. The editing dialogs contain fields to enter shape parameters and other thermal information. For example, the shape of a paraboloid may be modified by specifying the minimum and maximum radius, angular span and height. The second method takes advantage of the unique CAD features incorporated into the Thermal Desktop.
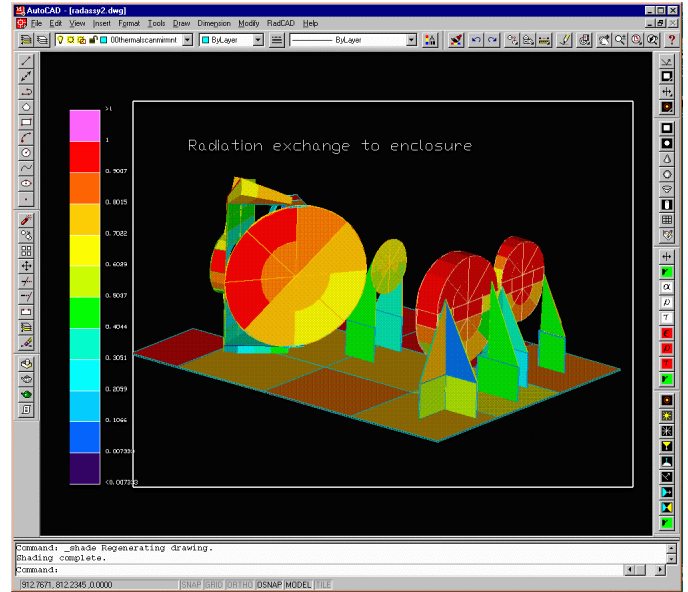

Figure 3. Thermal results for example model

Thermal analysis geometry is easily constructed from design geometry using two powerful features of the Thermal Desktop: snap points and grip points. Snap points are defined for all graphical objects and may be used whenever a point location is required for input. Snap points are located at key locations on an object such as the ends, middle, center, and intersections.

For example, the move command requires selecting the objects to be moved, and then inputting a "from" point and a "to" point. This point can be entered directly on the command line as an (x,y,z) triple, or the snap feature may be used to interactively select a location directly on the geometry. As the cursor is moved around the screen, snap points automatically highlight to show the user the available snap locations. An advanced expression utility may also be used to generate points not at snap locations (such as half way between two snaps).

Grips are used to modify a geometric object interactively. When an object is selected, the grip points appear, as shown by the small squares in Figure 4. Each grip point controls some aspect of the geometric object's location, shape, or size. When a grip is selected, the object will change dynamically as the cursor is moved about the graphics screen.

Figure 4 shows a Thermal Desktop (TD) cone surface. The cone is divided into three nodes in the height direction and two nodes in the angular direction. Solid lines show the nodal boundaries, and dotted lines pass through the nodal centers. The grip point at the base controls the base radius, and likewise, the grip at the top controls the top radius. Other grips are available for setting the angular span, height, location, and orientation of the surface.

If the base radius grip is selected, the cone will dynamically change size as the grip point is moved. The new location of the grip point may be entered at the

command line, arbitrarily placed on the current workplane with the mouse, or placed by selecting a snap point. For example, the radius of the cone can be made to precisely match a cylinder that it is on top of by selecting the cone radius grip, then moving the cursor over the cylinder until a suitable snap point is highlighted. Selecting the snap point completes the operation.
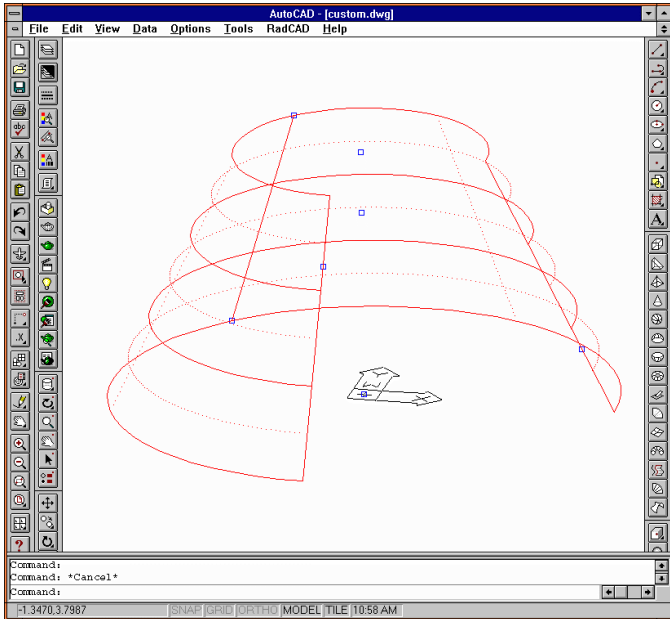

Figure 4. Snap points for TD cone

CONCURRENT ENGINEERING

A primary goal of the Thermal Desktop is to bring thermal analysis into the concurrent engineering fold. Two key areas must be satisfied for a successful concurrent engineering environment: integration with the design database, and exchange of analysis models and data between engineering disciplines.

Integration with the design database means the ability to generate analysis models directly from design geometry, and to react quickly to the inevitable changes in the design. Integration with other engineering disciplines means the ability to perform multi-discipline analysis from the same analysis model, or the tight coupling of data from one analysis model to another.

INTEGRATION WITH THE DESIGN DATABASE - The Thermal Desktop satisfies integration with the design database through its direct implementation in the CAD environment. Engineering shops using Autodesk's AutoCAD or Mechanical Desktop share the same drawing files that are generated by the designer. These design drawings can be linked externally to analysis files so that they are updated automatically when changes are made.

The Thermal Desktop was also designed to work with UNIX based CAD packages such as I-Deas or Pro/E. Thermal Desktop's IGES import capability allows analysis geometry to be constructed directly from the CAD design database. Design geometry is easily imported to form the basis for generating analysis models.

Geometry suitable for analysis may be directly converted to thermal analysis geometry. And, as is the usual case, a simplified representation of the design geometry for analysis is easily constructed using Thermal Desktop's snap/grip methods. Generating models by hand from 2D paper drawings, ruler, and calculator are replaced by efficient interactive graphical operations performed directly on the 3D design geometry.

The low cost of AutoCAD running on today's inexpensive and powerful PC's is an attractive alternative to costly high-end UNIX systems. AutoCAD does not compare to higher end solutions in terms of the functionality for the designer, but provides all of the needed viewing, IGES import, and CAD building methods for constructing analysis models. In addition, the evolution of the PC platform from a 16-bit DOS environment to a true 32-bit multitasking system has allowed AutoCAD to significantly expand its performance and functionality compared to previous versions.

The user interface and modeling features specifically designed for thermal engineers and AutoCAD's relative simplicity makes for a much faster learning curve compared to training engineers as designers on more expensive CAD platforms. AutoCAD is included as part of the Thermal Desktop.

INTEGRATION WITH OTHER ENGINEERING DISCIPLINES - A typical scenario in engineering organizations is the use of a CAD package that includes an integrated tool for constructing Finite Element Method (FEM) models. Some FEM modelers have the capability of basing meshes on design geometry and have the ability to automatically remesh based on design changes, a useful capability for rapidly responding to design changes.

Engineers use the FEM tool bundled with their in-house CAD system for model building and postprocessing. A typical pattern of usage is to export the FEM model for use with discipline specific "solvers" and then import results from their solver for postprocessing and data exchange.

The Thermal Desktop has solved the problems that have lead to the resistance of thermal analysts to adopt this engineering analysis approach. The Thermal Desktop may import and analyze FEM models directly, without the typical ad-hoc "element centroid" conversion process. FD and FEM methods are supported simultaneously, within the same thermal model, and temperatures are predicted using the industry standard thermal analyzer, SINDA/FLUINT. Thus, Thermal Desktop integrates into the CAD-based engineering process in the same way as tools that support analysis for other engineering disciplines.

The Thermal Desktop may be used as the engineering organization's thermal "solver", importing FEM meshes and performing radiation, orbital heating, and conduction and capacitance calculations. This data is then used by SINDA/FLUINT for steady state and transient temperature analysis. SINDA/FLUINT routines for producing appropriate input files to FEM packages are bundled with the Thermal Desktop so that temperature data may be transferred to other engineering discipline's models.

However, integrating with the standard in-house FEM package is only one mode of usage of the Thermal Desktop. Thermal Desktop contains the ability to generate FEM meshes directly, perform model-to-model data translation, and provides full postprocessing operations. A unique feature of Thermal Desktop is that it supports both traditional types of FD modeling primitives (conic surfaces, arbitrary nodes, and arbitrary conductors) and FEM modeling primitives simultaneously. This can often lead to advantages in system level modeling.

A full system level model constructed from a detailed FEM model will consist of many more nodes than a model constructed using traditional approaches. With the Thermal Desktop, specific areas of the hardware to be analyzed may be modeled using FEM, and other areas modeled using FD methods. For example, in an integrated thermal-structural-optical analysis of a telescope, the optical components can be modeled using FEM so that data may be transferred for thermal distortion analysis and optical performance predictions. The surrounding enclosure and spacecraft may be modeled using more CPU efficient FD methods. The geometric basis for the model may come from both design geometry and other engineering discipline's FEM models.

Despite pressures from the need to integrate tighter with other engineering disciplines, FE methods have been resisted by the thermal analysis community. One of the reasons is a perception that FE methods are not as physically based as traditional FD approaches (in part because of the generation of "negative conductors", which are viewed as physically unrealistic). The other reason is simply that current FE codes do not satisfy all of the requirements for performing system level thermal analysis. However, a failure on the part of FE *codes* to supply needed functionality does not negate the advantages of the finite element *method*.

Integrating FEM into current thermal analysis approaches, including a new first law formulation of the method was presented in detail in reference [4]. The derivation and approach will not be repeated here, but a few of the extensions to the finite element method implemented in the Thermal Desktop that make its application appropriate for thermal analysis are presented in the following section.

THERMAL SPECIFIC EXTENSIONS TO FEM

A common complaint against FEM is that an excessive number of nodes are generated. Nodalization is often driven more by geometric fidelity to the actual hardware rather than by thermal accuracy concerns. For example, a curved surface must be approximated by many flat elements. Traditional modeling primitives such as cylinders and cones allow nodal regions to be curved, permitting only the number of nodes as necessary for thermal accuracy to be used, while retaining geometric fidelity.

Typical FEM meshing utilities operate on detailed solid model representations of the design, leading to meshes that capture unnecessary detail for thermal analysis. Details of interest to a structural engineer calculating stress in a part may not be important thermally.

The Thermal Desktop employs two unique features to reduce the complexity of models generated using FEM. One is the formulation of a new set of thermal specific linear elements on curved surfaces. The other is a new thermal super element formulation.

THERMAL SUPER ELEMENTS - Thermal super elements may be constructed from any set of ordinary finite elements. A subset of ordinary nodes in the super element may be specified to be super nodes. The Thermal Desktop will compute conductance and capacitance data for the super nodes on the super element for use in SINDA/FLUINT. Correspondence between the super nodes and the sub nodes on the super element is maintained so that temperatures may be assigned to the sub nodes for post processing and for mapping data to other analysis models.

An example may illustrate the process. Consider the bracket shown in Figure 5, which contains holes to reduce weight. Suppose that the bracket is thermally attached only at the four end points.
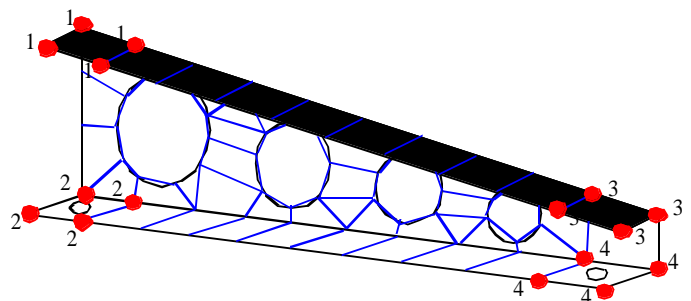

Figure 5. Detailed mesh simplified with super element

Thermally, only the conduction paths between the four mounting points are of interest. Using the traditional approach, the temperature distribution within the part is deduced, and equivalent conduction areas and lengths are calculated. Such estimation often requires considerable skill and experience to arrive at an accurate approximation.

Using thermal super elements, the part is first discretized with enough elements to capture the geometric detail and the set of elements is labeled as a super element. Next, super nodes are assigned to selected sub nodes of the mesh, which in this case are the four mounting locations. Conduction and capacitance data for the four super nodes are calculated automatically by Thermal Desktop. The super node data is then used by SINDA/FLUINT for temperature calculations.

Super elements work by numerically calculating the form of the temperature distribution over the super element. A Numerically computed shape function for each super node is used for the calculation of conduction and capacitance terms using a first law apportioning method[4].

In a regular finite element, the temperature distribution over the domain of element is assumed to have some simple mathematical form, such as a bilinear interpolation for a quadrilateral element. In a super element, the temperature distribution is not approximated by simple interpolation, but rather by a temperature solution calculated using the entire super element mesh.

The shape function for each super node is derived in turn by performing a steady state solution with the super node set to unity and the remaining super nodes set to zero. Because the problem is linear, superposition holds and the resulting shape functions can be used to compute the steady state temperature distribution for any set of super node temperatures.

Figure 6 shows the temperature distribution computed using super element shape functions compared to the temperature distribution performed using the full mesh. As can be seen, the temperature distributions are exact. The shape function for one of the super nodes is shown as a height plot in Figure 7.

In this example, the nodes along each flat edge were specified as super nodes, reducing the part down to three SINDA/FLUINT nodes. The choice of super nodes gives Thermal Desktop additional knowledge of the temperature distribution in the super element, which is used to reduce the complexity of the part. Specifying that all of the nodes along an edge are a single super node tells Thermal Desktop that the temperature distribution is isothermal in this area. The remaining edges are assumed to be adiabatic.

If this region were incorporated into a larger surface, the assumption of an adiabatic edge may not be correct. The Thermal Desktop also allows sub nodes to be specified as being interpolated from super nodes. For example, the temperature distribution along a curved edge can be specified to be linearly interpolated from the end points, which are specified to be super nodes.

This allows the computer to perform the work that is normally done by hand. The specification of super

elements and super nodes provides Thermal Desktop with additional knowledge about the form of the resulting solution, which is used to reduce the number degrees of freedom in the problem.
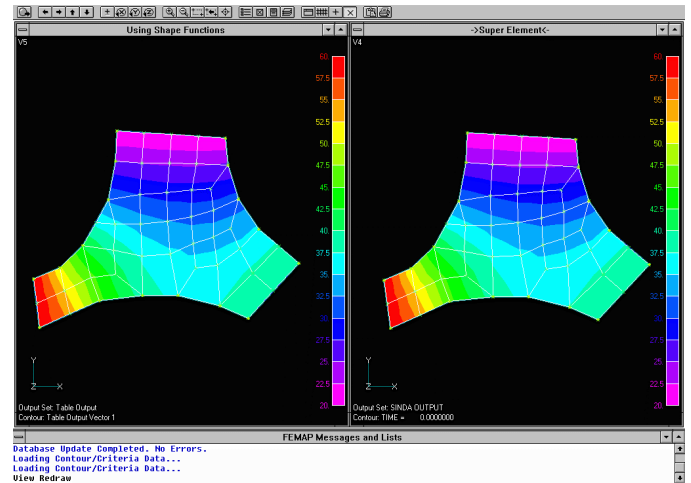

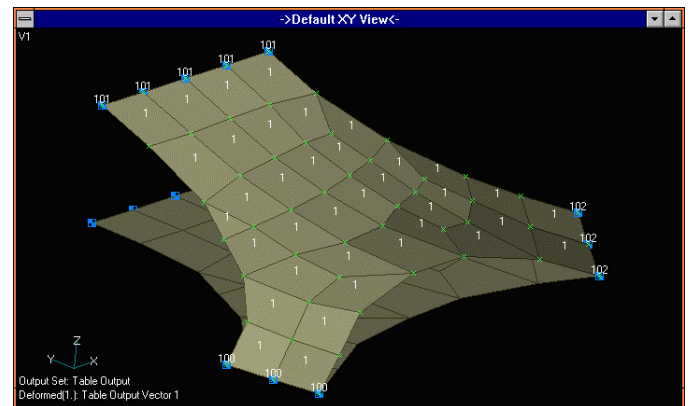Figure 6. Comparison of super element solution

Figure 7. Numerically computed shape function

The choice of super elements and super nodes is governed by the same considerations that the engineer uses in nodalizing any thermal problem. In a macro sense, energy is always conserved, and "bulk" properties of the region are accurate regardless of the size of the super element. Like FD nodes and regular finite elements, larger regions trade local fidelity for faster run times.

LINEAR CURVED FINITE ELEMENTS - The Thermal Desktop has integrated finite element meshes with finite difference based modeling. Thermal Desktop's first law based formulation of FEM shows that nodes represent control volumes and can be used in conjunction with traditional modeling methods. The lines that connect the element centroids delimit the control volume for a node. Direct radiation analysis of FEM meshes also avoids problems associated with ad-hoc conversion of valid FEM meshes to element-centroid networks.

Radiation analysis is performed using the shape functions to weight ray energy. When a ray is emitted from an element, its energy is determined by the shape

functions for each of the nodes on the element. When a ray strikes an element, the energy is distributed to each of the nodes on the element using the receiving element's shape functions[5]. Thus, radiation is modeled *non-isothermally*, providing better accuracy for the same nodal density as codes that are restricted to isothermal radiative exchange.

The problem remains; however, of the need to use many flat elements to approximate curved surfaces. Curved element formulations do exist, implemented with higher order shape functions. However, these parabolic elements are still more complicated than necessary for thermal concerns.

Nodal densities are greatly increased compared to using familiar TRASYS surfaces. In a radiation dominated problem, this can quickly lead to intractable models. The obvious solution is to implement curved, linear finite elements. The advantages for postprocessing, model-to-model data mapping, and conduction and capacitance generation are gained without sacrificing radiation performance.

Imagine a conduction problem in which a flat region subjected to some boundary conditions produces a temperature distribution. This temperature distribution does not change if the flat surface is somehow made into a curved surface (without stretching). Topologically, the problems are equivalent. As another example, conduction terms calculated by hand for a cylinder are identical to conductors calculated for a flat plate.

However, the problem is entirely different when one considers the stress problem. A flat plate subjected to external forces responds considerably different than a cylindrical section. This is the reason this simple solution has not been implemented in present FE based tools.

Most FE codes have their origins in solving structural problems. To implement a custom type of element that satisfies a narrow market of radiation dominated thermal problems has not been a priority among providers of FEM based tools.

The Thermal Desktop implements curved finite element based surfaces in the same manner as traditional conic FD based surfaces. From a modeling standpoint, the only difference is that nodal points span the entire surface, rather than just lying in interior regions. Each node is still viewed as being the center of a control volume, to which any other types of conduction, convection, and radiation connections may be made.

The main advantage of FEM conic surfaces over regular TRASYS-like surfaces becomes apparent when these surfaces are to be coupled together conductively. Conduction modeling based on FD surfaces requires artificial nodes to be generated at the boundaries of the surface. Sometimes these artificial intermediate nodes may be eliminated from the model, other times they may not. FEM based surfaces eliminate many ambiguous

conditions and provide a simpler interface since intermediate nodes do not have to be created. For surfaces that share a common edge, or solids that share a common face, nodes occupying the same location can be merged into the same node, or connected together through a contact conductance.

For compatibility with existing models and for those that prefer the familiar FD approach, Thermal Desktop also supports full conduction and capacitance generation for FD based radiation surfaces.

INTEGRATING THERMAL MODELING APPROACHES

The Thermal Desktop simultaneously supports FD, FEM, arbitrary network, and procedural modeling methods. An advanced radiation analysis tool, RadCAD, is integrated within the Thermal Desktop. RadCAD works with both FD and FEM based geometry. The use of the industry standard thermal analyzer, SINDA/FLUINT is retained, allowing complete user control and simulation capabilities along with advanced fluid flow modeling.
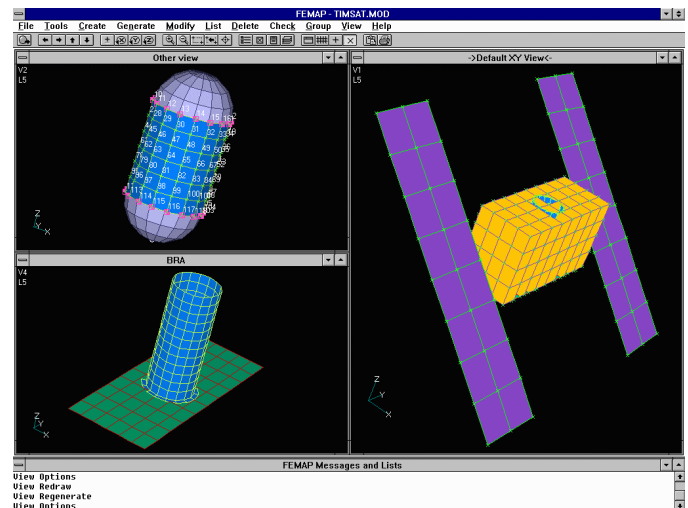

Figure 8. Simple FD/FEM test model

An example analysis of a fictitious spacecraft is presented to illustrate the combined use of these modeling methods simultaneously in a thermal analysis model. Figure 8 shows a simple box shaped spacecraft with two solar panels. Inside the box is a battery, mounted in a sleeve that is attached to a radiator panel at the base.

The battery was converted into a two node super element using the top and bottom edges of the cylindrical section as the super nodes. The battery was placed into the sleeve with the battery mesh and the sleeve mesh nodes non-aligned.

The interior of the sleeve and the base of the sleeve were specified as having a contact conductance. The Thermal Desktop performed the calculations to connect the sleeve to the battery and the sleeve to the radiator panel through these contact surfaces. A heatpipe on each side of sleeve connects the outside of the sleeve to

the radiator panel. The heatpipe was connected using FD calculated conductors.

The edges of the four long rectangles of the enclosing box were coupled together with a high contact conductance. The square ends were coupled to the long section with a thermally poor contact conductance. A single network type conductor modeled the conduction between the solar panel and the spacecraft body.

Radiation conductors were computed for both the interior and exterior of the spacecraft. Orbital heating rates were computed and the temperatures predicted using SINDA/FLUINT. Contour plots of the analysis results are shown Figures 9-10.
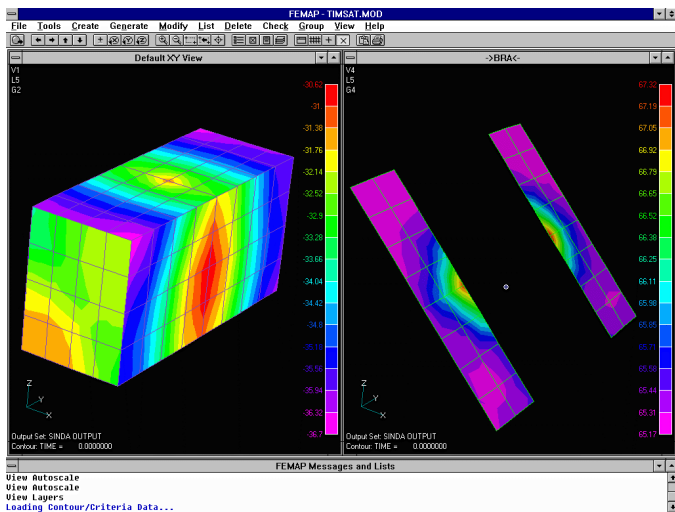
Figure 9. Temperature results for exterior of model

The radiation exchange with the warm battery and the enclosure can be seen, as well as the interaction between the solar panels and the ends of the box. The effects of the heatpipe on the battery sleeve and radiator panel are also apparent.
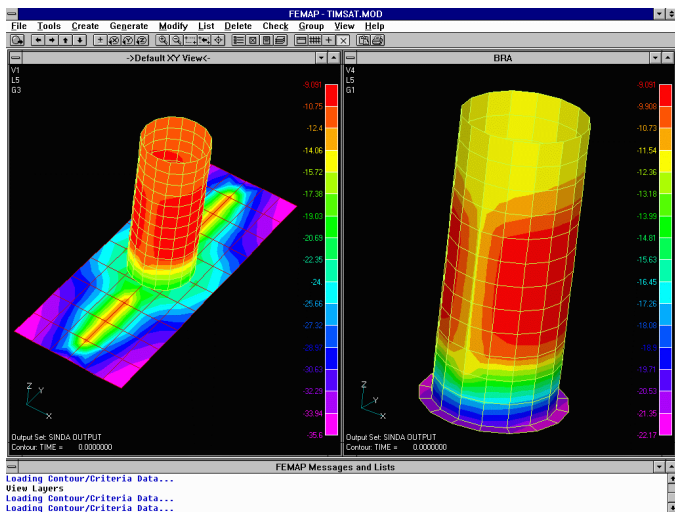

Figure 10. Temperature results for interior of model

CONDUCTION/CAPACITANCE GENERATION

An overview of the user's interaction with the Thermal Desktop has been presented in previous sections. This section presents a few of the user interface dialog forms and techniques for controlling the generation of conduction/capacitance data.

Analysis geometry is edited by selecting the geometry with the mouse and choosing the edit command from the pull-down menu or by clicking the edit icon on the toolbar. A single entity may be modified, or a group of entities may be edited together.

The edit command invokes the tabbed dialog shown in Figures 11-13. This form contains tabs for setting the nodal breakdown, numbering, optical properties, and active sides for radiation analysis. The "Surface" tab allows parameters to be input for the shape of the surface. The "Cond/Cap", "Contact", and "Insulation" pages are used for conductor and capacitance generation.

Nodal capacitances and intra-entity conduction hookups are controlled by the "Cond/Cap" page, as shown in Figure 11. Node-to-node conductors for nodes on the solid or shell entities will be generated using the data input on this page.

Surfaces have the additional capability of being modeling as a simple thin shell, or as two shells separated by a small distance coupled with a "through" conductivity. For example, the conduction and capacitance terms for a honeycomb panel constructed from two face sheets and a core material may be modeled by a single graphical entity. Orientation angles may be specified for anisotropic materials.
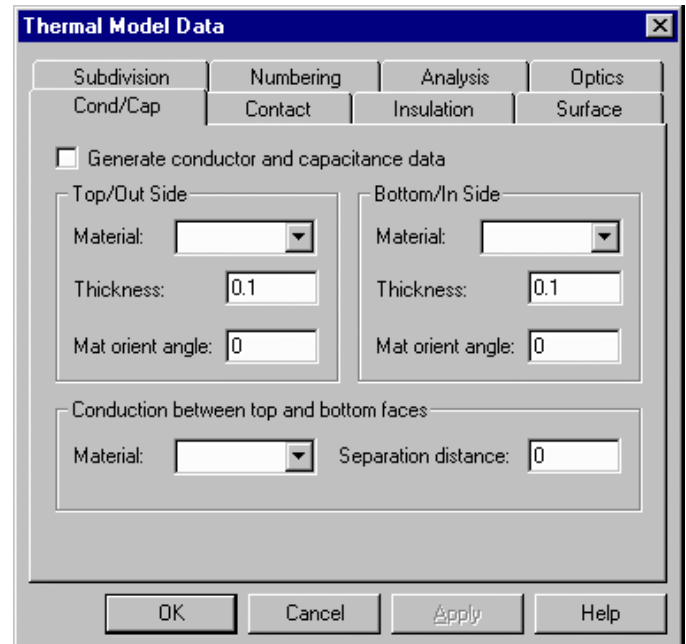

Figure 11. Conduction/Capacitance parameters

Figure 12 shows the input page to control contact conductance. Contact boundary conditions may be applied to faces or edges of thermal modeling entities. For example, the base of an electronic box on a panel may be specified to have a certain contact conductance. Likewise, the edges of surfaces used to represent computer cards that attach to the panel may also be specified as having a contact conductance. Thermal Desktop will integrate along the faces and edges of these entities, searching the other entities in the model, and generate the appropriate connections to adjoining nodes. The nodal meshes do not have to be aligned.

Insulation is often used in spacecraft and cryogenic applications. Multi Layer Insulation (MLI) is commonly used for spacecraft, relying on reducing the radiative coupling to the environment. Foam type insulations are often used in cryogenic applications. The input form permits insulation to be placed on the top or bottom sides of a surface, or selected sides of a solid. The insulation can be characterized by a combination of effective radiative emissivity and effective conductivity.
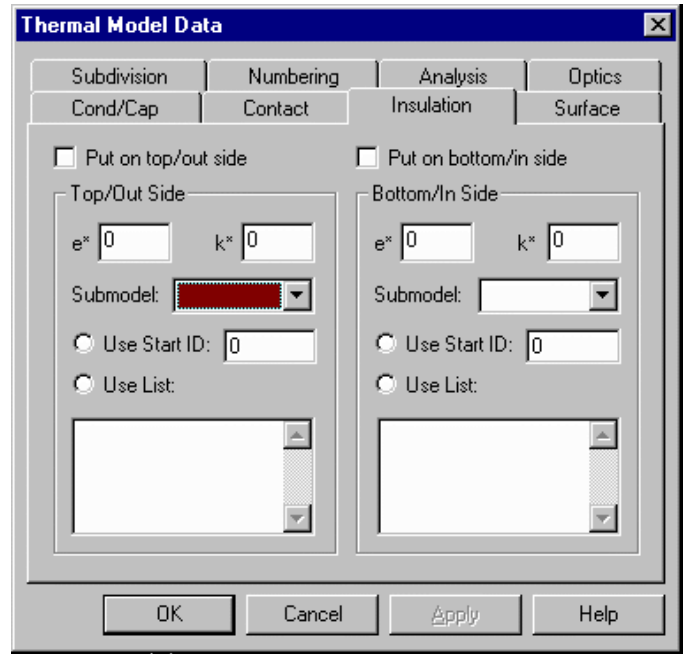

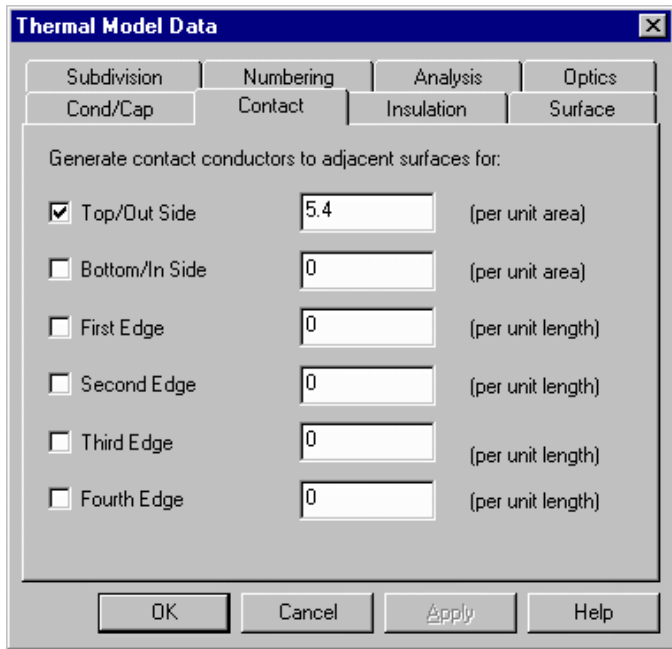Figure 13. Surface insulation parameters

Figure 12. Contact conductance parameters

Conduction and capacitance data along with analysis results may be graphically displayed, as shown in Figure 14. Spheres are drawn at the nodal centers, and tubes are drawn to represent node-to-node heat flow paths.

Both size and color are used to represent nodal and heat flow data. Five different quantities may be displayed simultaneously using the color of the nodal surface, the color and size of the nodal sphere, and the color and size of the nodal heat flow path.
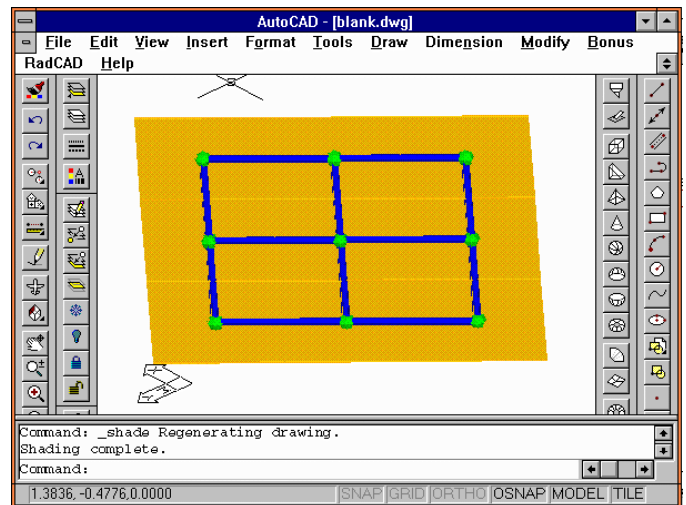

Figure 14. Conductor/Capacitance postprocessing

RADIATION MODELING

Thermal Desktop may be used for conduction and capacitance generation and results postprocessing. An advanced radiation analysis module, RadCAD, is optionally available for calculation of radks and orbital heating rates. RadCAD works with both traditional conic surfaces and with FEM generated meshes. Import and Export of TRASYS models is supported.

RadCAD employs both raytracing and a unique progressive radiosity algorithm. Specular, angular dependent and refractive optical properties are supported. Articulating geometry including star, sun, and planet tracking are provided. An overview of RadCAD and its unique computational algorithms are described in reference [6].

SUMMARY

A new thermal analysis system has been presented that addresses the problems associated with integrating thermal engineering into the concurrent engineering environment. Present radiation modeling techniques are preserved and directly integrated into a CAD environment.

Conduction and capacitance generation from these surfaces along with full support for FE models expands the set of modeling tools available to the thermal engineer and permits closer coupling with other engineering disciplines.

New types of custom finite elements have been developed to address specific thermal needs, as well as a unique super element formulation to reduce complicated meshes into a simpler SINDA/FLUINT network. Thermal Desktop along with SINDA/FLUINT forms a complete thermal analysis and fluid flow solution, that also integrates with existing in-house CAD systems and FE based tools.

The most up to date information regarding the release and availability of Thermal Desktop can be found at http://www.webcom.com/crtech.

REFERENCES

1. Lockheed Engineering and Management Services Company, "Thermal Radiation Analysis System Users Manual," JSC 22964, Contract NAS 917900, October 1991.
2. Cullimore, B., "Optimization, Data Correlation, and Parametric Analysis Features in SINDA/FLUINT Version 4.0", SAE paper 981547, 28th ICES Conference, July 1998.
3. Panczak, T., et al., "Thermal Synthesizer System: An Integrated Approach to Spacecraft Thermal Analysis," SAE paper 911582, 22nd ICES Conference, July 1992.
4. Panczak, T., "The Failure of Finite Element Codes for Spacecraft Thermal Analysis," SAE paper 961450, 26th ICES Conference, July 1996.
5. Chin, J. H., Panczak, T. D., and Fried, L., "Finite Element and Raytracing in Coupled Thermal Problems," *Proceedings of the Sixth International Conference on Numerical Methods in Thermal Problems*, pp. 683-791, Pineridge Press, Swansea, U.K. 1989.
6. Panczak, T. and Ring, S., "RadCAD: Next Generation Thermal Radiation Analyzer", SAE paper 972241, 27th ICES Conference, July 1997.