

FEM/SINDA:

COMBINING THE STRENGTHS OF MSC/NASTRAN, SINDA, SUPERTAB, AND PATRAN FOR THERMAL AND STRUCTURAL ANALYSIS

P. Richard Zarda, Ted Anderson, Fred Baum
Martin Marietta Missiles Systems
Computer Aided Analysis Group
P.O. Box 5837 MP 135
Orlando, Florida 32855-5837
(407)356-5715, or (407)356-2288

ABSTRACT

This paper describes the interface/integration between FEM/SINDA, a general purpose geometry driven thermal analysis code, and the FEM software: SUPERTAB, PATRAN, and MSC/NASTRAN. FEM/SINDA brings together the advantages of the finite element method to model arbitrary geometry and anisotropic materials and SINDA's finite difference capability to model thermal properties, loads, and boundary conditions that vary with time or temperature. SUPERTAB and PATRAN thermal entities are directly supported since FEM/SINDA uses the nodes of the FEM model as the point at which the temperature is determined. Output from FEM/SINDA (as well as the FEM/SINDA input deck) can be used directly by MSC/NASTRAN for structural analysis.

INTRODUCTION

The industry standard thermal analysis codes SINDA and MITAS are known for their versatility in solving a wide range of thermal analysis problems. The input to these codes, however, generally involves tedious hand calculations of nodal capacitances and conductances. The CAE group at Martin Marietta Missile Systems in Orlando, Florida has developed a finite element - finite difference hybrid thermal analysis code which can take finite element models developed in SUPERTAB or PATRAN and produce a finite difference network model which is then solved with MITAS, Martin Marietta's version of SINDA (from this point forward, any reference to SINDA implies MITAS as well).

A FEM/SINDA input deck can be generated from an I-DEAS universal file or a PATRAN neutral file using the SUPERTAB-to-FEM/SINDA translator or the PATRAN-to-FEM/SINDA translator. FEM/SINDA can then be run to produce *nodal temperatures at the finite element nodes*. The solution algorithm to determine the nodal temperatures is SINDA's finite difference network solution. The conductors and capacitances used in the SINDA network solution are mathematically equivalent to the thermal conductivity and thermal capacitance matrices generated by using finite element techniques. *A node in the finite element model will necessarily be a node in the SINDA model.*

This method will allow, at the SUPERTAB or PATRAN level, the mixing of 1-D (rod), 2-D (shell) and 3-D (solid) elements and will generate the conductivity network that this connectivity implies.

Copyright©1988 Martin Marietta Corporation, all rights reserved. Published by MSC, with permission.

SUPERTAB is a registered trademark of SDRC. PATRAN is a registered trademark of PDA Engineering.

MSC/NASTRAN is a registered trademark of MSC. FEM/SINDA is a trademark of Martin Marietta Corp.

This is in direct contrast to centroidal methods which require the creation of additional elements when mixing 1-D, 2-D and 3-D elements (for example, the connection between a shell coming into two nodes of a solid requires the creation of one or more shell elements on the face of that solid).

Working with the true finite element nodes (versus the centroidal node) also allows boundary conditions to be easily handled. Specified temperatures can be applied at the finite element nodes which are generated on the true boundary of the object. Applying boundary conditions to centroidal nodes can lead to erroneous answers since the node location is probably not at the proper boundary. In addition, the thermal boundary conditions and loads (such as convection, heat fluxes, radiation, etc.) can be specified in SUPERTAB or PATRAN using the current entities available in each of the pre-processors. In SUPERTAB or PATRAN the user can also specify whether the properties are isotropic or orthotropic, and whether they are constant or vary with temperature. Boundary conditions and loads are also specified by the modeler to either be constant or vary with time and/or temperature.

FEM/SINDA will automatically generate a SINDA input deck for the subsequent finite difference analysis. This deck can be automatically combined with a SINDA deck that has, for example, a table that could specify how a thermal property (for example, a thermal conductivity already flagged in SUPERTAB or PATRAN) would vary with temperature. The complete flexibility of SINDA is therefore available to the thermal analyst. Use of FORTRAN subroutines and tables to account for thermal properties or boundary conditions that vary with time and/or temperature is one of the strengths of SINDA.

FEM/SINDA is also integrated with TRASYS, a well-known code (developed by Martin Marietta) for determining both radiation view factors and solar and planetary heat fluxes. TRASYS has over ten years of development activity and is an industry standard. The SUPERTAB or PATRAN user can simply select which faces of shell or solid element radiate. FEM/SINDA will generate the necessary input deck to TRASYS for view factor calculations. A subsequent TRASYS run will return SINDA radiation conductors. These radiation conductors will reflect the view factors between the various radiating elements selected in SUPERTAB or PATRAN. Moreover, the radiation conductors are between the finite element nodes and can be combined with the SINDA deck of thermal conductors for a system analysis involving conduction, convection and radiation.

Existing SUPERTAB or PATRAN stress and dynamic models may also be used, with some or no modification, to drive FEM/SINDA. This will then insure, for example, that the temperature field is determined at the nodes of a stress model. A subsequent thermal stress analysis is therefore automatic since nodal temperatures are available. A centroidal method, on the other hand, would require the interpolation/extrapolation of the centroidal temperatures to determine the nodal temperatures – a possible source of misinterpretation and/or error.

Output from FEM/SINDA (either steady state or transient analyses) can be brought back into SUPERTAB or PATRAN for post-processing (also available is the ability to read a FEM/SINDA input deck into SUPERTAB or PATRAN). Another feature of FEM/SINDA is that the input deck can be either in free field and/or fixed field, and the card image format is almost identical to a NASTRAN input deck. Existing NASTRAN decks, with slight modification, could therefore be used as input to FEM/SINDA.

In short, the integration of SUPERTAB, PATRAN and MSC/NASTRAN with FEM/SINDA for thermal analysis combines the power of finite element pre- and post- processing and discretization techniques with the industry accepted SINDA code, taking advantage of the strengths of both while preserving completely the conventional input to SINDA. This allows the FEM user to completely specify his/her thermal model in SUPERTAB or PATRAN (conduction, convection, radiation) and

allows for boundary conditions, loads and thermal properties to vary with time and/or temperature.

FEM THEORY

In order to understand the basic architecture of FEM/SINDA, a short review of some of the basic techniques in finite element theory is in order. Consider the simple triangular element shown in Figure 1a.

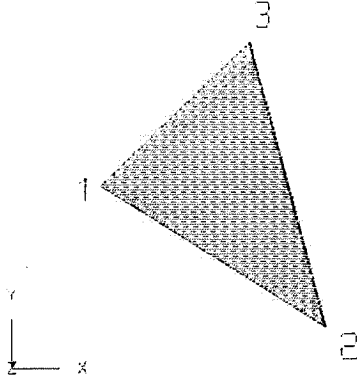


Figure 1a. Triangular Element

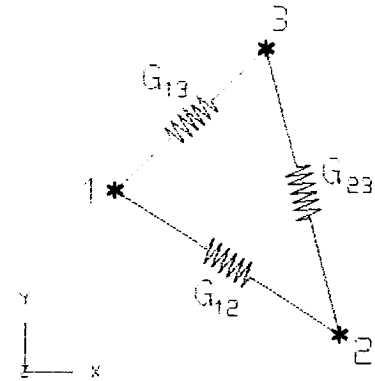


Figure 1b. Equivalent Conductors

The triangle has a constant thickness t and an isotropic thermal conductivity of k . The temperature field within the element is assumed to be a linear function of the nodal temperatures: T_1, T_2 , and T_3 . It can be shown (see Reference 1) that the temperature field T at any point (x, y) within the element is given by

$$T(x, y) = \frac{1}{2A} [a_1 + b_1x + c_1y \quad a_2 + b_2x + c_2y \quad a_3 + b_3x + c_3y] \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} \quad (1)$$

where

A = Area of triangle

t = Thickness of triangle

$$a_1 = x_2y_3 - x_3y_2 \quad (2)$$

$$b_1 = y_2 - y_3$$

$$c_1 = x_3 - x_2$$

and $a_2, b_2, c_2, a_3, b_3, c_3$, are obtained by permuting the indices in Equation 2 (for example, $b_2 = y_3 - y_1$). If the (x, y) coordinate in Equation 1 equals a nodal coordinate, $T(x, y)$ will reduce to that nodal temperature. Note also that the temperature field of equation (1) is linear.

Next, based on variational principles (Reference 1), the thermal conductivity matrix $[K]$ of the element can be determined. For this triangular element, it is given by (Reference 1)

$$[K] = \frac{kt}{4A} \begin{bmatrix} (b_1^2 + c_1^2) & (b_1b_2 + c_1c_2) & (b_1b_3 + c_1c_3) \\ & (b_2^2 + c_2^2) & (b_2b_3 + c_2c_3) \\ \text{SYM} & & (b_3^2 + c_3^2) \end{bmatrix} \quad (3)$$

Note that the matrix is symmetric and not all the values in the matrix are independent. It can be shown (based on the fact that a constant temperature can be maintained with no heat input) that the sum of the values on any row (or column) must add up to zero. Stated another way, the diagonal term on any row is minus the sum of all the off-diagonal terms of that row. Thus, for

$$[K] = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ & k_{22} & k_{23} \\ \text{SYM} & & k_{33} \end{bmatrix} \quad (4)$$

once the upper triangular values, k_{12} , k_{13} , and k_{23} are known, all the other entries are determined.

Next consider a conductor network between the same set of nodes as shown in Figure 1b. The conductor values, g_{12} , g_{13} , and g_{23} can be found such that this conductor network is equivalent to the finite element of Figure 1a and Equation 4. This can be shown by recalling that a conductor g between any two nodes A & B has a thermal conductivity matrix given by:

$$[K] = \begin{matrix} & \text{A} & \text{B} \\ \text{A} & \begin{bmatrix} g & -g \\ -g & g \end{bmatrix} \end{matrix} \quad (5)$$

The thermal conductivity matrix for the three conductors of Figure 1b is assembled by applying Equation 5 to each conductor. Then the assembled 3 x 3 conductivity matrix for the three conductors of Figure 1b is

$$[K] = \begin{matrix} & \begin{matrix} 1 & 2 & 3 \end{matrix} \\ \begin{matrix} 1 \\ 2 \\ 3 \end{matrix} & \begin{bmatrix} G_{12} + G_{13} & -G_{12} & -G_{13} \\ -G_{12} & G_{12} + G_{23} & -G_{23} \\ -G_{13} & -G_{23} & G_{13} + G_{23} \end{bmatrix} \end{matrix} \quad (6)$$

Notice that the conductors are assembled in the matrix consistent with the conduction matrix of Equation 5. Also the matrix exhibits the topology of all conductivity element matrices: the matrix is symmetric and the sum of the off-diagonal terms on any row is equal to minus the diagonal term of that row. Finally, the conductivity matrix of the finite element of Figure 1a will exactly match that of the conductor network of Figure 1b by equating Equation 6 to Equation 4. Only the upper triangular terms need to match (all the others will then necessarily match). This gives

$$\begin{aligned} G_{12} &= -k_{12} \\ G_{13} &= -k_{13} \\ G_{23} &= -k_{23} \end{aligned} \quad (7)$$

Equation 7 simply states that the conductor value between any two nodes i and j is simply minus the off-diagonal i - j term of the thermal conductivity matrix of that element. That is,

$$G_{ij} = -k_{ij} \quad (8)$$

Equation 8 applies not only for the triangular element but for any element. For example, Figure 2a shows a quadrilateral shell element, and the six conductors between the four nodes exactly correspond to the six upper triangle values of the thermal conductivity matrix shown in Equation 9.

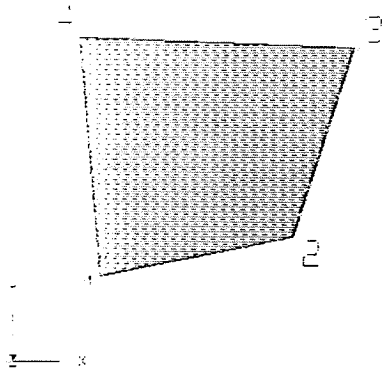


Figure 2a. Quadrilateral Element

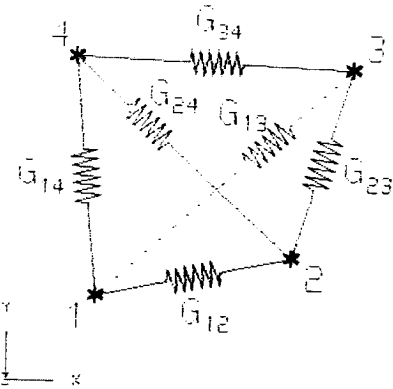


Figure 2b. Equivalent Conductors

$$[K] = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ & k_{22} & k_{23} & k_{24} \\ & & k_{33} & k_{34} \\ \text{SYM} & & & k_{44} \end{bmatrix} \quad (9)$$

For any element (rod, shell or solid) the thermal conductivity matrix can be determined and the conductivity network is given by Equation 8. The thermal conductivity matrix for most elements can be found in either Reference 1 or Reference 2. Once the conductor network for each element is determined, FEM/SINDA looks towards SINDA (a finite difference code) for solving the system of equations. This is in direct contrast to a finite element code that generally solves a linear system of equations of the form

$$[K]\{T\} = \{Q\} \quad (10)$$

where $[K]$ is a thermal conductivity matrix of size N (N is the total number of nodes in the model), $\{T\}$ is a vector of nodal temperatures, and $\{Q\}$ is a vector of nodal heat flows. The finite element method requires first the assembly of the system thermal conductivity matrix $[K]$ of Equation 10 and then the simultaneous solution to the set of Equations 10. FEM/SINDA does not assemble the matrix $[K]$. It simply determines the conductivity matrix of an individual element and then generates the appropriate SINDA conductors. The SINDA conductors can vary with time or temperature and hence handle nonlinearities that are common in thermal analysis. On the other hand, finite element techniques are not nearly as efficient (or even capable) in handling nonlinearities (MSC/NASTRAN thermal analysis package, for example, is significantly slower than SINDA in solving nonlinear transient problems, and will not handle something as fundamental as a heat transfer coefficient varying with time).

A code such as SINDA requires as input the conductor value between two nodes. For the triangular element of Figure 1a, Equation 3 (applying Equation 8) gives the conductor values. Thus the three conductors are

$$\begin{aligned}
G_{12} &= k \left[\frac{t}{4A} (b_1 b_2 + c_1 c_2) \right] \\
G_{13} &= k \left[\frac{t}{4A} (b_1 b_3 + c_1 c_3) \right] \\
G_{23} &= k \left[\frac{t}{4A} (b_2 b_3 + c_2 c_3) \right]
\end{aligned}
\tag{11}$$

These values (those given by Equation 11) can be input into SINDA in one of two ways. If the thermal conductivity k is constant, FEM/SINDA will generate the following SINDA card:

CONDUCTOR #, NODE_{*i*}, NODE_{*j*}, G_{*ij*}
EXAMPLE: 37, 2, 3, 4.278

where the CONDUCTOR # is some unique label number, NODE_{*i*} and NODE_{*j*} are the nodes that the conductor is between, and G_{*ij*} is the conductor value which is given by Equation 11. If k is not constant (but is to vary with temperature) the following SINDA card is generated by FEM/SINDA:

CGS CONDUCTOR #, NODE_{*i*}, NODE_{*j*}, ARRAY #, (A/L)_{*ij*}
EXAMPLE: CGS 97, 1, 3, A4, 0.789

where CGS implies a conductor that will vary, the ARRAY # (in the example, array A4) is a table of conductivity vs. temperature that specifies how the thermal conductivity is to vary with temperature, and (A/L)_{*ij*} (a single number) is the "geometric part" of the conductor and is the term in brackets in Equation 11. The table of k vs. T is added separately to the SINDA deck.

Capacitance for each node of each element uses the "lumped mass" approach that is often used in finite element structural analysis. Essentially this means that, for the triangular element of Figure 1a, each node is assigned 1/3 of the mass of that element. For other elements the lumping of mass (and hence capacitance) is similar and can be found in Reference 1 and 2. FEM/SINDA will automatically generate the appropriate capacitance for SINDA.

The above procedure for determining the "finite element" conductors and capacitances for each element is used in a similar way to handle convection and radiation. Convection and radiation will lead to additional conductors in the network and will automatically be generated by FEM/SINDA. SINDA radiation conductors can also include view-factor calculations based on a TRASYS run (the conductors are automatically generated by the TRASYS run).

FEM/SINDA will generate the conductors for each element used in the FEM model. When two elements produce conductors between the same nodes, those conductors are combined (in cases where the conductors are not constant but are referencing a different thermal conductivity, they are not combined). The sorting and summing is performed by FEM/SINDA not only for conductors (conduction, convection and radiation conductors) but also for capacitance and loads. This will generate a compact conductor network for the subsequent thermal analysis.

SUPERTAB and PATRAN MODELING

The thermal analyst can define his/her entire thermal model within SUPERTAB or PATRAN and then subsequently generate a FEM/SINDA input deck. The key to the ease of generating a FEM/SINDA input deck from a FEM model is simple: a node in the FEM model will necessarily be a node in the

SINDA network. The SUPERTAB entities available in I-DEAS 4.0 and the PATRAN entities available in PATRAN 2.3 will, in general, be used to directly to drive the FEM/SINDA model. In particular, the SUPERTAB and PATRAN entities shown in Table 1 are directly supported by FEM/SINDA.

SUPERTAB/PATRAN entity	FEM/SINDA entity
Cartesian coordinate system	CORDR
Cylindrical coordinate system	CORDC
Spherical coordinate system	CORDS
Isotropic material table	MATI
Orthotropic material table	MATO
Spring physical table	PCOND
Rod/Bar physical table	PROD
Shell physical table	PSHELL
Solid physical table	PSOLID
Node	NODE
Node-to-node translational spring	CONDUCT
Lumped mass	CAPAC
Linear rod/bar	ROD
Linear thin-shell triangle	TRIA
Linear thin-shell quadrilateral	QUAD
Linear solid tetrahedron	TETRA
Linear solid wedge	WEDGE
Linear solid brick	BRICK
Nodal heat source	NHEAT
Edge influx/Dist. heat source	EFLUX
Face influx/Dist. heat source	FFLUX
Distributed heat generation	VFLUX
Edge convection	ECNVECT
Face convection	FCNVECT
Edge radiation	ERADS
Face radiation	FRADS or FRADT
Nodal temperature	TEMP

TABLE 1. SUPERTAB/PATRAN entity and corresponding FEM/SINDA entity.

The property and material values in SUPERTAB can be used and the corresponding FEM/SINDA input deck will be properly generated. Some of the material values that are supported in SUPERTAB directly are isotropic and orthotropic thermal conductivity, specific heat and material density. Convective heat transfer coefficients and the emissivities (for radiation calculations) are also input directly in SUPERTAB in the "ANALYSIS_CASES" task. Note that by supporting both edge entities and surface entities (as Table 1 shows) both 2-D and 3-D models can be fully generated in SUPERTAB and analyzed by FEM/SINDA. Heat loads, convection and radiation can be applied using SUPERTAB's heat transfer loads (see Table 1). SUPERTAB's modeling of conductivity, specific heat, loads and boundary conditions that vary with time or temperature is supported by entering a negative integer value for that property. The FEM/SINDA translator (which translates a universal file into a FEM/SINDA input deck) interprets all negative integer values for conductivity, specific heat, loads and boundary conditions as a SINDA array reference (the SINDA array # is the absolute value of the integer). The SINDA input deck must then include an array which describes how that value is to vary with time or temperature.

```

TITLE=      $ Give some title
OUTPUT=    $ SUPERTAB, PATRAN, NASTRAN, and/or PLOT
ITEMP=     $ Initial temperature
RTEMP=     $ Temperature of radiation infinity node
SFILF=     $ Name of Sinda_formatted input file
MINCOND=   $ Minimum abs(A/L) value to be written
BEGIN BULK
$
$ Data enclosed in [ ] may be either a real constant
$ or reference an array
$
$ Coordinate systems:
CORDR, CID, A1, A2, A2,, B1, B2, B3, +Ccid
+Ccid, C1, C2, C3
CORDC, CID, A1, A2, A2,, B1, B2, B3, +Ccid
+Ccid, C1, C2, C3
CORDS, CID, A1, A2, A2,, B1, B2, B3, +Ccid
+Ccid, C1, C2, C3
$
$ Nodes:
NODE  ID    CID    X      Y      Z
FNODE ID
$
$ Material properties:
MATI  MID  {K}    rho    {Cp}
MATO  MID  {Kx}   {Ky}   {Kz}   rho    {Cp}
$
$ Physical properties:
PCOND  PID  {KA/L}
PROD   PID  MID    AREA
PSHELL PID  MID    THICKNESS
PSOLID PID  MID
$
$ Elements:
CAPAC  N1    MASS  {Cp}
CONDUCT N1   N2    {KA/L}
FCOND  FN1   FN2   {MDOT} {Cp}
$
ROD    EID   PID   N1    N2
TRIA   EID   PID   N1    N2    N3
QUAD   EID   PID   N1    N2    N3    N4
TETRA  EID   PID   N1    N2    N3    N4
WEDGE, EID, PID, N1, N2, N3, N4, N5, N6
BRICK, EID, PID, N1, N2, N3, N4, N5, N6, +Beid
+Beid, N7, N8
$
$ Boundary Conditions:
NHEAT  NODE  {HEAT}
EFLUX  EID    EDGE#  {FLUX1} FLUX2
FFLUX  EID    FACE#  {FLUX1}
VFLUX  EID    {FLUX1}
ECNVECT EID    EDGE#  {h1}    h2    FNODE  TYPE
FCNVECT EID    FACE#  {h}    FNODE  TYPE
ERADS  SID   EID    EDGE#
FRADS  SID   EID    FACE#
ENCLOSE SID1 e1    SID2  e2    F12    TB
FRADT  SID   EID    FACE#  e      I
TEMP   NODE  {TEMP}

```

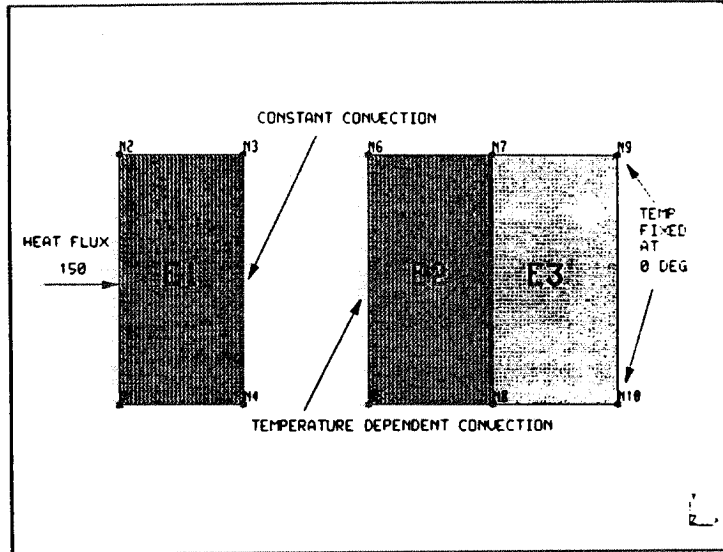


Figure 4. Simple Model Generated in SUPERTAB

```

TITLE = PLATE MODEL SAMPLE CASE
ITEMP = 50.0                $ Initial temperature
SFILF = PLATE_SAMPLE.MIT   $ Sinda-formatted input file
OUTPUT = SUPERTAB          $ SUPERTAB Univ. file output
MINCOND= 1.0e-15          $ Eliminate all conductors
$                            with ABS(A/L) < 1.0e-15
BEGIN BULK
MATI*    1    10.    200.    .2
MATI*    2    20.    100.    .4
PSHELL   1    1    0.5
PSHELL   2    2    0.5
NODE     1    0    0.0    0.0    0.0
NODE     2    0    0.0    4.0    0.0
NODE     3    0    2.0    4.0    0.0
NODE     4    0    2.0    0.0    0.0
NODE     5    0    4.0    0.0    0.0
NODE     6    0    4.0    4.0    0.0
NODE     7    0    6.0    4.0    0.0
NODE     8    0    6.0    0.0    0.0
NODE     9    0    8.0    4.0    0.0
NODE    10    0    8.0    0.0    0.0
QUAD     1    1    1    2    3    4
QUAD     2    1    5    6    7    8
QUAD     3    2    8    7    9    10
TEMP     9    0.0
TEMP    10    0.0
EFLUX    1    1    150.    150.
ECNVECT  1    3    10.    10.    1000    0
ECNVECT  2    1    A1    1000    1

```

Figure 5. Input Deck Generated from Model Shown in Figure 4

Figure 3. General Description of FEM/SINDA Input Deck

The PATRAN interface to FEM/SINDA supports almost all FEM/SINDA entities which, like SUPERTAB, allows the user to input the entire model in the preprocessor. Nodes and elements are generated with the standard GFEG and CFEG commands. Element properties and material properties are entered with PROP and PMAT commands, respectively. Two PMAT options are supported: thermal isotropic (TIS) and thermal anisotropic (TAN). Material properties which vary with temperature may reference a SINDA array by entering a negative array number in the PMAT command for that property. Boundary conditions are entered with the standard DFEG command and may reference a SINDA time-varying array by entering the array number in the UID field of the DFEG command. The only exception is convection in which the array reference goes in the data field and the convection option (time or temperature dependent) goes in the UID field.

SUPERTAB or PATRAN modeling used in conjunction with FEM/SINDA allows the thermal analyst to easily model his/her problem with the tools that are available in the pre-processors. The mapped and free mesh generation, application of loads and boundary conditions to geometric entities, mixing of rod, shell and solid elements are just a few of the FEM's features that can be used (without playing games) to generate a thermal model. Fundamental tasks such as free edge plots can be used meaningfully to show the absence of thermal connections (this is in direct contrast to centroidal methods and any method which does not use the nodes of the finite element model as the point at which the temperature is to be determined). FEM/SINDA's interface with SUPERTAB and PATRAN truly allows the modeler to use the pre-processor software consistent with its design, and hence makes the thermal analyst more efficient in his/her modeling.

FEM/SINDA INPUT DECK

The FEM/SINDA translator will read a SUPERTAB universal file or a PATRAN neutral file of a thermal model and generate a FEM/SINDA input deck. The input deck to FEM/SINDA looks similar to a MSC/NASTRAN input deck (hence present MSC/NASTRAN decks can be used, with slight modification, to perform a thermal analysis). Figure 3 shows a quick reference guide describing a FEM/SINDA input deck, and Figure 5 shows an input deck for the simple problem shown in Figure 4. This deck was completely generated from the SUPERTAB model shown by first generating a universal file from SUPERTAB and then running the FEM/SINDA translator (similar techniques apply for PATRAN). The card image input is self explanatory (Figure 3 can be used as a quick guide for the field description). The "SFILE" shown in Figure 5 is the name of a supplementary SINDA file. The SFILE can contain SINDA array definitions, FORTRAN subroutines, etc. that will augment the conductor network generated from the finite element model to produce the SINDA input deck. This file could contain old SINDA decks that will be thermally combined with the new finite element input deck. The ECNVECT card shown in Figure 5 defines the heat transfer coefficient to the air gap (see Figure 4) as a function of temperature to be defined by array "A1". This array is specified in the SFILE.

The quick reference guide (Figure 3) indicates which fields of the data input can vary (data enclosed in {}) with time or temperature and hence reference an array. For example, the edge flux card (EFLUX) allows for the flux to be specified by an array.

Radiation is specified (for which TRASYS will calculate the view factors and generate the nodal radiation conductors) by the FRADT card. The radiation conductor network returned from TRASYS is included with the SINDA input deck to form a complete system network which models the integration of the conduction, convection and radiation thermal model.

Once the SINDA analysis is complete, a universal file or neutral file is generated by SINDA that contains all of the nodal temperatures for post-processing. In a transient thermal analysis, this

file will contain a temperature data set for each output time step. SINDA can also generate a set of MSC/NASTRAN "TEMP" cards that can be included with a MSC/NASTRAN input deck for thermal stress analysis.

FEM/SINDA EXAMPLES

The following examples of FEM/SINDA will help to illustrate the advantages of the SUPERTAB/PATRAN-to-FEM/SINDA combination to the thermal analyst.

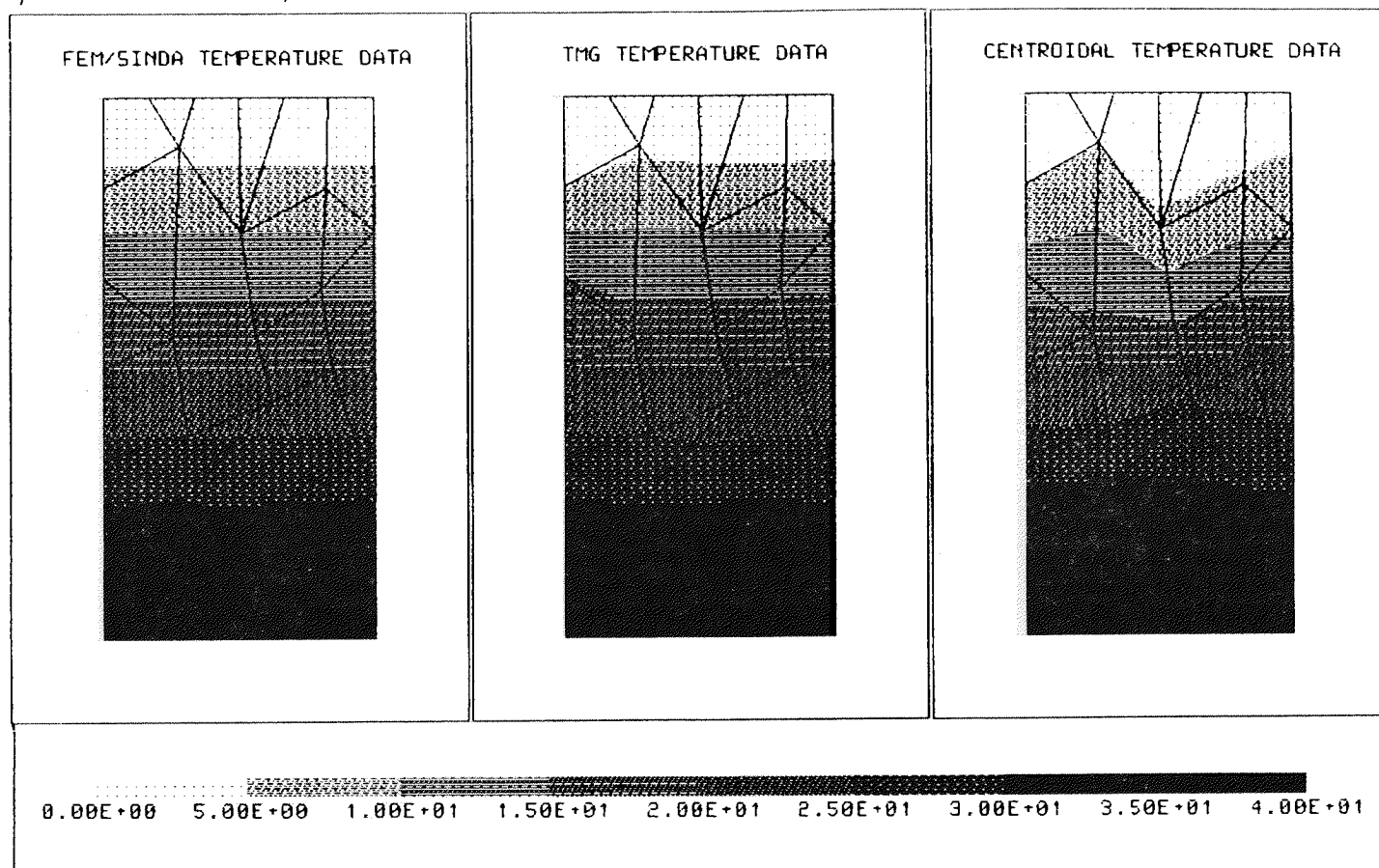


Figure 6. Linear Temperature Field Example

Figure 6 shows the temperature contours for a 2-D model of a rectangular region. Heat is input at the bottom of the region and the top is held at a constant temperature of zero. The thermal conductivity is constant. The grid shown (using 2-D shell elements) was purposely made irregular to illustrate the strength of the finite element method. The temperature field is linear for this model. FEM/SINDA will model a linear temperature field exactly because of the finite element description of the conductor network (see Equation 1). Three contour plots are shown: (a) FEM/SINDA results, (b) TMG results, and (c) a centroidal method. The FEM/SINDA results give the exact solution, and the TMG results are reasonably close to the exact solution. TMG uses a single "thermal node" per element, but the "node" is not at centroid but at the intersection of the perpendicular bisectors of the sides (assuming a triangular element - a quadrilateral can be broken up into triangles). It can be shown that these "thermal node" points will model a linear temperature field exactly. The apparent discrepancy (from Figure 6) is that TMG will not use these points when the bisector intersection falls outside the triangle. The resulting TMG conductor network is therefore not guaranteed to model the temperature field exactly (a trivial change to the code could remedy this). Despite this, the TMG temperature field is acceptable. This is not the case for the the centroidal temperature field show in Figure 6c. The conductor network for this model is based on an in-house code that uses the centroid

as the "temperature node". The results are unacceptable and clearly show that the irregular finite element grid dramatically affects the results (a rectangular grid would give the exact solution).

If the analyst were to use a centroidal method (rather than FEM/SINDA), the modeling for a large model could be complicated and very cumbersome. For example, besides the needed shell elements shown in Figure 6 to model the 2-D conduction region, bar elements must be used at the top and bottom boundary to designate the boundary conditions. This process carried over to 3-D models requires shell elements to be put on the face of solid elements to handle boundary conditions—a process that can add significant modeling time and that is cumbersome. These "additional" elements are sometimes needed even within a solid region; for example, at the interface of two materials with different conductivity. Failure to do so will cause interpolation algorithms to inadequately predict finite element nodal temperatures from the "element" temperatures. Modeling convection and radiation can also require the addition of elements on the appropriate boundaries. Mixing of 1-D, 2-D, and 3-D conduction elements also requires the "additional" elements when such elements join (a 2-D shell coming into two nodes of a solid requires the addition of a shell on the face of that solid to force the thermal connectivity). Overall, these thermal "games" can significantly affect the thermal analyst's productivity in SUPERTAB or PATRAN and can hinder the graphical verification of his model.

Figure 7 shows a radiation-conduction problem that was performed both with FEM/SINDA and MSC/NASTRAN. The top body is held at a constant temperature of 100 degrees and the bottom

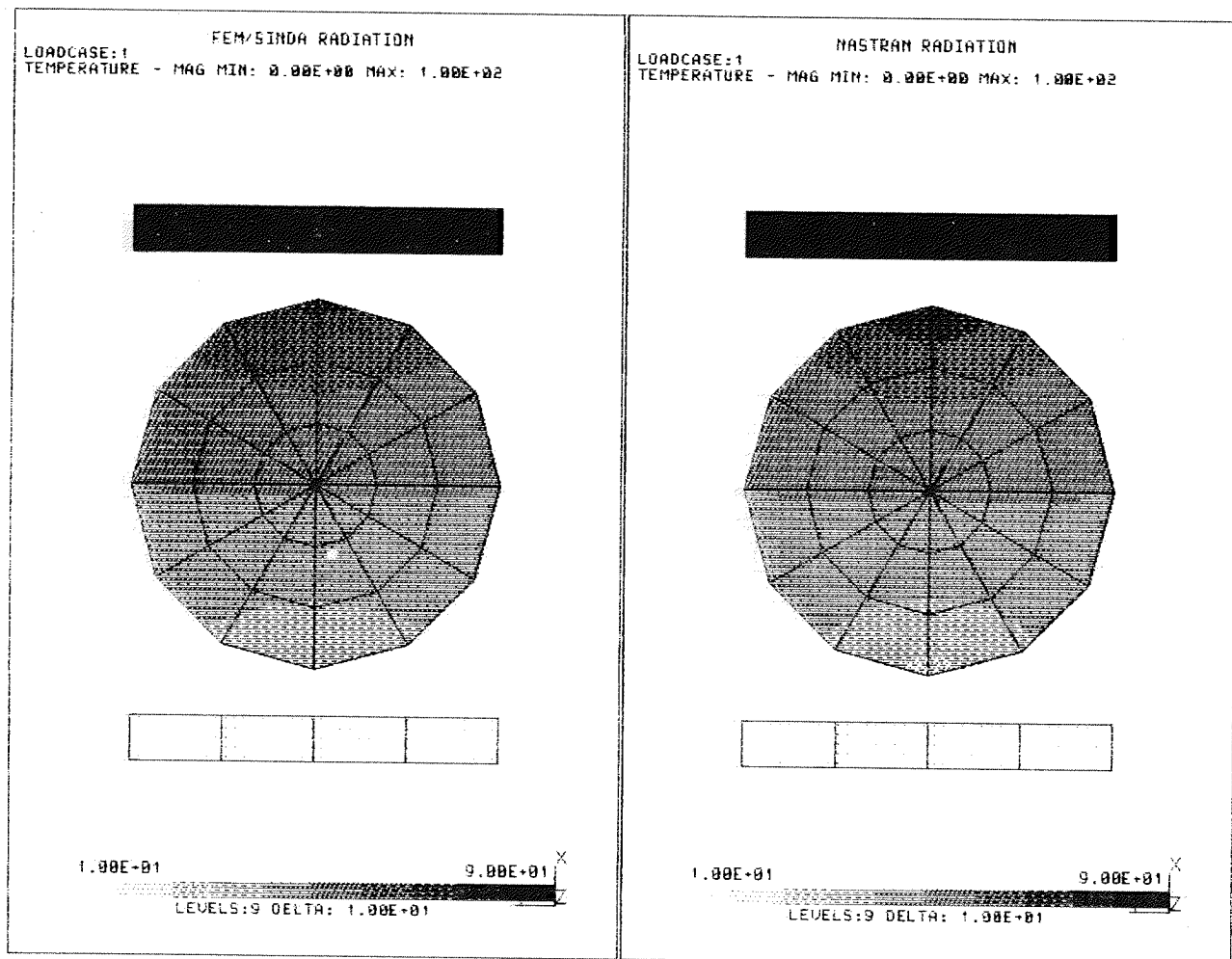


Figure 7. Radiation-Conduction Example

body at 0 degrees. The circular region has a low thermal conductivity and a unit depth is used. Space is at a temperature of 50 degrees. This model was generated in SUPERTAB including the designation of the radiating surfaces. FEM/SINDA generated the TRASYS run which produced the view factors and the SINDA radiation conductors. Good agreement is shown between FEM/SINDA and MSC/NASTRAN for the relatively coarse grid used.

Figure 8 shows an example of a transient analysis. It consists of a splice ring used to attach sections of a missile together. Normally, the thermal protection requirements of a missile are determined by a 1-D analysis through a typical portion of the missile skin. In this case two dimensional effects are considered important where the splice section and the bolt area join. For this example, a 2-D mapped mesh model was constructed. Different thermal properties were used for the splice ring, bolt and filler elements. Aerodynamic heating was applied to the outer surface (top) by means of a time-varying adiabatic wall temperature and convection coefficient. The outer surface was also allowed to radiate to the sky. The inner surface (bottom) had constant free convection applied. The results of the five second transient analysis are shown in two forms - four temperature contour plots at various points in time and as a surface temperature versus time plot. The surface temperature time trace compared favorably with the results from a 1-D in-house finite difference code, called F86, which is also shown in the plot.

A practical example showing the use of FEM/SINDA is the model of a TV camera of an electro-optical system that is shown in Figure 9. This model is composed of 2849 nodes and 2834 elements which generated 19289 SINDA conductors (the largest model to date with FEM/SINDA was 4897 nodes and 5423 elements). The model shown is a mixture of rods, shells, and solid elements. Convection loads the exposed surfaces. Heat is input in the mounting bracket (shown in the foreground) because of a direct connection between the bracket and an electronics module. The results shown here represent the steady state temperature distribution. The detail shown in the finite element model was needed for structural analysis. The deflections of the optical train were driven by the temperature distribution. The determination of the temperature distribution at the finite element

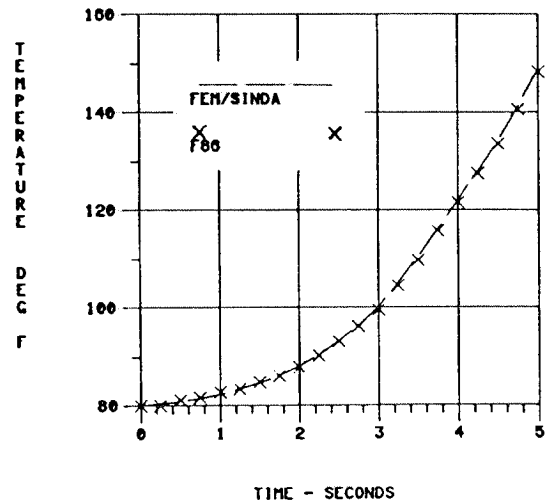
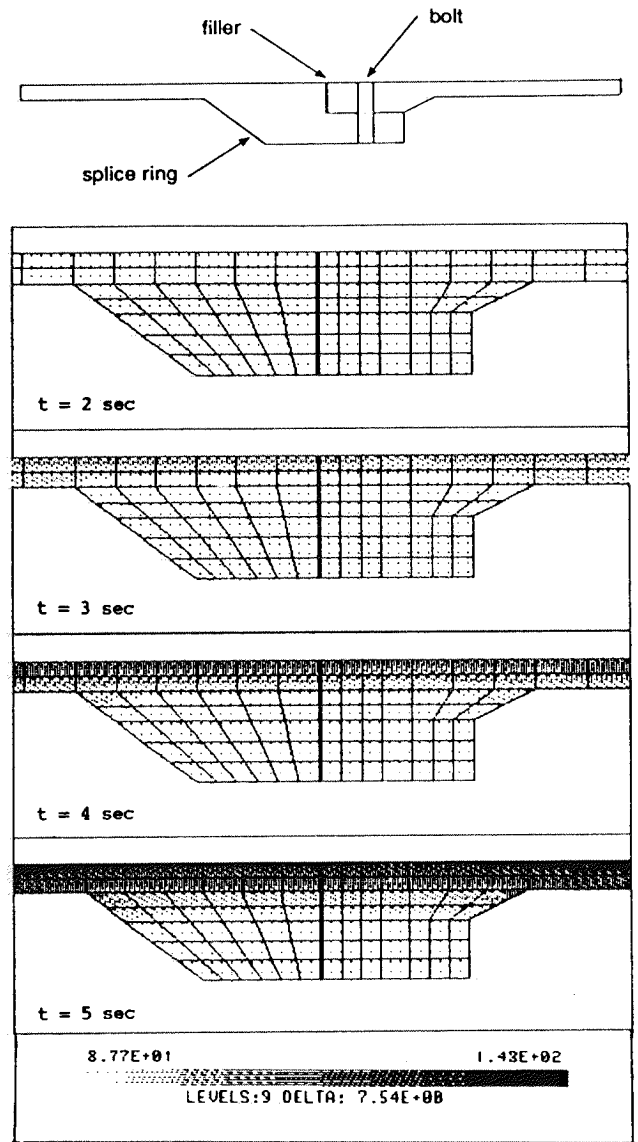


Figure 8. Thermal Transient Analysis of a Missile Splice Ring

nodes (performed by FEM/SINDA) made the interface between the structural and thermal model a trivial matter. The other important feature that is automatic in this model was the mixing of various element types. For example, a rod coming into one node of a shell is thermally allowed and easily modeled in PATRAN or SUPERTAB. This connectivity is also easily verified in PATRAN or SUPERTAB.

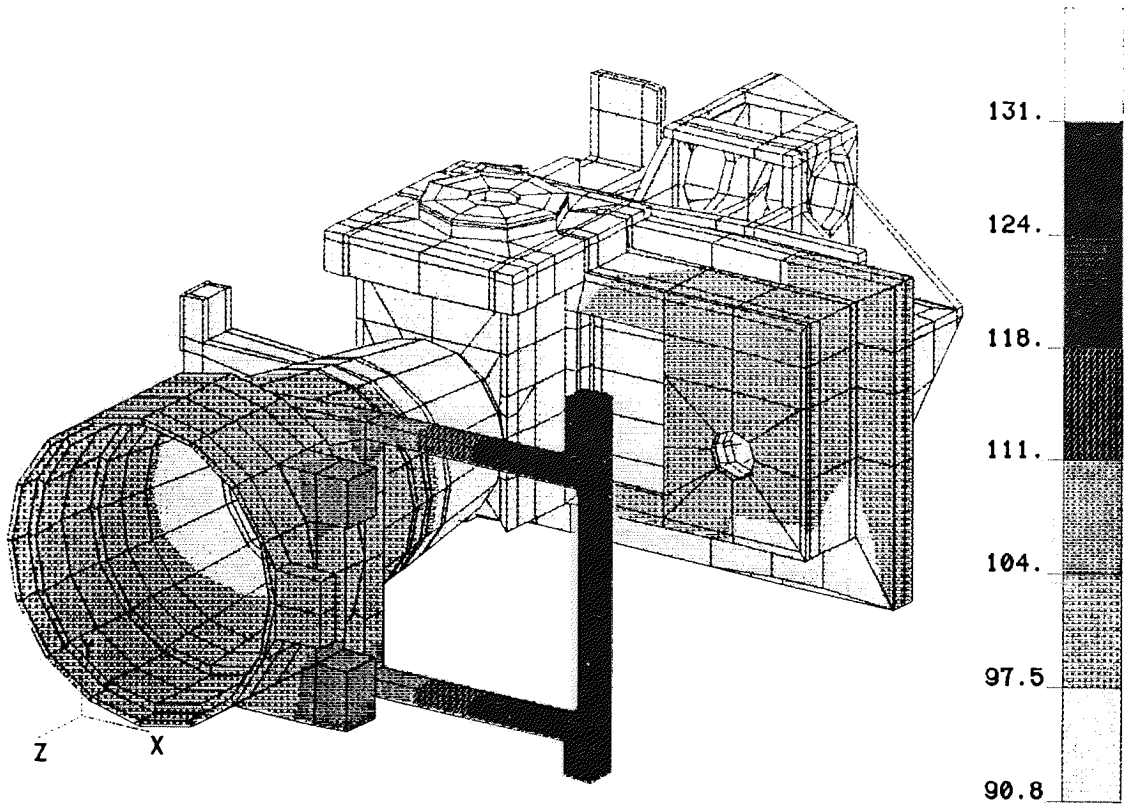


Figure 9. TV Camera Model

CONCLUSIONS

FEM/SINDA provides a general purpose geometry driven thermal analysis code to the thermal analyst. Because of the finite-element-type input to the code (essentially identical to a MSC/NASTRAN input deck), its interface with SUPERTAB, PATRAN and MSC/NASTRAN is complete: each FEM/SINDA entity corresponds naturally to a SUPERTAB or PATRAN entity, and in most cases to a MSC/NASTRAN entity (hence MSC/NASTRAN input decks can, with little or no modification, be used as an input deck to FEM/SINDA). The nodal temperatures determined from FEM/SINDA can be used directly to drive a thermal loading condition in MSC/NASTRAN. FEM/SINDA combines the power of finite element techniques with the thermal community's tested and well accepted workhorse: SINDA. This mix of the finite element-finite difference worlds takes advantage of the strengths of both methods: the finite element method's ability to handle arbitrary geometry, model non-homogeneous regions with different element types, and model linear temperature fields exactly; and SINDA's finite difference capability to handle time and temperature dependent material properties, loads, and boundary conditions, and add user written FORTRAN routines. FEM/SINDA's interface with SUPERTAB and PATRAN allows the thermal analyst to take full advantage of all of a finite element modeler's capabilities in a manner consistent with the design of the FEM pre- and post-processors. The key to that interface/integration is that a node of the finite element model will necessarily be a node in the thermal conductor network. Therefore this technique does not compromise the inherent modeling integrity of FEM geometric discretization, and will easily allow the algorithms of both old and new finite element technology (for example, both in meshing applications and finite element matrix manipulations) to be applied to general purpose thermal analysis.

REFERENCES

- (1) The Finite Element Method, O. C. Zienkiewicz, McGraw-Hill Book Company, 1977
- (2) Concepts and Applications of Finite Element Analysis, Robert D. Cook, John Wiley & Sons, Inc., 1981