THERMAL CAPABILITIES AND GRAPHICAL OUTPUT OF PAFEC

J. E. Akin
University of Tennessee
and
PAFEC Engineering Consultants, Inc.
Knoxville, Tennessee

SUMMARY

The computation of heat transfer in structures is enhanced by the utilization of passive and interactive graphics. These capabilities of the PAFEC system are presented and future developments are outlined. This finite element system is shown to have significant thermal capabilities in support of its general structures.

INTRODUCTION

The Program for Automatic Finite Element Calculations - PAFEC - is a powerful, general purpose code designed for easy use in thermal and structural analysis. Data preparation for PAFEC is perhaps simplest and fastest of all such finite element codes. PAFEC employs free format input with engineering key words, powerful mesh general facilities, and extensive plotting options. PAFEC has restart capabilities and control options for maximum versatility and economy. The PAFEC modules are particularly easy to learn and remember, making the use of PAFEC attractive for the beginner or occasional use.

PAFEC Interactive Graphics Suite - PIGS - interactively views and modifies the PAFEC data base. PIGS employs a graphics terminal with cursor control and optionally, a digitizing tablet. PIGS can be used for data generation and to study the pre- and post-solution results.

PAFEC (Level 4) offers the following types of analysis:

Interactive graphics, free format input
User defined program control steps
Steady state, transient heat transfer
Boundary element methods
Linear static, stress and displacements
Modes and frequencies calculations
Direct dynamic time integration
Frequency response analysis
Elastohydrodynamic lubrication
Large deflection analysis, buckling
Creep and plasticity analysis
Substructures, cyclic symmetry

Comparisons of the general PAFEC (Level 3) capabilities with those of other major finite element codes, like NASTRAN and ANSYS, can be found in reference [1]. A more specific tabular summary of its heat transfer features is included in reference [2]. In this paper the emphasis will be on the thermal analysis capabilities of PAFEC, its built-in pre- and post-processing features, and user control options. Future enhancements will also be outlined.

PAFEC is released in Levels that are always upward compatible. The current, Level 3, version was released in 1979. Some of the new features to be presented here are in the Level 4 release that is scheduled for January 1982. This FORTRAN software is installed on most computer hardware ranging from small computers, e.g. Prime 250, through the super mainframes, e.g. CRAY 1.

ANALYSIS WITH PAFEC

The theoretical basis for a finite element thermal formulation has been well established and is given in typical texts, such as reference [3]. Commonly utilized computational procedures for applied thermal analysis are presented in reference [4], and elsewhere. Most of the large finite element codes offer similar capabilities. The differences that are most noticeable center around the ease of use and the model building aids.

PAFEC uses English keywords (which can be shortened to four characters) and completely free format data. Data input in PAFEC is done in modular form, where only those modules actually needed for a job are used. The modules can be in any order and repeated as desired. Thus, the typical names of the modules in Table 1 imply information about the types of data available in the PAFEC thermal analysis option.

The advantages of the modular data system are many. From the standpoint of the user, the data are clear and intelligible without decoding them from the documentation. Copying data from files produced by other finite element programs or using input data prepared for other finite element programs can be done with far less editing. The numerical values are in free format with commas or spaces as separators between data items. The computer program thrives on the modular data, too. A scan is made of the module headers. This information indicates which phases to run and which subroutines and libraries are to be loaded in each phase.

The module names are known to the program by only the first four characters. Hence, anything that follows the first four characters can be thought of as merely comment material. The modular data construction also allows for constant properties to be inserted just after the header card with the form 'property' = numerical values. In the program input, the modules may be in any order. However it is recommended that modules serving a like function, e.g. mesh generation, be grouped together to aid in the user's grasp of data structure.

The keywords used to identify typical data modules employed in a thermal analysis indirectly indicate the capabilities of the PAFEC system. To be more specific the thermal analysis options include both transient and steady state solutions. Temperature dependent properties can be utilized for nonlinear thermal problems. The available boundary conditions include time dependent nodal temperatures or heat flux. Before considering the specific thermal options some of the general model building aids will be discussed.

MODEL BUILDING AIDS

PAFEC offers extensive mesh generation and data supplementation options. Eight default coordinates systems are available as user-defined axes. The generation of nodes on lines and arcs is included. The isoparametric generation of meshes for 2-D elements, surfaces and 3-D solids is included in the use of PAFBLOCKS. There are transitions for mesh refinement, and user defined spacing ratios and holes. PAFBLOCKS permit the independent creation of a continuous geometric model of curved blocks. Additional data control the separate subdivision of the blocks into finite elements. Thus once the geometric model is established the subdivision data are easily changed. The use of PAFBLOCKS is available in both interactive and batch modes. Other features allow for the generation of repetitive mesh segments by translation, rotation, or scaling. Powerful boundary condition generators are also included in PAFEC. PAFEC offers extensive warning and error messages in the data validation and geometry checks. Several mesh plotting options are included in PAFEC. Most of these plots allow user selected windows for more detail.

For nodal coordinates generation it is most direct to use the NODES module, but other modules are also used in describing the nodal coordinates. These are:

NODES	gives coordinates of nodes in any axis set.
AXES	describes the axis sets used for giving coordinates.
SIMILAR.NODES	Once a group of nodes has been described, this module may be used to locate other nodes which happen to be similar to them.
LINE.NODES ARC.NODES	are used to force any number of nodes to lie on the same line or circular arc, respectively.
PAFBLOCKS MESH	are used in conjunction with each other to cause blocks of both nodes and elements to be generated.

Several facilities are used in description of element topology, types, material properties, and thickness. Elements may optionally be referred to by GROUP.NUMBER to facilitate assignment of properties, making drawings, and other uses.

ELEMENTS	describes the properties, element type, group number and topology for individual elements.

GROUP.OF. SIMILAR. ELEMENTS	copies selected elements into new element groups. This facility may be used to describe repeated geometry.
REFERENCE.IN. PAFBLOCK	permits treating exceptions to elements and nodes generated with PAFBLOCKS.
PLATES.AND. SHELLS	sets element thickness, material number, and other information.

There are ten types of PAFBLOCKS available for model building. The most common, Type 1, block is shown in figure 1. It can be utilized to generate surfaces, solids, or sets of diaphragms. In figure 1, the quantities N1, N2, N3, N4, N5 refer to spacing ratios which are given in the MESH module; nodal point numbers specify the order in which the topology is listed in the PAFBLOCKS module. Dotted lines show resulting meshes with triangular or wedge elements. Numerous element types can be generated using PAFBLOCKS.

The PAFEC element library contains over 80 elements. Most involve linear to cubic isoparametric interpolation but hybrid and semi-loof elements are also employed. For thermal analysis options both the linear and quadratic surface and solid elements are available.

PIGS can be used interactively to generate and modify PAFEC data. When generating data, curved and straight boundaries may be defined and, if required, the individual points and node numbers may be specified. However, it is more common to generate the minimum number of nodes required to define a two or three dimensional structure and mesh the structure using an interactive PAFBLOCK. Where appropriate, any part of the mesh may be replicated, scaled, rotated and translated to form new parts of the structure with a minimum of effort. Information can be input from cursor, keyboard or digitizer and the interactive plot can be manipulated in many ways to ensure that the user sees as clearly as possible those sections of the structure which are of most interest.

It is also possible to have PAFEC automatically create an interface file for the MOVIE.BYU [5] program. This allows interactive hidden line plots of perspective or isometric views to be generated. A post-processing MOVIE interface is also available. Since the PAFEC model building capabilities are so powerful and popular at least one PAFEC site has developed an interface to create NASTRAN data. Often one page of PAFEC data would create ten to fifteen pages of NASTRAN input.

THERMAL MATERIAL DATA

In many types of analysis, the materials are isotropic with constant properties. For user convenience, ten standard material types are built into PAFEC, and include a typical steel, stainless steel, cast iron, aluminum alloys, titanium, glass, and concrete. The standard data may be overridden, or other materials may be added with the MATERIALS module.

Occasionally, nonuniform or anisotropic material descriptions are needed. Some PAFEC modules for this include:

ORTHOTROPIC. MATERIAL	permits specification of the nine components of the compliance matrix, and orthotropic thermal properties.
LAMINATES	Layered ply material may be described here.
VARIABLE.MATERIAL TABLES	describes the temperature dependence of user selected properties. Linear interpolation is used between tabulated values.

The experienced user can also supply subroutines to replace or expand these options. Thus specific nonlinear material responses or alternate constitutive relations can be introduced.

Convection coefficients are described in the standard MATERIALS modules. However, the forced mass flow convection conditions are not standard input. This type of property would also have to be defined by a user supplied subroutine.

THERMAL ANALYSIS OPTIONS

PAFEC may be used for finding temperature distributions in order that thermal distortions and stresses can be found, or for other applications requiring the solution of heat transfer.

There are two main types of thermal calculations: steady state and transient. In transient problems there is usually a thermal shock and it is required to know how the temperature varies with time. At any point in time it may be supposed that the temperature distribution is known completely; a finite element solution is needed to determine how the temperature will vary during a short interval of time. The transient temperature solution involves marching forward in time. For the process to begin, temperatures are required at a start time which is conveniently taken at time, $t = 0$. Initially, all temperatures may be known and input as data for the problem, or, alternatively PAFEC may have to carry out a steady state calculation as a prelude to the transient analysis.

During the steady state solution at each node either the temperature or the heat flux entering the structure is an unknown. For most nodes in the structure there will probably be an unknown temperature and the heat flux entering the structure from external sources will be zero. The following modules are used to describe the thermal boundary conditions:

TEMPERATURE	gives the nodal temperatures. Nodes not mentioned are assumed to be unknown.
FLUX	gives the heat inputs at nodes. For any node at which neither the flux nor the temperature is specified it is assumed that the flux input is zero.

FACTOR.LOADS	can be used to sum various thermal load cases together. This can also be used to simulate unsymmetrical cases by combining results of symmetrical and antisymmetrical cases.

For a transient thermal analysis it is assumed that the initial temperature field is completely specified. If this is not the case then a steady state solution should usually be run first.

The following modules are used in a transient calculation:

TEMPERATURE	gives the initial temperature distribution if a steady state solution was not run. Any node not mentioned is assumed to be at zero temperature.
THERMAL.SHOCK	describes the variations with time of any nodal temperatures which are prescribed.
NODAL.FLUX.SHOCK	gives the variation with time of prescribed heat fluxes at nodes.
UNSTEADY.THERMAL. TIMES	is used to define the time step selected and the time at which the solution is to end. It also controls the times for printed and graphical output.

Since the thermal analysis is not the default operation the program must be told to execute the desired option. This is accomplished by placing the commands CALC.STEADY.TEMPS and/or CALC.TRANS.TEMPS in the control module. If both are present then the steady state solution is automatically used as the starting condition for the transient analysis. In that case the TEMPERATURE module only describes boundary conditions on the steady state solution. If the calculated temperatures are to also be utilized in a structural analysis then the control command SAVE.TEMPS will cause the required files to automatically be created and named. As will be discussed later, the results of a thermal analysis can be displayed graphically in both passive and interactive formats.

The treatment of a radiation boundary condition is not currently easily included in a PAFEC analysis. This nonlinear boundary condition can be introduced by special user supplied subroutines. The program does not calculate any radiation view factors. However, the interactive program described at this conference, reference [6], could aid an analyst in obtaining the necessary view factors.

ADDITIONAL BOUNDARY CONDITIONS

To apply constraints at certain nodes, the following modules are used:

RESTRAINTS	can be used for temperatures that are known. It can describe the constraints at one node, or all the nodes on a line or a whole plane can be constrained at once.
REPEATED.FREEDOMS	is used to constrain two or more nodes to have unknown temperatures that are the same.
GENERALIZED. CONSTRAINTS	permits writing arbitrary linear functions relating temperatures among any number of nodes.

PASSIVE GRAPHICS

Extensive passive graphics options are available. It is also possible to interactively preview these plots before transmitting them to a plotter. Numerous options are available to display the results of the model building. These include standard mesh plots, exploded mesh views, boundary plots, etc. The orientation of the plots can be specified but the program will select a default view for each plot.

Various items of information can be displayed on the plots. These include node, element, and material numbers; active DOF and restraints; wave front position; and all axis sets. For a thermal analysis the output plot options include steady state and transient temperature contours. Specific element groups can be selected for display. The ability to plot temperatures along a user defined nodal path is another useful feature. Modules for the selection of PAFEC graphical output are listed here:

IN.DRAW	is used for drawings of the structure and the constraints. This controls the information contained on the drawing, and the groups of elements to be drawn.
OUT.DRAW	produces drawings of the temperature after solution.
SELECT.DRAW	supplements IN.DRAW and OUT.DRAW by allowing a spatial selection or selection by element type for drawing some portion of the structure.
GRAPH	gives plots of temperature as ordinates with selected nodes as abscissas.

INTERACTIVE GRAPHICS

In the post-processing mode, PIGS is used for displaying deformed shapes, temperatures, stresses, and mode shapes. Any number of load cases or modes may be stored and retrieved. The temperature at individual nodes can be displayed simply by hitting that node with the cursor. Menu options are chosen in the same way.

Three different contouring options are available and individual faces of structure may be drawn selectively to classify the output. Any PIGS drawing may be reproduced on a suitably interfaced plotting device, and, conversely, plot files created in PAFEC may be viewed on an interactive graphics terminal.

All PIGS facilities are selected by the user from a menu of available options. Only one menu is displayed at any time, occupying a column on the left-hand side of the screen. Options are displayed in alphabetical order. Whenever the selected option is hit it will be underlined. There are four different menus, the ROOT menu, the ANALYSIS branch menu, the VIEW branch menu, and the DIGITIZE menu. The VIEW menu not only offers finite element mesh viewing but also data modification and generation facilities which constitute the most useful options in PIGS. The DIGITIZE menu allows the direct input of nodal coordinates and element or PAFBLOCK topology. It utilizes special hardware in the form of a digitizing tablet. The ANALYSIS menu, for post-solution interactive graphics, permits the displaced shapes, temperatures, stresses, and mode shapes to be displayed. Most of the many options in these menus employ cursor input. However, the analyst can request typed input and prompting assistance.

Figure 2 shows a selected segment of a three-dimensional mesh that has been rotated and displayed by PIGS. Upon request the temperature contours on these elements can be displayed as illustrated in figure 3. If the analyst desires more specific information then individual nodes can be selected with the screen cursor. When this is done the node number and computed temperature is added to the display. Figure 4 shows that display format. These and other features in PIGS makes it a very user friendly system.

Another interactive option is available for very inexperienced users. That is the Automatic data Preparation and Edit Systems - APES. It aids in digitizing the model, prompts the user for material and boundary condition data, etc. While this is useful for a beginner an experienced finite element user would quickly outgrow the need for such an option.

DISCUSSION

A new feature of PAFEC is the option to utilize Boundary Element Formulations in conjunction with the standard finite element thermal solutions. This is well suited to semi-infinite regions and other specialized treatments of the Poisson and bi-harmonic equations. Thermal results using this capability will be reported in the near future.

The PAFEC and PIGS systems provide a powerful thermal and structural analysis capability. It is a well documented and easy to use system. However, it currently has a weakness in the thermal area, that is, the lack of a user friendly treatment of radiation problems. Such nonlinear applications have been solved with PAFEC. But this is usually done by way of user supplied subroutines. The system is designed to easily accept such routines via the CONTROL module. However an experienced user is usually needed for a radiation analysis. Hopefully this current shortcoming will be overcome in the near future. In conclusion, the PAFEC system provides another useful tool for the computation of heat transfers in structures.

REFERENCES

1. Akin, J. E., and Dewey, B. R.: PAFEC Compared with Other Major Finite Element Codes. Structural Mechanics Software Series, Vol. IV. Perrone, N., and Pilkey, W. D. (Eds), Univ. Virginia Press, 1981.

2. Noor, Ahmed K.: Survey of Computer Programs for Heat Transfer Analysis. Computational Aspects of Heat Transfer in Structures. NASA CP-2216, 1982. (Paper 27 of this compilation.)

3. Huebner, K. H.: The Finite Element Method for Engineers. John Wiley & Sons, 1975.

4. Akin, J. E.: Application and Implementations of Finite Element Methods. Academic Press, London, 1982.

5. Christiansen, H.: MOVIE.BYU, A General Purpose Computer Graphics Display System. Brigham Young Univ., 1980.

6. Emery, A. F.; Mortazavi, H. R.; and Kippenhan, C. J.: Interactive Computation of Radiation View Factors. Computational Aspects of Heat Transfer in Structures. NASA CP-2216, 1982. (Paper 14 of this compilation.)

TABLE 1 TYPICAL DATA MODULES

A) Model Building

- AXES
- ARC.NODES
- ELEMENTS
- GROUP.OF.SIMILAR.ELEMS
- LINE.NODES
- MESH
- NODES
- PAFBLOCKS
- REFER.TO.PAFBLOCK
- SIMILAR.NODES

B) Thermal Material Properties

- LAMINATES
- MATERIALS
- ORTHOTROPIC.MATERIAL
- TABLE.OF.PROPERTIES
- VARIABLE.MATERIAL

C) Thermal Boundary Conditions

- FACTOR.LOADS
- FLUX
- GENERALIZED.CONSTRAINT
- NODAL.FLUX.SHOCK
- OMIT.FROM.FRONT
- REPEATED.FREEDOMS
- RESTRAINTS
- TEMPERATURE
- THERMAL.SHOCK
- UNSTEADY.SOLUTION

D) Passive Graphics

- GRAPHS
- IN.DRAW
- OUT.DRAW
- SELECT.DRAW

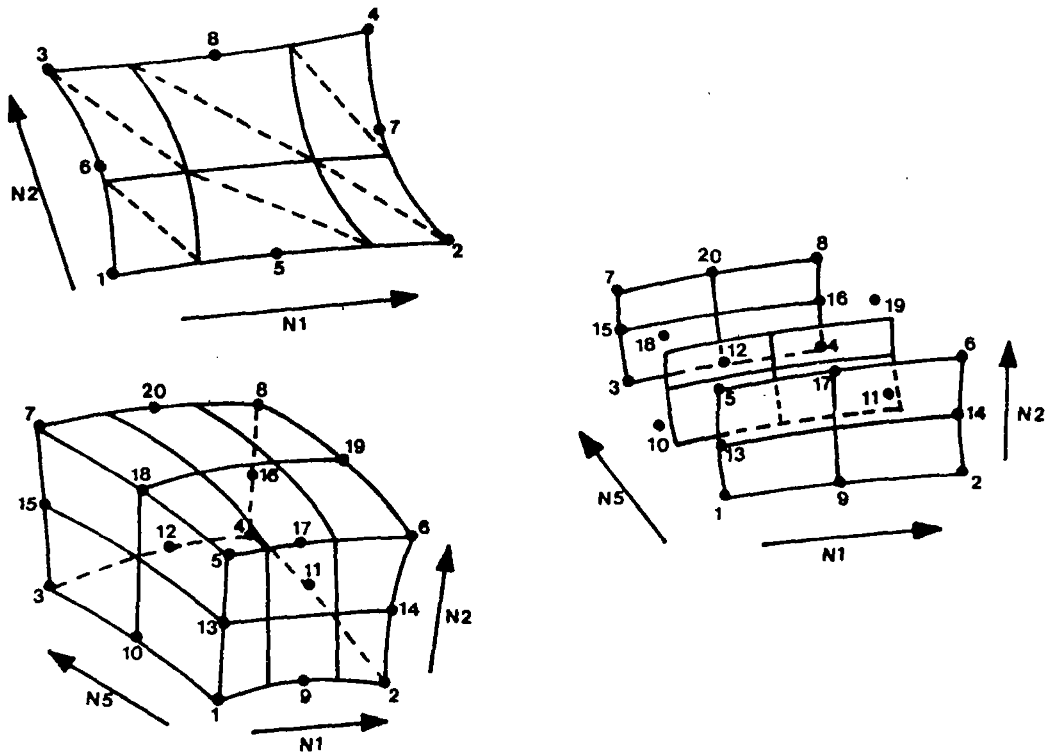

Figure 1.- A typical PAFBLOCK available for model building.

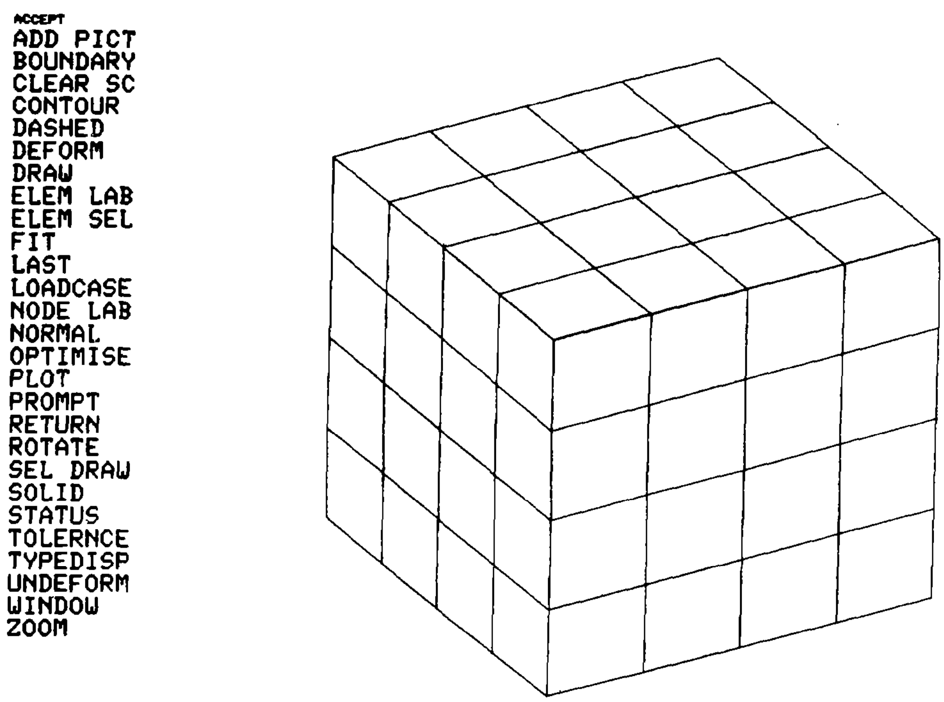

Figure 2.- Interactive display of 3-D mesh segment.

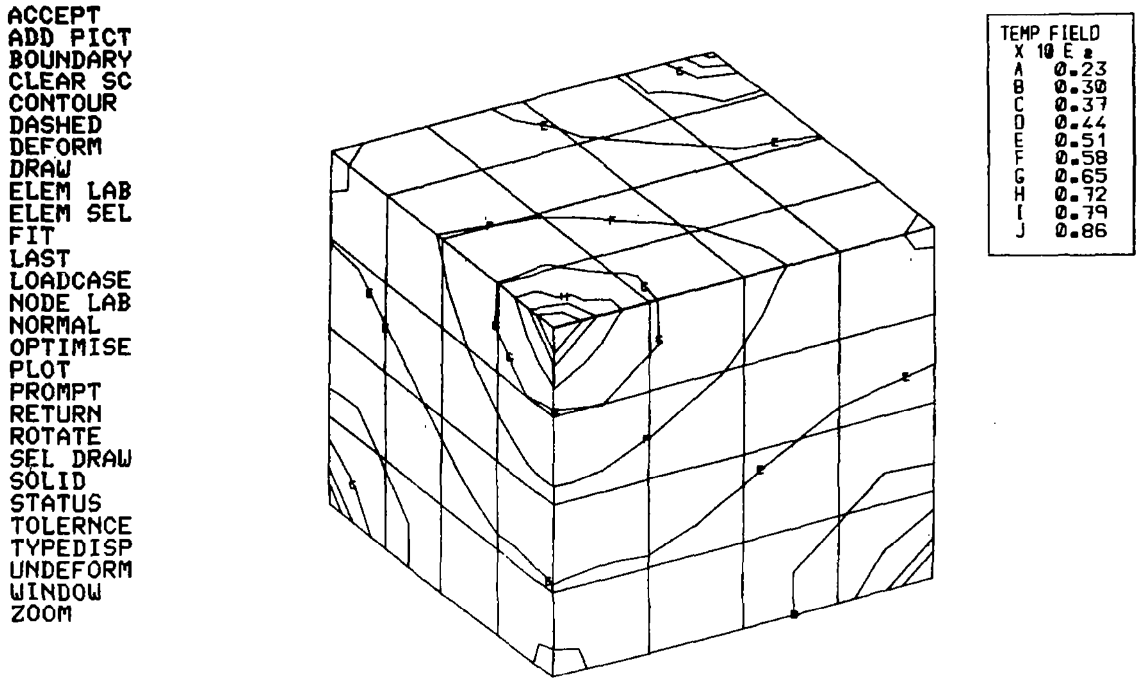

Figure 3.- Temperature contours added to mesh.

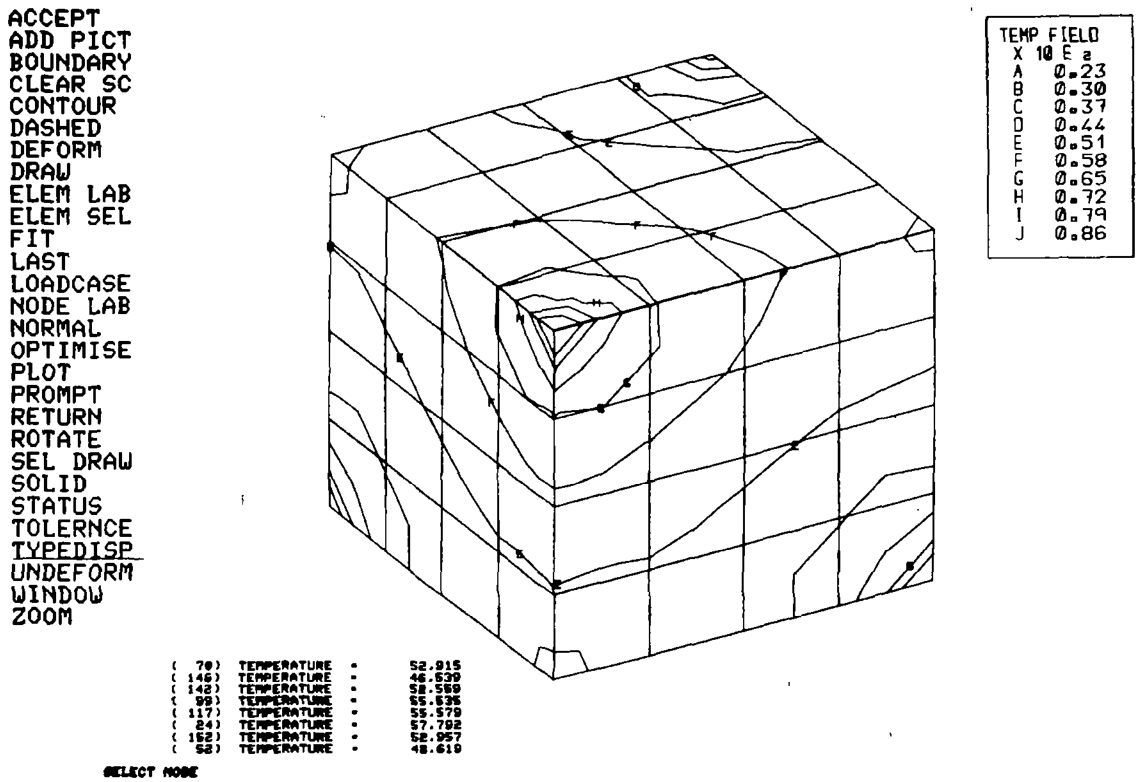

Figure 4.- Temperature values displayed during interactive node selection.