LEGIBILITY NOTICE

A major purpose of the Technical Information Center is to provide the broadest dissemination possible of information contained in DOE's Research and Development Reports to business, industry, the academic community, and federal, state and local governments.

Although a small portion of this report is not reproducible, it is being made available to expedite the availability of information on the research discussed herein.

LA-UR-89-2661

Los Alamos National Laboratory is operated by the University of California for the United States Department of Energy under contract W-7405-ENG-36

LA-UR--89-2661

DE89 016776

TITLE PATRAN AS AN ENGINEERING DESIGN VISUALIZATION TOOL: DESIGN AND ANALYSIS OF A HEAT EXCHANGER SUPPORT

AUTHOR(S) DANIEL J. WEINACHT, MEE-13

SUBMITTED TO

PATRAN USERS CONFERENCE
2975 REDHILL AVENUE
COSTA MESA, CA 92626

MASTER

Los Alamos National Laboratory
Los Alamos, New Mexico 87545

FORM NO. 836 R4
ST. NO. 2629 5/81

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

PATRAN AS AN ENGINEERING DESIGN VISUALIZATION TOOL:
DESIGN AND ANALYSIS OF A HEAT EXCHANGER SUPPORT

Daniel J. Weinacht
Los Alamos National Laboratory
Los Alamos, NM 87545

ABSTRACT

The effective communication of engineering results from the finite element modeling analyst to project managers and nontechnical program sponsors often presents a significant challenge to the analyst. Also, the ability to visualize trends in finite element model results is difficult when the analyst is confronted by reams of numerical output. Recent advances in computer graphics and the proliferation of sophisticated commercially-available software have greatly enhanced the ability of the analyst to assess the results of engineering calculations and present those results to others.

The use of PATRAN by an engineering mechanics group as a pre- and postprocessor for finite element modeling will be discussed. Specifically, the design and analysis of a heat exchanger support that experiences severe thermally-induced stresses will be examined. Both heat transfer and stress analyses for two different design concepts will be presented. The use of PATRAN as a visualization/presentation tool will be emphasized.

THE ENGINEERING MECHANICS section of the Advanced Engineering Technology group provides analytical and computational support to many research and development projects at Los Alamos. Described in this paper are the thermal and structural analyses of a component for an advanced research and development project. Specifically, two design concepts for supporting a heat exchanger were examined. Models for both concepts were built using PATRAN. ABAQUS was the analysis code. The feasibility of each design was assessed based on thermal behavior and stresses and deflections in the structures. PATRAN, with its vivid portrayals of the variations of these quantities through the structures, was instrumental in these assessments.

COMPUTER SYSTEMS

A VAX 11/785 was used as the host for finite element model construction using PATRAN. Tektronix 4208 and 4115 graphics devices were utilized in the pre- and postprocessing stages. The Advanced Engineering Technology group has been using PATRAN for finite element model construction and interpretation of results since May 1986. All analyses using ABAQUS were performed on a CRAY XMP-48.

PATRAN AS AN ENGINEERING DESIGN VISUALIZATION TOOL: DESIGN AND ANALYSIS OF A HEAT EXCHANGER SUPPORT

Daniel J. Weinacht
Los Alamos National Laboratory
Los Alamos, NM 87545

ABSTRACT

The effective communication of engineering results from the finite element modeling analyst to project managers and nontechnical program sponsors often presents a significant challenge to the analyst. Also, the ability to visualize trends in finite element model results is difficult when the analyst is confronted by reams of numerical output. Recent advances in computer graphics and the proliferation of sophisticated commercially-available software have greatly enhanced the ability of the analyst to assess the results of engineering calculations and present those results to others.

The use of PATRAN by an engineering mechanics group as a pre- and postprocessor for finite element modeling will be discussed. Specifically, the design and analysis of a heat exchanger support that experiences severe thermally-induced stresses will be examined. Both heat transfer and stress analyses for two different design concepts will be presented. The use of PATRAN as a visualization/presentation tool will be emphasized.

THE ENGINEERING MECHANICS section of the Advanced Engineering Technology group provides analytical and computational support to many research and development projects at Los Alamos. Described in this paper are the thermal and structural analyses of a component for an advanced research and development project. Specifically, two design concepts for supporting a heat exchanger were examined. Models for both concepts were built using PATRAN. ABAQUS was the analysis code. The feasibility of each design was assessed based on thermal behavior and stresses and deflections in the structures. PATRAN, with its vivid portrayals of the variations of these quantities through the structures, was instrumental in these assessments.

COMPUTER SYSTEMS

A VAX 11/785 was used as the host for finite element model construction using PATRAN. Tektronix 4208 and 4115 graphics devices were utilized in the pre- and postprocessing stages. The Advanced Engineering Technology group has been using PATRAN for finite element model construction and interpretation of results since May 1986. All analyses using ABAQUS were performed on a CRAY XMP-48.

DESIGN TASK

The basic issue was to design a structure that would support a 5000-pound heat exchanger, yet thermally isolate the heat exchanger from the components below it. Isolating the heat exchanger would maintain its elevated temperature resulting in maximum thermal efficiency for the overall system. As shown in Fig. 1, the design also had to permit flow around the vessel.

OBJECTIVES

The goal of the analyses was to assess the structural and thermal behavior of two different conceptual designs. Specifically, the temperature profiles and the thermally-induced stresses were calculated. Deflections of the structures were also of interest. These three parameters--stress, deflection, and heat leak--were used to assess the suitability of each design.

CONCEPTUAL DESIGN #1

THE CONCEPT - The initial conceptual design consisted of an inner and an outer ring connected by welded struts. The heat exchanger pressure vessel would bolt to the top of the inner ring while the outer ring would bolt to ground. Thermally the struts would act as resistors, tending to isolate the inner from the outer ring. The openings between the struts would serve as air coolant passages.

THE FINITE ELEMENT MODEL - PATRAN was used to construct the finite element model of a 30° segment of the structure, as shown in Fig. 2. This repeatable pattern was then used to quickly construct the full model shown in Fig. 3. The ability of PATRAN to use named entities to create new Phase II portions of the model proved especially useful. Obviously, a full model was not necessary from an analytical perspective. However, visualization of thermally-induced behavior (deflections and stresses) was considerably enhanced through the use of complete models of the structure. After Phase I and Phase II model construction in PATRAN were completed, the PATABA translator was used to create an ABAQUS input file. The modeling process took approximately 20 hours. The finite element model was composed of approximately 1500 eight-noded solid brick elements. Material properties for stainless steel were assigned to the elements in the model.

THERMAL ANALYSIS - The boundary conditions on the thermal model included isothermal surfaces, convectively-cooled surfaces, and insulated surfaces. Nodes on the top of the inner ring were held at $650^{\circ}F$, which was the estimated temperature of the outside surface of the heat exchanger pressure vessel. Nodes at the bottom of the outer ring were held at $170^{\circ}F$ to simulate the ground condition. Because the purpose of the "two rings separated by struts" concept was to thermally isolate the two rings from each other, yet allow air flow around the heat exchanger, there was some question about whether the struts should be covered with an insulating material or be left bare to be cooled by the passing fluid. Hence, both of these boundary conditions were investigated. All other surfaces in the model were perfectly insulated ($\partial T/\partial X = 0$).

Figures 4 and 5 present the temperature profiles for the insulated and convectively-cooled strut situations, respectively. PATRAN has been used as the postprocessor. As expected, the struts act as thermal resistors, thermally isolating the two rings from each other. The temperature in the inner ring is fairly constant, as shown in Fig. 4. Similarly, the outer ring also remains relatively isothermal, but at a much lower temperature than the inner ring. The differential thermal expansion that occurs between the two rings results in excessive stresses in the inner ring and in the struts. A comparison of Figs. 4 and 5 shows that convectively cooling the struts is an effective mechanism for removing heat from this structure. However, this cooling increases the <u>difference</u> in temperature between the inner and outer rings resulting in an even more severe stress problem. Based on the thermal profiles for the two design options, as presented with PATRAN, it was determined that insulating the struts would be necessary.

STRESS ANALYSIS - The linear elastic stress analysis was performed with the complete model shown in Fig. 3. Nodes at the bottom of the outer ring were constrained in the vertical degree of freedom (Z-direction). Point loads simulating the weight of the heat exchanger were assigned to the nodes at the top of the inner ring. The temperature profile shown in Fig. 6 was also imposed. Note that the initial temperature of the structure was 70°F.

Results in the form of a von Mises stress contour plot and a deformed mesh plot are presented in Figs. 7 and 8. Inspection of Fig. 7 shows that stresses exceeding the yield strength (55 ksi) of the material are present in the struts and in the inner ring. Figure 8 provides some justification for these excessive stresses by giving a feel for the deformation behavior of the structure. Presented in this figure is the deformed finite element mesh. The inner ring undergoes the largest displacement because it experiences the largest temperature excursion (70°F to 650°F: $\Delta T = 580^{\circ}$F). The outer ring does not "grow" as much as the inner ring. This differential thermal expansion causes the struts to load the inner ring in a "torsional" fashion. In response to this loading, the inner ring "rolls out" as shown in the figure. PATRAN was instrumental in visualizing the deformation behavior of this structure.

FEASIBILITY ASSESSMENT - The thermal behavior of the structure was predictable. The struts serve as effective thermal resistors, isolating the two rings from each other. As shown in Fig. 7, a linear elastic stress analysis calculates excessively high stresses that are really artifacts of the analysis assumption. In an actual structure, plastic deformation would occur once the material yield point was exceeded. However, this level of stress still could not be tolerated. The nature of the application (repeated thermal cycling with very limited opportunities for periodic inspections) dictates that stresses remain below the yield point of the material. The finite element results, as graphically displayed with PATRAN, demonstrated that conceptual design #1 was not acceptable.

CONCEPTUAL DESIGN #2

THE CONCEPT - The previous analysis indicated that the large differential thermal expansion between the inner and outer rings would have to be accommodated. Figure 9 presents the second-generation design concept, which consisted of two rings connected by pins that were free to slide in the outer ring. Such an arrangement would accommodate the differential radial expansion of the two rings.

THE FINITE ELEMENT MODEL - Figure 10 displays the dimensions of the 15° segment of the structure that was modeled. The finite element mesh developed with PATRAN is shown in Fig. 11. This model, composed of approximately 700 twenty-noded brick elements, took approximately 24 hours to create. Note that the bolt holes in the top surface of the inner ring have been included and the outer ring has been modeled as a "pedestal". Link elements were utilized to model the connection between the sliding pin and the pedestal. After Phase I and Phase II model construction using PATRAN were completed, the PATABA translator was used to create an ABAQUS input file. Material properties for stainless steel were assigned to the elements in the model.

THERMAL ANALYSIS - The nodes on the top of the inner ring were held at 650°F and the nodes at the bottom of the outer ring were held at 170°F, as before. All other surfaces, including the circumference of the pin were perfectly insulated ($\partial T/\partial X = 0$). Results from a steady-state thermal analysis are presented in Fig. 12, using PATRAN as the postprocessing software package. The pin acts as a resistor, thermally isolating the inner ring from the outer ring. Heat leak from the inner ring to the outer ring through the pin is less than the heat leak through the struts of the previous design. Thus, the second conceptual design is better from a thermal perspective than the first.

STRESS ANALYSIS - A linear elastic stress analysis was performed with the model shown in Fig. 11. Nodes at the bottom of the outer ring "pedestal" were constrained in the vertical (Z) direction. Point loads were applied to the nodes at the top of the inner ring to simulate the weight of the heat exchanger. Symmetry boundary conditions were invoked at the cutting plane locations. The temperature profile shown in Fig. 12 was imposed on a structure with an initial uniform temperature of 70°F. Link elements were used to simulate the pin-outer ring connection. Note that the pin was free to slide over the outer ring pedestal in the radial (X) direction, thus modeling unconstrained radial expansion of the inner ring.

Results are presented in the form of von Mises stress contours in Fig. 13. PATRAN was again used for the postprocessing and clearly indicates the areas of high stress. The peak stresses in this configuration are, however, well below the yield point of the material. This particular conceptual design requires that contact stresses between the pin and the outer ring be examined. In fact, a detailed effort to assess these effects was made, but it will not be discussed here.

FEASIBILITY ASSESSMENT - The thermal behavior of the second conceptual design was as expected. The pins serve as very effective thermal resistors, isolating the two rings from each other. Temperature profile plots developed with PATRAN graphically display the temperature drop across the length of the pin. The linear elastic stress analysis indicated that this conceptual design was superior to the previous design. Allowing the two rings to expand independently significantly reduced the stresses in the structure.

CONCLUSIONS

The finite element method is a powerful analysis tool used by engineers to assess the thermal and structural behavior of components. In the past, a considerable portion of the modeling effort was consumed in the mesh generation and result interpretation phases. Recent advances in computer graphics and the proliferation of sophisticated commercially-available software packages (such as PATRAN) have greatly enhanced the productivity of the analyst. Specifically, with the use of PATRAN, an analyst can quickly construct finite element meshes, interpret the results of engineering calculations, and present those results to others for numerous different conceptual designs. This paper has demonstrated how PATRAN serves as a vital tool to accomplish these tasks in the engineering mechanics section of a research and development organization. The heat exchanger support analysis presented is but one example of the myriad of applications in which visualization using PATRAN has proven indispensable.

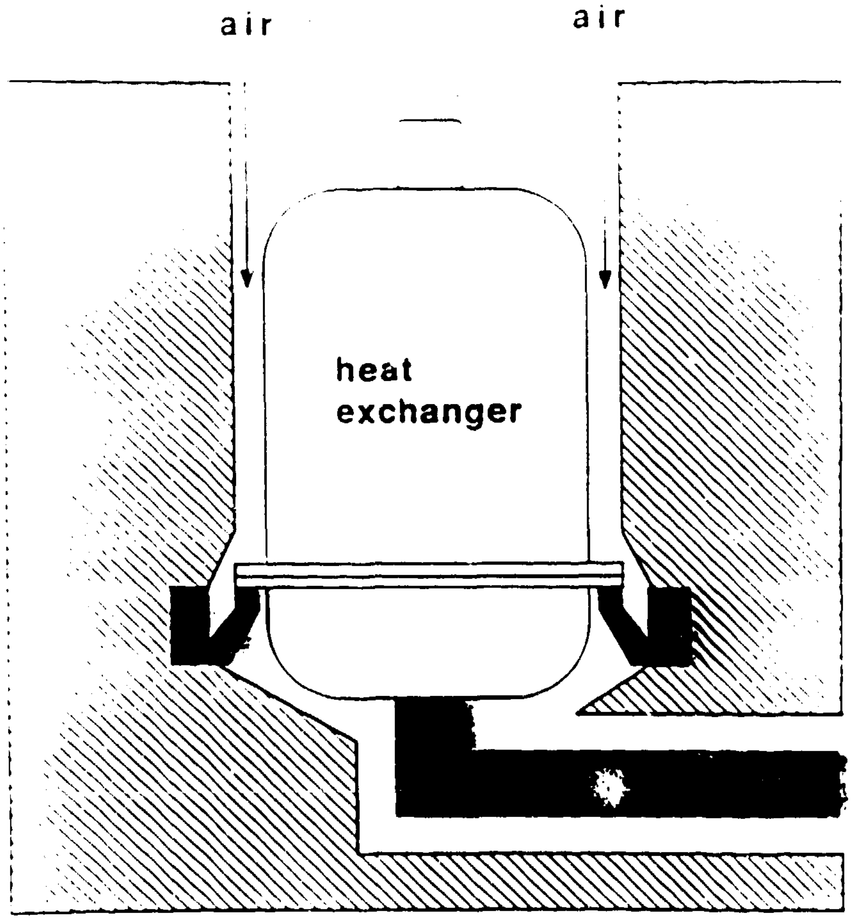

Fig. 1. Heat Exchanger Configuration.

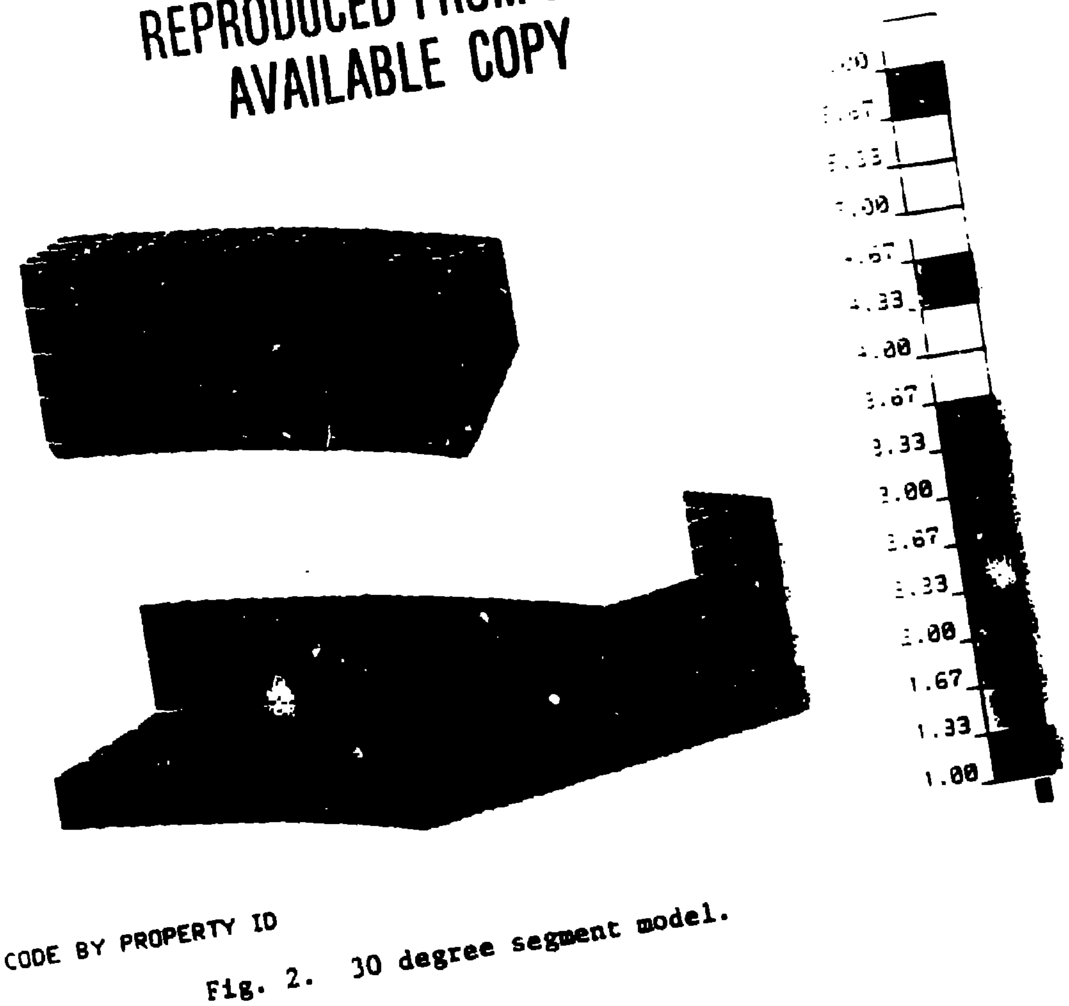

Fig. 2. 30 degree segment model.

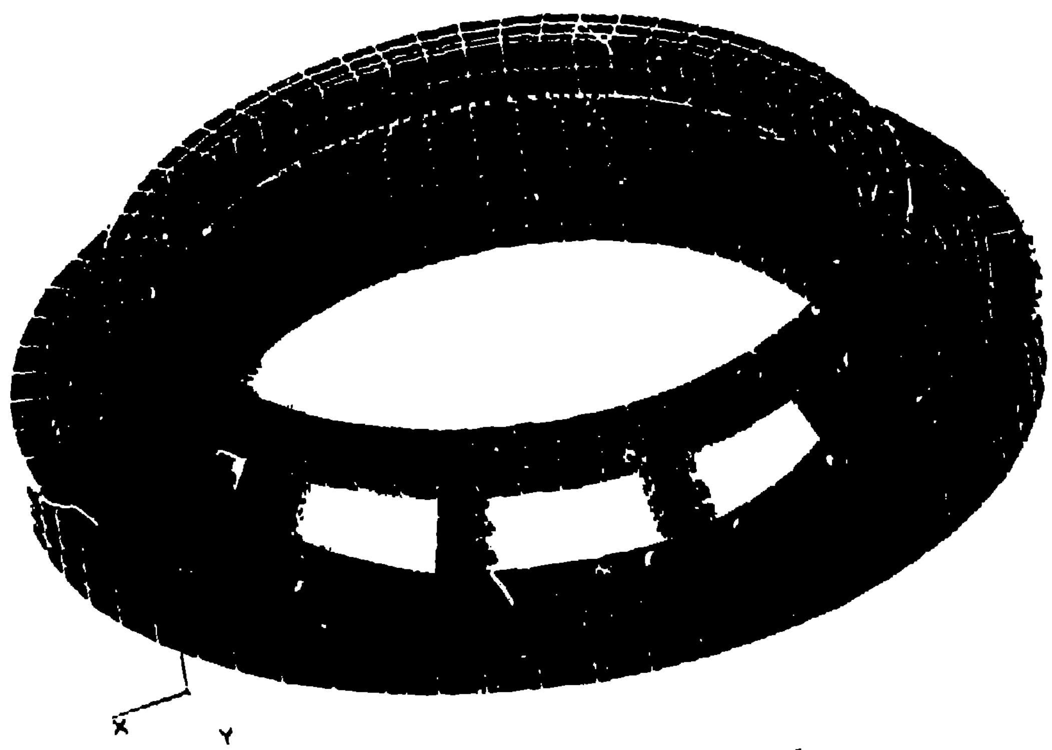

Fig. 3. Full (360 degree) model.

Fig. 4. Temperature contours - insulated strut.

Fig. 5. Temperature contours - convectively-cooled strut.

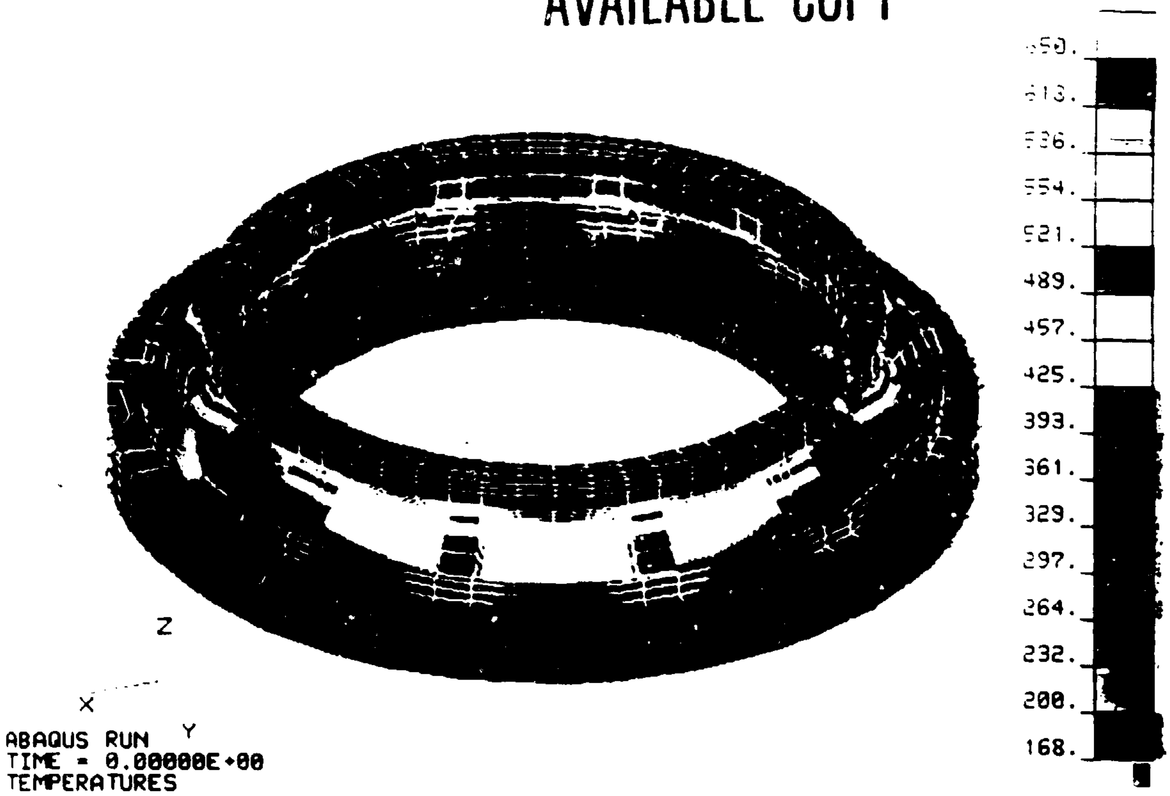

Fig. 6. Full model temperature contours.

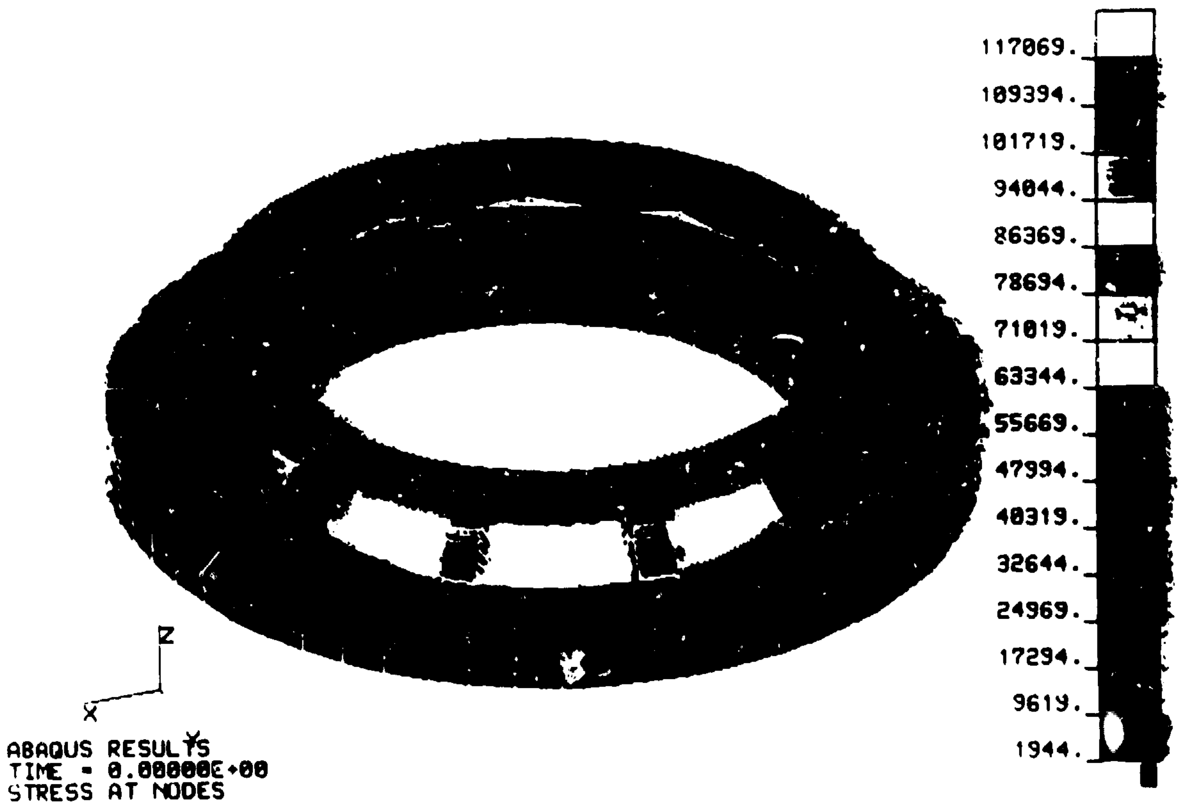

Fig. 7. Full model von Mises stress contours.

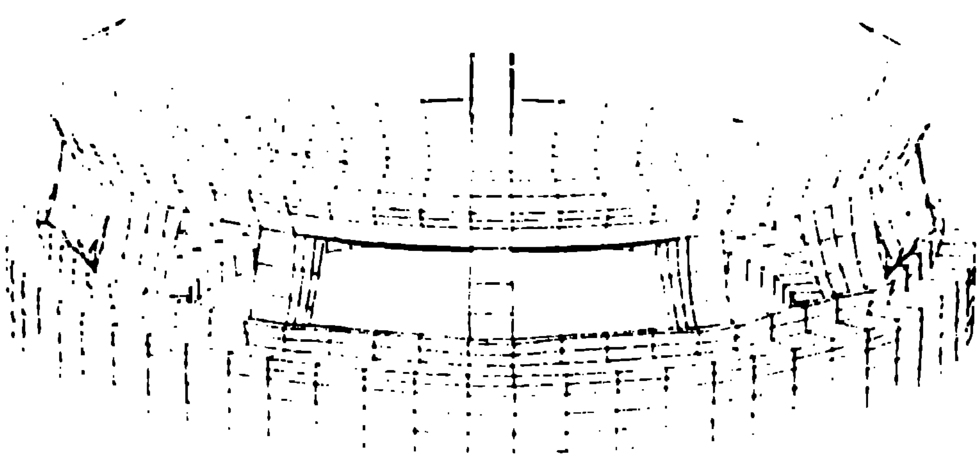

Fig. 8. Deformed Mesh.

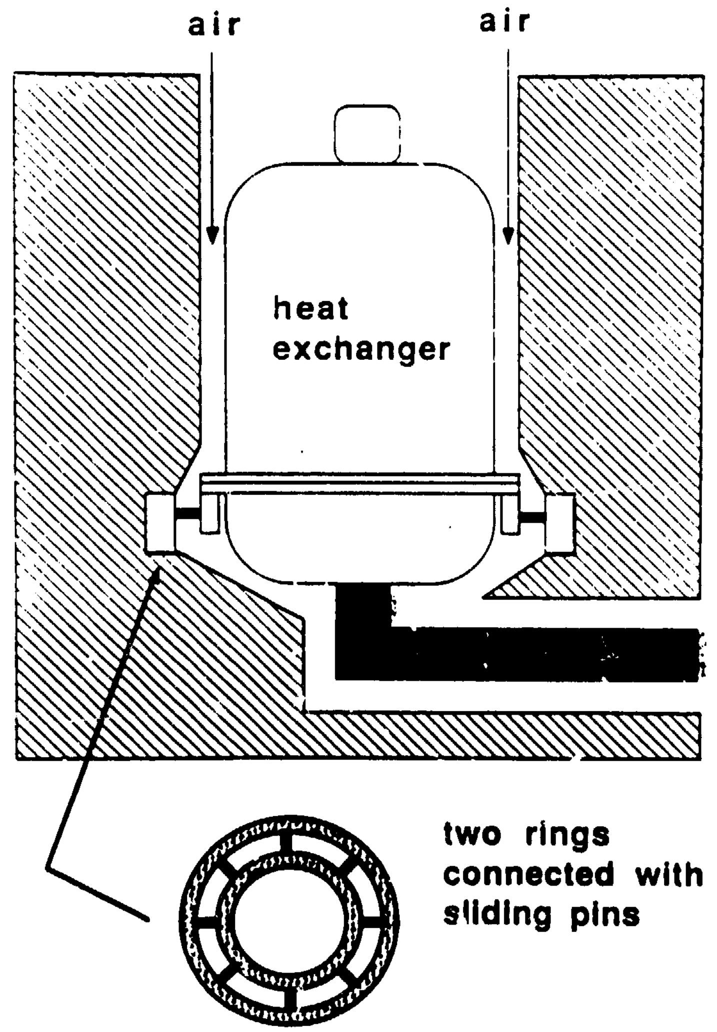

Fig. 9. Conceptual heat exchanger support, Design #2.

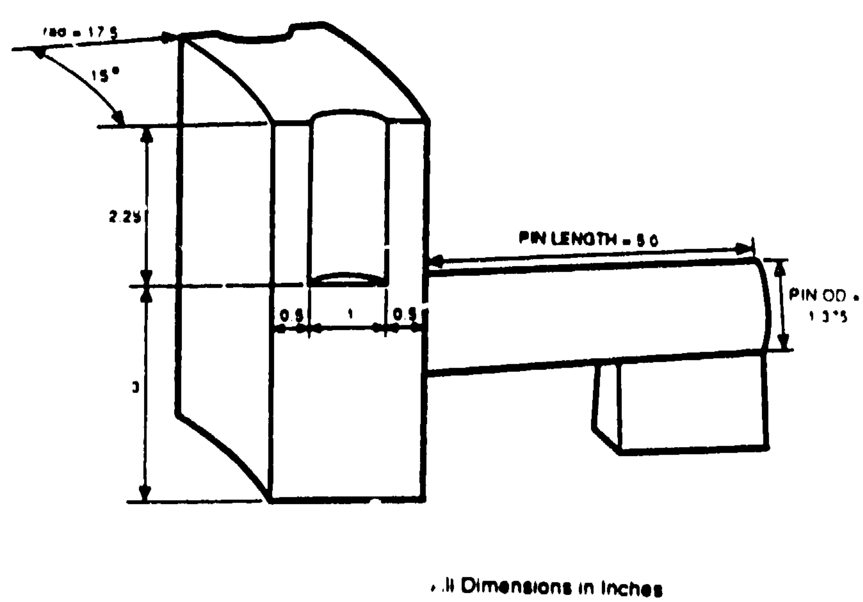

Fig. 10. Dimensioned 15° ring support.

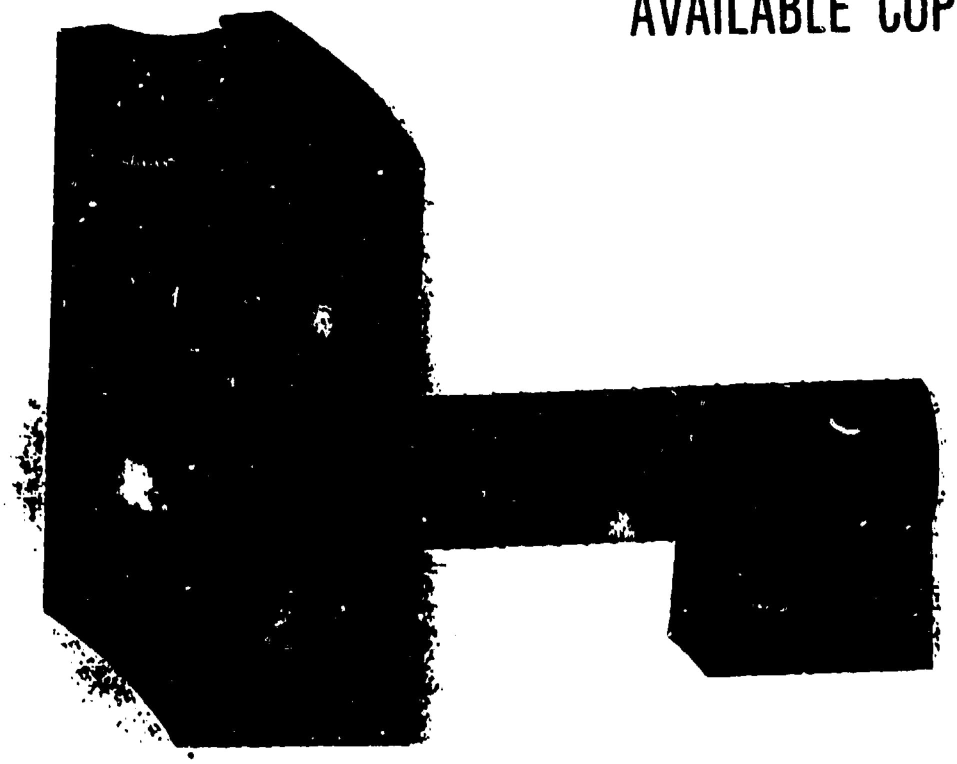

Fig. 11. Finite element mesh.

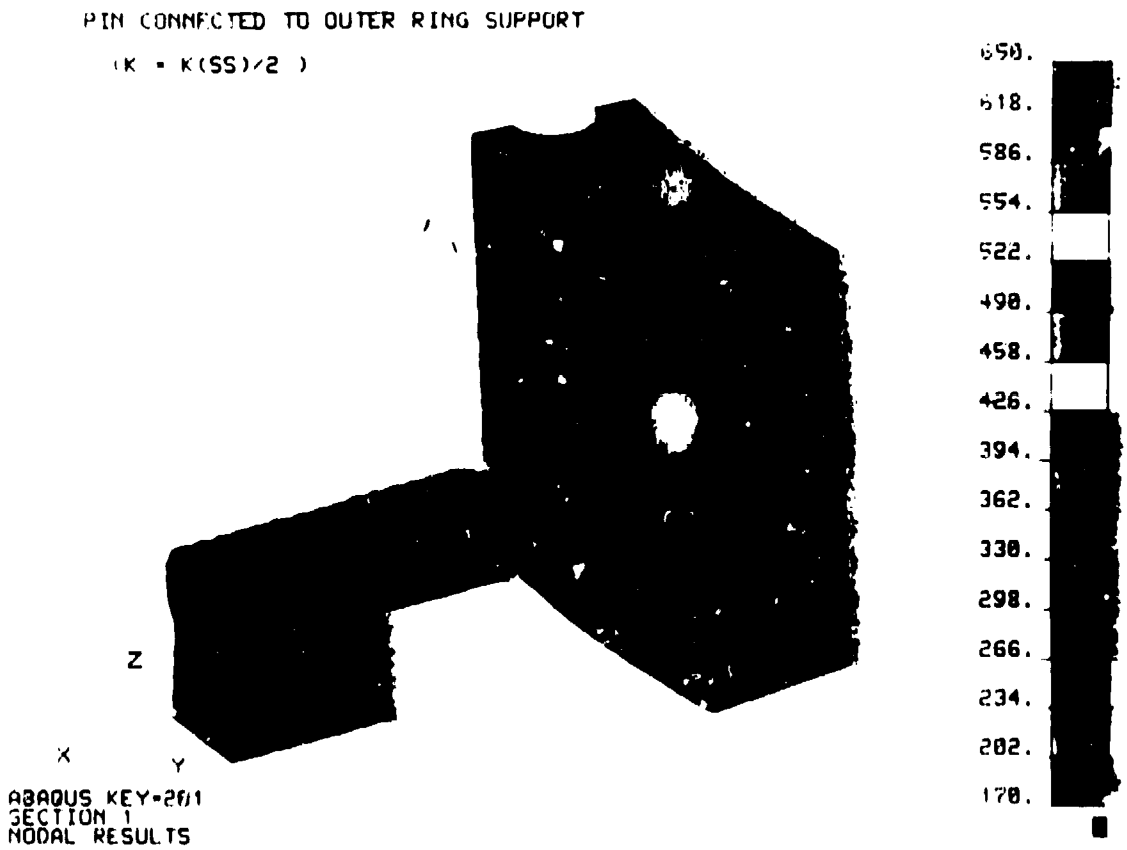

Fig. 12. Temperature profile (Deg. F).

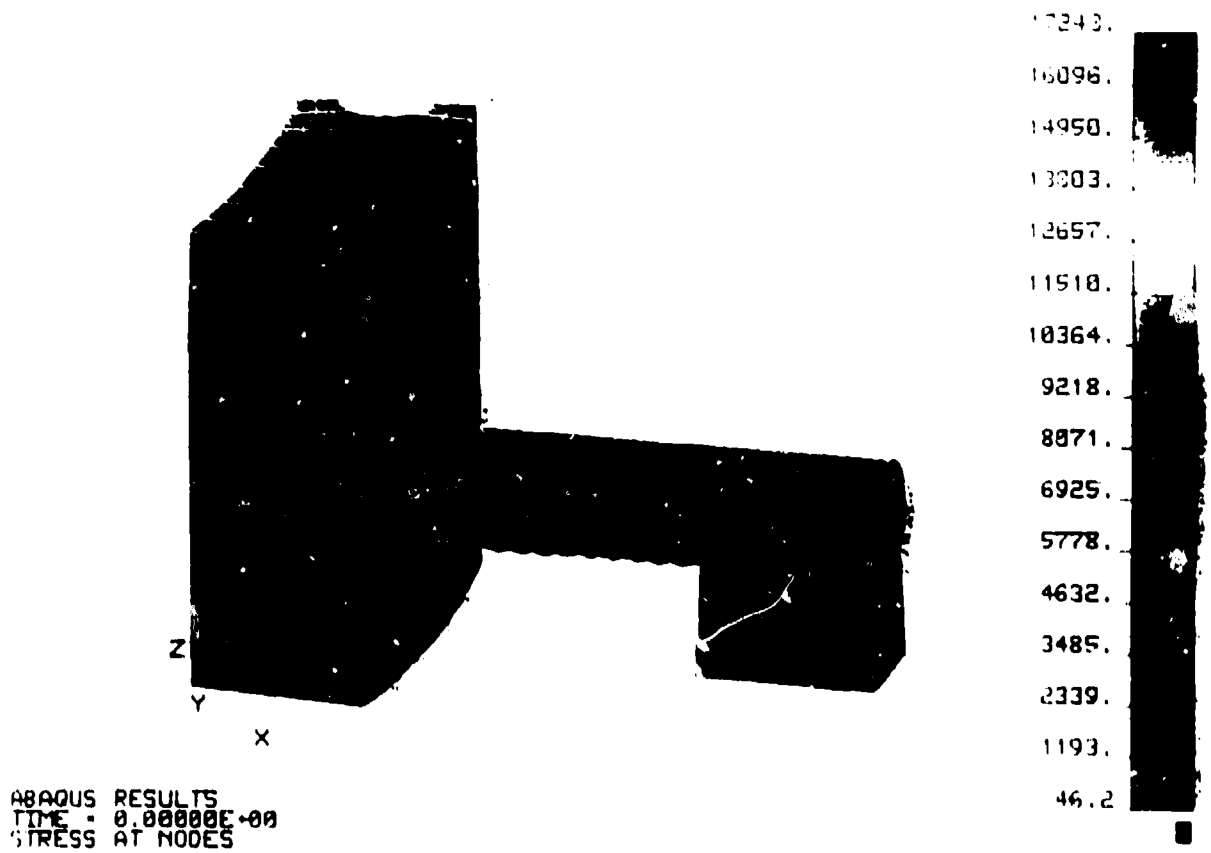

Fig. 13. von Mises stress profile (PSI).

Daniel J. Weinacht

B.S. Mechanical Engineering with/Highest Honors, South Dakota School of Mines and Technology, May 1984

Graduate Research Assistant, Los Alamos National Laboratory, Summer 1984.

M. S. Mechanical Engineering, University of Illinois @ Urbana-Champaign, May 1986.

Staff Member, Los Alamos National Laboratory, June 1986-Present.

Currently enrolled in the PhD program in Mechanical Engineering, University of Illinois @ Urbana-Champaign.