IMPACT OF THE BOUNDARY ELEMENT METHOD ON ENGINEERING QUALITY AND PRODUCTIVITY

A. Wanderlingh
Mechanical Products Department
United Technologies Corporation,
Hamilton Standard Division
Windsor Locks, Connecticut, USA

INTRODUCTION

Two extremely important challenges confronting the aerospace industry in the 1990's, and beyond, is its ability to produce high quality products and, to deliver them to market rapidly. When one considers the enormous global social changes taking place, such as, the newly unified Europe in 1992, and the opening up of the Eastern Bloc and Asian markets, the incentives to the aerospace industry become quite clear. That is, the companies that are the first, with the best, will capture the largest market share.

To this end engineering companies spend vast amounts of dollars on high speed state-of-the-art mainframe and workstation computers, the latest in CAD (computer aided design) tools, and a variety of analysis programs in order to help the engineer answer difficult and time consuming questions such as:

1. How does one rapidly traverse the path of preliminary design, production design, and analysis?

2. How reliable are the analysis results?

3. How much will the analysis cost?

4. Can the design and analysis be done in a timely fashion?

5. Is the design acceptable? Optimum?

During the 1980's, Boundary Element Method (BEM) Computer Programs, such as BEASY [1], have proven themselves to be quite effective in helping engineering companies face the challenges of the nineties. The primary advantages over other approaches, such as finite element, are that:

- The programs are easy to learn while not requiring the engineer to read volumes of user manuals.
- The programs are easy to use while offering a high level of sophistication in the capabilities they provide for solving simple and complex engineering problems.
- The Boundary Element Method delivers reliably accurate surface results.
- The method communicates far more easily with available engineering CAD systems.
- The use of this method dramatically reduces analysis cycle time because of less geometry generation/modification.

This chapter will illustrate, with the use of actual engineering applications, how the Boundary Element Method has contributed to product quality and engineering productivity. Finally, a discussion on what the future direction method development should take, in order to further advance our ability to effectively compete, will be presented.

DESCRIPTION OF A PRODUCT DESIGN CYCLE

Product Design Flowchart

A somewhat typical product design cycle is best described by the flowchart [2] shown in Figure 1. The cycle beings with a customer defining the performance and packaging requirements of a particular product. The Design/Drafting Group then translates the requirements into a geometric representation with the use of a computer-based CAD system. An example is shown in Figure 2. From such a geometry definition an analytical model is constructed with the use of either a direct translator or through a neutral file system, such as, IGES (Initial Graphics

Exchange Specification). With the analytical model now completed, an analysis is performed. In the case of a structural analysis, a maximum stress is identified and compared to Low Cycle Fatigue (LCF) and High Cycle Fatigue (HCF) material allowables. If the allowables are not exceeded, a prototype is produced by the Manufacturing Department and then tested.

If, however, the customer requirements are not met, the design cycle is repeated until a satisfactory product can confidently be built and tested. Once the tests confirm that the product meets all of the necessary conditions, the product enters into mass production.

Three important points emerge from the examination of the product design cycle flowchart. They are:

1. The time it takes to obtain a satisfactory design can potentially be very long depending on how many times steps are repeated.

2. The quality of the tools, and how they are integrated with respect to each other, will determine the design cycle time.

3. Analysis and CAD tools must be accurate in order to guarantee a one-pass design cycle.

Therefore, an engineering organization must strive to have the highest quality tools and have them function as one in order to remain competitive.

BEM's Key Role in the Design Cycle

Use of the Boundary Element Method during the design cycle has contributed to the competitiveness issue by dramatically reducing pre- and post-processing, and analysis time and cost without compromising the quality of analysis. In fact, pre- and post- processing times have been so greatly reduced that a more detailed analysis can be performed thereby improving product quality.

Pre-processing: The CAD Connection **CAD systems and** the Boundary Element Method work extremely well together because each uses identical geometric entities, such as, points, lines, splines, circles, and patches (3D only). The significance of this is quite profound when considering that a product's **exact**

geometry can be used in the analysis phase. An exact representation of a product's geometry is essential, for example, for determining stress concentration effects.

There are currently three methods employed by engineers to transfer CAD data to a boundary element analysis computer program:

1. IGES

2. Commercially marketed translators

3. In-house developed translators

IGES appears to be the most widely used method of transferring geometric entities from CAD to a boundary element analysis. Unfortunately, this approach has a drawback. IGES scrambles line and patch directions. Consistent entity direction is critical in determining the type of analysis, finite or semi-infinite, to be performed. Scrambled entity definition will prevent an analysis from being successfully executed. Therefore, the analyst must spend time adjusting the data prior to attempting an analysis.

This deficiency has been eliminated at Hamilton Standard by using an in-house developed computer program which is linked directly to a CAD program called CADAM [3] and BEASY. This link permits analysis specifics, such as, zone identification, element break-up, applied traction conditions, and constraints to be defined directly on the CADAM geometry. The link then interprets this information and generates an analysis-ready program input file. This method significantly reduces model generation time. An axisymmetric example is shown in Figure 3.

Commercially available translators, while not providing the same level of linkage, nonetheless do provide a reliable means of bridging geometry generation programs to the boundary element analysis program. Such a translation has been demonstrated between BEASY and the commercial pre- and post-processing product called I-DEAS [4].

It is worth noting that, despite any minor shortcomings that may exist regarding CAD and the Boundary Element Method, this connection is ultimately more robust than the CAD and Finite Element Method connection. Despite recent advances in finite element automeshing techniques, it is still quite difficult, for example, to obtain a satisfactory 3-D finite element model for a complex structure.

Many times exact geometric detail must be sacrificed in order to populate the volume with elements. Also, element shapes are often distorted thus causing a loss of solution accuracy. Therefore, an engineer that understands the pitfalls of the finite element approach will spend a great deal more time building a finite element model than a boundary element model.

There have been documented cases where the deficiencies of the finite element modeling approach have led to major delays in a product's development because during the test phase some of those deleted geometric features proved to be highly stressed.

Analysis: Confidence **in the Results** Here lies the heart of the design cycle. It simply does not make any sense to rapidly construct an analytical model only to get the wrong answers fast. Fortunately, this is not the case for boundary element computer programs like BEASY. In fact, many designers after having used boundary elements refuse to go back to using finite element or finite difference approaches. It is worth examining the reasons why.

Based on discussions with engineers, three key reasons emerge as to why they prefer the Boundary Element Method:

1. The reduction of dimensionality. No internal element meshes are required as in Finite Element or Finite Difference Methods.

2. Discontinuous boundary elements, in 2-D and Axisymmetric cases, provide an excellent means of assessing solution convergence rapidly.

3. Removal of the mesh point compatibility constraint in 3-D cases.

Removal of internal element meshes completely eliminates the concerns of element distortion and mesh density. The engineer no longer worries about results being corrupted because an internal element violates shape rules or, is so large that surface results are compromised due to the extrapolation that is commonly performed in Finite Element and Finite difference Methods.

With the Boundary Element Method, the age old question of how many internal elements are needed to properly calculate through-the-thickness gradients, such as bending in structural analysis, is put to rest. The primary concept of the Boundary Element Method is that the volume can be represented by surface integrals. This simply means that if the solution on the surface can be obtained, the solution in the interior of a body is automatically determined and can be recovered in a far more simple manner with the use of internal points.

One of the most annoying deficiencies demonstrated by Finite Element and Finite Difference Methods over the years is the lack of a clear and easy way to determine solution convergence. Developers of these two methods would have engineers run a series of cases in which the element mesh density is increased each time a model is analyzed. With a minimum of three analyses, it is hoped that the solution indicates a trend to a converged solution. Unfortunately, design schedules cannot accommodate this approach because of the amount of time it takes to generate and interpret the necessary data. Many times only one analysis can be performed because of time and dollar constraints.

Use of discontinuous elements in the Boundary Element Method has remedied this completely in 2-D and Axisymmetric cases. An engineer, by inspecting X-Y plots of surface results can get an immediate indication of solution accuracy by examining the differences of the element values at each mesh point. An example of such a plot is given in Figure 4. If the differences at the peak value location(s) is greater than fifteen to twenty percent, the engineer must run another analysis with a finer element mesh. If the differences, however, are less than fifteen percent, the solution has converged. Typically, I have found that the first analysis, particularly in 2-D and Axisymmetric cases, converges to within one percent of the correct answer. This is due in large part to the fact that the unknowns, such as displacement and tractions in structural analyses, are solved for directly in the Boundary Element Method. Therefore, if quadratic elements are used, a quadratic function represents both the displacement and traction unknowns. In the Finite Element Method, displacement functions are differentiated in order to obtain strains. This differentiation causes a loss in accuracy in obtaining stresses. So, if the engineer is interested in stresses, a finer element mesh must be used to capture the proper gradient.

This convergence check capability profoundly impacts the end quality of the product because, test data is not generally available until the final development phases of the design cycle. Therefore, the more confident one can be of the analytical predictions early on, the greater the chances are that only one pass through the design cycle will be necessary.

The example of an internal spur gear tooth [5], presenting Figures 5 through 9, best illustrates the importance of the first two reasons given. A 2-D plane stress finite element model of the structure is shown in Figure 5. The model consists of 1,060 constant strain elements. A point load is applied at the tooth pitch diameter and the tooth boundary is completely constrained from motion. The boundary element model, with the same boundary conditions, is shown in figure 6. This model uses only forty-one quadratic surface elements to describe the structure topology.

Surface stresses along the tooth fillet, for both models, are shown in Figure 7. The boundary element peak stress is five percent higher than the finite element result. While the results of these two methods compared well, there is no way of determining whether the finite element solution, by itself, represents a converged solution. Therefore, several additional models of the internal spur gear tooth were constructed to determine stress sensitivity due to element mesh size. The models, which were analyzed using an in-house finite element computer program and MSC/NASTRAN[6], are shown in Figure 8.

The element density of the finite element meshes was increased in the vicinity of the tooth fillet until the peak fillet stress did not change more than five percent. While the initial boundary element X-Y plot of the surface stress results indicated solution convergence, two additional analyses, with finer element meshes, were performed for the purpose of confirmation. The comparison of peak fillet stress results for all models is shown in Figure 9. It can be clearly seen that the boundary element results have indeed converged. The in-house finite element program, on the other hand, did not converge until the third analysis. The MSC/NASTRAN solution converged, but at a higher peak stress level. This maybe due, in part, to the extrapolation of element results to the surface. However, regardless of the magnitude of the peak stress, the key point which needs to be emphasized once again, is that additional time consuming analyses must be performed when using the Finite Element Method.

In 3-D boundary element cases, removal of mesh point continuity has drastically reduced analysis time without compromising solution quality. For example, if mesh point continuity was a requirement the model shown in Figure 10, would have resulted in an unacceptable burden on computer space resources due to the increased element mesh density required to describe the surface. Instead, several design iterations of this structure were possible in days rather than weeks. Geometry modeling has truly become extremely easy without this cumbersome restriction. It is worth taking note that nodal point compatibility is absolutely essential in both the Finite Element and Finite Difference Methods.

Post-processing: The Importance of Presenting Results in a Meaningful Way

Typically, analyses generate volumes of information which is output in printed form. It has been recognized for many years that this type of output is cumbersome and does not lend itself to rapid study. Color graphic output, such as that illustrated in Figure 11 and 12, have gained wide acceptance in engineering as being the post-processing method of choice.

However, there are limitations to color graphics output. For example, many software programs represent varying levels of a quantity as color-coded contour bands. From these color bands the engineer is unable to identify the direction of the surface quantity (i.e. stress), thereby forcing a return to the computer printout. Of course, this assumes that the particular information required is listed. For example, consider the case where an engineer needs to determine strain gage placement. A color contour plot of principal stresses would not provide the direction in which the stress is acting. If the principal angels are not present in the computer printout, the engineer must now perform the necessary calculations in order to properly orientate the gages. Another costly alternative might be to use strain gage rosettes in a number of places to insure that the peak stress(s) are captured.

Another limitation of color contour bands is their inability to exactly identify the magnitude and precise location of the peak stress. This would require a large number of stress level bands in a high gradient area. Typically, default band levels used are generally inadequate. This can ultimately lead to an incorrect fatigue analysis assessment and incorrect strain gage placement.

To offset this deficiency, X-Y plots are used. This may provide a better way of identifying the peak quantity, assuming the engineer has the ability to vary the number of division on the plot so that the peak can fall on an identified division. This avoids any time-consuming interpolation between division. The presentation of a limited amount of data in tabular form on the computer screen further compliments the X-Y plot information. Exact values can be obtained in this manner.

A good post-processing system must be able to provide the engineer with the most global, as well as the most detailed, picture of a problem as rapidly as possible. System flexibility in displaying data is essential in reducing design cycle time.

ENGINEERING APPLICATIONS

In this section several typical engineering applications which best illustrate the effectiveness of the Boundary element Method in solving a variety of problems, will be discussed.

Application 1 - 2-D Plane Stress: Stress Concentration Factor Determination

The boundary element computer program BEASY was used to provide engineering guidelines on how to identify stress concentration (K_t) magnitudes. Two-dimensional (2-D) cases were selected from Peterson's Stress Concentration Handbook [7]. The geometries selected are shown in Figures 13A and B. The first case (Figure 13A) represents a tension bar with infinite rows of semi-circular edge notches. Under a tension load the maximum stress occurs at the base of each semi-circle. The second case (Figure 13B) represents a U-shaped member subjected to a spreading load. Depending on the structure geometry and distance of the applied load from the center of curvature, the maximum stress can occur either at the base of the U or near the tangency of the radius to the vertical arm.

The boundary element model for the first case is shown in Figure 14. Geometry parameters follow:

A/D = .20
b/a = 3.503

Peterson would predict a Kt = 2.32. As expected, the boundary element analysis results in a maximum stress which occurs at the base of each semi-circle; see Figure 15. In order to obtain the boundary element nominal stress, stresses through the minimum section were plotted and the area under the curve was determined using a FORTRAN program. By dividing this nominal value into the maximum stress, a boundary element Kt = 2.32 was obtained.

Two boundary element models were constructed for the second case; see Figure 16. Geometry parameters for each model follow:

Model 1	Model 2
d/r = 1.0	d/r = 2.0
w = r	w = r
P at m = 3r	P at m = 3r

For Model 1, Peterson predicts a Kt = 1.243 at the position 1 location. For Model 2, Peterson predicts a Kt = 1.285 at the position 2 location which is approximately 20° off the horizontal axis.

The boundary element analysis of Model 1 results in a maximum stress at position 1; see Figure 17. The same Kt extraction procedure discussed in Case 1 results in a boundary element Kt = 1.26 which is 1.26% higher than the Peterson prediction. Analysis of Model 2 results in a maximum stress at position 2; see Figure 18. In this case the nominal stress used to calculate a Kt is calculated in the narrow section of the model. A Kt of 1.293 was calculated. This is .58% higher than the Peterson result. A clear understanding on how the K_t should be determined, was achieved. The nominal stress must always be calculated at the minimum section closest to the maximum stress location. Otherwise, a significant error in the calculation of K_t will occur.

Application 2 - 2-D Plane Stress: Optimization of a Ceramic Vane Pump Rotor Desire

In a paper entitled, "BEASY Used for Optimization of a Rotor Stress" [8], Mr. P. Hearn describes how, with the use of BEASY, he was able to significantly improve the cyclic fatigue life of this component. In the course of this project he determined that the Boundary Element Method is an effective tool in performing rapid geometric iterations and is capable of providing a highly accurate result with the first analysis. He based his findings on the comparison of the boundary elements approach to that of the finite element approach.

The ceramic vane rotor shown inside the cam block is illustrated in Figure 19. Two loading conditions which combine to produce the maximum to minimum stress range in the key slot fillet is illustrated in Figure 20. The initial analysis was conducted using the finite element computer program MSC/NASTRAN. The original model is shown in Figure 21. In the process of constructing a second finite element model with modified geometry, it was determined that the finite element approach was taking too much time and that the design schedule could not be met if this course of action was followed. A decision was then made to use the Boundary Element Method in hopes that the schedule could still be met. The BEASY model, which matched the NASTRAN model, is shown in Figure 22. Comparison of the peak slot stress revealed a fifteen percent difference. The finite element results were fifteen percent lower. The net cause of this discrepancy was the finite element mesh was not fine enough. The final finite element fine mesh, utilizing quadratic elements, is shown in Figure 23. This model produced results consistent with the original BEASY model. With this resolved, the boundary element optimization of the ceramic vane pump rotor proceeded to a successful conclusion. The project was completed ahead of schedule due to the use of this method.

The author concluded that "The Boundary Element Method was found to be an effective tool for both geometric iterations and accurate first cut prediction of peak elastic stresses. It subsequently proved to be invaluable in predicting a final peak elastic stress first missed when analyzed with MSC/N-ASTRAN. The Finite element Method was able to match this peak stress but only after a deliberate and highly concentrated effort that was instigated by a mismatch with the Boundary Element Method results. This analysis and resulting study of meshing provided insight into the benefits of the Boundary Element Analysis Method and its relative insensitivity to mesh density."

Application 3 - 2-D Plane Stress: Displacement and Stress Reduction

A 2-D boundary element model of a section through a propeller control unit is shown in Figure 24. The section is subjected to a variety of internal pressures. A boundary element model of the original section, along with the imposed boundary conditions is shown in Figure 25. The purpose of this analysis was to minimize wall stress, especially around interior holes, and to keep outer wall displacements under a design allowable. Displacement and stress contours of the original section are shown in Figures 26 and 27 respectively.

The project objective was easily met by simply adding more material to the outer wall and inside the major cavity located at the outer wall and inside the major cavity located at the lower right-hand comer of the structure. The modified boundary element model is shown in Figure 28. Displacement and stress contour plots of the modified geometry are shown in Figures 29 and 30. Stresses and displacements fell within the prescribed limits.

Use of this method enabled the engineer to not only modify the geometry rapidly, but to analyze and modify four other sections as well in a very short amount of time. Speed and analysis accuracy led to the optimization of all sections thereby producing a higher quality product.

Application 4 - Axisymmetric Analysis: Stress Analysis of a Threaded Member

In a paper entitled, "Determination of the Maximum Thread Stress in a Threaded Structure Using the Boundary Element Method" [9], the author was able to dramatically improve the state-of-art of fatigue analysis of these types of structures by accurately modeling the actual threads. Prior attempts to accurately model this type of geometry using the Finite Element Method proved to be unsatisfactory and time consuming. In an attempt to minimize this deficiency, an approximate method which combined a substantially simplified finite element model with the Heywood Method [10] was adopted. Unfortunately, this approach was later proven to be too conservative when compared to actual field service data.

The boundary element model of a threaded member is shown in Figure 31. By applying an assumed thread load distribution, a more reasonable maximum thread fillet stress was obtained. The resulting fatigue life calculation, using this improved maximum thread fillet stress was obtained. The resulting fatigue life calculation, using this improved maximum stress, more closely matched actual field data. With a better correlated analysis, product weight reductions were realized in later designs.

Subsequent boundary element analyses of these types of structures have included the mating parts. Contact analysis has further improved the results because reliance on assumed thread load distributions are no longer necessary.

Application 5 - Three-Dimensional (3-D) Analysis: Distortion of a Fuel Control Housing Due to a Pressed in Bushing

A portion of jet engine fuel control housing, with an inserted bushing (upper right), is shown in Figure 32. Bore diameter (larger diameter-right side) measurements made after the bushing is pressed into the hole, located at the top of the housing, indicate bore diameter out-of-round distortions of .0004 to .0005 inches. This substantially exceeds the maximum established design tolerance. When this maximum is exceeded, the control is unable to function properly.

One of the options explored to avoid excessive bore diameter distortion was to vary the bushing outer diameter. The boundary element approach made this type of parametric study extremely fast and easy. Using contour plots of the bushing and housing, similar to the ones shown in Figures 33 and 34. An appropriate bushing press-fit, which minimized the bore diameter distortions, was determined.

FUTURE DEVELOPMENT DIRECTIONS

The tremendous success of the Boundary Element Method has left engineers with a strong desire for more capabilities. With additional capabilities, there is definitely the opportunity for further increases in engineering productivity. Some of the most needed capabilities include:

Automated Shape Optimization

Cyclic Symmetry

Automated Fatigue Analysis

Improved Post-processing

Normal Modes Analysis

Orthotropic/Anisotropic Material Definition

Temperature Dependent Material Properties

Automated Shape Optimization

Two key design parameters in the aerospace industry are weight and size. Automated optimization of these parameters would result in lighter and more competitive products. An example of an axisymmetric jet engine fuel control housing [2] cover, which was manually designed with these two parameters in mind, is shown in Figure 35. While the final design satisfies all of its functional requirements, there is no guarantee that it represents an optimum (lightest weight and size) design.

In the conventional design process there generally is only enough time to focus on reducing the maximum stress, for example. Unfortunately, the low stress areas go unmodified. This means that the component is usually heavier than necessary. An automated computer design shape optimization program, with user defined objective functions and constraints, could provide a minimum weight design much faster than any conventional design approach. The net result will reduce cycle time while providing a highly competitive product.

Cyclic Symmetry

The aerospace industry produces a significant number of rotating components, such as the propeller system shown in Figure 36. While the geometry is symmetrical, the loading conditions on such systems, at times, is anti-symmetrical. Rather than modeling the entire structure, which would dramatically increase the computational time and tax the best computer systems, an alternate approach would be to perform a cyclic symmetry analysis. This technique has commonly been performed using the Finite element Method for many years. The technique allows for the modeling of the smallest symmetric geometric segment while accounting for any and all anti-symmetric loading conditions. This reduces the problem to something a lot more manageable.

Automated Fatigue Analysis

One often forgets that to obtain the maximum stress value, given a set of conditions, is only part of the design process. A large majority of products must continually undergo a number of load cycles during its useful fatigue life. It would therefore, be extremely beneficial if an automated means of performing a fatigue analysis were developed.

It is envisioned that the engineer would input, interactively, the material fatigue allowable data (usually a Modified Goodman Diagram) in the post-processing system. Steady and cyclic stress load cases would than be defined. These may be a single load case, or, a linear combination of a series of load cases. The post-processor would automatically compare the steady and cyclic stress pairs to the fatigue allowable line, identifying which pairs violated the fatigue limits.

This capability would facilitate an accurate assessment of a product's ability to meet its intended design fatigue life, especially in 3-D cases in which it is not often obvious where the worst combination of steady and cyclic stresses may be located.

Improved Post-processing

While the current tool kit is adequate for evaluating analysis results, there are several improvements which should prove to be very useful. For example, cursor picks directly from X-Y plots, would significantly enhance the engineer's ability to quickly identify exact numerical values.

It would also be useful if the following types of contour plots were added to the tool kit:

Shaded contour bands rather than solid bands. This would quickly indicate the nature of any existing gradients.

Arrow contour plots. This would aid the engineer in identifying strain gage placement and orientation.

Some consideration should also be given to selecting mesh points directly from the boundary element geometry for X-Y plotting purposes.

Normal Modes Analysis

It would be extremely valuable to identify the natural frequencies that a product may exhibit. Without this capability, the engineer will be forced to construct a finite element model in order to obtain this vital information.

For years the Finite Element Method has provided outstanding techniques for extracting eigenvalues. Two excellent eigenvalue extraction methods are:

LANCZOS
MODIFIED GIVENS

Either one of these approaches would be quite acceptable.

Orthotropic/Anisotropic Material Characterizations

One way that aerospace companies are addressing component weight is by using light-weight high-strength composites and directionally solidified metals. The manner in which these materials are aligned with respect to the primary load path can have a substantial impact on a product's ability to meet its fatigue life requirements. If the Boundary Element Method is to be a major player in this rapidly growing area, it must be able to correctly model these types of materials.

Temperature Dependent Material Property Capability

Many aerospace products must be able to function in temperatures that can vary from several hundred degrees below zero (Fahrenheit) to several thousand degrees above zero. With such temperature variations, many materials exhibit significant property changes in modulus and poisons ratio, for example.

To further complicate matters, the temperature variation on some products, at any given time in a flight cycle, can be tremendous. This type of va variation inducts large component stresses. Space vehicles, for instance, which have one side exposed to direct sunlight and the other to the cold darkness of space, is a perfect illustration of an extreme temperature gradient.

A means, therefore, must be found to account for this type of thermal environment without compromising the Boundary Element Method's solution accuracy and procedural simplicity.

CONCLUSION

The principal theme throughout this chapter has been improved international competitiveness through the reduction of engineering cycle time. A case has clearly been made that the use of the Boundary Element Method is contributing to this ever-present challenge.

With regard to geometry modeling, the Boundary Element Method, with its tight coupling to CAD systems, enables the engineer to rapidly generate the truest of product representations. This has a tremendous benefit in determining an accurate fatigue life. It also greatly aids in critical weight reduction and packaging studies.

By providing a simple means for assessing solution accuracy, the Boundary Element Method dramatically impacts product quality and design cycle time. With solution confidence greatly increased, the engineer can spend more time developing a better product. The need to run a series of analyses to determine solution convergence has been eliminated, thereby, shortening design time.

The continual drive to reduce weight and increase product strength by using increasingly exotic composite materials and high strength metals, such as single crystal, emphasizes the need for further Boundary Element Method advances. Dynamics, Cyclic Symmetry, Temperature-Dependent Material Characterizations, and Automated Fatigue Analysis will all contribute to making the Boundary Element Method more useful in aerospace applications.

In the past twenty years, we have seen the Boundary Element Method develop from a pure research activity to a very useful and productive engineering tool. I believe that in the next twenty years, this method will definitely be the method of choice for a very broad range of linear and non-linear engineering aerospace applications.

REFERENCES

1. BEASY is a product of Computational Mechanics, Inc., Southampton England.

2. Wanderlingh, A. Specifications for a Useful Boundary Element Analysis Tool for Industry, Boundary Elements (Ed. C.A. Brebbia), Volume 3, Stress Analysis, pp. 579 to 590.

3. CADAM is a product of CADAM, Inc., USA.

4. I-DEAS is a product of Structural Dynamic Research Corporation, OH, USA.

5. Wanderlingh, A. Comparison of Boundary Element and Finite Element Methods for Linear Stress Analysis-Technical Program Results, Boundary Elements VII (Ed. C. A. Brebbia, G. Marin), Volume 1, pp. 7-33 to 7-52.

6. MSC/NASTRAN is a product of the MacNeal Schwendler Corporation, Los Angeles, CA., USA.

7. Peterson, R. E., Stress Concentration Factors, John Wiley & Sons, New York, London, Lyndey & Toronto, 1974.

8. Hearn, P. BEASY Used for Optimization of a Rotor Stress, Advances in Boundary Elements (Ed. C. A. Brebbia, J. J. Connors), Volume 3 Stress Analysis, pp. 435 to 450.

9. Wanderlingh, A. Determination of the Maximum Thread Stress in a Threaded Structure Using the Boundary Element Method, Advances in Boundary Elements (Ed. C. A. Brebbia, J. J. Connors), Volume 3 Stress Analysis, pp. 377 to 392.

10. Heywood, R. B. Designing by Photoelasticity, First Edition, Chapter 7, Stress Concentration in Screw Threads, Bolts and Nuts, 1952.

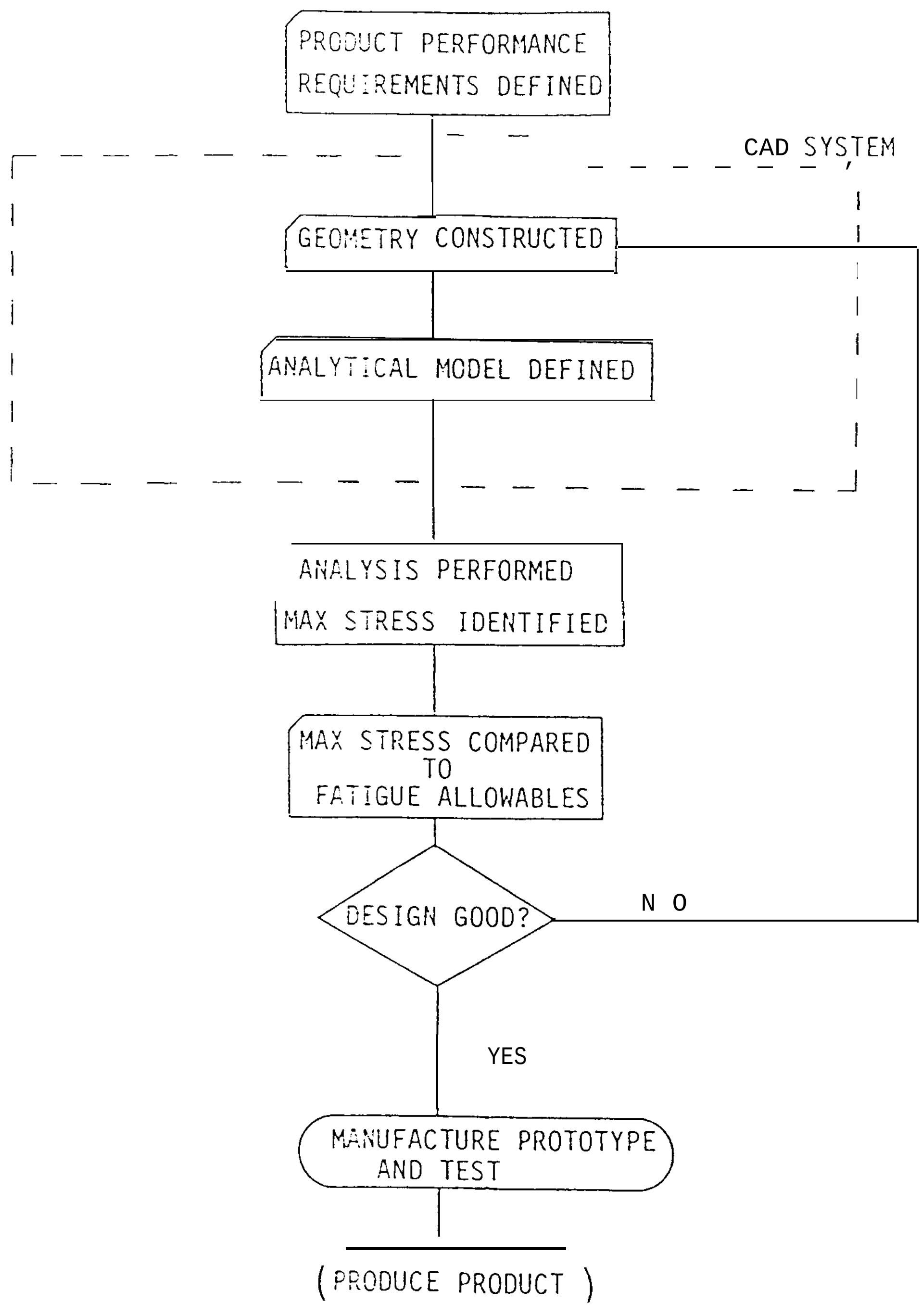


FIGURE 1. TYPICAL PRODUCT DESIGN CYCLE FLOW CHART

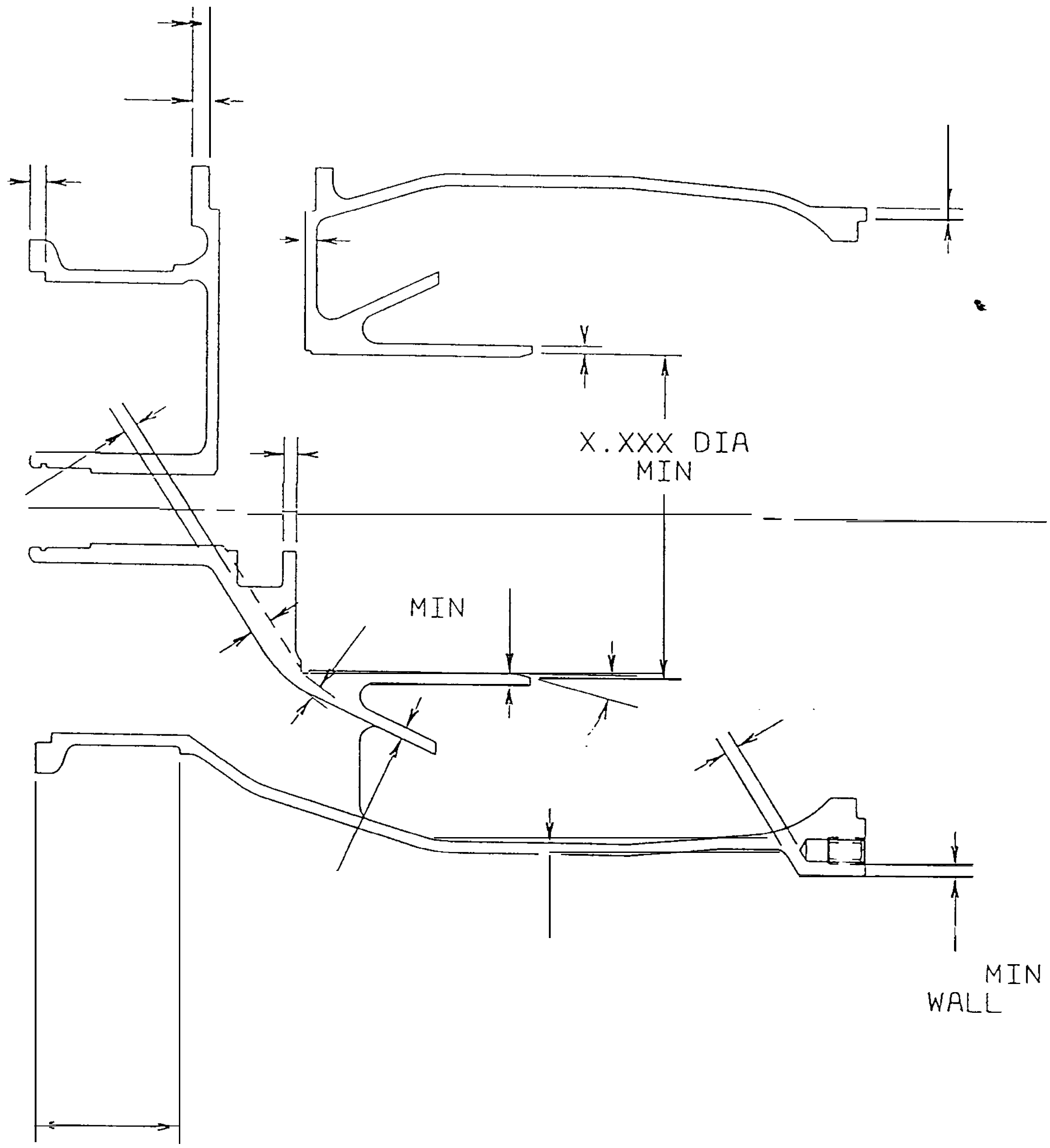


FIGURE 2. CAD REPRESENTATION

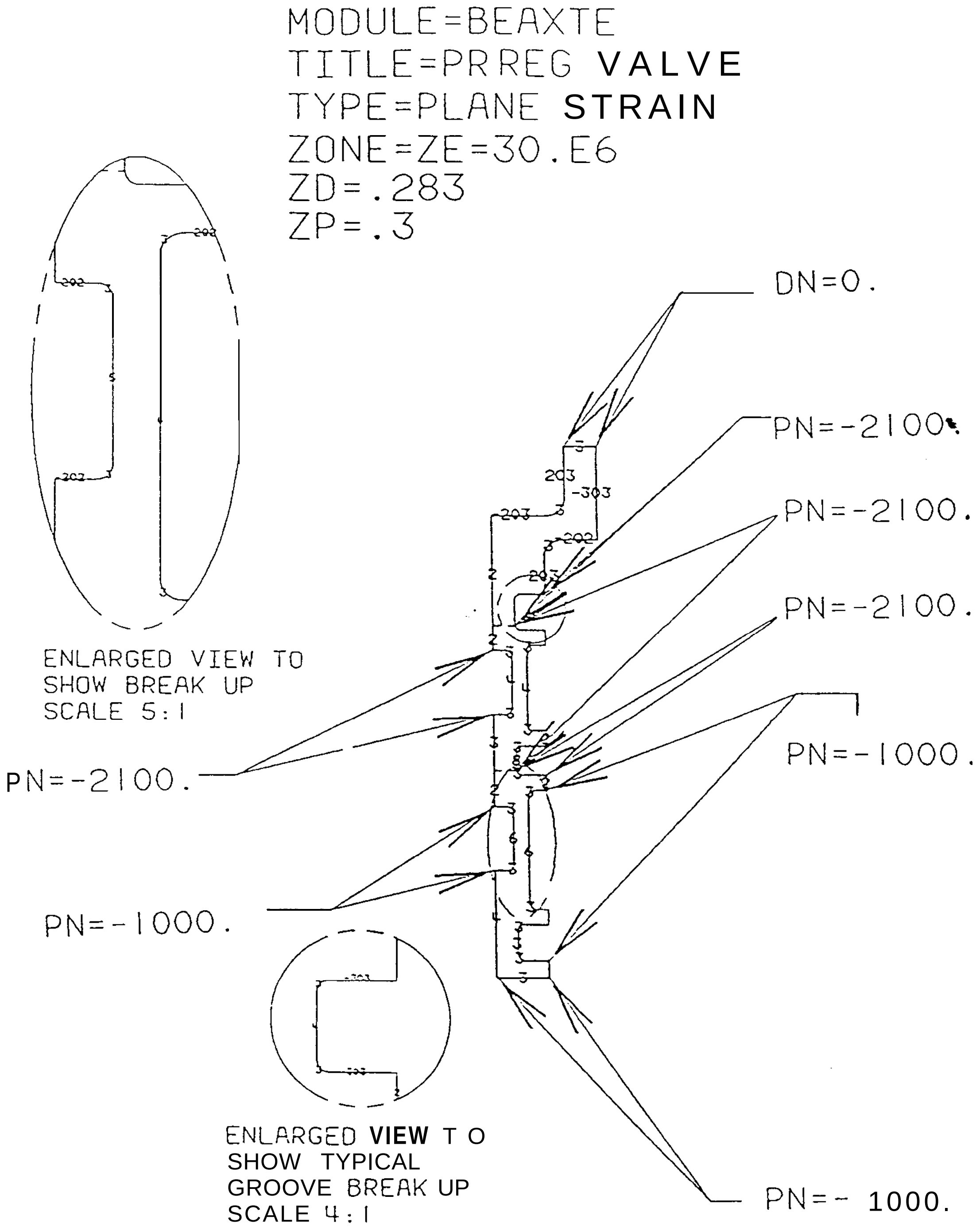


FIGURE 3. CADAM REPRESENTATION OF A BEASY MODEL

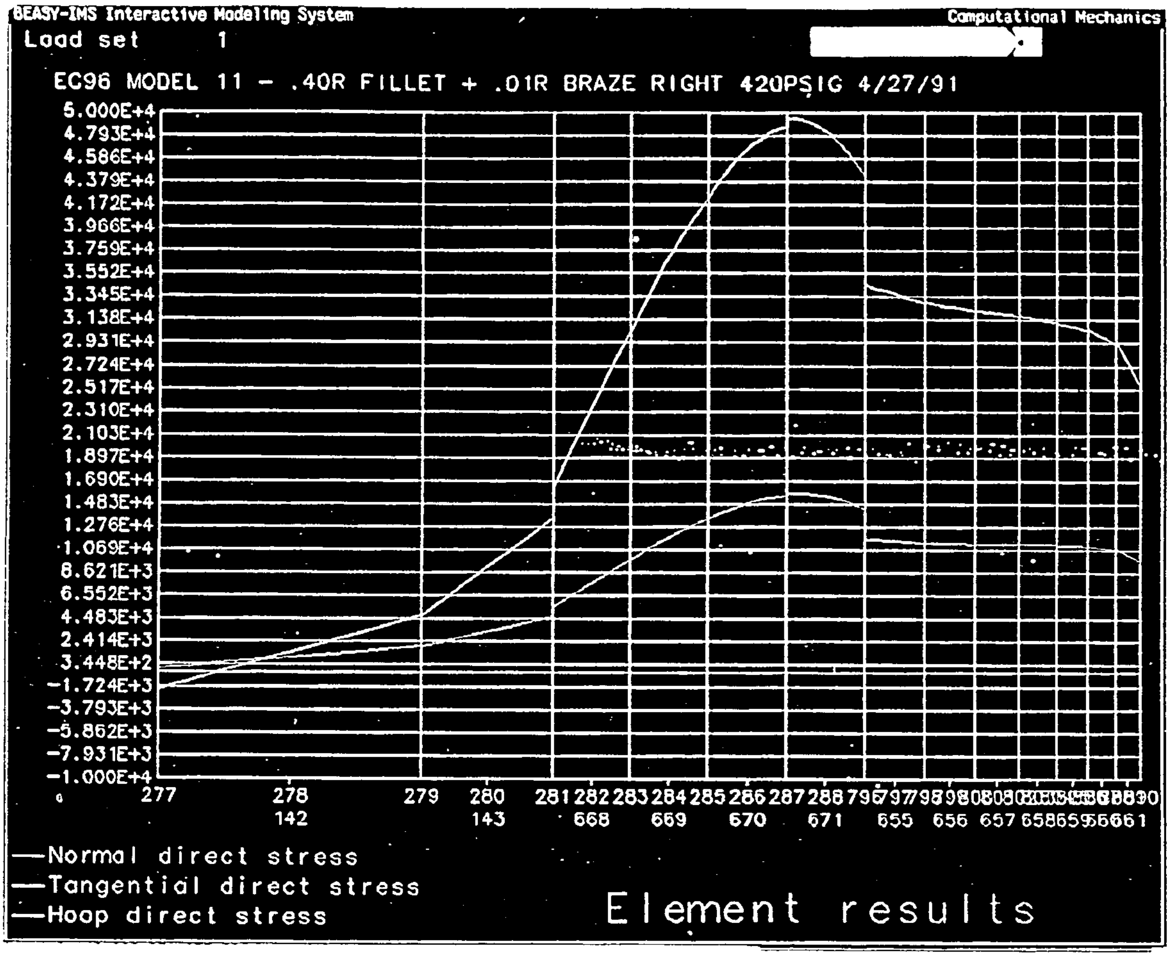


FIGURE 4. X-Y PLOT OF NORMAL STRESSES

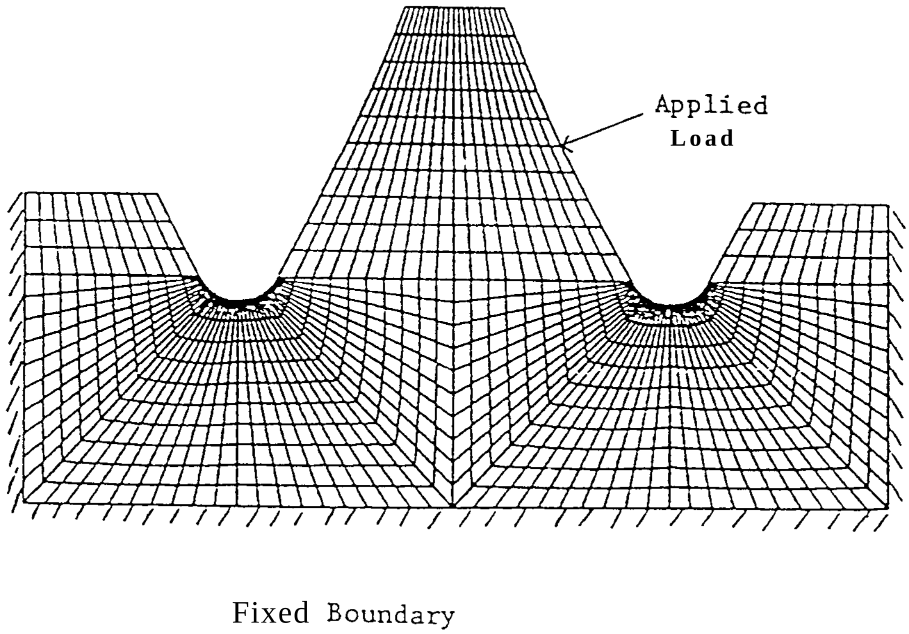


FIGURE 5. FINITE ELEMENT MODEL OF INTERNAL SPUR GEAR TOOTH

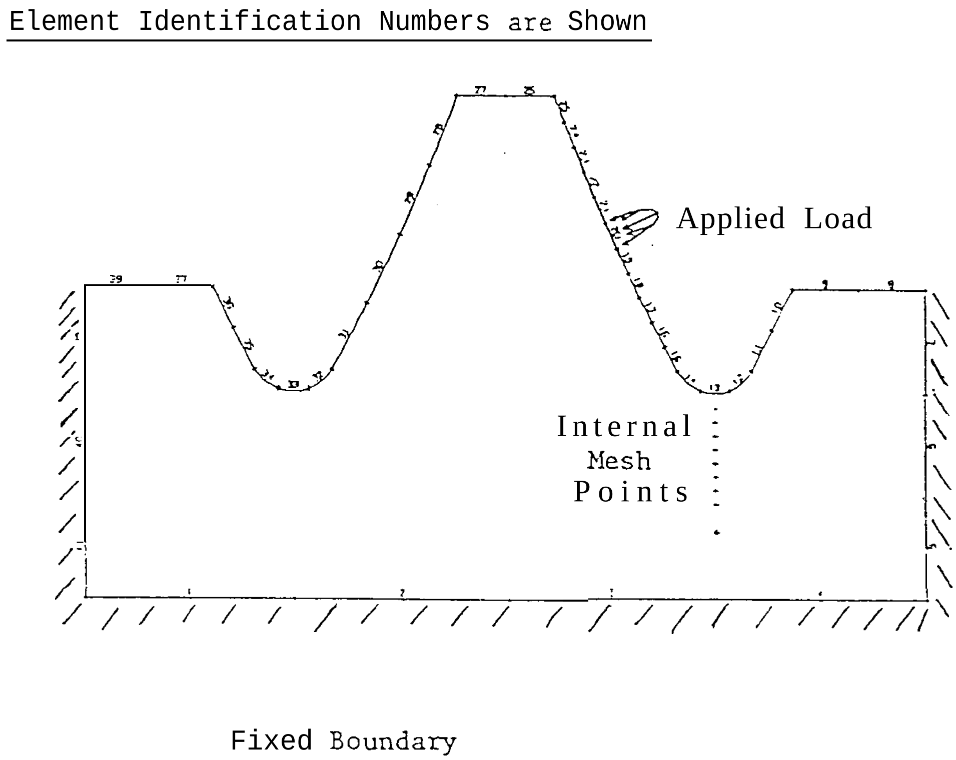


FIGURE 6. BOUNDARY ELEMENT MODEL OF INTERNAL SPUR GEAR TOOTH

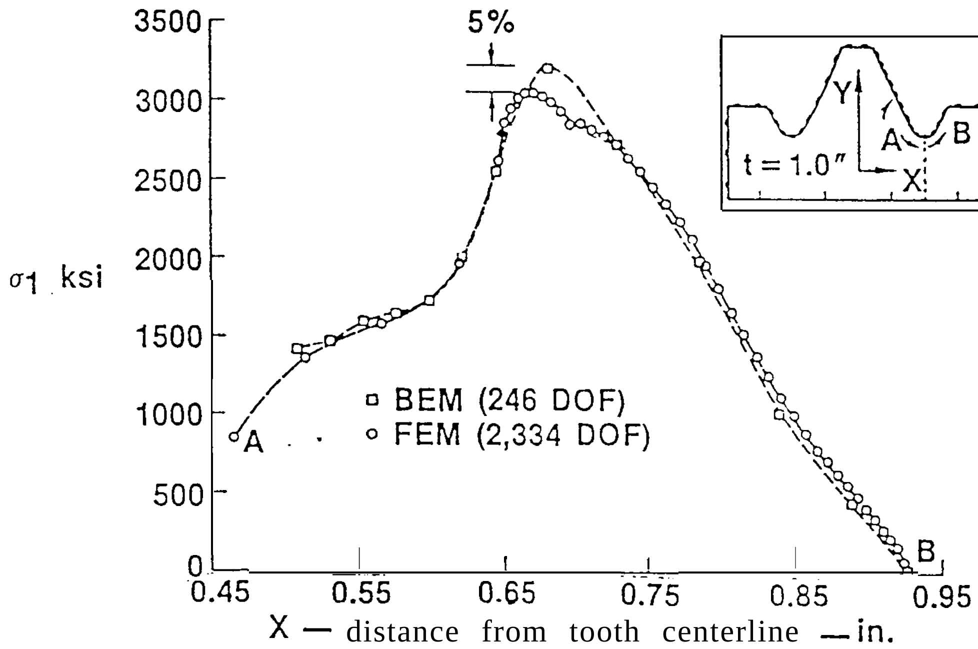


FIGURE 7. TOOTH FILLET SURFACE STRESS

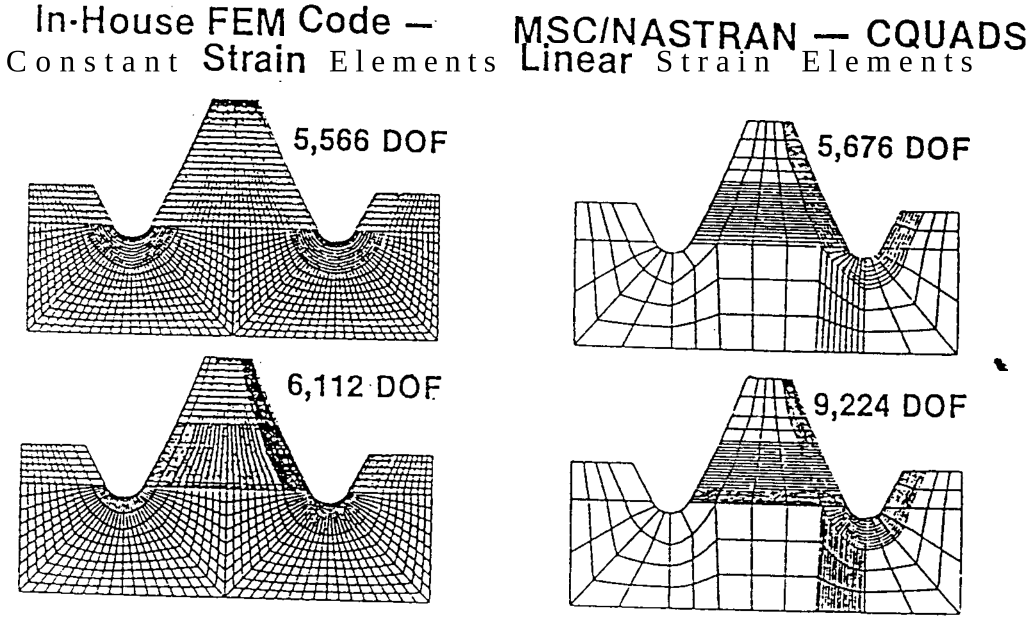


FIGURE 8. FINITE ELEMENT TOOTH MODELS

Code	Size (DOF)	CPU (sec)	σ max (psi)	Δ %
BEM-Beasy	246	19	3,210	-
	444	112	3,184	0.8
	528	164	3,193	0.3
FEM-in-house	2,334	18	3,053	
	5,566	67	3,374	10.5
	6,112	61	3,232	4.2
FEM-MSC/ Nastran	5,676	164	3,260	
	9,224	280	3,299	1.2

FIGURE 9. MAXIMUM STRESS CONVERGENCE FOR TOOTH MODEL

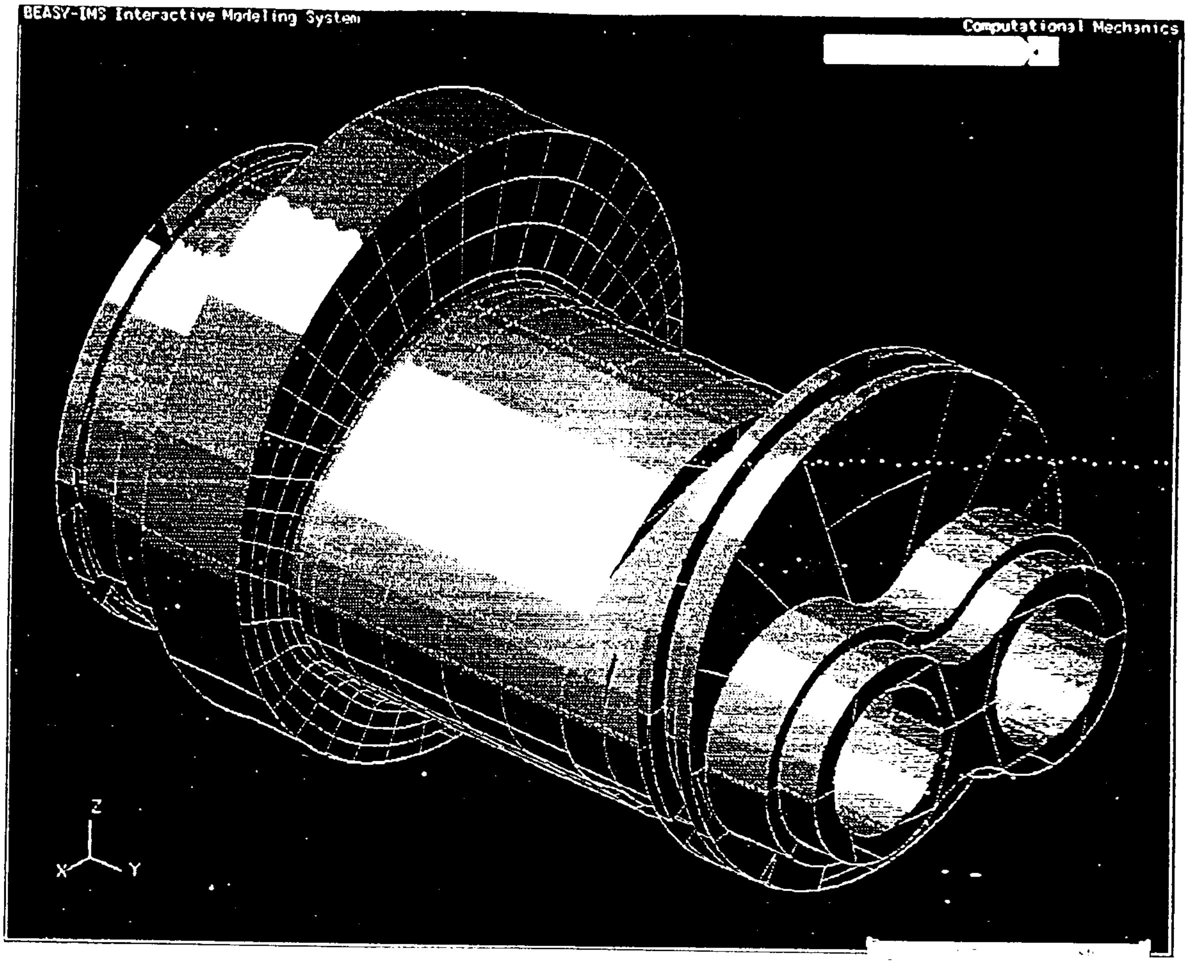


FIGURE 10. 3-D BOUNDARY ELEMENT ILLUSTRATION OF ELEMENT DISCONTINUITY

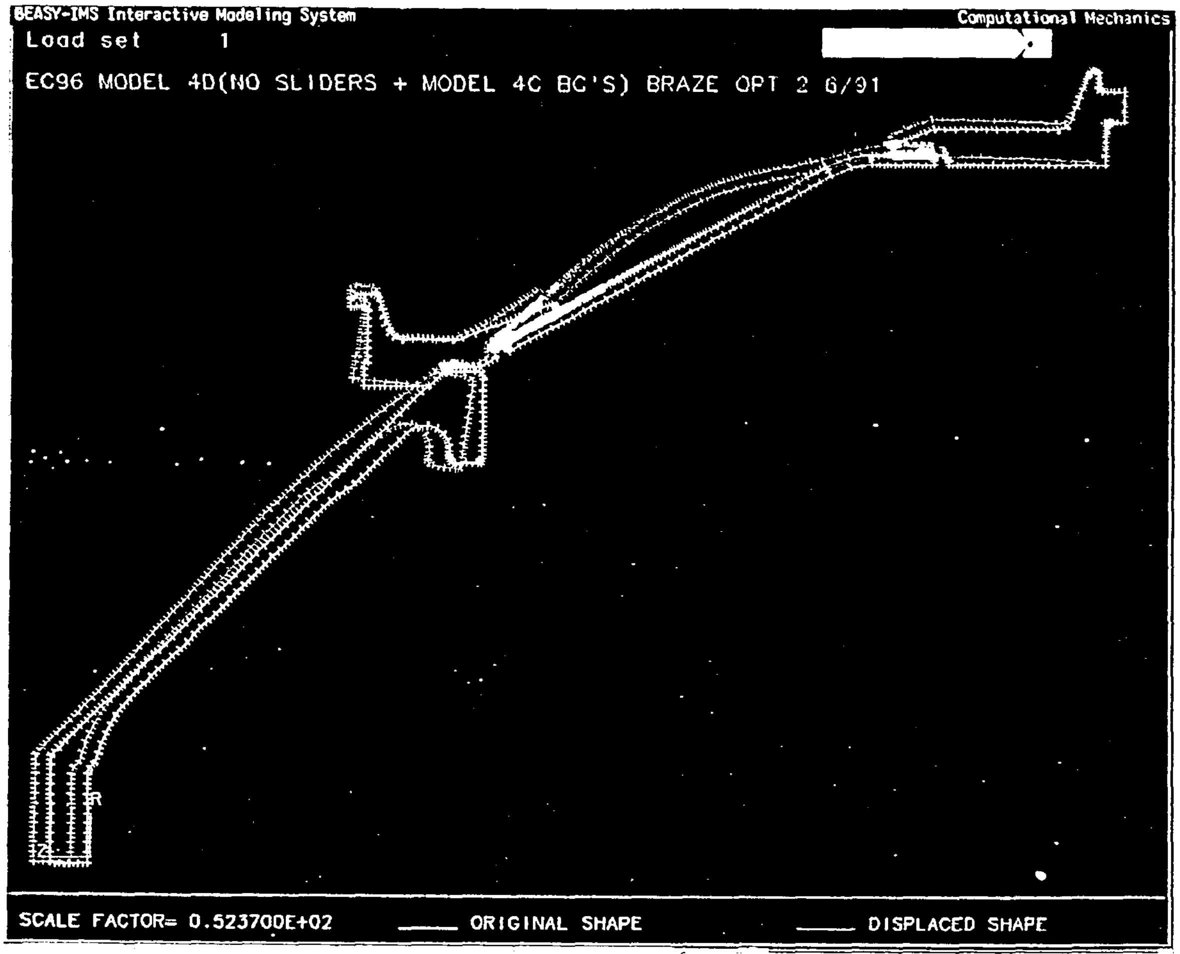


FIGURE 11. COMPARISON OF DEFORMED SHAPE VERSUS UNDEFORMED SHAPE

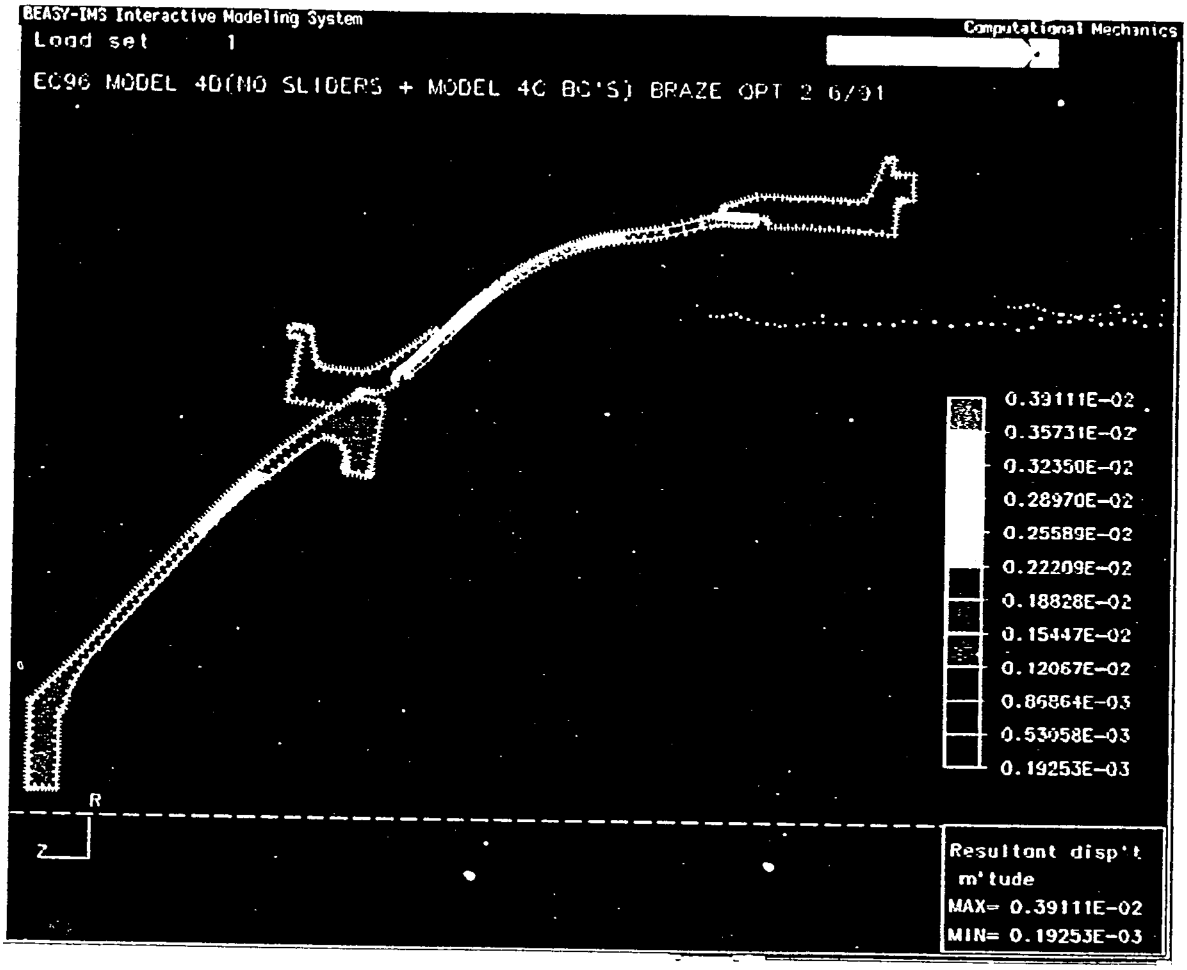


FIGURE 12. CONTOUR PLOT OF RESULTANT DEFORMATIONS ON AN AXISYMMETRIC CHECK VALVE BODY

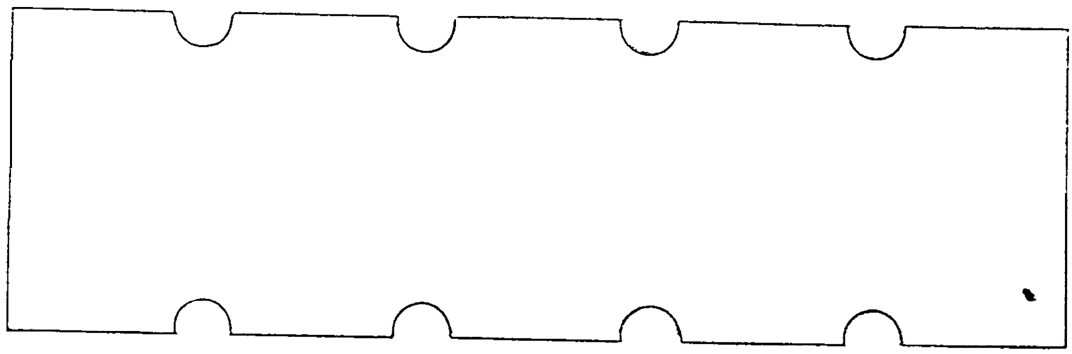


FIGURE 13A. TENSION BAR WITH FINITE ROWS OF EMI-CIRCULAR EDGE NOTCHES

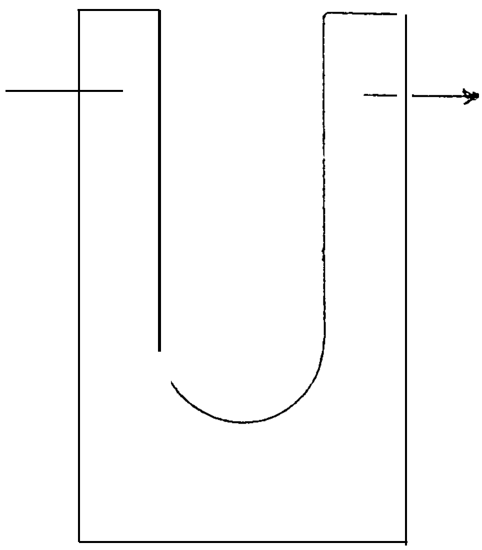


FIGURE 13B. U-SHAPED MEMBER SUBJECTED TO A SPREADING LOAD.

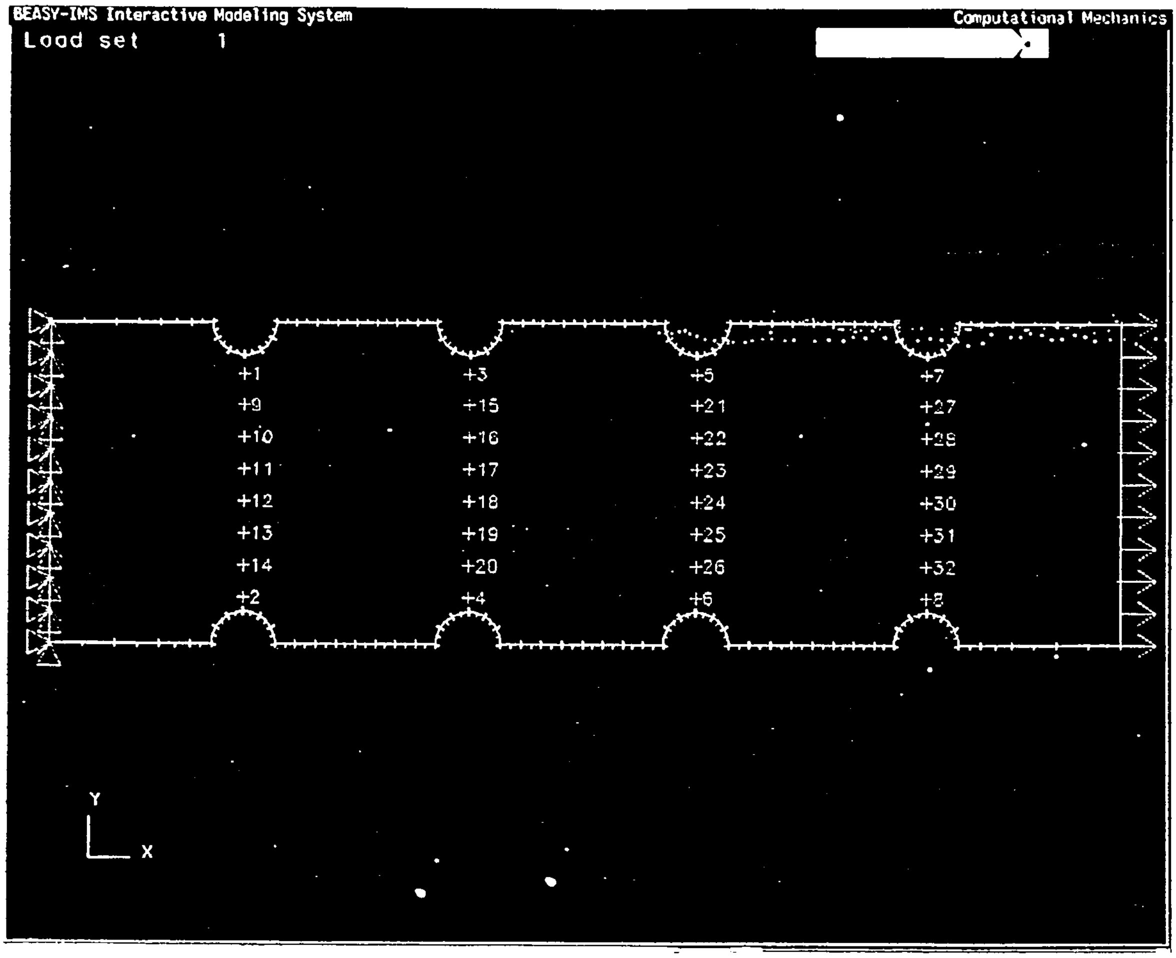
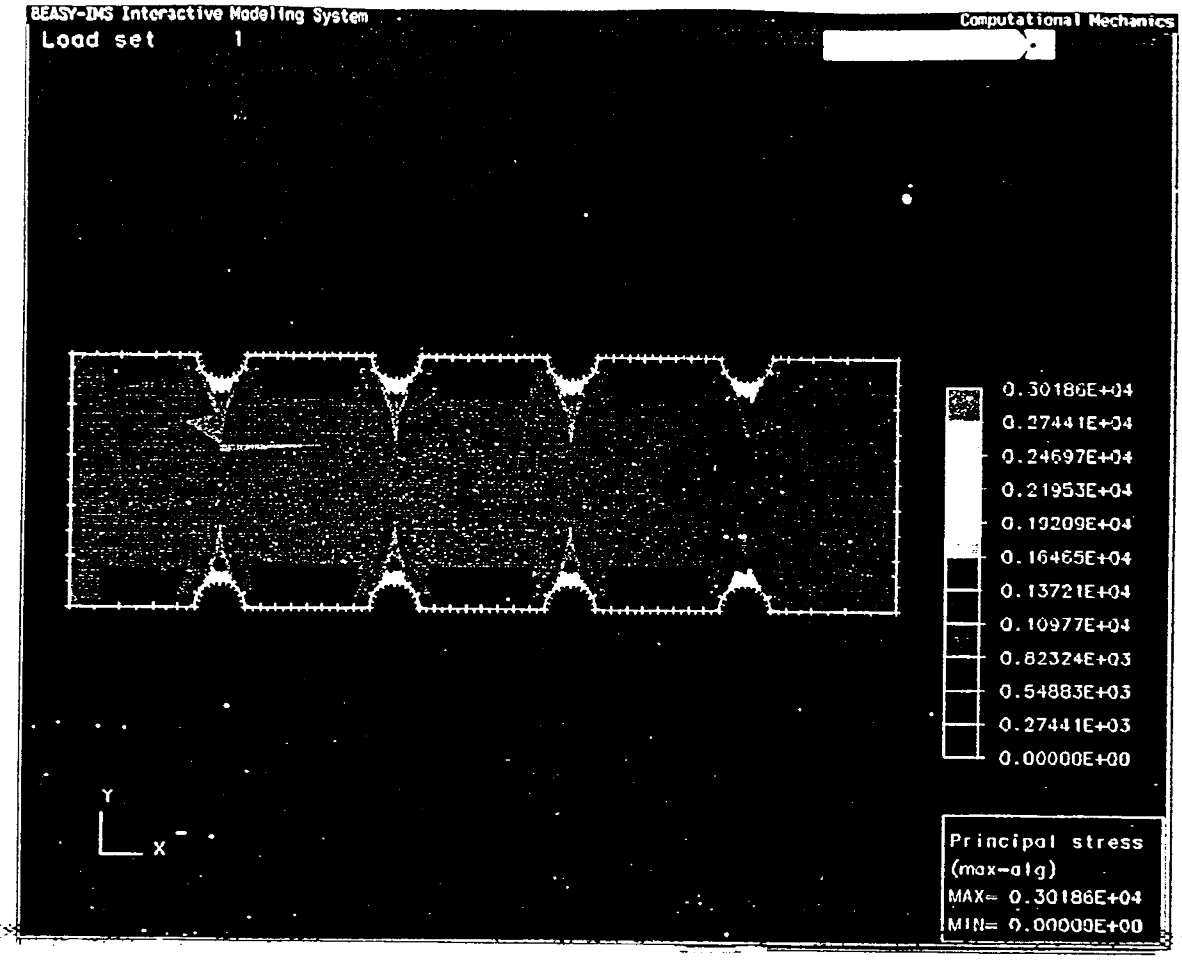


FIGURE 14. BOUNDARY ELEMENT MODEL OF TENSION BAR WITH SEMI-IFINITE SEMI-CIRCULAR EDGES SUBJECTED TO AXIAL LOAD



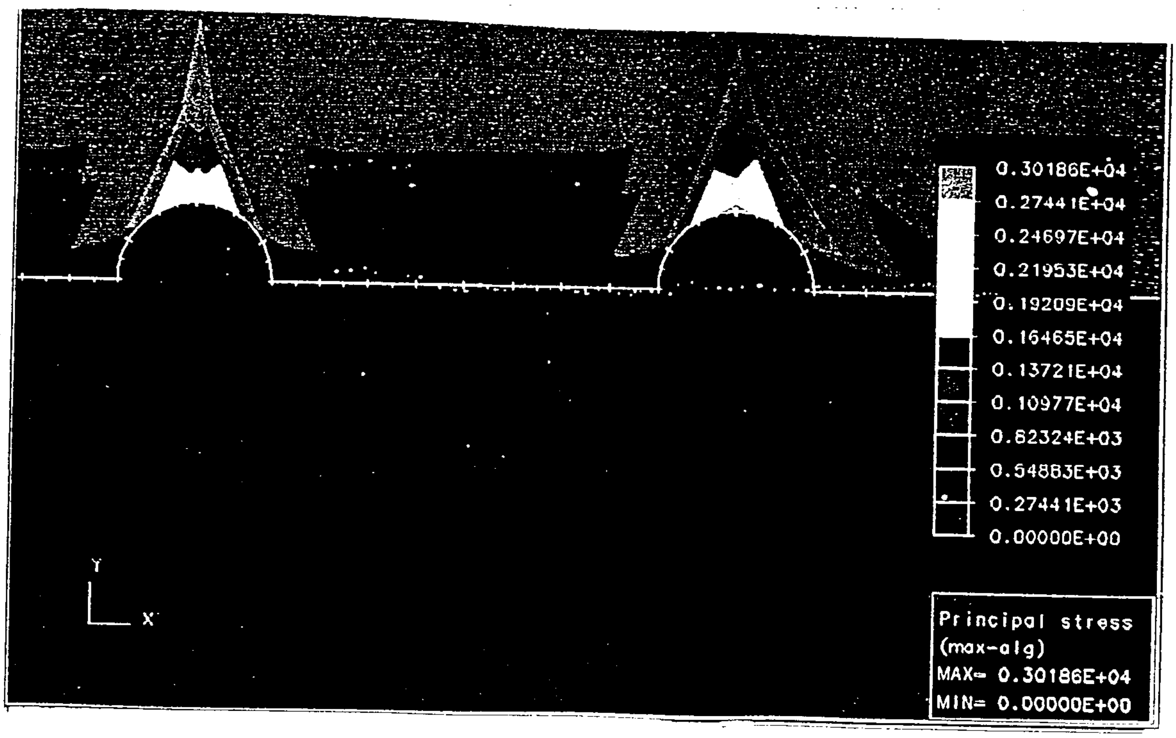
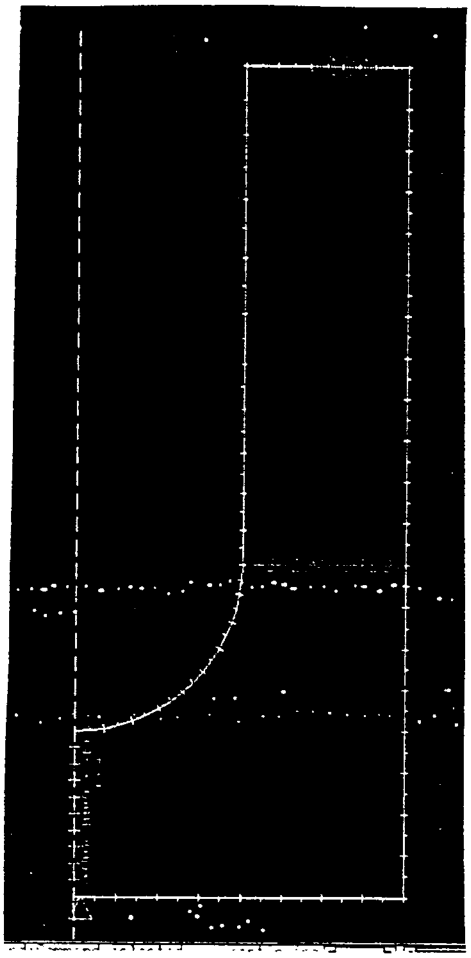


FIGURE 15 CONTOUR PLOT OF MAJOR PRINCIPAL STRESS DEPICTING MAXIMUM STRESS LOCATIONS



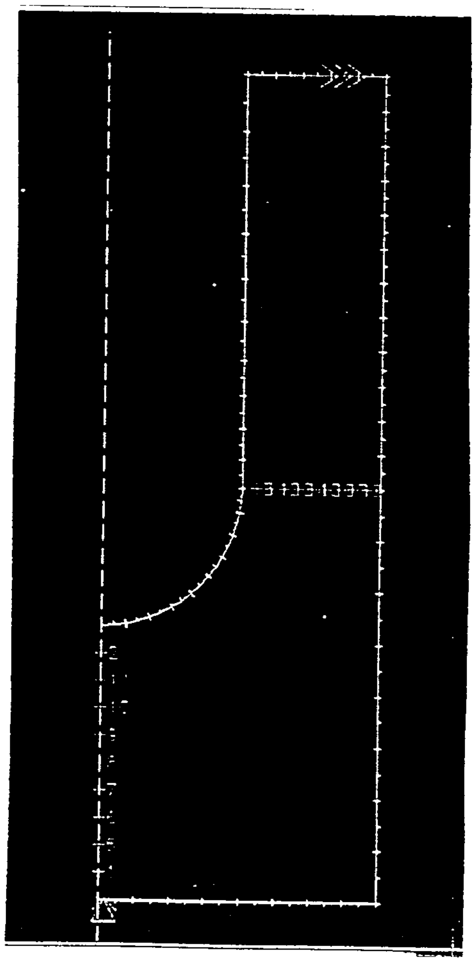
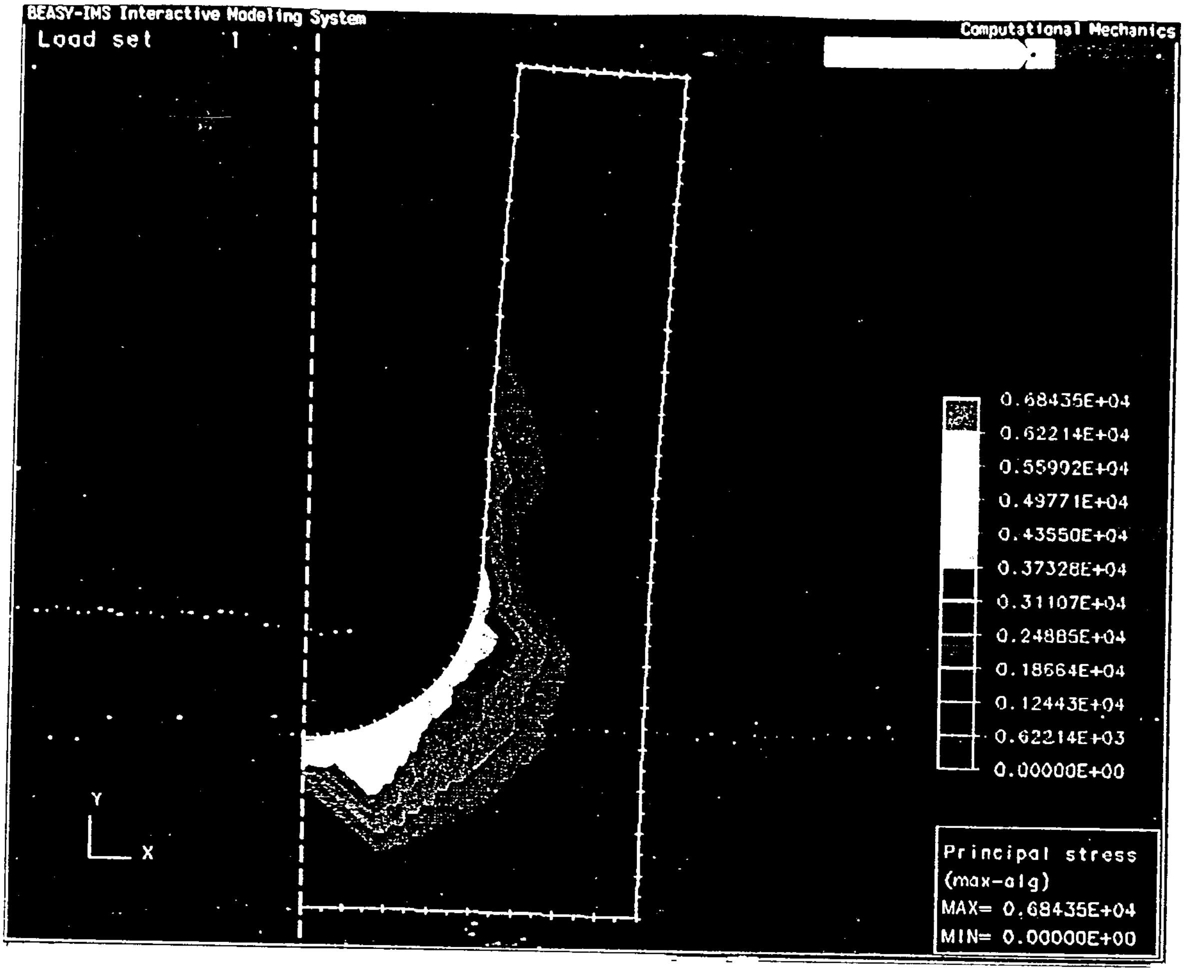


FIGURE 16. BOUNDARY ELEMENT MODELS OF U-SHAPED MEMBER SUBJECTED TO SPREADING LOAD



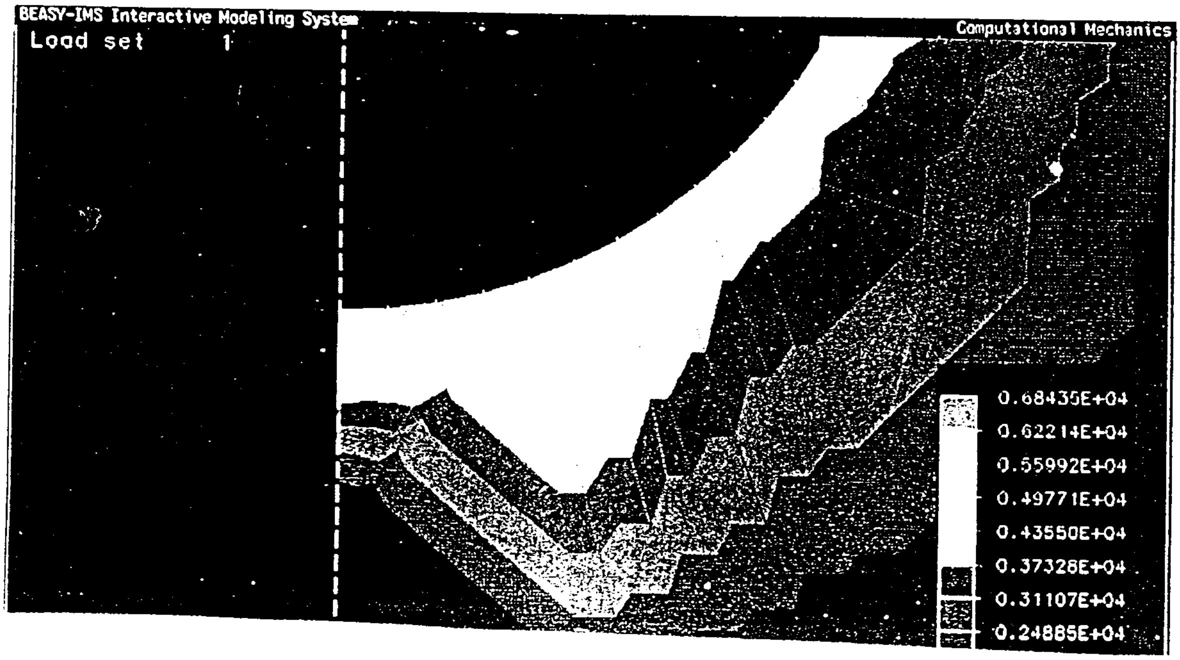


FIGURE 17. MODEL I CONTOUR PLOT OF MAJOR PRINCIPAL STRESS DEPICTING MAXIMUM STRESS LOCATION

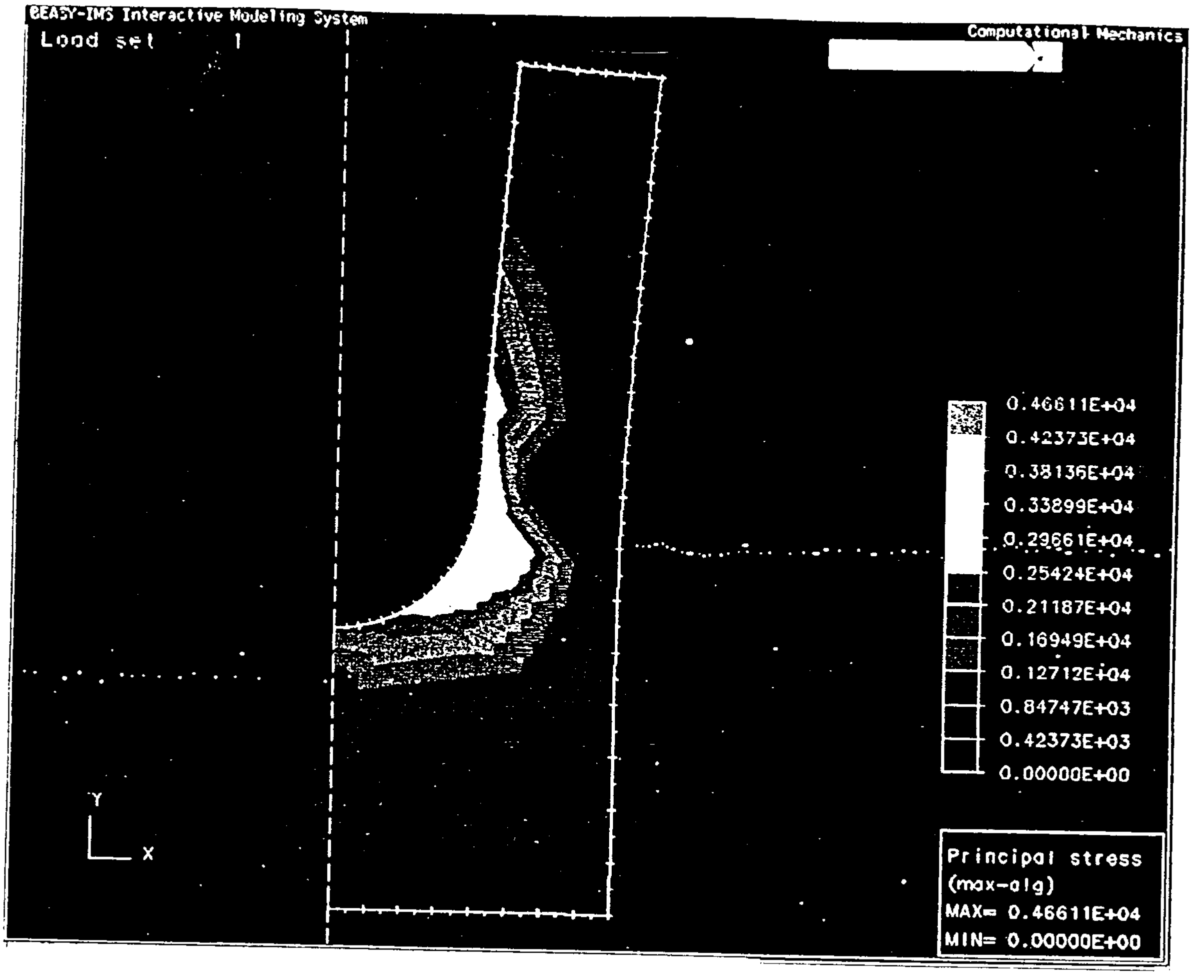
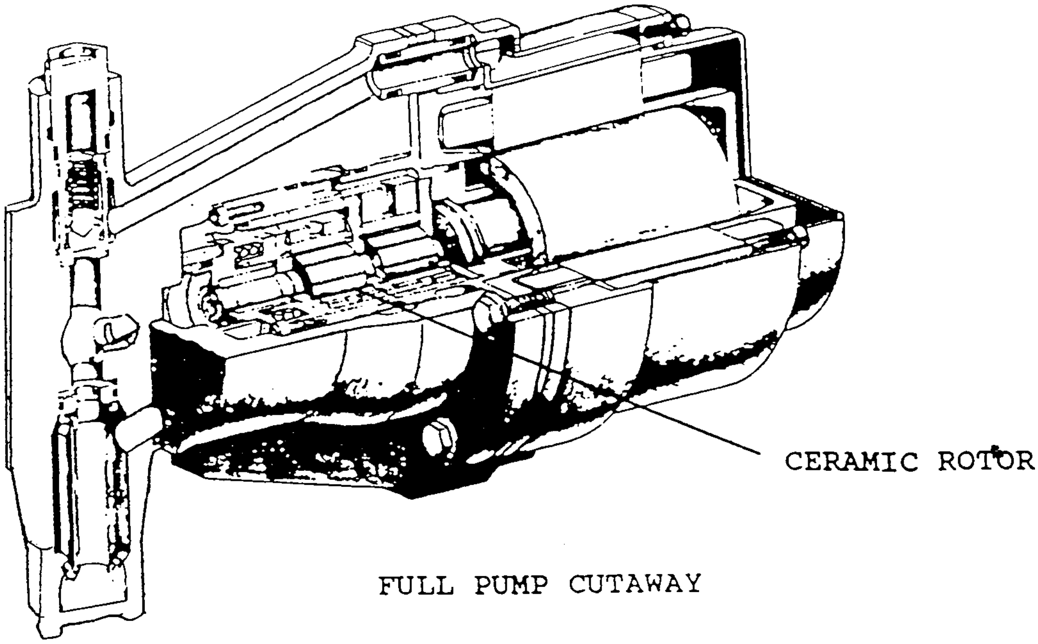
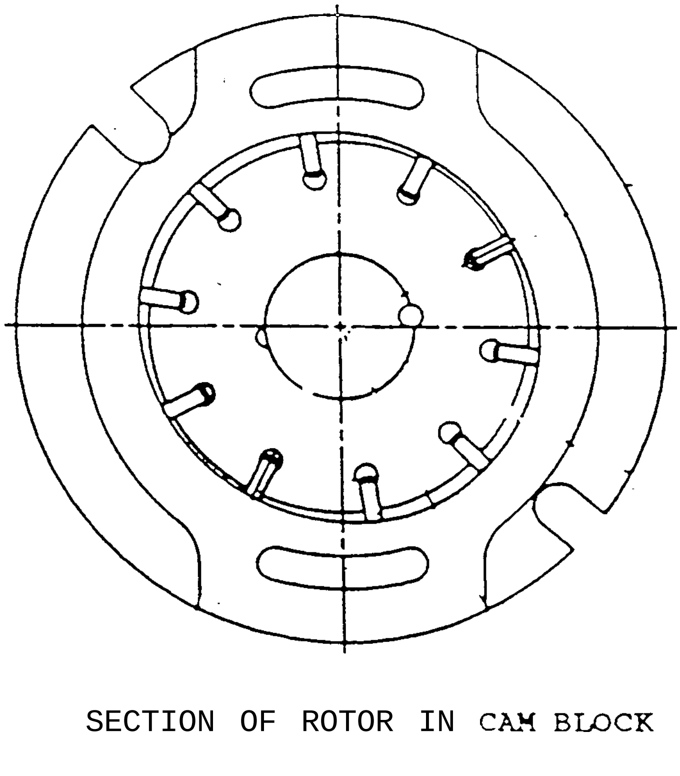


FIGURE 18. MODEL 2 CONTOUR PLOT OF MAJOR PRINCIPAL STRESS DEPICTING MAXIMUM STRESS LOCATION



FULL PUMP CUTAWAY



SECTION OF ROTOR IN CAM BLOCK

FIGURE 19. CERAMIC VANE ROTOR ILLUSTRATION

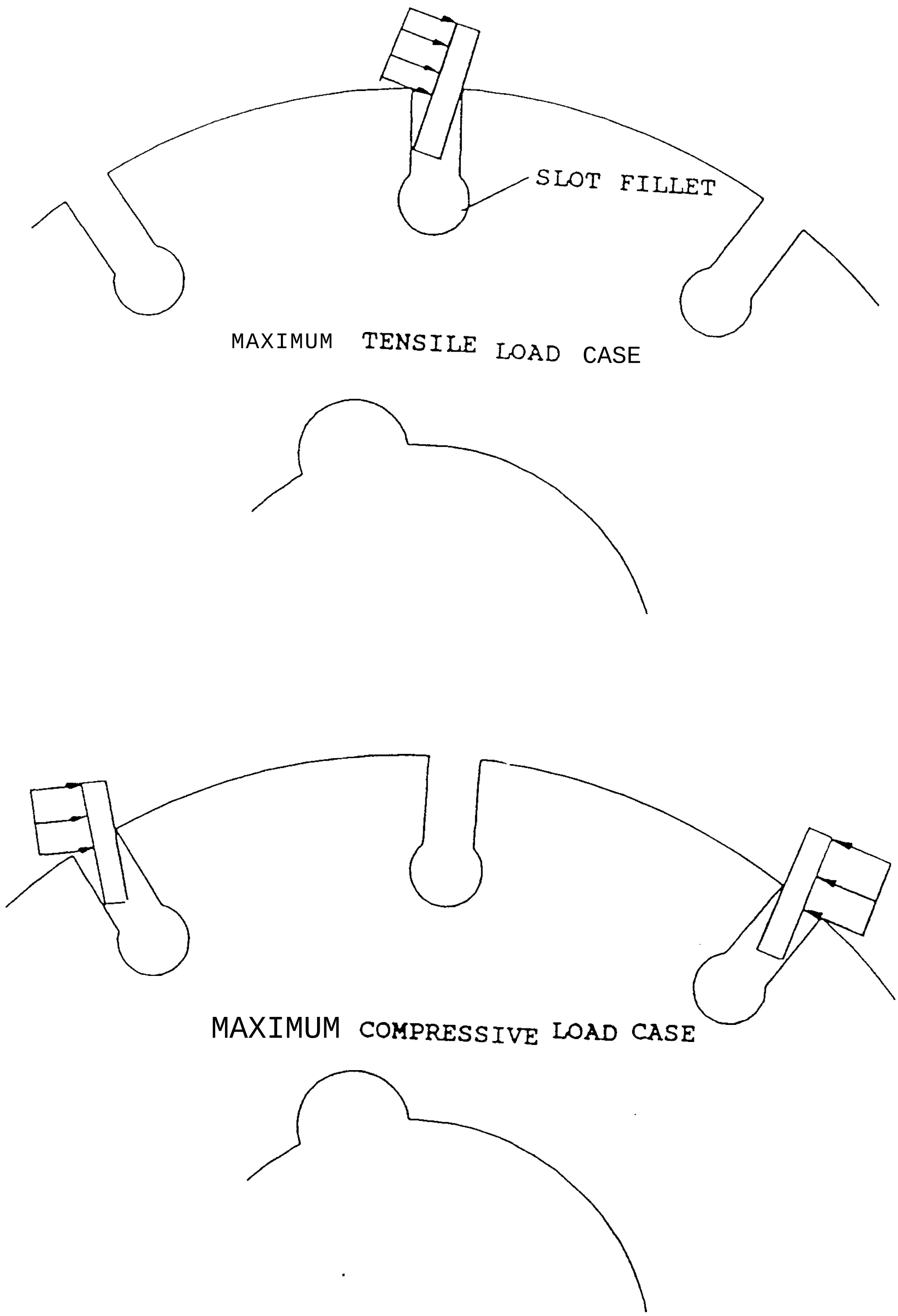


FIGURE 20. ILLUSTRATION OF LOADING CONDITIONS THAT PRODUCE MAXIMUM AND MINIMUM STRESSES IN KEYSLOT

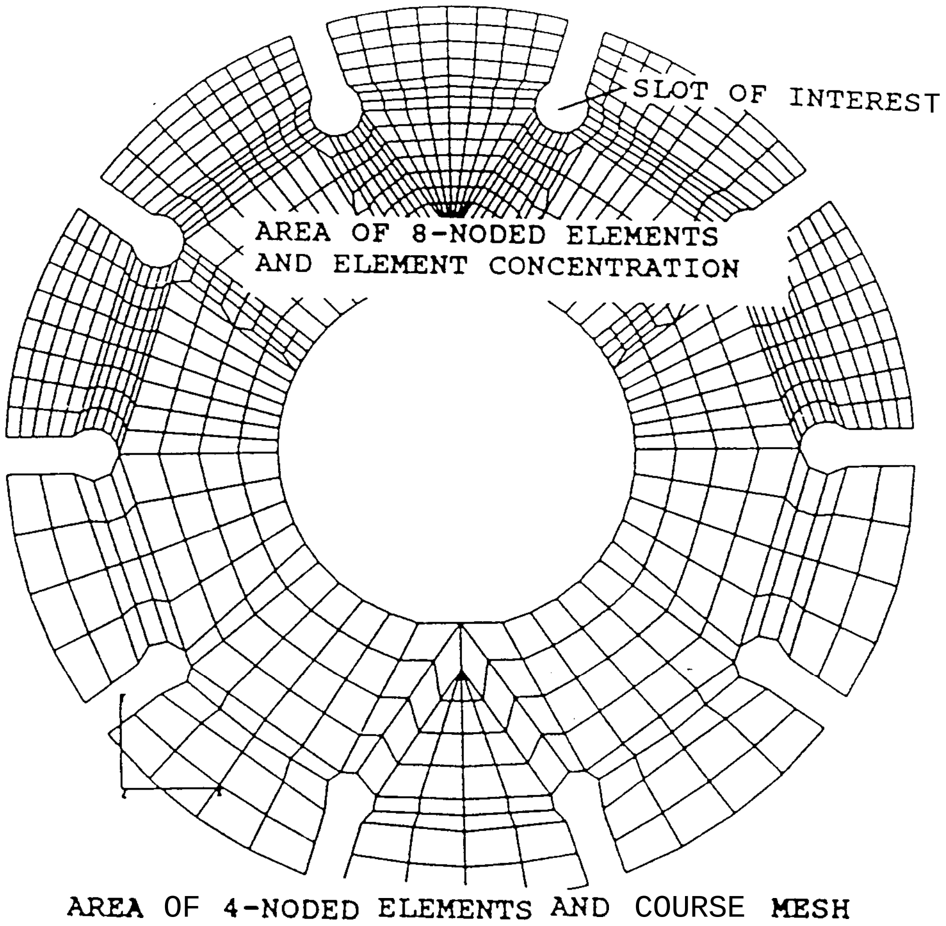


FIGURE 21. ORIGINAL FINITE ELEMENT MODEL

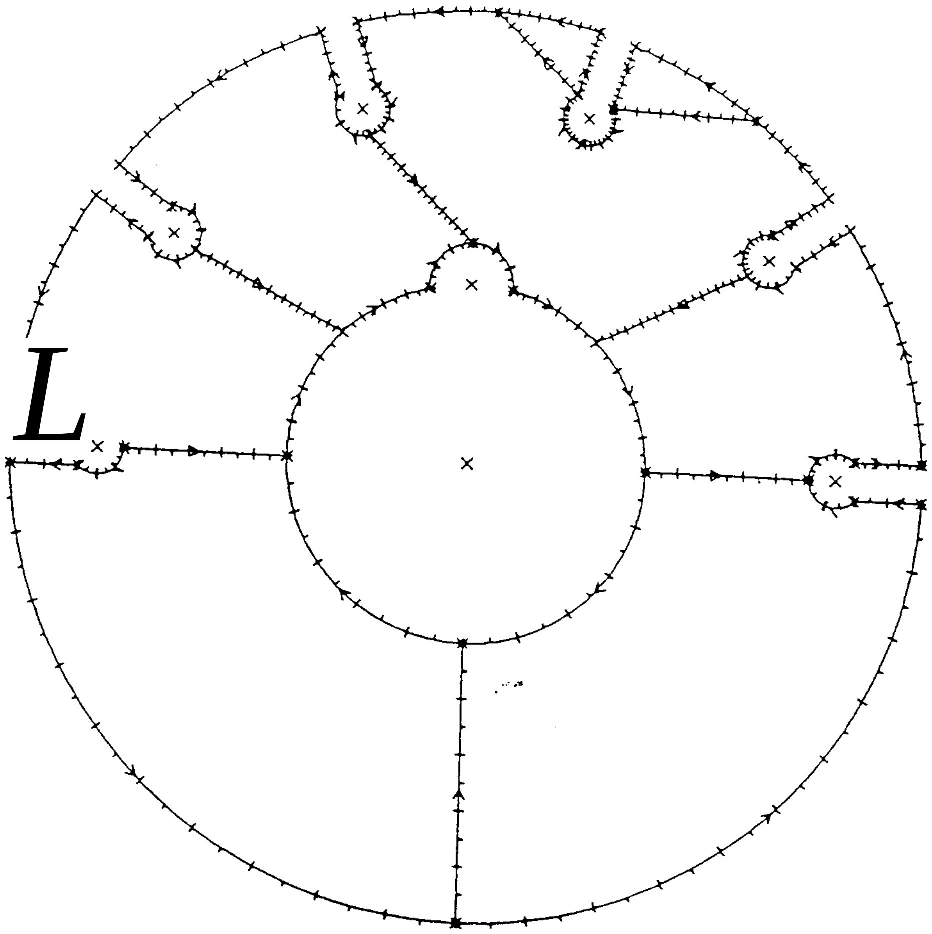


FIGURE 22. INITIAL BOUNDARY ELEMENT MODEL

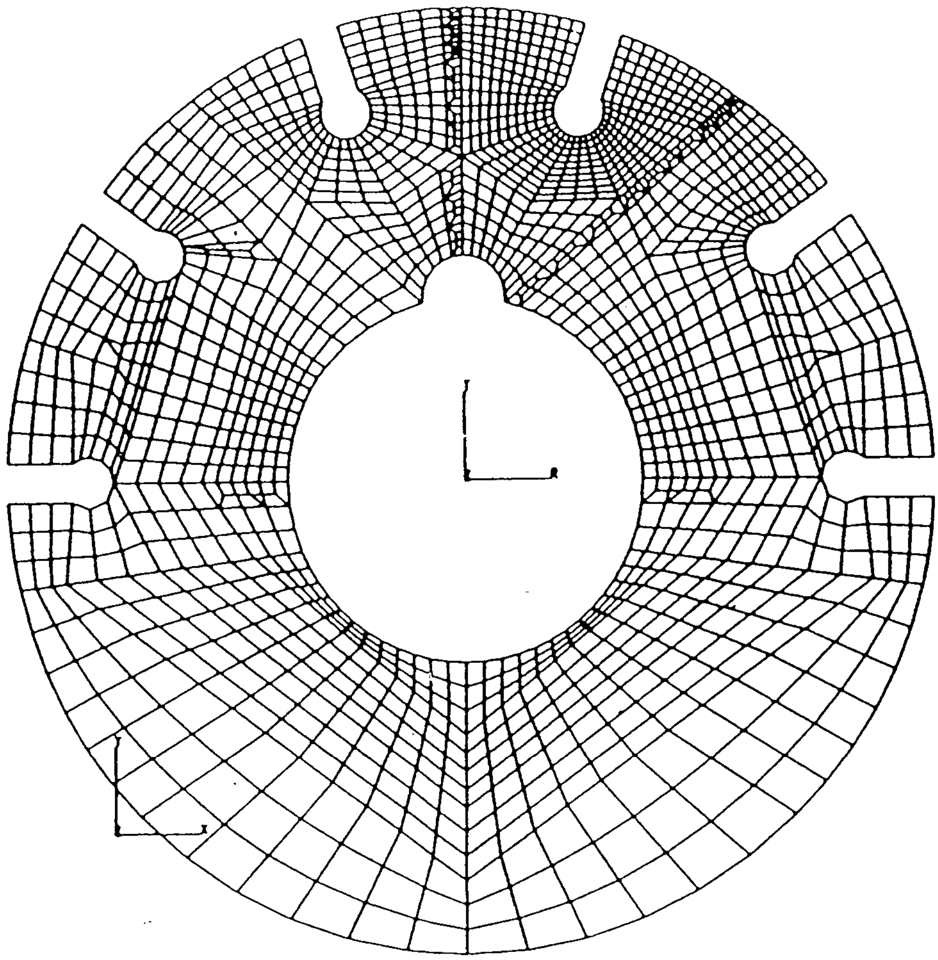


FIGURE 23. FINAL FINITE ELEMENT MESH NECESSARY TO OBTAIN SOLUTION CONVERGENCE

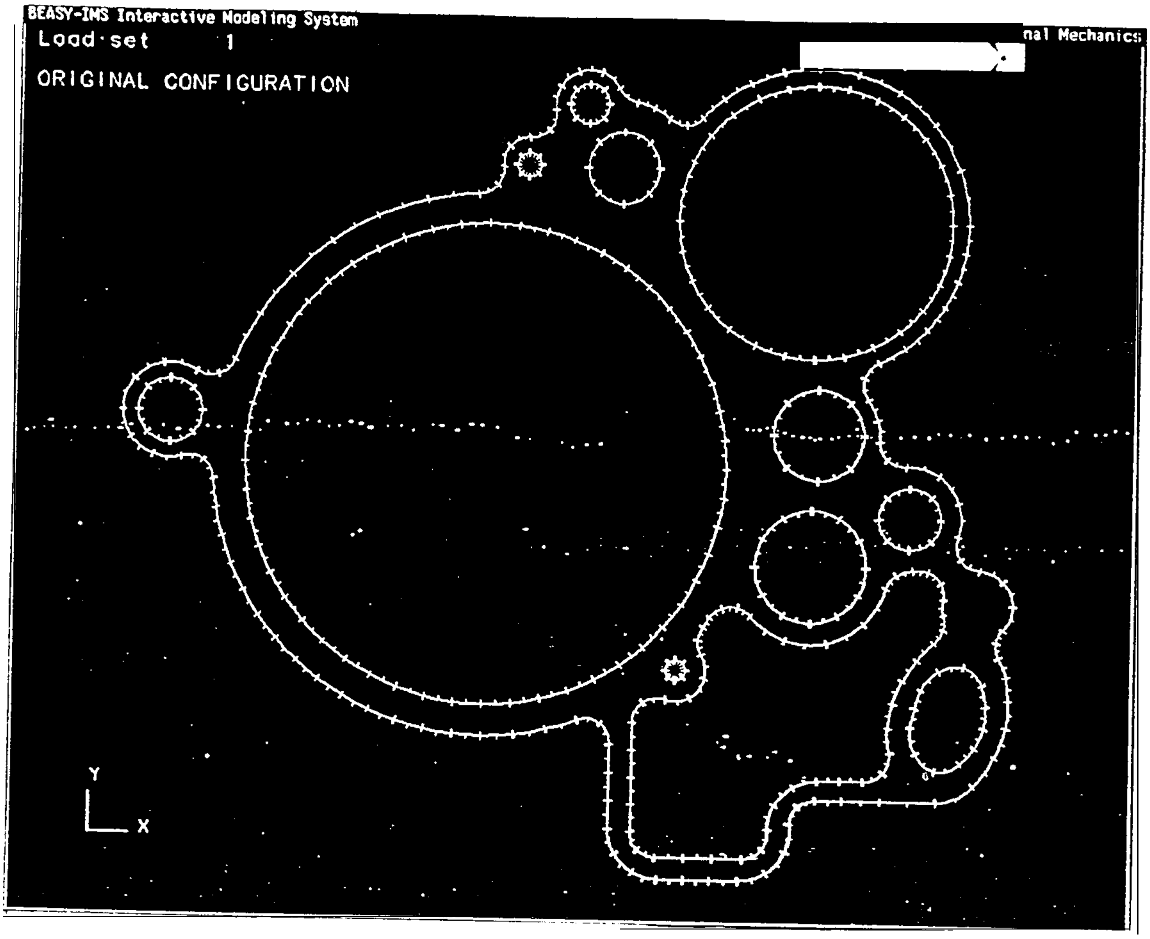


FIGURE 24. 2-D SECTIONBOUNDARY ELEMENT MODEL OF A PROPELLER CONTROL UNIT

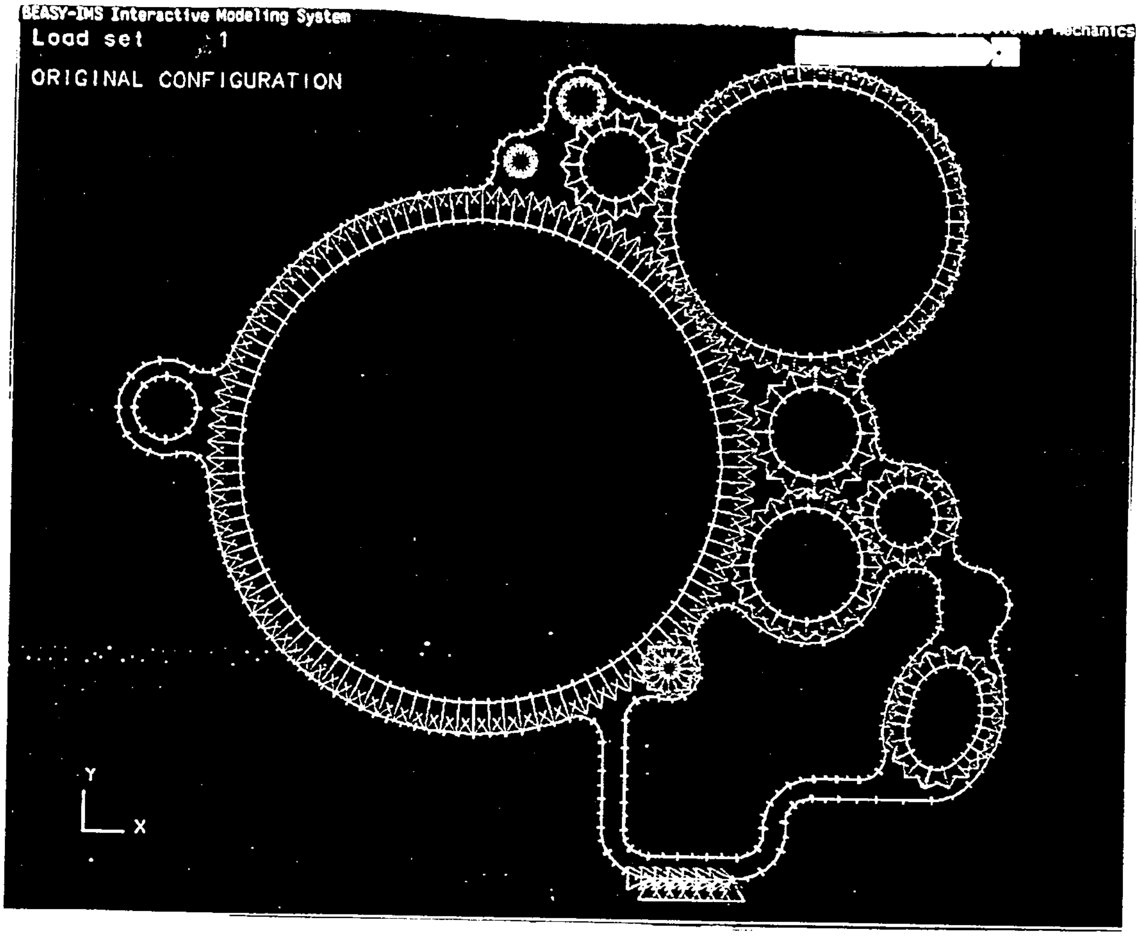


FIGURE 25. ILLUSTRATION OF IMPOSED BOUNDARY CONDITIONS ON THE ORIGINAL PCU SECTION

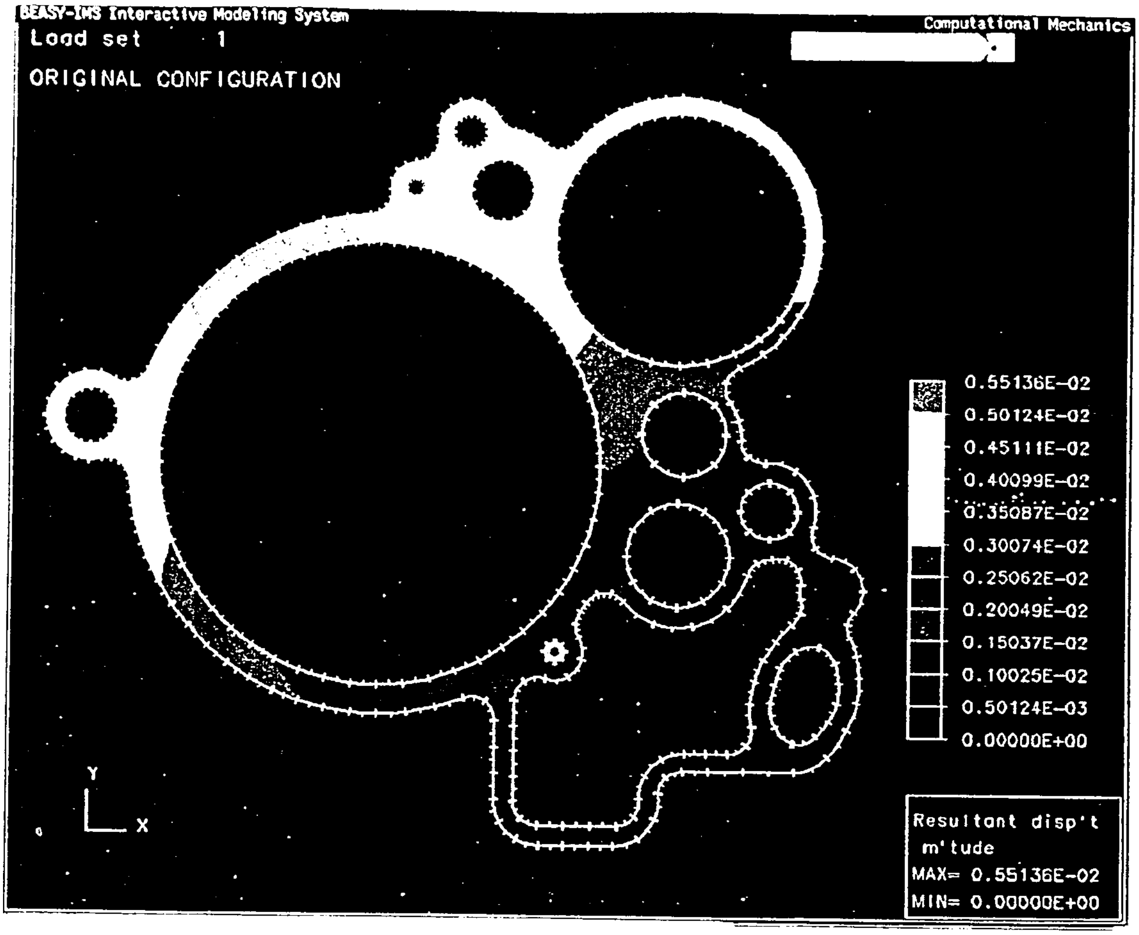


FIGURE 26. CONTOUR PLOT OF RESULTANT DISPLACEMENT DUE TO INTERNAL PRESSURE

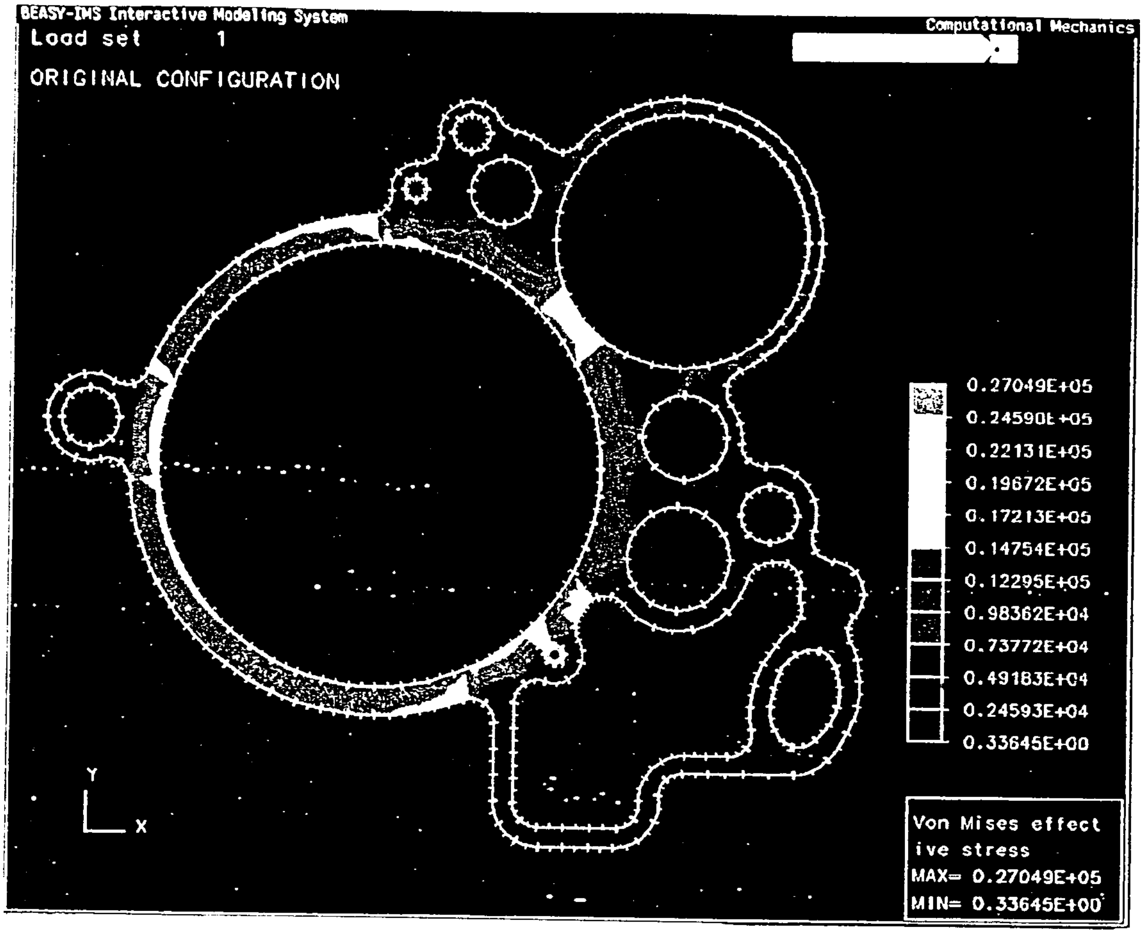


FIGURE 27. CONTOUR PLOT OF VON MISES STRESSES DUE TO INTERNAL PRESSURE

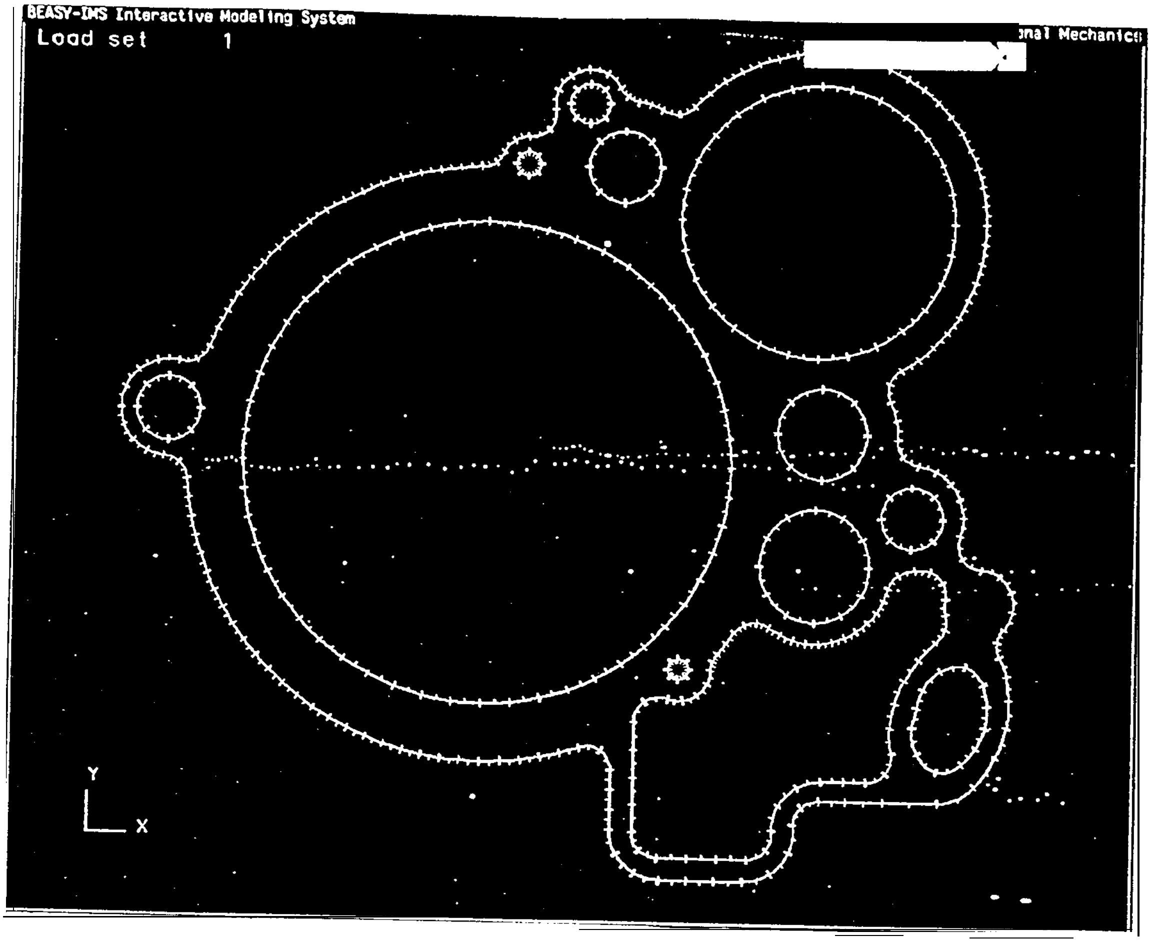


FIGURE 28. BOUNDARY ELEMENT MODEL OF INCREASED WALL THICKNESS PCU SECTION

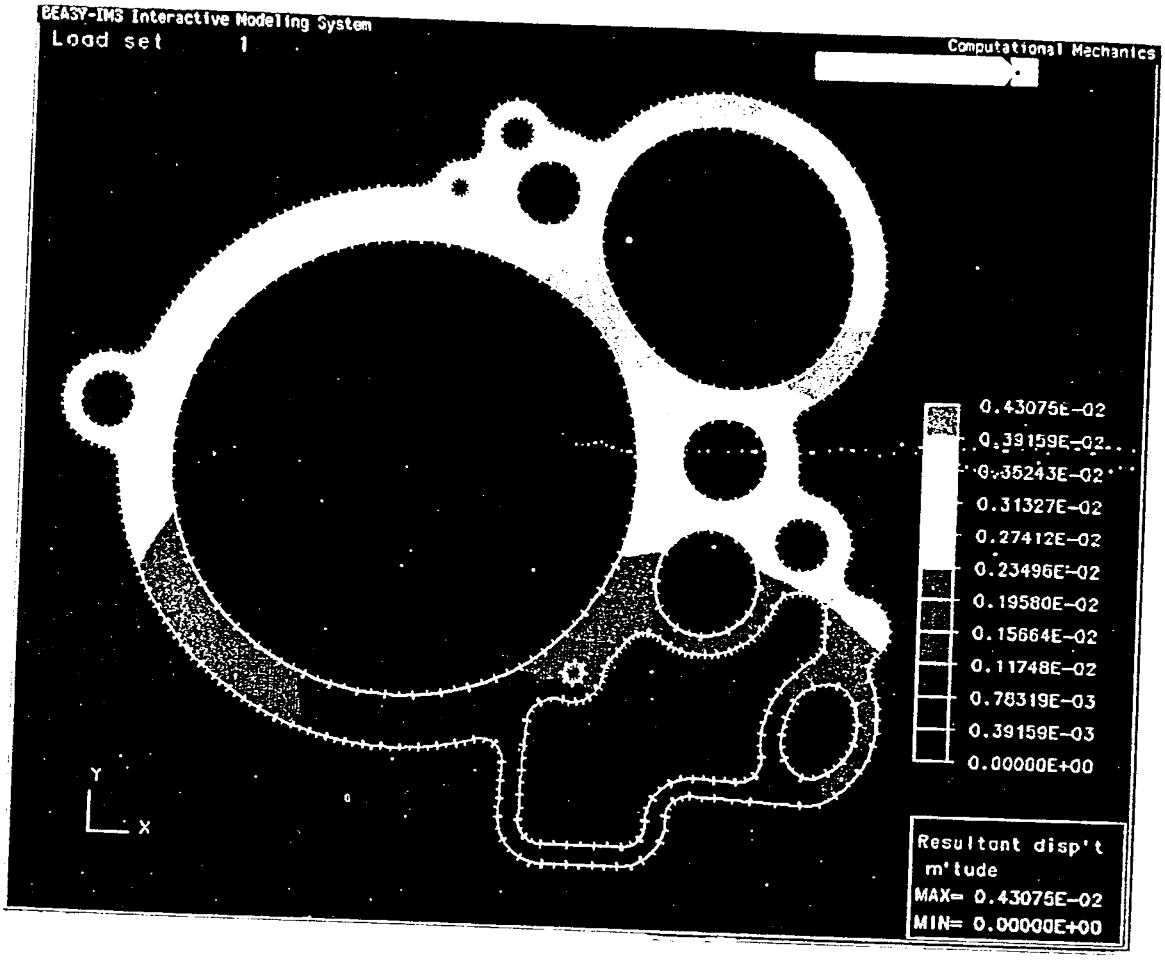


FIGURE 29. CONTOUR PLOT OF RESULTANT DISPLACEMENTS DUE TO INTERNAL PRESSURE ON THICKENED PCU SECTION

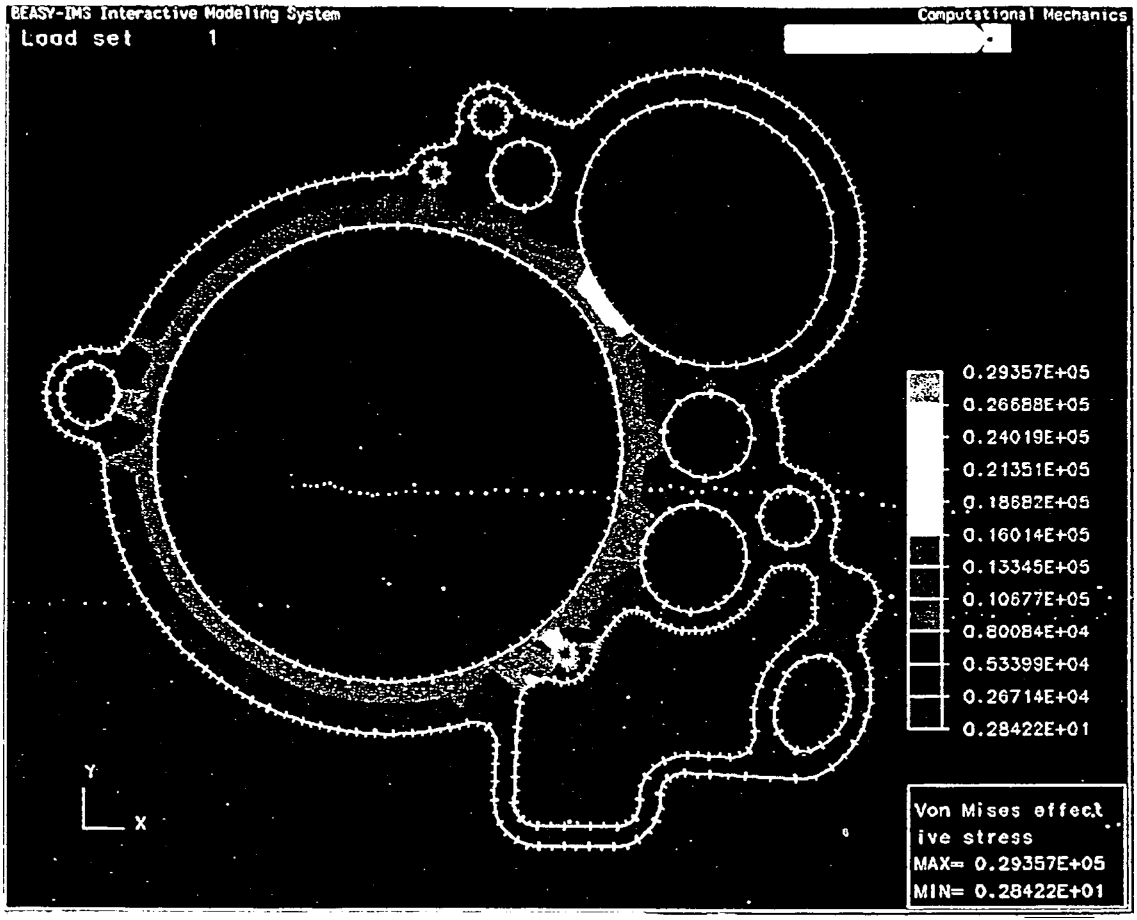


FIGURE 30. CONTOUR PLOT OF VON MISES STRESSES DUE TO INTERNAL PRESSURE ON THICKENED PCU SECTION

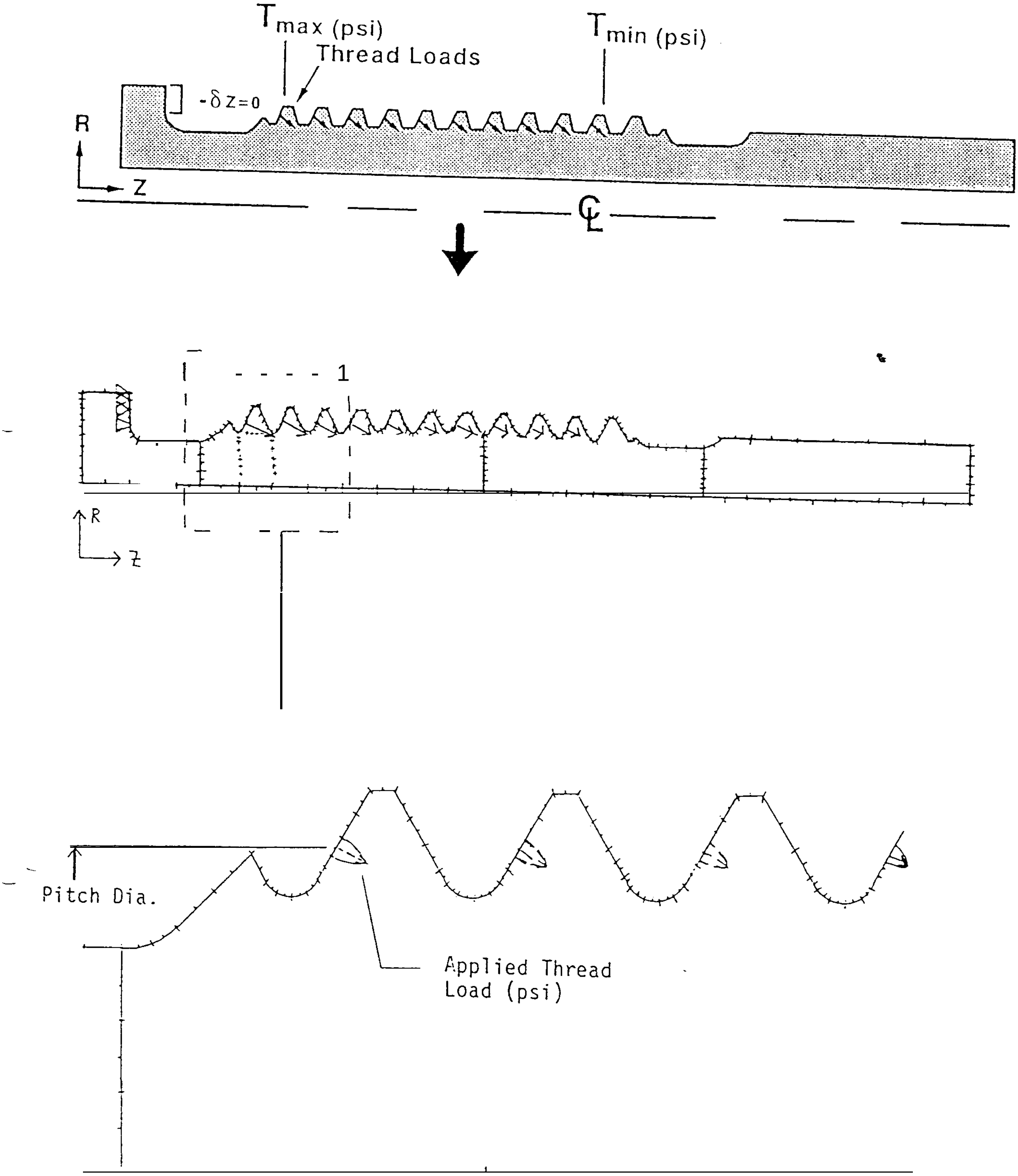


FIGURE 31. BOUNDARY ELEMENT MODEL USED TO DETERMINE THE MAXIMUM THREAD STRESS

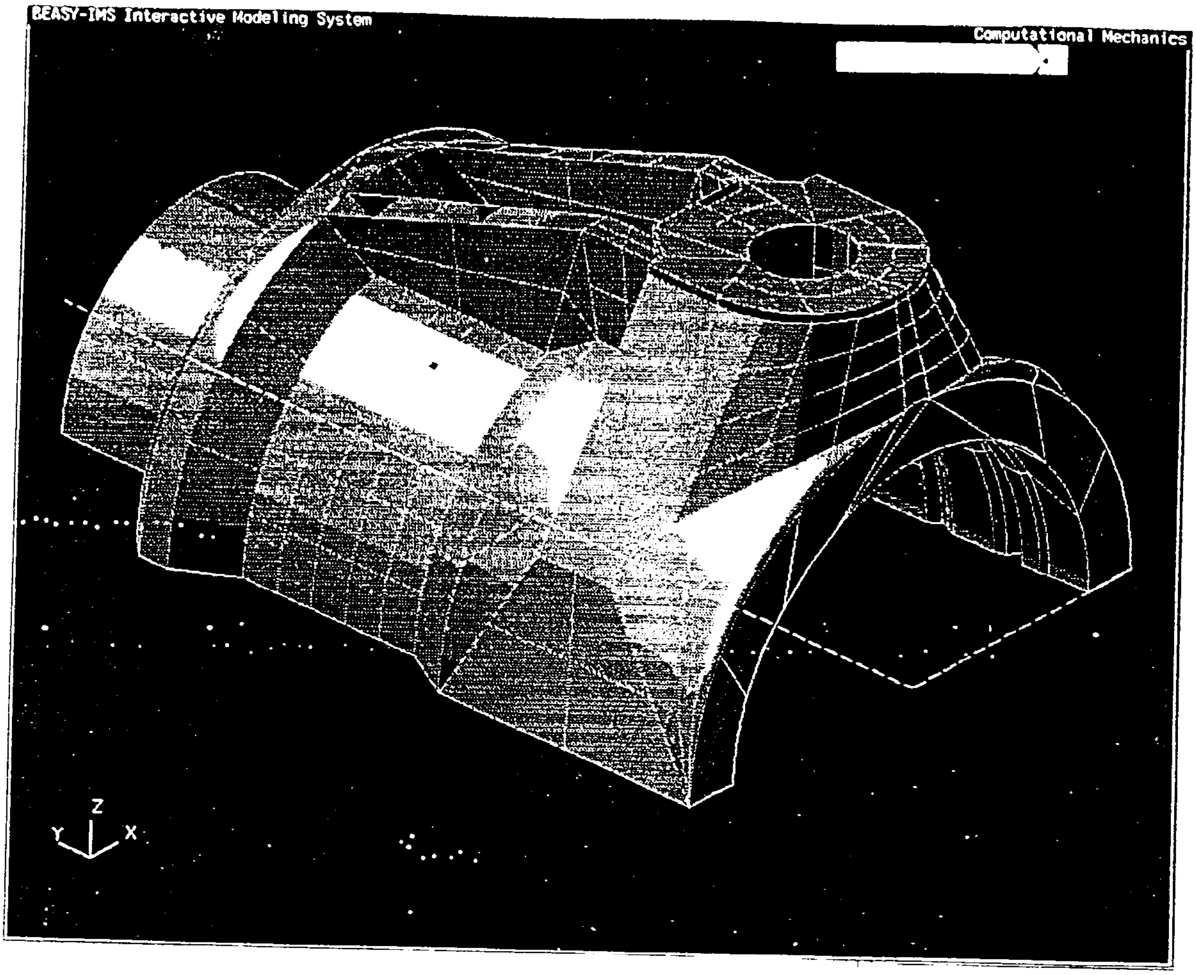


FIGURE 32. BOUNDARY ELEMENT MODEL OF A JET ENGINE FUEL CONTROL HOUSING SECTION WITH AN INSERTED BUSHING

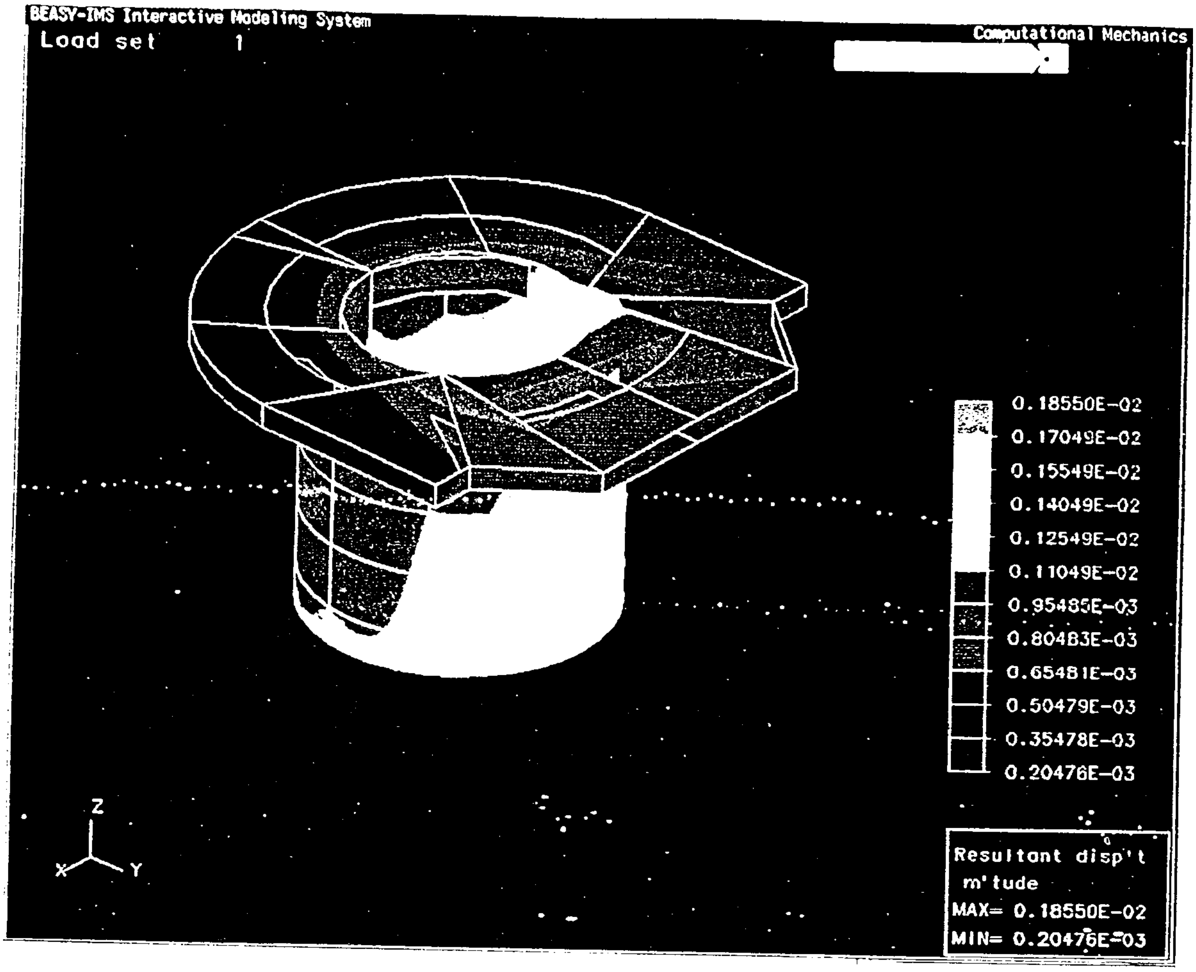


FIGURE 33. CONTOUR PLOT OF RESULTANT BUSHING DISPLACEMENTS

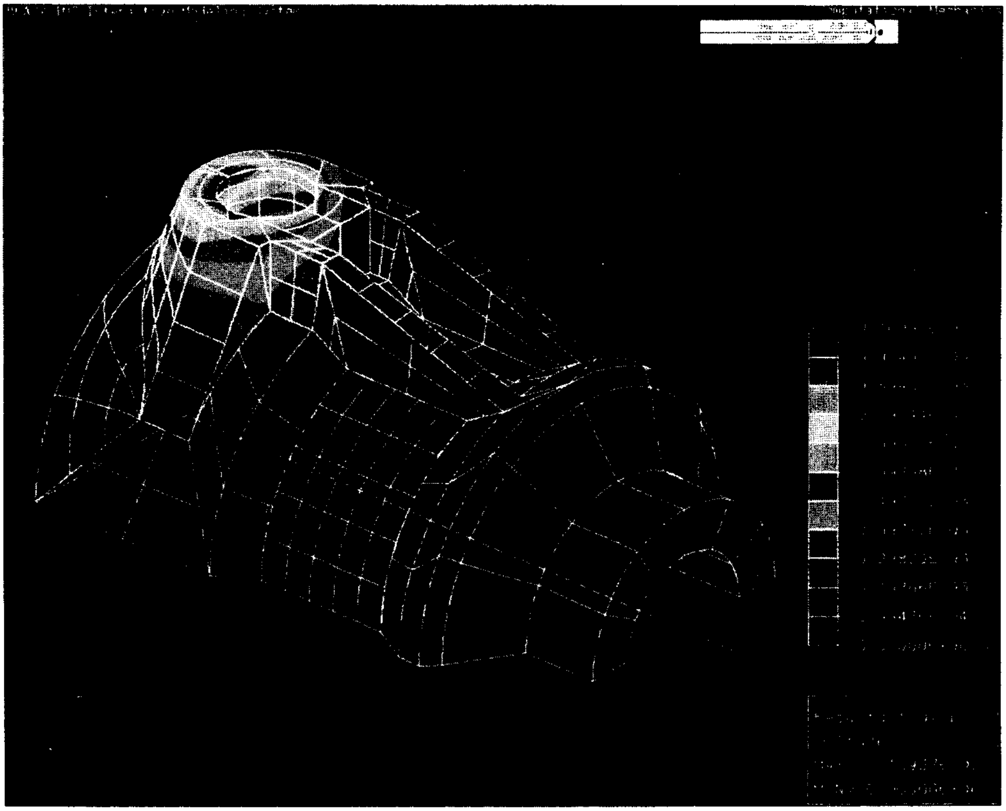


Figure 34: Contour plot of resultant housing displacement due to the pressure-fit of bushing

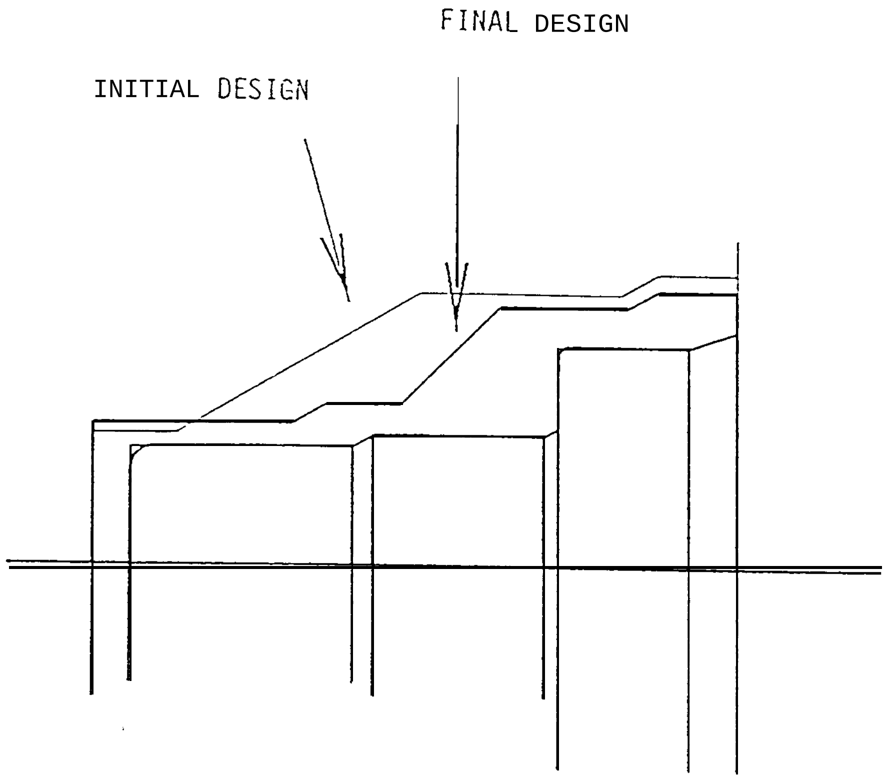


FIGURE 35. A COMPARISON OF THE INITIAL AND FINAL DESIGNS OF AN AXISYMMETRIC JET ENGINE FUEL CONTROL COVER

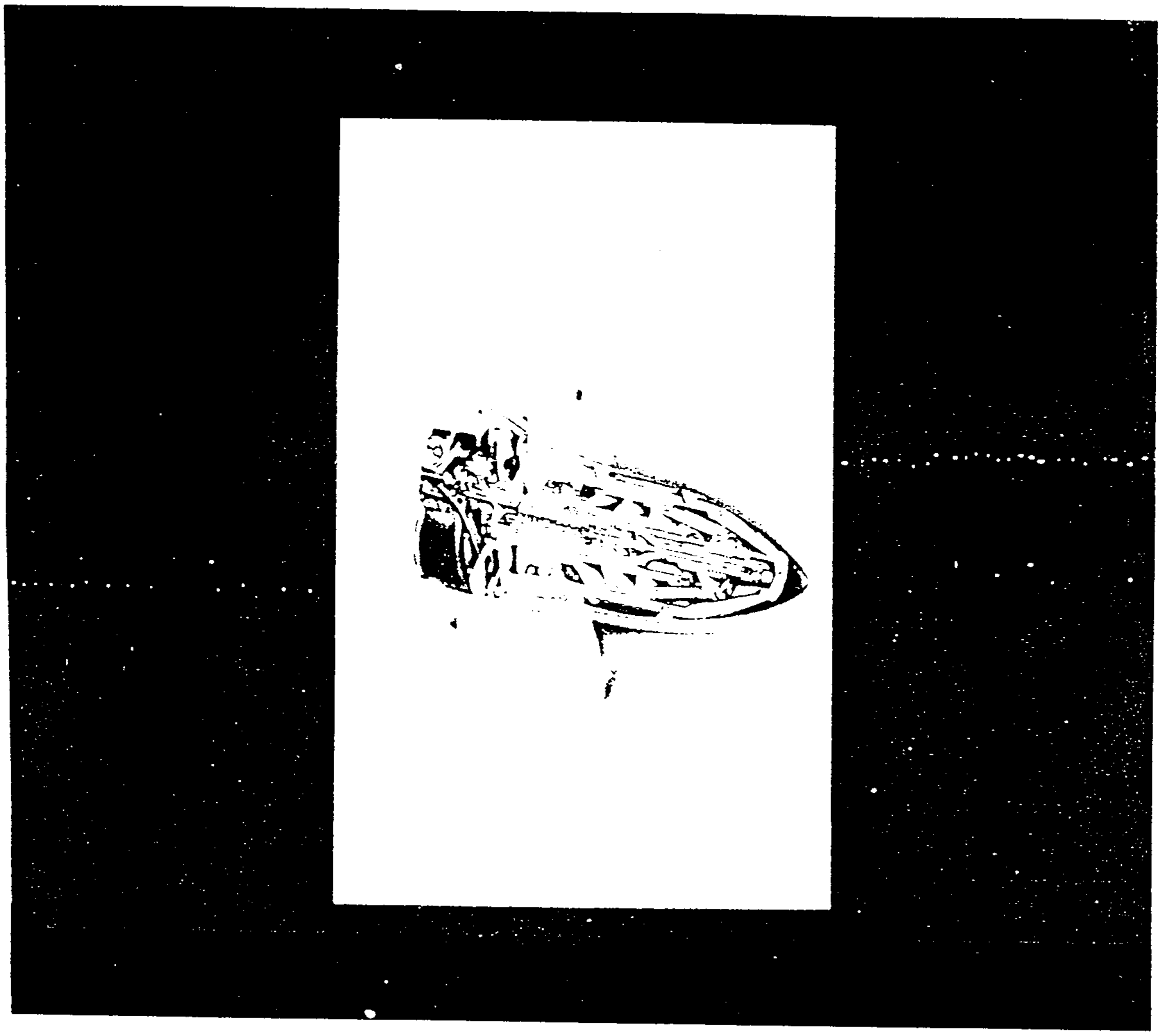


FIGURE 36. ILLUSTRATION OF A PROP-FAN PROPELLER SYSTEM