New Jersey Institute of Technology

Digital Commons @ NJIT

Winter 1-31-1994

Semi-automated approach to shape optimal design of planar structures

Gurinder Pal Singh
New Jersey Institute of Technology

Follow this and additional works at: https://digitalcommons.njit.edu/theses

Part of the Mechanical Engineering Commons

Copyright Warning & Restrictions

ABSTRACT

This paper presents a general algorithm, using IDEAS programmability features to integrate different phases of shape optimization process. The present work is mainly concentrated on planar structures, defined by thin or shell finite elements. Interactive programs are generated to convert input data to geometric models, to define mesh areas, to generate finite element mesh and to execute adaptive mesh refinement techniques. The importance of automated mesh generation and refinement is highlighted and various optimization methods are considered. Further, a special attention is given to integration issues of shape optimization capabilities into CAD environment. To demonstrate the developed algorithm, an automotive rear suspension torque arm is selected for finite element analysis and shape optimization. The practical example highlights the important features of IDEAS programs and analysis results are explained by plotting stress contours for different steps during the optimization procedure. As a whole, the stress has been given to m develop a general approach, rather than concentrating on specialized areas of shape optimal design process.

SEMI-AUTOMATED APPROACH TO SHAPE OPTIMAL DESIGN OF PLANAR STRUCTURES

by
Gurinder Pal Singh

A Thesis
Submitted to the Faculty of the
New Jersey Institute of Technology
in Partial Fulfillment of the Requirements for the Degree of
Master of Science in Mechanical Engineering

Department of Mechanical and Industrial Engineering

January 1994

Blank Page

APPROVAL PAGE

Semi-automated Approach to Shape Optimal Design
of Planar Structures

Gurinder Pal Singh

Dr. Raj Sodhi, Thesis Adviser / Date
Director of Manufacturing Engineering Programs
New Jersey Institute of Technology

Dr. Nouri Levy, Committee Member Date
Associate Professor, Department of Mechanical and Industrial Engineering
New Jersey Institute of Technology

Dr. Zhiming Ji, Committee Member Date
Assistant Professor, Department of Mechanical and Industrial Engineering
New Jersey Institute of Technology

BIOGRAPHICAL SKETCH

Author: Gurinder Pal Singh

Degree: Master of Science in Mechanical Engineering

Date: January, 1994

Undergraduate and Graduate Education:

- Master of Science in Mechanical Engineering,
  New Jersey Institute of Technology, Newark, NJ, 1994

- Post Graduation in Mechanical Engineering,
  Punjab Engineering College, Chandigarh, India, 1992

- Bachelor of Engineering in Mechanical Engineering,
  Guru Nanak Dev Engineering College, Ludhiana, India, 1990

Major: Mechanical Engineering

This work is dedicated to AMRITA
. . . my inspiration.

ACKNOWLEDGMENT

TABLE OF CONTENTS

TABLE OF CONTENTS

(Continued)

TABLE OF CONTENTS

(Continued)

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Introduction to Shape Optimal Design

Shape Optimal Design can be defined as the solution to an optimization problem involving structural geometry as a design variable. Every shape optimal design problem is formed to achieve a predetermined set of design objectives, subject to geometrical and/or behavioral constraints. Most of the times, the primary design objective is set to minimize the structural weight, without violating the geometrical and functional constraints. The constraints may include specific dimensions, and maximum stresses.

In case of a pure structural optimization problem, the objective is achieved by varying the sizes of design variables, such as plate thickness, cross sectional area, moment of inertia etc. The point to be stressed is that the geometry of the finite element model, material distribution and topology remain unchanged. Such structural problem is a non linear mathematical programming problem, to which standard minimization techniques could be applied.

In case of planar structures, thickness of shell element as only design variable, weight can not be reduced beyond a limit, due to either a minimum gauge requirement or constant thickness requirement, imposed by manufacturing process. The boundary curves defining the shape of the structure are selected as design variable with an imposition of maximum stress or displacements or natural frequency as optimization constraint.

The exterior and interior boundaries are controlled to alter the shape of the structure. The continuously changing shape of the finite element model requires careful consideration to represent the boundary shape, to maintain the integrity of the finite element mesh, to refine the finite element mesh as distortion of elements occur, and to enhance the accuracy of the sensitivity analysis. Some of these problems are not present in

structural optimization problems or are easy to solve, as the geometry is unchanged. In this view shape optimization problem is more complex than a sizing optimization.

Minimizing the weight of the structure is an important consideration in any design process. This work also deals with weight minimization of any planar or two dimensional structure, as design objective, using changes in the boundary shapes. Considerations of limited energy and material resources, technological competition, and special functional requirements as in aerospace and bio-medical applications, are the driving force to current research in this area, and indicate growing significance for the field in the future.

1.2 Objective and Scope of Present Work

The primary objective of this work is to develop an integrated approach to the independent steps of shape optimization process. The field of study is limited to planar or thin shells and two dimensional structures only. IDEAS software, a widely accepted Design and Analysis software tool is selected to develop an algorithm, which addresses the different problem areas of shape optimization procedure such as, geometrical model description, mesh areas formation, finite element mesh generation and adaptive mesh refinement, model solution, optimization parameters setup, and execution of shape optimization solution. Considerations are also given to different shape representation methods and optimization techniques.

To implement the proposed general algorithm and to demonstrate its flexibility, an automotive rear suspension torque arm is selected for the purpose. This work also attempts to present an integrated approach including CAD computer codes and finite element software having shape optimization capabilities.

CHAPTER 2

REVIEW OF RELATED LITERATURE

One of the first treatments of the problem of obtaining optimum shape of a structure without compromising with its functional and geometrical constraints was done by Zienkiewicz and Cambell. They used the features of finite element analysis with node coordinates as design variables to find an optimum shape. With the advent of digital computers and availability of general numerical analysis methods, interest in this field was increased greatly. Furthermore, a number of commercial optimization softwares based on well established finite element computer codes have been introduced. Some of the widely used softwares in this class include; ANSYS, IDEAS, MSC-NASTRAN, PATRAN, ABAQUS, SAMTECH etc.

The advantages of shape optimization procedure have resulted in keen interest to develop applications in automotive, aerospace, and bio-medical industries. This chapter describes briefly the previous research and development work, with respect to present study, in this field. The present work also surveys the problem areas encountered in shape optimization, and which are absent or easier to deal in structural optimization. These problems could be classified into two broad areas.

First, due to the continuously changing boundaries of the finite element model, it is difficult to ensure that the accuracy of the analysis remains satisfactory, throughout the design process. Secondly, more processing time is required to obtain good sensitivity derivatives with respect to shape design variables than with respect to sizing variables.

A literature survey presented by Ding(1986) reviewed various numerical and analytical methods for shape optimization of structures with special attention paid to different steps involved in shape optimization process. The steps considered were model description, selection of the objective function and shape variables, representation of the

boundary shape, finite element mesh generation and refinement, sensitivity analysis and solution methods.

An approximation concept approach to shape optimal design was given by Braibant and Fleury(1985), in which a convenient geometric representation to describe the boundaries of the structure by Bezier or B-splines curves was described. A general algorithm for shape optimization was devised to combine mixed approximations and dual method. A general method to describe complex geometries in a compact way by a set of design variables was also suggested.

Botkin(1982) presented a new approach to shape optimization problem of plate and shell structures. This work provided many useful guidelines to develop automatic programs for planar structures, during present research.

Several computational methods for optimization of structural shapes were considered. A more accurate approach for shape optimal design sensitivity was given by Hou, Cheng, and Sheen(1988). Also a numerical method using direct integration and B-splines for shape optimization problem including torsional elements, was given by Walter(1993).

The first work in the area of composite laminated plates was carried out by Kikuchi and Lee(1989). Another important area of shape optimization process is mesh generation and refinement. Kikuchi(1985) presented adaptive finite element methods for shape optimization of linearly elastic structures. The quantitative effect of element distortion near the design boundaries was identified in terms of interpolation error associated with the finite element mesh. A computer program was developed to combine numerical grid generation, an automatic remeshing with the grid adaption and design change.

The problem of linking geometrical description with the mesh generation capabilities was addressed by Bannet and Botkin(1985). A description format was developed, which used only the boundary information and was connected to a finite

element mesh generator, which required only the boundary information to generate the finite element mesh. Thus, leading to a more accurate estimate of the true solution. A general methodology for structural shape optimization problems using automatic adaptive remeshing was given by Bugeda(1993).

The concept to integrate shape optimization and CAD was given by Rasmussen(1982). A structural optimization system CAOS was developed for the integration of structural optimization facilities into a computer-aided design environment. A similar approach matching CAD and shape optimization concepts was developed at aerospace laboratory, University of Liege, Belgium. According to this method, the structure to be optimized, was decomposed into a set of simple sub regions. The shape of these sub regions was described by master nodes. The master nodes positions were selected as design variables, i.e. the unknowns of the optimization procedure. Sometimes these sub regions were also represented by Bezier or B-splines blending functions. This optimization capability for shape optimization combined a parametrical representation of the regionalized design elements which model the structure, a rigorous sensitivity analysis formulation, and an approximation concept approach for solving the optimization problem.

Though, the present work is mainly focused on planar structures or two dimensional problems, the concept of shape optimal design of three dimensional solid components was also taken into consideration. Imam(1982) presented a a general approach for 3-D shape optimization of structural components, which can only be defined using solid or thick shell type elements. Three dimensional shape optimization problem is geometrically found more complex as compared to planar shape optimal design.

This chapter reviewed the related literature, which formed some of the basic guidelines for the present work. Now, the next chapters would present each concerned topic in detail. The very next chapter highlights some of the concepts used in automated mesh generation and refinement techniques.

CHAPTER 3

AUTOMATED MESH GENERATION AND REFINEMENT

3.1 Mesh Generation

Automated mesh generation and refinement is an integral part of any shape optimization algorithm. There has always been an increase interest in the development of automatic mesh generation algorithms capable of discretizing any geometry into a valid finite element mesh without user intervention. One factor contributing to this is the availability of advanced geometric modeling systems which have greatly increased the efficiency of the design process, thus making the finite element mesh generation portion of the analysis process and even more obvious bottlenecks. A second factor is the need to improve the robustness of the entire finite element modeling process so it can be reliably used by designers that are not finite element experts. The only way to meet these goals is to automate the finite element modeling process. For purposes of this discussion an automated finite element process accepts a geometric description of the problem with analysis attributes tied to it as input and produces results, to a prescpecified level of accuracy, as output. At this time such systems are not so popular, however, an active work is still going on for the development of the various components that are needed to construct such systems. This present work is an attempt to develop an automatic mesh generator using computer aided engineering analysis software IDEAS to perform shape optimization procedures on planar structures.

The selection of an algorithmic approach to automatic mesh generation begins with the determination of the requirements that it must satisfy. Items that must be considered in the selection of meshing approach for use in automated finite element modeling procedure include:
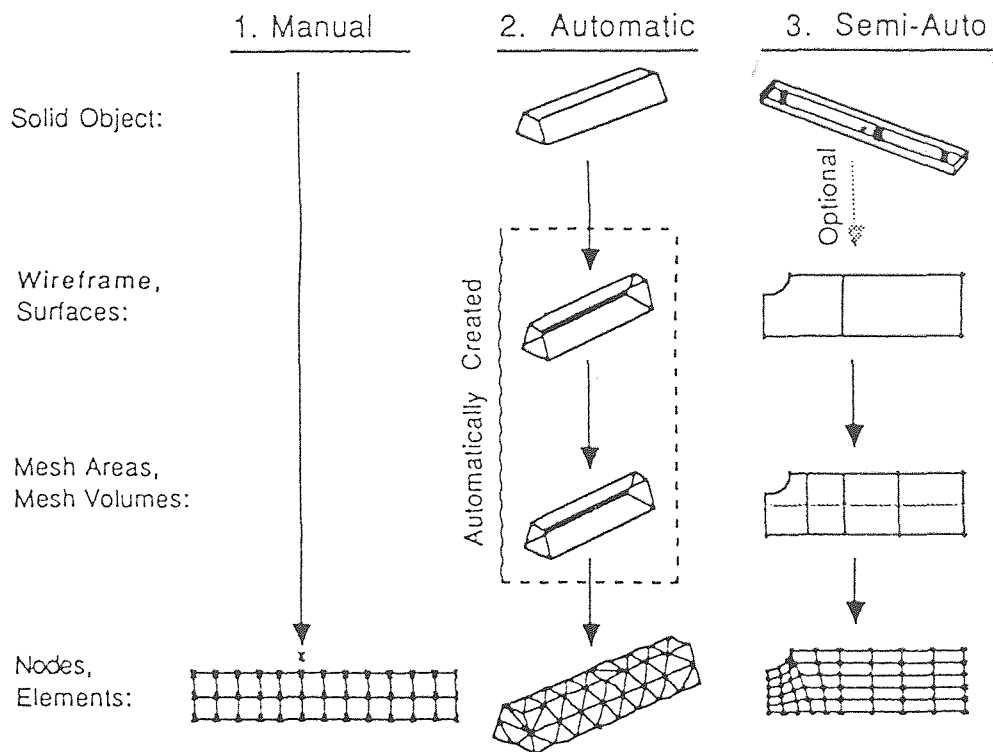
Fig 3.1 Various approaches for mesh generation

1. The geometric modeling systems to which the mesh generator will be integrated influences the selection since the computational effort required for the needed geometric operations, and the difficulty of providing those operators, is a function of both the geometric modeler and the finite element mesh generator. In general, some geometric modelers will not currently support the geometric operations needed by some meshing generation algorithms.

2. The type of finite element mesh desired may influence the selection of a meshing algorithm. For example, some mesh generators can not produce extremely coarse meshes while others may have a computational growth rate that causes their use for fine meshes with many elements to be computationally prohibitive.

3. The class of analysis to be carried out and the finite element solution procedures have a strong influence on the types of meshes that should be generated.

4. The form of mesh improvement desired during adaptive analysis as well as the development of efficient resolution procedures also have an influence on the meshing algorithm to be selected.

3.2 Algorithms for Automated Mesh Generation

The problem of mesh generation is to convert the geometry to a form understood by a finite element solver (a finite element mesh), in as automatic a manner as possible. Fig. 3.1 shows various approaches for mesh generation. Depending on the type of application considered, the element type is selected. Fig. 3.2 describes various element types which could be utilized to define different finite element models. There are various popular ways of generating these meshes and they can be classified into the following categories:

1. Laplacian Methods - A set of simultaneous nonlinear equations for the position vectors of the interior nodes with respect to the neighboring nodes is solved using iterative techniques. A starting grid is required, which can be improved using this method. This may be used to smooth meshes created using other methods.
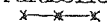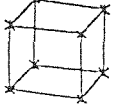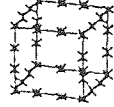
9

Fig 3.2 Types of elements

2. Mapping Methods - A function is used to map the given geometry into a simple geometry. This simple geometry is meshed and all the node points are mapped back to the original geometry. Various mapping functions have been used such as isoparametric mapping and the Schwarz-Christoffel transformation. Mapping methods do impose a number of restrictions on the geometry of the object.

3. Cell Decomposition or Spatial Enumeration - This involves dividing up the space enclosed by an object into regular shapes using the octree or quadtree methods. These are then modified or rearranged to get a valid mesh. This method is particularly suited to the Cell Decomposition type solid (CSG) modelers.

4. Surface or Volume Triangulation - This involves cutting up a given surface or volume into subsurfaces or subvolumes using standard methods. This "cutting up" is continued until acceptable sized finite elements are obtained. This technique can handle arbitrarily complex geometries and so can be automated to a higher degree. It is most suited to be used with boundary representation type solid modelers, but may also be used with all of the types.

3.3 Adaptive Mesh Refinement

Adaptive meshing is a way to automatically change a mesh of nodes and elements to refine it. Often, a final grid of elements is required in areas of high stress or strain energy. Using IDEAS, adaptive meshing task could be programmed to refine a mesh implementing either the results obtained from analysis or element distortion values as the basis for the refinement. The general procedure involving adaptive mesh refinement to reduce element distortion is displayed in Fig. 3.3.

There are four basic approaches to adaptively improve a finite element mesh including:

1. subdividing selected elements (h-refinement)
2. increasing the polynomial order of selected elements (p-refinement)
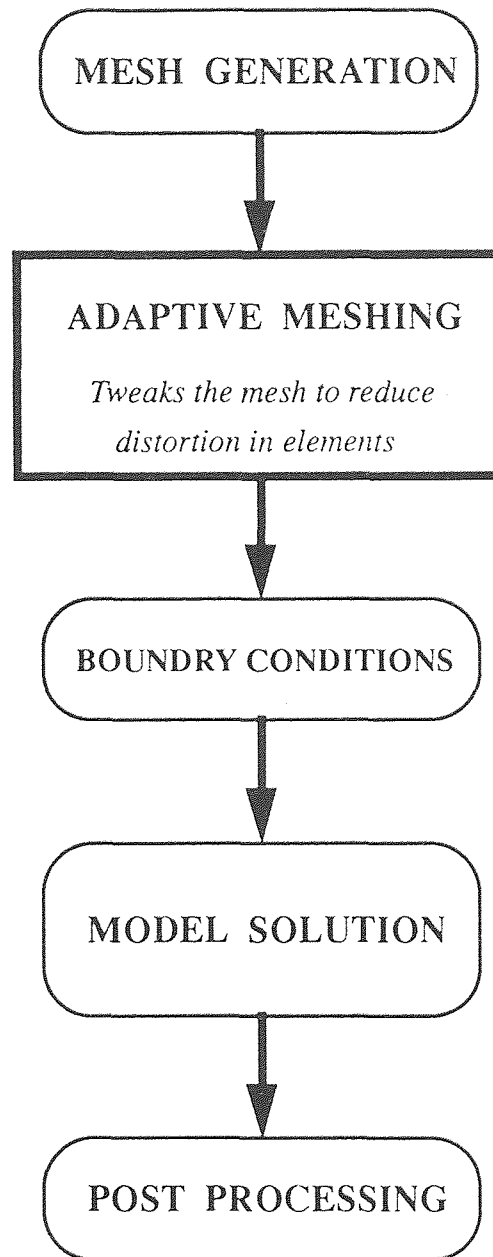
Fig 3.3 Application of Adaptive mesh refinement to reduce element distortion

DISTORTED ELEMENTS   REFINED ELEMENTS
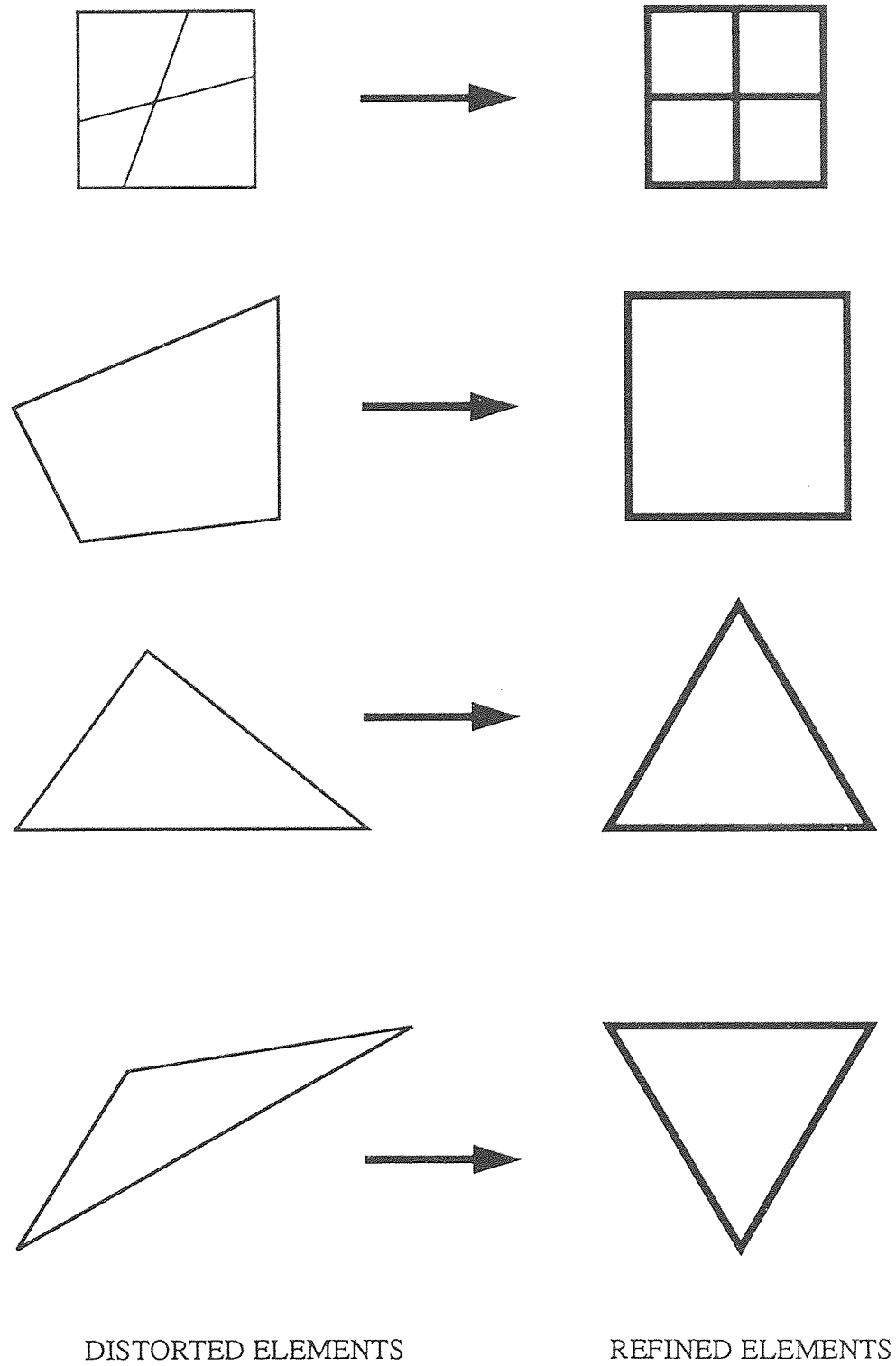
Fig 3.4 Smoothing of distorted elements by Optimizing method Selection

3. moving node points in a fixed element topology (r-refinement), and

4. defining a new mesh having a better distribution of elements.

The most important step in the adaptive mesh refinement is to identify the regions which require mesh refinement. There are mainly two approaches being widely used to select the region for mesh refinement.

1. The first approach considers the potential energy of the trial finite element solution for selecting the critical region. It is argued that since the approximate solution gives an upper bound on the true value of potential energy, the best grid may be defined as the one that gives lowest possible upper bound. In practice however, the formal solution of the problem is avoided because of the highly nonlinear form of the objective and of the geometry constraints that depend on nodal locations. Optimality conditions are normally too complicated to be operationally useful and, rather than working with these equations directly, several authors have developed guidelines that approximate the true optimality conditions and at the same time are easy to implement computationally.

2. In the second approach, the finite element model accuracy is improved by an adaptive mesh refinement scheme using strain energy density gradients to identify regions which require mesh refinement. A contour plant of the Strain Energy Density (SED) for the object is taken. The areas with undesirably high SED variation are identified and the elements belonging to those regions are refined using various techniques.

The value of SED variation above which an element will be refined is obtained from the following:

$$CV = \Delta E_{av} + \beta (\Delta E_{max} - \Delta E_{av})$$

where CV is the SED difference cut off value, ΔE_{av} the average SED variation for all elements, ΔE_{max} the maximum SED variation in an element and β a parameter to be selected based upon the problem (generally between 0 and 0.5).

Fig. 3.5 displays the general process of implementing adaptive mesh refinement techniques, based on analysis results.
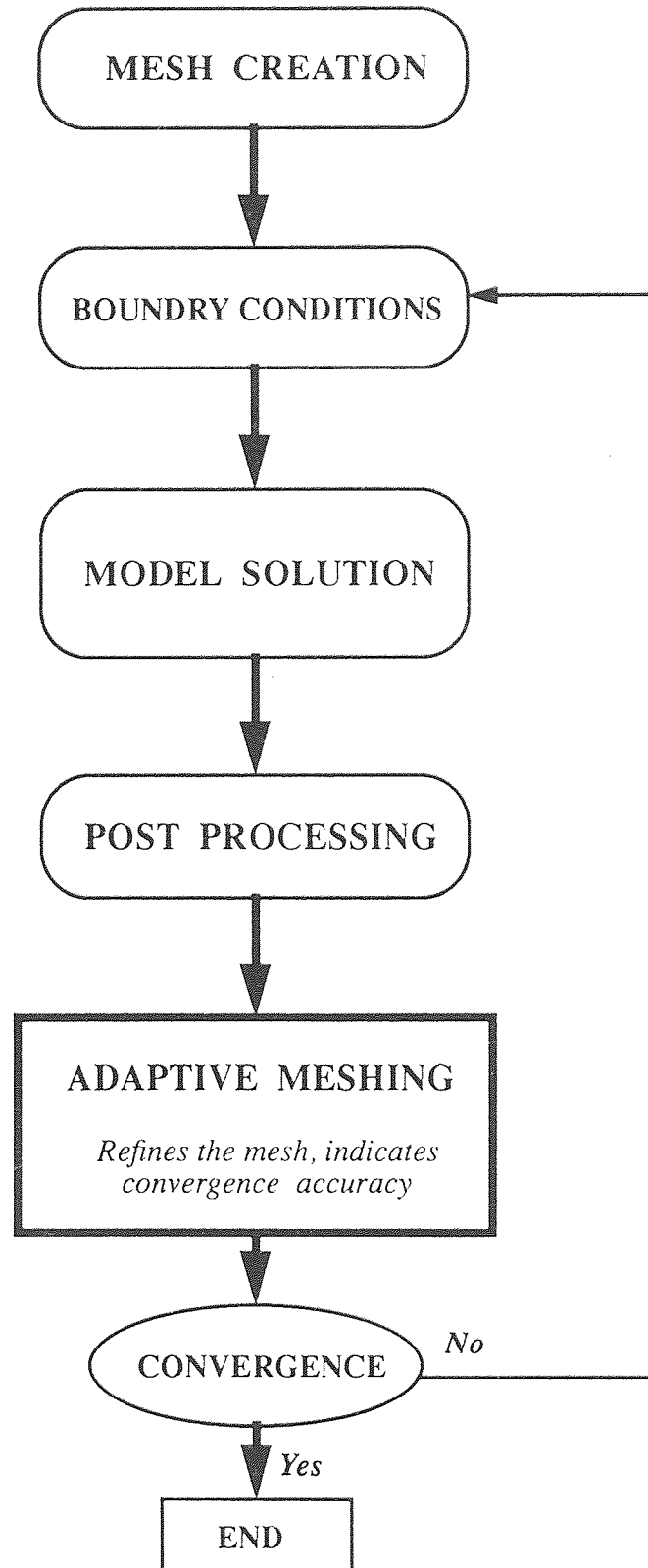
Fig 3.5 Application of Adaptive mesh refinement based on analysis results.
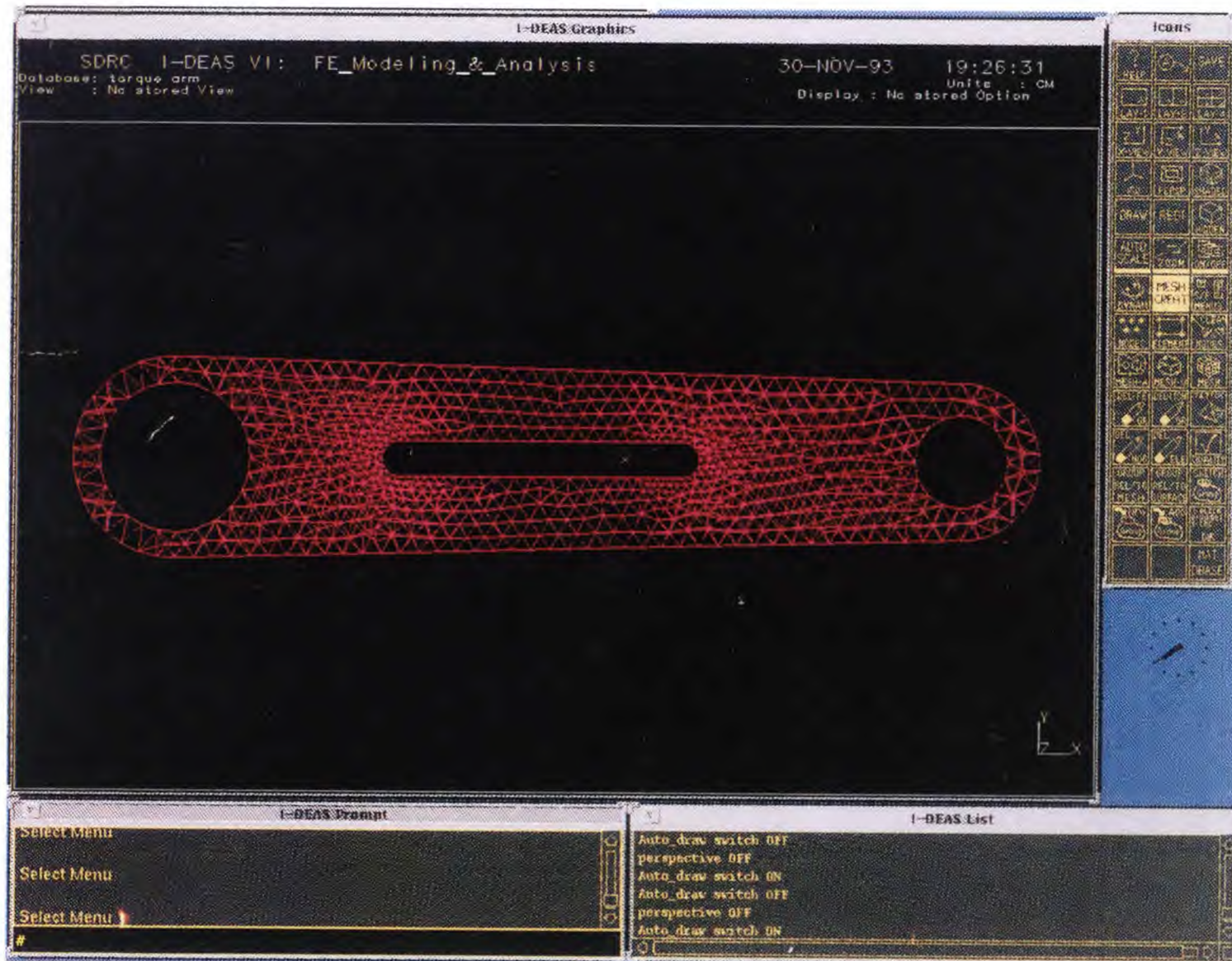
Fig.3.6 Element distortion in finite element mesh
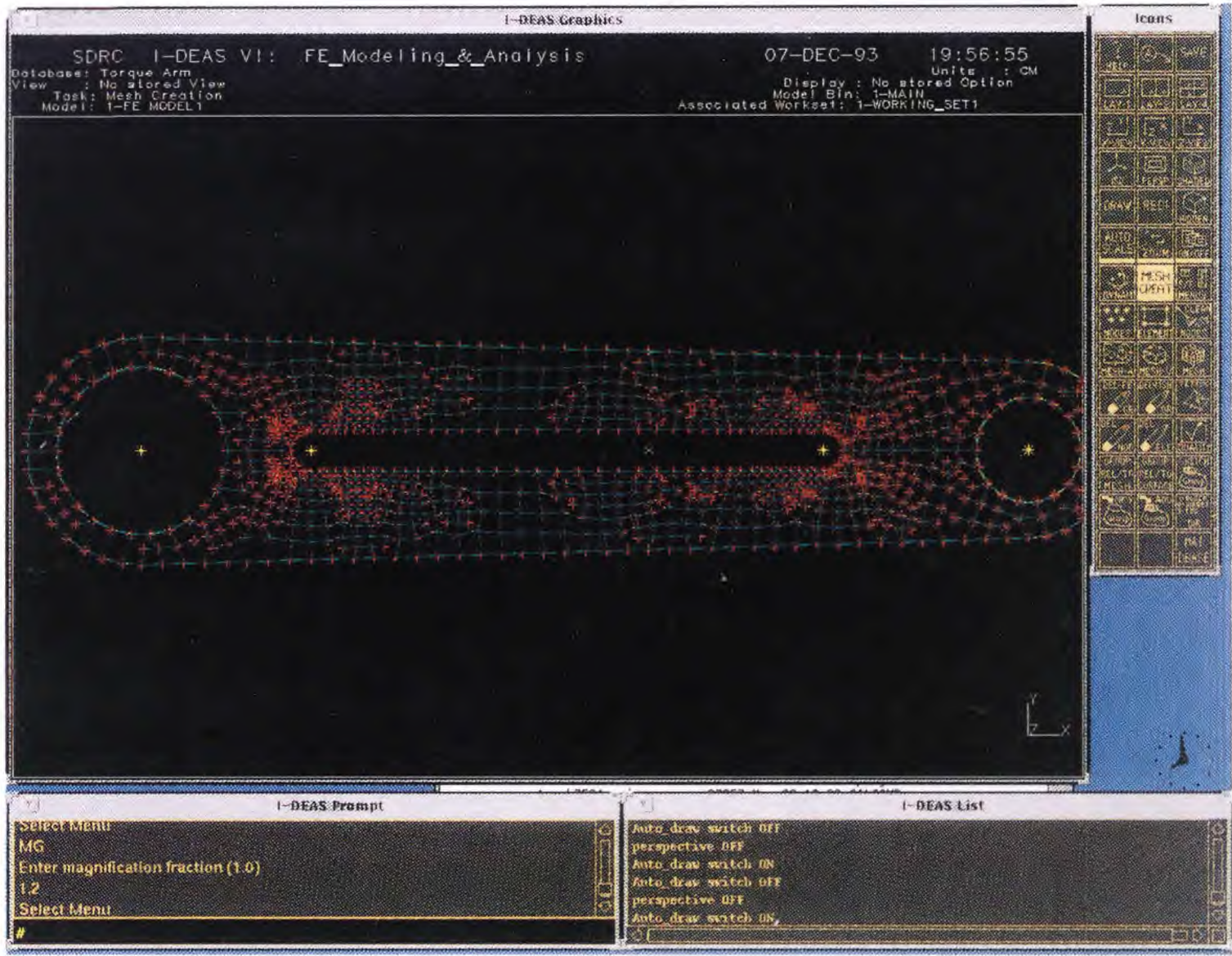
Fig 3.7 Adaptive mesh refinement using IDEAS

Various combinations of these approaches are also possible. Determining which of these approaches is the best for a particular class of problems is a complex decision which must consider the cost of the entire solution process. In an automated finite element modeling procedure, this cost includes the generation of finite element model, the adaptive improvement of that model, the determination of the a posteriori error measures and the solutions to the algebraic equations resulting from the various finite element models that must be analyzed during the process. Although the majority of investigations to date have considered measures in terms of the number of degrees of freedom in the finite element model versus the solution accuracy, or the cost of the solution of the resulting equations versus the solution accuracy, they have not attempted to measure the total cost of the entire finite element modeling process. Since the cost of mesh generation in an automated finite element modeling system can be on the order of the cost of the solution to the finite element equations, the selection of a mesh generation procedure and its interaction with the adaptive mesh improvement procedures is a critical consideration. Fig. 3.7 portrays an example of mesh refinement using adaptive methods under finite element modeling module of IDEAS.

Important considerations for automating mesh generation and refinement process were discussed. Application of IDEAS adaptive mesh refinement capabilities were considered. Next chapter presents an approach of integrating shape optimization procedure with a computer-aided design environment. Various shape representation techniques are also considered.

CHAPTER 4

INTEGRATION OF CAD WITH SHAPE OPTIMIZATION

4.1 Shape Optimization and CAD

Inspite of considerable development in the field of computer-aided design, the available systems are still considered to be the first generation of a long row of computer-integrated manufacturing systems. The future systems will provide an integrated environment for design, analysis, and fabrication of products. Thus, the CAD system could be simply regarded as a data base for geometrical information, equipped with a number of tools to facilitate the design procedure. Among these tools are facilities for structural analysis and optimization, with standard CAD features such as, drawing, modeling, and visualization tools. The result of this integration would be CAD systems for rational design in which structural optimization is an important design tool. The major problems for CAD-integrated shape optimization include:

1. There are many possible formulations of the shape optimization problem as the design objective could be selected as minimize weight, stress, compliance, displacement, or any property derived from the geometric model. Mathematically, different formulations lead to very different optimization problems.

2. To use a mathematical programming technique to solve the problem, the continuous shape of the geometry must be described by a finite, number of design variables. This problem is connected with the data structure of the CAD system, which is not flexible enough to allow for the shape changes, required by the optimization module. The solution to this problem is that the interface to the optimization system must provide a translation of CAD data to a form more convenient for shape optimization.

3. The geometrical information is interchanged between CAD model and the optimization application, rather than just passed on and the optimized geometry goes the opposite way, i.e. from Optimization module to CAD model

18

4. In most cases, the initial geometry possesses certain measures and shapes, which are important for functional or geometrical constraints and, therefore, cannot be altered during the shape optimization process. A method must be devised to maintain the functionality of the geometry, throughout the optimization process.

5. For finite element analysis, the initial CAD model of the structure must be converted to mesh areas, and this finite element mesh must conform to the changes of the geometry, as the optimization process progresses.

In present work an attempt has been made to generate mesh areas and finite element mesh, from the given geometric information. The IDEAS program takes geometric data for boundary curves and inner cutouts, as input and creates effective mesh areas to produce finite element mesh. This program can be implemented to produce finite mesh areas for three dimensional components also, with small modifications.

4.2 Shape Representation Techniques

For the geometric definition of any object, it is first described by indicating its geometrical boundaries. During shape optimal design process, the boundary of the structure continuously varies, leading to many complexions. It is difficult to maintain an adequate finite element mesh for analysis, keeping elemental distortion values within allowable range. Also proper care is required to be taken to enhance the accuracy of the sensitivity analysis. In most of the cases, the problems are associated with reducing stresses at a boundary by altering the boundary. Therefore, the manner in which the boundaries are represented is a key element in the process of obtaining optimum shape. The important methods to represent shape of any structure include:

1. Boundary representation by boundary nodes

2. Polynomial representation of boundaries

3. Spline representation of boundaries

4. The Design element concept

5. Boundary representation by spline blending functions

First method is the simplest approach to represent boundary of a two dimensional structure. Generally, the design variables of the shape optimal problem are chosen as the node coordinates of the finite element model. Besides simplicity, another advantage offered is to obtain a general curved boundary that is automatically followed by the finite element mesh without depending on the necessary shape required to obtain the minimum weight. This approach leads to many problems such as, increase in the number of design variables, tendency to produce unrealistic designs and, problems related to element distortion leading to inaccurate results.

The boundary curves may also be represented in the form of polynomials and, polynomial coefficients as design variables to characterize the shape. A more general approach is to define the boundary as a linear combination of shape functions with the coefficients as the design variables. This approach will surely reduce the net number of shape variables but may result in an oscillatory boundary shape with high order polynomials due to the numerical instability of the higher order curves. This problem can be eliminated by using the spline representation of the boundaries.

The splines are composed of low-order polynomial pieces, combined to give smoothness. The natural choice to define a moving boundary can be a cubic spline function, which has two continuous derivatives at every point and also possesses minimum mean curvature. The advantage of using spline representation is, better sensitivity accuracy and , application of the Bezier and B-spline blending functions provide great flexibility for the geometrical description. Another advantage with the B-spline formulation is, boundary regularity requirements are taken into consideration automatically.

One of the newest approach to achieve an adequate finite element model is to use the design element concept. In this approach the structure is divided into a few regions. These regions, or design elements can be described by a set of master nodes, that controls the geometry. Associated with the design element is a set of design variables, that describe the location of the master nodes, which orient during the shape optimization process. Each design element consists of

DESIGN ELEMENT BOUNDARY
OTHER FINITE ELEMENT BOUNDARIES.
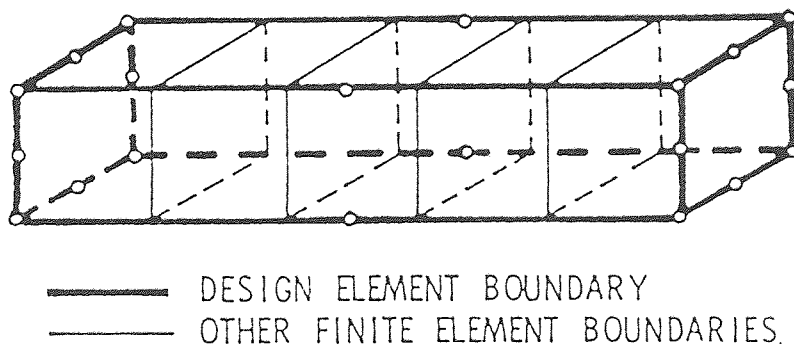
Fig 4.1 The design element

many finite elements as depicted in Fig 4.1. The boundary of a design element can be described by using two-dimensional isoparametric finite element interpolation functions or, spline blending functions. The major advantage of using design element concept is to describe complex geometries and three dimensional components.

The current chapter presented some of the important features of integration concept for CAD system and shape optimization process. The problems in this area were also highlighted. Various shape representation techniques were studied and for the practical example of torque arm, the geometric boundary was represented using boundary nodes method and the spline representation for outer boundary curves. Next chapter deals with the mathematical representation of shape optimization problem. Different optimization methods are discussed with respect to shape optimal design of planar structures.

CHAPTER 5

MATHEMATICAL REPRESENTATION AND OPTIMUM SOLUTION

5.1 Problem Formulation for Shape Optimization

The shape optimization problem can be represented mathematically as

$$\min F(S_1, S_2, \ldots, S_n) \tag{1}$$

subject to:

$$d_i (S_1, S_2, \ldots, S_n) = 0 \qquad i = 1, \ldots p$$

$$e_j (S_1, S_2, \ldots, S_n) \leq 0 \qquad j = 1, \ldots q$$

$$S_k^l \leq S_k \leq S_k^u \qquad k = 1, \ldots n$$

where:

F :   Objective function

d_i: Equality constraint function describing ith structural response

e_i: Inequality constraint function describing ith structural response

S_i: Vector of n design variables representing shape of the object

S_k^l: Lower limit of shape variables

S_k^u: Upper limit of shape variables

p :   Total number of equality constraints

q :   Total number of inequality constraints

n :   Total number of shape variables

5.2 Definition of Objective Function

The objective function is specified by the user as a part of optimization specification and can be selected in various ways, depending on which variable is required to be optimized.

(i) Weight, Mass or Volume Optimization

In most of the cases, the weight of the object is selected as the objective function which can be represented as

$$F(S) = \sum_{e=1}^{NE} \rho_e \Omega_e(S) \qquad (2)$$

where $\Omega_e(S)$ is the volume of eth finite element, and in general varies nonlinearly with respect to S_k.

(ii) Maximum Von Mises Stress

The objective function can also be selected in order to determine the shape which has the maximum Von Mises stress along a given part or a whole part of the boundary. Mathematically,

$$F(S) = \text{Max } \sigma_{vm} \qquad (3)$$

(iii) Difference Between Maximum and Minimum Tangential Stresses

$$F(S) = \sigma_{\theta max} - \sigma_{\theta min} \qquad (4)$$

where $\sigma_{\theta max}$ and $\sigma_{\theta min}$ are the maximum and minimum tangential stresses at all the sampling points.

(iv) Stress Leveling

This objective function can be represented in the form

$$F(S) = \phi (\sigma - \sigma_a)^2 \, dA \qquad (5)$$

where σ is maximum principal stress and σ_a is the average stress at initial shape and A is the part of a whole surface of the body.

(v) Weighted Objective Function

$$F(S) = (0.5\Omega / \Omega_0) + ((\phi (\sigma - \sigma_a)^2 \, dA) / (\phi (\sigma_0 - \sigma_a)^2 \, dA)) \qquad (6)$$

where Ω is the volume of the object, Ω_0 represents volume of the initial shape, σ is the maximum principal stress and σ_a, σ_0 represents average and maximum principal stresses, respectively of the initial shape.

Other examples of possible objective functions are maximum elastic displacement at any point in the structure and compliance. The use of the objective function reduces the effect of stress concentration in the altered boundary. Along with that it controls the continuity of changing boundary shape.

Mathematically, different objective functions lead to very different optimization problems. Elastic displacement and stress are ordinary scalar quantities that can be derived directly from the output from the finite element analysis. Minimizing weight is of the integral type and require some post-processing of the results.

In this work the objective function is selected to minimize the weight or volume of the structure.

5.3 Optimization Techniques

The same considerations that affect traditional structural optimization will be important in shape optimization; that is, the number of analyses should be small and the derivatives should be calculated as efficiently as possible.

5.3.1 Treatment of Stress Constraints

There are inherent difficulties associated with treating stress constraints in discretized structures which are compounded when shape is used as a design variable. If the stress data point is a continuous function of the design variables, the finite element stress results can only approximate this function which results in a highly nonlinear constraint behavior characterized by many local peaks and valleys. To some extent, this nonlinearity is inevitable as the remeshing guarantees some of this behavior.

In addition, some decision must be made as to how the constraints are to be defined. Associating the constraints with a finite element is unattractive since the finite element mesh will change during the design process. Clearly the stress constraint needs to be associated with some point in the structure. For the present time, the following limited approach has been chosen. It is assumed that the maximum stress occurs along a

boundary element. For this purpose each boundary element can be broken into a predetermined number of stress constraint segments. In each segment, the stress constraint is taken as the maximum stress in any finite element touching that segment of the boundary.

5.3.2 Method of Optimization

The traditional methods of nonlinear optimization have two severe drawbacks for extensive shape optimization. First, they permit large excursions in the design variables which may be not justified based on the starting finite element analysis. This radically altered design may be so infeasible as to seriously compromise the convergence of the optimization. Second, they tend to spend most of the computational time tracking active constraints. The irregularity of the constraints will seriously compromise this convergence process. In addition, of course, the direct methods require an excessive number of finite element solutions.

The approximation concept impose intermediate move limits on the design variables, large changes in the shape can be limited. This also allows for an orderly introduction and updating of the extrapolation for the constraint values based on coarse mesh solutions as described earlier. The idea that is being proposed is deficient in that, as the mass is reduced, the approximation used to predict the stress level becomes more unconservative. Therefore, at the end of each step, the design is usually infeasible and, in general, more infeasible than the approximations predict.

When the approximation concepts are used, the success of the optimization is often determined by the side constraints imposed on the approximate problem, which are usually called move limits. The particular implementation of move limits used is based on a percentage of the total motion allowed. Then, for the jth subproblem,

$$x_{i_{upper,lower_j}} = (x_i)_{j-i} \pm \Delta (x_{i_{upper}} - x_{i_{lower_0}}) \tag{7}$$

$$i = 1, 2, \ldots, \text{number design variables}$$

where the subscript 0 refers to the original, nonsubproblem upper and lower bounds and Δ is a global move parameter that refers to all design variables. In this formulation, the amount of motion is not dependent on the current value of the design variable. This is important because some design variables of the double cubic may pass through zero, and a move limit given as a percentage of the current value would produce very slow motion around zero.

5.3.3 Stress Approximation

As described earlier, an extrapolation had been used to approximate the actual stresses from an unrefined analysis based on a previously refined analysis. This will be handled by the following relationship:

$$\sigma_a{}^e = \sigma_c{}^e + \lambda A_c{}^{0.5} \tag{8}$$

where $\sigma_a{}^e$ is the approximate value of the actual maximum element stress, $\sigma_c{}^e$ the maximum element stress for the coarse mesh, A_c the coarse element area, λ the stress approximation parameter, and

$$\lambda = \frac{(\sigma_f{}^e)_n - (\sigma_c{}^e)_n}{(A_f)_n{}^{0.5} - (A_c)_n{}^{0.5}} \tag{9}$$

where the subscripts f and c refer to fine and coarse meshes and n indicates that λ is updated at step n.

Clearly, if significant changes in the design are experienced before λ is updated, the updated design will be significantly infeasible and instabilities in the design algorithm will result. One approach is to update frequently, which, to some extent, defeats the purpose of using this approximation. Another alternative is to select an arbitrary value of λ, say λ_0, to be used at all times when λ is calculated to be smaller than λ_0. If this λ_0 is judiciously chosen, it tends to protect against excessive reduction in material leading to the

previously mentioned instability. For the example considered in the present work, $\lambda_0 = -0.5$ with updates every two steps has proven this satisfactory.

5.3.4 Geometric Behavior Constraints

An aspect of shape optimization that has caused difficulty in the past is how to keep boundaries from intersecting as the design changes. Many times, typical behavior constraints such as stress or displacement will not control boundary movement sufficiently, resulting in the boundaries intersecting each other. The way this has been handled in the past is to put side constraints on the dimensions. This, of course, works well as long as the boundaries are a function of a single dimension only.

Otherwise, the solution to the problem is to define a new kind of behavior constraint referred to as the geometric behavior constraint and defined to be the distance between boundary segments. A constraint is assigned for each boundary segment in combination with every other segment, except for those segments which are on a common closed boundary curve. The relationship for computation of the number of constraints is

$$\text{ngc} = 0.5[(n-1)n - \Sigma (m_k - 1)m_k] \qquad (10)$$

in which n is the total number of boundary design elements, k the number of closed boundaries which are composed of more than one boundary design element, and m_k the number of boundary elements on boundary k.

The constraint values are calculated by a double loop through the coordinates of all boundary points, which have been stored continuously at the beginning of the list, computing the minimum distance between each point and every other point, and then retaining only the minimum value for each boundary segment. The distance computation is further refined by computing the actual distance between tangents to the discrete segments.

The mathematical representation of shape optimization problem was presented in detail. Various optimization methods were also discussed. The next chapter deals with

description of major features of IDEAS software, which was implemented to demonstrate the proposed algorithm. It presents a brief picture of Finite Element Modeling module of a versatile computer aided engineering analysis software, IDEAS and how these features are useful in shape optimal design process .

CHAPTER 6

INTRODUCTION TO IDEAS FINITE ELEMENT MODELING

6.1 Introduction to Finite Element Modeling & Analysis

Finite Element Analysis (FEA) is a process which predicts deflections and other effects of stress on a structure. Finite Element Modeling (FEM) divides the structure into a grid of "elements" which form a model of the real structure. Each of the elements is a simple shape (such as square or a triangle) for which the finite element program has information to write the governing equations in the form of a stiffness matrix. The unknowns for each element are the displacements at the "node" points, which are the points at which the elements are connected. The finite element program will assemble the stiffness matrices for these simple elements together to form the global stiffness matrix for the entire model. This stiffness matrix is solved for the unknown displacements, given the known forces and boundary conditions. From the displacements at the nodes, the stresses in each element can then be calculated.

A finite element model can be defined as the complete idealization of the entire structural problem, including the node locations, the elements, physical and material properties, load and boundary conditions. The model will be defined differently for different types of analysis: static structural loads, dynamics, or thermal analysis.

A finite element model is often made of more than one element type. The finite element model is made to mathematically model the deflection of the structure, not to look like it. Parts of a structure might be best modeled with beam elements, and other parts with thin shell elements.

The accuracy of the resulting solution will depend on how ell the structure was modeled, the assumptions made for loads and boundary conditions, and the accuracy of the elements used for the given problem. In general, the solution will be more accurate as

30

the structure is subdivided into smaller elements. The only sure way to know if you have sufficiently converged on the final solution is to make more models with finer grids of elements and check the convergence of the solution.

6.2 Steps in Finite Element Analysis

Finite element modeling consists of three steps. These are:

- Pre-processing
- Solution
- Post-processing

Pre-processing includes the entire process of developing the geometry of a finite element model, entering physical and material properties, describing the boundary conditions and loads, and checking the model.

The solution phase can be performed in the Model Solution Task of IDEAS Finite Element Modeling & Analysis, or in an external finite element analysis program. IDEAS Model Solution can solve linear statics, linear dynamics, conduction heat transfer, and potential flow analysis. For other types of analysis such as non-linear statics, the finite element model information can be written in the format required for an external finite element solver such as NASTRAN, ANSYS, or ABAQUS.

Post-processing involves plotting deflections and stresses, and comparing these results with failure criteria imposed on the design such as maximum deflection allowed, the material static and fatigue strengths, etc.

There are many possible sources of error in the user's model, such as the coarseness of the finite element, the type of elements used, or incorrect material properties. This is why post-processing should include checking for errors that might not have been detected while building the model.

6.3 Finite Element Modeling and Analysis

Finite element modeling and analysis is one of the families in IDEAS. There are several tasks in this family: pre-processing the model, solution, post-processing, and optimization. The basic tasks used for the three steps of finite element modeling include:

- Pre-processing
- Mesh creation task
- Geometry modeling task
- Boundary conditions task
- Solution
- Model solution task
- Post-processing
- Post-processing task

The basic geometry for a model is most easily built using IDEAS Solid Modeling. Geometry can also be created in the Finite Element Modeling Geometry Creation Task. However, the real advantage of using IDEAS as an integrated MCAE package is that the geometry from Solid Modeling can easily be shared between applications.

6.4 Finite Element Geometry Construction

The Geometry Modeling task is used to create and manipulate the wire frame geometry which will be used for defining mesh areas. This wire frame geometry can be created in this task, or it can be transferred from an object created in Object Modeling. Wire frame geometry is used as construction geometry. Nodes and elements are not created in this task.

Wire frame geometry includes points and various kinds of curves. Curves can be created as lines, arcs, circles, fillets, and splines. Any of these curves can be used to define mesh areas.

6.5 Generation of Mesh Areas

Mesh areas will be created from (and bounded by) wire frame geometry. A mesh area is an N-sided, closed area bounded by curves. Mesh areas are used for automatically generating nodes and elements and for defining mesh volumes. Mesh areas can be manually created by picking three or more curves that form a closed loop, or they can automatically be created by the program. The present work is focused on automatic mesh generation and refinement tasks. An IDEAS program was created to generate after geometry of the object is created.

Surfaces can be attached to mesh areas to describe the surface which the elements are to follow inside the mesh area boundary. If the mesh area is planar, surfaces are not required.

6.5.1 Mapped and Free Mesh

Nodes and elements are generated on mesh areas by one of two methods, mapped or free mesh. Mapped meshing requires the same number of elements on opposite sides of the mesh area, and requires that mesh areas be bounded by three or four "edges". If the user defines a mapped mesh area with more than four curves, the program will prompt the user to indicate the starting point for each edge, and will merge curves to form edges if necessary. Mapped mesh areas with three edges will generate triangular elements in one corner. The mesh density is controlled by the number of elements per edge, and biasing of element size toward one edge or the center.

Free meshing allows more flexibility in defining mesh areas. Free mesh areas can be much more complicated than mapped mesh areas. Mesh density is controlled by the "Global Element Size" and "Local Element Sizes" set on different points on the boundary curves. By varying these size settings, the user can have substantial control over the mesh density in different areas. The mesh will automatically be created by an algorithm which tries to minimize element distortion (deviation from a perfect square).

For free mesh areas with internal holes, the user must define a curve that connects the hole with the outer boundary. When picking connecting curves to define the mesh area, "walk" around the outer boundary in one direction and in to the center hole; walk around this hole in the other direction, and then back out to the starting position. This type of geometry would have to be further subdivided to use mapped meshing. Mesh areas created automatically from the solid object geometry will have the holes connected to the outer boundary of the mesh area.

6.5.2 Generating Meshes

Nodes and elements are generated on mesh areas with the GENERATE command. The program will ask the user if he wants to accept the displayed elements. If not, the user can go back and change the mesh specifications and repeat the procedure again, whenever required.

Nodes and elements will lie on a surface which is attached to the mesh area. Mapped meshing requires a surface, and will generate one using a "Coon's Patch Algorithm" if a surface does not exist. If the geometry is defined in the Object Modeling and transferred into Finite Element Analysis, the surfaces of the objects will automatically have surfaces defined, unless they were purposely deleted.

6.5.3 Mesh Volumes

Mesh volumes are defined by closed regions bounded by mesh areas. Mesh volumes can be either mapped or free mesh, and all the mesh areas must also be the same type, either mapped or free. Mapped mesh volumes are more restrictive than free mesh volumes, since the volume must be bounded by five or six mesh areas. In the case of six mesh areas, the interior is topologically a "box" and solid brick elements will be generated. If five mesh areas enclose the volume, the volume is wedge shaped. Elements generated will be solid brick elements except at the last edge, where edge elements will be used.

6.6 Preparing the Model for Solution

6.6.1 Material Properties

Each element contains a material property ID which refers to a table of material properties. Every element must reference one material table. One table may be referenced by many elements. Material properties can be isotropic, orthotropic, or anisotropic. Material properties can be defined before creating elements, or during the element creation process. If no material table has yet been created, IDEAS will force the user to create one. The default material properties represent steel.

6.6.2 Physical Properties

Physical properties are also referenced by elements. These represent factors like element thickness and beam cross-section properties. The default physical properties are usually meaningless and should not be used. Some elements do not need any additional physical properties, but the element must still reference a "dummy" physical table.

6.6.3 Model Checking

The Mesh Creation Task also contains several checks to help the user identify modeling errors in the finite element model. Typical problems that can be checked are duplicate nodes, duplicate or missing elements, and highly distorted or warped elements. The element checking commands are found under the ELEMENT menu, under the command QUALITY_CHECKS.

One of these checks is an element free edge check. This check will plot the free edges of elements not connected to another element. This can be a very useful check in finding element connectivity problems. Normally, this will plot the outer boundary of the model, which is where the elements are not connected to others. If elements adjoin each other edge to edge but reference duplicate coincident nodes rather than share the same

nodes, an extra line will show up in the free edge plot. This represents "crack" in the model. Duplicate elements defined by the same nodes will cause neither element to be plotted in this check and a missing line may show up in the plot.

Element distortion is another popular check. Values are reported by the distortion check from -1.0 to 1.0. A value of 1.0 represents a perfect square (a circle fits inside). Values less than 0.0 are horrible. A typical rule of thumb is that values should be between 0.5 and 1.0, but there is no exact cut-off for what is not acceptable. It depends on the type of analysis to be performed and where the badly distorted elements are located in the model. Avoid highly-distorted elements in important areas such as high stress locations. Sometimes due to the geometry the user is modeling, distorted elements can not be avoided.

Other element quality checks include checks for warping out of plane, interior angles, midside node placement, and coincidental elements.

Under the NODE menu, there is a coincident node check to detect coincident nodes within a small tolerance supplied by the user. This command will optionally renumber adjacent elements so that they share the same nodes. This is called "merging" out the duplicate nodes. IDEAS will ask the user if he wants to delete the unused nodes after renumbering the elements.

6.6.4 Boundary Conditions

The Boundary Conditions Task is used to build analysis cases containing loads and restraint boundary conditions to apply to the model. An analysis case is a collection of DOF sets, constraints, restraints, structural loads, and heat transfer loads. For most structural problems, only structural loads and restraints are needed.

Structural loads can be nodal forces (forces directly at a node) or pressures on the face or edge of an element (which are converted to nodal forces internally). A nodal force has six values, for the three forces and the three moments. To create nodal forces, the

user will be prompted to select the nodes. This selection can be done individually, one at a time, or by selecting nodes by other methods, such as outlining a screen area by diagonal points or a screen polygon. It will help to select the most appropriate view first to help in selecting nodes, depending on the method used.

Restraints are used to restrain the model to ground. Restraints also have six values at nodes for three translations and three rotations. Each entry can either have a value for the fixed displacement (0.0 means the degree of freedom can not move), or pick the menu FREE or reenter the letter "F" to say the degree of freedom is free to move. Nodes are selected to apply restraints the same way as applying forces. The values given for restraints apply to the displacement coordinate system for the nodes, not the global coordinate system. A model should normally be held in space by restraints so that it is not free to move in any direction even if there are no applied forces in that direction, or the problem may not solve.

Forces and restraints are graphically illustrated using arrows on the model. Forces have closed arrowheads and restraints have open arrow heads. Rotations in either case have a double arrowhead.

6.7 Finite Element Solution

6.7.1 Steps in Using Model Solution

The steps to perform in Model Solution to solve the model are:

1. Select the appropriate solution type.
2. Select the desired execution options such as using a batch or interactive solution.
3. Select the case set to used for the analysis.
4. Select the method, such as verification or solution.
5. Select the output datasets to save, such as displacements and stresses.

6. Solve.

To solve the finite element model for linear statics, the user must have the model properly restrained. Improperly creating restraints is a common user error. Even if a load is not applied in a particular direction, the user must restrain the model against all six possible rigid body motions or singularities will result and the solution will abort.

6.7.2 Interfacing Other FE Codes

When the model has been completed, check to make sure everything is connected properly, and built an analysis case containing loads and boundary conditions, the user is ready to solve the model. This can be done with IDEAS Model Solution or by using an external solver such as NASTRAN or ANSYS.

6.8 Post-Processing

Post-processing is the display and interpretation of results after the solution is finished. The steps required to make a post-processing display are as follows:

1. Group Elements - Either make an existing group of elements "current" or create a new group of elements to use for the display.

2. Analysis Dataset - Make sure one of the stored datasets "current". The selected dataset may contain deflections or stresses. The menus will change depending on what display types are valid for the current analysis dataset.

3. Display Form - Select the menu for the form of display, such as deformed geometry contour.

4. Data Component - For stress data, select what component to display, such as maximum principle stress, or Von Mises stress. (This does not apply to deflection.)

5. Display Option - Choose the display option such as continuous tone, free face, or fringe contours for stress data; line, hidden line, free face, or shaded image for deflection data.

6. Execute Display - Execute the display either using the EXECUTE command under the display option, or set the display type as global and execute the display with the GD-GLOBAL_DISPLAY command.

This chapter discussed the various features available under Finite Element Modeling module of IDEAS. The next chapter deals with the implementation of these features to optimize the shape of selected component, torque arm. IDEAS programs were also developed to execute various steps of shape optimization using IDEAS. Chapter 7 also highlights the actual working of these programs and possibility of integration of all such programs, leading to a general approach to shape optimal design of planar structures.

CHAPTER 7

IMPLEMENTATION OF SHAPE OPTIMAL DESIGN PROCEDURE USING IDEAS VI

7.1 Initial Design

To illustrate the proposed algorithm, an automotive rear suspension torque arm was selected for shape optimization. Fig. 7.1 describes the proposed initial design of the component under observation. It is approximately 50 x 11 cm and 0.3 cm thick and could be well treated as a planer structure. As a critical part of the rear suspension system the torque arm is attached to the chassis frame through its smaller end and other end is fixed to the rear axle of the automobile.

For present work, the component was supposed to be subjected to a nonsymmetric static loading condition in the X-Y plane and the forces act on the smaller end of the component. The numeric values of the acting forces were considered as 2789 N in the X direction and 5066 N in the Y direction. The larger end of the torque arm connected to the rear axle was constrained against the translation and rotation around the hole.

The material of the torque arm was assumed to be uniform and isotropic. Some of the important material properties like, modulus of elasticity and mass density, were also provided as a part of input data to execute the analysis. For shape optimization process, minimum weight of the component was focused as an objective function, and an equivalent Von-Mises stress of 80,000 N/cm^2 was imposed as a constraint function. By careful examining the stress analysis of the initial design it was decided to remove the material from both exterior and interior boundaries as a part of the shape optimization process, in order to reduce weight. The initial weight of the component was noted, through the summation of all the constituent elements, about 918 gms.
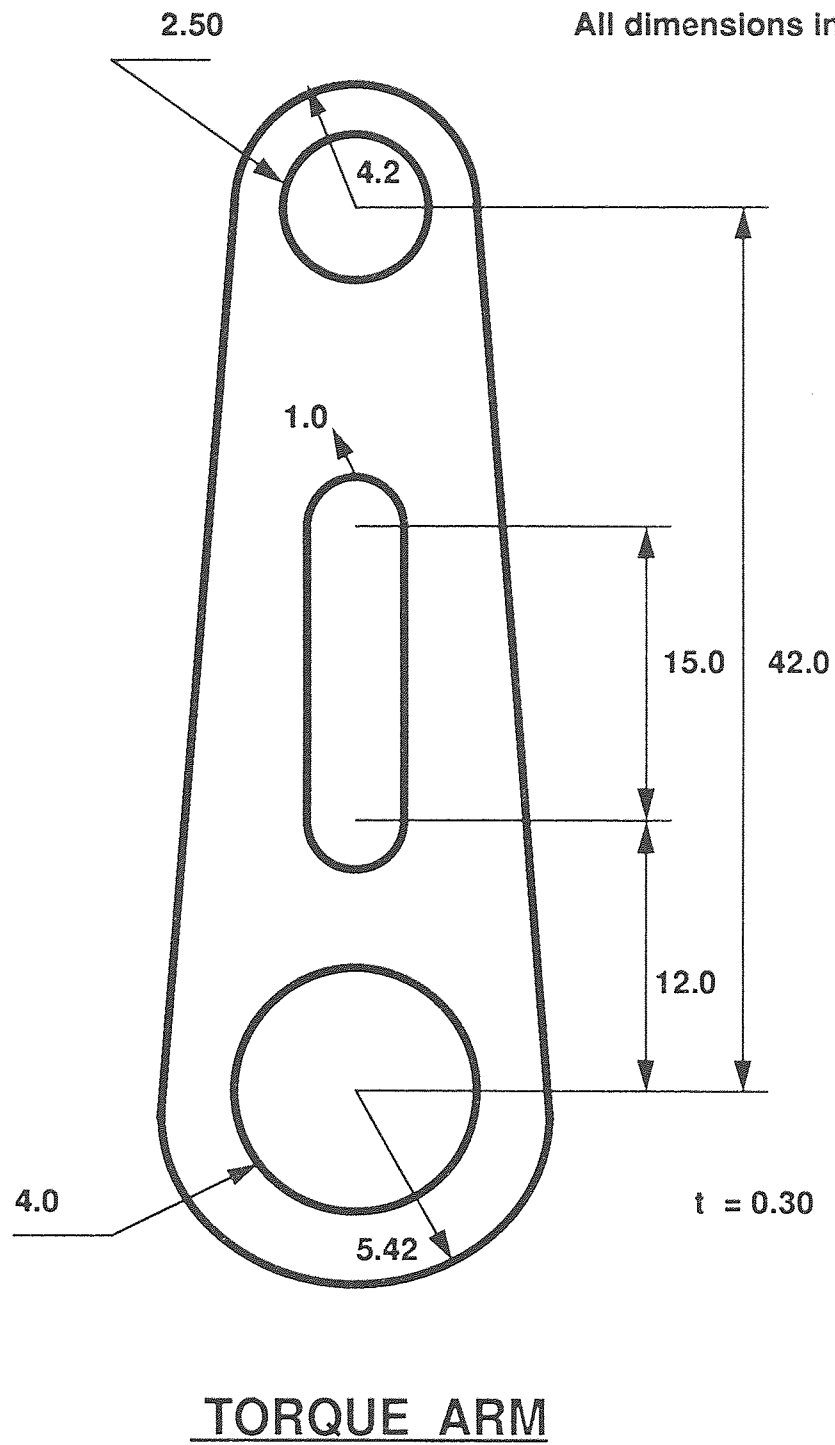
Fig 7.1 Initial design of the Torque Arm.

7.2 General Procedure of Shape Optimization using IDEAS VI

The flow chart presented in Fig 7.2 schematically shows all the steps involved in shape optimization process, using IDEAS VI. The initial geometry of the component acts as the starting point. There are two options available to the user; either to make a solid model of the given object or to proceed with the creation of the wire frame directly under Finite Element Modeling module. As this work is more related to planer structures, creation of wire frame approach was found more appropriate to initiate the process. Once the wire frame of model was finished, mesh areas were defined to mark the boundary of the region to be divided into grid of elements. Free meshing technique with isoparametric rectangular elements was used for analysis.

To provide a finer mesh in the regions of high stress concentration, specially near the holes and inner slot, local elements were also defined. After generating the finite element mesh in the active mesh area, quality checks for the elements and their respective nodes were performed. These checks ensure for element distortion within allowable limits, no coincident elements and free edges. Based on these checks adaptive refinement for the mesh could be selected, where distorted elements can be modified or in extreme cases a complete automatic remeshing process can be carried out. Before applying the necessary boundary conditions, physical and material properties are checked and modified, if necessary. Under Physical properties, the thickness of isoparametric quadrilateral thin elements is set equal to the thickness of the main part under consideration. Material properties considered were Modulus of elasticity and Mass density.

As described in section 7.1, the large end of the torque arm is constrained against the rotation and translation. To apply this condition, all the boundary nodes present on the inner edge of the larger hole(4.0 cm radius), were included in a group known as Restrain Set 1, and the allowable translation and rotation in X, Y, and Z directions were set to zero. To apply the given loads, another group of nodes was created; Load Set 1, and the
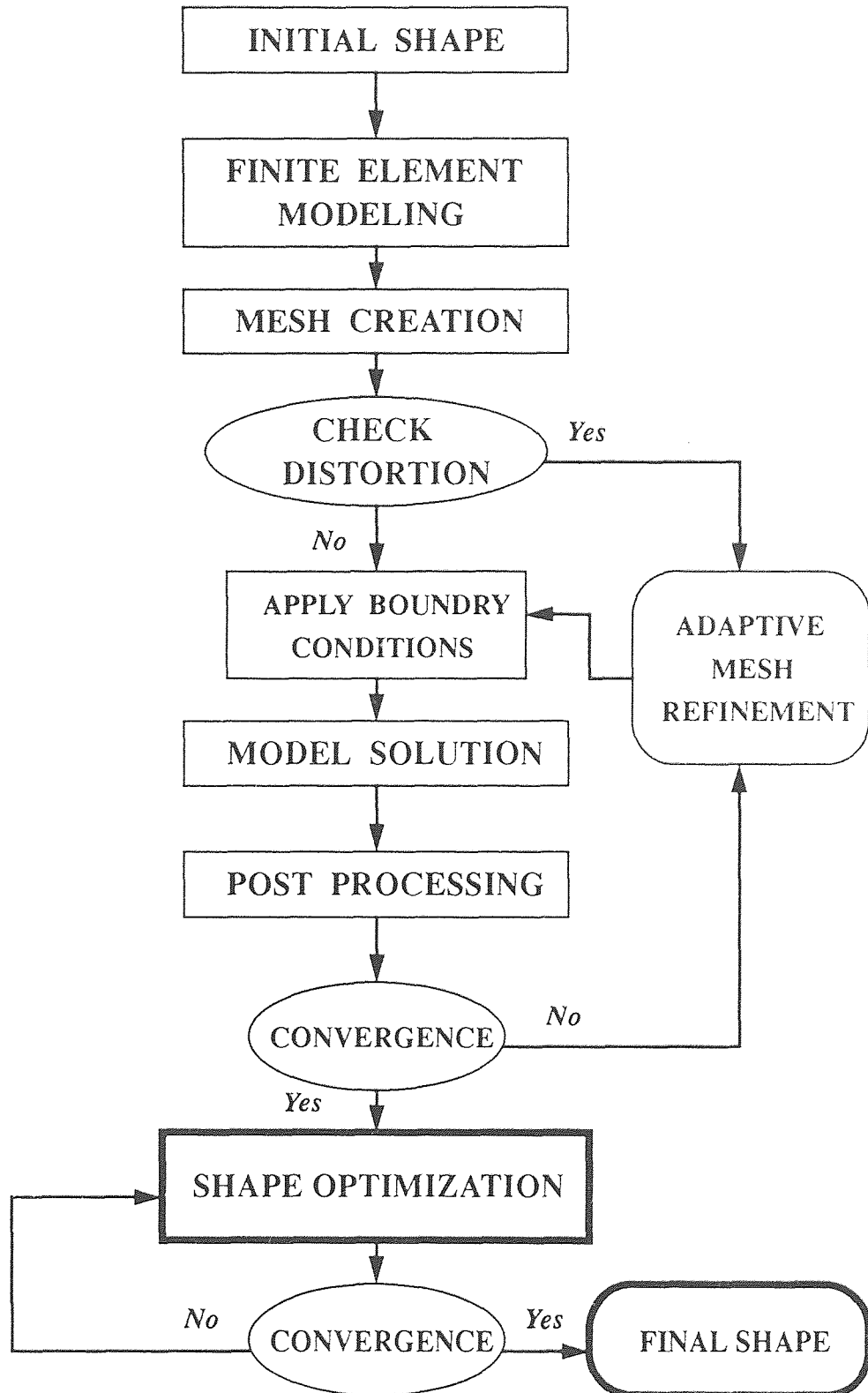
Fig 7.2 General procedure for Shape Optimization using IDEAS VI

nodes in the first quadrant of smaller hole(2.5 cm radius) were included in the set, Force in the X and Y directions were set to the input limits. Force in the Z direction and Moments in all three directions were set to zero. Next was to create a Case Set, to include the Restraint Set 1 and Load Set 1. This Case Set was then put to Model Solution task. The method selected was 'Solution No Restart' and for the output entries, Stresses, Displacements, and Strain Energy Density were picked and output mode was to store their final values.

The Model Solution was executed on The current Case Set. No errors or warnings were displayed which indicates the model is ready for Post Processing stage. During this step, main interest was to plot the stress contours and to check for the upper stress limit. The deformed geometry was also displayed and max. displacement was also checked to be in permissible limits.

In the present example of shape optimization, constraint function was defined as max. equivalent Von Mises stress and objective function as minimum weight. The model was solved again after displacing the optimization boundary nodes, which govern the actual shape of the part. Depending on the results of optimization solution, updating of node movement was carried out for the entire model, resulting in new shape and reduced weight. This procedure was repeated till enough convergence is achieved. A detailed view of the shape optimization procedure is presented in section 7.6.

This section reviews the whole process of shape optimization and highlights the major steps involved. The subsequent sections deal with these steps in detail.

7.3 Creation of Wire Frame Model

To automate the geometric modeling task, a general program was created, capable of generating arcs, splines and tangent lines, which define the boundary of an object. For the current example, the data input required is position of centers, radii of arcs and hole

diameters. To generate arcs and curves, center point, start point and end point coordinates were used. The program also takes care of the symmetry of the object about X axis.

There were two options available to the user; either to generate the two dimensional wire frame model, or to make a solid model of the same size and prepare it for mesh generation without defining even the mesh areas. Though the present work is mainly dealing with planer or thin structures, the IDEAS program could also be utilized for three dimensional objects. Figure 7.3 shows the geometric model created, using both approaches. This feature adds to the versatility of the program and could also be considered as a step closer to the integration of Shape Optimization and CAD as an important tool for Design.

As the program creates geometric entities based on the input data, the wire frame model could be modified anytime, by changing the input numeric values. This feature adds to another advantage, elimination of the need for changing optimization algorithm in case the initial design is altered. For example, if there is a need to shift the position of the holes or even the size of the holes, the corresponding changes in the wire frame model can easily be implemented.

7.4 Mesh Generation Program

To generate a finite element mesh, it is necessary to define the mesh area, a domain where elements would be created. A mesh area is well marked by the boundary curves and lines forming a closed loop. As shown in Fig. 7.1, the geometry of the torque arm could be classified as; one outer loop of boundary curves, indicating the shape of the object and three inner loops which define the holes and the elliptical slot. The effective mesh area in this case is, outer loop minus the three inner loops. This formula could be implemented in IDEAS program in the Mesh Generation task, under Mesh Area module.

As a whole, four mesh areas were defined, for every closed loop. To create a mesh area, a label or an identifier was assigned to that particular mesh area. Element type with
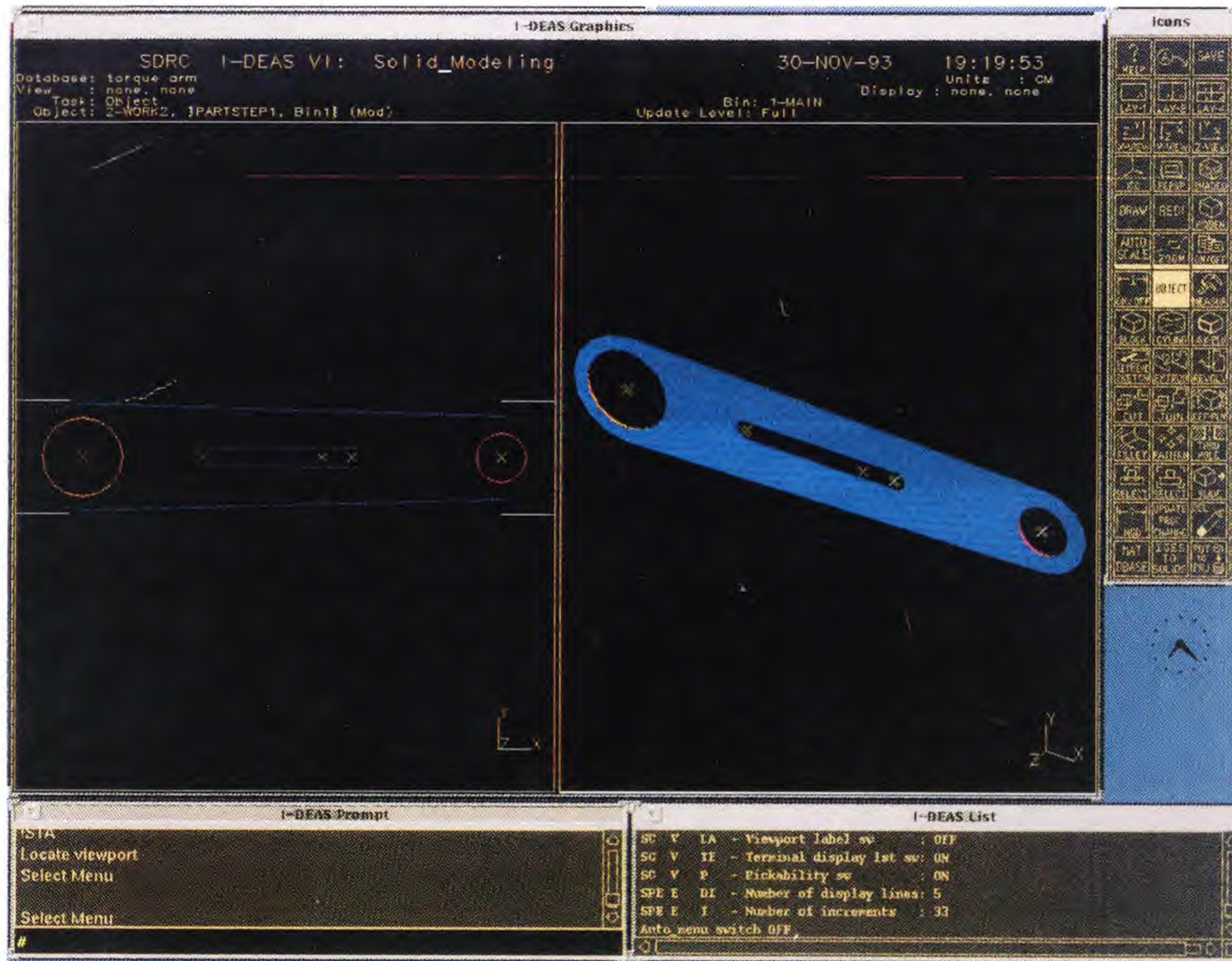
Fig. 7.3 Two approaches to generate geometric wire frame model

physical and material properties were mentioned, followed by highlighting the continuous curves forming a closed loop. The program repeated this process for every mesh area till the four independent mesh areas were obtained. These mesh areas were merged together to generate the final effective mesh area. The option to delete the parent mesh areas was also provided and could be set as default. As a part of the input data, global element and local element sizes were given. Tolerance limits for curvature based elements were also provided. The IDEAS program automatically identifies the effective region to be meshed with the help of assigned labels and generates finite element mesh of thin shell quadrilateral elements. Quality checks for generated element and nodes were performed. No coincident elements were stored as output group and distortion values were found within allowable limits. Adaptive refinement was not required at this stage. The elemental bandwidth and nodal wavefront were also optimized through the program. The weight and the volume of initial design, could be noted by checking the solid properties of elements

The Physical and Material properties were also modified based on the input values. Important physical property considered was element thickness, which was set to the thickness of torque arm. Under material properties comes the modulus of elasticity, poisson's ratio, and mass density. The numeric values defining property table were stored as a group and could be retrieved and modified by referring to the corresponding group.

7.5 Boundary Conditions Application and Model Solution

Next important step towards the finite element model solution is to apply the boundary conditions. These constraints were applied through two sets namely, Restraint Set 1 and Load Set 1. The restraint set was a group of boundary nodes at the larger hole and the allowable movement in form of translation and rotation was set to zero in all three principal directions. The node selection method implemented was to pick all the nodes on the circle(4.0 cm radius), by defining enclosed Screen Area.

Defining the load set was relatively difficult and approximation techniques were put into use. According to the problem statement, all the forces act in the positive direction of X and Y axes, thus only the nodes in the first positive quadrant were assumed to be affected and subjected to loading. Load applied in the X direction was set to 2789 N and 5066 N in the Y direction. No moments in any principal direction were considered and the force applied in the Z direction was also set to zero. To check the error caused, due to approximation in loads, various combinations of loads sets were created and combined with Restraint Set 1 to generate different Case Sets. Verification of results indicated error to be less than 5 %. The Model Solution was executed on the Case Set 1. Solution method opted for was Solution No Restart. The Output selection entities included Stresses, Displacements, and Strain Energy Density. The values of these entities were put to store for Post Processing task.

The Model Solution could also be run by selecting the solution method as Verification Only, to list any error or warning occurred during the solution process. In the current model solution no errors or warning were reported. Based on the Output Selection entities, three Analysis Datasets were produced. Under Post Processing task each dataset was plotted and recorded for optimization process. More attention was given to the contour plot of equivalent Von Mises stress, the constraint function for the shape optimization. Fig 7.8 shows the stress contour with Von Mises stress as a data set, for the initial design of the torque arm. Deformed geometry was also plotted to note the max displacement. The purpose of generating Strain Energy Density dataset was to make a basis for adaptive mesh refinement during optimization procedure, if required.

7.6 Shape Optimal Design of Torque Arm

The important steps involved in shape optimization task are schematically portrayed in Fig 7.4. The analysis of the initial design presents a clear picture of low stress regions and the areas, from where boundary nodes could be oriented to change the shape.
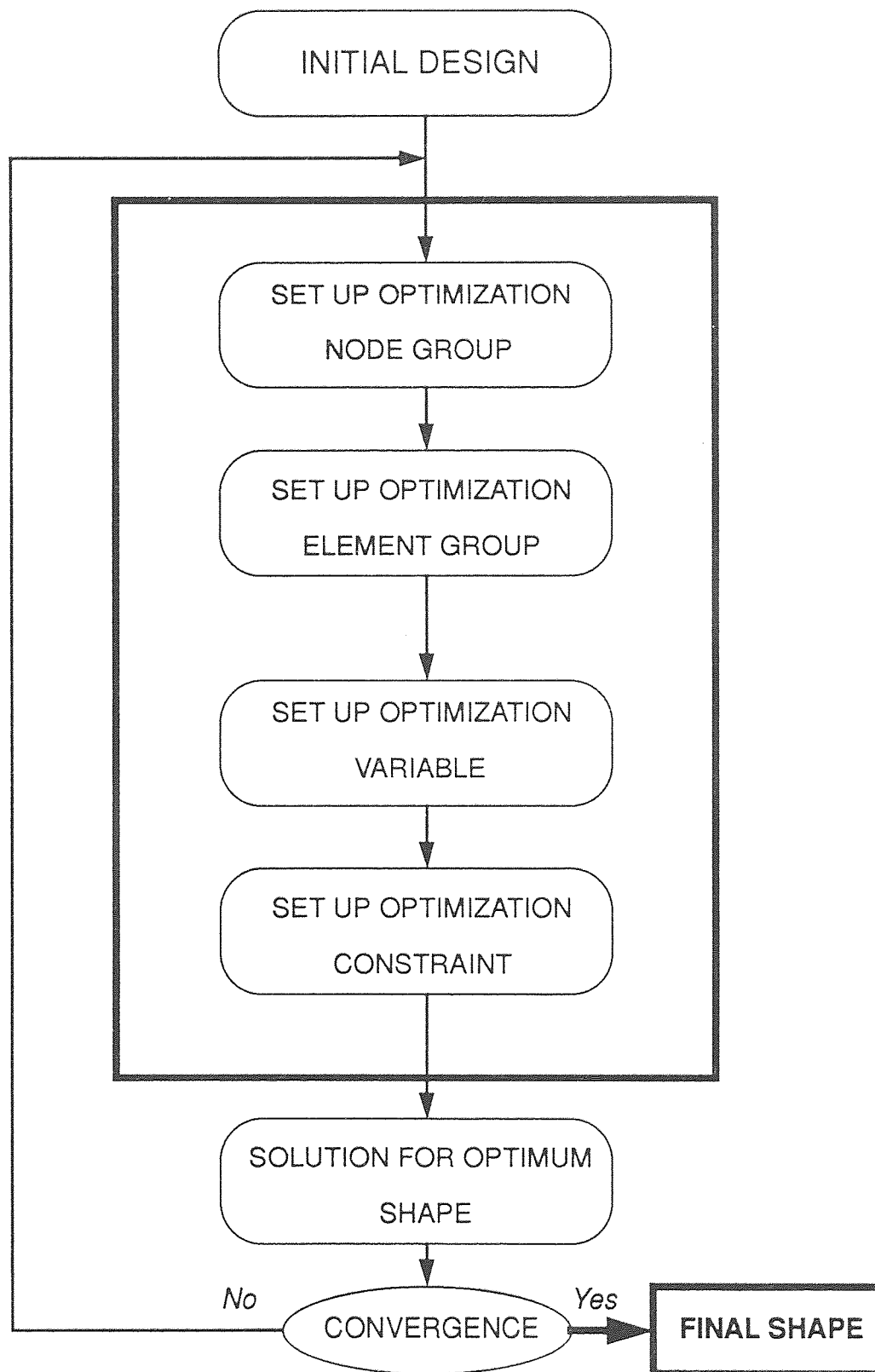
Fig 7.4 General steps of Shape Optimization process using IDEAS VI

Movement of boundary nodes was selected as main approach to define the variable shape and various concepts to define the nodal movement were considered. Definition of shape optimization problem could be divided into four independent but related activities:

1. Set up optimization node group

2. Set up optimization element group

3. Set up optimization variable

4. Set up optimization constraint

The optimization node group is a collection of all those boundary nodes subjected to some form of nodal movement. The group is stored under a label, used for identification purposes. The nodes could be added, removed and their features could be modified within a group. Different methods to define movement of nodes include:

1. Movement along a vector

2. Movement of nodes as a function to initial nodal position

3. Movement in a radial direction

4. Restrained movement of nodes

Figures 7.5, 7.6, and 7.7 describe the various examples for nodal movement. The nodal movement as a function of initial nodal position has been shown in fig 7.5, where the boundary nodes on the right wall of the object shift according to the governing function. In cases of objects having grooves, notches or recesses, method of restrained movement of nodes (Fig 7.7) is applied to maintain the original shape format. The objects with slots or holes are subjected to radial nodal movement (Fig. 7.6) to change the shape of such geometric entities.

In current problem the nodes were oriented using Vectors and Radial Movement approaches. For the nodes present on the outer boundary, the movement was defined along a vector having its origin as the initial position of the node, and magnitude as a percentage of the chord length. the advantage of using this method was that it provides a better control to regulate the nodal displacement. Nodes present near the smaller end were
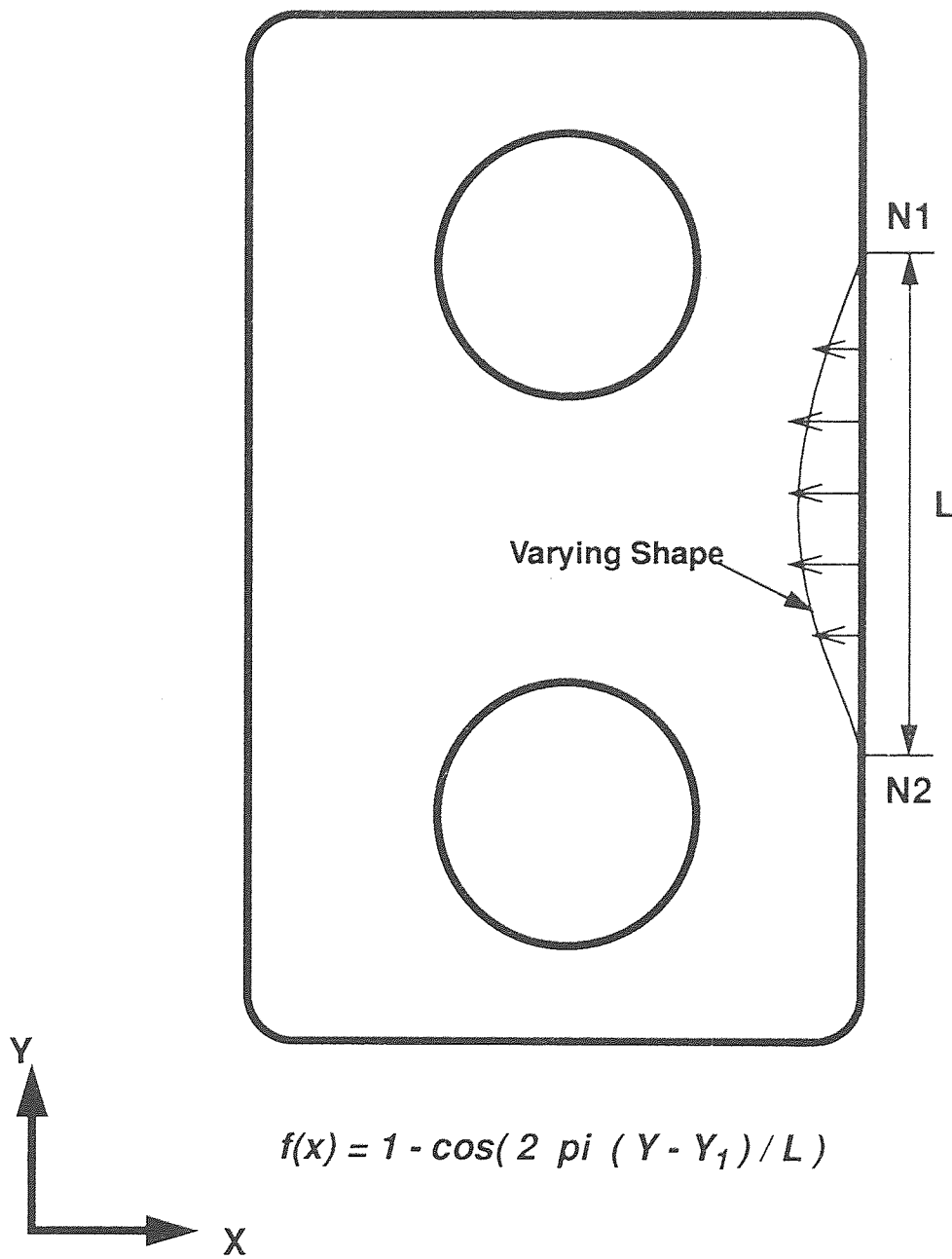
$$f(x) = 1 - cos(2\ pi\ (Y - Y_1) / L)$$

Fig 7.5 Movement of nodes as a function of initial nodal position
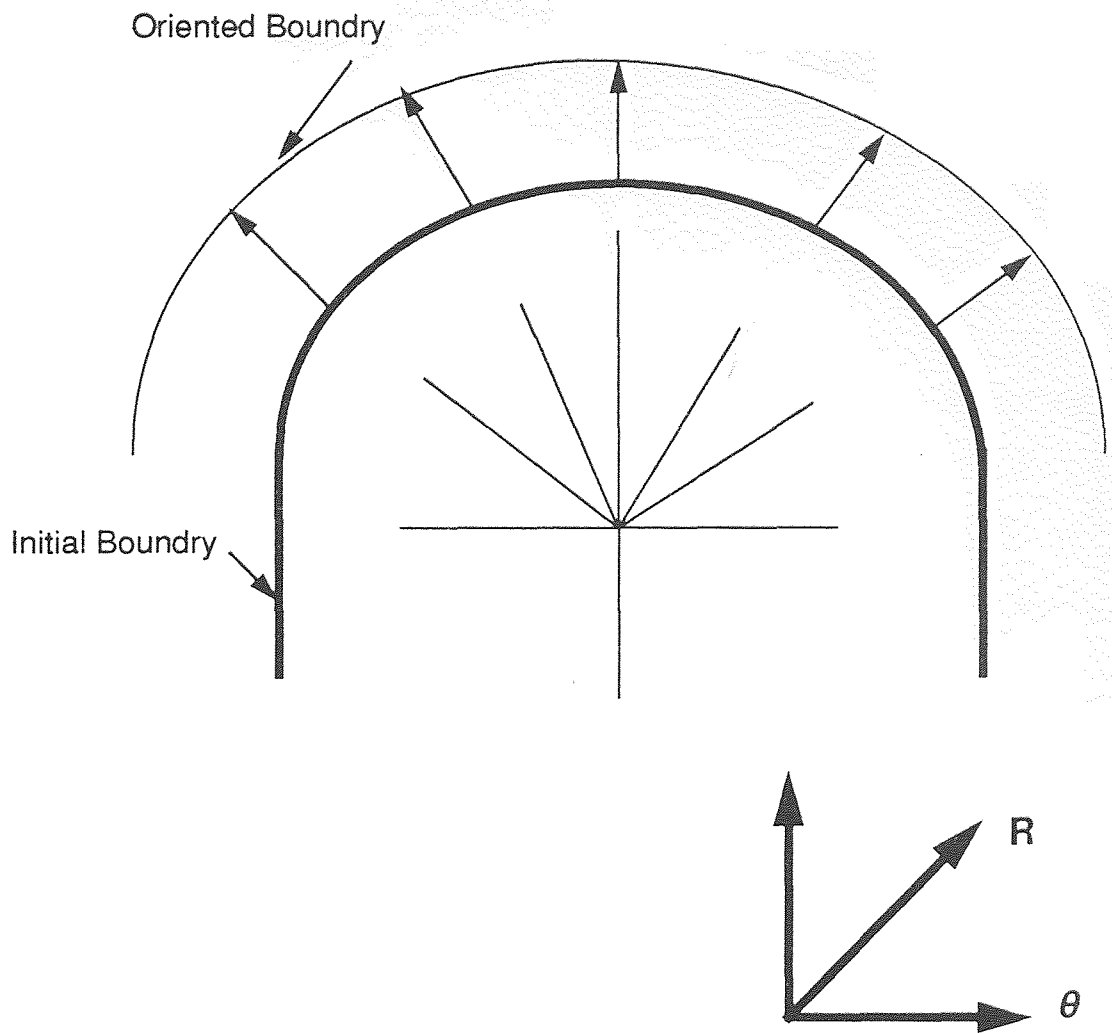
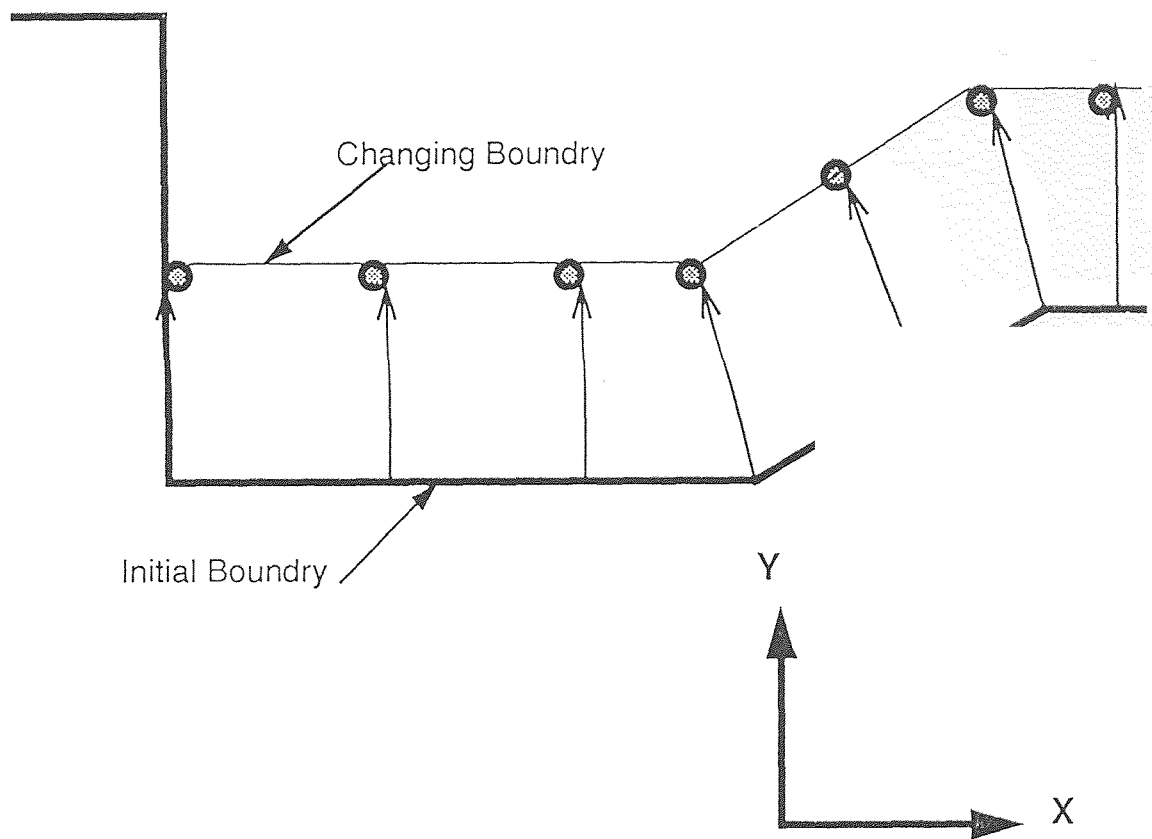Fig 7.6 Movement of nodes in radial direction

Fig 7.7 Shape variable set up with restrained node movement.

subjected to more inner displacement as compared to nodes on the opposite end. Defining respective movement through independent vectors surely provides a solution to that. Moreover, it was easy to maintain the symmetry of the object even when shape was varying.

Radial movement was needed to displace the nodes present on the inner slot. A global cylindrical coordinate system was defined at the center of the arc, and nodes were oriented in the radial direction to alter the shape of the slot. Fig 7.6 explains the concept of radial movement of nodes. After defining all the nodes to be moved, the affected elements could be sketched to view the altered shape and new positions of the nodes. A rough idea of element distortion could also be obtained from this sketch.

The optimization element group contains the elements which would be associated with the optimization constraint. In current example of torque arm, all the elements were selected to form optimization element group. These elements were grouped under the free selection procedure. Other grouping criteria could be based on similar physical or material properties or a combination of both. A label was also assigned for reference uses. As in this case, every optimization solution would contain all the elements, a permanent group of elements could be formed to avoid setting up optimization element group each time.

Shape redesign was selected as optimization variable and redesign limits were set parametrically from -1 to +1. Setting up the optimization constraint was the most important step in the shape optimization process. In present case, upper limit for equivalent Von Mises stress was set to 80,000 N/cm^2 as optimization constraint. While setting up optimization constraint, load set and element group label ranges were also included.

A few options of Solution Control were selected to execute the optimization solution. The solution method to be used was selected as Linear Statics and Shape redesign. Iteration control was adjusted to default settings. Output control settings were
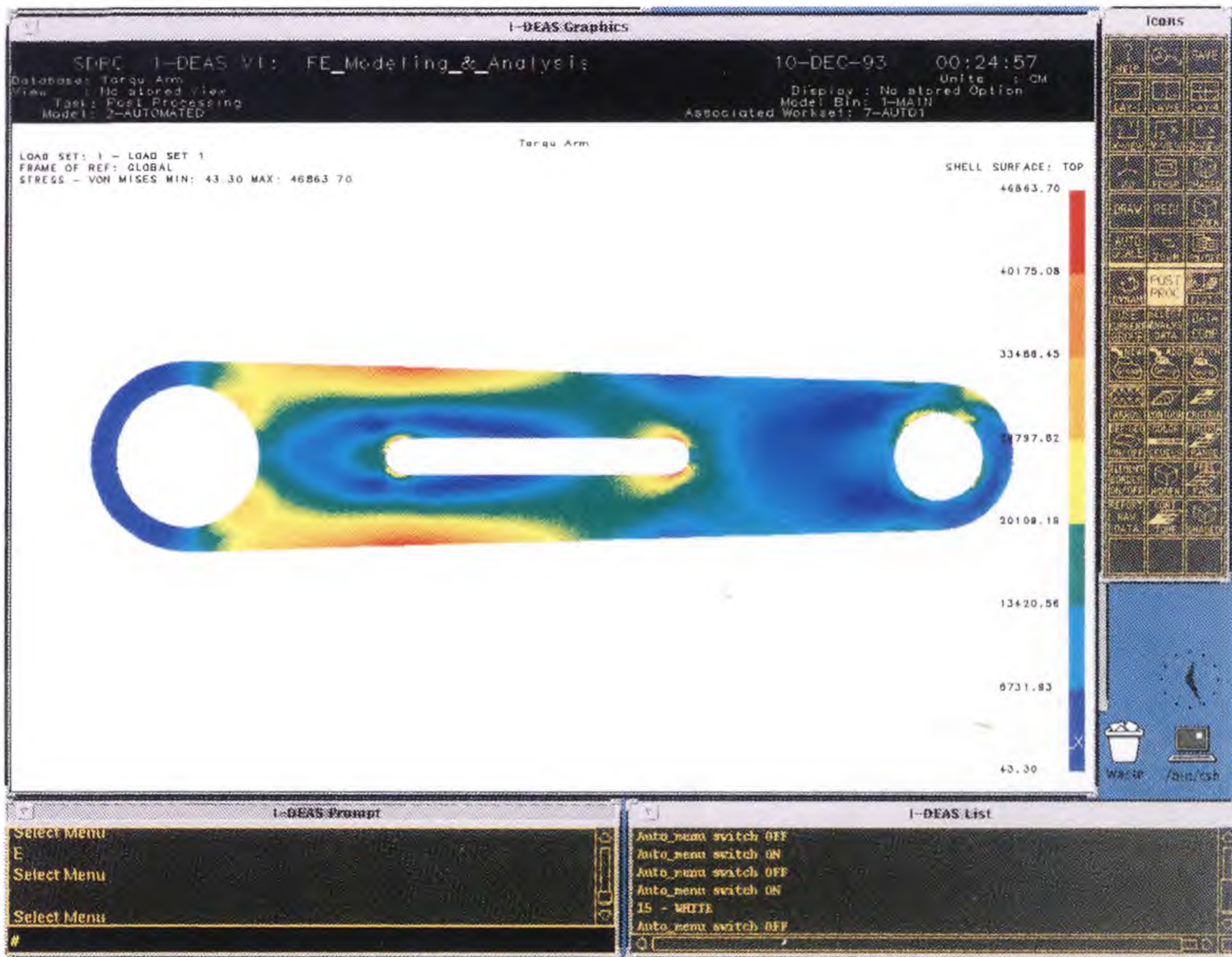
Fig 7.8 Stress contour plot for initial design

adjusted as it was done in Model Solution. After this, the model was put to optimization solution based on set up parameters. The final results were displayed and plotted for analysis. Based on satisfactory results, global node updating was carried out to change the existing design shape. The new model was put to model solution to obtain the values for major datasets. Maximum equivalent Von Mises stress was plotted for the new solution.

The distortion values for the elements were checked and if necessary, adaptive mesh refinement was executed. At certain instances, mesh refinement also resulted in increased accuracy in final results. Based on element distortion, an option to completely remesh the object was also available. This process was repeated till results exhibited considerable convergence and as a whole twelve iterations were executed to achieve considerable weight reduction.

A brief review of the implementation of shape optimization process has been depicted in this chapter. Main advantages of using interactive IDEAS programs are discussed. The programmability feature adds to the flexibility and versatility of application area. The highlights of the whole shape optimization process with major steps involved are also described in detail. With the help of illustrations and figures, various methods of nodal movement are explained. The concept of linking CAD geometry to mesh generation task is also taken into consideration and special attention is paid to use the IDEAS program for mesh generation and refinement. The following chapter deals with the discussion of results and conclusions withdrawn.

CHAPTER 8

RESULTS AND CONCLUSIONS

8.1 Explanation of Results

Fig. 8.1 shows the solid model of torque arm before the shape optimization process was executed. The initial design of the torque arm had a weight of 918 gms. Fig 8.2 describes the wire frame geometric model of the initial design, which was created using the IDEAS program to generate geometrical entities. It is clearly portrayed in this figure the shape and location of holes and inner cutout. Figure 8.3 shows the Von Mises equivalent stress distribution in stress contour format for the initial design of the torque arm.

As indicated by the color bar, red and orange areas highlight the maximum stress zones. On the other hand, dark and light blue colors identify the low stress zones. For the analysis of initial design, the global element size of 0.8 cm was used to have higher accuracy. Due to small element size, the number of elements and the nodes increased considerably, thus leading to long processing time. For further analysis, the global element size was selected to be 1.3 cm, which was a good compromise between adequate accuracy and processing time for model solution. The maximum stress for the initial design was noted to be 4.68E04 N/cm2. Fig. 8.4 shows the plot for stress distribution over the nodes present in the effective mesh area for the initial design.

Fig. 8.5a shows stress contour diagram after the second iteration of optimization process was applied. In this step, the boundary nodes near the smaller end were shifted using the vector nodal displacement method and the nodes present on the inner slot were moved using radial nodal movement method. As a result of this global nodal displacements, the inner slot was altered to the similar shape of outer boundary curves of the torque arm. The nodal displacement in the blue zone near the smaller end caused a neck formation. The maximum stress in this case was recorded to be 4.97E04 N/cm2,
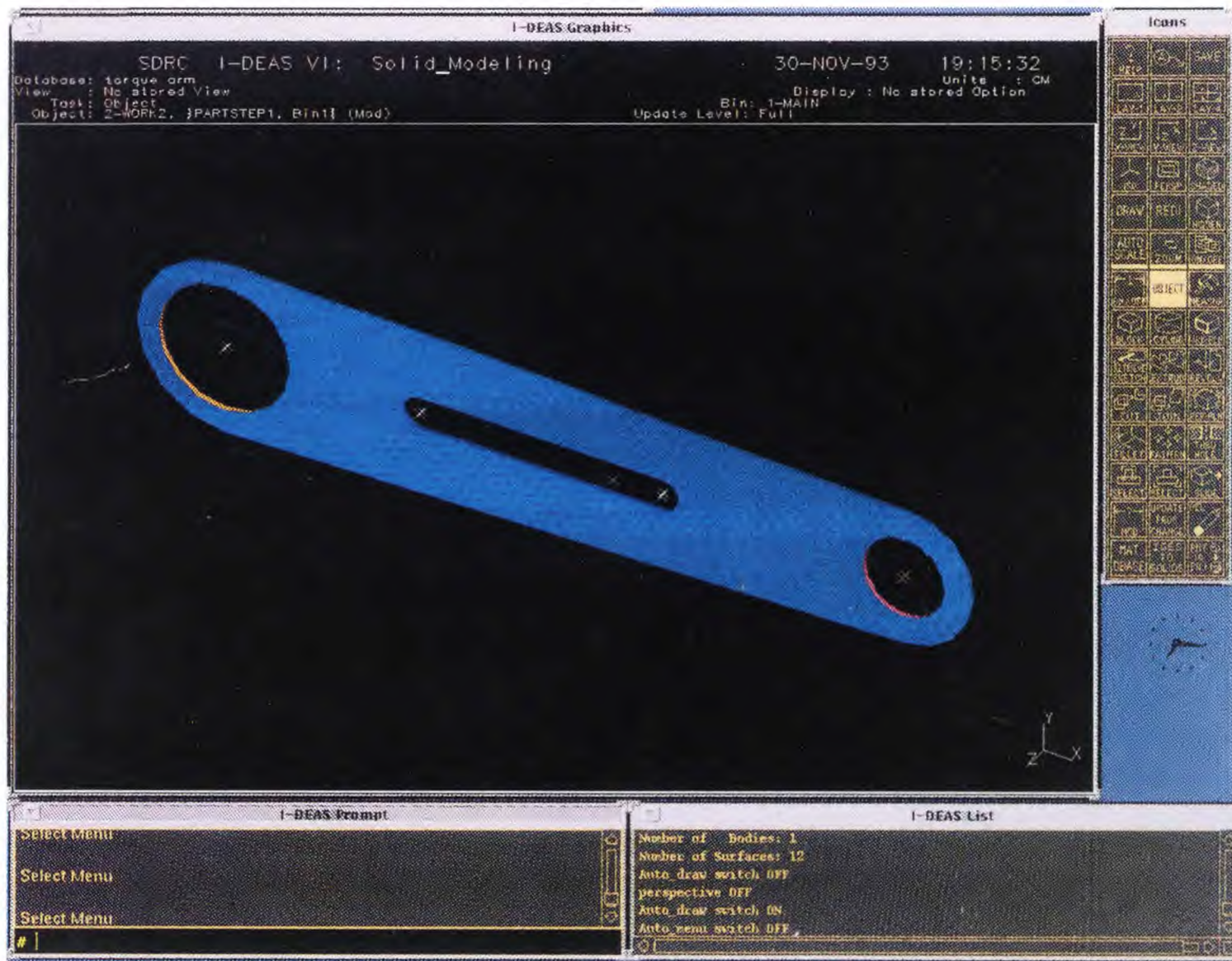
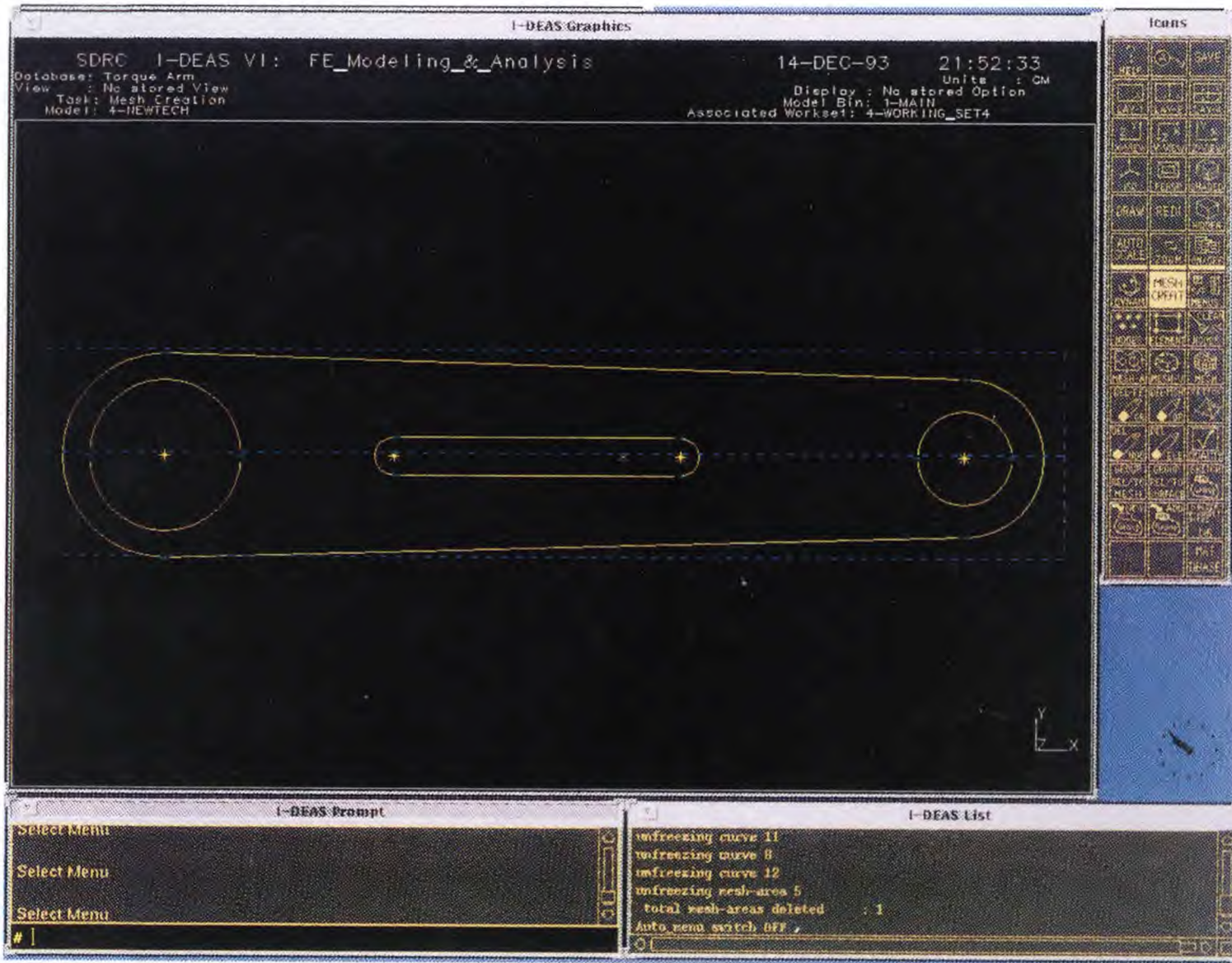Fig.8.1 Solid model for initial design of torque arm

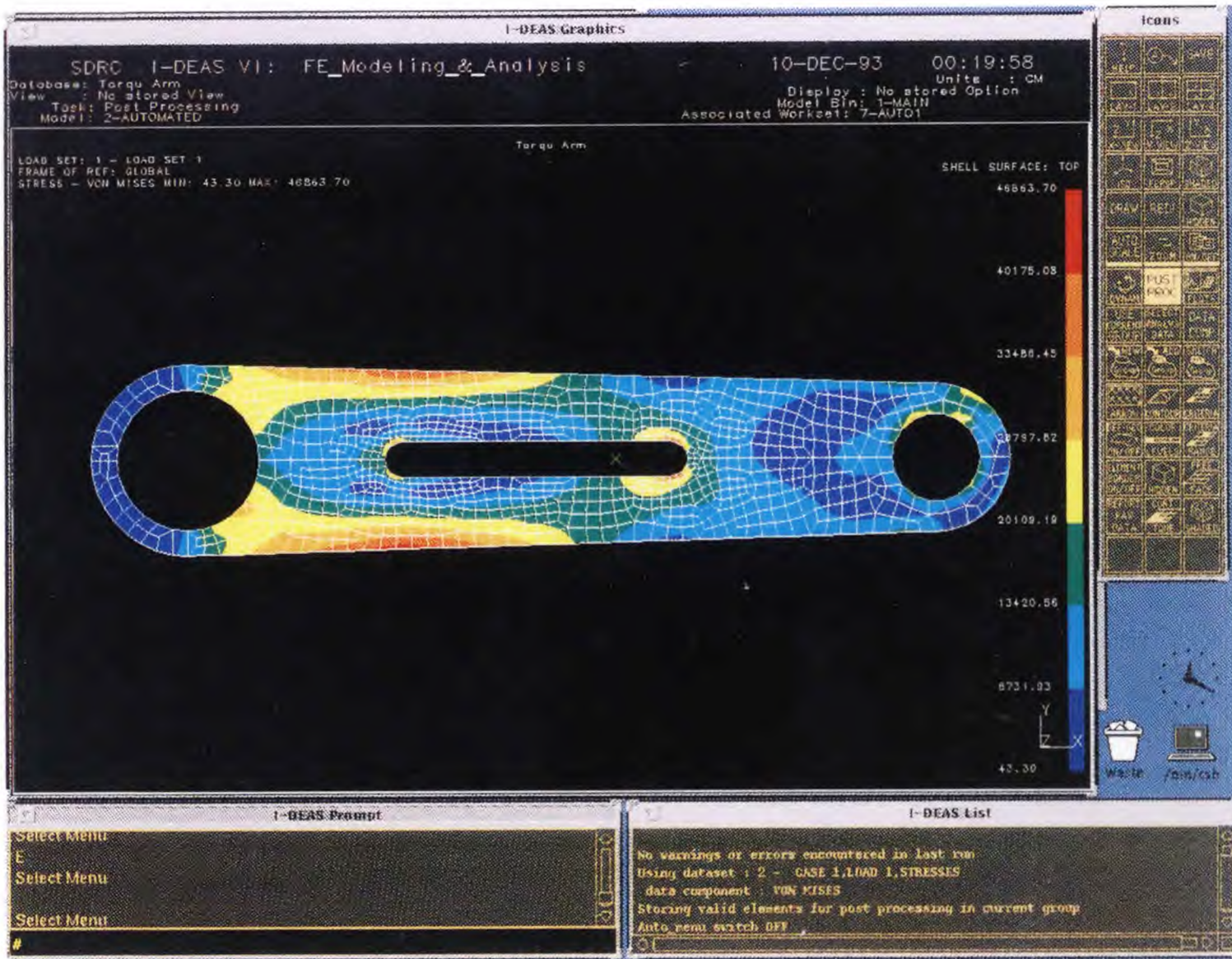Fig 8.2 Creation of geometric wire frame model for initial design

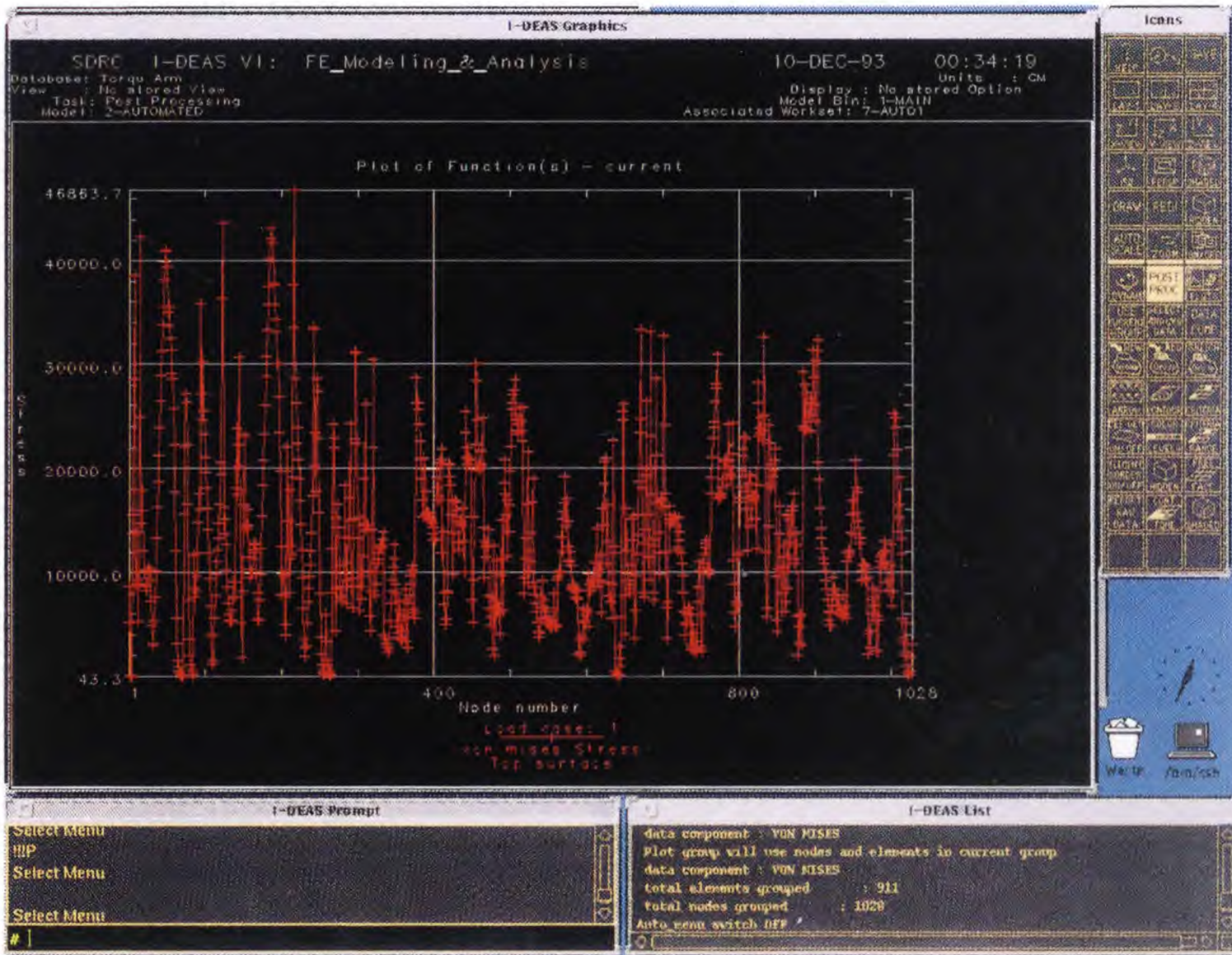Fig 8.3 Stress analysis for initial design of torque arm

Fig 8.4 Maximum stress plot over the nodes present in the mesh area

which is far below the stress constraint limit. The weight for this design case was calculated to be 850 gms.

To increase the length of inner slot, nodes towards the smaller end were translated using vector nodal displacement method. The neck formation near the smaller end was further deepened. The resulting stress distribution diagram is presented in Fig 8.5b. The element distortion resulted in higher stress values. The maximum stress value was noted about 5.74E04 N/cm2. The weight for this design was 815 gms. To eliminate the effect of element distortion, adaptive mesh refinement techniques were applied, causing a drop in maximum stress value to 5.68E04 N/cm2. A slight reduction of 10 gms in structural weight was observed. Fig. 8.5c indicates the stress distribution plot for fifth step of shape optimization process. The weight was reduced to 775 gms in this case, and mesh refinement further drops the maximum stress value.

Further steps of shape optimization process included widening of inner slot, radial expansion of slot end near the large hole, and the refinement of neck formation near the smaller end. The finite element mesh was regenerated as the higher elemental distortion occurred during these steps. Fig. 8.5e describes the stress contours plotted for the changed shape in which the inner slot was further changed and smoothening of smaller end neck was done. The weight was reduced to 688 gms, and maximum stress indicated was about 6.57E04 N/cm2.

Further orientation of nodes on the smaller end caused considerable increase in the maximum stress. So it was decided to trim some of the material from large end , where low stress zone existed. Fig. 8.5f shows the stress distribution plot for this step. Some of the boundary nodes near the middle section were also oriented to normalize the effect of stress concentration. The weight of this shape was 663 gms, and maximum Von Mises equivalent stress was reported 6.78E04 N/cm2, still within the allowable limits.

Fig 8.6 shows the final shape of torque arm. The nodes displaced in previous step were further smoothened. The elemental distortion values were checked and found within
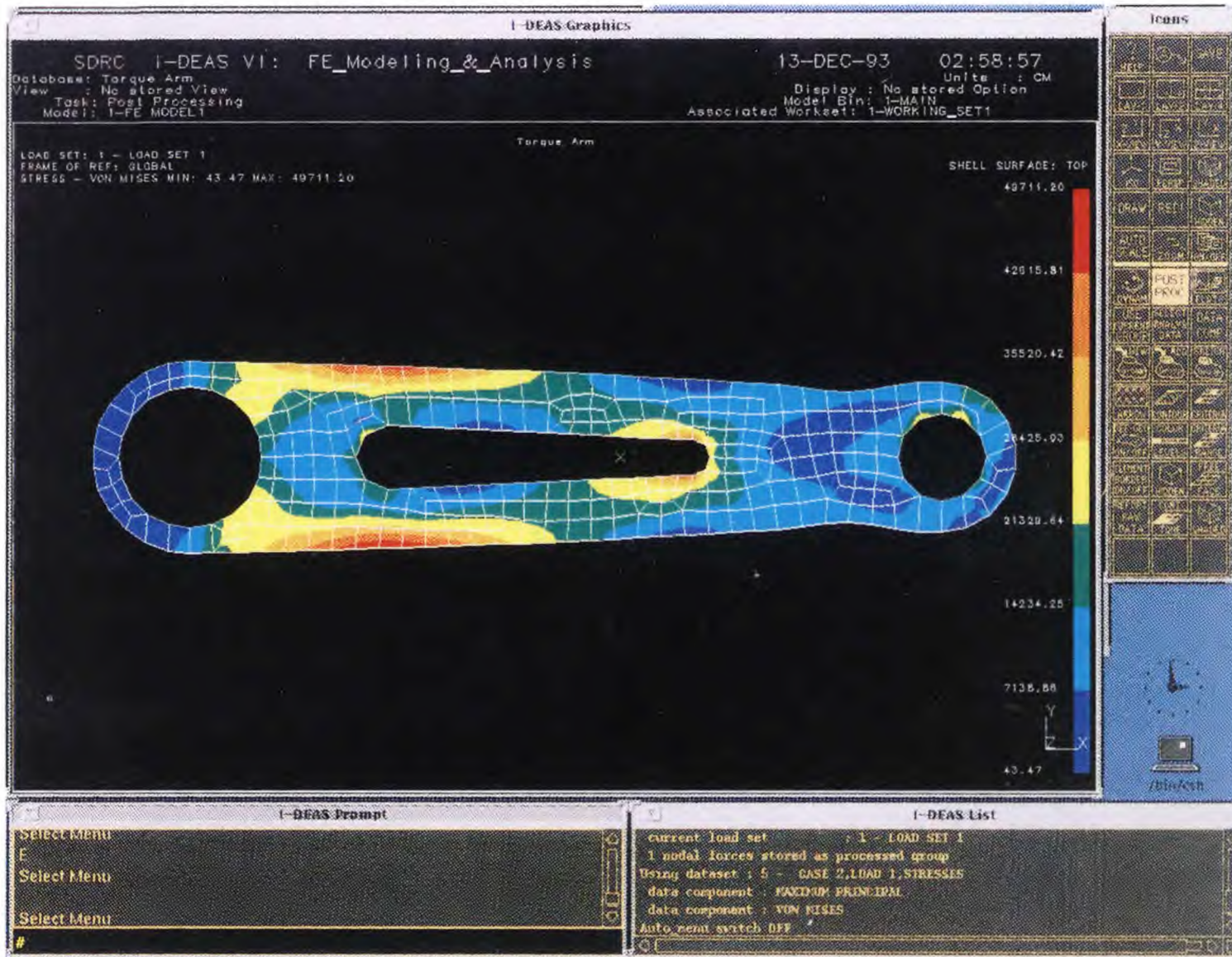
Fig 8.5a Stress contour plot for the intermediate step of shape optimization
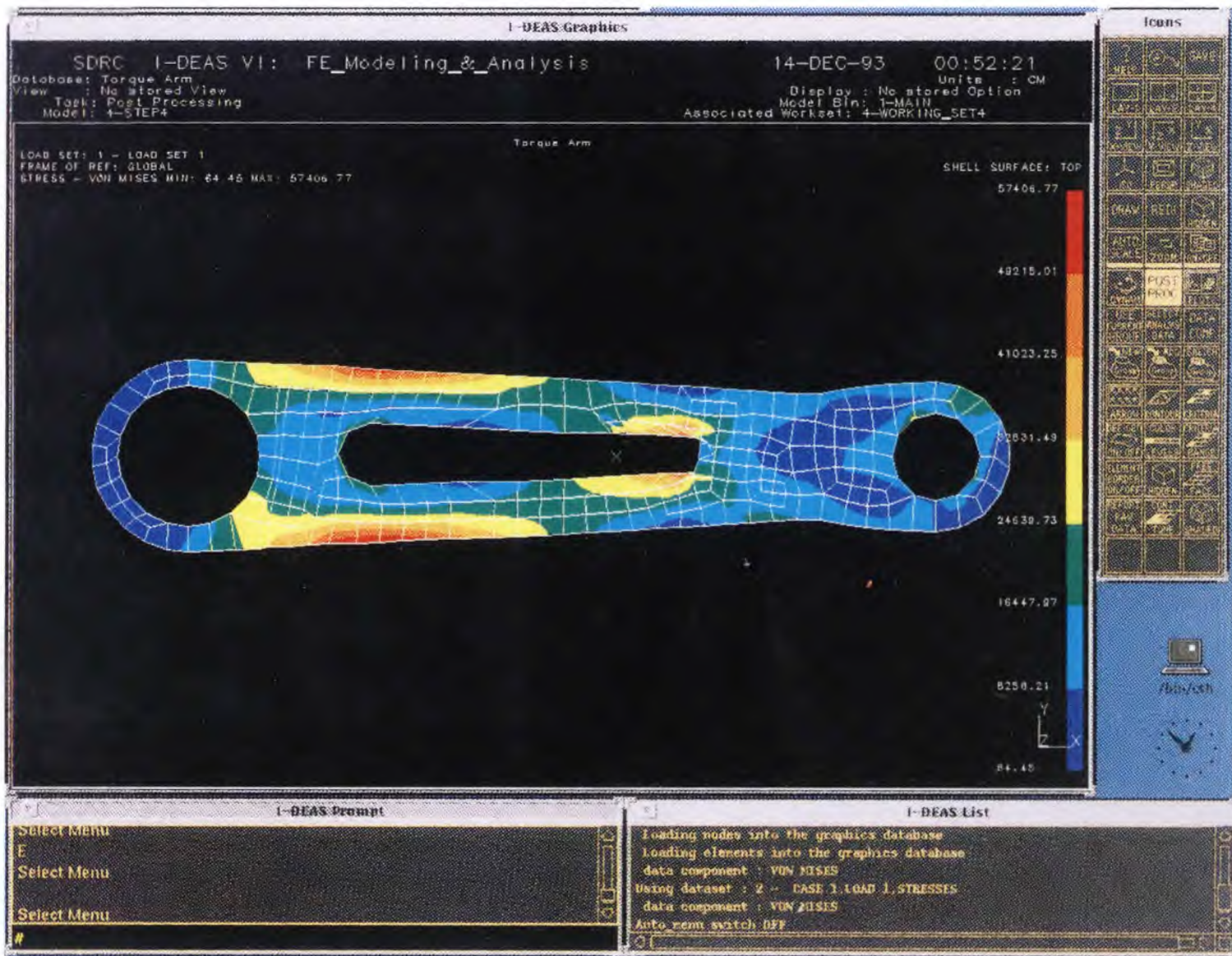
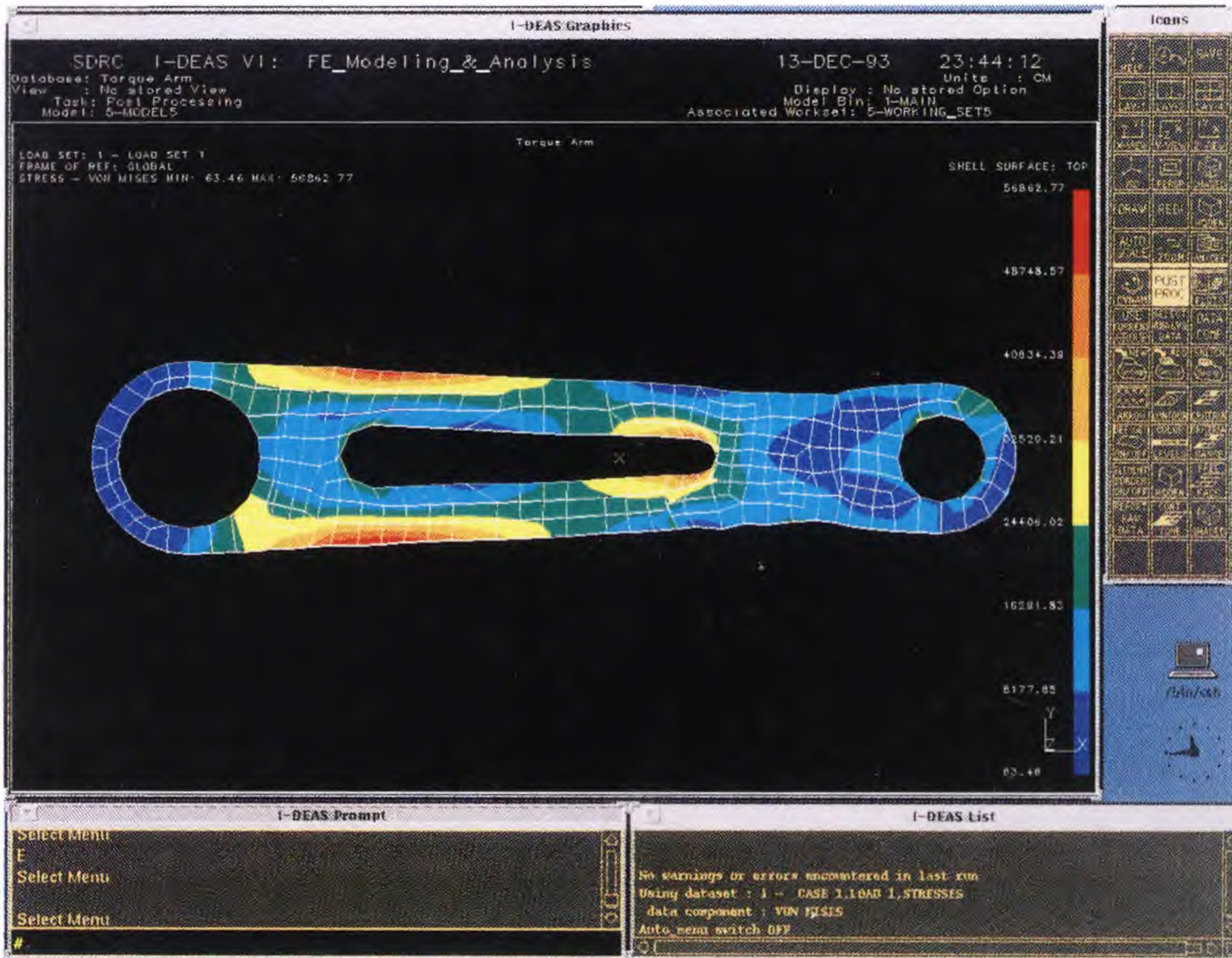Fig 8.5b Stress contour plot for the intermediate step of shape optimization

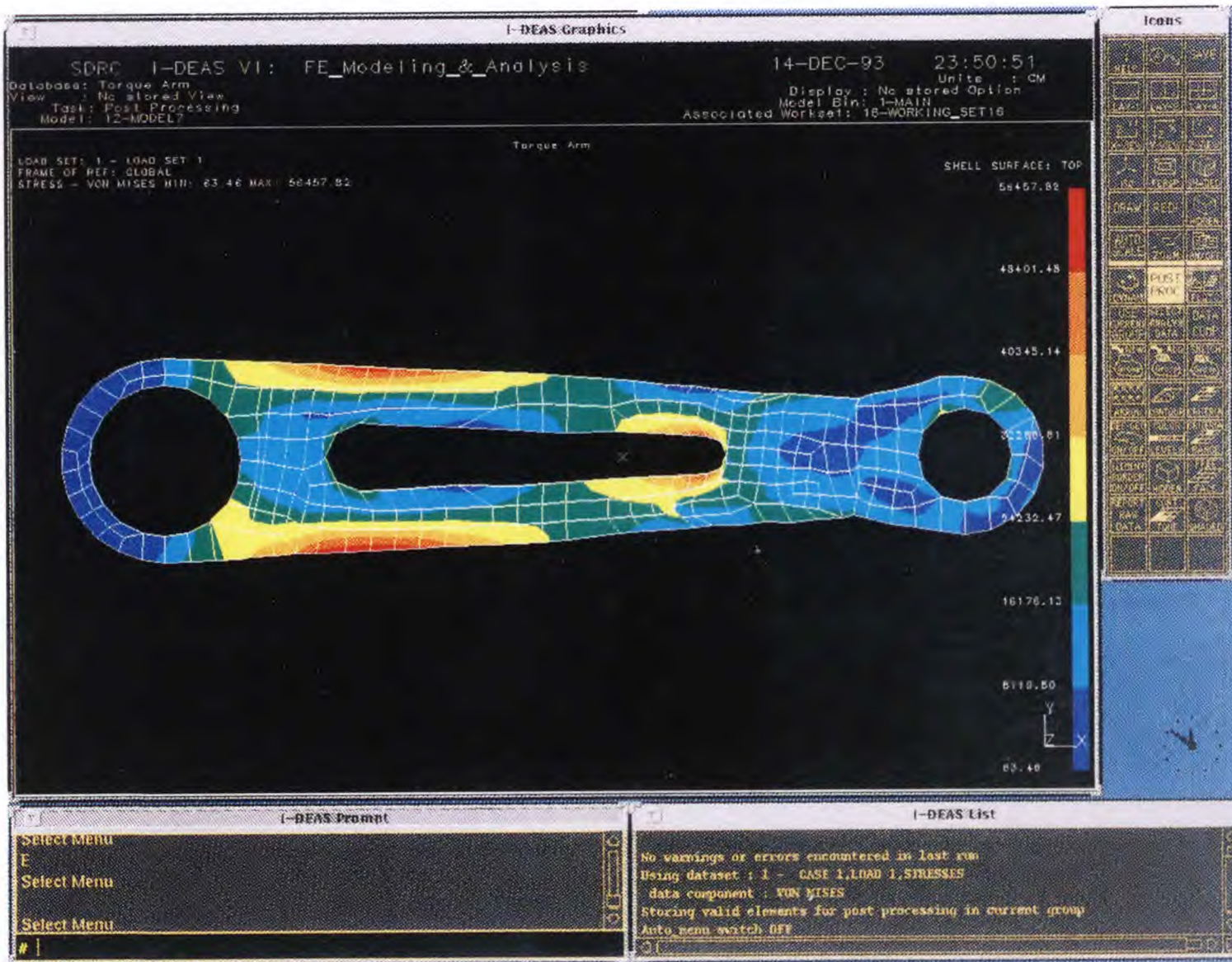Fig 8.5c Stress contour plot for the intermediate step of shape optimization

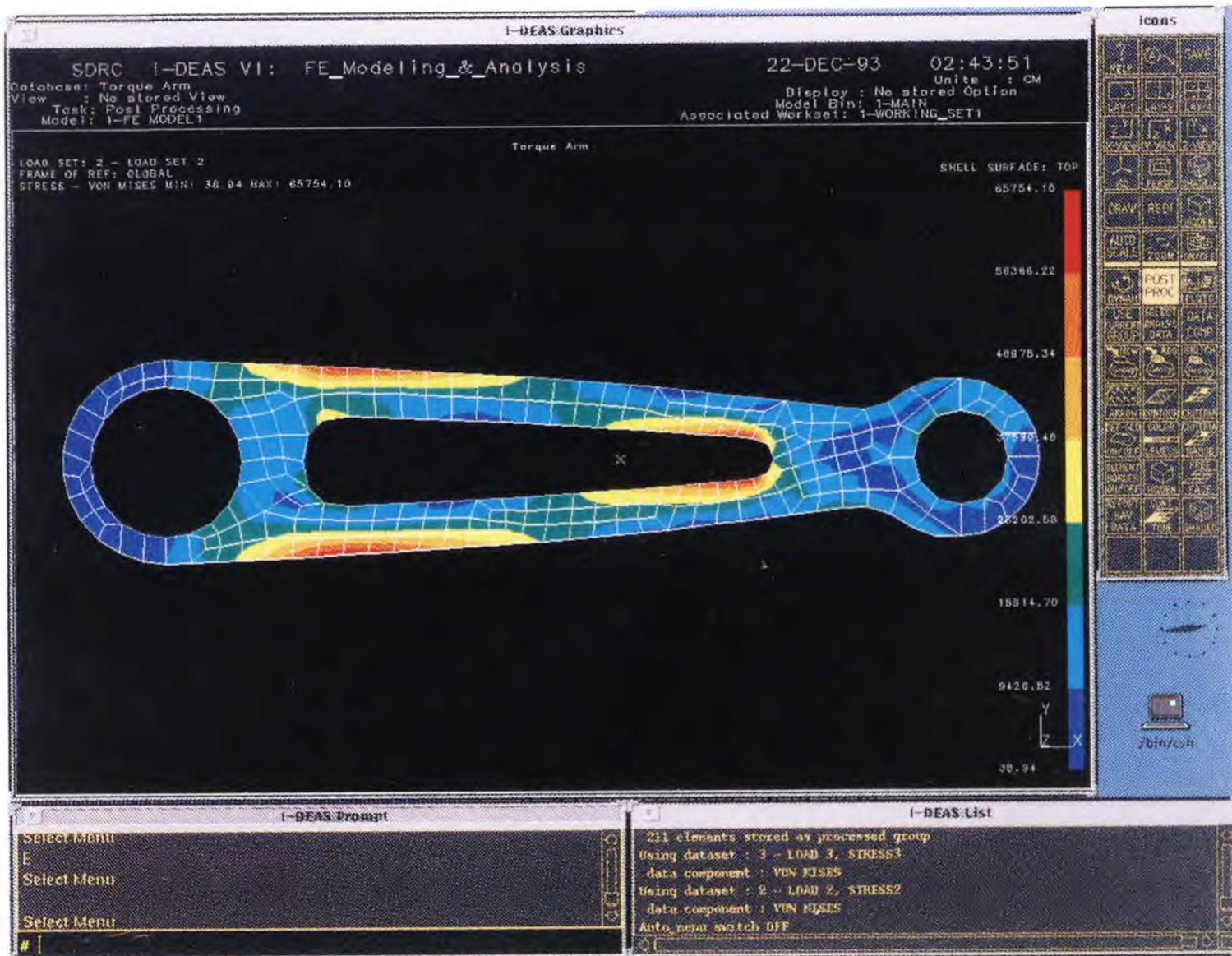Fig 8.5d Stress contour plot for the intermediate step of shape optimization

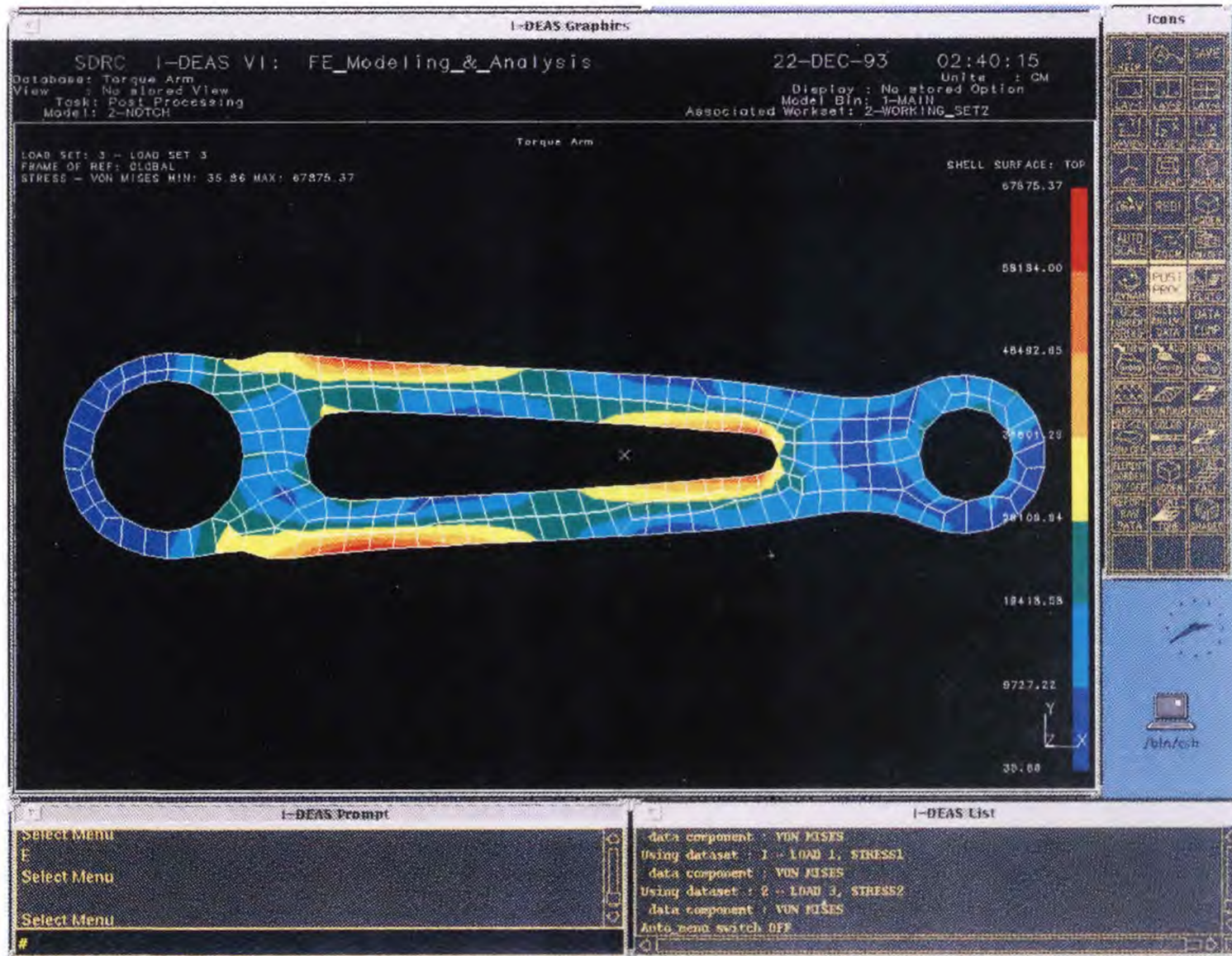Fig 8.5e   Stress contour plot for intermediate step of shape optimization

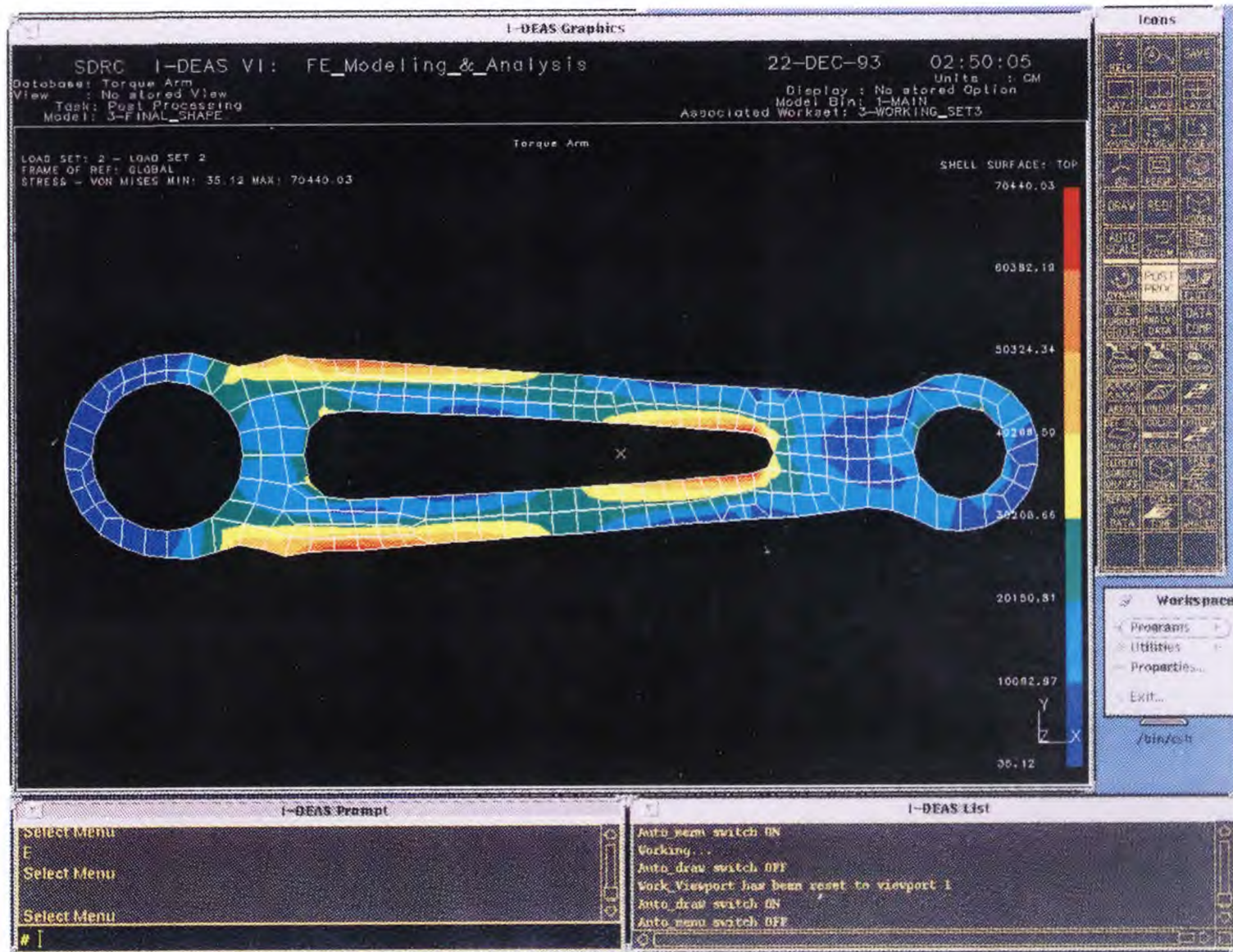Fig 8.5f Stress contour plot for intermediate step of shape optimization process

Fig 8.6 Stress contour plot for the final design

allowable range. The maximum stress value was 7.04E04 N/cm2 and the final weight was reduced to 655 gms, resulting in net weight reduction of 263 gms. Fig 8.7 shows the plot, indicating the weight reduction pattern over the shape optimization process.

8.2 Conclusions

The programmability feature of IDEAS was implemented to develop a semi automatic algorithm for shape optimization, of planar structures. Interactive programs were generated to create geometric wire frame models, to integrate geometric entities with mesh areas to merge mesh areas, to generate finite element mesh, to refine finite element mesh, to set up boundary conditions and finally to control the shape optimization parameters. However, selection of nodes to be oriented during the boundary change and identification of lower stress zones and magnitude of nodal displacement is done manually. The attempt was made to integrate different phases of shape optimization process with these programs. Various techniques to represent the boundary shape were considered and different approaches to automated mesh generation and refinement were also studied.

To demonstrate the developed algorithm, a practical example of automotive rear suspension torque arm was selected for shape optimization. A net reduction of 28.6% in structural weight was successfully achieved. Major advantages of this method includes flexibility and compact geometric description capability. The method also provides an integrated approach to link the geometrical information from CAD module to automatic mesh generator. This advantage could be of considerable importance when implementing industrially oriented shape optimization capabilities.
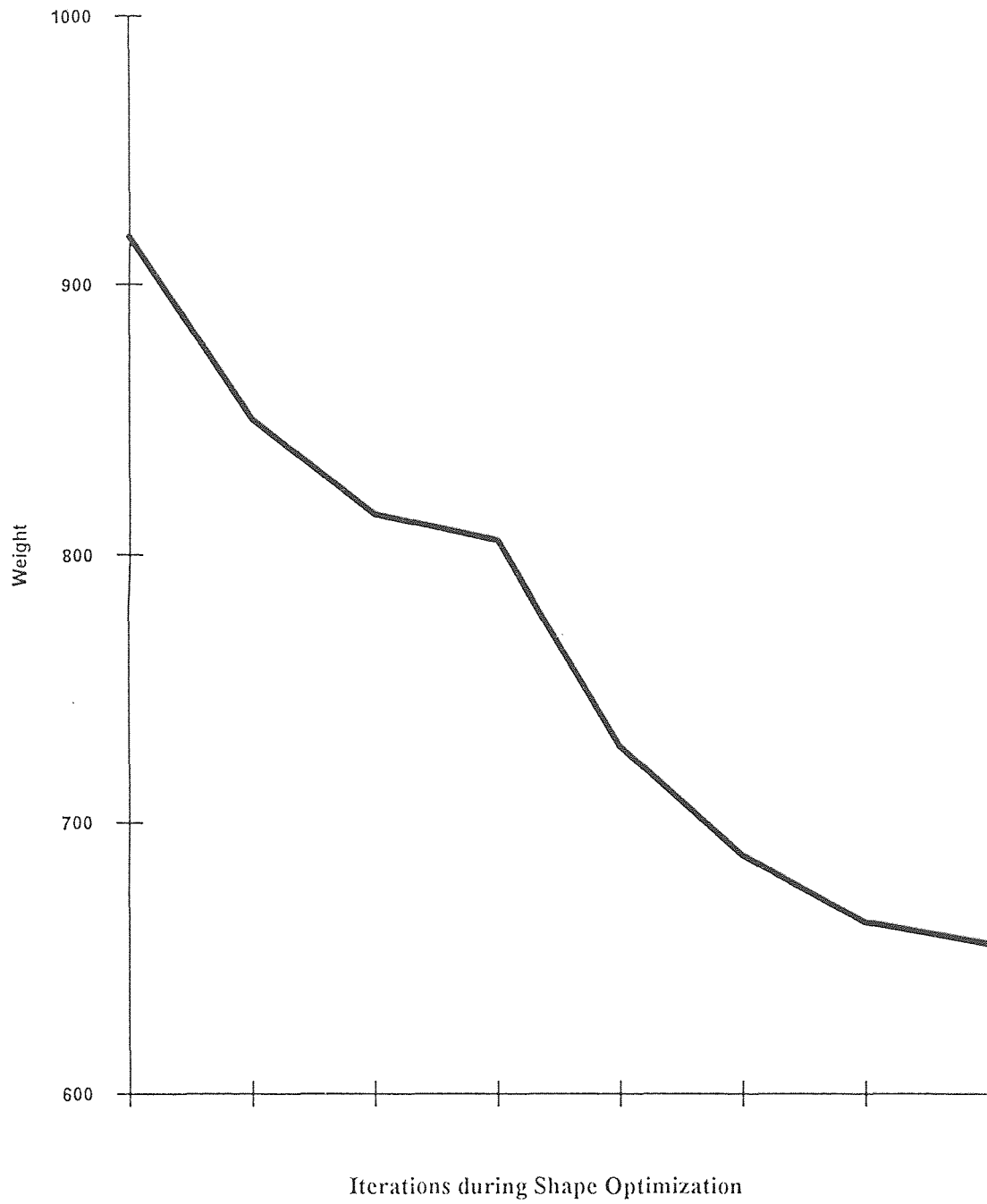
Fig 8.7 Variation of structural weight during Shape Optimal Design process

8.3 Scope for Future Work

The present work is more focused on two dimensional applications or thin structures. However, similar approach could also be applied to those mechanical structures, which can only be defined using three dimensional solid brick elements. The need for light weight economical structures and limited energy resources is one of the driving force for design optimization procedures. Aerospace, Automotive and Bio Medical industries are among the vast fields of applications.

The integration of shape optimization with CAD could be the most valuable tool in the design process. A continuing research is going on to present a unified automated approach to address this challenge. Another area to be pursued to fully automate the shape optimal design process is to improve the method of analysis to reduce the time takes by each analysis. This work could be treated as a step towards the development of fully automatic shape optimization process which could be added as another important tool of any CAD environment.

REFERENCES

Amresh, J. "Modern Methods for Automatic Finite Element Mesh Generation,"
   Proceedings of ASCE National Convention, Boston, October 1986.

Asce M., W. R. Spillers, and S. Singh. "Shape Optimization: Finite Element Example,"
   ASCE Structural Division, v107, no. 10, 1981, pp. 2015-2025.

Bagga, C. S. "Computer-Aided Shape Optimization Using Finite Element Analysis,"
   Master's Thesis Dept. of Mechanical & Industrial Engineering, NJIT,
   January 1992.

Balwin, K. "Modern Methods for Automating Finite Element Mesh Generation,"
   Proceedings of ASCE National Convention, Boston, October 1986.

Bennet, J. A. and M. E. Botkin. "Structural Shape Optimization with Geometric
   Description and Adaptive Mesh Refinement," *AIAA Journal*, v23, March 1985,
   pp. 458-465.

_____. "Automated Design for Automotive Structures,"
   *AIAA/ASME/ASCE/AHS Structures, Structural Dynamics and Materials
   Conference*, Nevada, 1983.

Bhavikatti, S. S. and C. V. Ramakrishnan. "Optimum Shape Design of Rotating Discs,"
   Computer Structures, v11, 1980, pp. 397-401.

Botkin, Me. E. "Shape Optimization of Plate and Shell Structures," *AIAA Journal*, v20,
   1982, pp. 268-273.

Braibant, V. and C. Fleury. "An Approximation - Concepts Approach to Shape Optimal
   Design, " *Computer Methods in Applied Mechanics and Engineering*, v53, 1985,
   pp. 119-148.

_____. "Shape Optimal Design Using B-Splines," *Computer Methods
   in Applied Mechanics and Engineering*, v44, 1984, pp. 247-267.

_____. "Shape Optimal Design: Performing CAD Oriented
   Formulation," AIAA/ASME/ASCE/AHS Structures, Structural Dynamics &
   Materials Conference, California, May 1984.

Burnett, D. S. Finite Element Analysis, from Concepts to Applications. New York:
   Addison-Wesley Publishing Company, 1987.

REFERENCES

(Continued)

Cavendish, J. C., D. A. Fields and W. H. Frey. "Automating Three-Dimensional Finite Element Mesh Generation," *Proceedings of ASCE National Convention*, Boston, October 1986.

Chavez, P. F. "Automatic Mesh Generation and Optimization from the Solids Models Database," *Proceedings of ASCE National Convention*, Boston, October 1986.

Chung, K. Y., T. Torigaki and J. Taylor. "Shape Optimization of Elastic Structures to Meet Local Criteria and Constraints," *22nd Annual Meeting of the Society of Engineering Science*, Pennsylvania, 1985.

Diaz, A. R., N. Kikuchi, P. Papalambros and J. Taylor. "Design of an Optimal Grid for Finite Element Methods," *Journal of Structural Mechanics*, v2, 1983, pp.215-230.

_____. "A Method of Grid Optimization for Finite Element Methods," *Computer Methods in Applied Mechanics and Engineering*, v41, 1983, pp. 29-45.

Ding, Y. "Shape Optimization of Structures, a Literature Survey," *Computers and Structures*, v24, no. 6, 1986, pp. 985-1004.

Esping, B. J. D. "Minimum Weight Design of Membrane Structure Using Eight Node Isoparametric Elements and Numerical Derivatives," *Computers and Structures*, v19, 1984, pp. 591-604.

Felippa, C. A. "Optimization of Finite Elements Grids by Direct Energy Search," *Applied Math Modeling I*, 1976, pp. 93-96.

_____. "Numerical Examples in Finite Element Grid Optimization by Direct Energy Search," *Applied Math Modeling I*, 1977, pp. 239-244.

Finite Element Handbook. New York: McGraw-Hill, 1987.

Hou, J. W., J. L. Chen and J. S. Sheen. "Computational Method for Optimization of Structural Shapes," *AIAA Journal*, v24, 1986, pp. 1005-1012.

Haftka, R. T. and R. V. Grandhi. "Structural Shape Optimization, a Survey," *Computer Methods in Applied Mechanics and Engineering*, v57, 1986, pp. 91-106.

REFERENCES

(Continued)

Imam, M. H. "Minimum Weight Design of 3-D Solid Components," *Proceedings of the 2nd International Computer Engineering Conference*, ASME, v3, California, August 1982.

_____. "Three Dimensional Shape Optimization," *International Journal for Numerical Methods in Engineering*, v18, no.5, 1982, pp. 661-673.

Kikuchi, N., K. Y. Chang, J. E. Taylor and T. Torigaki. "Adaptive Finite Element Methods for Shape Optimization of Linearly Elastic Structures," *Computer Methods in Applied Mechanics and Engineering*, v57, 1986, pp. 67-89.

_____. "Optimal Modification of Shape for Two Dimensional Elastic Bodies," Journal of Structural Mechanics, v11, 1983, pp. 111-135.

_____. "Shape Optimization in Laminated Composite Plates," *Computer Methods in Applied Mechanics and Engineering*, v72, 1989, pp.29-55.

Kirsch, U. Optimum Structural Design. New York: McGraw Hill, 1981.

Luchi, L. M. "An Interactive Optimization Procedure Applied to Design of Gas Turbine Discs," *Computers and Structures*, v11, 1980, pp. 629-637.

Luss, H. "Optimization: Methodology, Algorithms and Applications," *AT&T Technical Journal*, v68, May-June 1989, pp. 3-6.

Morris, A. J. "Foundations of Structural Optimization: A Unified Approach," *Proceedings of NATO ASI Meeting*, Iowa City, 1980.

Oda, J. "Pattern Transformation Method for Shape Optimization in its Application to Spoked Rotary Discs," *International Symposium on Optimal Structural Design*, Arizona, 1981.

Oliver, J. and G. Bugeda. "A General Methodology for Structural Shape Optimization Problems Using Automatic Adaptive Remeshing," *International Journal for Numerical Methods in Engineering*, v36, 1993, pp.3161-3185.

Olhoff, N. "On Structural Optimization," *Journal of Applied Mechanics*, v50, December 1983, pp. 1139-1151.

REFERENCES

(Continued)

Prager, W. and J. E. Taylor. "Problems of Optimal Structural Design," *ASME Journal of Applied Mechanics*, v35, 1968, pp. 102-106.

Quea, J. P. "Two Dimensional Shape Optimal Design by the Finite Element Method," *International Journal for Numerical Methods in Engineering*, v15, 1980.

Rao, S. S. The Finite Element Method in Engineering. New York: Pergamon Press, 1982.

Rasmussen, J. "The Structural Optimization System CAOS," *Structural Optimization*, v2, 1990, pp. 109-115.

Reddy, J. N. An Introduction to Finite Element Method. New York: McGraw-Hill, 1984.

Schmit, L. A. "Structural Synthesis - Its Genesis and Development," *AIAA Journal*, v19, 1981, pp. 1249-1263.

Schramm, U. and B. P. Walter. "Structural Shape Optimization for the Torsion Problem Using Direct Integration and B-Splines," *Computer Methods in Applied Mechanics and Engineering*, v107, 1993, pp. 251-268.

Tinucci, J. P. "A Mesh Generator for the Modeling of Rock Discontinuities," *Proceedings of ASCE National Convention*, Boston, 1986.

Vanderplaats, G. N. "Structural Optimization - Past, Present and Future," *AIAA Journal*, v20, 1982, pp. 992-1000.

Venkayya, V. B. "Structural Optimization: A Review and Some Recommendations," *International Journal of Numerical Methods in Engineering*, v30, 1978.

Yang, R. J. "Accuracy of Finite Element Based Shape Design Sensitivity Analysis," *Journal of Structural Mechanics*, v13, 1985, pp. 223-239.

Zienkiewicz, O. C. and J. S. Campbell. Optimum Structural Design. New York: John Wiley, 1973.