ABSTRACT

Title of Thesis: Computer Aided Shape Optimization Using Finite Element Analysis.

Thesis By: Charanpreet Singh Bagga,
Master of Science in Mechanical Engineering

Thesis Directed By : Dr. Raj S. Sodhi,
Associate Professor,
Department of Mechanical Engineering

Various theories behind Computer Aided Shape Optimization are studied. A special attention is paid to the shape representation, mesh generation and refinement during the shape optimization process. Shape optimization process for a wall bracket is performed using the Computer Aided Engineering package of I-DEAS. Finite element analysis of the wall bracket is done and the stress distribution over the bracket is studied. The bracket is trimmed in the selected low stress regions using shape optimization capability of I-DEAS and an optimized shape of the wall bracket is obtained. The end product is evaluated using Supertab module of I-DEAS as well as the Finite Element Analysis package of ANSYS.

COMPUTER AIDED SHAPE OPTIMIZATION USING FINITE ELEMENT ANALYSIS

by :
Charanpreet Singh Bagga

A Thesis
Submitted to the Faculty of the Graduate Division of the
New Jersey Institute of Technology
in Partial Fulfillment of the Requirements for the Degree of
Master of Science in Mechanical Engineering
January, 1992

APPROVAL SHEET

Title of Thesis: Computer Aided Shape Optimization Using Finite Element Analysis

Name of Candidate: Charanpreet Singh Bagga
Master of Science in Mechanical Engineering, 1992

Thesis and Abstract Approved :

12/19/91

Dr. Raj S. Sodhi — Date
Associate Professor,
Department of Mechanical Engineering,
Director of Manufacturing Engineering.

Signatures of other members of the thesis committee :

12/19/91

Dr. B. Koplik, — Date
Professor and Chairman,
Department of Mechanical Engineering.

Dec. 19, 91

Dr. N. Levy, — Date
Associate Professor,
Department of Mechanical Engineering.

VITA

Name : Charanpreet Singh Bagga

Adress : 272, Ampere Parkway, Bloomfield, NJ 07003

Degree and date to be conferred : MSME, Jan. 1992

Date of Birth :

Place of Birth :

Secondary Education : Khalsa College, Amritsar, Punjab, India.

Professional Institutions Attended	Dates	Degree	Date of Degree
Punjab Engineering College Chandigarh, India.	1986- 1990	BEME	June,1990
New Jersey Institute of Technology	1990 1991	MSME	January,1992

To

My

Parents

ACKNOWLEDGEMENT

The author wishes to express his sincere gratitude to his advisor, Dr. Raj Sodhi for his guidance and moral support throughout his research.

TABLE OF CONTENTS

CHAPTER	PAGE
1. INTRODUCTION	1
1.1 An Introduction to Shape Optimization	1
1.2 Need for Shape Optimization	2
1.3 Scope and purpose of this Thesis	3
2. AN INTRODUCTION TO FINITE ELEMENT METHODS	4
2.1 An Introduction To Finite Element Methods	4
2.2 Steps Involved in the Finite Element Analysis of a Typical Problem	5
2.2.1 Step 1	5
2.2.2 Step 2	5
2.2.3 Step 3	7
2.2.4 Step 4	7
2.2.5 Step 5	7
2.2.6 Step 6	8
2.2.7 Step 7	9
2.2.8 Step 8	9
2.2.9 Step 9	10
2.2.10 Step 10	11
2.3 Guidlines for Element Usage	11
2.3.1 Aspect Ratio	12

3. MATHEMATICAL PROBLEM FORMULATION AND SOLUTION 14

3.1 Mathematical Problem Formulation 14

3.2 Objective Function 15

3.3 Method of Solution 17

4. SHAPE REPRESENTATION 20

4.1 Shape representation 20

4.2 Use of Boundary Nodes for Shape Representation 21

4.3 Polynomial Representation 21

4.4 The Design Element Technique 24

4.4.1 The Design Element 24
4.4.2 Generic Shape Optimization Model 28

4.5 Super Curves Technique 30

4.6 Boundary Shape Described by Spline or Spline Blending Functions 33

4.6.1 Bezier's and B-Splines in the Definition of a Design Element 38

4.7 The Structural Optimization System CAOS 41

5. MESH GENERATION AND REFINEMENT 44

5.1 Mesh Generation 44

5.2 Mesh Refinement 48

5.2.1 Complete Remesh 48

5.2.2 Localized Mesh Refinement 48

5.3 Adaptive Mesh Refinement 49

6. COMPUTER AIDED ENGINEERING PACKAGE I-DEAS 53

6.1 CAE Package of I-DEAS 53

6.2 Geometry Modeling Task 54

6.3 Mesh Creation Task 54

6.4 Boundary Conditions Task 56

6.5 Model Solution Task 56

6.6 Post Processing Task 58

6.7 Adaptive Meshing 58

6.8 Optimization 60

7. COMPUTER AIDED SHAPE OPTIMIZATION OF WALL BRACKET USING I-DEAS 64

7.1 Solid Modeling of the initial object 64

7.2 Finite Element Modeling 64

7.2.1 Creation of wireframe 67

7.2.2 Choice of meshing 67

7.2.3 Formation of mesh areas 67

7.3 Mesh Generation 67

7.4 Boundary Conditions 68

7.5 Model Solution 68

7.6 Post Processing 68

7.7 Shape Optimization of Wall Bracket 69

8. RESULTS AND CONCLUSIONS 87

8.1 Results 87

8.2 Evaluation of I-DEAS package for Shape Optimization 88

8.3 Conclusion 89

8.4 Scope for Future 89

9. BIBLIOGRAPHY 90

List of Figures

Fig. #	Description	Page #
4.1	Three Dimensional Isoparametric Element	22
4.2	The Design Element	23
4.3	Generated Mesh for an initial shape	26
4.4	Generated Mesh for a final shape	27
4.5	Concept of Generic Modeling	30
4.6	Shape Representation by Parametric Curves	31
4.7	Shape Superposition Technique	32
4.8	The Concept of Design Element	34
4.9	Bezier Curve	36
4.10	Bezier Curves : Zero and First Order Continuity	37
4.11	Bezier Curve and B-Spline	39
4.12	B-Splines	39
4.13	Design Element with Bezier curves and B-Splines	40
5.1	Initial Mesh Shape	45
5.2	Final Mesh Shape	46
6.1	Initial Design of Wall Bracket	65
6.2	Flow Chart of Shape Optimization Process	66
8.1	Initial Shape of the Wall Bracket	70
8.2	Final Shape of the Wall Bracket	71
8.3 to 8.16	Intermediate Steps of the Shape Optimization Process	72 to 86
8.17	Stress Distribution over the Optimized Shape using ANSYS	87

CHAPTER 1

INTRODUCTION

(1.1) An Introduction to Shape Optimization

The problem of finding the optimum shape of a structural component can be called shape optimization. In other words, shape optimization may be defined as the rational establishment of a structural design that is the best of all possible designs within a prescribed objective and a given set of geometrical and/or behavioral limitations.

Shape optimization is different from structural optimization as structural optimization is limited to resizing of structural members to obtain optimum crossections or thickness. Shape optimization solves another class of optimization problems involving continuous structural components where optimum shape of boundaries and surfaces of components is determined. Shape optimization is more complex than pure sizing optimization. Since the shapes are continuously changing in the design process, a special attention has to be paid to the following areas :

(1) To describe the changing boundary shape.

(2) To maintain an adequate finite element mesh.

(3) To enhance the accuracy of the sensitivity analyses.

(4) To impose proper constraints.

(5) To utilize existing optimization methods to solve the shape optimization problems.

Analytical methods for solving shape optimization problems have been used for a long time. Perhaps the best known early work is the study by Michell(1904). Zienkiewicz and Campbell(1973) were among the first to use numerical methods for selecting an optimum shape of the structure. They utilized the finite element method with node co- ordinates as the design variables to find an optimum shape. Since then several authors have published practical successful applications of general optimization. Furthermore, a number of commercial optimization systems based on well established finite element codes have been introduced. Systems such as ANSYS, IDEAS, OASIS, MSC-NASTRAN, SAMTECH and NISAOPT are widely known examples.

(1.2) Need for Shape Optimization

The need for shape optimization has been there since long and has attracted many people into this field but the urge for shape optimization grew stronger with the availability of high speed digital computers. Considerations of limited energy resources, shortage of economic and material resources, strong technological competition and environmental problems motivate the considerable current research going on in the field of shape optimization, and indicate increasing significance for the field in future.

The principal motivating forces are tough market competition and limited resources. Optimized product among other things can imply lesser material cost, lighter weight which may result in many secondary benefits. For example decreasing the weight of transportation systems means lesser energy consumption, higher speed, lesser vibrations and noise and lower pollution. Similarly, it has great significance in the biomedical field. A lighter artificial limb means better comfort for the patient who is using it .

(1.3) Scope And Purpose of this Thesis

The objective of this thesis is twofold. First, to study the shape optimization process using finite element analysis. Second objective is to use I-DEAS package to optimize a given wall bracket.

Before discussing the shape optimization problem, it is necessary to understand how a shape can be represented in the definition of the problem. Similarly it is very important to study how the finite element mesh is generated over the defined shape. There is no common approach being used for shape representation, so various methods of shape representation will be studied. The second objective we have is to use a commercial software to solve a practical problem using shape optimization and I-DEAS is selected for the purpose. Thus the first part of the thesis deals with the theoratical aspect of shape optimization, while the second part deals with its practical application.

CHAPTER 2

AN INTRODUCTION TO FINITE ELEMENT METHODS

(2.1) AN INTRODUCTION TO FINITE ELEMENT METHODS

Finite element method is a method of finding an approximate solution to a boundary and/or initial value problem by assuming that the domain is divided into well defined elements and the unknown function of the state variable is defined approximately within each element. With these individually defined functions matching each other at the element nodes or at certain points at the interfaces, the unknown function is approximated over the entire domain.

There are many other approximate computational methods like Galerkin method, Rayleigh-Ritz method, finite difference method, least squares method etc. to solve the boundary value problems but the basic difference between finite element method and most other methods is that in the finite element method, the approximation is confined to relatively small subdomains. It is, in a way, a localized version of Rayleigh-Ritz method. Instead of finding an admissible function satisfying the boundary conditions for the entire domain, which is often difficult for irregular domains, the finite element methods define the admissible functions over element domains with simple geometry, which obviates the complications at the boundaries. This is one of the reasons that finite element method has gained superiority over the other approximate methods.

(2.2) STEPS INVOLVED IN THE FINITE ELEMENT ANALYSIS OF A TYPICAL PROBLEM

Regardless of the physical nature of the problem, a standard finite element method involves the following steps. Each step requires a great deal of different planning and operations depending upon the physical nature and mathematical modeling of the problem.

(2.2.1) Step 1:

Definition of the Problem and its Domain.

(2.2.2) Step 2:

Discretization of the Domain into a Collection of Preselected Elements.

This step can be subdivided into the following three steps

(1.) Construct the finite element mesh of the preselected elements.

(2.) Number the nodes and elements.

(3.) Generate the geometric properties (e.g.,coordinates, crossectional areas etc.) needed for the problem.

An important part of this step is to decide the number of nodes and elements. It is true that finer the mesh (with smaller elements), the more accurate the solution should be. But the finer mesh means more number of nodes and elements and results in larger number of equations to be solved and rather decreases the accuracy apart from taking more computational time. Thus one needs to have an optimal mesh This problem has been solved to a great extent by automatic mesh generation and adaptive meshing. Research studies indicate that best mesh is the one based upon strain energy distribution.

Physical Problem	Conservation principle	State variable	Flux	Material constants	Source	Constitutive equation
Deformation of an elastic body	Equilibrium of forces	Displacement or forces	Stress or strain	Young's modulus of elasticity, Poisson,s ratio	Body forces or surface	Hooke's Law
Electric network	Equilibrium of currents	Voltage or current	Electric flux	Electrical conductivity	External electric charge	Kirchhof's Law
Torsion	Conservation of potential energy	Stress function or warping function	Rate of twist	Shear	-2 x angle of twist	Hooke's Law
Heat transfer	Conservation of energy	Temperature	Heat flux	Thermal conductivity	Internal or external heat	Fourier's Law
Fluid flow	Conservation of momentum	Velocity	Shear stress	Viscosity	Body forces	Stoke's Law
Flow through porous media	Conservation of mass	Hydraulic head	Flow rate	Permeability	Fluid sources	Darcy's Law
Electrostatics	Conservation of electric flux	Electric potential	Electric flux	Permittivity	Charge	Coulomb's Law
Magnetostatic	Conservation of magnetic poten-	Magnetic potential	Magnetic flux	Magnetic permeability	Current	Maxwell's Law

Table (2.1) Classification of various physical problems

(2.2.3) <u>Step 3:</u>

Identification of Physical(State) Variable(s).

Until this step, no reference has been made to the physical nature of the problem. Whether it is a heat- transfer problem, fluid or solid-mechanics problem or an electricity problem etc. . Table (2.1), Ref. (17) presents various physical problems with associated state variables and constitutive equations.

(2.2.4) <u>Step 4:</u>

Establishment of Coordinate Systems.

There are primarily two reasons for choosing special coordinate axes for the elements in addition to the global axis for the entire system. The first is the ease of constructing trial functions for the elements and the second is the ease of integration within the elements. However, since the elements have to finally assembled in the global frame for calculations, this step introduces additional computations in the form of coordinate transformations. Still it is better the than the complicacies of having the entire finite element analysis be carried out directly in the global system.

Depending upon the element shape, one can choose from cartesian, cylinderical or spherical coordinate system. Other coordinate systems known as natural coordinates such as area or volume coordinates are often employed in the finite element analysis since numerical integration is much simpler in respect to these coordinate systems.

(2.2.5) <u>Step 5:</u>

Construction of Approximate Functions for the Elements.

Once the state variable and the local coordinate system has been choosen, the function can be approximated in numerous ways. It is to be noted that there are two entities that need to be approximated. The first is physical (the state variable) and the second is geometrical (the shape of the element). The analyst has to decide whether to approximate physics (state variable) and geometry (element shape) equally or to give preference to one or the other in the various regions of the domain. This leads to three different categories of elements as follows.

Let "r" and "s" represent degree of approximation for element shape (coordinate tranformation) and interpolation (state variable),then:

(a) *Subparametric elements:* $r < s$

(b) *Isoparametric elements:* $r = s$

(c) *Superparametric elements:* $r > s$

It has been recommended in the literature that:

(1.) The local functions be so constructed that their discontinuities (in terms of their derivatives as well) should not make the functional itself undefined over the entire domain. In other words, not only the local functions but the derivatives of one order less than that occurring in the functional must be continuous.

(2.) The intergrand of the functional must be single-valued and represent a constant as the element size approaches zero.

(2.2.6) <u>Step 6:</u>

Derivation of Element Equations for all Typical Elements in the Mesh.

(a) Construct the variational formulation of the given differential equation over the typical element.

(b) Assume that a typical dependent variable 'u' is of the form

$$u = \sum_{i=1}^{n} u_i \psi_i$$

and substitute it into (a) to obtain element equations in the form

$$[K^{(e)}]\{u^{(e)}\} = \{F^{(e)}\}$$

(c) Derive or select, if already available in the literature, element interpolation functions and compute the element matrices.

Ref. (37) gives a detailed account of the above three steps.

(2.2.7) Step 7:

Assembly of Element Equations to Obtain the Equations of the whole Problem.

(a) Identify the interelement continuity conditions among the primary variables (relationship between the local degrees of freedom and the global degrees of freedom- connectivity of elements) by relating element nodes to global nodes.

(b) Identify the equilibrium conditions among the secondary variables (relationship between the local source or force components and the globally specified source components).

(c) Assemble element equations using (a) and (b) and the superposition.

(2.2.8) Step 8:

Imposition of the Boundary Conditions of the Problem.

(a) Imposition of Loads.

(b) Imposition of Restraints.

With the application of boundary conditions, the complete set of equations obtained in step7 are condensed to its final form, ready for solution.

(2.2.9) <u>Step 9:</u>

Solution of the Assembled Equations.

Until this step, no reference has been made to whether the problem is linear or nonlinear, or to whether it is an eigen value problem or not. Regardless of the nature of the problem, the finite-element methods eventually yield the solution of a set of simultaneous equations. The solution ,procedure for simultaneous equations can be categorized into three parts:

(1) *Direct*

(2) *Iterative*

(3) *Stochastic*

The direct solution techniques consist of a set of systematic steps and are used a good deal in finite element solutions. The accuracy of results is largely determined by the condition of the equations, the number of equations and the computer. The Guass elimination and Cholesky's factorization (LU decomposition) are the most commonly used direct procedures. These methods are well suited to a small or moderate number of equations.

When the systems are of a large order, iterative procedures such as Gauss-Seidel or Jacobi iterations are more suited. Iterative methods are generally self correcting and the accuracy of the solution depends on the number of iterations. The solution time is considerably less than that required by direct procedure. When the set of equations to be solved is nonlinear, the modified Newton-Raphson iteration method is the most commonly used method.

Stochastic solution procedures have received very little attention because the finite element methods are generally applied to deterministic rather than to probabilistic problems.

(2.2.10) Step 10:

Postprocessing/Interpretation of the Results.

This is the decision making step and is probably the most important step in the entire process. Two important questions have to be dealt with at this point: How good the results are ? and What should be done with them ? The first requires the estimation of error bounds, and the second involves the physical nature of the problem. The answers to these questions either terminate the analysis or require that certain steps be repeated. In some cases, the reanalysis begins with step1, until a satisfactory result is obtained.

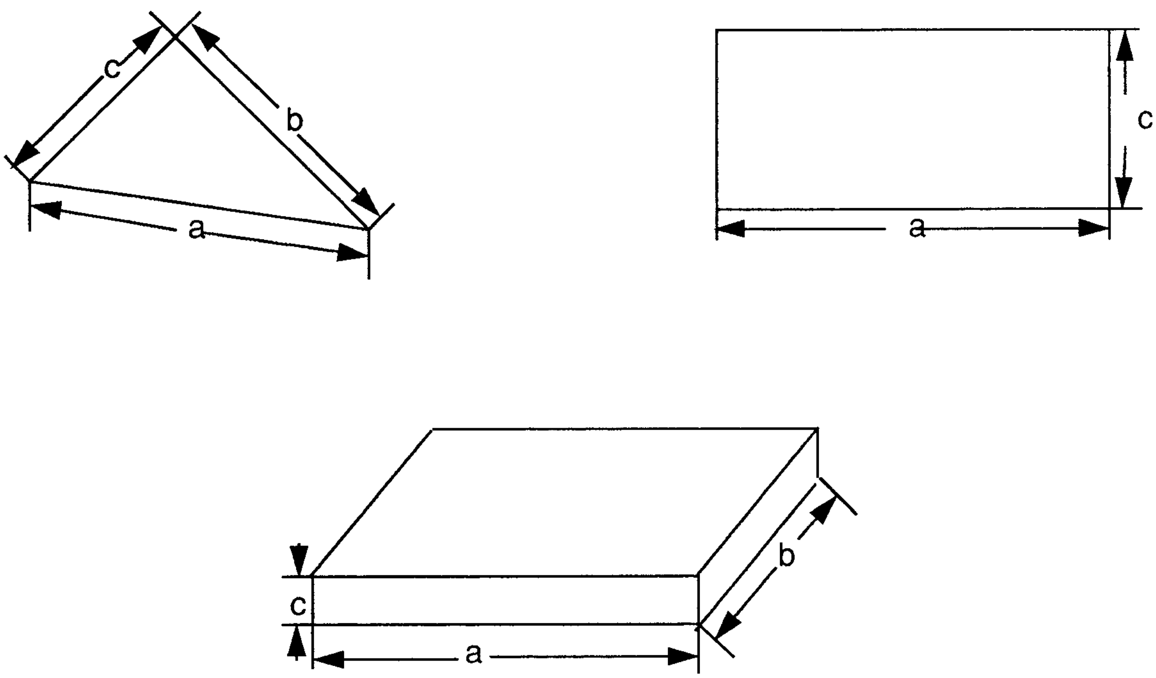
(2.3) Guidelines for Element Usage

Since selection of proper elements is an important first step in finite element modeling, it was considered necessary to discuss it in brief over here.

It is always desired for elements to have ideal shapes , which involve a little or no error in numerical computation of individual stiffness matrices. It would be convenient if triangles could always be equilateral, quadrilaterals always be squares, and hexahedra always be cubes. However, it is almost impossible to model complex systems with a mesh of ideally shaped elements. Therefore it is always wise to match the mesh refinement to stress gradients and deformation patterns. This means that elements must vary in size, have unequal side lengths, and, possibly, be distorted. We now discuss the modeling problems associated with elements having unequal side lengths, distorted elements, and transitioning patterns for varying refinements.

(2.3.1) Aspect Ratio :

The element aspect ratio is the ratio between the longest and the shortest element dimensions. Acceptable ranges for aspect ratio are element and problem dependent but generally it is considered safe to put a ceiling of 3 : 1 for stresses and 10 : 1 for deflections Actually there is no hard and fast rule governing all elements. The limit to aspect ratio is affected by the order of the element displacement function, the numerical integration pattern for stiffness, the material behaviour and even the resulting deflection and stress solution patterns.



ASPECT RATIO = a / c

Elements with higher order displacement functions and higher order numerical quadratures for a given displacement function are less sensitive to large aspect ratios. Elements in the regions of material nonlinearities are more sensitive to changes in the aspect ratio than those in the linear regions.

The best gauge according to Ref (17) for aspect ratio is the ability of the element to simulate the deflection and stress gradient of the given problem. In a general stress field with gradients in all directions, most elements should have aspect ratios near 1 : 1. Since no particular direction dominates, the mesh refinement must be nearly equal in all directions. If a problem has a deflection or stress gradient dominant in a single direction, elements may have relatively high aspect ratios, provided the shortest element dimension is in the direction of the maximum gradient.

Since an element's sensitivity to aspect ratio is dependent upon both element developement and actual problem limits, general tests and problem dependent checks must be implemented before using any of the elements. Simple "patch tests" for constant stress and similar tests under linear or other stress gradients can be run for each element type. However a user should also create simple models with loadings to simulate expected problem distortions and stresses. The problem like tests are necessary to develop cost-effective and accurate models.

In this chapter, we discussed the basic concepts behind the finite element analysis. Finite element analysis is the base on which the shape optimization process works. Various steps involved in the finite element analysis were studied. The choice of element is a very important step as the accuracy of the solution depends on it. Thus a special emphasis was placed on the choice of elements. Now the next step in the shape optimization problem is the problem formulation itself. The next chapter discusses the mathematical formulation of the shape optimization problem and its solution.

CHAPTER 3

Mathematical Problem Formulation and Solution

(3.1) Mathematical Problem Formulation

The shape optimization problem can be stated mathematically as

Find minimum $F(S_1,S_2,..,S_n)$

Subject to

$$h_j(S_1,S_2,..,S_n) = 0, \qquad j = 1,..,p$$

$$g_k(S_1,S_2,..,S_n) \leq 0 \qquad k = 1,..,q$$

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

................(1)

Where

F : Objective function

h_i : Equality constraint function describing i_{th} structural response.

g_i : Inequality constraint function describing i_{th} structural response.

S_i : Vector of n design variables defining shape of the object.

S_k^l : Lower limit of shape variables.

S_k^u : Upper limit of shape variables.

S_k : Shape variable.

p : Number of equality constraints.

q : Number of inequality constraints.

n : Number of shape variables.

(3.2) Objective Function

Objective function can be defined depending upon the optimization requirement.

(a) Weight or Volume Optimization :

In most of the cases, the weight or volume of the object is choosen as the objective function, which can be defined as :

$$F(S_1,S_2,..,S_n) = \sum_e e_e \Omega_e(S_1,S_2,..S_n)$$

Where $\Omega_e(S_1,S_2,..S_n)$ is the volume of the e_{th} finite element, and is normally a nonlinear function with respect to S_k.

(b) <u>The maximum Von Mises Stress</u>, that is :

$$F(S_1,S_2,..,S_n) = \text{Max } \sigma_{vm}$$

(c) <u>The difference between the maximum and the minimum tangential stresses</u> , that is :

$$F(S_1,S_2,..,S_n) = \sigma\theta_{max} - \sigma\theta_{min}$$

(d) <u>Stress leveling</u> , that is :

$$F(S_1,S_2,..,S_n) = \oint_A (\sigma - \sigma a)^2 \, dA$$

Where σ is the maximum principal stress and σa is the average stress at the initial shape, A is the part in question.

(e) <u>Weighted objective function</u> , that is :

$$F(S_1,S_2,..,S_n) = 0.5\ \Omega/(\Omega o) + \left(\oint (\sigma - \sigma a)^2 \, dA\right) \Big/ \left(\oint (\sigma o - \sigma a)^2 \, dA\right)$$

Where

Ω :	The volume of the object.
Ωo :	The volume at initial shape.
σ :	The maximum principal stress.
σa :	The average stress at the initial shape.
σo :	The maximum principal stress at initial shape.

The pupose of using the objective functions is to decrease the peak of stress concentration in the changed boundary and simultaneously to consider the control of boundary shape.

In most cases, the objective function is to minimize the weight or volume. If an objective function is to be maximized, one may just substitute F = - G and the rest of the process is just the same.

The resulting shape optimization problem is a nonlinear mathematical programming problem to which standard minimization techniques can be applied. However this problem exhibits some characteristics that make it complicated when practical design applications are considered. The main difficulties arise from the large number of design variables and the large number of nonlinear inequality constraints that are computationally burdensome implicit functions of the design variables. Moreover their precise numerical evaluation requires a complete finite element analysis. Since the solution scheme is iterative, it involves a large number of structural reanalysis and the computational cost often becomes prohibitive when large structural systems are dealt with.

(3.3) Method of Solution

The functions F, h_j and g_k depend both on design and state, and the equality constraints comprise the state equations. Assuming F, h_j and g_k to be continuously differentiable functions of S, we may formally solve the problem (1) using the calculus of variations. Based on F, we form an augmented objective function F* to be minimized.

$$F^* = F(S_1,S_2,..,S_n) + \sum_{j=1}^{p} \lambda_j . h_j(S_1,S_2,..,S_n) + \sum_{k=1}^{q} \mu_k [g_k(S_1,S_2,..,S_n) + S_k^2] \qquad \text{. . . . . (2)}$$

where λ_j and μ_k are Langrangian multipiers.

The conditions of stationarity of F* with respect to arbitrary admissable variations of $(S_1,S_2,..,S_n)$ lead to a number of n optimality conditions :

$$\frac{\partial F}{\partial Si} + \sum_{j=1}^{p} \lambda_j \frac{\partial}{\partial Si} hj + \sum_{k=1}^{q} \mu_k \frac{\partial}{\partial Si} gk = 0 , \qquad i = 1,..,n$$

. . . . **(3)**

and stationarity with respect to Langrangian multipliers λ_j, j= 1,..,p, recovers the equality constraints in (1),

$$h_j(S_1,S_2,..,S_n) = 0 , \qquad j= 1,..,p$$

. . . . **(4)**

Stationarity of F* with respect to S_k, k=1,..,q, yields the so called switching conditions $\mu_k.S_k = 0$, and the necessary conditions $\frac{\partial^2}{\partial Sk2}F^* \geq 0$ for a minimum of F* imply that the Langrangian multipliers μ_k must be non-negative,

i.e., $\mu_k \geq 0, k = 1,. . ,q.$

A combination of the latter result with the switching conditions and the defining equations for S_k yields the conditions :

$$\mu_k = 0 \qquad \text{if } g_k(S_1,S_2,..,S_n) < 0$$

$$k = 1, . . , q$$

$$\mu_k \geq 0 \qquad \text{if } g_k(S_1,S_2,..,S_n) = 0$$

. . . .**(5)**

Which are seen to imply simplification in (3) if one or more of the inequality are not tight. Equations (3) - (5) constitute the formal set of governing equations for the shape optimization problem and are often called *the generalized Kuhn - Tucker conditions*. The governing equations derived here are generally not sufficient conditions for global optimality. Sufficient conditions are possible only in the case of simple cases where a linear relationship exists. In reference (32), Prager and Taylor have derived sufficiency conditions for a variety of such problems by making use of extremum priniciples of structural mechanics. Thus for a vast majority of the problems, it is necessary to apply iterative numerical methods of solutions. The numerical methods commonly used for solution of problems where optimality conditions are contained in the discretized or initially discrete set of governing equations are usually termed as optimality creterian methods. References (25,29,39,40) can be seen for more detailed description.

For complex shapes, optimality conditions together with the state equations and other constraint conditions form such a large and complicated set of algebraic equations that it may be advantageous to apply a purely numerical solution procedure from the very outset. Direct procedures of this type are identified as methods of mathematical programming, such as linear, nonlinear, geometric or integer programming. References (25,29,38,39) contain accounts of these methods.

In this chapter we discussed the shape optimization problem formulation. It is very important to define the problem and objective function accurately in order to obtain accurate solution. The mathematical equations involved in the solution were also discussed here. The next task is to define the shape of the boundary. The boundary shape is continuously changing in the shape optimization process. Thus the definition of the boundary has to be such that it can accomodate those changes. Various researchers have devised different techniques for shape representation. The next chapter discusses those techniques in detail.

CHAPTER 4

SHAPE REPRESENTATION

(4.1) Shape Representation

The shape representation is a very important step in the process of obtaining an optimum shape. If the shape variables are not carefully selected, the reliability of the results is affected seriously. In the following are listed some of the major techniques being used for the shape representation of an object during shape optimization.

(1) *Boundary nodes are used for shape optimization ;*

(2) *Boundary shape is described by piecewise polynomials ;*

(3) *Design element technique ;*

(4) *Surfaces are defined by curves known as Super Curves ;*

(5) *Boundary shape is described by spline or spline blending functions ;*

(6) *The structural optimization system CAOS.*

(4.2) Use Of Boundary Nodes For Shape Representation

Use of coordinates for boundary nodes in the finite element model as shape variables is the earliest used method. The approach is simple and instinctive, and associated with the finite element method. This choice of design variables has however the following severe drawbacks :

(1) *The number of design variables often becomes very large which leads to high costs and difficult optimization problems to solve :*

(2) *It is difficult to assure compatibility and slope continuity between boundary nodes, which may lead to an undesirable or impractical shape ;*

(3) *It is difficult to maintain an adequate finite element mesh during the optimization process.*

This idea, therefore is not practical except for some special cases where there may not be any other choice.

(4.3) Polynomial Representation

Many researchers use polynomials to describe the boundary shape when solving shape optimization problem in two or three dimensions. There are several possibilities. A few boundary nodes may be used to control the boundary shape. Coordinates or moving directions of the control may be used as design variables. Shape functions are used to define the shape of the boundary between those control nodes.

The use of polynomials with control nodes for shape representation can obviously reduce the total amount of shape variables, but can result in oscillatory boundary shape with high order polynomials due to the numerical instability of the higher order curves.

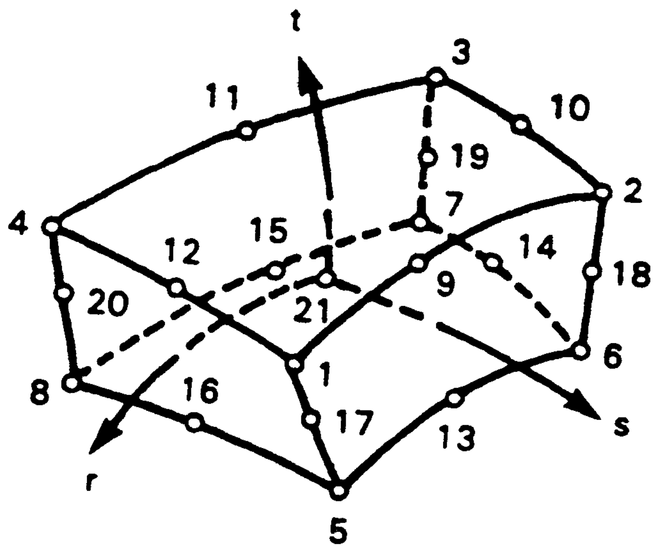


Fig. (4.1) Three Dimensional Isoparametric Element

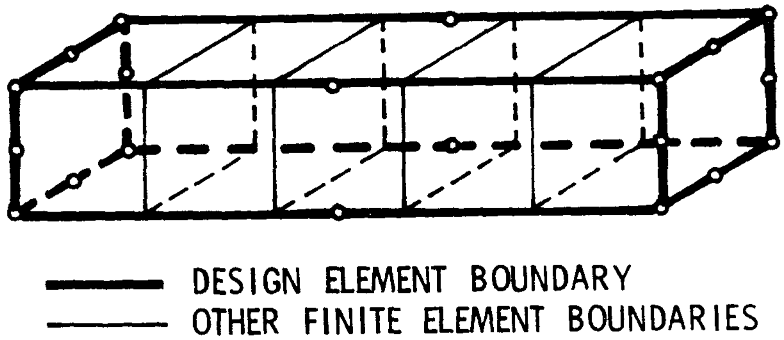


Fig. (4.2) The Design Element

(4.4) The Design Element Technique

The proponent of this technique is M. H. Imam. He has done an extensive work using the design element technique. References (19,20,21) give a detailed account of his works.

(4.4.1) The Design element

The concept of a design element in shape optimization for 3-D solids emerges from the fact that 20 noded isoparametric solid finite element shown in Fig. (4.1) can be used to represent the shape of a curved hexahedron. The shapes of its six surfaces can be varied by moving some or all of the 20 nodes which describe the element shape. The shape representation is inherent in the isoparametric formulation which uses the following set of equations to compute the x, y, z coordinates of any point on the outside surfaces or inside of the element :

$$x = \sum_{i=1}^{N} h_i x_i$$

$$y = \sum_{i=1}^{N} h_i y_i$$

$$z = \sum_{i=1}^{N} h_i z_i$$

where N is the number of nodes used to describe the element geometry; x_i, y_i and z_i (i= 1 to N) are the nodal coordinates of the i_{th} node ; and h_i's are the quadratic functions of of the parametric variables r, s and t. These parametric quantities (see Fig. (4.1)) uniquely identify the points on or inside of the element.

The design element shown in the Fig. (4.2) is a 20 noded isoparametric solid element used for shape representation of complex components. One design element may describe the shape of the whole structural component like a cantilever beam or, for more complex shaped components, it may be used as a building block. The concept is very useful for shape optimization because it simplifies the numerical representation of complex surfaces. Design elements with extreme aspect ratio and distorted shape can be used because the design element model is not used for analysis. The outside surfaces are approximated as piecewise parabolic, but the errors in the mass and the structural properties of the component due to this approximation have been observed to be small.

The idea of using the isoparametric finite element shape representation can be extended to avoid discontinuity at the element interface and to reduce the number of design variables by introducing the concept of a design element, which consists of more than one three dimensional finite elements. The whole structural component or parts of it are considered as one single 20 noded element for the purpose of shape representation as shown in Fig (4.2) by thick lines. The finite element mesh is contained in this design element. The isoparametric shape representation as used for the individual elements, is used to determine the coordinates of any point on the surfaces of the design element. The coordinates of the 20 nodes of the design element are the shape variables in this case. The parametric quantities, r, s and t, locate any point on the six surfaces. This technique also allows the determination of the coordinates of node points inside the design element for finite element mesh generation, which is required every time the shape is changed during the optimization. One apparent drawback of this technique is the restriction of the shape of the surfaces and edges to be no more than parabolic.

The finite element mesh can be generated automatically within each design element by specifying the number of elements along the natural (r, s, t) coordinates of the elements. The mesh is generated by the method of isoparametric mapping the details of which have been covered in Ref. (21) . The generated mesh is a function of the location of the design element nodes. The finite element mesh points are relocated every time the design element nodes move and therefore the finite element mesh distortion due to shape change during optimization process can be kept to a minimum. Care must be taken, however when using the design element as a building block to match the number of generated nodes and elements at the design element boundaries. This idea of design element modeling with automatic mesh generation has been presented in Ref. (4) for plate and shell components modelled by triangular flat plate elements.Its use for the 3-D solid components has been observed to be very helpful because it eliminates the

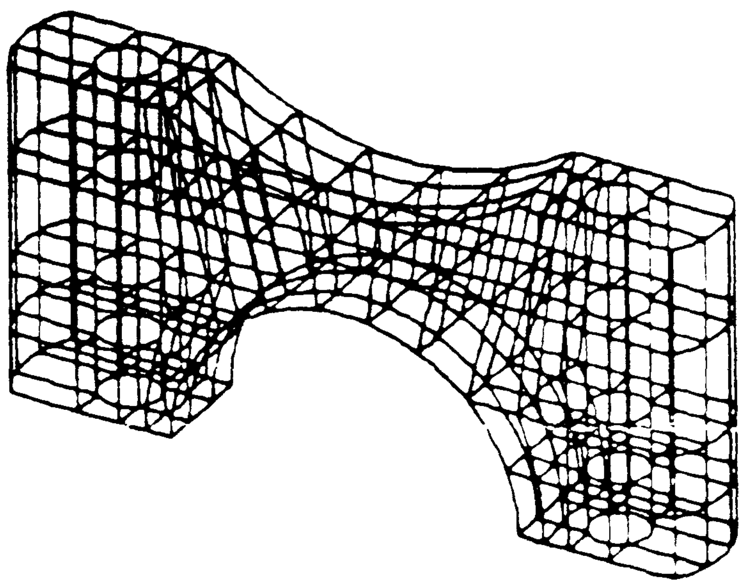


Fig. (4.3) Generated Mesh for Initial Shape

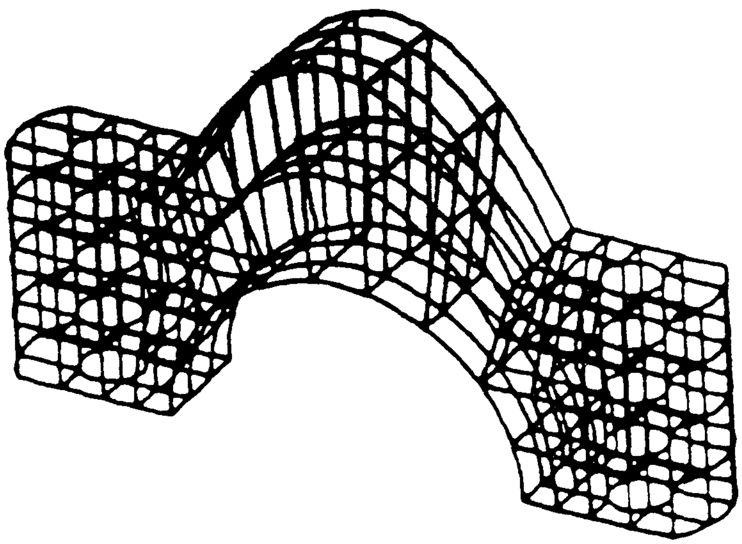


Fig. (4.4) Generated Mesh for Final Shape laborious finite element modeling which otherwise had to be done either manually or by the use of some interactive graphic capability. MH Imam has done a considerable amount of work in this field and Figs. 4.3 and 4.4 have been taken from one of his works (ref. 20,21) for illustration. Fig. 4.3 shows the changes in the mesh configuration from one design to another during the optimization process, where both designs are being represented by the same design element model. Fig. 4.4 shows the design element model of an engine main bearing cap and the generated finite element meshes of two different refinements.

(4.4.2) ***GENERIC SHAPE OPTIMIZATION MODEL***

In a production environment, the design of a component is represented by a drawing with detailed dimensioning. Some of those are key dimensions and are considered as design parameters because the mass of the component and its structural properties are sensitive to them. The designer thinks of changes in those key dimensions when considering any modification in the design or shape of the component. The design element technique discussed in the previous section requires the description of the component in terms of the x, y, and z coordinates of the design element nodes. Such a description of the component obviosly is very awkward from a designer's point of view. For complex parts it is very laborious and time consuming to build a design element model starting from the drawings. Also, any change in the design, which may be very simple in terms of the actual dimensions of the part shown on its drawing may require substantial changes in the design element model .

The problems mentioned above can be avoided if the diifficult task of finite element modeling is completely automated so that the required input information is only in terms of the key dimensions of a component . The design element technique with automated mesh generation is a major step in this direction because it automates the finite element modeling (for a given component whose design model has been built).

The basic idea in the generic modeling concept is to allow the user to specify the shape and topological details of the component in terms of the key dimensions only. For this

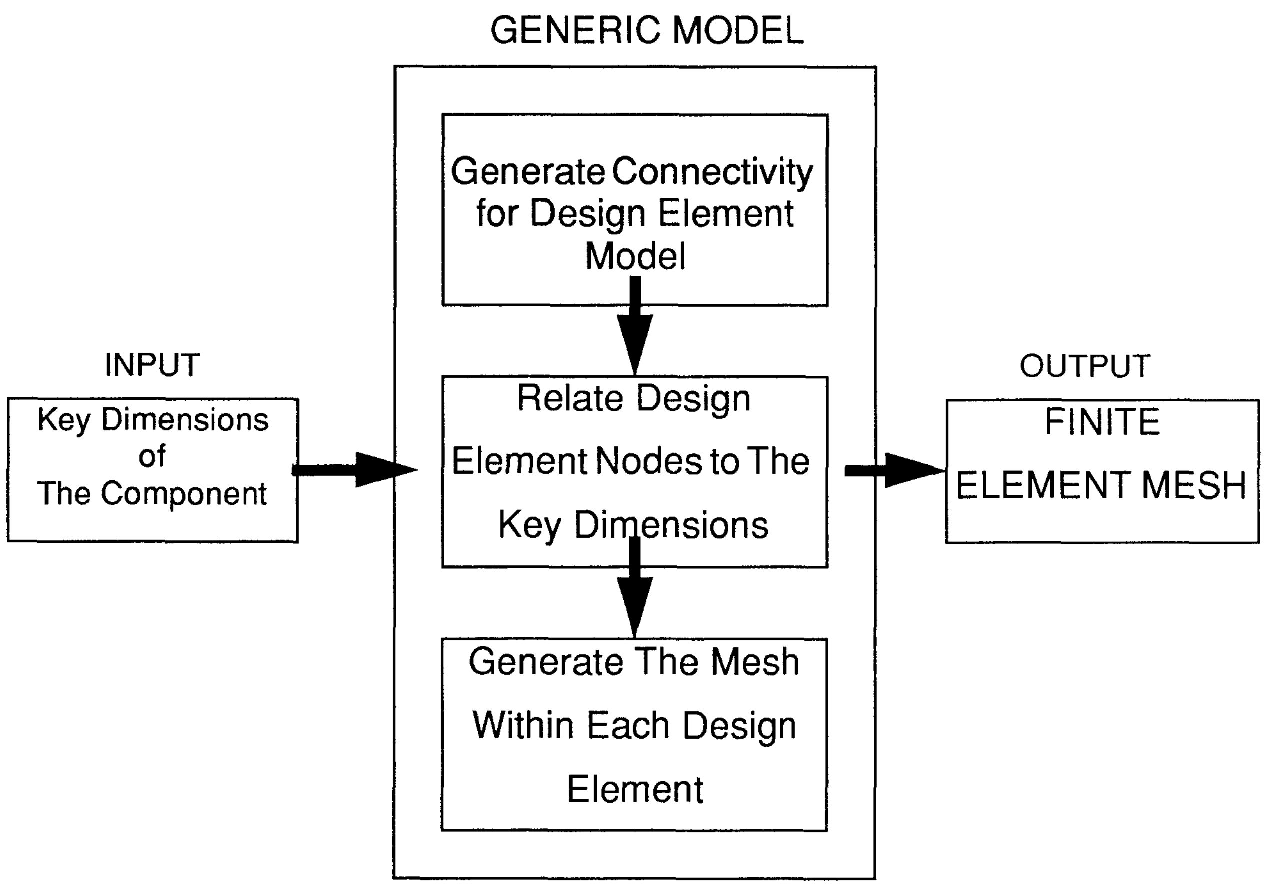


Fig. (4.5) Concept of Generic Modeling, Ref. (19)

purpose, a general design element model of a given component is first created in such a way that all feasible shapes and topological configrations can be described by the same model simply by moving the node points (offcourse there will be a limit on the shape changes which can be allowed, and a drastic change in the configration cannot be accomodated by the same model). The locations of these node points are are expressed as functions of the preselected key dimensions of the component. Thus, any changes in these key dimensions cause relocation of the nodes of the design element model to represent the new shape. A block diagram has been shown in Fig. (4.5) to explain this concept. It must be noted that data for the loads, the boundary conditions and the constraints on stresses and displacement are also regenerated as a result of any shape change. Therefore the generic shape optimization model must have built into it all necessary information relating to the key dimensions to all such details.

(4.5) Super Curves Technique

In the shape optimization problem, the shape of a surface is fully determined sif the shapes of a few curves on the surface are determined because on;y the points on those curves are required to construct a finite element model. For example, in FIG(4.6), the shape of the top surface is fully defined (for finite element modeling) if the shapes of the three curves (reffered to as 'super curves') are fully determined. In such cases, the parametric representation of the curves with polynomial expressions may be used as described in the following :

$$x = a_0 + a_1 s + a_2 s^2$$

$$y = b_0 + b_1 s + b_2 s^2$$

$$z = c_0 + c_1 s + c_2 s^2$$

. In most cases, it is possible to reduce the representation of the shape of a surface to the representation of the shapes of a few super curves. This approach also has the

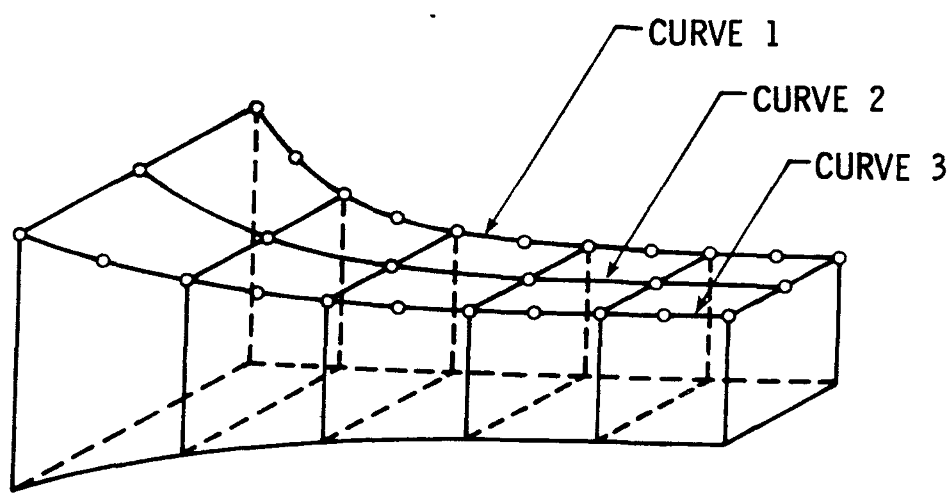


Fig. (4.6) Shape Representation by Parametric Curves advantage of allowing higher order surfaces without the use of complex polynomial expressions. Each curve can be handled individually with relatively simpler polynomial expressions. The coefficients of the polynomial are the shape or design variables.

A further step in this direction is the idea of super position of curves. The shape algorithm may consist of a table of numbers, input by the user and a computer subroutine to read them in and compute the co-ordinates of the selected points which determine the shape. This idea can be used to to superimpose two or more shapes (specified in terms of nodal locations of points on a curve or surface) in varying proportions to to generate a variety of shapes. If $d_1, d_2, \ldots, d_N$ are the shape variables associated with N different shapes given by the coordinate vectors $\{x_1\}, \{x_2\}, \ldots, \{x\}$, then a general shape $\{x\}$ can be expressed as :

$$\{x\} = d_1\{x_1\} + d_2\{x_2\} + \ldots + d_N\{x_N\}$$

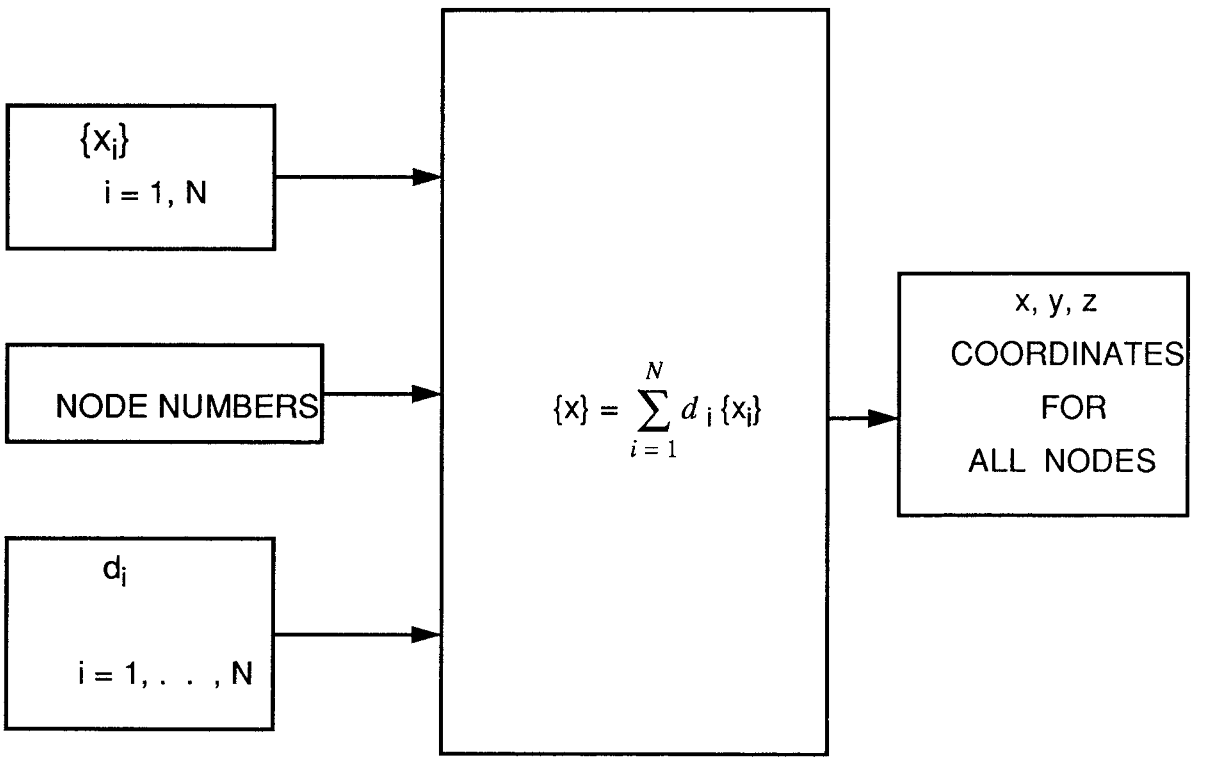


FIG (4.7) Shape Superposition Technique

{ Ref. (20) }

There are two basic ingredients which are required using this technique of shape representation :

1. Node numbers of a finite element mesh which are on a curve or a surface.
2 The vectors {xi} which give the nodal co-ordinates of the respective nodes for each input shape.

The figure on the last page shows the diagram for this technique. A similar technique was used by Vanderplaats for airfoil optimization. It allows representing complicated shapes without increasing the number of design variables. It is necessary to mention the following two points overhere :

1. A constant and a linear function should always be included among the input shapes to allow for a translation and rotation of the shapes.

2. It may be required to constrain the nodes the nodes to move between certain limits. These constraints are complex functions of design variables and cannot be expressed as simple side constraints on the design variables as can be done with the previous three techniques of shape representation explained before. Therefore, the constraints are treated as behavior constraints. These additional constraints are linear and they have no significant on the speed of the optimization process.

(4.6) Boundary Shape described by Spline or Spline blending functions

The technique used over here is similar to the design element technique used by Imam. The region of the structure that will be modified during the optimizaton process is also defined by one or more design elements which still contain a part of the mesh (see Fig. 4.8).

However the two approaches differ in the representation of the design element. Instead of using the shape functions of a two - dimensional finite element, blending functions

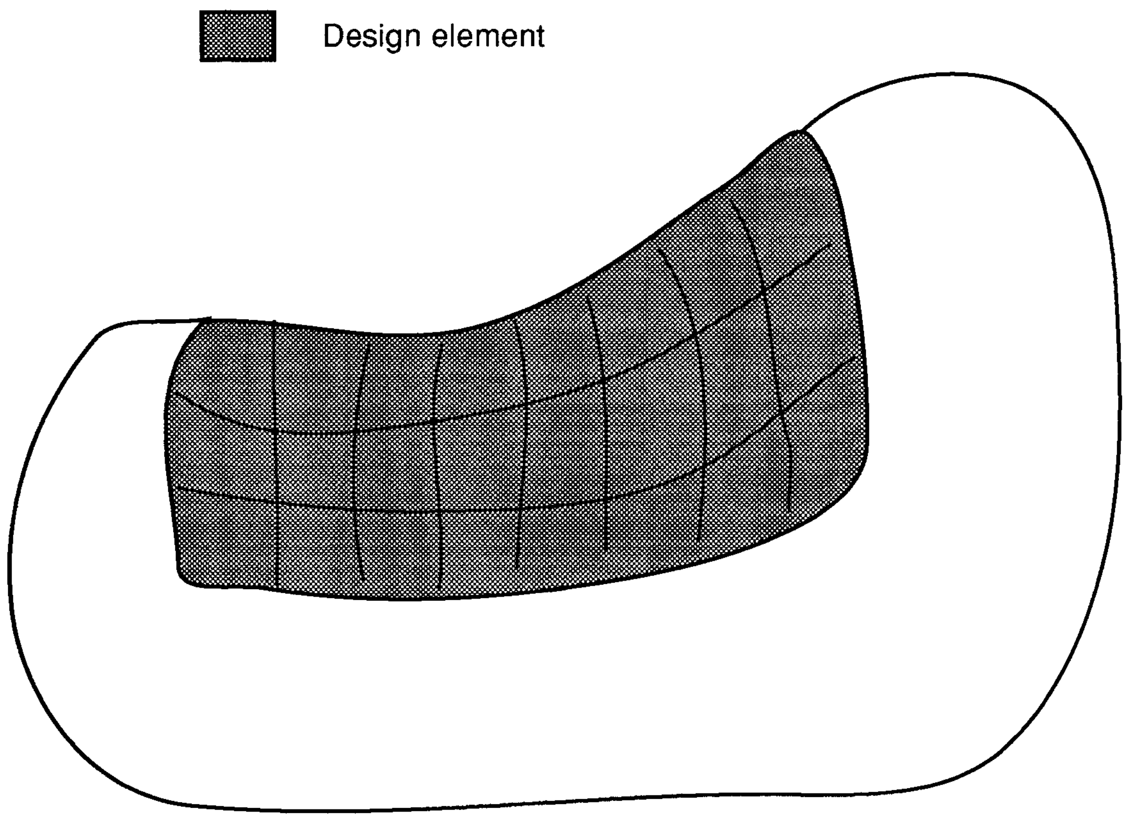


Fig. (4.8)

typical of computer graphic methods are employed to determine the coordinates of any point inside the design element or on its boundaries. More precisely, the blending functions are those used in the the *Bezier or the B - spline techniques.* Therefore the shape variables are no longer the positions of the the nodes of an isoparametric element but the points which control two familiesof curves whose Cartesian product defines the design element. Moreover, the degree of the boundaries may be more than cubic.

At this point, it becomes necessary to discuss Bezier curves or B- splines in brief to get a brief idea of what this technique really means, so next few lines will discuss these issues.

(a) <u>Bezier Curves</u>

In the following are discussed some of the highlights of the Bezier curves

Bezier curves are variation diminishing :

Each curve lies within the convex hull of control points that define it.

Axis independence :

A Bezier curve is independent of the coordinate system used to measure the location of the control points.

Multiple Values :

The parametric formulation of Bezier curves permits representation of very general functions (including multivalued functions such as spirals, closed curves, ...).

At the end points, the curve is tangent to the corresponding edge of the polygon of control points (Fig. 4.9).

Two characteristics of the blending functions limit the flexibility of this type of curve

(1) The number of control points fixes the degree of the polynomial which defines the curve. For instance, the four points of Fig. (4.9) define a cubic. The only way to reduce the degree of the curve is to reduce the number of vertices. Conversely, the only way to increase the degree of curve is to increase the number of vertices. There

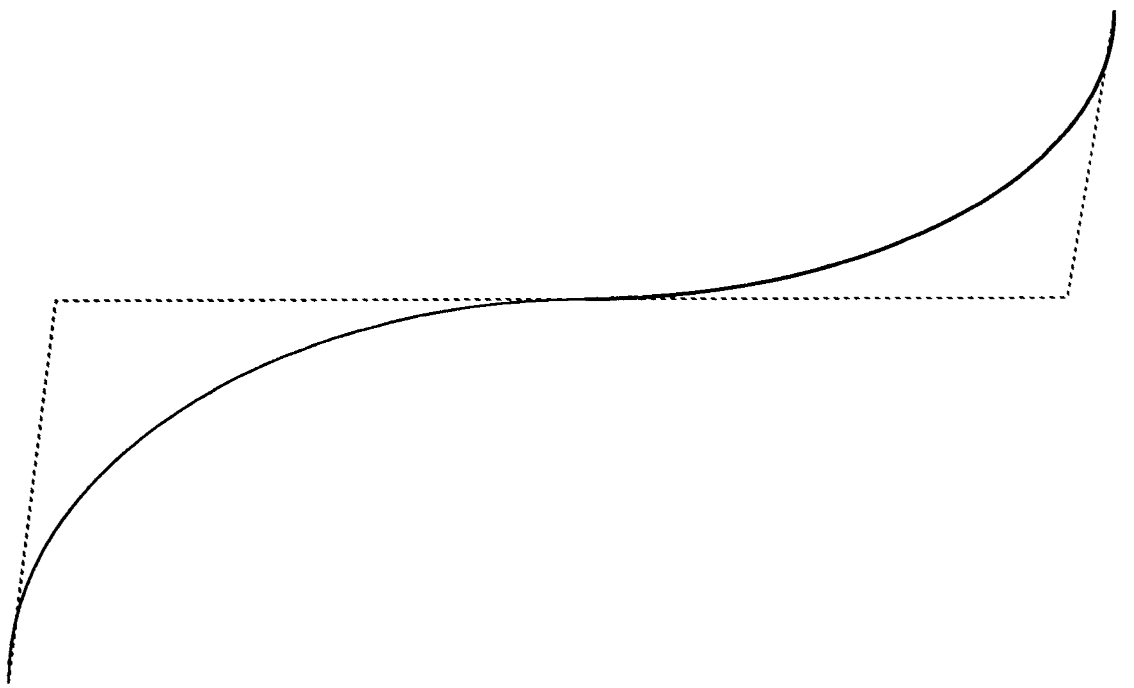
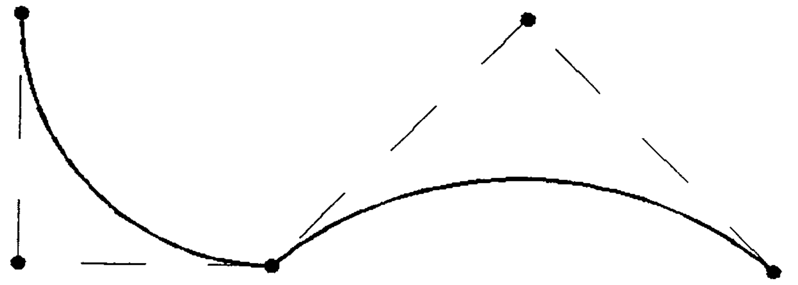
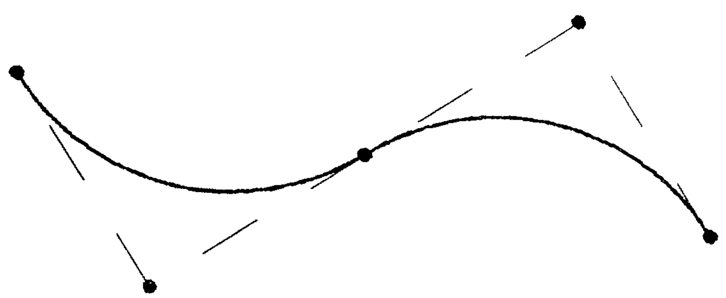


Fig. (4.9)



Zero Order Continuity



First Order Continuity

Fig. (4.10)

exist thus two ways of desribing a complex geometry : the first one consists in using high degree curves ; in the second one, Bezier curves of modest order are pieced together using simple geometric rules to insure continuity at at the different joints. For instance, to achieve zero order continuity at the at a joint, it is sufficient to impose the end control points of the curve to coincide. First-order continuity can be obtained by stating that the edges of the two polygons adjacent to the common end point must lie on a straight line (Fig. 4.10).

(2) The second major characteristic of the Bezier curves is that they do not provide local control : moving any control point will change the shape of the every part of the curve. this can be seen from the blending functions being nonzero everywhere. Consequently the location of each control point influences the whole curve.

(b) <u>B - spline curves</u>

B - splines share many of the characteristics of the Bezier curves : axis independence, variation diminishing property etc. The main advantages of the the B - splines are on one hand that local control of the curve shape can be achieved by using a set of blending functions that have local support only, and on the other hand, that additional control points can be introduced without increasing the degree of the curve. B-splines offer more parameters to the designer than Bezier curves : the degree can be selected (Fig. 4.11), as well as the multiplicities of control points (Fig. 4.12). Consequently, complex shapes may be represented by the quadratic or cubic splines which are automatically pieced together to form the B- spline. On the other hand, for a given number of vertices, the degree fixes the smoothness of the curve.

(4.6.1) <u>Bezier's and B-splines in the definition of a design element</u>

The formulations of Bezier and B-spline curves are easily extended to the generation of surfaces. A surface may be defined by the cartesian product of two curves so that the properties of the blending functions are not modified.
Applications of this approach show that the B-spline parametric curves representation is

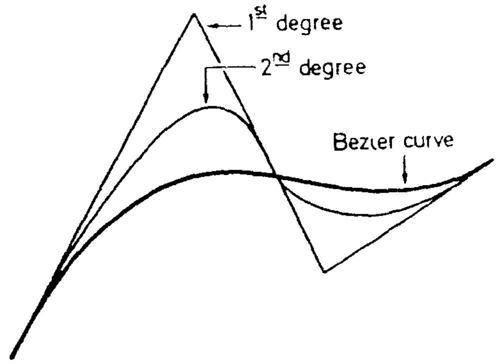


Fig. (4.11)

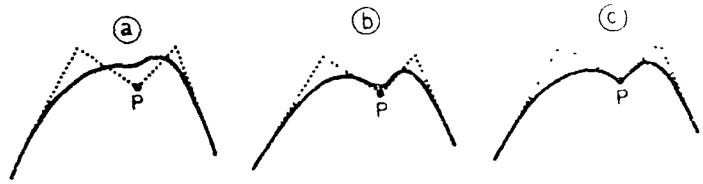


Fig. (4.12)

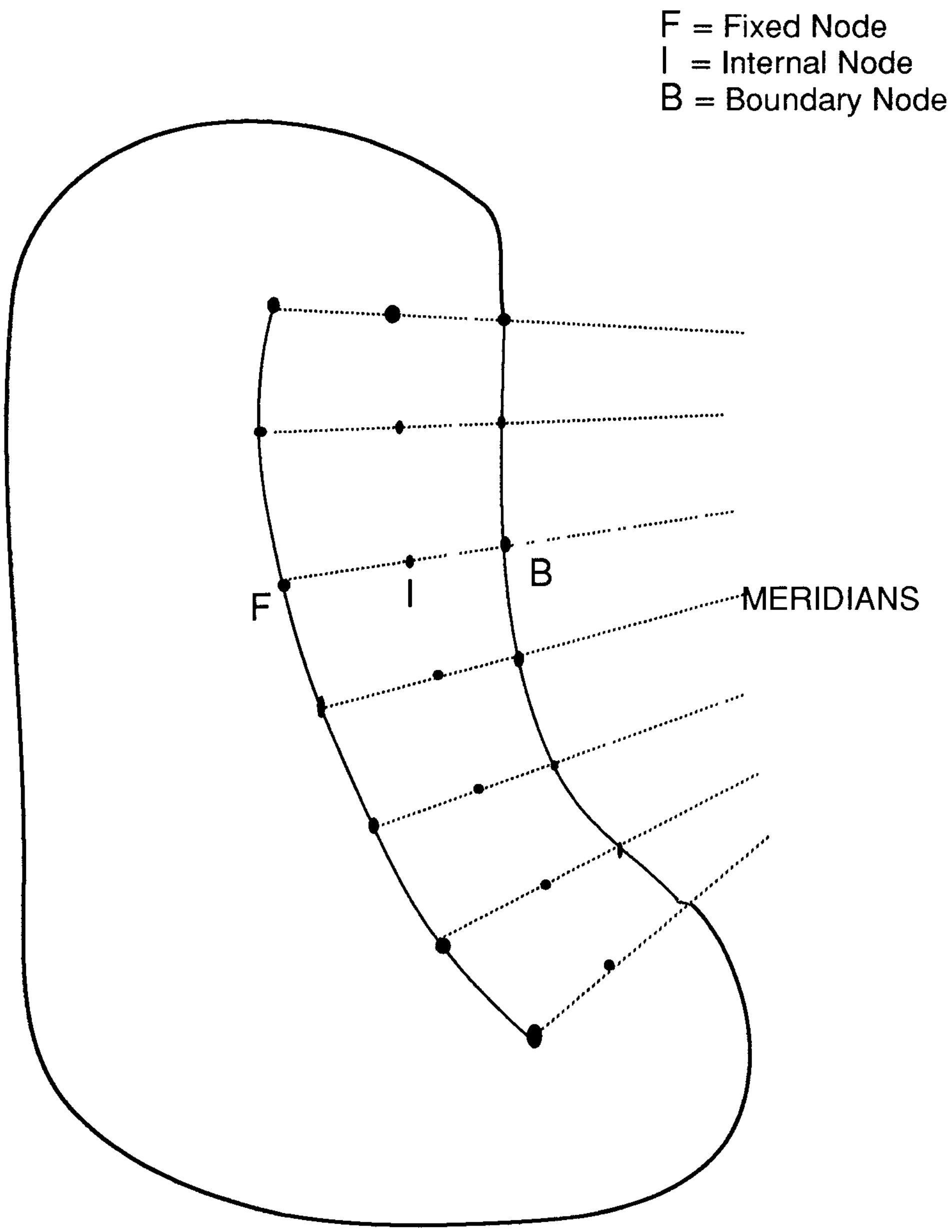


Fig. (4.13)

a very tractable tool for the shape optimization by the "design element technique" . By using the possibility of automatically piecing together splines of modest order to define a B-spline, and by controlling the continuity of the generated curve, design elements may exhibit very complex geometries. A few design elements are generally sufficient to fully describe the region that is modified during optimization. Moreover, selecting the degree according to the variation diminishing properties provides a rational scheme to avoid unrealistic designs.

At last the mesh may be easily updated inside the design element as follows : regular mesh is defined in the curvilinear coordinate system of the design element and coordinate transformations are applied to the definition of the mesh within it. When boundary nodes move, it is however necessary to actualize the positions of the the internal control nodes in order to insure a constant density. In the optimization code, it is assumed that all the design variables are the locations of the control nodes for a same edge of the design element. These vertices, together with the corresponding ones located on the opposite side, define 'meridian' directions along which the design control nodes are translated (see Fig. 4.13). Internal nodes that are located on these meridians are next moved homothetically.

(4.7) <u>The Structural Optimization System CAOS</u>

CAOS (Computer Aided Optimization of Structures) has been devised and developed by J. Rasmussen (Ref 35,36) using the foundations laid by the work of Bennet and Botkin (1), Esping (14), and Braibant and Fleury (5,6,7,8). The shape representation in CAOS is based on an adaption of the concept of design element as presented by Braibant and Fleury (5).

This approach is based on a subdivision of the geometry into a number of topologically similar quadrangular design elements. These elements have a number of attractive features :

1. The mesh generation is very easy using quadrangular elements. A number of randomly placed nodes on a boundary is the only input needed for a complete mesh

generation in the design element.

2. The boundaries of the design elements can be curves of almost any character. It is therefore very simple to generate relatively complicated geometries with a small number of design elements.

3. The shapes of the boundaries are controlled by a number of master nodes. This creates an evident connection between the design variables (namely the positions of the master nodes) and the shape of the geometry, thereby forming the necessary description of the shape by a set of design variables.

4. With the drawing aids of the CAD system it is very easy to draw the design elements in a separate drawing layer on the top of the original drawing.

Each of the four edges of a design element is a design boundary. Using these boundaries, the designer can control the outcome of the optimization process in two ways :

1. Design boundaries with a predefined function can be forced to take on a certain shape, e.g. a circular arc or a piecewise straight line.

2. If the designer desires a boundary to locally or globally maintain its shape, one or several of the master nodes can be fixed. This is simply accomplished by not assigning any design variable to the particular master node.

Design elements are simply drawn on top of the original geometry using the drawing tools of the CAD system. This is performed with a number of predefined geometrical entities. These entities have attributes assigned to them. Attributes are variable numbers or text strings that describe the entity. This is a very common facility which is normally used for storing dimensions etc. of standard components on a drawing. The CAD system can automatically scan the drawing for attributes of a given type and generate a text file

containing the data. In this particular case, the attributes describe curve types, number of finite element nodes, state variables etc. ; in other words, all the specification necessary to define an optimization problem.
When the definition of the optimization problem is completed, the system automatically generates a number of text files containing the information. Based on these files, the optimization system generates its own copy of the geometry and starts optimizing.
The widely used CAD system AutoCAD has been used for actual implementation of CAOS. AutoCAD is well suited for the present purpose because of its advanced LISP programming facilities. However, the integraton method does not rely on theCAD system data stucture. This makes the method sufficiently general to work in connection with a number of different CAD systems.

Various techniques for shape representation were discussed in this chapter. The generic model and the CAOS were discussed to consider the practical applications. The next most important step in shape optimization is to accurately generate the mesh. The basic principles were discussed in Chapter(2) so the next chapter pays more attention to advanced topics. Automatic mesh generation and adaptive mesh refinement are discussed in detail in the next chapter.

CHAPTER 5

MESH GENERATION AND REFINEMENT

(5.1) Mesh Generation

The problem of finite element mesh generation in shape optimization is due to the fact that in most cases the definition of a finite element mesh is manual rather than an automatic process. That is, the analyst uses judgement and experience based intution to select the mesh. Often the mesh is changed based on the result of a trial analysis, which reveals regions where the mesh needs to be refined. This manual approach is not adequate for shape optimization problems, because the analyst needs to define the mesh for a series of structures, without knowing their shape.

According to Ref. (17), in conventional finite-element models, the finite element mesh must satisfy the following three requirements :

(1) ***The essential geometric details of the object to be modeled must be represented.***

This is a very difficult task in the shape optimization as the shape of the object is continously changing.There are two basic solutions to the problem of adapting the mesh to the changing boundaries.

The first is to use simple modification rules for deforming the initial mesh. But such simple modification rules often run into trouble. This has been illustrated by

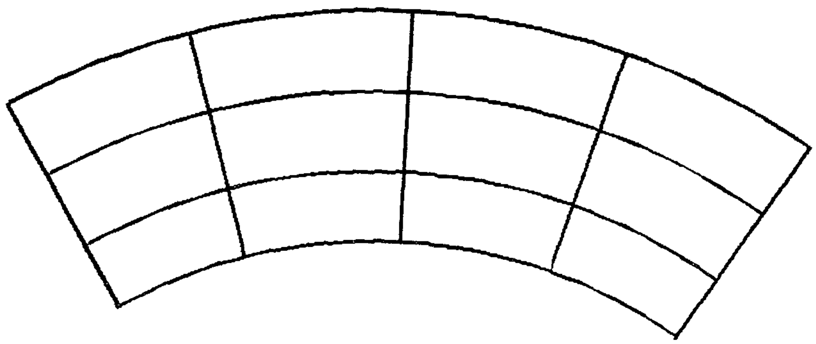


Fig. (5.1) Initial Design

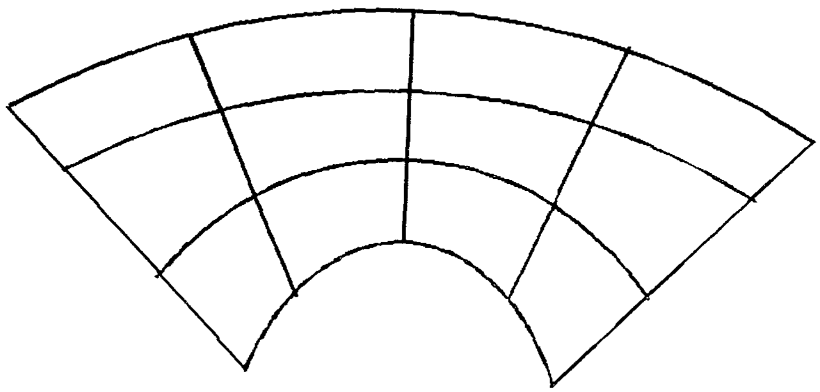


Fig. (5.2) Final Design

simple mesh in Fig. (5.1) . The 3 x 4 mesh for initial design is adequate for stress calculation. In the second Fig. (5.2) , the mesh is deformed so as to preserve a 3 x 4 uniform mesh. However, the final design has sharp corners and the mesh is not adequate for accurate stress calculation. Also this approach may lead to some elements having undesirably high aspect ratio. Therefore, when a simple modification rule is used, it is often necessary to stop the optimization process and remesh manually.

The second approach is based on the use of sophisticated automated mesh generation techniques, which generate mesh and adaptively improve it based on the calculated response.

(2) ***The element size must be sufficiently small to keep the error of approximation within acceptable bounds.***
This does not mean that very small elements should be used as they create even more problems. Chapter (2) discusses this issue.

(3) ***The aspect ratios of the elements should be close to one in order to avoid degradation of their numerical performance.***
Chapter (2) discusses this issue in detail.

Taken together, these requirements can create considerable practical problems . Which impose serious limitations on the usefulness of three dimensional models in engineering practice. In two dimensional analysis, it is practical to grade finite element meshes, so that fine meshes are used only at the critical areas. In the three dimensional case, mesh grading is a far more difficult task, and the topological constraints usualy force the analyst to use fine meshes on the entire domain if a fine mesh is required over one or more subdomains.

(5.2) MESH REFINEMENT

Due to the continuous change of boundaries in the shape optimization process, some of the elements distort badly, the finite element model becomes incapable of evaluating high stresses in the valleys of the wavy boundary. Once the mesh is not capable of accurately modeling the problem, a refined mesh should be created. Mesh refinement can be done in two ways :

(1) *Complete Remesh ;*

(2) *Localized Mesh Refinement.*

(5.2.1) Complete Remesh

Oda and Yamazaki (30) regenerated the mesh after a number of optimization iterations. Yang et al. (41) manually remeshed the optimum shape design and then restarted the optimization procedure with that design for a final correction by the optimizer.

(5.2.2) Localized Mesh Refinement

A natural way of improving the quality of finite element mesh is to increase the number of degrees of freedom. The new degrees of freedom are added in the selected regions by either increasing the order of the polynomial approximation inside the elements or by subdivision of elements. In the following are listed some of the widely used methods of mesh refinement :

(1) *h - method ;*

(2) *p - method.*

(3) *degrees of freedom fixed*

(1) h - method

In the h - method, the new degrees of freedom are added by selectively subdividing elements into the regions where finite element approximation is less accurate. An example of the implementation of this method is presented in references (17,22,26).

(2) p - method

In the p - method, the new degrees of freedom are added by increasing the order of polynomial approximation inside the selected elements

(3) degrees of freedom fixed

The objective of finite element mesh refinement methods is to improve some aspects of the selection of a discretized finite element model, in order to facilitate the best possible finite element solution.The addition of degrees of freedom to the finite element model is a natural way of improving the quality of the approximate solution. In much of the grid optimization literature, new degrees of freedom are added by either subdividing selected elements into smaller ones or by selectively incresing the order of polynomial approximation inside some elements. In many cases, however it may be desired to keep the number of degrees of freedom fixed and, under this limitation, to obtain the best possible finite element solution. Diaz et al. (11,12) did an appreciable work in this type of grid optimization. It consists of finding the location of nodes that yield the best possible finite element solution for a given number of elements and a specified order of polynomials.

(5.3) ADAPTIVE MESH REFINEMENT

Adaptive mesh refinement is a very important tool for shape optimization process using completely automated mesh generation. With this concept, information from an analysis with a trial mesh is used to identify regions of the finite element mesh element mesh which need further improvement (refinement). This refinement can take either the form of adding additional elements in the area to be refined or of increasing the order of finite element as discussed in h & p methods. The finite element mesh points are relocated whenever the boundary shape changes, and thus individual element distortion due to shape changes during the optimization process can be kept to a minimum. Thus, a good adaptive mesh refinement strategy can avoid jagged shape otherwise produced by using the coordinates of the finite element grid as design variables.

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 (11,15,16).

(2) In the second approach, the finite element model accuracy is improved by an adaptive mesh refiement scheme using strain energy density gradients to identify regions which require mesh refinement. A contour plot 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 techniques which have been already discussed.

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

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

Where

CV : SED difference cut off value ;

ΔE_{av} : The average SED variation for all elements ;

ΔE_{max} : The max SED variation in an element ;

β : A parameter to be selected based upon the problem but generally lies between 0 and 0.5.

This concept has been extended by Botkin and Bennett (1) to three dimensional structures. The mesh generation is not an inexpensive part of optimization ; it took approximately one third of the total CPU time in the applications reported by Botkin and Bennett (1). However, the subsequent improvements in the efficiency of the mesh generator reduced its cost to a few percent of the total.

Because automated mesh generation must be an integral part of the shape optimization, optimal mesh refinement is a closely related concept to optimum shape design. The two concepts of optimum mesh and optimum shape converge in the field of Kikuchi, Taylor and their coworkers. Thus their earlier work on optimal grids (12) and optimal shape modification (24) led to the combination of the two (10,23). Luch et al. (27) used automatic mesh generation at each step of optimization in designing a gas turbine disc. Queau and Trompette (33) used an automated mesh generation for several two dimensional problems. The idea of local automatic mesh generation during the process of design has been presented by Botkin (4) for plate and components modelled by triangular flat plate elements. The techniques of 3-D shape optimization for solid components have been reported by Imam (20) and demonstrated on simple cantilever beams modelled by 3-D solid finite elements. Applicationn of those techniques to the engine main bearing cap has been reported by Imam. An integrated shape design program was developed by Bennett and Botkin (1,2) for 2-D problems. The program includes finite element analysis, automatic mesh generation, and structural optimization as an integrated package.

This chapter discussed the mesh generation and refinement during shape optimization process. By now, we get a fair idea of the theory behind shape optimization process. Thus we have achieved our first objective of understanding the shape optimization process. Now we go to the practical application of shape optimization. The next chapter discusses the various capabilities of I-DEAS and how shape optimization can be achieved using the finite element analysis capability of I-DEAS.

CHAPTER 6

COMPUTER AIDED ENGINEERING PACKAGE I-DEAS

(6.1) CAE Package of I-DEAS

I-DEAS is a comprehensive Computer Aided Engineering package. It is a complete package in itself which can be used right from the concept to the final production of the product. In the following are listed some of the major features of I-DEAS :

(1) *Solid Modeling (Geomod) ;*

(2) *Drafting (Geodraw) ;*

(3) *Engineering Analysis (Supertab) ;*

(4) *Graphic Numerical Control (G.N.C.).*

The major advantage with I-DEAS is that it is interactive and menu driven . Thus it is very user friendly and convenient. The whole software has been divided into families, like Solid Modeling is one family and Engineering Analysis is other. Each family has been further divided into tasks and each task has its own subdivisions. Here our study will be restricted to the Finite Element Analysis family. To be more precise, our study will be on shape optimization and related tasks.

(6.2) Geometry Modeling Task

Geometry modeling task is used to create the finite element model of the object or to modify an existing finite element model. It is similar to the construction geometry task of the solid modeling family but is less powerful. The finite element model of an object can be created by two ways :

(1) Transfer from object modeling ;
(2) Creation of a new model.

(1) This method is used when the geometry is simple and easily transferably.

(2) This method is used in two cases
(a) When the geometry is too complex to be transferred from objet modeling.

(b) When the geometry is too simple (especially for 2-d modeling).

In most cases, there is a partial transfer of the geometry from the object modeling and the rest of the geometry is completed in the geometry modeling task. The major part of the geometry modeling is done using create wire, copy and orient, modify and delete menu's.

(6.3) Mesh Creation Task

Mesh creation task is used to create mesh areas & mesh volumes, to generate mesh, to create and modify nodes and elements and to define the material and physical properties of the object.

Creation of Mesh Areas :

The geometry created in the geometry modeling task is used to form mesh areas. Formation of mesh area decides the type of element and the type of mesh to be used for mesh generation. There are two types of mesh :

(1) Manual or Mapped mesh ;

(2) Automatic or free mesh.

In manual mesh, the user has to define the number and size of the elements along each curve while in automatic or free mesh, the user just has to define the size of the element and the I-DEAS package automatically generates the mesh. The automatic mesh can be refined at a particular point/area by giving a new local element size at that particular point. I-DEAS also has the capability of identifying a hole while defining the mesh area For that, the mesh area is created using auto_create option.

Once the mesh is formed, it can be checked by exploding it using explode option to check that the mesh area is made of right curves.

Creation of Mesh Volumes :

Mesh volumes are needed when we are dealing with 3-D object. The mesh volume for a particular piece of object is made up of the mesh areas bounding that piece. Thus the mesh volume for a cube would be made up of six mesh areas which are defined by its six faces. Once created, the mesh volumes can be checked by exploding them.

Generation of Mesh :

Generation of mesh is done using generate mesh menu. Nodes and elements are generated on the specified mesh areas and mesh volumes using generate mesh menu. Another feature of generate mesh menu is that it can generate mesh directly from the solid model of the object for simple shapes and thus saves all the trouble of creating geometry, forming mesh areas and mesh volumes and defining mesh size. Thus in case of simple objects, all one has to do is to make a solid model of the object in the solid modeling family and then generate the mesh in Finite Element family just by giving one command.

Nodes and Elements :

Nodes and elements are created by generating the mesh. Also, they can be created independently by creating nodes and then creating elements from nodes. I-DEAS also allows the option of modifying nodes and elements once they have been generated. Also, if the user doesn't like the generated mesh, he can generate a new mesh but before that, he has to delete all the existing nodes and elements in that particular mesh area or volume.

Material and Physical Properties :

At the time of creation of mesh areas, I-DEAS gives user the option of defining the Material and Physical properties of the object or accept the default values. Most of the times, user just accepts the default values to concentrate more on mesh areas at that time. Default values are actually the values for ordinary Steel. There are two separate menu's in the mesh creation task which manage the Physical and Material properties which can be changed anytime by the user.

(6.4) Boundary Conditions Task

The boundary conditions task in I-DEAS is used to define the constraints and restraints on an object. It is also used to define the magnitude and type of loading acting on the object. Since the object may be under different loading conditions at different times, IDEAS provides case management. A case set defines one particular condition of loads and restraints on an object and more than one case sets can be defined for an object.

(6.5) Model Solution Task :

The finite element problem is solved in model solution. Model solution task has further subtasks :

(1) Linear Statics Task ;

(2) Normal Mode Dynamics Task ;

(3) Constraint Mode Dynamics Task ;

(4) Heat Transfer Task ;

(5) Forces Response Task ;

(6) Potential Flow Task ;

For static loading, linear statics task is used so only linear statics task will be discussed in detail here.

Linear Statics Task :

Linear statics task is used for solving finite element problems with static loading. First the user has to specify the case set he wants to use for solution, then he has to select the executive options, whether he wants the solution in interactive mode or batch mode etc. Then the user has to select the method of solution. Verification_Only method is used when the user just wants to check the any error in the model. Most of the time, Solution_No_Restart method is used . In this method, the solution stops the moment some error is found thus avoids unnecessary calculations. The most important task before the problem is solved is to select the types of output needed. Output_Selection menu is used to select the type of output needed, for example displacement, stresses, reaction forces, strain energies etc. . Ultimately when all the formalities are done, Solve_Linear_Statics provides the solution. Depending upon the complexity of the problem, I-DEAS might take one to twenty minutes in general to get the solution.

(6.6) Post Processing Task

Once the solution has been prepared in the model solution task, the results are viewed, interpreted and processed in the post processing task. The first step is to go to analysis dataset selection menu to choose the type of result which the user needs to view. Then depending upon the requirement, there are Contour, Criterian and Deformed Geometry options to view the results. A plot of stresses can be taken using XYZ_Plot menu. The most important and most commonly used option is Contour. It gives the distribution of stresses or forces or strain energies over the selected group of elements and is best to visualize the results.

(6.7) ADAPTIVE MESHING

Adaptive meshing task is used to get an optimal mesh for getting best results. Adaptive meshing can be done on the following two basis :

(1) Elemental Distortion

(2) Analysis Results

(1) Elemental Distortion :

Sometimes the mesh generated in the mesh creation task has elements with distortion exceeding the allowable limit. Adaptive meshing technique analysis the distortion summary of the existing mesh, locates the elements to be modified and then modifies the mesh to bring the elemental distortion down to the allowable level.

(2) Analysis Results :

This is the most common use of adaptive meshing to get a better solution. Most of the time, the refinement is done based upon the strain energy distribution. This technique analysis the strain energy distribution over the object and identifies the regions which need refinement for getting a better solution and then refines the selected regions. The desired refinement may not take place in a single step, so it is an iterative process and can be continued till acceptable results are obtained.

Method of Modification :

The method of adaptive meshing can be choosen depending upon the type of the problem. I-DEAS uses the following methods for adaptive meshing :

(1) By moving nodes

(2) By splitting elements

(3) Complete Remesh

The first method just shifts the node positions to get optimal mesh and is used only when smal changes in mesh are required. The second method splits the current elements to get more elements in the required region to refine the mesh and the third method generates a completely new mesh. The user can combine these methods to get better results. Most of the times, method one and method two are used in tandom to get the benefits of both. The second method works only when a surface is attached to the selected mesh area. Because of this reason, the second method is mostly used when we start with mapped meshing as mapped meshing generates a surface on the mesh area using Coon's patch technique. Complete remesh method is usually used when the user starts with automatic free mesh.

(6.8) OPTIMIZATION

After the part has been completely analyzed, optimization task is used to optimize the design. The following steps have to be followed by the user to achieve optimization :

(1) To create an optimization design model

(2) To setup optimization

(3) To control solution

(4) To solve

(5) To check for solution errors

(6) To view the results

(7) To update Finite Element model

To Create An Optimization Design Model :

In this step, the user has to create or activate an optimization design model which will store all the optimization results. This is done in the Manage_Designs menu of the optimization task.

To Setup Optimization :

This is the most important step in the optimization process. This is done in the Setup-Optimization menu. The whole optimization problem is defined at this step. It has the following sub menus :

(a) *Optimization node group ;*

(b) *Optimization element group ;*

(c) *Optimization variables ;*

(d) *Optimization constraint set.*

(a) Optimization Node Group :

Optimization node group includes all the nodes which are to be included in the optimization problem. This group is very important for shape optimization problem as it optimizes the shape by node movement at the boundary. All the required node movement has to be specified in this menu. Care has to be taken avoid the distortion of the elements. This group only includes those nodes which are required to be moved for optimization.

(b) Optimization Element Group :

Optimization element group includes all the nodes which are to be included in the optimization problem. In case of shape optimization, since the whole object has to be optimized to obtain an optimal shape, all the elements are included in the optimization element group.

(c) Optimization Variables :

This menu is used to define all the variables for the given optimization problem. In case of shape optimization, shape is the variable so user chooses shape redesign as the variable.

(d) Optimization Constraint Set :

Optimization constraint set is created to define the constraints of the optimization problem. Most common constraints are stress constraint, deformation constraint and mass constraint.

<u>To Control Solution</u> :

Once the optimization problem is defined, it is necessary to specify the type of output desired, which is done in the Control Solution menu. The following selections can be made using I-DEAS :

<u>Method</u> :

The user can define the method according to the type of problem, for example Linear Statics is choosen for a problem with static loading.

<u>Iteration Control</u> :

Iteration control can be provided to limit the iterations. Also convergence is defined here. In the case of shape optimization problems, since there is no iterations, it is specified as zero.

<u>Output Selection</u> :

Here the user chooses the type of output needed like stresses, displacement etc.

<u>To Solve</u> :

Finally when everything has been set, the problem is given to optimization solver for solution.

<u>To View the Results</u> :

After the optimization problem has been solved, the results are viewed by going to Display_Results menu. The optimized object can be viewed, its iteration history, mass history, stress history can be observed apart from other things. This menu is similar to post processing task.

<u>Update Object</u> :

Once the optimized object has been viewed, if the user is satisfied with the result, he can transfer the result to his original finite element model. In other words, he can update his finite element model. Sometimes in shape optimization problem, a situation might arise where the user may want to modify the shape in the pattern specified but he might like to modify the object at a smaller magnitude or may be bigger magnitude, thus an option is provided in I-DEAS at the time of update by which the user can do so. Thus after the original finite element has been updated, the user has the optimized object for presentation. But in the case of shape optimization, since it is not iterative process, the updated model is only a step towards the final desired shape. The whole optimization process is repeated again and again until the final desired optimized shape is obtained.

In this chapter we discussed the use of I-DEAS for shape optimization problem with emphasis on shape optimization. The next chapter discusses the use of I-DEAS for the shape optimization of a given design of wall bracket.

CHAPTER 7

COMPUTER AIDED SHAPE OPTIMIZATION OF WALL BRACKET USING I-DEAS

Fig (7.1) gives the detailed geometry of the Wall Bracket before the optimization process Flow chart (7.2) shows the process involved in the Shape Optimization using I-DEAS. The following steps are involved :

(7.1) Solid Modeling of the initial object

The given wall bracket has a uniform thickness, so we make a ***profile*** of the wall bracket in the ***construction geometry*** task using the given dimensions and the ***extrude*** it in the ***create*** menu of the ***object modeling*** task to obtain the desired solid model of the object as shown in the fig. ().

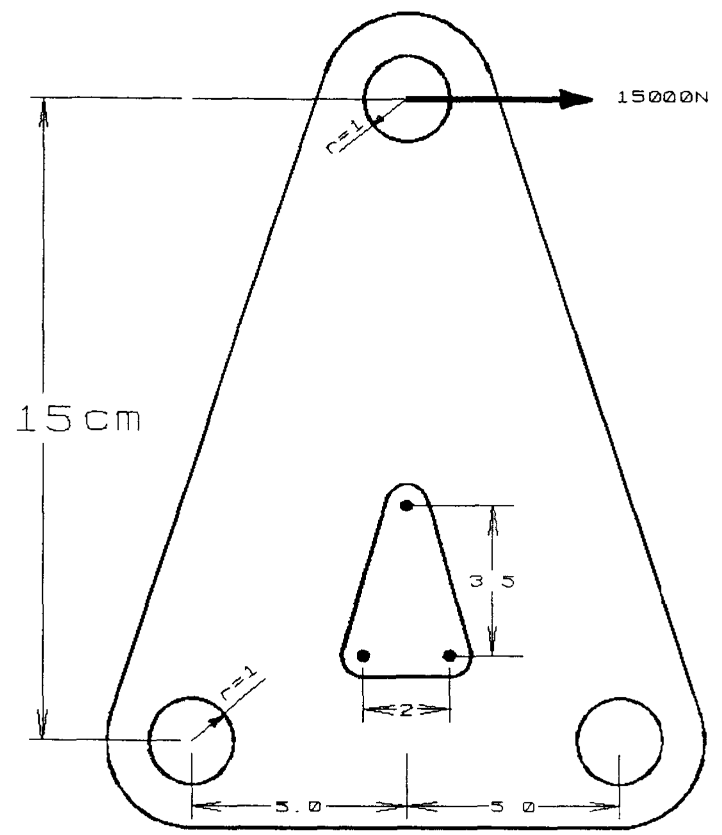
(7.2) Finite Element Modeling

There are two ways to start the Finite Element Modeling of an object :

(1) Transfer the data from solid modeling ;

(2) Creation of a new wireframe in the geometry modeling task of the finite element module.

Both ways are quite feasible in the Finite Element Analysis Wall Bracket but since the thickness of the wall bracket is too small and uniform, it was decided to analyze it by 2-d



Yield Stress = 8.0E08 N/m2
Density = 0.00761 Kg/cm2
Thickness = 0.3 cm
E=20.74E06

INITIAL DESIGN

Fig. (7.1)

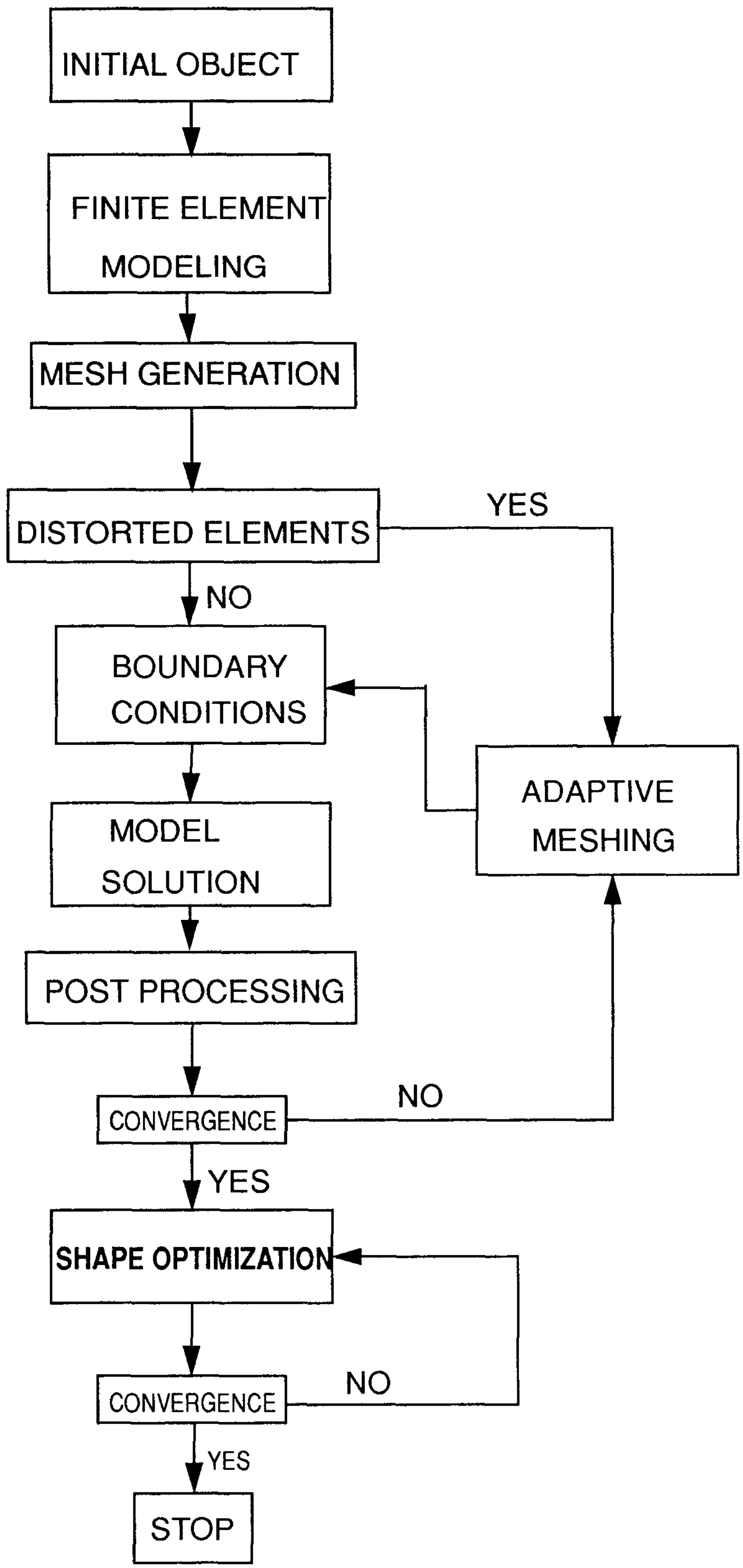


Fig. (7.2) Shape Optimization using I-DEAS

modeling (easy and quick), for which creation of a new wireframe is quite convenient so a new wireframe was created for finite element modeling.

(7.2.1) Creation of wireframe

Wireframe was created in the ***create wire*** menu of the ***geometry modeling*** task. It was done by first making circles at the three points, then tangents were drawn to these circles, unwanted curves were trimmed in the ***modify*** menu using ***divide curves*** method and then ***join curves*** option to get the final desired wireframe.

(7.2.2) Choice of meshing

It was decided to have ***automatic*** and ***free*** meshing because of the obvious and already discussed benefits they provide.

(7.2.3) Formation of mesh areas

This is done in ***mesh generation*** task. ***Auto-create*** option was used to create a single mesh area for the whole object. The advantage of this method was easy and accurate definition the three holes present in the object. Without ***auto-create*** option, it is very difficult and cumbersome to define a hole in finite element modeling.

(7.3) Mesh Generation

This is also done in the ***mesh generation*** task. The most important step at this point is the right choice of element type. ***Thin quadrilateral shell elements*** have been choosen in our case. Local mesh elements were defined at the two support holes to give a finer mesh at those areas for better results.

With generation of the required mesh, the next step is to check the nodes and elements The mesh area was checked for ***free edges*** and no free edges were found. ***Nodal coincidence*** was checked to remove unwanted coincident nodes. No coincident nodes were found. Similarly ***element coincidence*** was checked and no coincident elements were found. Also all the elements were checked for ***interior angle*** and ***distortion*** and all the elements were found to be within the allowable limits. The elemental ***bandwidth*** and ***nodal wavefront*** were also optimized.

(7.4) Boundary Conditions

This is done in the ***boundary conditions*** task. At this step, we define the loading and the restraints on the wall bracket as shown in fig. . The nodes on two support holes are fixed and the load is applied on the top of hole as shown. After creating the ***load set*** and the ***restraint set***, a ***Case Set*** is defined to be used for Model Solution.

The last step before going for the model solution is to define the Physical and the Material Properties of the object. This can be done right at the time of formation of the mesh area but can be postponed till the end by accepting default values at that time. Most important ***Physical property*** is the thickness and important ***Material Properties*** are Young's Modulus of elasticity, Poisson's Ratio, ultimate strength, yield strengh etc.

(7.5) Model Solution

The case set defined in the boundary conditions task is used in the model solution task to obtain the final solution. The ***analysis data set*** is formed in the ***Execution Options*** menu to select the type of output needed and then model solution is obtained for further analysis in the ***post processing*** task.

(7.6) Post Processing

This is done in Post Processing task. First we go to ***Analysis Dataset Selection*** to select the type of output to be processed from the analysis dataset. Then we go to ***contour*** menu to view the results in different formats. Deformed geometry can be viewed by going to ***deformed geometry*** menu. In the post processing, the display options play major role in the depicting of results in various forms. ***Arrow plot*** menu gives us the option of having the arrow plot. A plot of stresses is taken by going to ***XYZ Plot*** menu

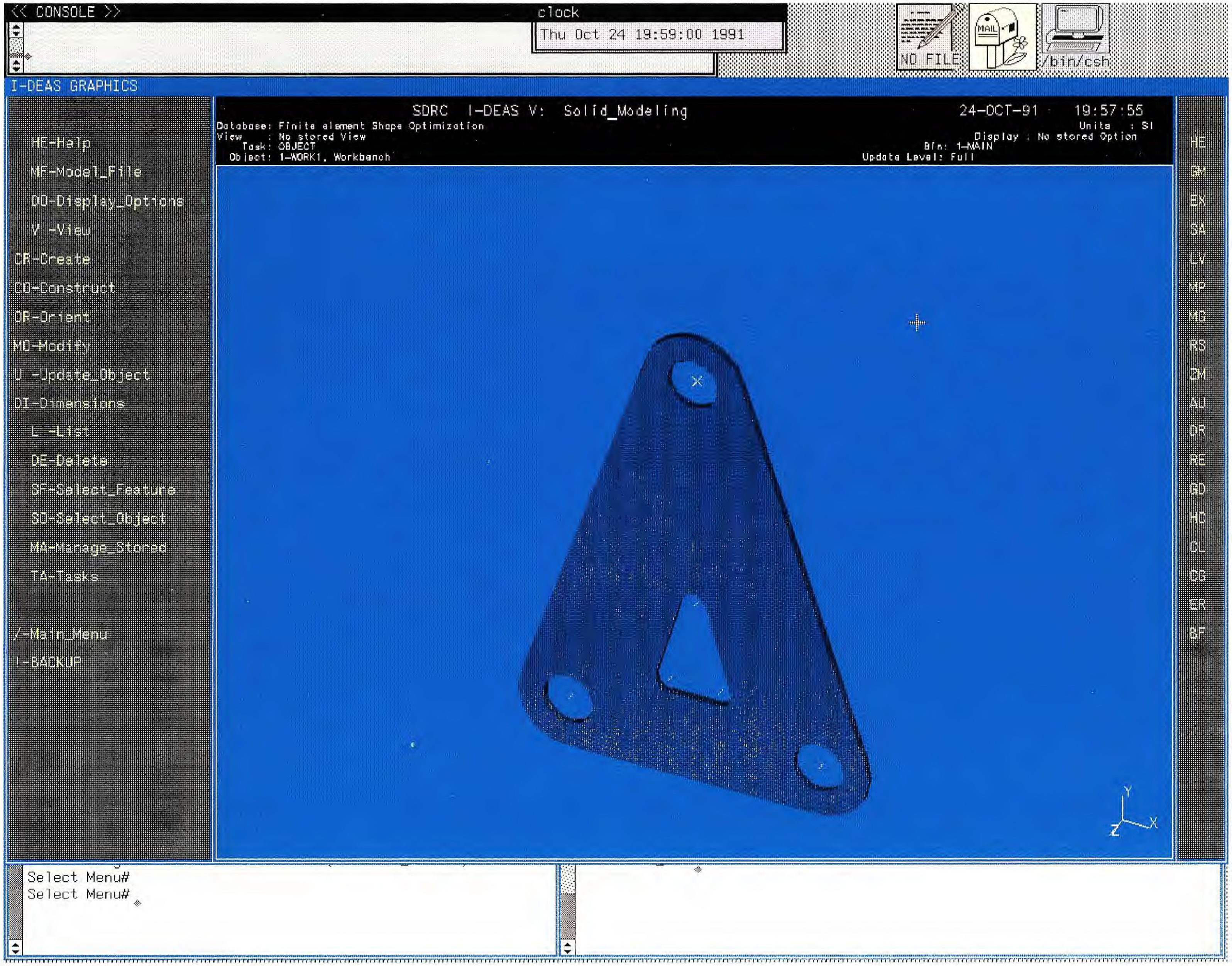
(7.7) Shape Optimization of Wall Bracket

Till now, we were just analyzing the initial wall bracket. Now comes the real task of shape optimization. Once we have analyzed the existing design, we get a fair idea of the areas which could be trimmed to get the final optimized shape. First we create a design model using ***manage designs***. The next step is to define the optimization problem, so we go to ***setup optimization*** menu. First we create an ***optimization node group*** to select all the nodes for which we need to change the position. Then we define the displacement of each of the selected nodes. This is a very tedious process as we don't know how the displacement of each node is going to affect all the related elements and we have to keep the element distortion within limits to avoid misleading results. The best thing is to ***sketch*** the modified nodes after the changes to visualize the node movement. The next thing is to form an optimization element group. In this problem we include all the elements in the optimization node group. Nextly we have to define the optimization variables, we choose ***shape redesign*** as the variable. The last thing in the setup routine is to define the ***optimization constraint set***. Since we just have to bother about the upper limit of the stress, so Max. Stress is the only constraint we have.

After the problem is defined, the type of solver to be used and the type of output required is defined in the Control Solution menu. ***Linear Statics*** method of solution is used. The kind of output needed like stresses, displacement etc. are chosen in the output selection menu. Finally the problem is given to the solver to sove.

After the solution has been obtained, the results are viewed the most important things in this case are the ***elemental distortion, max. stress*** and the ***weight reduction***. If the results are good then the finite element model is updated to include the optimization changes, and the next optimization cycle is started until the final optimized shape is obtained.

This chapter discussed the procedure involved in the shape optimization of wall bracket. The next chapter discusses the results and the conclusions drawn from that results.



<< CONSOLE >>
clock
Thu Oct 24 19:59:00 1991
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
SDRC I-DEAS V: Solid_Modeling
24-OCT-91 19:57:55
Database: Finite element Shape Optimization
View : No stored View
Task: OBJECT
Object: 1-WORK1, Workbench
Units : SI
Display : No stored Option
Bin: 1-MAIN
Update Level: Full
HE-Help
MF-Model_File
DO-Display_Options
V -View
CR-Create
CO-Construct
OR-Orient
MO-Modify
U -Update_Object
DI-Dimensions
L -List
DE-Delete
SF-Select_Feature
SO-Select_Object
MA-Manage_Stored
TA-Tasks
/-Main_Menu
!-BACKUP
HE
GM
EX
SA
LV
MP
MG
RS
ZM
AU
DR
RE
GD
HC
CL
CG
ER
BF
Y
X
Z
Select Menu#
Select Menu#

FIG. (8.1)

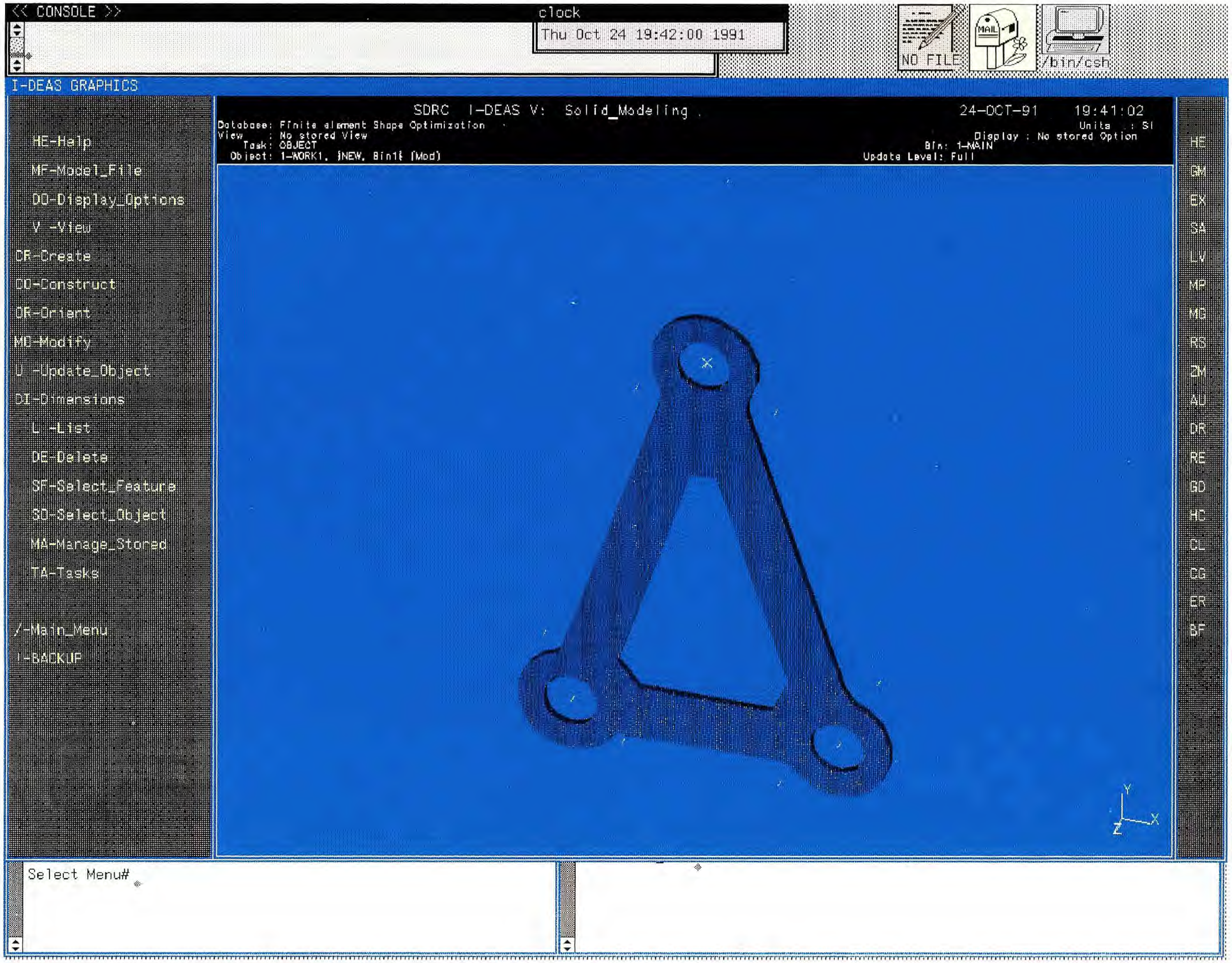

<< CONSOLE >>
clock
Thu Oct 24 19:42:00 1991
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
SDRC I-DEAS V: Solid_Modeling
24-OCT-91 19:41:02
Database: Finite element Shape Optimization
View : No stored View
Task: OBJECT
Object: 1-WORK1, (NEW, Bin1) (Mod)
Units : SI
Display : No stored Option
Bin: 1-MAIN
Update Level: Full
HE-Help
MF-Model_File
DO-Display_Options
V -View
CR-Create
CO-Construct
OR-Orient
MO-Modify
U -Update_Object
DI-Dimensions
L -List
DE-Delete
SF-Select_Feature
SO-Select_Object
MA-Manage_Stored
TA-Tasks
/-Main_Menu
!-BACKUP
HE
GM
EX
SA
LV
MP
MG
RS
ZM
AU
DR
RE
GD
HC
CL
CG
ER
BF
Y
X
Z
Select Menu#

FIG. (8.2)

71

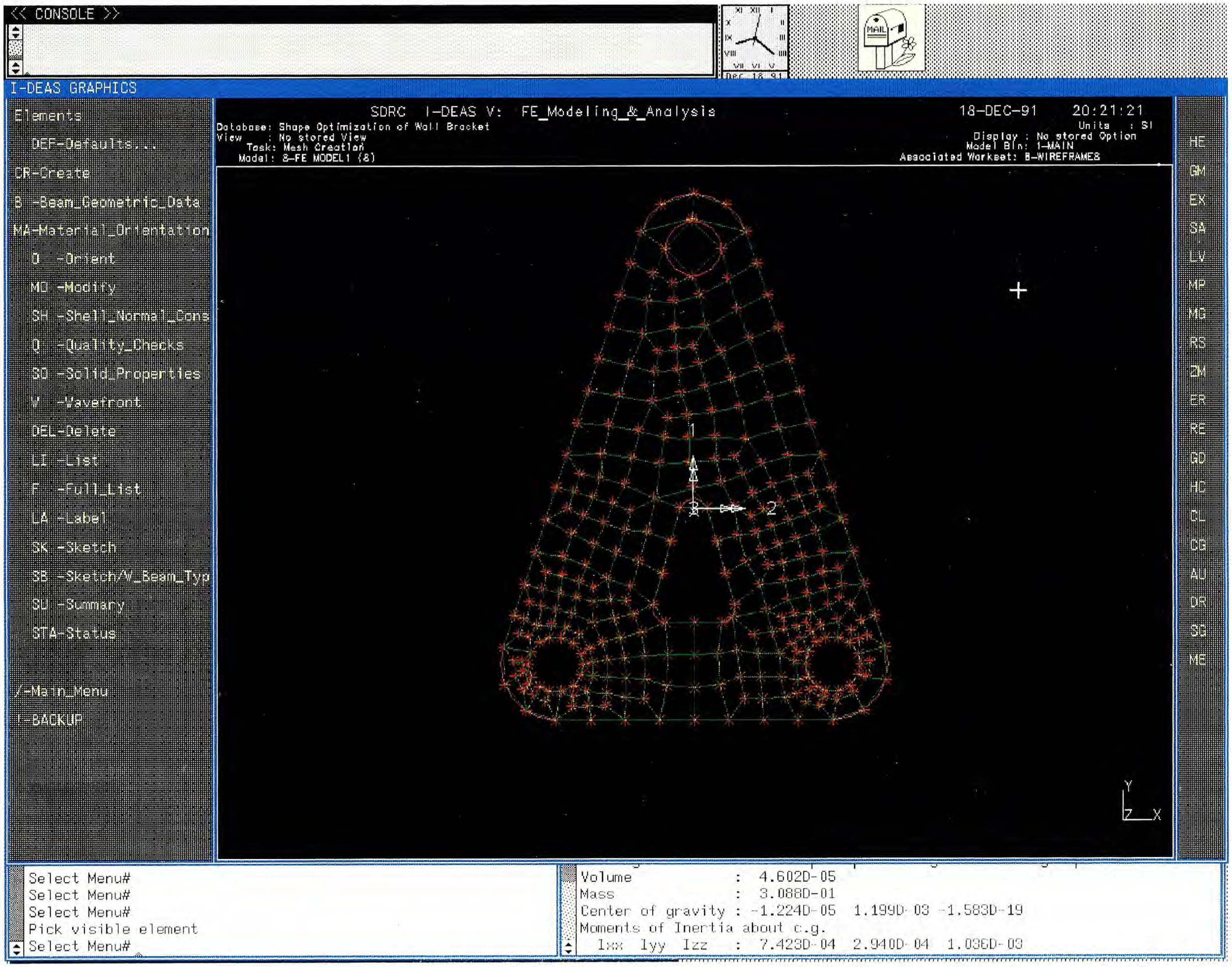
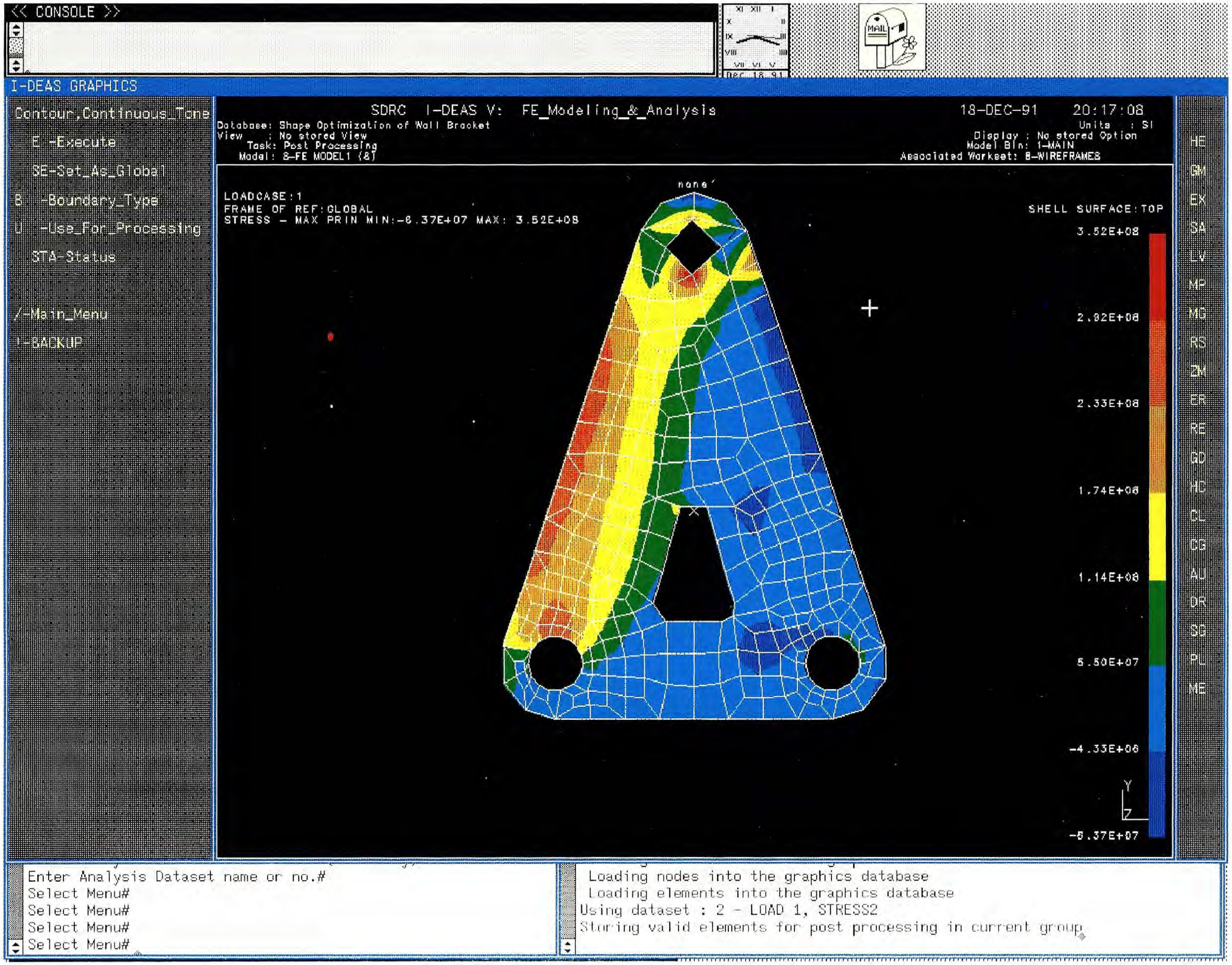

<< CONSOLE >>
I-DEAS GRAPHICS
Elements
DEF-Defaults...
CR-Create
B -Beam_Geometric_Data
MA-Material_Orientation
O -Orient
MO -Modify
SH -Shell_Normal_Cons
Q -Quality_Checks
SO -Solid_Properties
V -Wavefront
DEL-Delete
LI -List
F -Full_List
LA -Label
SK -Sketch
SB -Sketch/W_Beam_Typ
SU -Summary
STA-Status
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
18-DEC-91 20:21:21
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Mesh Creation
Model: 8-FE MODEL1 (8)
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 8-WIREFRAMES
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
1
2
Y
Z X
Select Menu#
Select Menu#
Select Menu#
Pick visible element
Select Menu#
Volume : 4.602D-05
Mass : 3.088D-01
Center of gravity : -1.224D-05 1.199D-03 -1.583D-19
Moments of Inertia about c.g.
Ixx Iyy Izz : 7.423D-04 2.940D-04 1.036D-03

FIG. (8.3)



<< CONSOLE >>
I-DEAS GRAPHICS
Contour,Continuous_Tone
E -Execute
SE-Set_As_Global
B -Boundary_Type
U -Use_For_Processing
STA-Status
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
18-DEC-91 20:17:08
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 8-FE MODEL1 (8)
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 8-WIREFRAMES
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-6.37E+07 MAX: 3.52E+08
SHELL SURFACE:TOP
3.52E+08
2.92E+08
2.33E+08
1.74E+08
1.14E+08
5.50E+07
-4.33E+06
-6.37E+07
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Enter Analysis Dataset name or no.#
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Loading nodes into the graphics database
Loading elements into the graphics database
Using dataset : 2 - LOAD 1, STRESS2
Storing valid elements for post processing in current group

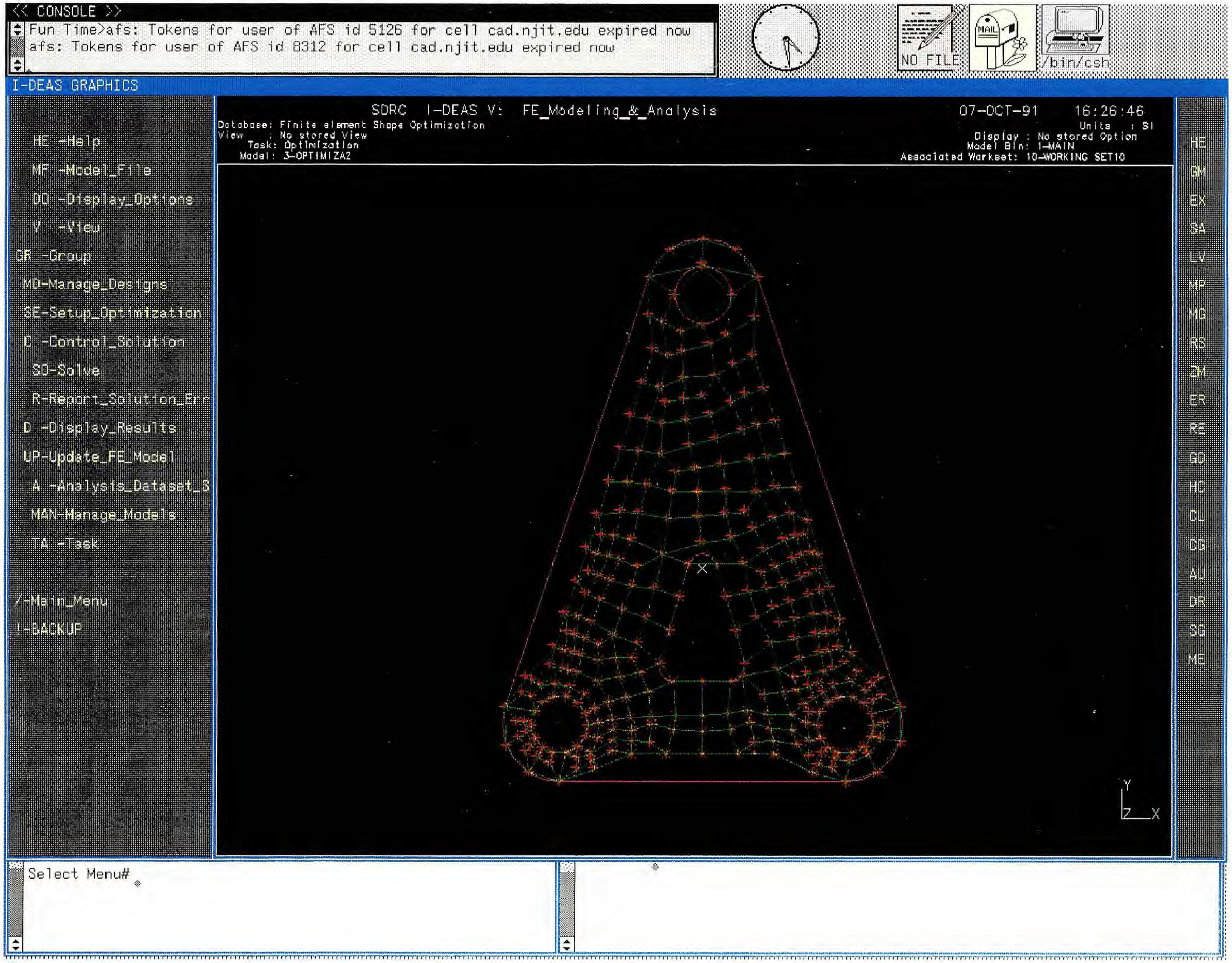
FiG. (8.3a)


<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 15:50:05
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 2-OPTIMIZA1
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 9-WORKING SETS
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
Finite element Shape Optimization
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-68334.77 MAX: 3.94E+05.
SHELL SURFACE:TOP
3.94E+05
3.28E+05
2.62E+05
1.96E+05
1.30E+05
63784.67
-2275.05
-68334.77
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Select Menu#
Iteration Number
Actual Mass
Feasible Mass
Lower Bound Mass
Non-structural Mass
0
2.684E-01
2.684E-01
2.424E-01
0.000E+00

FIG. (8.4)


<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
Display_Results,Mass_Hi
LI-List
XY-XY_Plot
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 15:40:04
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 4-OPTIMIZA3
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 11-WORKING SET11
Finite element Shape Optimization
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-1.19E+05 MAX: 5.61E+05
SHELL SURFACE:TOP
5.61E+05
4.64E+05
3.67E+05
2.69E+05
1.72E+05
75234.78
-21890.14
-1.19E+05
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Select Menu#
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Iteration Number
Actual Mass
Feasible Mass
Lower Bound Mass
Non-structural Mass
0
2.424E-01
2.424E-01
2.424E-01
0.000E+00

FIG. (8.5)



<< CONSOLE >>
Fun Time>afs: Tokens for user of AFS id 5126 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8312 for cell cad.njit.edu expired now
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
07-OCT-91 16:26:46
Database: Finite element Shape Optimization
View : No stored View
Task: Optimization
Model: 3-OPTIMIZA2
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 10-WORKING SET10
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_3
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
HE
GM
EX
SA
LV
MP
MG
RS
DM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Y
Z X
Select Menu#

FIG. (8.6)

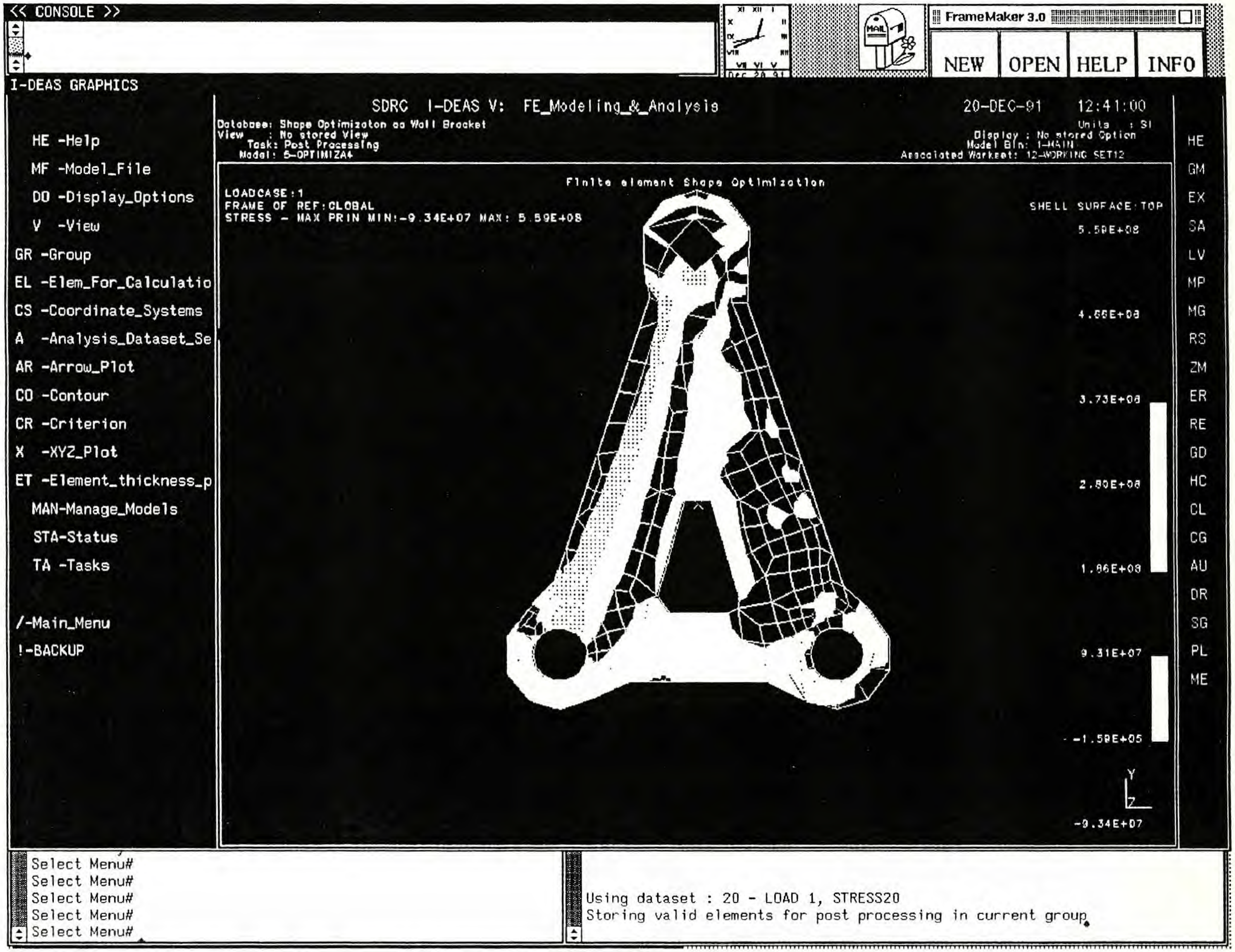

<< CONSOLE >>
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
20-DEC-91 12:41:00
Database: Shape Optimizaton as Wall Bracket
View : No stored View
Task: Post Processing
Model: 5-OPTIMIZA4
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 12-WORKING SET12
Finite element Shape Optimization
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-9.34E+07 MAX: 5.59E+08
SHELL SURFACE:TOP
5.59E+08
4.66E+08
3.73E+08
2.80E+08
1.86E+08
9.31E+07
-1.59E+05
-9.34E+07
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
EL -Elem_For_Calculatio
CS -Coordinate_Systems
A -Analysis_Dataset_Se
AR -Arrow_Plot
CO -Contour
CR -Criterion
X -XYZ_Plot
ET -Element_thickness_p
MAN-Manage_Models
STA-Status
TA -Tasks
/-Main_Menu
!-BACKUP
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Using dataset : 20 - LOAD 1, STRESS20
Storing valid elements for post processing in current group

FIG. (8.7)

<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 15:26:57
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 6-OPTIMIZA5
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 13-WORKING SET13
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
none
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-93390.09 MAX: 5.59E+05
SHELL SURFACE:TOP
5.59E+05
4.66E+05
3.73E+05
2.80E+05
1.86E+05
93056.30
-186.90
-93390.09
Y
Z
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Select Menu#
Iteration Number
Actual Mass
Feasible Mass
Lower Bound Mass
Non-structural Mass
0
2.424E-01
2.424E-01
2.401E-01
0.000E+00

FiG. (8.8)

<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 15:22:09
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 7-OPTIM6
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 14-WORKING SET14
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
Finite Element Analysis
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-93512.17 MAX: 5.55E+05
SHELL SURFACE:TOP
5.55E+05
4.63E+05
3.70E+05
2.77E+05
1.85E+05
91850.13
-831.02
-93512.17
HE
GM
EX
SA
LV
MP
MC
RS
2M
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Select Menu#
Iteration Number
Actual Mass
Feasible Mass
Lower Bound Mass
Non-structural Mass
0 2.401E-01 2.401E-01 2.328E-01 0.000E+00

Fig. (8.9)

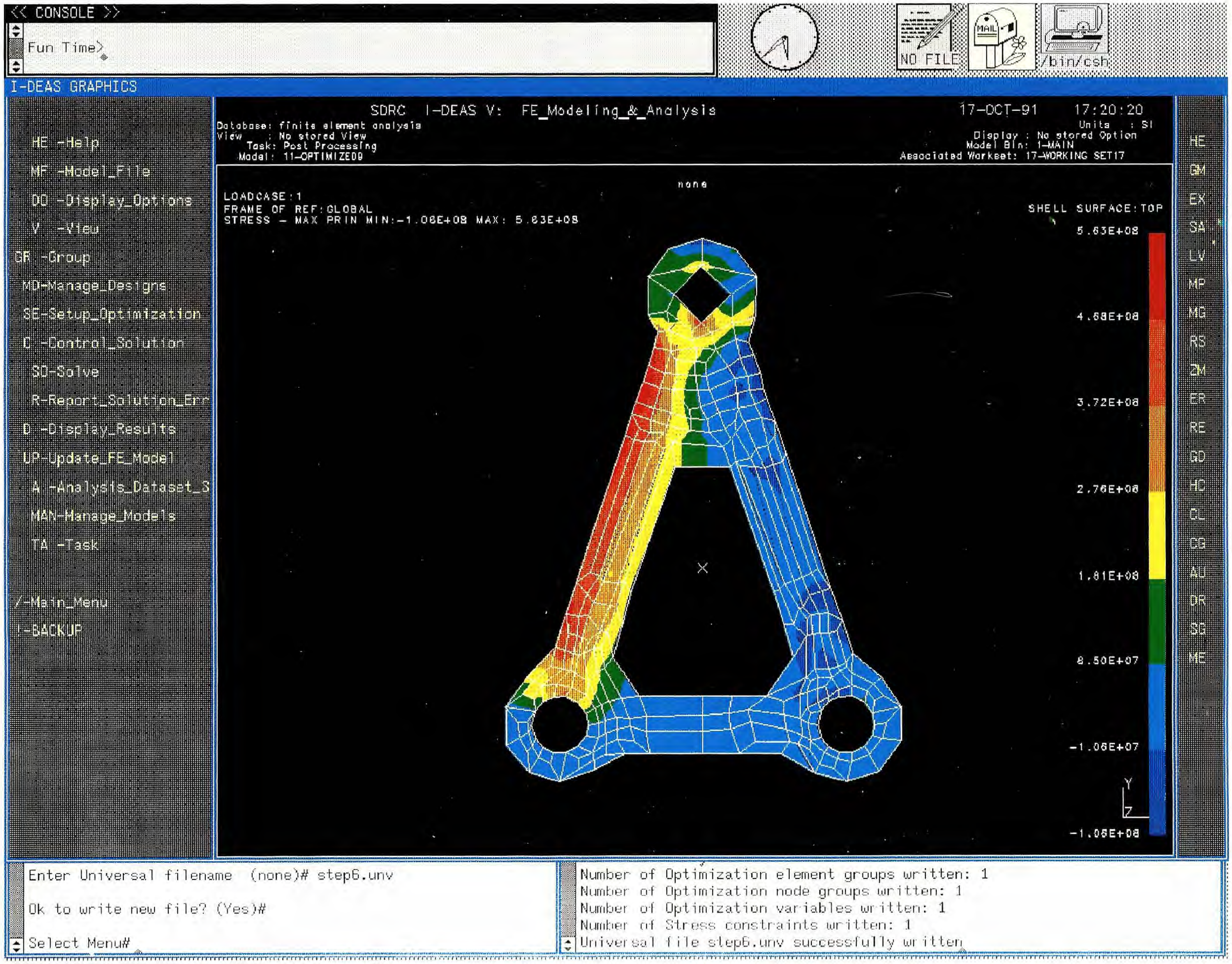
<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 14:58:21
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 9-OPTIM7
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 15-WORKING SET15
Finite Element Analysis
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-94482.82 MAX: 5.52E+05
SHELL SURFACE:TOP
5.52E+05
4.60E+05
3.67E+05
2.75E+05
1.83E+05
90282.41
-2090.21
-94482.82
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
HE GM EX SA LV MP MG RS 2M ER RE GD HC CL CG AU DR SG ME
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Select Menu#
Iteration Number 0
Actual Mass 2.312E-01
Feasible Mass 2.312E-01
Lower Bound Mass 2.013E-01
Non-structural Mass 0.000E+00

FIG. (8.12)


<< CONSOLE >>
afs: Tokens for user of AFS id 8790 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8186 for cell cad.njit.edu expired now
fmprintdriver.ps: Using /afs/cad/sun4/frame/.fminit2.0.ps
/bin/csh
FrameMaker 3.0
NEW
OPEN
HELP
INFO
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
13-DEC-91 14:51:00
Database: Shape Optimization of Wall Bracket
View : No stored View
Task: Post Processing
Model: 10-OPTIM8
Units : MM
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 16-WORKING SET16
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
none
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-94462.82 MAX: 5.52E+05
SHELL SURFACE:TOP
5.52E+05
4.60E+05
3.67E+05
2.75E+05
1.83E+05
90282.41
-2090.21
-94462.82
Y
Z
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Select Menu#
Select Menu#
Enter Iteration range (All)#
Select Menu#
Select Menu#
Iteration Number
Actual Mass
Feasible Mass
Lower Bound Mass
Non-structural Mass
0
1.984E-01
1.984E-01
1.877E-01
0.000E+00

Fig. (8.11)

Fig. (8.12)


<< CONSOLE >>
Fun Time>
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
MD-Manage_Designs
SE-Setup_Optimization
C -Control_Solution
SO-Solve
R-Report_Solution_Err
D -Display_Results
UP-Update_FE_Model
A -Analysis_Dataset_S
MAN-Manage_Models
TA -Task
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
17-OCT-91 17:20:20
Database: finite element analysis
View : No stored View
Task: Post Processing
Model: 11-OPTIMIZED9
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 17-WORKING SET17
none
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-1.06E+08 MAX: 5.63E+08
SHELL SURFACE:TOP
5.63E+08
4.68E+08
3.72E+08
2.76E+08
1.81E+08
8.50E+07
-1.06E+07
-1.06E+08
Y
Z
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
ME
Enter Universal filename (none)# step6.unv
Ok to write new file? (Yes)#
Select Menu#
Number of Optimization element groups written: 1
Number of Optimization node groups written: 1
Number of Optimization variables written: 1
Number of Stress constraints written: 1
Universal file step6.unv successfully written

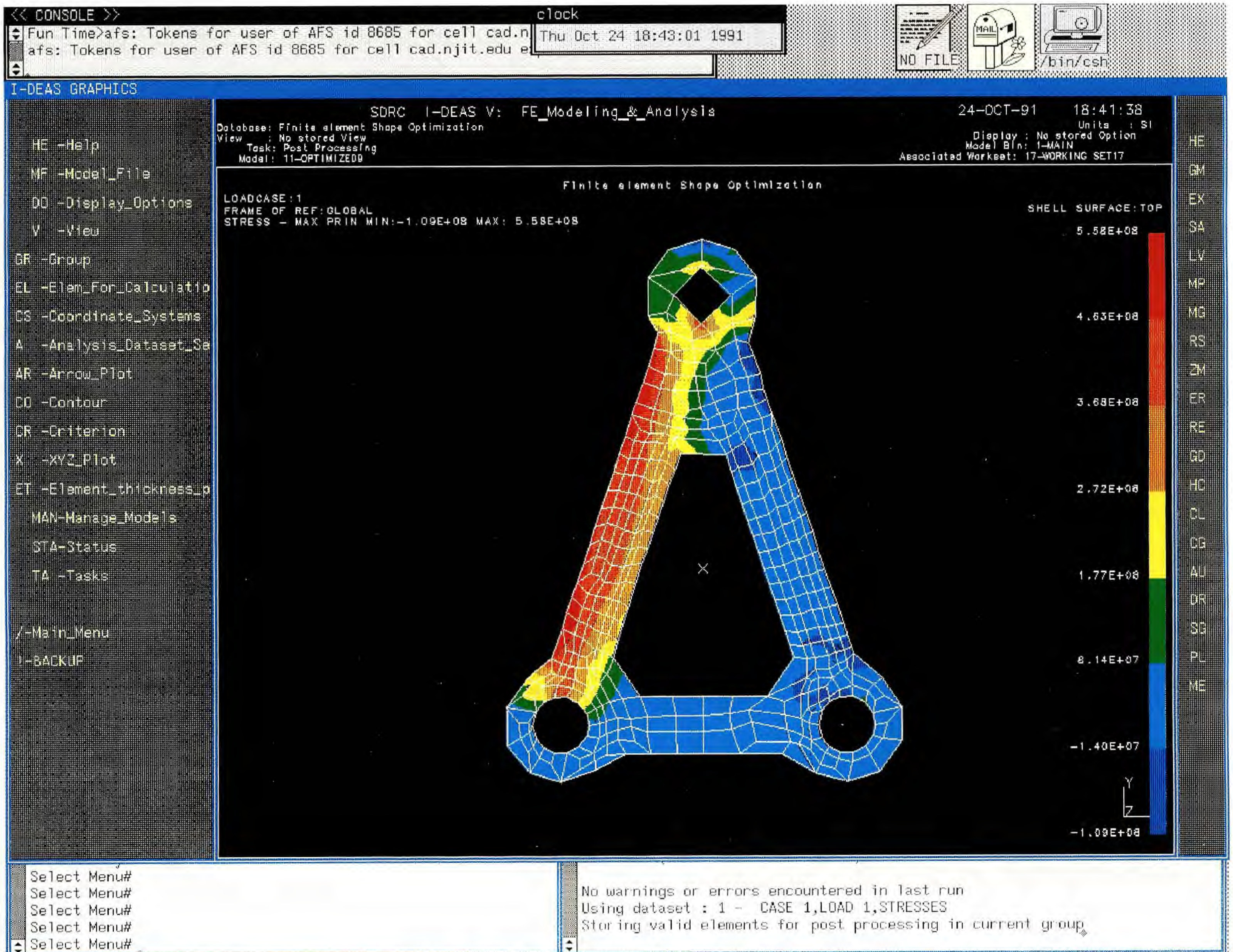

<< CONSOLE >>
clock
Fun Time>afs: Tokens for user of AFS id 8685 for cell cad.n
afs: Tokens for user of AFS id 8685 for cell cad.njit.edu e
Thu Oct 24 18:43:01 1991
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
HE -Help
MF -Model_File
DO -Display_Options
V -View
GR -Group
EL -Elem_For_Calculatio
CS -Coordinate_Systems
A -Analysis_Dataset_Se
AR -Arrow_Plot
CO -Contour
CR -Criterion
X -XYZ_Plot
ET -Element_thickness_p
MAN-Manage_Models
STA-Status
TA -Tasks
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
24-OCT-91 18:41:38
Database: Finite element Shape Optimization
View : No stored View
Task: Post Processing
Model: 11-OPTIMIZED9
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 17-WORKING SET17
Finite element Shape Optimization
LOADCASE:1
FRAME OF REF:GLOBAL
STRESS - MAX PRIN MIN:-1.09E+08 MAX: 5.58E+08
SHELL SURFACE:TOP
5.58E+08
4.63E+08
3.68E+08
2.72E+08
1.77E+08
8.14E+07
-1.40E+07
-1.09E+08
Y
Z
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Select Menu#
No warnings or errors encountered in last run
Using dataset : 1 - CASE 1,LOAD 1,STRESSES
Storing valid elements for post processing in current group

FIG. (8.13)

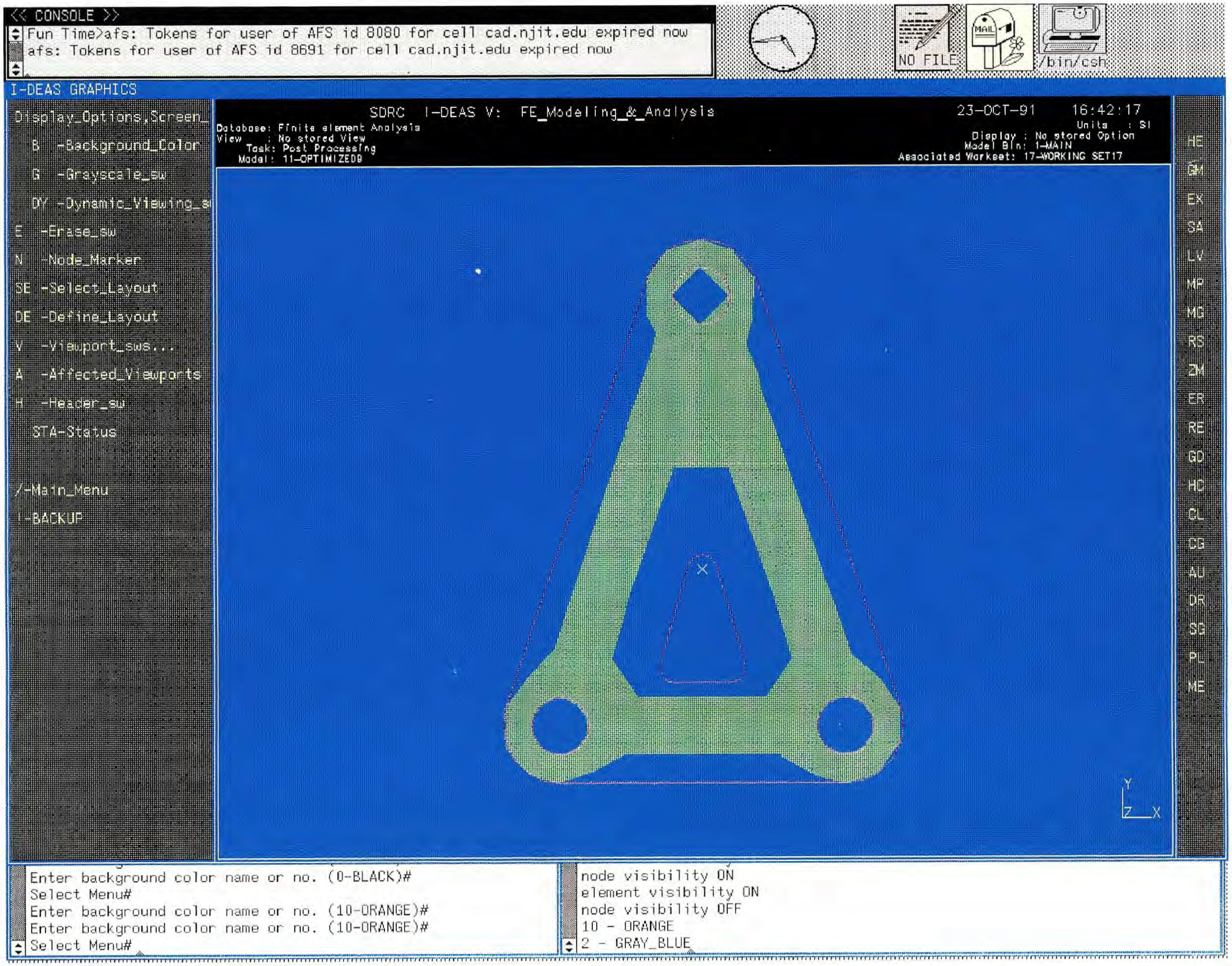
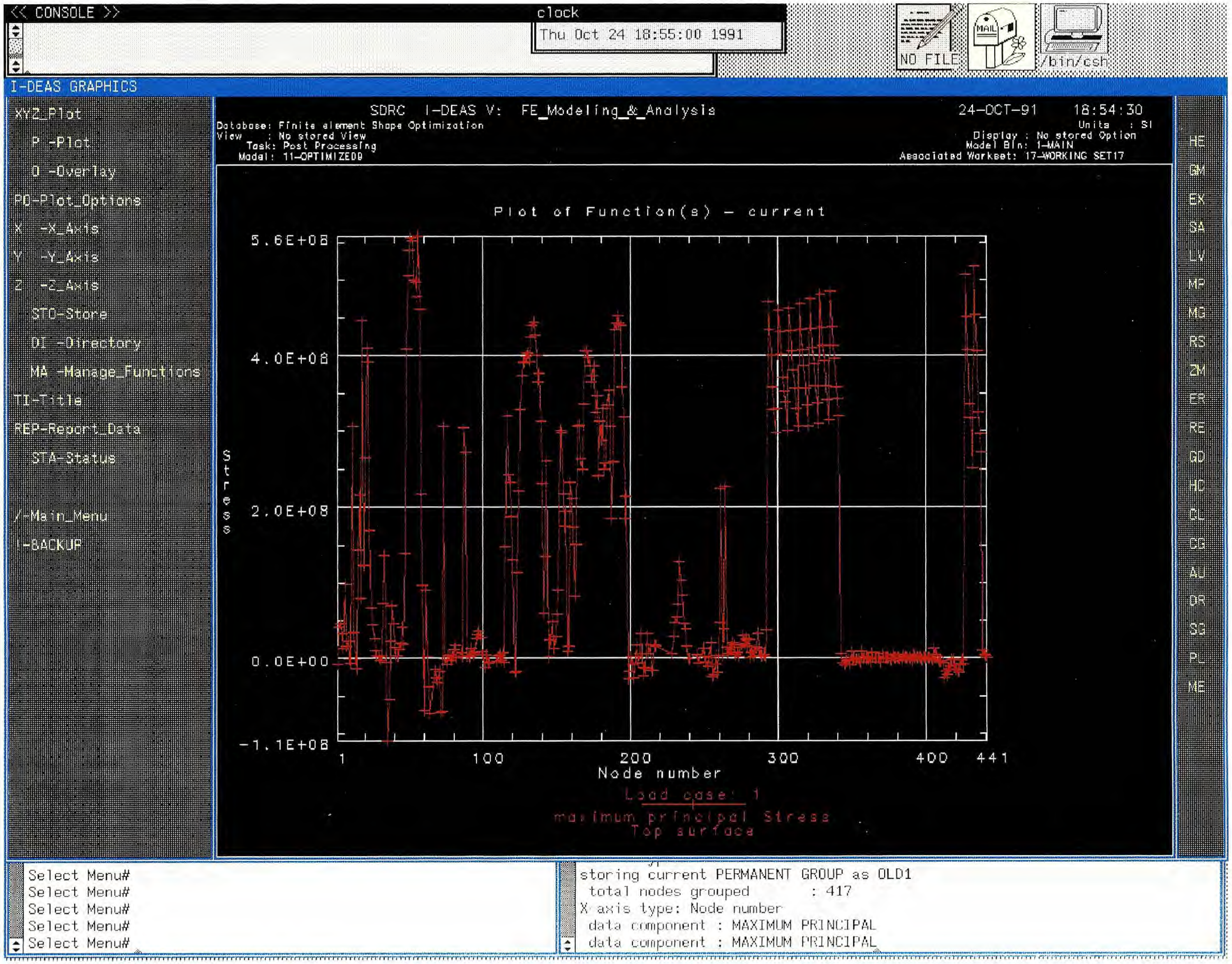

<< CONSOLE >>
Fun Time>afs: Tokens for user of AFS id 8080 for cell cad.njit.edu expired now
afs: Tokens for user of AFS id 8691 for cell cad.njit.edu expired now
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
SDRC I-DEAS V: FE_Modeling_&_Analysis
23-OCT-91 16:42:17
Database: Finite element Analysis
View : No stored View
Task: Post Processing
Model: 11-OPTIMIZED9
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 17-WORKING SET17
Display_Options,Screen_
B -Background_Color
G -Grayscale_sw
DY -Dynamic_Viewing_s
E -Erase_sw
N -Node_Marker
SE -Select_Layout
DE -Define_Layout
V -Viewport_sws...
A -Affected_Viewports
H -Header_sw
STA-Status
/-Main_Menu
!-BACKUP
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Y
Z
X
Enter background color name or no. (0-BLACK)#
Select Menu#
Enter background color name or no. (10-ORANGE)#
Enter background color name or no. (10-ORANGE)#
Select Menu#
node visibility ON
element visibility ON
node visibility OFF
10 - ORANGE
2 - GRAY_BLUE

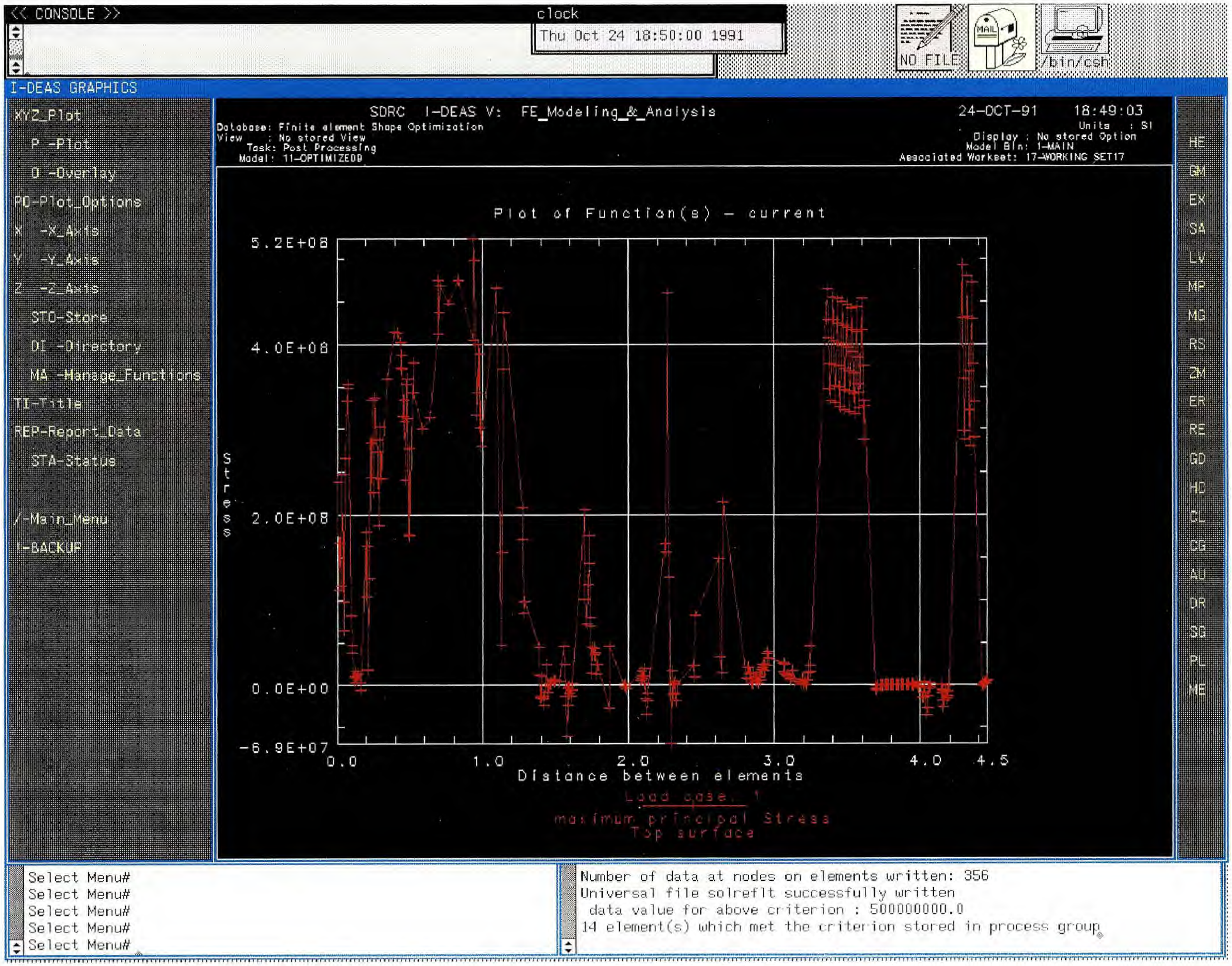
FIG. (8.14)

84

FIG. (8.15)



<< CONSOLE >>
clock
Thu Oct 24 18:55:00 1991
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
XYZ_Plot
P -Plot
O -Overlay
PO-Plot_Options
X -X_Axis
Y -Y_Axis
Z -Z_Axis
STO-Store
DI -Directory
MA -Manage_Functions
TI-Title
REP-Report_Data
STA-Status
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
24-OCT-91 18:54:30
Database: Finite element Shape Optimization
View : No stored View
Task: Post Processing
Model: 11-OPTIMIZED9
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 17-WORKING SET17
Plot of Function(s) - current
5.6E+08
4.0E+08
2.0E+08
0.0E+00
-1.1E+08
Stress
1
100
200
300
400
441
Node number
Load case: 1
maximum principal Stress
Top surface
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Select Menu#
storing current PERMANENT GROUP as OLD1
total nodes grouped : 417
X axis type: Node number
data component : MAXIMUM PRINCIPAL
data component : MAXIMUM PRINCIPAL

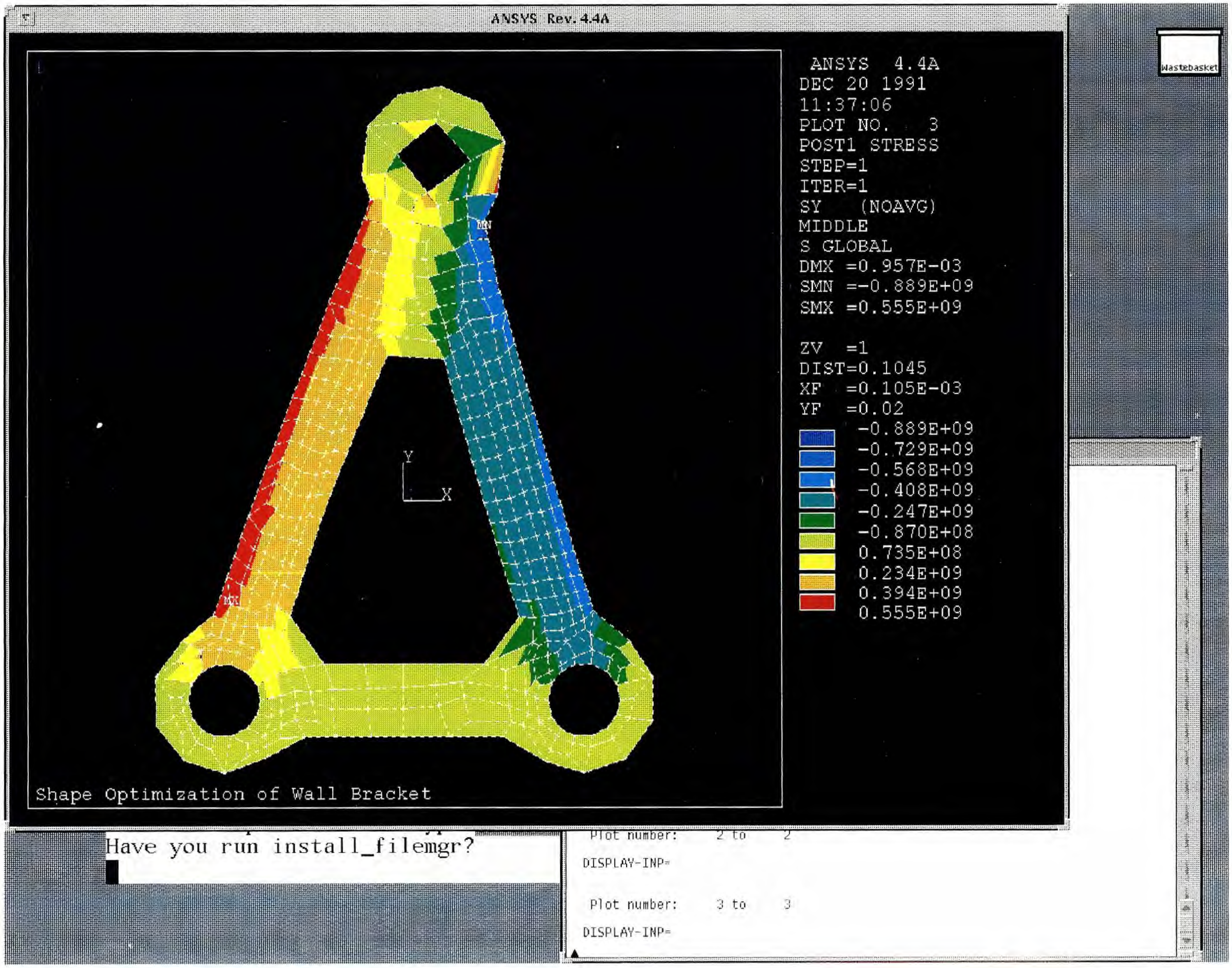


<< CONSOLE >>
clock
Thu Oct 24 18:50:00 1991
NO FILE
MAIL
/bin/csh
I-DEAS GRAPHICS
XYZ_Plot
P -Plot
O -Overlay
PO-Plot_Options
X -X_Axis
Y -Y_Axis
Z -Z_Axis
STO-Store
DI -Directory
MA -Manage_Functions
TI-Title
REP-Report_Data
STA-Status
/-Main_Menu
!-BACKUP
SDRC I-DEAS V: FE_Modeling_&_Analysis
24-OCT-91 18:49:03
Database: Finite element Shape Optimization
View : No stored View
Task: Post Processing
Model: 11-OPTIMIZED9
Units : SI
Display : No stored Option
Model Bin: 1-MAIN
Associated Workset: 17-WORKING SET17
Plot of Function(s) - current
5.2E+08
4.0E+08
2.0E+08
0.0E+00
-6.9E+07
Stress
0.0
1.0
2.0
3.0
4.0
4.5
Distance between elements
Load case 1
maximum principal Stress
Top surface
HE
GM
EX
SA
LV
MP
MG
RS
ZM
ER
RE
GD
HC
CL
CG
AU
DR
SG
PL
ME
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Select Menu#
Number of data at nodes on elements written: 356
Universal file solreflt successfully written
data value for above criterion : 500000000.0
14 element(s) which met the criterion stored in process group

FIG. (8.16)

98

FIG. (8.17)



ANSYS Rev. 4.4A
ANSYS 4.4A
DEC 20 1991
11:37:06
PLOT NO. 3
POST1 STRESS
STEP=1
ITER=1
SY (NOAVG)
MIDDLE
S GLOBAL
DMX =0.957E-03
SMN =-0.889E+09
SMX =0.555E+09
ZV =1
DIST=0.1045
XF =0.105E-03
YF =0.02
-0.889E+09
-0.729E+09
-0.568E+09
-0.408E+09
-0.247E+09
-0.870E+08
0.735E+08
0.234E+09
0.394E+09
0.555E+09
Y
X
Shape Optimization of Wall Bracket
Wastebasket
Have you run install_filemgr?
Plot number: 2 to 2
DISPLAY-INP=
Plot number: 3 to 3
DISPLAY-INP=

CHAPTER 8

RESULTS AND CONCLUSIONS

(8.1) Results:

Fig. (8.1) shows that the wall bracket before the optimization has a weight of 310 grams. Fig. (8.2) shows the shape optimized wall bracket, which has a weight of 188 grams, so the net weight reduction due to the optimization process is 39.35% which are good savings.

Fig. (8.3) shows the stress distribution over the initial wall bracket, from where we identify the areas with lower stresses, which are at the sides, bottom and inside hole. Fig.(8.4) shows the shape and stress distribution after two steps, the weight at this stage is 268 grams, max. stress is 3.94E08 N/m2. Fig.(8.5) shows the shape and stress distribution after three steps, the weight at this stage is 242 grams, max. stress is 5.61E08. Fig. (8.6) shows the finite element mesh at this stage, which shows a lot of distorted elements, so the next few steps only try to minimize the distortion of the elements. Figures (8.7) and (8.8) show the stress distribution over the object at these steps. It can be noticed that due to minor change in shape at the upper neck, the weight has been reduced slightly to 240 grams and the more accurate solution gives the maximum stress at 5.59E08. Fig. (8.9) shows a further decrease in weight, to 233 grams by taking out material at the inner section which is virtually stress free and also the stress has reduced to 5.55E08, which is just because of change in element shape. Practically, the weight loss at this step doesn't affect the max. stress at all. Fig. (8.10) shows the results of the next shape optimization step. The weight has been reduced to 200 grams. An interesting observation made here is that the maximum stress has rather reduced to 5.52E08, which is due to change in element shape. Practically, the weight loss at this step doesn't affect the max. stress at all. Fig. (8.11) shows the final step output of the shape optimization. The weight has been reduced to 188 grams and the maximum stress is shown as 5.52E08. A closer look at the solution shows that there is a great possibility of inaccurate solution because of high elemental distortion, which was proved right as the next optimization step tries to minimize the elemental distortion for the same shape, and the results are as expected. Fig. (8.12) shows that maximum stress is increased to 5.63E08N/m2,which is still within the allowable range. At this stage, the side elements have high aspect ratio, so more elements are added at the side and the mesh is further refined to get a final max. stress at 5.58E08 N/m2. Fig. (8.13) shows the final shape and the stress distribution over the final shape. Fig. (8.14) shows the shaded image of the final shape. Fig.'s (8.15) and (8.16) show the plots for stress distribution over the nodes and the elements respectively.

INITIAL WEIGHT = 310 grams

FINAL WEIGHT = 188 grams

%age Weight Reduction = 39.35%

(8.2) Evaluation of I-DEAS package for Shape Optimization

Fig. (8.17) shows the output of the same model using ANSYS. The results obtained from Ansys are similar to those obtained from I-DEAS. ANSYS does not work well when there is elemental distortion, which is unavoidable in Shape Optimization. Otherwise, the results from ANSYS are more accurate. Thus ANSYS is more accurate while I-DEAS is more flexible as well as user friendly. This thesis tries to take the advantage of both.

As is clear from the excellent results obtained for shape optimization of wall bracket, I-DEAS is a very good package for shape optimization problem. Still there are certain areas which can be improved to make it a better package. The shape optimization process, as done by I-DEAS is semi-automatic. Only the mesh generation is automatic, if desired. The identification of lower stress areas as well as boundary change is manual and may not be efficient. It is suggested that it should have adaptive shape change capability. It is a current research problem but there is still one area, which could be improved. The suggestion is to have an algorithm for adaptive mesh refinement during the shape optimization process to avoid high elemental distortion. It is possible as I-DEAS does have adaptive mesh refinement capability. The only thing to be done is to combine it with shape optimization process. It will make the shape optimization process more convenient and faster and thus increase its efficiency.

(8.3) Conclusion :

The thesis was started with two objectives :

1. To study the shape optimization process.
2. To use I-DEAS package for shape optimization of wall bracket.

The process of shape optimization was studied in detail. The methods of representing the boundary shape during shape optimization were studied. The recently developed techniques for automatic mesh generation were also studied.

In the second part of the thesis, shape optimization of wall bracket was done using I-DEAS. As discussed before, the weight of the wall bracket was considerably reduced. Thus both the objectives were satisfactorily achieved.

(8.4) Scope for Future :

Shape optimization as of today is limited mostly to 2-dimensional applications. It is applied to three dimensions only in very simple cases. There is a tremendous scope of shape optimization as applied to complex three dimensional shapes. Artifitial limbs, Aerospace, Automobile industry etc. are only a few of the vast applications it can have with three dimensional capability. A lot of research is going on in this field and even more research is needed to achieve the desired goal. I personally feel that shape optimization has a tremendous scope for the future.

CHAPTER
9

BIBLIOGRAPHY

Bibliography

1. Bennett, J .A. and Botkin, M.E., " *Structural Shape Optimization with geometric description and adaptive mesh refinement*," AIAA Journal, Vol 23, March 1985, pp 458-465.

2. Bennett, J .A. and Botkin, M.E., " *Automated design for automotive structures*," AIAA /ASME /ASCE /AHS 24th Structures, Structural Dynamics and Materials Conference, AIAA -83 -0941, Lake Tahoe, NV, 1983.

3. Bhavikatti, S.S. and Ramakrishnan, C.V.," *Optimum Shape Design of rotating discs*," Computer Sruct., Vol. 11, 1980, pp. 397-401.

4. Botkin, M.E., " *Shape optimization of plate and shell structures*," AIAA J20, 1982, pp. 268-73.

5. Braibant, V. and Fleury, C.,"*An approximation - concepts approach to shape optimal design*,' Computer Methods in Applied Mechanics and Engineering, Vol. 53, 1985, pp. 119-148.

6. Braibant, V. and Fleury, C.," *Shape optimal design using B-Splines,*" Computer Methods in Applied Mechanics and Engineering, Vol. 44, 1984, pp. 247-267.

7. Braibant, V. and Fleury, C.," *Shape optimal design : a performing CAD oriented formulation,*" AIAA /ASME /ASCE /AHS 25th Structures, Sructural Dynamics & Materials Conference, May 14-16, 1984, Palm Springs, California.

8. Braibant, V. ,Fleury, C. and Beckers, P.," *Shape optimal design : an approach matching CAD and optimization concepts,*" Report, SA - 109, Aerospace Laboratory of the University of Liege, Belgium (1983).

9. Burnett, D.S.," *Finite Element Analysis, from concepts to applications,*" Addison-Wesley Publishing Company, 1987.

10. Chung, K.Y., Torigaki, T. and Taylor, J.E.," *Shape optimization of elastic structures to meet local creteria and constraints,*" 22nd annual meeting of the Society of Engineering Science, State College, P.A., 1985.

11. Diaz, A.R., Kikuchi, N. and Taylor, J.E.," *A method of grid optimization for finite element methods,*" Computer Methods in Applied Mechanics and Engineering, Vol. 41,1983, pp. 29-45.

12. Diaz, A.R., Kikuchi, N., Papalambros, P. and Taylor, J.E.," *Design of an optimal grid for finite element methods,*" Journal of Structural Mech., Vol. 2, 1983, pp. 215-230.

13. Ding, Y., " *Shape optimization of Structures, a literature survey,*" Computers & Structures,Vol. 24, No. 6, 1986, pp. 985-1004.

14. Esping, B.J.D.," *Minimum weight design of membrane structures using eight node isoparametric elements and numerical derivatives,*" Computer Sruct., Vol. 19, 1984, pp. 591-604.

15. Felippa, C.A.," *Optimization of finite elements grids by direct energy search*," Applied Math Modeling I, 1976, pp. 93-96.

16. Felippa, C.A.," *Numerical examples in finite element grid optimization by direct energy search*," Applied Math Modeling I, 1977, pp. 239-44.

17. Finite Element Handbook, Macgraw- Hill, New York, 1987.

18. Haftka, R.T. and Grandhi, R.V.," *Structural shape optimization, a survey*," Computer Methods in Applied Mechanics and Engineering, Vol. 57, 1986, pp. 91-106.

19. Imam, M.H.," *Minimum weight design of 3-D solid components*," Proceedings of the Second International Computer Engineering Conference, ASME, Vol. 3, San Diego, California, August 1982, p p. 119-126.

20. Imam, M.H.," *Three dimensional shape optimization*," International Journal for Numerical Methods in Engineering, Vol. 18, No. 5, 1982, pp. 661-673.

21. Imam, M.H.," *Three dimensional shape optimization*," General Motors Research Laboratories, Research Publications, GMR 3675, 1981.

22. Kikuchi, N., Chung, K.Y., Torigaki, T . and T aylor, J.E.," *Adaptive finite element methods for shape optimization of linearly elastic stuctures*," Computer Methods in Applied Mechanics and Engineering, Vol. 57, 1986, pp. 67-89.

23. Kikuchi, N., Chung, K.Y., Torigaki, T . and T aylor, J.E.," *Adaptive finite element methods for shape optimization of linearly elastic stuctures*," General Motors Conference : The Optimal Shape, Warren, MI, 1985.

24. Kikuchi, N. and T aylor, J.E.," *Optimal modification of shape for two dimensional elastic bodies*," Journal of Struct. Mech., Vol. 11, 1983, pp. 111-135.

25. Kirsch, U.," *Optimum Structural Design,*" Mcgraw Hill, New York, 1981.

26. Lee, M.S., Kikuchi, N. and Scott, R.A.," *Shape optimization in laminated compositeplates,*" Computer Methods in Applied Mechanics and Engineering, Vol. 72, 1989,pp. 29-55.

27. Luchi, L.M., Pogglialin, A. and Persiani, F.," *An interactive optimization procedure applied to the design of gas turbine discs,*" Computer and Structures, Vol. 11, 1980, pp. 629-37.

28. Luss, Hanan," *Optimization : Methodology, Algorithms and applications,*" AT&T Technical Journal, Vol. 68, May/June 19 89, pp.3-6.

29. Morris, A.J.," *Foundations of Structural Optimization : A Unified Approach,*" Proceedings of Nato ASI Meeting, Iowa City, 1980, Noordhoff, The Netherlands, 1981.

30. Oda, J.and Yamazaki, K.," *Pattern transformation method for shape optimization in its application to spoked rotary disks,*" International symposium on optimal structural design, Tuscan, Arizona, 1981, pp. 429-435.

31. Olhoff, N. and Taylor, J.E.," *On Structural Optimization,*" Journal of applied Mechanics, Vol.50, December 1983, pp. 1139 -1151.

32. Prager, W. and Taylor, J.E.," *Problems of Optimal Structural Design," ASME Journal of Applied Mechanics*, Vol. 35, 1968, pp. 102-106.

33. Quea, J.P. and Trompette, Ph.," *Two dimensional shape optimal design by the finite element method,*" nternational Journal for Numerical Methods in Engineering, Vol. 15, 1980, pp. 1603-1612.

34. Rao, S.S.," *The finite element method in engineering,*" Pergamon Press, 1982.

35. Rasmussen, John," *Shape optimization and CAD*,"

36. Rasmussen, John," *The structural optimization system CAOS*," Structural Optimization, Vol. 2, 1990, pp. 109-115.

37. Reddy, J.N.," *An Introduction to finite element method*," Macgraw- Hill, New York, 1984.

38. Schmit, L.A.," *Structural Synthesis - Its genesis and developement*," AIAA Journal, Vol. 19, 1981, pp. 1249-1263.

39. Vanderplaats, G.N.," *Structural Optimization - Past, Present and Future*," AIAA Journal, Vol. 20, 1982. pp. 992-1000.

40. Venkayya, V.B.," *Structural Optimization : A review and some recommendations*," International Journal of Numerical Methods in Engineering, Vol. 13, 1978, pp. 205-228.

41. Yang, R.J. and Choi, K.K.," *Accuracy of finite element based shape design sensitivity analysis*," Journal of Struct. Mech., Vol. 13, 1985, pp. 223-239.

42. Zienkiewicz, O.C. and Campbell, J.S.," *Optimum Stuctural Design*," (Edited by Gallagher, R.H. and Zienkiewicz), John Willey, New York, 1973.