870872

The Computer Aided Design Analysis and Structural Optimization of Engine Accessory Components

R. I. DeVries
Structural Analysis Dept.
Advanced Vehicle Engineering Technology
Ford Motor Co.
Dearborn, MI

H. V. Radziwon
Powertrain NVH & Design Analysis Dept.
Advanced Powertrain Engineering
Ford Motor Co.
Dearborn, MI

P. Aghssa
Light Truck Design Analysis Dept.
Truck Operations
Ford Motor Co.
Dearborn, MI

ABSTRACT

This paper describes the integration of computer graphics, finite element analysis including design sensitivity analysis, and structural optimization technology to automate the design analysis and design optimization of engine accessory components. By use of MSC/NASTRAN's design sensitivity analysis and structural optimization methodology, the design of vehicle components is facilitated to achieve both design improvement and a significant increase in engineering efficiency.

The application of this technology to two typical engine accessory brackets for minimum weight design subject to constraints on stresses due to applied loads as well as natural frequencies is described.

THE POTENTIAL OF computer based methods of design analysis has been widely demonstrated. Finite element analysis to provide design assurance can reduce hardware iterations in the early design of structural components or systems. Computer graphics software facilitates both the preprocessing of data for finite element analysis (eg. automated mesh generation) and the post processing of the structural response as computed by the finite element method. More recently, design sensitivity analysis, which computes the sensitivity of this structural response to changes in the design of the structure, has become an available feature of several finite element programs , including MSC/NASTRAN (1). Optimization methodology has been developed to interface analysis, design sensitivity analysis, and mathematical programming to permit automated structural optimization. This paper describes the integration and coordinated use of this methodology for the efficient design analysis and structural optimization of engine accessory components.

Figure 1 is a flow chart illustrating the integration of these computer aided engineering tools. A finite element mesh is generated using the automated mesh generation available in programs such as PATRAN (2) or PRIME/PDGS (3) and modelling the component so that each sub-component has a unique property specification. A finite element analysis is performed on the model for suitable loads and constraints using MSC/NASTRAN. Salient design constraints are determined using computer graphics software to identify relevant response variables. Design sensitivity analysis then computes the derivatives of these response variables with respect to changes in the sizing of the sub-components comprising the structure. Linear programming on the linearization of the structural response computed by the design sensitivity analysis then provides an improved or approximate optimal design. The process then iterates to converge to an optimal set of design parameters.

Each of these software tools is reviewed with particular attention paid to its interface with the other tools integrated into this process. The application of this process to two examples involving engine accessory brackets is described illustrating this method in greater detail.

MESH GENERATION

The initial task in the process of finite element analysis, sensitivity analysis, and optimization is the generation of the mesh required for finite element analysis. This mesh includes grid point coordinates, connectivity of elements, and property specification. Required mesh characteristics, including its distribution and refinement, is dependent on not only the geometry of the component but also the type of analysis, specific elements used, applied loads, constraints and potential design changes.

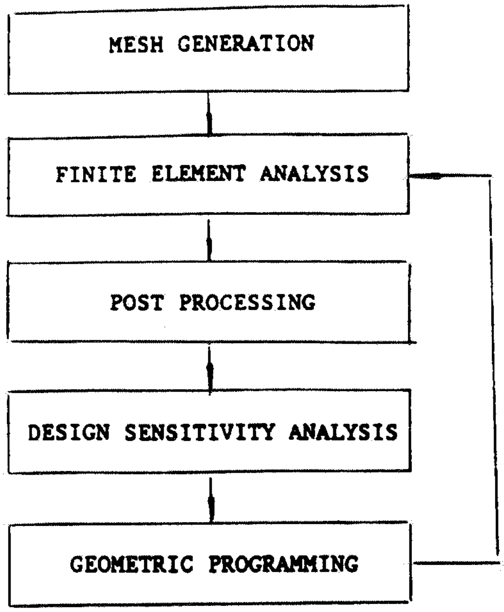

Figure 1. Design Analysis - Optimization Overview

```
NASTRAN FILES=(DB01), BUFFSIZE=4609
ID DSA,EXAMPLE
SOL 53
TIME 10
DBMGR //0//5000////DB01 $
DBMGR //$
CEND
SENSITY(PRINT)=ALL
TITLE=SAMPLE DSA INPUT WITH DATA REPLICATION
BEGIN BULK
$
$ STRUCTURAL RESPONSE VARIABLE ( 1ST MODE FREQUENCY )
$
DSCONS          1MODE1    FREQ            1        3
$
$ DESIGN VARIABLES AND LINKAGE TO PROPERTY CARDS
$ ( THICKNESSES OF ELEMENTS WITH PROPERTY IDS =11 THRU =29 )
$
DVAR            11RIB11          10       11
=               *(1)   =         =        *(1)
=(17)
DVSET           11PSHELL         4        1.       1.       11
=               *(1)     =       =        =        =        *(1)
=(17)
ENDDATA
```

Figure 2. MSC/NASTRAN Design Sensitivity Analysis Input Using Data Replication

Generation of this required mesh is ilitated by use of preprocessing software such PRIME/PDGS or PATRAN that interactively produce a finite element model. PRIME/PDGS is a menu driven computer graphics program that operates on a combination of Prime computer and Lundy graphics hardware. A primary advantage to the use of PRIME/PDGS is that a finite element mesh can be generated using the geometry data already created in a computer aided drafting process. PATRAN is a command driven pre and post processor that is particularly useful for generic models of a component. Its use of color graphics is particularly useful for the modelling of complicated structures and for post processing of a finite element analysis.

Specific details of finite element mesh generation are not discussed in this paper; however, the special reuirements with respect to the finite element model's use in a structural analysis - optimization scheme are identified. If sub-structuring, or a so called " superelement processing ", will be required in the analysis of the structure, all parts of the structure that will vary in the redesign process as well as all grid points and elements at which structural response is relevant must be modelled as part of the residual structure. This is required because the MSC/NASTRAN finite element program currently places this restriction on its design sensitivity capability. Each portion of the structure that will potentially be redesigned independently uires its own individual property ification (eg. MSC/NASTRAN property card ıدntification numbers). Use of sequential property identification numbers will allow

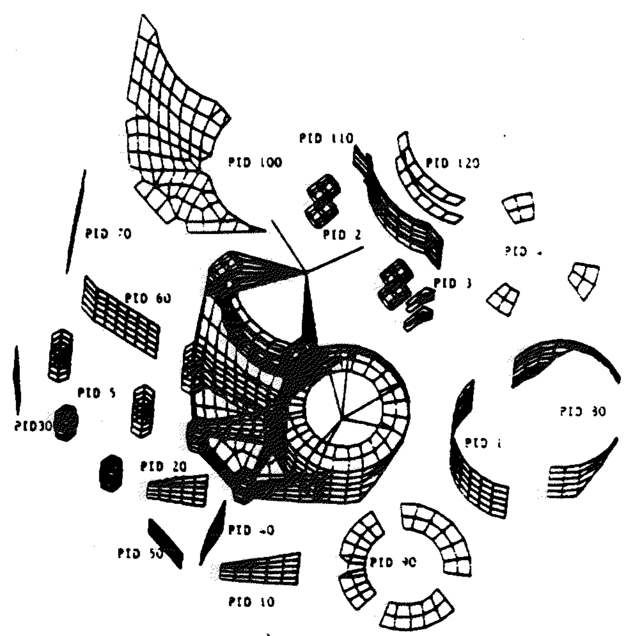

Figure 3. Specification of Unique Property ID's for Each Design Region of Component

simplification of the input to a subsequent MSC/NASTRAN design sensitivity analysis as shown in Figure 2.; however, this requirement is optional. In the redesign process it may be possible to delete portions of the design; therefore, the inital finite element should be modelled to facilitate this possibility. Graphics illustrating each design region of a component facilitate the identification of manufacturing constraints for the design optimization of the component.

Figure 3. illustrates an exploded view of a finite element mesh of a typical engine accessory bracket. this mesh was generated using PRIME/PDGS. Each design region has its own property identification, the entire component is modelled in the residual of the finite element model, and the model is clearly displayed for future reference in the optimization. Any part of the structure which the design optimization process determines should be deleted is easily removed from the mesh.

FINITE ELEMENT ANALYSIS

The design analysis via the finite element method of an engine component or accessory requires the analysis of the component under a variety of loads and constraints approximately simulating service or test conditions. For engine accessories these analyses include determination of natural frequencies, forced response, stress resultants under expected use, and displacement of the component relative to the other engine components. Appropriate loads and constraints are added to the mesh and initial property specification created by the above described mesh generation. Salient analyses for the examples described in this paper include both stresses under static loadings and the natural frequency derived from a normal modes analysis.

The MSC/NASTRAN finite element program is used for these analyses in that it is both the principal finite element program used at Ford and has a design sensitivity analysis capability. The only special requirement for analysis of the component salient to its inclusion in a design sensitivity and optimization process is that the analysis is performed using super-element processing (ie. substructuring), even if the entire component is included in the residual structure. This is required, as explained in a subsequent section, because the sensitivity analysis is restarted from a baseline structural analysis and requires a database containing the baseline solution as well as the assembled and factored mass and stiffness matrices. In order to minimize the size of this database, only relevant structural responses need to be stored for the subsequent design sensitivity analysis (eg. modal displacements and eigenvalues must be stored to later produce sensitivity coefficients related to natural frequencies). Details as to which quantities must be stored by explicit requests in the NASTRAN case control are given in references 1 and 4. Only analyses involving static analysis, normal modes analysis, buckling, or inertial relief are currently supported with a design sensitivity analysis capability.

PROCESSING

Prior to either performing a design sensitivity analysis or formulating a structural optimization problem, salient design constraints must be identified. To facilitate this task, appropriate computer graphics software can be utilized. Figure 4, generated by the PATRAN program and displayed on a TEKTRONIX color graphics terminal, is a black and white copy of a color graphics plot of computed stresses in an engine accessory bracket identifying high stress areas under loading of the bracket. Design objectives and sensitivity analysis can thereby be limited to stress response in only the high stress areas. Displays of mode shapes, see Figure 5, corresponding to the natural frequencies of a structure as computed by a normal modes analysis facilitate selection of relevant modes for inclusion in sensitivity analysis and optimization.

Limiting design sensitivity analysis and optimization to only those design objectives that are approaching their critical values, reduces the cost of the sensitivity analysis and simplifies the resulting optimization problem for easier solution. After optimization the entire structural response is recomputed and all design constraints validated, including those deleted from the optimization process.

DESIGN SENSITIVITY ANALYSIS

Finite element analysis analytically computes the displacements, stresses, natural frequencies, and other structural response parameters of a structure under specified loads and constraints. Design sensitivity analysis then computes the derivatives, or sensitivity coefficients, of these structural response variables with respect to changes in the parameters defining the structural design, providing a linear model relating the structural response variables to these design variables (Equation 1).

$$z_i - z_i^o = \sum_{1}^{J} c_{ij}(b_j - b_j^o) \qquad (1)$$

where

z_i^o = Baseline value of response variable

z_i = Value of response variable

b_j^o = Baseline value of design variable j

b_j = Value of j th design variable

c_{ij} = Design sensitivity coefficients

Although in general the functional relationship between the design variables and structural response is non-linear, this linear model yields reasonable predictions of structural response for modest changes in design variables or for large changes in those cases where the relationship between the design variables and the salient structural response is monotone and quasi-linear. In either of these situations this linear model can be directly used to identify necessary changes in those parameters to achieve desired

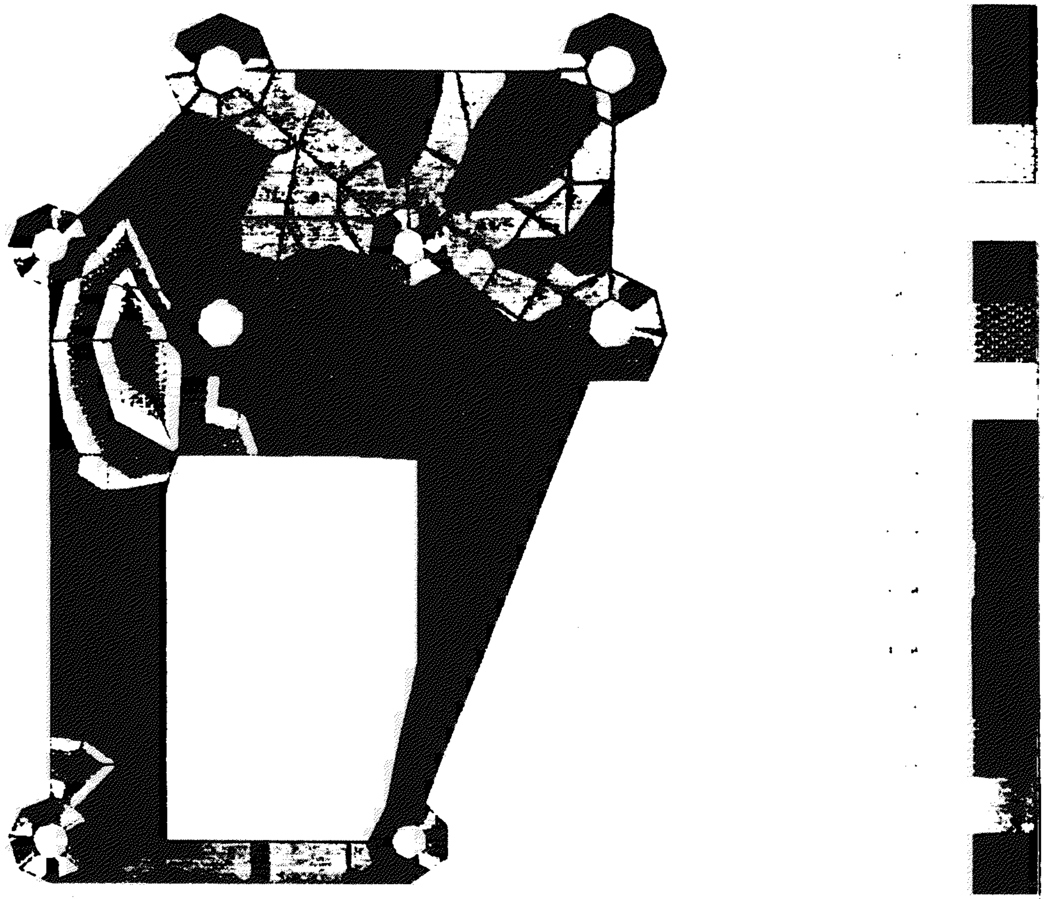

Figure 4. Color Graphics Display
Example 2. Static Stresses

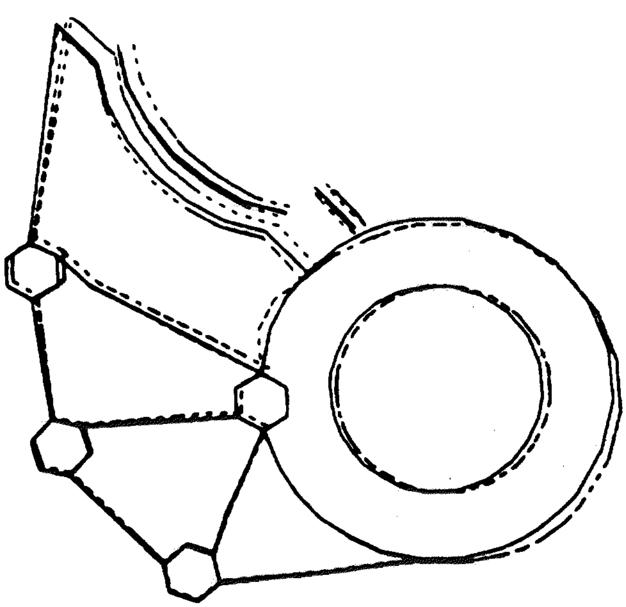

Figure 5. Mode Shape Display
Example 1. First Mode

structural response. The design sensitivity coefficients can also be used in a linear programming scheme or other optimization algorithm to achieve these design objectives.

Design sensitivity analysis is a capability of the MSC/NASTRAN finite element program and provides sensitivity coefficients for structural response parameters derived from static (SOL61), normal modes (SOL63), and buckling (SOL65) analyses with respect to changes in either material properties or sizing parameters (eg. thicknesses, cross sectional area). The formulation of this capability is derived in equations 2 through 5 by implicitly differentiating the finite element equilibrium equations and solving for the sensitivity coefficients in terms of both the baseline solution and the derivatives of the fundamental finite element matrices.

For static analysis,

$$K_b u_b = P_b \qquad (2)$$

where

K_b - stiffness matrix

u_b - displacement vector

P_b - load vector

so that

$$\frac{\partial u}{\partial b} = - K^{-1} \left(\frac{\partial K}{\partial b} u - \frac{\partial P}{\partial b} \right) \qquad (3)$$

For normal modes:

$$K_{gg} \phi^i_g = \lambda_i M_{gg} \phi^i_g \quad (4)$$

where

λ_i - i th eigenvalue

ϕ^i_g - mode shape associate with λ_i

so that

$$\frac{\partial \lambda}{\partial b} = \frac{1}{m_i} \left(\phi^T \frac{\partial K}{\partial b} \phi - \lambda_i \phi^T \frac{\partial M}{\partial b} \phi \right) \qquad (5)$$

where

$$m_i = \phi_i^T M \phi_i$$

A similar expression results from the equations for buckling analysis, which is symbolically identical to equation (4).

Since the sensitivity coefficients are expressed only in terms of the derivatives of the fundamental finite element matrices and the solution of the baseline design, they can be economically computed by using this previously computed data and factored stiffness matrix. Experience with typical problems, including those examined in this paper, demonstrates that the sensitivity analysis computational costs are an order of magnitude less than the baseline analysis. Further, the single sensitivity analysis replaces a large number of reanalyses of the entire structure previously required to determine the effect of design parameter changes.

An MSC/NASTRAN design sensitivity analysis is restarted from a super element analysis (substructuring) of a baseline design of the structure using the database containing both the solutions of the finite element analysis and its factored stiffness matrix. Salient structural response variables that have been identified by post processing this baseline analysis are flagged in the design sensitivity input. Specification of design variables in the sensitivity analysis is facilitated if, during mesh generation and generation of the finite element model, the numbering of property cards are unique to each design parameter and are numbered as sequentially as practical. If numbered uniquely and sequentially, the input can be generated either through a simple computer program or via NASTRAN data replication features (Figure 2). Subsequent engineering effort and computational cost to produce a design sensitivity analysis from the database created in the baseline analysis is minimal.

OPTIMIZATION

Finite element analysis computes the overall structural response of a particular design of a component. Software and hardware to post process this data facilitates identification of salient design constraints and objectives. Design sensitivity analysis can provide a linear model relating these constraints to design parameters via equation (1). Structural optimization methodology then employs this data to determine an improved or optimized design relative to these design objectives. Structural optimization methodology solves the following generic problem:

Find $b=(b_1 ,\ldots, b_n)$ - design variables (eg. structural sizing parameters)

To minimize the objective function

$$F(b,z) \longrightarrow \min \text{ (eg. weight)} \qquad (6)$$

Subject to the constraints (eg. max stress, minimum natural frequencies):

$$G_i(b,z) \leq 0 \quad i = 1,\ldots,m \qquad (6.a)$$

And side constraints:

$$b_i^{min} \leq b_i \leq b_i^{max} \quad i=1,\ldots,n$$

Where

$$z=(z_1 ,\ldots, z_m) \text{ - structural response}$$

General purpose software (5,6) has been developed to solve this generic problem linking MSC/NASTRAN with a variety of optimization algorithms. Most of these optimization methods require design sensitivity coefficients to prescribe an improved design. Large scale optimization problems and problems having highly nonlinear constraints can be successfully optimized using such software.

Many structural optimization problems, however, such as the engine accessory components examined in this paper, exhibit constraints and design objectives that are both monotone and closely approximated by the linear model given in Equation 1.. For such problems, sequential linear programming (7) efficiently solves the structural optimization problem. Since linear programming is readily available, either as computer software or hand calculations, it represents a feasible approach for the successful solution of such design optimization problems.

Sequential linear programming is illustrated in Figure 6. The objective function and design constraints are approximated by their linearization about the baseline design.

Critical constraint surfaces ($G_i = 0$) are approximated and an optimum determined by solving this linear sub-problem. This linear sub-problem necessarily has its optimum at a vertex of the simplex of the constraint surfaces defining the domain of feasible designs. The process is repeated with an analysis about this approximate optimum, another sensitivity analysis, and linearization. In the problems examined in this paper, a maximum of two iterations were required for this process to converge to the exact optimum.

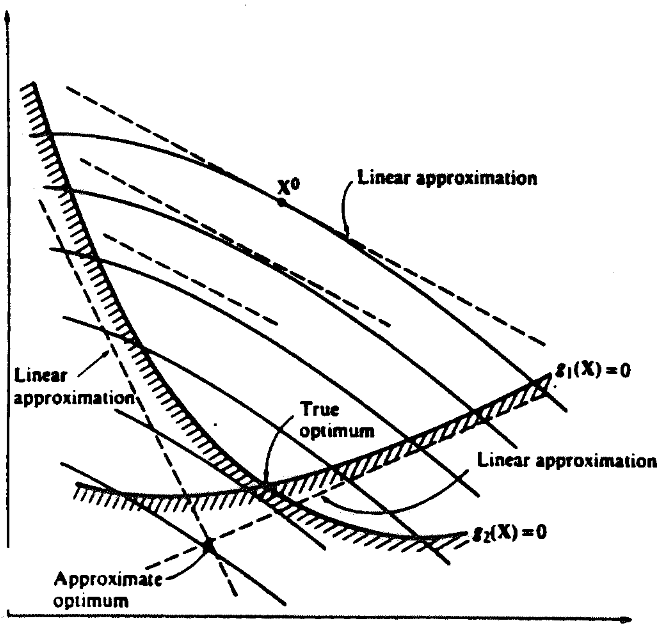

Figure 6. Linear Sub-Problem (SLP)

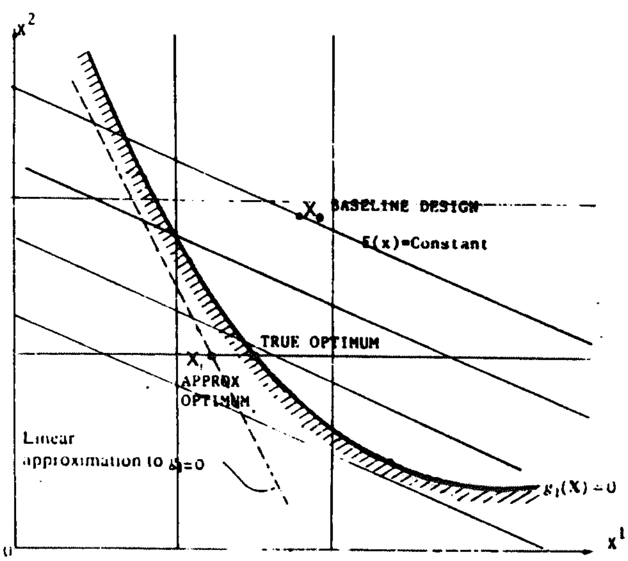

Figure 7. SLP - Linear Objective & Single Constraint

The problem is further simplified if both the objective function (eg. weight) is linear in the design parameters (eg. Thicknesses of components of the structure) and only a single constraint is critical. This simplification is illustrated in Figure 7. In this case the optimum necessarily exists at one of the vertices corresponding to the side constraints, which are generally determined by manufacturing limits. This point can be determined by calculating the coefficients:

$$d_{ij} = (\partial G_i / \partial b_j)/(\partial F/\partial b_j) \quad (7)$$

If the objective, or cost, function F is the weight of the structure, these coefficients represent the sensitivity of the i-th constraint to changes in distribution of material in the area effected by the j-th design variable. The optimum is determined by upgaging those design variables yielding the most design benefit per unit cost and downgaging those giving the least or negative benefit per unit cost. Rank ordering these coefficients identifies the appropriate changes.

For problems involving more than one critical constraint, this problem can be solved using the simplex method through manipulation of tables of design variables, constraints, and slack variables. Large scale problems, however, are best solved by using automated optimization systems such as OPUS (5,6) and employing its sequential linear programming capability. The trade off of using an automated approach vis-a-vis hand computations includes not only the data preparation required for the former versus the hand calculations required by the latter but also some loss of engineering insight in the automated approach (eg. Identification of additional manufacturing constraints and design opportunities). Two applications of sequential linear programming to solve the optimization of engine accessory components are described the examples.

EXAMPLES

EXAMPLE 1: Air Conditioning / Power Steering Accessory Bracket Optimization- A typical design of an engine accessory bracket, holding an air conditioning compressor and power steering pump, was optimized with the technology described in this paper for minimum weight subject to stress and natural frequency related constraints . A finite element mesh illustrating this component is shown in Figure 8. This bracket was designed to be manufactured from die cast aluminum; therefore, the design variables included the thicknesses of all ribs in the design as well as the thicknesses of the continuous regions. Design objectives considered in the finite element analysis and optimization of this component included component weight, a first mode frequency 20 hertz above the first harmonic frequency of the engine to which the accessory bracket was attached, computed stresses below both the fatigue and yield stress of the bracket

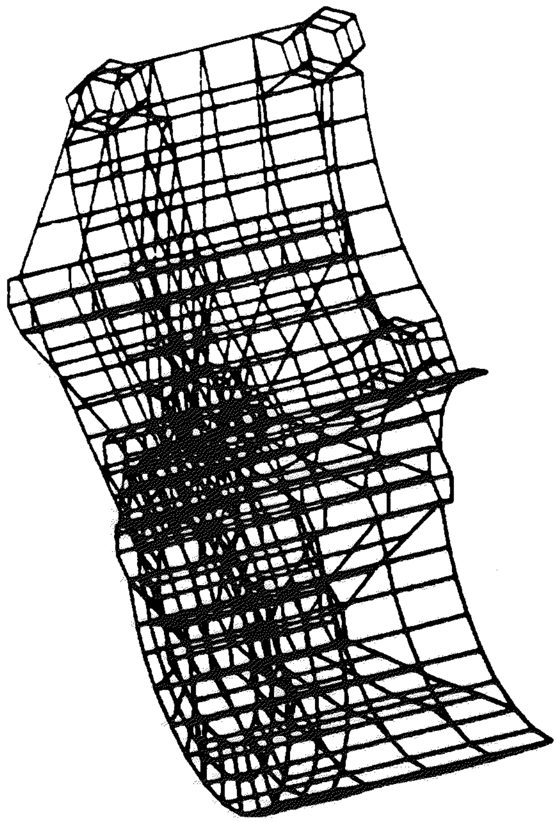

Figure 8. FE Mesh Power Steering - Air Conditioning Bracket

material, thickness and draw limits imposed by the material and die cast process by which the bracket would be produced, and packaging constraints including clearance to other engine components and attachment requirements.

A finite element mesh was generated for the bracket with, as illustrated in Figure 3, each potential design area identified with a unique property identification. Baseline finite element analyses, including both a normal modes analysis and a static analysis, were performed with results stored in a database. The total mass of the bracket, exclusive of the mass of the air conditioning compressor and power steering pump, was 2.6746 x 10^{-3} Mg. The stress distribution through the part was plotted using both color graphics and stress contours to identify high stress areas. Calculated stresses were approximately 12% of critical stress levels. The first mode frequency for this baseline design was 241 hertz, which represented an over design of 21 hertz and potential weight savings. This first mode demonstrated an overall bending of the bracket from the attachment points as illustrated in Figure 4. Many manufacturing constraints were ritical in that most regions were at the maximum icknesses allowed by the die casting process nd the type of aluminum used. Design sensitivity analyses were then performed.

A sensitivity analysis of the calculated stress in the areas identified as high stress regions indicated that no change in the design parameters would lead to stresses that would violate the applicable stress constraints; therefore, the design optimization was limited to the natural frequency constraint, side constraints on the thicknesses of the different areas of the bracket, and an objective function of component weight.

The design sensitivity coefficients for weight and first mode frequency are listed with their quotient, representing projected frequency change per unit mass added in the respective design regions, and summarized in rank order in Table 1.

TABLE 1.

Design Sensitivity Analysis Results
First Mode Frequency DSA
Mass DSA

Design* Variable	$\partial\lambda/\partial B_i$ Freq DSA hz./mm	$\partial M/\partial B_i$ Mass DSA 10^{-5}Mg/mm	$\partial\lambda/\partial M_i$ Quotient 10^{+4}hz./Mg
10	.10384	2.3412	0.4435
50	.10365	.9843	1.0530
120	.11276	.6838	1.6490
60	.81393	3.6543	2.2273
80	1.18830	5.0929	2.3332
100	1.47720	4.7155	3.1326
110	1.60660	4.8539	3.3099
90	.82039	2.3201	3.5360
70	1.39960	2.9368	4.7657
40	.92717	1.5707	5.9029
20	1.10490	1.7843	6.1923
30	1.02850	1.4339	7.1727
3	.77532	.5750	13.4833

* Unchanged design variables omitted

Since the initial design exceeded the minimum first mode frequency constraint, the optimal design could be determined, in the manner described in the previous section, by down sizing the parameters yielding the lowest frequency increase per unit mass add in the order inidcated in Table 1 until this constraint is violated and subsequently up sizing those parameters yielding a large increase in frequency per unit mass added. In this example, each part was down sized in the order indicated in Table 1. The results of the optimization are summarized in Table 2. The frequency predicted by the design sensitivity analysis was 229 hertz, an 11.46 decrease in frequency (4.8 %). The projected weight savings was 32.95 x 10^{-5} Mg (approximately .725 lbs.). A reanalysis indicated the actual frequency from the prescribed design changes was 220. hertz with no significant change in the calculated stresses.

Table 2.
Example 1 Optimization
Results

Design Level	First Mode Frequency (Hertz)	Mass (Mg.)
Baseline Design (FEA Results)	241.	2.6746 10^{-3}
Optimized Design* (Linear Approximation)	229.	2.3451 10^{-3}
Optimized Design* (FEA Results)	220.	2.3451 10^{-3}
Design Goals	> 220.	Minimum

* All sizing at minimum permitted by manufacturing process

Example 2. Air Conditioning Compressor Engine Mounting Bracket- A typical design of an engine accessory bracket to secure an air conditioning compressor was optimized using the · hnology described in this paper. A finite ent mesh of this design is shown in Figure 9. A in the previous example, this component was designed to be manufactured from die cast aluminum and each reinforcing rib and continuous region was allowed to vary in the design process. Manufacturing constraints included maximum and minimum material thicknesses in each region. Each rib and each flange was also allowed to be deleted completely. Design constraints included a 240 hertz minimum frequency for the first mode and a stress constraint for the static loadings.

A mesh was generated with each rib, flange, and continuous region with a unique property identification. Baseline finite element analyses for both static loadings and normal modes were performed. The total mass of this baseline bracket was 1.232 lbs. The stress distribution as computed by the static analysis is displayed in Figure 5 with the highest calculated stress only 12.6% of the maximum allowable stress levels; therefore, stress was eliminated as an active constraint in the optimization process. The first mode frequency of the bracket was 264 hertz which was 24 hertz higher than the design goal of a 240 hertz first mode.

A design sensitivity analysis for first mode frequency and mass was performed (MSC/NASTRAN SOL53). The caluculated design sensitivity coefficients ranked by their quotient are listed in Table 3. Although in general the design sensitivity coefficients are valid only for small ges of the design variables, the linearized l projected, with these coefficients, the e ect of deleting a rib or flange completely.

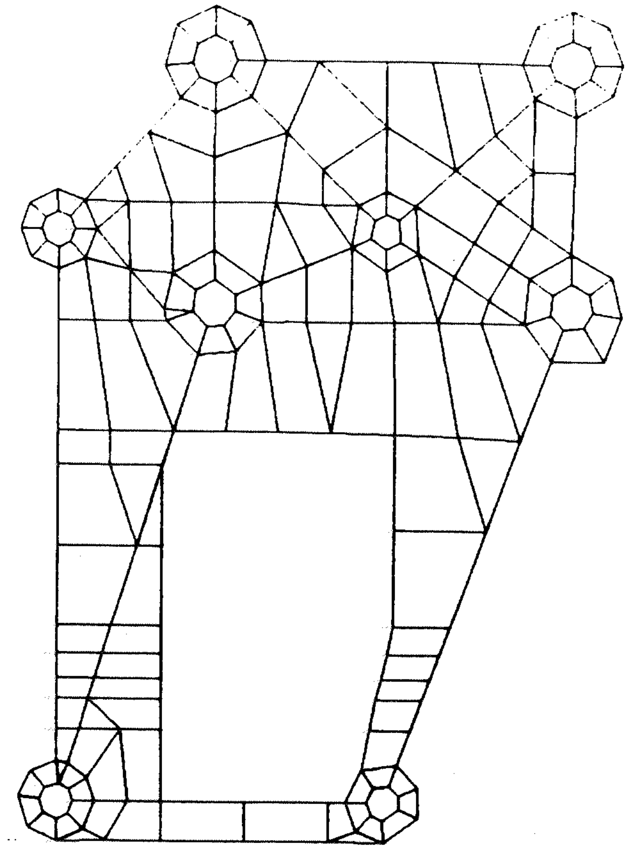

Figure 9. FE Mesh Air Conditioning Engine Bracket

Using the method reviewed in the previous section, the design components were either deleted or upgaged to achieve an optimal design. The design changes are summarized in Table 4. Although a large number of changes were made, including the complete removal of a number of ribs and flanges, the linearized model agreed well with a reanalysis of the optimal design as summarized in Table 5. The weight savings achieved from this process was approximately 40%.

The total computer time required for the baseline analysis of this second engine bracket was 240 cp seconds on Ford's Cray XMP-II computer. The sensitivity analysis cost only 7 cp seconds. Had the sensitivity of changes of all 24 design regions been determined by repeated analyses of the design, the cost would have been 5760 cp seconds plus the manpower required to perform these analyses. The design sensitivity analysis, therefore, extends finite element analysis to an automated design process at extra minimal cost in computer usuage or manpower.

CONCLUSIONS

The integrated use of automated mesh generation, finite element analysis, computer graphics for post processing, design sensitivity analysis, and optimization represents a practical tool for the automated design optimization of engine components. Significant improvements in design quality and component costs are achieved in the application of this technology. Design ensitivity analysis and optimization gnificantly reduce the time and computer xpense required to go from a preliminary design to a fully optimized design.

TABLE 3.

Example 2.
Design Sensitivity Analysis Results
First Mode Frequency DSA
Mass DSA

Design Variable	$\partial\lambda/\partial B_i$ Freq DSA	$\partial M/\partial B_i$ Mass DSA	$\partial\lambda/\partial M_i$ Quotient
	hz./mm	10^{-5}Mg/mm	10^{+4}hz./Mg
11	.01692	.2200	0.7691
12	.00318	.2200	.1445
13	2.59600	.4400	59.0000
14	.03742	.2200	1.7007
15	2.31700	.4400	52.6590
16	2.60550	.4400	52.9159
17	2.39320	.2200	108.7818
18	.81214	.2200	36.9154
19	.59723	.2200	27.1468
20	1.02190	.6700	15.2522
21	1.70290	.2200	77.4045
22	.83110	.2200	37.7772
	.96366	.2200	43.8037
[illegible]	.01384	.1700	.8138
32	.29080	.3300	8.8121
33	1.44760	.3300	43.8666
34	-.01037	.1700	-.6100
35	1.16650	.3300	35.3484
36	.09798	.1700	57.5880
37	.14037	.3300	42.5360
38	.19870	.3300	6.0212
39	.96721	.1700	56.8947
40	5.15330	.6700	76.9149
41	6.9438	1.8300	37.9442

ACKNOWLEDGMENTS

The authors express their appreciation to Messers. V. J. Borowski, G. C. Campbell, J. P. McDonald, and L. I. Nagy of Ford Motor Company for technical advice on both engine design analysis and finite element analysis of engine components. The continuing support and encouragement of Messers. M. P. Anderson, A. A. Butkunas, N. Kazan, R. W. Rankin, P. Suruli-narayanasami, and D. H. Yun of Ford Motor Company is sincerely appreciated.

REFERENCES

. , "MSC/NASTRAN Application Manual", NASTRAN Version 65, The MacNeal-Schwendler Corporation, Los Angeles, CA, 1986.

2. , "PDA/PATRAN Reference Manual", PDA Engineering, Santa Anna, CA, 1986.

3. , "PDGS Finite Element Modeling Guide Release 7.0", Prime Computer Inc., Natick, Mass, 1986.

4. W. J. Anderson et al., "Study Guide Video Lecture Series 1003 Design Sensitivities and Optimization", The MacNeal-Schwendler Corporation, Los Angeles, CA 1986.

5. Adelberg, DeVries, et al., "OPUS - A Programming System Approach to Structural Optimization", Proceedings of the 4th International Conference on Vehicle Structural Mechanics, Nov. 18-20, 1981, PP99 SAE Society of Automotive Engineers, Inc., Warrendale, PA, 1982

6. DeVries, R. I. and Chon C. T., " Structural Optimization Technology at Ford Motor Company", Proc. of the Int. Conf. on Modern Vehicle Design Analysis, London, UK, June 22-24, 1983.

7. G. N. VanderPlaats, " Numerical Optimization Techniques for Engineering Design ", McGraw-Hill, New York, NY 1984

Table 4.
Redesign Summary with Projected
First Mode Frequency and Weight Changes

Design* Variable	Baseline Thickness (mm)	Redesign Thickness (mm)	Freq. Change (hz)	Mass Change (Mg)
34	6.0	0.0	+.0622	-.00001
12	4.5	0.0	-.0479	-.00001
31	6.0	0.0	-.1309	-.00001
11	4.5	0.0	-.0761	-.00001
14	4.5	0.0	-.3753	-.00001
36	6.0	0.0	-.5878	-.00001
20	4.5	0.0	-4.5985	-.00003
19	4.5	0.0	-2.6874	-.00001
35	6.0	0.0	-6.9990	-.00002
18	4.5	0.0	-3.6540	-.00001
22	4.5	0.0	-3.7399	-.00001

* Design variables 37,38,32, and 41 not changed due to manufacturing constraints

Table 5.

Design Level	First Mode Frequency (Hertz)	Mass (lbs.)
Baseline Design (FEA Results)	264.	1.2420
Optimized Design (Linear Approximation)	240.	.8900
Optimized Design (FEA Results)	244.	.8900
Design Goals	> 240.	Minimum