N88-19122 | 511-61
125797
68.

DESIGN MODELING FOR SHAPE OPTIMIZATION

M.E. Botkin
Engineering Mechanics Department
General Motors Research Laboratories
Warren, MI 48090-9057

ABSTRACT

Some important aspects of design modeling for shape optimization will be discussed for both stamped sheet metal components and cast solid components. For stamped components the basis for the modeling approach is a boundary design function. Design parameters control the shape of two-dimensional regions. For more complex, folded plate components, the two-dimensional regions can be assembled using translation and rotation operations. The analysis model is automatically created using a mesh generation procedure requiring only boundary data. For less complex solid components, it was found that this approach is not suitable. For these structures, the finite element models are typically created using very sophisticated graphical modeling systems. A new approach which overlays a parameterized surface design model on an existing analysis model is described. To summarize, the future needs for solid shape design will be described in terms of an extension of the previously described two-dimensional capability.

Design Modeling for Large-Scale Three-Dimensional Shape Optimization Problems

R. J. Yang and M. J. Fiedler
Engineering Mechanics Department
General Motors Research Laboratories

ABSTRACT

Modeling three-dimensional automotive components for shape optimization is described. Shape optimization differs from sizing optimization in the type of structure, type of design variable, and sensitivity analysis employed. The key element of the shape optimization design model is the parameterization of the geometry by which the optimizer controls the structure dimensions. Efficient generation of the design model is very critical in the design process. A quick generation of a good optimization model combined with an efficient optimization system will result in a drastic design time saving. In this paper, three approaches to generating the design model are discussed. Emphasis will be placed upon a special modeling technique which overlays the design model onto an already existing finite element model. This technique is incorporated in a modular three-dimensional shape optimization system which uses NASTRAN for analysis. A realistic automotive steering control arm is used as an example to demonstrate the use of the technique.

INTRODUCTION

Optimization techniques have emerged as useful design tools in recent years. Structural optimization for sizing variables has been treated extensively in the literature. The problem of designing the shape of a structure for minimum mass constitutes another important class of optimization problems. Shape optimization differs from sizing optimization in several ways. First, sizing design variables are generally dimensions which do not affect the geometric configuration of the structure, such as cross-sectional dimensions of beam members (thickness, width, height, moment of inertia, etc.). Shape design variables define the geometry of two-dimensional plate and three-dimensional solid structures. As a result, shape design sensitivity analysis is much more complicated. In shape optimization, the boundary of the structure is variable, so parameterization of the geometry is the most important aspect of the shape design model. Modeling for shape optimization is more difficult because both the analysis and design models must completely describe the structure geometry. The design and analysis models for sizing optimization are inherently loosely coupled because there is little duplication of information. For an existing large analysis model whose surface is not parametrized, generating the design model is not trivial.

In the past, most effort has been put on shape design sensitivity analysis and most problems solved are limited to two-dimensional problems [1-4]. The importance of automatic creation of the design model was seldom found in the literature. Botkin et. al. [5] used computer graphics to generate shape design models for two and three-dimensional stamped structures. Only a limited amount of work has been accomplished in three-dimensional shape optimization using solid finite element analysis [5-7]. Refs. 6 and 7 generated design models manually and as a result, only simple geometries (cantilever beam, engine bearing cap, etc.) were optimized. Ref. 5 used an automatic mesh generator to create the design model and a more complicated engine connecting rod was optimized. However, connecting design variables to the geometry was still done manually. In the real world, three-dimensional problems are often complex and require large finite element analysis models. To be most effective in impacting the design process, the design model must be efficiently generated through an interface to a CAD system.

Many graphics oriented finite element preprocessors are available which can generate very complex finite element models. Unfortunately, these models cannot be used directly for optimization, since they offer no means of parameterizing the shape of the structure. Ideally, for shape optimization, the design and analysis models should be generated simultaneously using a CAD system. An alternative to this approach is an optimization system which generates the analysis model automatically from the the design model description of the structure. The major disadvantage to both approaches is that the present state-of-the-art in mesh generation is not of a level where they would be robust enough to function in a real world design environment. However, finite element analysis is an accepted and established part of the design process. For immediate impact, a shape optimization system should be able to take advantage of this fact. Hence, the third approach to design modeling which is presented in this paper is one which utilizes an already existing finite element model as the basis for the geometry description. A parameterization of key dimensions, edges, and surfaces is then overlayed on the finite element mesh.

In this paper, different design modeling approaches are first discussed. A new approach which can handle large-scale problems initially generated as analysis problems only is presented. A steering control arm is used as an example to demonstrate the use of the design modeling approach.

DESIGN MODELING APPROACHES

When evaluating any modeling approach, the robustness of the technique and the difficulty of integrating the system into the design process are the two major criteria. A robust design model will be general enough to include every possible shape which will satisfy the design constraints. At the same time, the constraints must be flexible enough to eliminate the consideration of any impractical designs from a manufacturing standpoint. It is also important that the coupling between the design and analysis models be of a nature that maintains the integrity of the finite element analysis through the iterations of the optimization process.

Two design modeling approaches were found in the literature: a boundary design element concept, and a design element approach or a generic model approach. The present approach differs from both of these in that they both use some form of mesh generation while our approach uses mesh manipulation.

The boundary design element concept was first proposed by Bennett and Botkin [8] for two-dimensional plates and by Botkin and Bennett [9] for three-dimensional folded plate structures. The basic idea of this approach is to parameterize a boundary segment with several design variables, assemble all segments to form the whole part, and generate a finite element mesh within this boundary. The key to the success of this approach is the availability of a two-dimensional fully automatic mesh generator [10]. With this capability, a more advanced step which considers the accuracy of finite element analysis with mesh refinement was made possible [8]. This approach is probably the most robust and attractive as the creation of the finite element mesh is transparent to the designer. However, the boundary description format cannot be extended to three-dimensional solids because a fully automatic mesh generation [11] which relies on surface data is not developed to the point where it can be routinely used in an automated fashion.

The design element approach for two-dimensional elasticity problems was first used by Botkin) [12] and also used by Braibant and Fleury [13], who employed Bezier and B-spline functions for boundary geometry. The design element or generic modeling scheme for three-dimensional shape optimization was used in Refs. 5-7. This approach can be thought of as a volume design element concept. In this approach, the geometry is described by design elements whose key dimensions are associated with the geometric design variables. The finite element mesh for analysis is then generated within each design element by an isoparametric mapping technique. The advantages of this approach are that no discontinuity exists at the element interface, relatively few design variables are needed, and interior points are automatically adjusted when a boundary moves. The main disadvantage of this method is the relatively inflexible mesh generation scheme. Mesh gradation is completely controlled by the number of generic elements and the mapping technique used. Since the generation technique creates a very uniform mesh, refinement in a local region can only be accomplished by adding more design elements. For complex geometries which cannot be modeled with a coarse mesh, the density of the generic model quickly approaches that of the analysis model. In effect, the designer has to generate a full-scale finite element model anyway. With increased complexity of the mesh, the number of necessary design variables also increases. Although the finite element mesh generation is largely transparent in this approach, the quality of the mesh may not satisfy the designer who is used to generating finite element models with a graphics preprocessor. One other drawback to this method is that the designer will be restricted to using the finite element types permitted by the mesh generator.

PRESENT APPROACH

In the present approach, the original finite element model is employed as the basis for the design model. There is a one-to-one correspondence between the finite element analysis and design model geometry descriptions. That is, the node numbers and locations for both models are identical. The design model attaches design variables to the node locations stored in the analysis model. As the optimizer changes the design, the analysis model is updated to reflect the change in node coordinates, and the design model is updated to reflect the change in the design variables.

All the additional data needed to describe the shape optimization model is stored in a single DESIGN file. The present model contains two key elements. The first is a list of design variables with upper and lower limits. When the optimization is performed, the design variable vector moves toward the optimal design. The second key element of this model is the type of geometric operators which give these numbers physical significance by relating them to actual part dimensions. This is done by manipulating the coordinates of the nodes which describe the finite element mesh. Three types of operators are included in the design model. LINK functions form the most direct relationship between the design variables and the part geometry. Each LINK function references a design variable or a linear combination of any number of the design variables as specified by the user. This dimension is then used to position a list of dependent nodes relative to some independent reference. The type of reference depends on the type of LINK function specified. For example, if a cylinder function is used, all the dependent nodes are positioned relative to an axis. Unlike LINK functions, POLY and GRID functions do not explicitly reference design variables. Therefore, they allow the designer to minimize the number of design variables necessary to completely describe a problem. Like LINK functions, both of these functions position nodes in a specified list relative to some independent nodes. POLY functions do this by putting a polynomial curve through the independent nodes and interpolating the dependent nodes onto it. GRID functions set chosen coordinates of the dependent nodes to a value determined by a linear combination of the independent node coordinates.

The most time-consuming and tedious part of this approach is locating and identifying the independent and dependent nodes used in the geometric functions. To expedite this process, an interface with a CAD system should be developed. With such a graphical system, the business of determining and attaching the node labels to the geometric functions would be transparent to the user. The designer would have to select the nodes graphically off the screen, while the computer internally stores the appropriate numbers and builds the DESIGN file. A key feature of the shape design modeler, which will be implemented in the future, is the ability to animate the geometric functions. This will allow the designer to instantly see the effect changing an individual design variable has on the part geometry.

The main advantage of this method is that it is applicable at any point in the design process. The designer does not have to sacrifice the time already invested in building the analysis model if he decides to run an optimization. Also, this method has been shown to work on real problems with technology that is currently available.

MODULAR SYSTEM FOR SHAPE OPTIMIZATION

The design modeling technique described in the previous section is incorporated with a three-dimensional modular shape optimization system which uses MSC/NASTRAN for finite element analysis [5,14]. The system flow chart is shown in Figure 1. Each step is an independently executable module. CONMIN [15] is called as a subroutine from SENSTY. A fifth module (not shown) forms the link between STEP 4 and STEP 1. Termination is controlled by an iteration counter and can occur after STEP 2 or after STEP 4, as specified by the user. Steps 2 and 4 can be run independently to test the design model without running an analysis. In STEP 1, a NASTRAN static analysis is run using superelement formulation. The nodal coordinates, internal/external node label list, and displacements are written to an output file for use in the next steps. In STEP 2 (ADJLOD), stress and displacement constraints are evaluated. For those constraints which are active, adjoint loads are calculated. In STEP 3, each adjoint load is submitted as a separate load case in a restart on the analysis performed in the first step. Displacements from this analysis are written to an output file and used in the next step to calculate sensitivities. In STEP 4 (SENSTY), the gradients of the cost function and active constraints with respect to the design variables are evaluated. This information is fed to CONMIN, which forms Taylor series approximations of these functions and performs an optimization to arrive at the next design iteration. The grid coordinates are updated to reflect the new design variables. Then a Laplacian smoothing operation is carried out on all interior corner grids to minimize element distortion. Finally, the midside grids are linearly interpolated between their respective corner grids, except those on the boundary surfaces . The new coordinates and design variables are written to the NASTRAN and DESIGN files, respectively.

STEERING CONTROL ARM

The forged steel steering control arm shown in Figure 2 was optimized. The arm is subjected to a single 9000 N steering load applied through a ball stud as shown. Constraints are applied around the strut tube on the upper and lower surfaces of the arm to simulate the welds. The NASTRAN model consists of 190 HEXA elements, 6 PENTA elements, and 30 BAR elements (used to model the ball stud). There are 1497 grids in the model which corresponds to roughly 4300 DOF. Young's modulus, Poisson's ratio, and the allowable octahedral shearing stress are $2.07\text{x}10^5 MPa$, 0.3, and $250 MPa$, respectively. The optimization model shown in Figure 3 uses 12 design variables, 31 link functions, 15 polynomial interpolating functions, and 21 grid link functions. The numbered arrows represent the design variables. The lettered points are key node locations and the dashed lines are movable boundaries. The design variables are described in Table 1. Design variable 5 is actually fixed, but is needed to locate point F. Quadratic interpolation functions are used to generate curves KLM, BCD, and FGH. Cubic Hermite curves AB and DE form smooth transitions between BCD and the outside radii at the ball stud and the strut tube. Only half the model is shown in the XY-plane because it is symmetric about the X-axis. Figure 4 is a partial listing of the DESIGN file for this model. Three geometric constraints have been included to prohibit the inside wall boundary from crossing the outside wall boundary. The initial design is infeasible as the

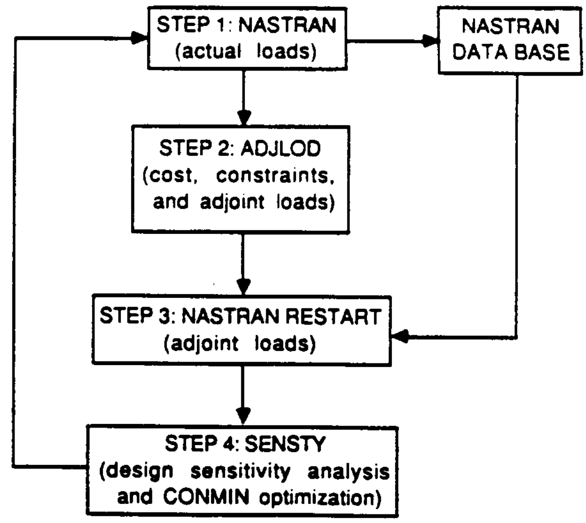


Figure 1. System flow chart

Table 1. Design Variables Description

Design Variable	Description
1	Floor thickness
2	Strut tube (MN) thickness
3	Midsection (L) thickness
4	Width of inside wall at ball stud (F)
5	Radius of inside wall of ball stud (fixed)
6	Width of inside wall at midsection (G)
7	Radius of inside wall of strut tube
8	Radius of fillet (HJ)
9	Position of fillet radius center
10	Width of outside wall at ball stud (B)
11	Width of outside wall at midsection (C)
12	Width of outside wall at strut tube (D)

part had a high stress value near the inside fillet radius at the strut tube (Figure 2). The peak stress in the part violates the stress constraint by 87.5%. The initial mass is 615.4 g. After 11 design iterations, the stress constraints were met and an 8% weight savings was achieved (final mass of 566.5 g). Table 2 lists the initial and final values of the design variables as well as the limits placed on them. A comparison of the initial and final geometries is given in Figure 5. The design histories of the mass and maximum stress constraint are shown in Figure 6.

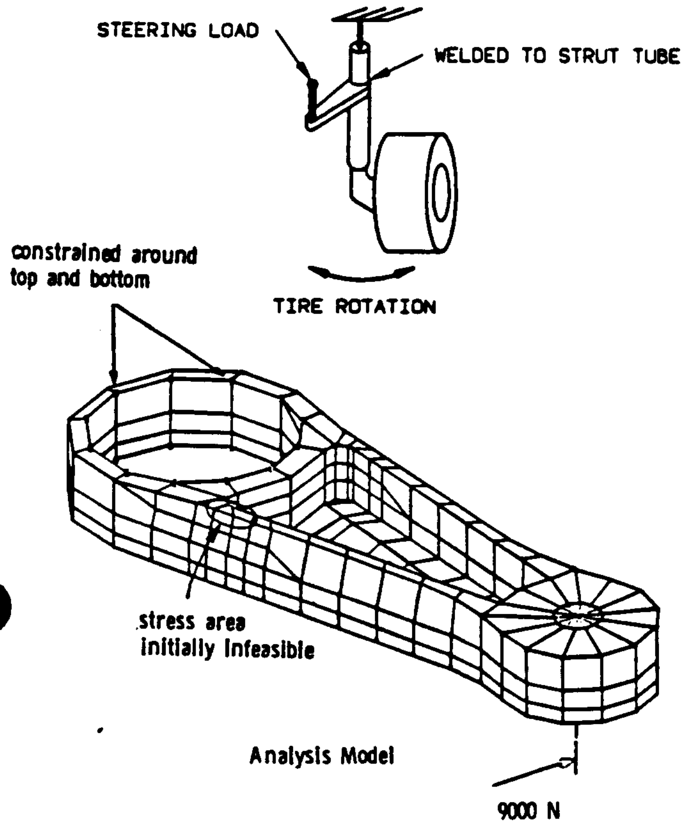


Figure 2. Steering control arm

Table 2. Design Variables for Steering Control Arm

No.	initial	final	lower bound	upper bound
1	4.20	2.50	2.50	10.00
2	20.00	20.00	20.00	40.00
3	20.00	22.12	11.00	40.00
4	8.97	8.64	2.20	17.00
5	20.75	20.75	20.75	20.75
6	15.86	17.95	2.20	26.00
7	34.30	30.46	30.40	40.00
8	4.00	5.72	1.00	10.00
9	15.29	11.30	2.20	18.00
10	18.61	19.00	4.20	19.00
11	21.98	22.26	4.20	30.00
12	26.21	25.46	4.20	29.00

unit: mm

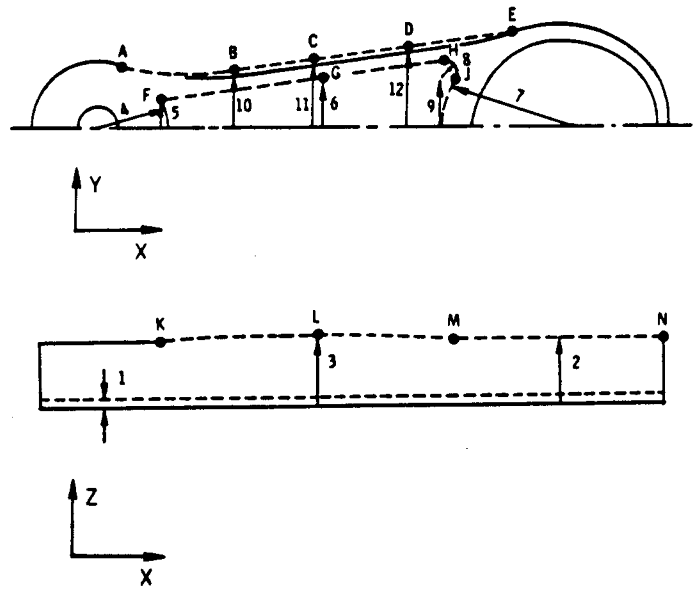
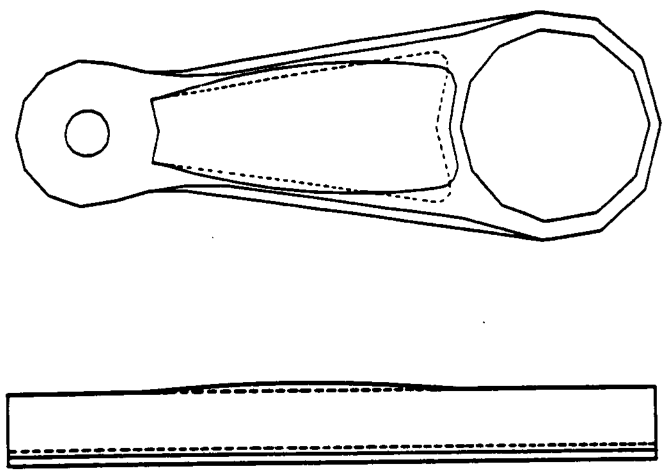


Figure 3. Design model for steering arm

```
$
PARAMETERS     ITER.NCOND / ICHECK.ISTOP.IDEBUG.ISMUTH.NSTR
    1    0
                              1    1    0     1    7
$
OPTIMIZATION PARAM    NACM.IFLERY.ILINC. EPS. PER. BMINLK.THETA
                       -1     -1     -1   0.10 0.10   0 05   -1 0
$
PROPERTY
    2 0684E05  0 3
    250
$
GEOMETRIC CONSTRAINTS
    2    5.
    -1.  10     1 0 4
$
GEOMETRIC CONSTRAINTS
    2    4.
    -1.  11     1.0 6
$
GEOMETRIC CONSTRAINTS
    2    5.
    -1.  12     1 0 9
$
STRESS CONSTRAINT REMOVAL
    70
    73  74  76  66  63  64  65  67  68  69  70  71  72  75
    80  77  56  51  46  45  47  52  57  60  61  62  79  78
    43  42  39  20  18  17  19  21  22  23  24  44  41  40
    207 202 201 206 204 219 225 226 221 212 211 210 209 208
    164 165 182 184 205 215 217 216 218 214 199 198 172 168
$
DESIGN VARIABLES  B4=6. B6=10. B9=10
    12
    1         2.500      4 200      4 200      10 000
    2        20 000     20 000     20 000      40 000
    3        11 000     20 000     20.000      40 000
    4         2.200      8 968      8 968      17 000
    5        20.750     20 750     20 750      20.750
    6         2 200     15 858     15 858      26 000
    7        30 400     34 300     34 300      40 000
    8         1 000      4 000      4 000      10 000
    9         2 200     15.285     15 285      18 000
   10         4 200     18.610     18 610      19 000
   11         4 200     21 976     21 976      30 000
   12         4 200     26 214     26 214      29 000
$
LINK                   FLOOR THICKNESS
    3  113   1   0 0
    1.0  1
    0.   0.   1
    76
    62   49  124  182  269  369  379  378  346  339  218  155   95   61
    63   50  125  183  270  297  316  315  29C  234  219  156   96   64
   440  515  590  679  779  903  936  947 1102
   453  528  603  692  792  916 1034 1060 1100
   458  533  608  697  797  921 1042
   448  523  598  687  787  911 1037
   427  502  577  666  766  897  975
   411  486  561  650  757  839  970  986 1008
   405  480  555  644  728  833  854  865 1010
  1223 1300 1391 1429 1445 1444 1436 1410 1332 1267 1157 1140 1074 1206
  1220 1298 1392 1430 1446 1447 1437 1411 1333 1265 1154 1142 1076 1208
$
LINK                   STRUT TUBE THICKNESS
    3   28   1   0 0
    1 0 2
    0    0    1
    76
  1232 1309 1401 1462 1493 1492 1474 1424 1342 1276 1166 1151 1084 1217
  1228 1306 1402 1463 1494 1495 1475 1425 1343 1273 1162 1150 1085 1216
$
```

Figure 4. DESIGN file for steering arm optimization



Initial dimensions shown as dashed lines

Figure 5. Initial and final designs of steering arm

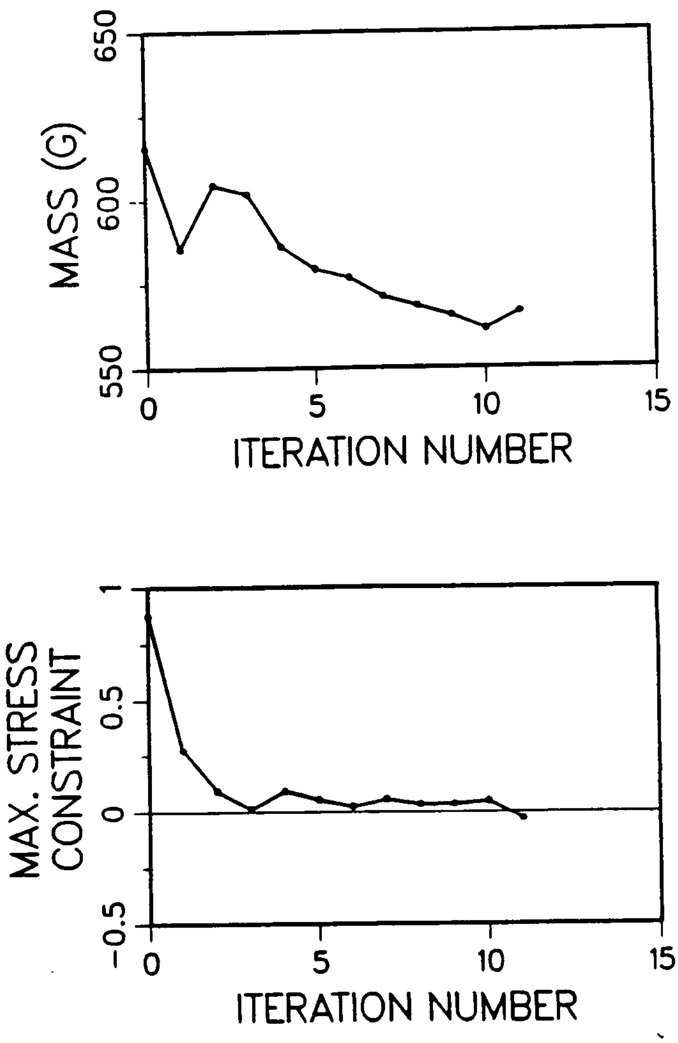


Figure 6. Design history of steering arm optimization

SUMMARY

Efficient creation of the the design model is crucial in three-dimensional shape optimization. In the ideal scheme, creation of the analysis model is completely integrated into the design model building process, thus eliminating any duplication of effort. At the same time, no compromise should be made with respect to mesh quality. For realistic three-dimensional parts, this technology is not available yet. In this paper, a design modeling approach was presented which takes advantage of the fully developed state of finite element analysis model building. In this method, the analysis model is the basis of the geometric description. Building the design model consists of overlaying a parameterization of the geometry onto the finite element mesh. This method is applicable with present technology. It has been used in a number of automotive component applications with success, one of which was presented here.

REFERENCES

1. Ramakrishnan, C. V. and Francavilla, A., "Structural Shape Optimization Using Penalty Functions," Journal of Structural Mechanics, 3(4), 1974-1975, pp. 403-422.
2. Haug, E. J., Choi, K. K., Hou, J. W., and Yoo, Y. M., "A Variational Method for Shape Optimal Design of Elastic Structures," New Directions in Optimum Structural Design, Ed. E. Atrek et al., Wiley, New York, 1984.
3. Haug, E. J., Choi, K. K., and Komkov, V., Design Sensitivity Analysis of Structural Systems, Academic Press, 1986.
4. Yang, R. J., and Botkin, M. E., "Comparison Between the Variational and Implicit Differentiation Approaches to Shape Design Sensitivities," AIAA, Vol. 24, No. 6, 1986, pp. 1027-1032.
5. Botkin, M. E., Yang, R. J., and Bennett, J. A., "Shape Optimization of Three-Dimensional Stamped and Solid Automotive Components," The Optimum Shape: Automated Structural Design, Ed. J.A. Bennett and M.E. Botkin, 1986.
6. Imam, M. H., "Three-Dimensional Shape Optimization," International Journal for Numerical Methods in Engineering, Vol. 18, 1982, pp. 661-673.
7. Imam ,M. H., "Minimum Weight Design of 3-D Solid Components", ASME Computers in Engineering, Vol. 3, 1982.
8. Bennett, J. A. and Botkin, M. E., "Structural Shape Optimization with Geometric Problem Description and Adaptive Mesh Refinement," AIAA, Vol. 23, No. 3, 1985, pp. 458-464.
9. Botkin, M. E. and Bennett, J. A., "Shape Optimization of Three-Dimensional Folded Plate Structures," AIAA, Vol. 23, No. 11, 1985, pp. 1804-1810.
10. Cavendish, J C., "Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method," International Journal for Numerical Methods in Engineering, Vol. 8, No. 4, 1984, pp. 679-696.

11. Shephard, M. S. and Yerry, M. A., "Automatic Finite Element Modeling for Use with Three-dimensional Shape Optimization," The Optimum Shape: Automated Structural Design, Ed. J.A. Bennett and M.E. Botkin, 1986.

12. Botkin, M. E., "Shape Optimization of Plate and Shell Structures," AIAA, Vol. 20, No. 2, 1982, pp. 268-273.

13. Braibant, V. and Fleury, C., "Shape Optimal Design, A Performing C.A.D. Oriented Formulation," Proceedings of AIAA/ASME/ASCE/AHS SDM Conference, CP No. 84-0857, Palm Springs, CA, May 14-16, 1984.

14. Yang, R. J. and Botkin, M. E., "A Modular Approach for Three-Dimensional Shape Optimization of Structures," AIAA, Vol. 25, No. 3, 1987, pp. 492-497.

15. Vanderplaats, G., "CONMIN - A Fortran Program for Constrained Function Minimization User's Manual," NASA, TM X 62,282, 1973.