Three-Dimensional Shape Optimization Using Fully Automatic Mesh Generation

M. E. Botkin*
General Motors Research Laboratories, Warren, Michigan 48090

Introduction

THE technique for associating design variables with mesh data is the most crucial factor in three-dimensional shape optimization. Previously, work in three-dimensional shape optimization involved specifying design variables by associating parameters directly with grid points on an existing mesh. For realistic problems this can be a very tedious (and error-prone) process. In the past, shape optimization capabilities have been developed based on a variety of design/analysis capabilities ranging from associating parameters with a mesh created manually[1] to associating parameters with control points of a mapped mesh generator.[2] Special techniques have also been developed to properly move internal grid points during sensitivity calculations.[3] More recently, a capability based on constructive solid geometry (CSG) has been developed,[4] but CSG representations are not particularly suitable for design optimization. A new optimization approach is demonstrated in this Note that uses a new, fully automatic mesh generation capability. The design model is developed based on design-oriented geometric primitives that represent recognizable features of a part and can be assembled into complete solid models that are defined in terms of a small set of design parameters.

Design/Analysis Modeling

This Note describes the integration of fully automatic mesh generation[5] with shape optimization. The geometric modeling capability described in Ref. 5 has certain advantages over other generic modeling approaches. Models can be assembled from a library of design primitives, some of which represent common features found in many parts; see Fig. 1. Primitives are selected according to a preassigned name, e.g., ARM or BOSS. Each design primitive is made up of a fixed number of *geometric model entities* (faces and edges) that are associated with a set of dimensions that specify the size of the object. Once the primitives have been assembled, there exists a fully parameterized, solid, geometric model of the part that can be meshed automatically using an OCTREE-based, three-dimensional mesh generation capability.[6] As the mesh is being generated, associations of nodes and elements with the geometric model entities are stored in the underlying data structure.

It should be pointed out that this is not a technique to exercise the ability to obtain extremely detailed parametric models. The purpose is to show how optimization can be used to select a few key dimensions of a model that embodies the important features of the actual part. Generally, solid models (CSG) that contain all of the detail of the part to be manufactured not only have too many parameters to be effectively optimized, but also the parameters are defined in such a way that they are not suitable to be used as design variables.

Received May 28, 1991; revision received Nov. 18, 1991; accepted for publication Nov. 19, 1991.

*Senior Staff Research Engineer, Engineering Mechanics Department, 30500 Mound Road.

Shape Optimization Methodology

Stress Constraint Definition

During the optimization iteration history, the number of constraints must remain constant. In several other shape optimization papers, a constraint is imposed on every finite element. This approach works fine as long as the number of elements remains constant throughout the iteration history. Using automatic mesh generation, however, allows the number of finite elements to vary throughout the convergence history. In this case, assigning a stress constraint to an element number would permit a constraint to exist at different physical locations at different steps in the optimization process. This would mean that a change in stress that has been detected by the optimizer may be due to that location change rather than to the design variable change. For this reason, the stress constraints are assigned to the *design model*, which has a fixed topology, rather than to the *mesh*. This can be easily accomplished due to the underlying data structure of the mesh generator, as noted earlier, which associates the mesh data with the model entities. At the time of constraint creation, then, the maximum stressed finite element is located for each model face and is used as the constraint for that face. Refinement of the constraint set can easily be obtained by removing the loaded and supported faces. Normally, elements directly adjacent to loads and boundary conditions have erroneously high stress values and therefore should not be used as constraints.

Table 1 lists the stress values for the model in Fig. 1. The table lists the maximum stress value on each *model face* for the ARM primitive and the element number (Elmax) that produced the maximum value. The final set of constraints used for optimization will be the values in the table normalized with respect to the limiting stress value. For supported or loaded faces, no optimization constraint will be defined. Also, any other face with Elmax equal to the element number on a supported or loaded face will not have an optimization constraint. The latter situation could occur on faces adjacent to supported or loaded faces.

Behavioral Sensitivities

The following relationship expresses the association between the geometric model $\boldsymbol{G}$ and the mesh $\boldsymbol{M}$:

$$\boldsymbol{G}(\delta) \rightarrow \boldsymbol{M} \tag{1}$$

in which δ represents the design variables. Whereas the model is an explicit function of the design variables, the mesh is only obtained through the operation ($\rightarrow$) in Eq. (1). Figure 2 represents a two-dimensional segment of a mesh, the associated geometry, and design variables using the notation of relationship of Eq. (1). The bold curve represents a typical shape variation. The subscripted variable $\boldsymbol{M}_o$ refers to mesh points on the surface geometry and variable $\boldsymbol{M}_i$ refers to points in the interior. It would not be desirable to regenerate an entirely new mesh for each design variable perturbation $\Delta\delta_i$ but only recompute the surface point locations $\boldsymbol{M}_o$ locally. Since the mesh is tied to the geometrical description through an operation, it is only indirectly associated with the design variables. For that reason a local remeshing procedure was implemented that operates only on the mesh data for specific edges and faces. Associativity information between the topology and the mesh allows only pertinent nodes to be identified for repositioning.

As described in Ref. 7 the behavioral sensitivities can be computed as follows:

$$\frac{\partial \boldsymbol{R}}{\partial \delta} = \frac{\partial \boldsymbol{R}}{\partial \boldsymbol{M}} \times \frac{\partial \boldsymbol{M}}{\partial \boldsymbol{G}} \times \frac{\partial \boldsymbol{G}}{\partial \delta} \tag{2}$$

in which $\boldsymbol{R}$ is a response quantity to be constrained. Reference 7 dealt with the second and third terms of Eq. (2). These terms obtain a perturbed mesh for each perturbed design variable

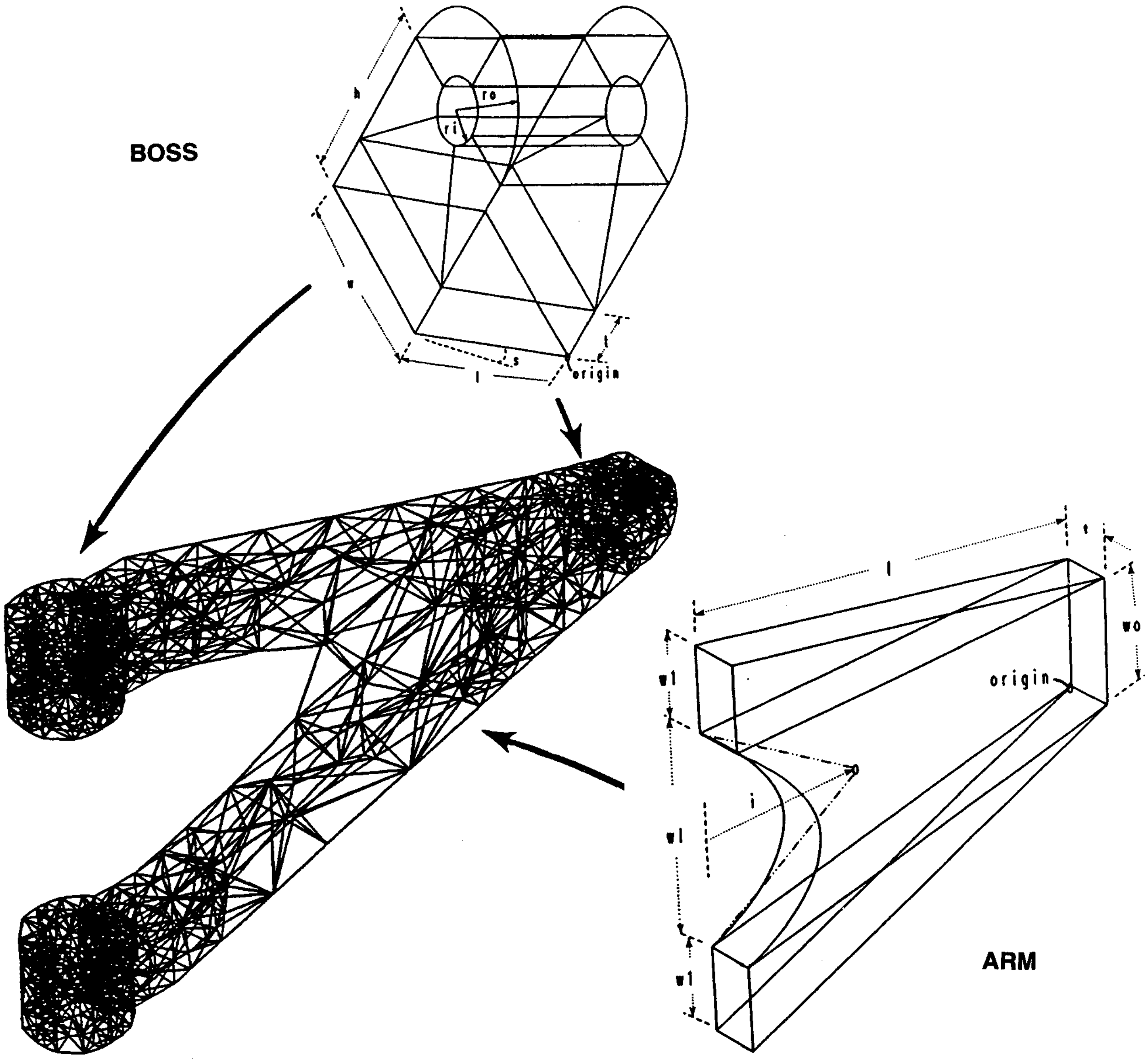

Fig. 1 Design/analysis models.

using the following steps in the process that must be done in batch mode and require interaction of the modeling program and optimization:

1) Increment design variable and perturb design variable value.

2) Call design-primitive-building routines and update model.

3) Call local remeshing routines to update mesh to match the model.

4) Subtract the new mesh from the original mesh to obtain mesh sensitivities.

5) Reset model to the original configuration and repeat for the next variable.

These data are used to create input data for the calculation of term one of Eq. (2) that is obtained by the finite element program.[8] It must be stressed that the modeling program must be an integral part of the optimization program.

Design Example

The following example was modeled using the mesh generation capability developed by Georges and Shephard[6] and documented in Ref. 5. Sensitivities and optimization were carried out within the finite element program.[8] All input data for the optimization run were created automatically based on the original geometrical design model.

This problem represents an idealization of a suspension steering knuckle. The following 17 parameters (in cm) determine the shape and are contained in the configuration data set[9]:

A01 = 6.00 ...HUB outer radius
A02 = 2.00 ...HUB thickness
A03 = 0.95 ...HUB shoulder thickness
A04 = 6.00 ...SLAB length
A05 = 1.00 ...BOSS (upper) radius
A06 = 2.00 ...BOSS (upper) width
A07 = 1.00 ...BOSS (lower) radius
A08 = 2.00 ...BOSS (lower) width
A09 = 1.00 ...BOSS (steer) radius
A10 = 2.00 ...BOSS (steer) width
A11 = 10.00 ...SLAB slope (deg)
A12 = 15.00 ...distance to upper BOSS (outer)
A13 = 9.00 ...distance to lower BOSS (outer)
A14 = 9.00 ...distance to steer BOSS load point
A15 = 3.00 ...HUB inner radius
A16 = 3.00 ...height of upper and lower BOSS
A17 = 0.50 ...inner radii of all BOSSes
NDV = 4 ...number of design variables

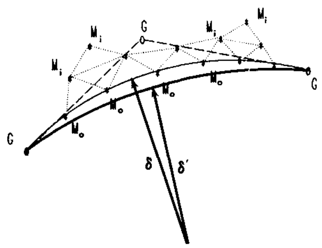

Fig. 2 Model/mesh relationship.

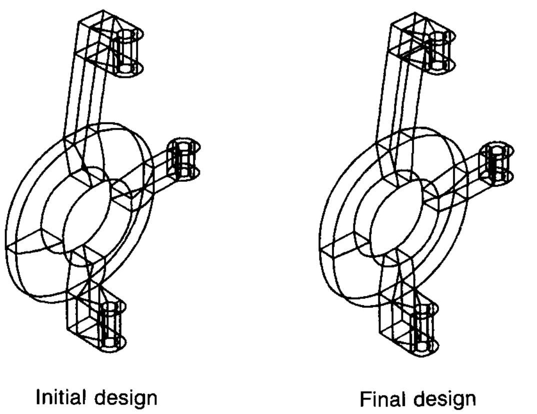

Fig. 3 Knuckle models.

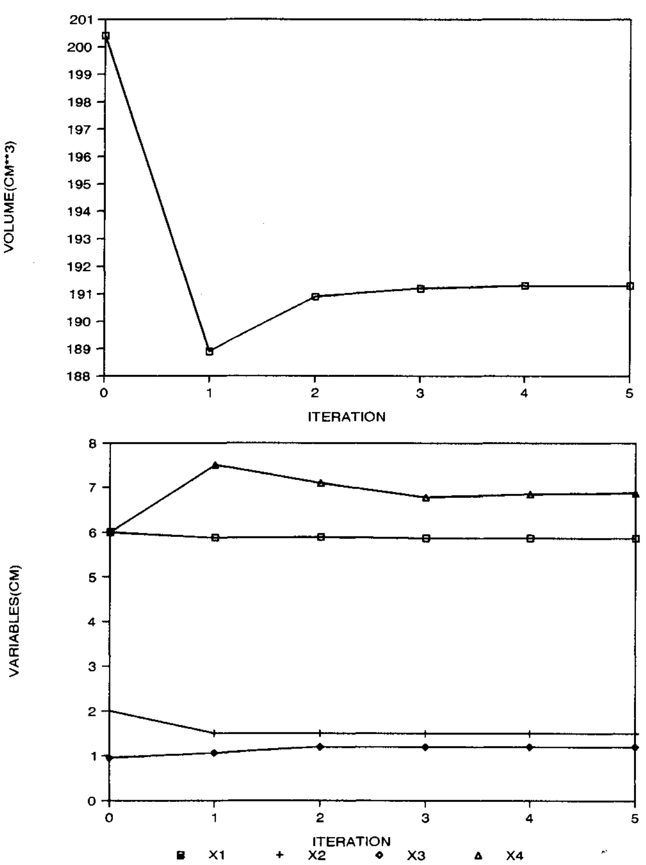

Fig. 4 Iteration history plots.

The design variables are the first four values as indicated by NDV. The design model faces and edges can be seen in Fig. 3 for the initial and final designs. There are 98 model faces resulting in 76 stress constraints when the 22 loaded or constrained faces are removed. The steering knuckle is loaded with 1000 N in the global z direction at the upper control arm boss, 3000 N in the y direction at the lower control arm boss, 1000 N in the y direction at the steering arm boss, and clamped at the center of the hub. The finite element model consists of 4513 linear (four noded) tetrahedral elements and 1403 nodal points. Young's modulus is $10.E+6$ N/cm^2 and the yield stress is 9000 N/cm^2. The maximum stresses occur on the hub at the locations where the arms attach. The optimization iteration history is shown in Fig. 4 for design variable bounds of $\pm 25\%$. The tendency was to thicken the shoulder of the hub to which the arms are attached. Since this is an idealized problem, the amount of mass savings is not as important as how well the program handles the problem, which can be seen by the smooth convergence in the last two steps when the stress constraints are active. The computer time for the entire iteration history was 7.5 CPU min on an IBM 3090 mainframe. Of that time only 79 s were spent in mesh generation. This indicates that mesh generation is a relatively small part of the overall cost of automated design.

Conclusions

The following conclusions can be drawn from the design study. Design models, complete with all data necessary to submit for an optimization run, can be created in a few hours. Mesh sensitivities obtained from the geometric model operations are adequate for design optimization. Design histories exhibited good convergence in less than 10 finite element analyses.

References

[1]Yang, R. J., "SHOP-3D: Shape Optimization of 3-D Structural Components," *Computers and Structures*, Vol. 31, No. 6, 1989, pp. 881-890.

[2]Imam, M. H., "Application of Shape Optimization to the Engine Main Bearing Cap," *ASME Computational Engineering Conference*, Vol. 3, American Society of Mechanical Engineers, New York, 1982, pp. 119-126.

[3]Rajan, S. D., and Belegundu, A. D., "Shape Optimal Design Using Fictitious Loads," *AIAA Journal*, Vol. 27, No. 1, 1989, pp. 102-107.

[4]Kodiyalam, S., Kumar, V., and Finnigan, P., "A Constructive Solid Geometry Approach to 3-D Shape Optimization," *Proceedings of the AIAA/ASME/ASCE/AHS/ASC 32nd Structures, Structural Dynamics, and Materials Conference*, Washington, DC, April 1991, (AIAA Paper 91-1211).

[5]Botkin, M. E., "Shape Design Modeling Using Fully-Automatic 3-D Mesh Generation," *International Journal of Finite Elements in Analysis and Design*, Vol. 10, No. 2, 1991, pp. 165-181.

[6]Georges, M. K., and Shephard, M. S., "Automatic Three-Dimensional Mesh Generation by the Finite Octree Approach," *International Journal of Numerical Methods in Engineering*, Vol. 32, 1991, pp. 704-749.

[7]Botkin, M. E., "Shape Sensitivities using Automatic 3-D Mesh Generation," *Third NASA Symposium on Multidisciplinary Analysis and Optimization*, San Francisco, CA, Sept. 1989, pp. 210-215.

[8]Miura, H., "Handbook for Structural Optimization: MSC/NASTRAN Version 66," MacNeal-Schwendler, Pub. MSR-87, Los Angeles, CA, Dec. 1988.

[9]Bajorek, D. "Generic Model Development for Component Design," B.S. Thesis, GMI Engineering & Management Inst., Flint, MI, April 1990.