Optimizing the Shapes of Mechanical Components

By generating basis shapes and using optimization techniques, it is possible for a designer to optimize the shape of a component so as to minimize an objective such as weight or peak stress. Basis shapes can be generated using available finite element pre- and post-processors.

Ashok D. Belegundu
Associate Professor
Department of Mechanical Engineering
Pennsylvania State University
University Park, Pa.

In optimization, the goal is to minimize an objective, such as weight or peak stress, subject to constraints on strength and performance. Optimization problems may be placed in three categories: sizing, shape, and topology. Sizing relates to determining values of design parameters such as thicknesses and cross-sectional areas and inertias. Shape optimization relates to determining the outline of a component and dimensions such as the radius of a fillet or hole. Topology addresses the basic question of where material should and should not be.

The focus here is on shape optimization with finite element models. The field has advanced to the point where vendors and users are showing keen interest in implementing such capability.

Research into the optimization of structures dates back to the 17th century classical treatise of Galileo, who investigated the optimum shapes of beams. In the 18th century, mathematicians posed certain classical shape-optimization problems. For example, the "isoperimetric problem" required finding the shape of a closed curve of fixed length that encloses the greatest area (answer: a circle). From 1960 through 1980, considerable research was done on using classical mathematical approaches to determine the optimum shapes of columns, plates, arches, and shafts. The objective in these efforts was to maximize the buckling load (or lowest natural frequency) or to minimize the displacement in the structure for a given amount of material.

In the last two decades, advances in finite element analysis and com-

Figure 1. The shape of this torque arm component is to be optimized for minimum weight using the basis shapes shown in Figure 2.
puters have shifted the focus to using numerical methods for shape optimization. Numerical methods are needed to optimize engineering components that generally have complex loading and geometry. Moreover, material reduction or performance optimization needs simultaneous consideration of design and manufacturing. Shape optimization has been successfully applied to a variety of engineering problems in the automotive, aircraft, and other industries. A few examples are fillets, turbine disks and blades, pre-forms in forging, channels or ducts, stress minimum forms for elastic solids, and airfoil shapes.

**Steps Toward Optimization**

Shape optimization involves three main steps: generation of basis shapes, optimizer execution, and controlling mesh distortion. A discussion of each of these steps follows.

We could change the shape of a component in an infinite number of ways. To make the problem manageable, we need to affect only a few shape variables. This is achieved by generating basis shapes. The optimization problem reduces to finding the right combination of these basis shapes to optimize the weight or other criteria. For example, consider the design of an airfoil. Many airfoils that have been tested to determine lift and drag characteristics already exist. Thus, even though we wish to design a new airfoil for a unique application, it is reasonable to use previous designs as basis shapes.

Basis shapes ultimately determine the optimum shape of the component. From structural, manufacturing, and aesthetic considerations, they have to satisfy many form-preserving requirements, such as straight edges remaining straight, portions of the boundary remaining unchanged in shape, and symmetry. Basis shapes must be smooth in order to avoid stress concentrations and must be able to reduce component weight and peak stresses.

Basis shapes can be generated in a number of ways. As mentioned, airfoil shapes from past designs can be used as basis shapes. Often, a designer has an outline or even a finite element model of a certain component and wishes to optimize its shape. In the approach presented here, basis shapes are obtained by generating variations in the shape of the original component. The key idea is to apply loads or prescribed displacements on the component and treat the resulting sets of deformed shapes as basis shapes.

Consider the automotive component shown in Figure 1, which is subjected to two point loads as shown. The design task is to minimize the volume of the component while limiting the von Mises stress in each element. The shape of the slot and of the outside boundaries is to be changed while the holes and semi-circular regions around the holes are to be unchanged. We first create a finite element model of the torque arm component. The model is constrained around the holes so that the resulting displacements will not change the hole size and locations, as per the designer's specifications. Now, eight sets of pressure loads are applied on the model, resulting in eight corresponding sets of displacements. These displacements are referred to as "velocity field" vectors in shape optimization. Basis shapes corresponding to each set of displacements are plotted in much the same way as deformed plots are obtained from displacement fields in finite element analysis. Four of the eight basis shapes for the torque arm are shown in Figure 2.

Once the displacements are generated, changes in the shape of the component are expressed as

\[
\text{GRID}_{\text{new}} = \text{GRID}_{\text{old}} + \sum_{i=1}^{ndv} b_i \mathbf{q}_i
\]

where \( \text{GRID} \) = a vector consisting of the \( x, y, \) and \( z \) coordinates of each node in the model; \( ndv \) = number of design variables; \( b_i \) = \( i \)th design variable; and \( \mathbf{q}_i \) = \( i \)th displacement vector. In the torque arm example, we have \( ndv = 8 \). The significance of Equation (1) is that the shape of the component is controlled by the magnitudes of a finite number of design variables, \( \{ b \} \). The actual choice of the design variables to improve the shape are determined by an optimizer.

The attractive feature in this approach is that basis shapes can be prepared using already available finite element models and post-processors. Also, the designer may make any necessary adjustments to the finite element model to obtain basis shapes that have the desired appearance and satisfy form-preserving requirements. Basis shapes can be generated for general problems using standard finite element input data preparation. Further, solid modellers can be integrated with this approach to generate basis shapes.

**Optimizer Execution**

The generation of a small set of basis shapes allows us to express the shape of the entire component as a combination of these shapes. Thus, the shape of the component can be controlled by only a small set of design variables. Quantities such as weight or inertia and response quantities such as stresses, displacements, and frequencies all depend on the selection of the design variables. Available optimization algorithms or interactive design procedures may be used to determine the best value of the design variables to improve the shape toward an optimum. Optimum means that an objective such as weight of the component or peak stress in the component is minimized subject to satisfying the design criteria. The shape-optimization procedure can be integrated with any finite element analysis and pre- and post-processing programs.

As the shape is Iteratively changed
in the optimization process, the finite element mesh gets increasingly distorted. Mesh distortion needs to be controlled so that finite element analysis and response calculations are reliable. Further, distortion control and remeshing allow for larger shape changes to occur and ensure that the shape process terminates gracefully. Although much work has been done on this topic in purely finite element applications, implementing such capabilities in shape optimization deserves further attention and is very important for developing a robust program. Here, we have two controls in place. First, the maximum shape change at any step is restricted so that the Jacobian for each finite element remains positive. Second, a rezoning scheme is used to alleviate mesh distortion. More work needs to be done on integrating distortion measures for diverse models, developing reliable rezoning formulas for three-dimensional components, and h- and p-adaptivity.

**Design Examples**

An in-house FORTRAN code has been developed for shape optimization. The code consists of the following: a module for finite element analysis; an optimization module; a module for generating basis shapes; a module for updating the shape based on combining the basis shapes; and a sensitivity analysis module, which interfaces with the optimization module.

The MSC/Nastran program has been used for finite element analysis. Since MSC/Nastran is a stand-alone system, a special command procedure on a Vax workstation has been written to execute MSC/Nastran and the other shape modules in a design loop. The DMAP feature provided in MSC/Nastran has been used to save and retrieve data blocks. Pre- and post-processing have been done using the MSC/XL and SDRC Ideas programs.

The shape-optimization procedure discussed earlier is now applied to two mechanical components. **Torque arm component.** The torque arm in Figure 1 has been optimized for minimum weight using the basis shapes shown in Figure 2. A limit on the maximum von Mises stress in an element is imposed. An optimum shape is obtained in 17 iterations with a 40 percent reduction in weight. The stress plots for initial and optimum shapes are shown in Figure 3. In these color plots, red represents high-stress regions, and blue represents low-stress regions. It can be clearly seen that the optimized design uses the material more efficiently, since larger portions of the component are stressed to the limiting value.

**Connecting rod problem.** A connecting rod loaded in tension is optimized for minimum weight subject to stress constraints. The initial and final shapes are shown in Figure 4. The weight reduction is 33 percent. Only one loading condition is considered here, corresponding to an axial tension load applied at the big end with the small end held fixed. However, multiple loading conditions that the connecting rod experiences in operation may be readily considered in the shape-optimization process for a more realistic design.

---

**Acknowledgments**

The author wishes to thank the MacNeal-Schwendler Corp. for funding part of this research. Thanks also to Ravikrishna Salagame, a graduate research assistant at Penn State, and Shenghua Zhang, a staff engineer at MacNeal-Schwendler Corp.

**For Further Reading**