DESIGN IMPROVEMENT BY SHAPE CHANGE USING A LINEAR FINITE ELEMENT METHOD

Ryan Adams*, Cees Bil*, Josef Tomas**
*School of Aerospace, Mechanical and Materials Engineering, RMIT University, Melbourne, Australia
**Advea Engineering Pty. Ltd. Melbourne, Australia

Keywords: *sensitivity, finite elements, design improvement, optimisation*

Abstract

Mathematical methods of optimization[1] have been used in the engineering workplace for a long time[2], but due to some theoretical problems with their application to the solution of discrete finite-element models, such methods have been slow to be adopted by non-academic users[3].

In recent years, various methods of design modifications based on finite-element methods of structural analysis have been developed. Some of these methods have been implemented in commercial computer programs, such as, for example, OptiStruct, NASTRAN, ReSHAPE and Tosca. The problems presented in this paper have been solved using the software ReSHAPE, developed by Advea Engineering, Australia[4].

1 Introduction

In design problems, various scalar response quantities can be defined, for example, stress, natural frequency, critical buckling force, reaction force, displacement. These responses are functions of the geometry of the structure. The relationship can be mathematically exploited for the calculation of a direction vector for each *node* location that will result in an increase or decrease of some scalar response quantities, while constraining the values of other responses.

Let's consider a structural component that is designed to handle certain loads, then analyzed using the finite-element method

(FEM), and finally manufactured to become part of a prototype for testing purposes. Following the prototype testing, the component is targeted for performance improvement, i.e. the increase or decrease of selected responses. It is at this stage where recent software developments have given engineers a whole new collection of tools to help them improve the performance of structural components.

This paper briefly discusses the theoretical basis of the methods used for shape improvement, followed by the solutions of two practical design improvement problems.

2 Sensitivity Analysis

The underlying method that forms the basis of shape changes to finite element models is the *sensitivity analysis*. Consider an FE model which consists of *nodes* in space, connected by *elements* which describe the geometry, materials and loads on the structure. Certain responses of interest are selected for possible improvement. The change of the position of one of the nodes in the model will have an effect on the responses. Each response r is a function of the geometry, $\mathbf{x}$, and the displacement response, $\mathbf{u}$:

$$r = r(\mathbf{x}, \mathbf{u}(\mathbf{x})). \qquad (1)$$

In order to determine the effect that moving a node has on the responses, the rate of change of each response with respect to the change of nodal coordinates is calculated as

$$\frac{dr}{d\mathbf{x}} = \frac{\partial r}{\partial \mathbf{x}} + \frac{\partial r}{\partial \mathbf{u}} \cdot \frac{d\mathbf{u}}{d\mathbf{x}}, \quad (2)$$

where $\partial r / \partial \mathbf{x}$ and $\partial r / \partial \mathbf{u}$ are easily calculated, and $d\mathbf{u} / d\mathbf{x}$ is evaluated from the finite element equation,

$$\mathbf{K.u} = \mathbf{f}, \quad (3)$$

where K is the stiffness matrix and $\mathbf{f}$ is the vector of loads.

In the case of eigenvalue responses (vibration or buckling) the required sensitivities are calculated differently. For example, the buckling sensitivity is solved from the characteristic equation

$$\left(\mathbf{K} + \lambda \mathbf{K}_g \right) \cdot \mathbf{u} = 0, \quad (4)$$

where K is the stiffness matrix, K_g is the geometry matrix λ is the eigenvalue and u is the eigenvector which are all functions of the geometry, x.

The equation is solved by calculating the eigenvector u and the eigenvalue λ. The eigenvector is normalized as

$$\mathbf{u}^{\mathrm{T}} \cdot \mathbf{K}_g \cdot \mathbf{u} = \mathbf{I}, \quad (5)$$

The equation (4) is multiplied from left by u^T, and differentiated with respect to the geometry, x, so that

$$\mathbf{u}^{\mathrm{T}} \left(\frac{d\mathbf{K}}{d\mathbf{x}} + \frac{d\lambda}{d\mathbf{x}} \cdot \mathbf{K}_g + \lambda \cdot \frac{d\mathbf{K}_g}{d\mathbf{x}} \right) \cdot \mathbf{u} = 0. \quad (6)$$

Finally, the sensitivity $d\lambda/dx$ of the eigenvalue to a change in the geometry is given as

$$\frac{d\lambda}{d\mathbf{x}} = \mathbf{u}^{\mathrm{T}} \cdot \left(-\frac{d\mathbf{K}}{d\mathbf{x}} - \lambda \frac{d\mathbf{K}_g}{d\mathbf{x}} \right) \cdot \mathbf{u}. \quad (7)$$

Various methods exist for the calculation of $d\mathbf{K} / d\mathbf{x}$ such as semi- and fully-analytical approaches. ReSHAPE uses a fully analytical approach (see [5] for further details).

In the presence of constraints, the sensitivity of the objective is projected onto the hyper-surface of the constraints, by the use of a projected gradient method.

3 Practical examples

Two examples have been selected as typical for structural shape design improvement.

The first example solves the problem of weight reduction of a typical automotive control arm in a suspension module, under the constraint of unchanged maximum stress.

The second example demonstrates the weight reduction of an aerospace component.

3.1 Weight reduction of a control arm

3.1.1 Overview
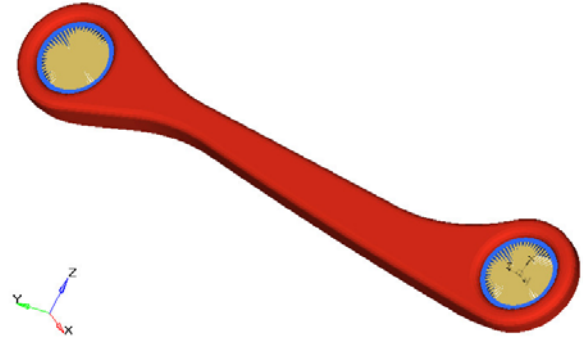A control arm of a suspension system of a small car was found to be too heavy.

Fig. 1. Finite-element model of a suspension control arm

As part of the normal analysis procedure, a finite element model has been created, shown in Fig. 1. The NASTRAN model consists of 4115 linear shell elements (CQUAD4, CTRIA3), 550 solid elements (CHEXA, CPENTA) and 2 rigid body elements.

The control arm is manufactured using a stamping process, from 4.0mm thick steel sheeting.

The FE model is used to validate the design before prototyping, sparing the need for expensive prototyping revisions. Typically a range of finite element analyses, both linear and non-linear, are employed.

The control arm was selected by the engineer for a possible weight reduction. The only efficient way of reducing the weight of components manufactured from sheet metal is to reduce sheet thickness. This would however, result in the increase of stresses. Therefore the weight reduction process has to be divided in

two stages: 1) stress reduction by shape change of the original component; 2) weight reduction by the reduction of the gauge of steel that the component is stamped, under the constraint that the original value of maximum stress must not be exceeded.

After examining the possible loads on the control arm, the five most critical loading conditions were selected to represent the most extreme loading that the control arm would ever undergo. The selected loading scenarios were:

- 1. Extreme cornering
- 2. Vertical loading
- 3. Braking while reversing
- 4. Horizontal loading
- 5. Gear changing under power

3.1.2 Structural Analysis

After reviewing the structural analysis results, two load cases were clearly critical (vertical loading and gear changing under power). The von Mises stress contour plot for the 'gear changing under power' case is shown in Fig. 2.
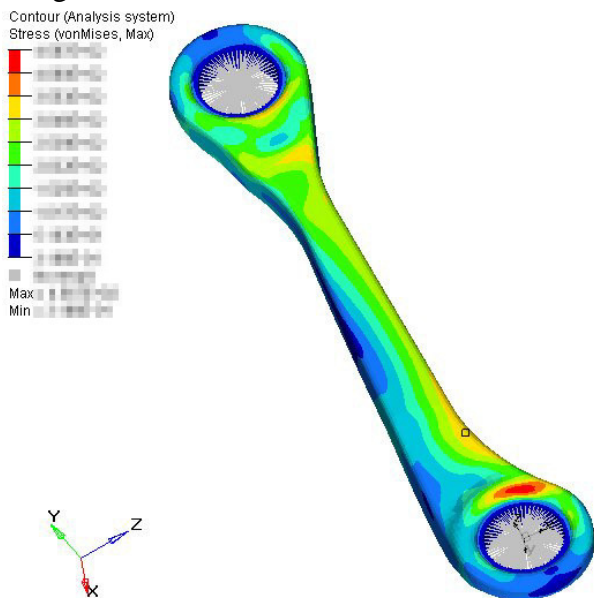

Fig. 2. Von Mises stress contour for changing gears under power

3.1.3 Sensitivity Analysis

A sensitivity analysis was carried out using ReSHAPE in two steps: first a sensitivity analysis on reducing the stress under loading scenario 5, and then a sensitivity analysis on reducing the stress under loading scenario 2.

The ReSHAPE command file for the sensitivity analysis is shown in the Appendix.

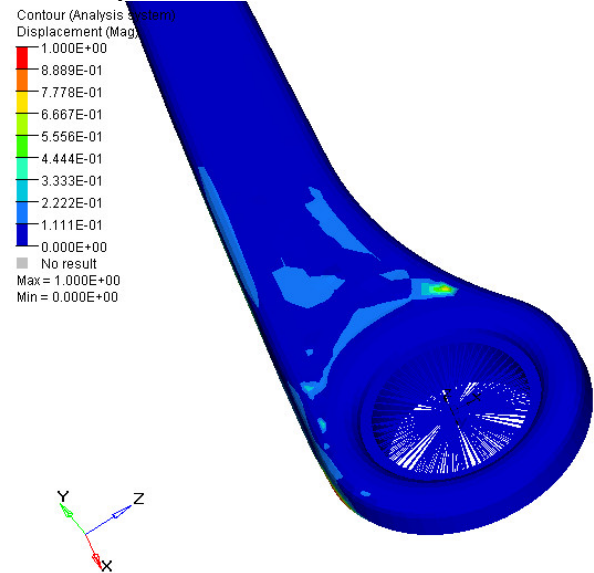Figure 3 shows the resulting sensitivity contour plot.

Fig. 3. Sensitivity contour plot for changing gears under power

The contour plot for the sensitivity of the second critical loading case (scenario2: vertical loading) is shown in Fig. 4.
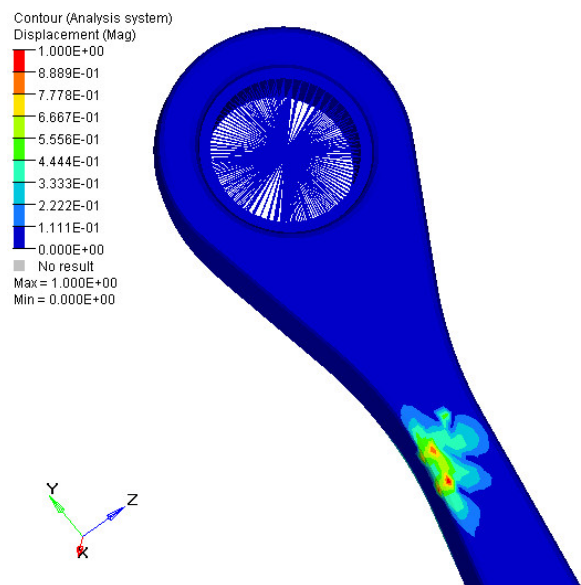

Fig. 4. Sensitivity contour plot for vertical load

Both sensitivity contour plots clearly indicate the region that should be the focus of a shape change in order to reduce the stress.

3.1.4 Shape Change

The next step is to perform the actual shape changes that will reduce the magnitude of the stress response, based upon the sensitivity analyses.

The first shape change was done for the gear changing loading scenario.

The method used for the shape change is the *method of influence functions.* A set of control points is defined by the user. From each of these points, a *virtual displacement field* emanates, defined by the influence functions. These functions have unlimited support, therefore internal influence functions are generated from points on domain boundary and used as constraints on the user defined functions. The product of the sensitivity vector with the vector of influence functions is sampled in the control points and then reflected back through the influence functions to nodal changes. In this way, a smooth, improved shape for the response is obtained.

The ReSHAPE command file for the design improvement is shown in the Appendix.

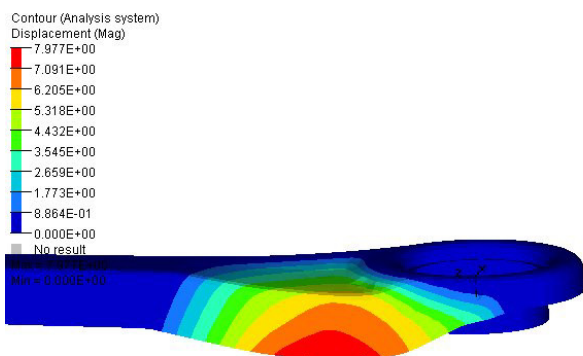The resulting shape is shown in Fig. 5.

Fig. 5. Shape change for reduced stress under gear changing load

A new NASTRAN model was then created, implementing the shape changes. Then the same method for design improvement was applied for the 'vertical loading' case starting with the new shape. The resulting shape is shown in Fig. 6.
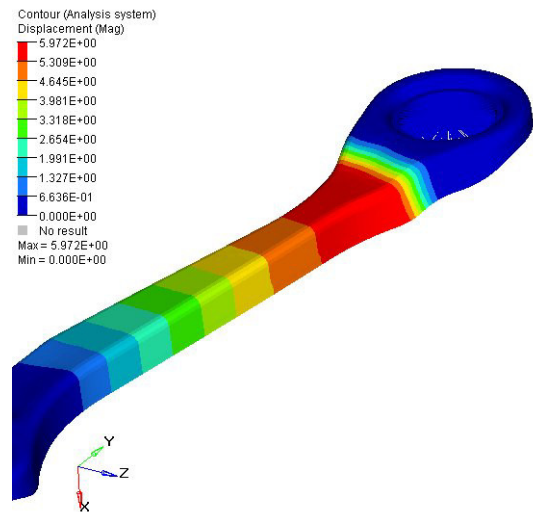

Fig. 6. Shape change for reduced stress under vertical load

The shape changes successfully reduced the maximum stress by more than 15%, which was immediately transferred to a weight saving by reducing the thickness of the component.

3.1.5 Discussion

The computer software used, ReSHAPE, also has the ability to perform shape changes under multiple loading situations at the same time. This automatic approach, however, should be used with care, and the experience shows that it is often more efficient to consider each loading case separately, as it was done in the presented case.

Some companies work to a standard for their computer aided designs. For example, some automotive and aerospace companies may only accept NASTRAN results as correct. In order to satisfy these requirements the CAE analyst performing the shape change can easily analyze the final model by NASTRAN, because the software ReSHAPE already uses the NASTRAN input deck format and generates the modified NASTRAN input deck. (On a side note, this also made learning ReSHAPE extremely fast, because no new format is to be learnt).

3.1.6 Conclusion

The weight of the final component was 10% lighter than the original.

It is important to note that a close relationship between the designer and the CAE analyst was essential to the success of this project, as the possible shape changes could be accepted quickly, or altered as required for manufacture, or to satisfy any packaging constraints.

3.2 Weight reduction of an aerospace component

3.2.1 Overview

In aerospace applications, the single most important design driver (after functionality of course) is weight. Aerospace companies strive for reduced weight vehicles in different ways, such as by implementing new and innovative materials or by designing components in a more efficient manner. This example addresses the latter method, in order to redesign an aerospace component for reduced weight. Fig. 7 shows the original design.
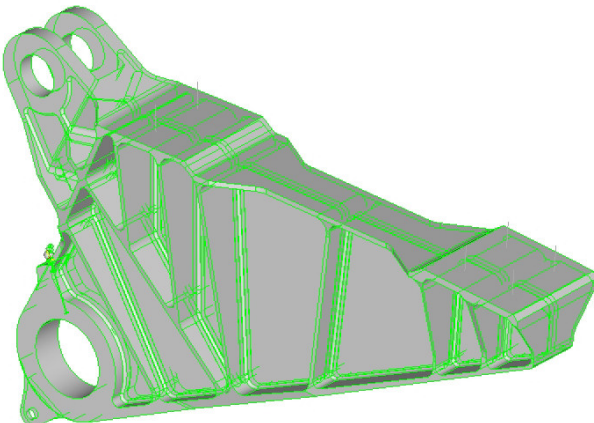
Fig. 7. Original design of an aerospace component

The stiffeners are used in the design mainly to arrest any fast crack growth in the case of shear web failure, to prevent buckling of the shear web, and to provide some torsional stability. They contributed very little to the stress response, and so they were removed for the initial shape investigation, to be replaced later. The component without stiffeners is shown in Fig. 8.
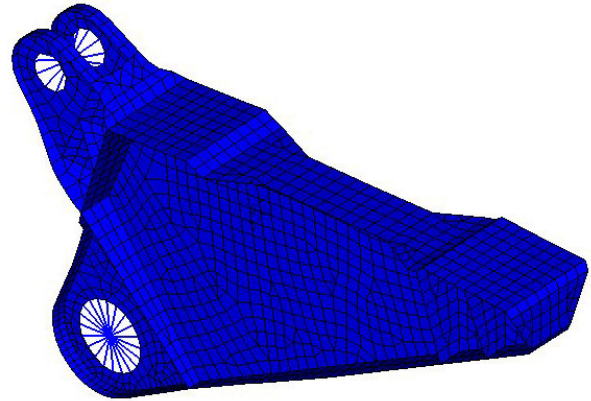
Fig. 8. Finite-element model with stiffeners removed

The model contains just 1822 linear shell elements, which means that this model will be solved with a minimum of computer time for the iterative shape improvement. The hinge/actuator connection points are modeled with RBE2 rigid elements.

The objective was to reduce the weight of the component but constrain the stress under multiple load cases.

3.2.2 Sensitivity Analysis

After removing the stiffeners, a sensitivity analysis is performed to see which area of the structure should be changed to have the most effect on reducing the weight (but keeping the stress unchanged). The volume reduction objective is subject to the constraint that the stress level must not increase. Mathematically this means that the sensitivity vector array for the objective is projected onto the sensitivity vector array for the constraint using a projected gradient method.

The sensitivity analysis to reduce volume and constrain stress is shown in Fig. 9.

The sensitivity analysis was restricted to a domain that didn't include the hinge and actuator support locations because there were no provisions for these areas to be re-designed.

Further, the change was restricted to changes parallel with the plane of symmetry of the component (the active domain was defined in a relevant local coordinate system).
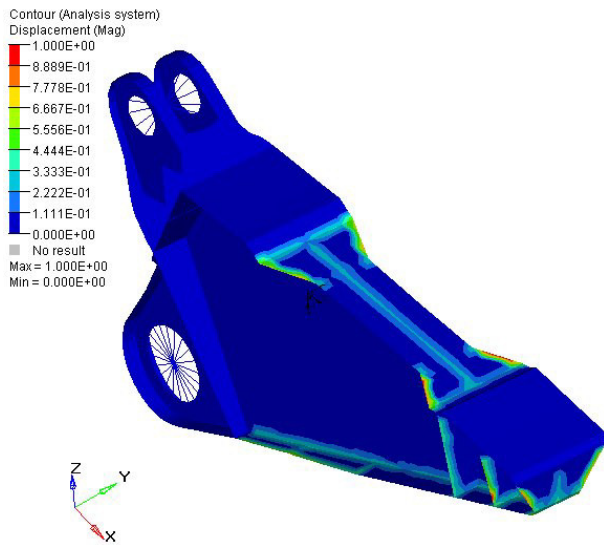
Fig. 9. Sensitivity analysis contour plot

3.2.3 Shape Change

Once the sensitivity has been checked to make sure there are no strange results from skewed elements, the shape change process can be started.

The method selected in ReSHAPE for shape change was again the method of influence functions. In this case however, an alternative of this method was used, with an automatic selection of control points.

Investigation of the model reveals that the active domain has many nodes on its boundary, i.e. the nodes not allowed to be change (boundary with the hinge and actuator support region, two support surfaces at the top). In this case, one control point is generated internally in the centre of the volume of the active domain and then this function is modified by influence functions generated from the boundary nodes.

As a remark, if there is not enough boundary nodes available, ReSHAPE provides another method which creates a grid of control points in 3D space around the object, providing good coverage and flexibility for the influence function sampling.

For an initial investigation, the first method was employed. The resulting shape is shown in Fig. 10.
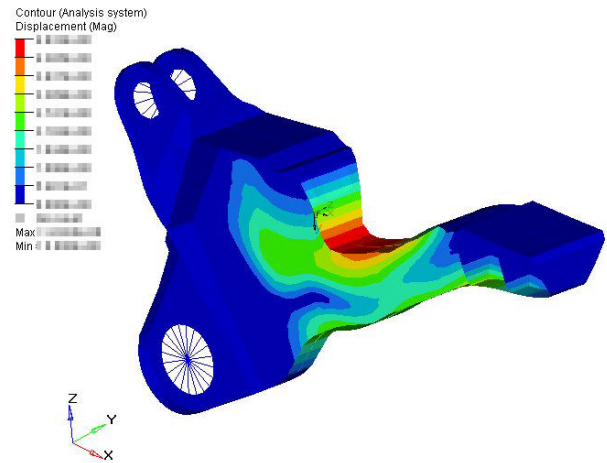

Fig. 10. Shape change with automatic control

The shape change clearly shows the load paths through the structure. The bolted connections at the front are quite clearly being supported through the bottom actuator, and the bolted connections at the back are supported by the hinge.

If it is desirable to have a smoother shape change, cubic-splines can be applied to the selected region as constraints to influence functions. Such shape change is shown in Fig. 11.
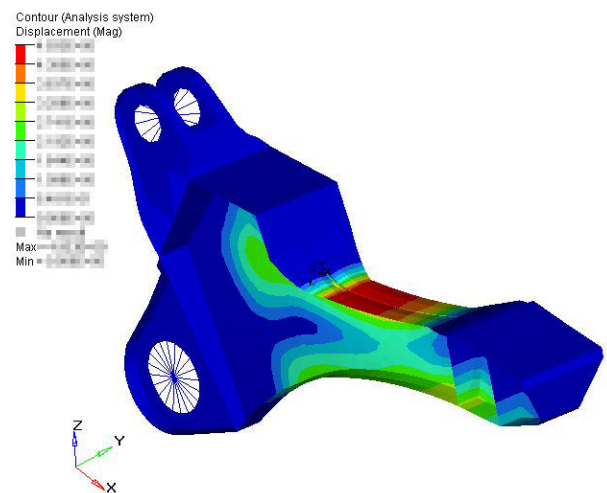

Fig. 11. Shape change with spline constraint

The shape change reduced the volume by more than 25%. Once the stiffeners were added back in, the final shape compared to the original shape was over 15% lighter, with precisely the same stress response.

Components of this nature are usually manufactured from a solid billet of a material using a milling machine. This means that more

complex shapes could be investigated, such as variable shear-web/stiffener thicknesses.

A suitable method for these kinds of investigations is so called *sizing*. A typical approach for an engineer would be to strive for the reduction of thickness where there is a low stress. This common sense approach becomes an extremely tedious procedure and it is much more efficient to use a computer program to find a solution, or identify whether the current design can be improved at all by this type of change. (Refer to [4] for sizing process in ReSHAPE.)

3.2.4 Discussion

The shape changes of the component were achieved under multiple loading scenarios – all loads applied in one solution run.

The inertia relief method was used in order to accommodate different boundary conditions for each load case.

3.2.5 Conclusion

The final component was estimated to be 15% lighter than the original component, after replacing the stiffeners. All of the required static analysis conditions for the design were met.

The final design was converted from a NASTRAN mesh to STEP 214 format, so that the design could easily be imported by the designer into a CAD package so that the changes could be implemented accurately on the original geometry.

References

[1] Beveridge G S G, Schlechter R S., *Optimization: Theory and Practice*, McGraw Hill, 1970.

[2] Tomas, J A, Optimum Design of a Gas Turbine Wheel for Some Ceramic Materials, *SAE*, Paper No. 760241, Detroit, 1976.

[3] Tomas J A, Trinder S M, Wille H C, Changing the Shape of Vehicle Components for Improved Performance, *Int. Seminar on Numerical Methods for Applied Science and Industry*, Polytecnico Torino, 1990.

[4] ReSHAPE Manuals and Case Studies, http://www.advea.com/

[5] Adams R, Tomas J, Bil C. Design Improvement of Components and Structures. *Engineering Mathematics and Applications Conference*, Melbourne, Australia, 2005.

Appendix

ReSHAPE program uses NASTRAN input file as input and creates NASTRAN file as output of the modified model.

The sensitivity and design improvement is control by *command files*. A command file contains specifications for objectives, constraints, side conditions, process selection and other control commands.

As an illustrative example to demonstrate the simplicity of the process, two command files are presented, both related to the first problem.

The *command file* for the ReSHAPE sensitivity analysis was

```
reshape(sensitivity)
  eset(active)=3
control
  subcase=5
responses
stress(target=min,var=vmmax,avgw=98%)=etype
(2d)
process()
  domain(xyz)=set(active)
end
```

The ReSHAPE command file used for design improvement was

```
reshape(improvement)
  eset(active)=3
  nset(cpoints)=4
control
  steps(number)=20
  subcase=5
responses
stress(target=min,var=vmmax,avgw=98%)=etype
(2d)
  disp(var=tt,bound=nolimit)=etype(2d)
process(function)
  domain(xyz)=set(active)
  fcontrol(wrt=xyz)=set(cpoints)
sideconditions
  rlimit(xl=-0.01,zu=0.01)=set(active)
end
```

The reader is referred to ReSHAPE Manuals [4] for a more detailed explanation of the commands and processes.