Design Improvement of Components and Structures

Ryan Adams[1], Christos Tsiangalis[1], Neil McLachlan[2], Josef A. Tomas[3]
[1] SAMME, RMIT University, Australia
[2]Australian Bell, Australia
[3]ADVEA Engineering, Australia

ABSTRACT

In recent years, various methods of optimisation based on finite-element methods of structural analysis have been developed. Some of these methods have been implemented in commercial computer programs, such as NASTRAN, OptiStruct, Tosca, and others.

Using some of these programs, however, requires the use of the complete package (suite) of programs, from pre- to post-processor, with analysis and optimisation modules in-between, all supplied by the program provider. For example, NASTRAN requires Patran to be used; OptiStruct requires HyperMesh; and similar with other programs. It is then difficult for an ANSA and μETA user to interface with the optimization modules of these programs.

The design improvement program ReSHAPE has been developed to be completely independent from any pre- and post-processor. It was then an easy task to couple ReSHAPE with ANSA and μETA into one solution process, as required by one customer.

This paper first explains the basics of two design improvement methods and then demonstrates the iterative improvement process on three selected problems using ANSA, ReSHAPE and μETA. The first problem is a concept layout design of an aerospace component to reduce weight, the second problem shows retuning a car component for noise reduction, and the last example demonstrates the improvement of a weapon system.

keywords: shape optimization, sensitivity, design improvement, topology

1. INTRODUCTION

The finite element method (FEM) is a tool that has revolutionized the engineering world, giving engineers the ability to model engineering problems from simple truss structures, to complex and unintuitive problems like electromotive force and magnetic fields.

Let's consider a structural component that is designed to handle certain loads, then analysed using FEM, and finally manufactured to become part of a prototype for testing purposes. Following the prototype testing, the component is targeted for performance improvements such as stiffness increase and fatigue performance enhancements, or weight reduction, and similar. 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.

Mathematical methods of optimization (see for example, [1]) have been commonplace in the engineering workplace for a long time, but due to some difficult issues when applying such methods to the solution of discrete FE models, such methods have been slow to escape the research environment, and enter the commercial industry.

Two types of FE optimization software products are offered in the market today:

(a) Software that uses “brute force” by running massive numbers of the same FE model, altering a small number of variables, and then searching through all the results with one of the direct search methods [2], or with Multiple Objective Genetic Algorithms (MOGA) or other such methods [3], looking for a solution to the problem. This has the advantage that it can be applied to both static and dynamic FE problems, linear and non-linear, but can take days of computations on large and expensive hardware, to solve a problem with only a small number of design variables.

(b) Software using mathematical strategies to improve the performance of a component by altering its configuration or shape iteratively. Most of these methods is suitable only for solving linear static stress/buckling/vibration type problems, but it is always possible to reduce a structural design problem to an equivalent linear problem (e.g. fatigue as stress, buckling as an eigenvalue problem, and similar). Then an experienced FE analyst is capable of improving an existing design by use of these iterative methods, and, if the problem is non-linear, to check the resulting design by non-linear analysis.

In this paper, the mathematical methods of structural design optimization will be discussed. This will include the initial design of components using the so-called ‘topology’ optimization, as well as shape improvement methods for design improvement of existing components.

2. SENSITIVITY TO DESIGN CHANGES

2.1. FINITE ELEMENTS

Even if a real life component is discrete at its molecular level, for mathematical analysis, it has to be considered as a continuum. On the other hand, a digital computer is discrete (in contrast with analog computers used un the past), therefore the mathematical continuum has to be discretised into a finite number of elements, each element defined by its ‘nodes’ (i.e. co-ordinates in space all over the structure) and by material properties. The nodes and elements, which form the ‘mesh’ over the structure, are usually created based on some CAD geometry.

In the general case, for each node there are 3 coordinates in space (x, y and z).

2.2. DESIGN SENSITIVITY [4]

In a static design problem, a scalar response quantity π is defined (stress, frequency, displacement, for example), to be used either as an objective or as a constraint. The response π is a function of nodal co-ordinates $\boldsymbol{x}$, and the displacement vector $\boldsymbol{u}(\boldsymbol{x})$:

$$\pi = \pi\left(\mathbf{x}, \mathbf{u}(\mathbf{x})\right). \qquad \textit{Equation 1}$$

If a node in the structure is moved, it will have an effect on the response π - this is called the *sensitivity* of the node. The sensitivity is calculated for each nodal co-ordinate in the model, in order to determine which nodes have the most effect on the response if perturbed:

$$\frac{d\pi}{d\mathbf{x}} = \frac{\partial\pi}{\partial\mathbf{x}} + \frac{\partial\pi}{\partial\mathbf{u}} \cdot \frac{d\mathbf{u}}{d\mathbf{x}}, \qquad \textit{Equation 2}$$

where $\frac{d\mathbf{u}}{d\mathbf{x}}$ is calculated from the finite element equation

$$\mathbf{K}.\mathbf{u} = \mathbf{f}\ , \qquad \textit{Equation 3}$$

where **K** is the stiffness matrix and **f** is the vector of loads.

Differentiating with respect to the nodal displacements yields:

$$\mathbf{K} \cdot \frac{d\mathbf{u}}{d\mathbf{x}} = \frac{d\mathbf{f}}{d\mathbf{x}} - \frac{d\mathbf{K}}{d\mathbf{x}} \cdot \mathbf{u}\,. \qquad \textit{Equation 4}$$

Due to the large number of right-hand sides to this equation for shape change (one for each node), this is best solved using the adjoint-variable method. $\frac{d\mathbf{u}}{d\mathbf{x}}$ in Equation 2 is replaced by rearrangement and substitution of Equation 4, resulting in

$$\frac{d\pi}{d\mathbf{x}} = \frac{\partial\pi}{\partial\mathbf{x}} + \frac{\partial\pi}{\partial\mathbf{u}} \cdot \mathbf{K}^{-1}\left(\frac{d\mathbf{f}}{d\mathbf{x}} - \frac{d\mathbf{K}}{d\mathbf{x}} \cdot \mathbf{u}\right). \qquad \textit{Equation 5}$$

The $\partial\pi / \partial\mathbf{u} \cdot \mathbf{K}^{-1}$ vector can be solved by the solution of the system (so that the inverse of the very large sparse positive definite symmetric matrix ***K*** need not be calculated)

$$\mathbf{K} \cdot \left(\frac{\partial\pi}{\partial\mathbf{u}} \cdot \mathbf{K}^{-1}\right) = \left(\frac{\partial\pi}{\partial\mathbf{u}}\right)^{T}, \qquad \textit{Equation 6}$$

where $\frac{\partial\pi}{\partial\mathbf{x}}$ and $\frac{\partial\pi}{\partial\mathbf{u}}$ are easily calculated. $\frac{d\mathbf{K}}{d\mathbf{x}}$ can also be calculated analytically with algorithms that have been derived for commonly used elements such as bars, shells and solids.

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

2.3. FREQUENCY SENSITIVITY

As an example, static modal vibration frequency (eigenvalue) sensitivities are calculated in the following way. The equation for solving the eigenvalue problem is

$$(\mathbf{K} - \lambda \cdot \mathbf{M}) \cdot \mathbf{u} = 0\,. \qquad \textit{Equation 7}$$

where ***K*** is the stiffness matrix, λ is the eigenvalue, ***M*** is the mass matrix, and ***u*** is the eigenvector of the model – all functions of the geometry, ***x***.

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

$$\mathbf{u}^{\mathbf{T}} \cdot \mathbf{M} \cdot \mathbf{u} = \mathbf{I}\,. \qquad \textit{Equation 8}$$

Then Equation 7 is multiplied through with $\boldsymbol{u}^{T}$, and then differentiated with respect to the geometry nodal coordinates, ***x***, so that

$$\mathbf{u}^{\mathbf{T}}\left(\frac{d\mathbf{K}}{d\mathbf{x}}-\frac{d\lambda}{d\mathbf{x}}\cdot\mathbf{M}-\lambda\cdot\frac{d\mathbf{M}}{d\mathbf{x}}\right)\cdot\mathbf{u}=0\,, \qquad \textit{Equation 9}$$

and , finally, the sensitivity can be calculated as

$$\frac{d\lambda}{d\mathbf{x}}=\mathbf{u}^{\mathbf{T}}\cdot\left(\frac{d\mathbf{K}}{d\mathbf{x}}-\lambda\frac{d\mathbf{M}}{d\mathbf{x}}\right)\cdot\mathbf{u}\,. \qquad \textit{Equation 10}$$

3. TOPOLOGICAL CONCEPT DESIGN

3.1. OVERVIEW

One optimization method that (because if it's relative simplicity) has been quick to leave the research departments and make an impact in the commercial industry is *Evolutionary Structural Optimisation (ESO)* or *Topology Optimisation*.

Topology is used in the concept design phase. In this phase, the engineer knows the packaging constraints of a new component to be designed (i.e. how much space the new design can take up), and the boundary conditions (i.e. the loads acting on the component, and any stiffness/vibration constraints, etc). The FE analyst creates a finite element model of the entire design space in full, with the known loads applied to the design space. The topology optimization then calculates which elements in the design space contribute the least to the work done by the component, and removes these elements from the model. The result is a concept shape of the new component, which is then transferred to the CAD engineer who creates a prototype CAD model.

A variety of methods have been developed for solving the topology problem, such as *Evolutionary Moment of Inertia Optimisation* or the *Homogenisation Method*, but the method that is featured in most commercial FE optimisation packages uses the process known as the *Density Method*.

In the *Density Method*, the volume and stiffness matrix of each element are multiplied by a factor, which varies between zero and one. Thus the element becomes more or less dense, depending on the sensitivity calculation. In a typical commercial package, the multiplier is usually considered as a density multiplier. Ultimately the multiplier becomes equal to either zero or one; i.e. the final model to be a real FE result, because each element has either full density, or zero density.

In ReSHAPE [5], the sensitivity of each element, for the given objective and constraints, is calculated in a similar fashion to the sensitivity calculation presented in chapter 2 of this paper, only the vector **x** relates to the element multiplying factors.

Several practical problems are faced when applying the topology procedure. For example, the final concept FE model produced is not a real FE model at all. The initial mesh has been chipped away at by the program, resulting in a very jagged edge mesh, which has high stress concentrations. For this reason, topology cannot be used with stress constraints/objectives.

The topology process also has the tendency to produce *checker boarding* in FE models with linear elements. This is the name given to the phenomena evident when linear elastic elements are only connected by adjacent elements with shared corner nodes. Although technically correct mathematically, this represents a false stiffness, and a nonmanufacturability of the structure. Methods have been developed to control this phenomenon [6].

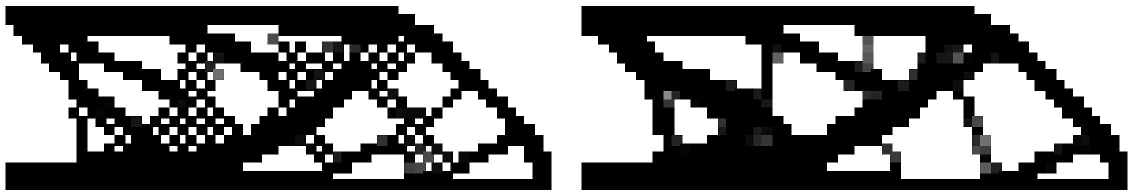

Figure 1 Checker boarding and checker board control models of a cantilever beam

Topology can also produce differing results for the same loading/boundary conditions depending on the mesh density of the model. A very finely meshed model often produces a much more acceptable engineering design, but can take a substantially longer amount of time to process, as expected. It is best to select as fine a mesh as processing time constraints allow. ReSHAPE actually removes elements from the stiffness matrix when the pseudo-multiplier falls below a certain level, so that each iteration becomes faster and faster as the process continues.

Despite these issues, the topology method produces very good concept suggestions which are invaluable for a design engineer. The topology result is a quick way for a design engineer to get an idea about the best way to design a component for a given set of loads and package space.

3.2. TOPOLOGICAL CONCEPT DESIGN APPLICATIONS

A wing spar section was conceptually designed to obtain an idea of an efficient shape for the load paths across the spar.

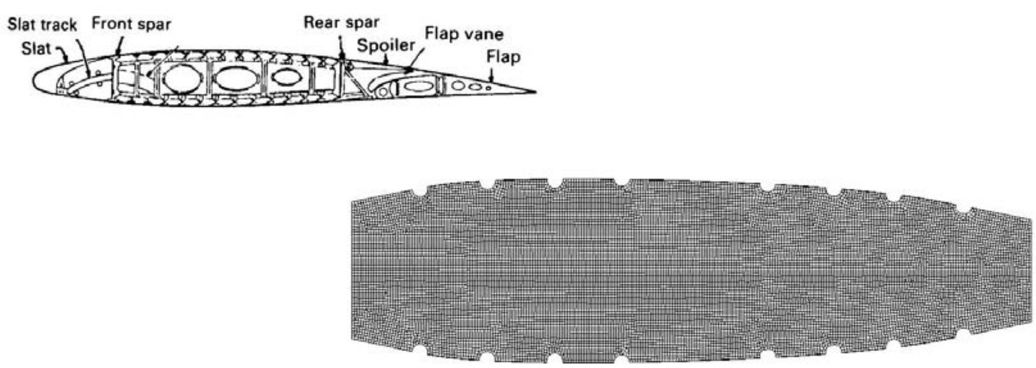

Figure 2 Wing spar section, showing FE model in ANSA

The wing spar was modeled with 16,859 linear shell elements (CQUAD4), with a constant thickness of 5mm. There is a bending load applied at the rear spar, and the model is constrained in all degrees of freedom at the front spar.

The objective of the topology procedure is to reduce the volume of the structure, but to constrain the displacement of the loaded nodes within a tolerance of 5%.

The results of the topological design process, using the software ANSA and ReSHAPE are depicted in Figures 3 to 5.

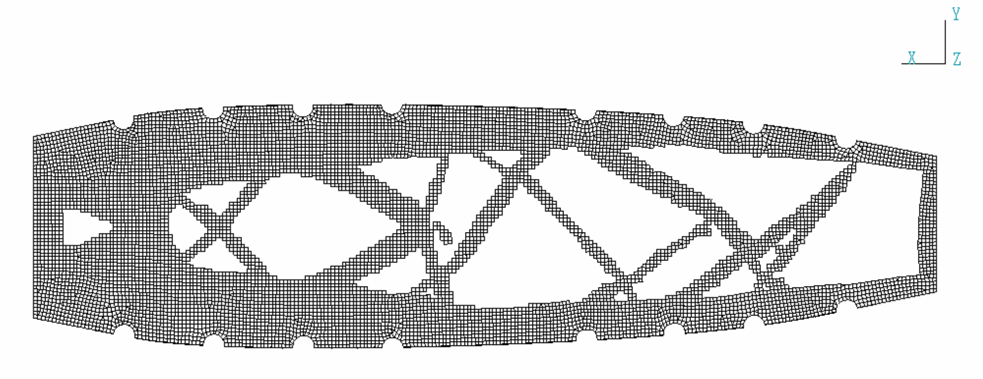

Figure 3 ANSA screenshot after 5862(35%) elements removed

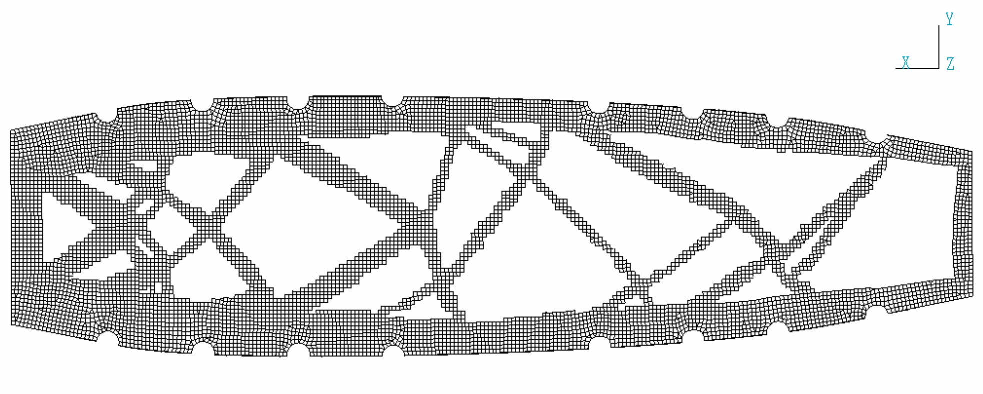

Figure 4 ANSA screenshot after 7299(43%) elements removed

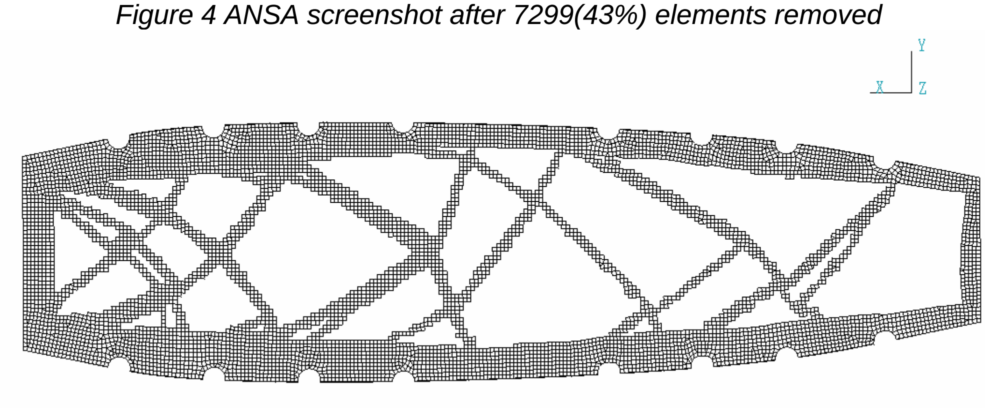

Figure 5 ANSA screenshot after 8094(48%) elements removed

In an industry situation, the final result is used by the CAD designer to help decide where holes need to be placed to allow objects such as hydraulic lines, and electrical wires to pass through.

4. RESHAPING OBJECTS

4.1. OVERVIEW

Compared to the relatively simple topology process, reshaping an existing component is a much more complex problem. For example, the resulting improved shape must be deemed satisfactory, i.e, it must be smooth, or have specific characteristic shapes in certain regions, which is difficult to achieve with only minimal input from the engineer.

So far nodal sensitivity for a certain objective has been discussed in chapter 2. This sensitivity can be applied directly to FE models, by simply stepping in the direction defined by the sensitivity vector for each node, and thus creating a new shape with improved performance. A new shape created with the direct application of raw sensitivities is often highly irregular, so this method is only suitable for improving models made entirely of beams and truss, '1D', types of elements.

Shells and solid meshes, however, are created by meshing CAD geometry, yet all information about the surfaces that the mesh was created from is removed. For example when using NASTRAN for the improvement process, there are no geometry information data in the NASTRAN input deck file.

So, in order to begin improving the shape of the component, the program must create some mesh to geometry association by introducing generalized coordinates, $\boldsymbol{q}$:

$$\mathbf{x} = \mathbf{x}(\mathbf{q}). \qquad \textit{Equation 11}$$

The nodal sensitivity of the finite-element mesh $\boldsymbol{x}$ can then be recalculated in the generalized coordinates $\boldsymbol{q}$:

$$\frac{d\pi}{d\mathbf{q}} = \frac{\partial \pi}{\partial \mathbf{x}} \cdot \frac{d\mathbf{x}}{d\mathbf{q}} \;. \qquad \textit{Equation 12}$$

Now the change in the generalized coordinates (to minimize or maximize the objective) can be related to the generalized sensitivity as

$$\Delta\mathbf{q} \approx \frac{\left(\dfrac{d\pi}{d\mathbf{q}}\right)^T}{\left\|\dfrac{d\pi}{d\mathbf{q}}\right\|} \;. \qquad \textit{Equation 13}$$

Finally the actual change of the mesh, $\Delta\mathbf{x}$, can be calculated from the change in generalized coordinates as

$$\Delta\mathbf{x} \approx \left(\frac{d\mathbf{x}}{d\mathbf{q}}\right)\Delta\mathbf{q} \;. \qquad \textit{Equation 14}$$

In order to satisfy the constraint, the change in generalized coordinates must be equivalent to an average change in the finite-element mesh. This is accomplished by orthonormalising the $\dfrac{d\mathbf{x}}{d\mathbf{q}}$ matrix by

$$\left(\frac{d\mathbf{x}}{d\mathbf{q}}\right)^T \left(\frac{d\mathbf{x}}{d\mathbf{q}}\right) = \mathbf{I}\,. \qquad \textit{Equation 15}$$

The $\frac{d\mathbf{x}}{d\mathbf{q}}$ matrix can be created in many different ways. The most popular methods are: creating basis vectors (several allowable shape changes created by the engineer); using non-uniform rational b-spline surface coefficients for shape improvement of shell based components, applying virtual displacement fields emanating from selected locations on the component, and similar.

4.2. NOISE, VIBRATION AND HARSHNESS APPLICATION

The differential gear teeth meshing excited the vibrations of a differential case, generating noise deemed by the design engineers to be irritating to the occupants of the vehicle. The aim of the design improvement was to increase the 2nd natural frequency of the case, moving it away from the resonant frequency, and hence reducing the noise of the component. The finite-element model of the case is shown in Figure 6. The model consists of 240,809 tetrahedral solid elements, 3240 shell elements plus several bar and rigid elements. The size of the **x** vector is 216,861.

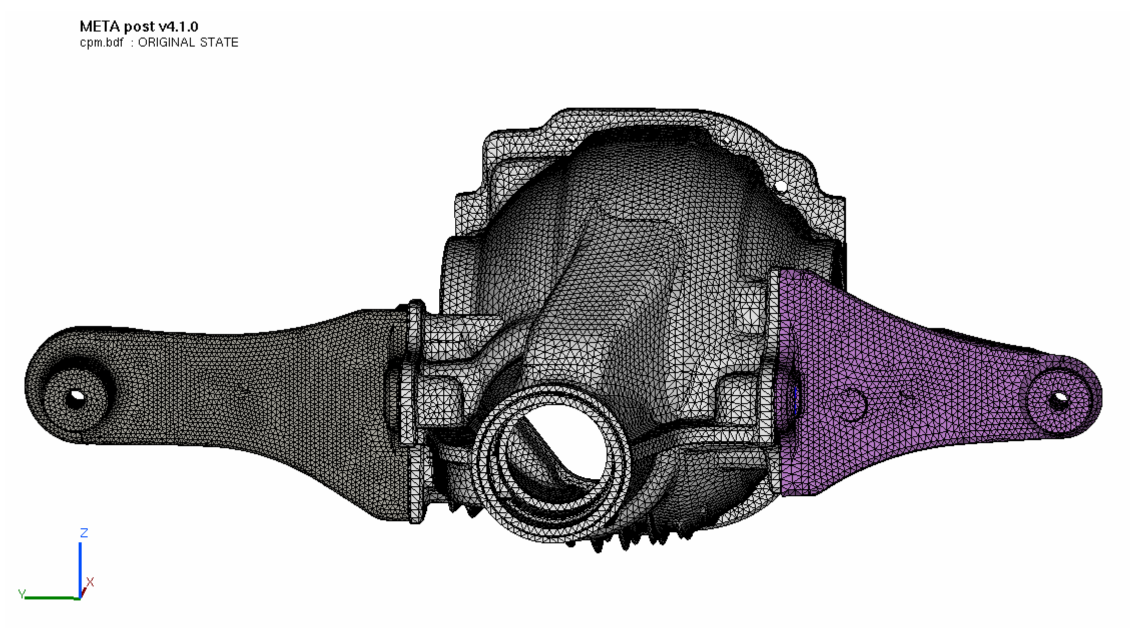

Figure 6 Differential case and bracket (Courtesy DANA Australia)

As a rule, the redesign began with a sensitivity analysis with respect to the design objective. The sensitivity analysis clearly showed the areas that affected the second modal frequency the most. The active domain for reshaping the component was selected in one of these regions, shown on the left section of the model in.

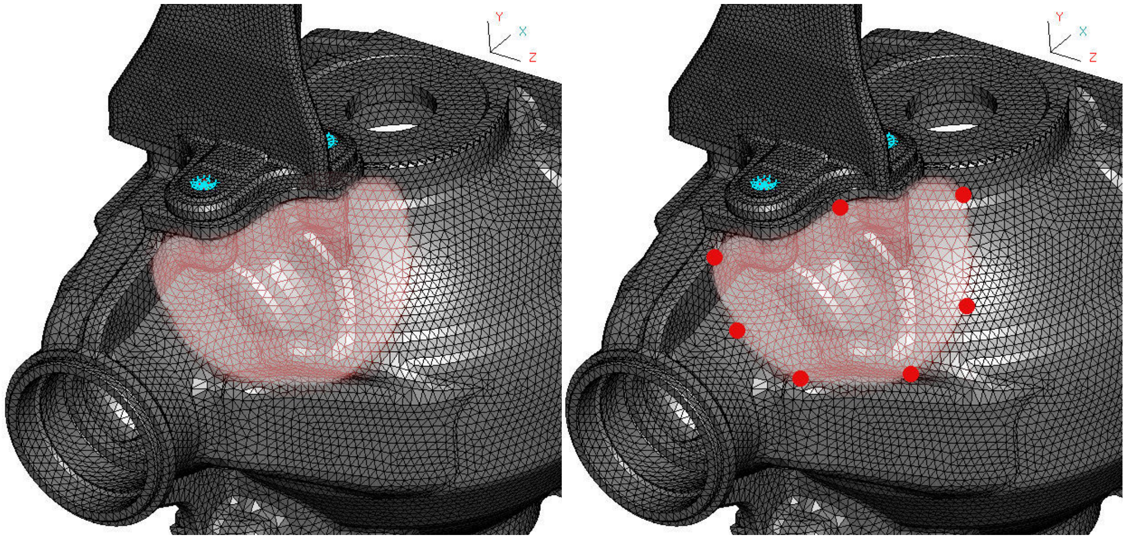

Figure 7 Active domain for reshaping the component, and the selected control points

The area was then reshaped using influence functions. Reshaping this section alone led to an increase in the second modal frequency by 30Hz.

After selecting several sensitive regions on the differential case, and subsequently reshaping these regions, the 2nd natural frequency was increased by 50 Hz. This was sufficient enough to reduce the noise to a suitable level. After each stage of design improvement, the modified model was discussed with designers, who inspected the proposed changes and suggested some adjustments so that the final design could be suitable for manufacturing and assembly. The final model is shown in Figure 4.

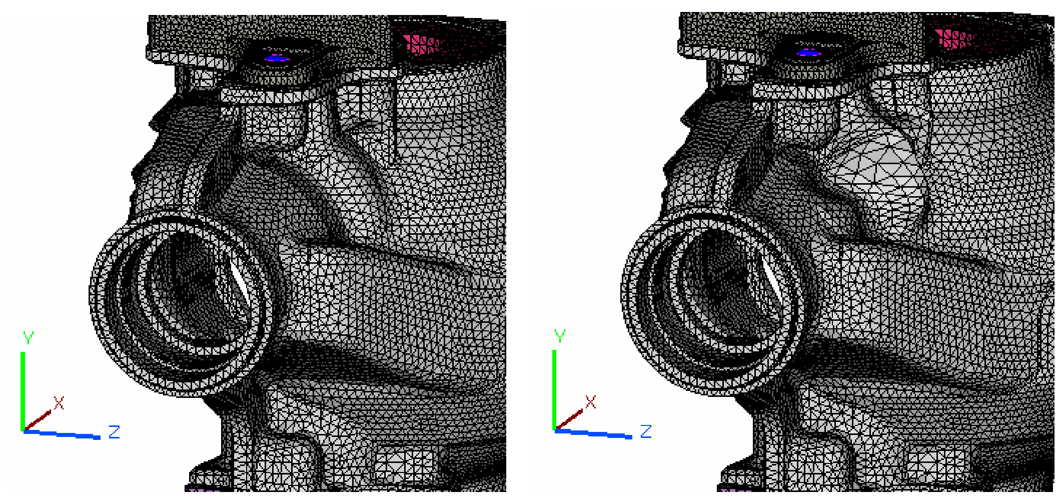

Figure 8 Before (left) and after (right) reshaping the first selected region
(Courtesy of DANA Australia)

The model was then handed over to the CAD designers, who used the results as a guide to add mass to the correct area of the case.

It is an interesting side note that this problem had existed for some time. Engineers had attempted to use their intuitive ‘feel’ to solve the problem by placing stiffening ribs on the structure, but this had nowhere near the level of effect as the careful mass placement result created by ReSHAPE.

4.3. APPLICATION TO THE DESIGN IMPROVEMENT OF WEAPONS SYSTEMS

The presented techniques for shape change have many varied applications. One application where the software ReSHAPE has been used is in a defence project by RMIT University in Melbourne, Australia for an improvement of a modern cannon system [7]. For confidentiality reasons, the method is demonstrated on the improvement of a 16th century cannon.
The cannon selected was a Spanish 42-pounder cannon. The behaviour of interest of the cannon was the transverse modes of vibration (see Figure 9). It was found that the accuracy of the cannon was highly affected by the displacement of the cannon after the explosion, excited by these particular vibration modes.

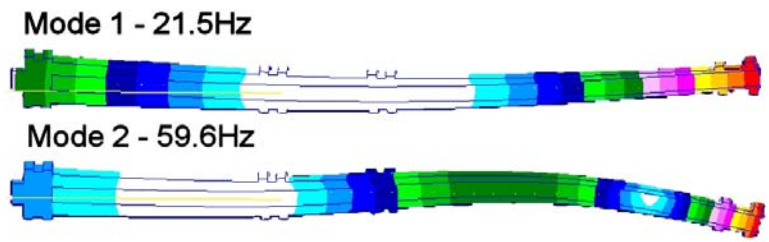

Figure 9 Displacement contours for transverse modes of vibration of the 16th century Spanish cannon

The objective was to get the first mode to vibrate with a phase angle of 180^o, while making sure that the stress levels did not exceed yield of the material. It was calculated that this would require a vibration frequency of approximately 40 Hz.

The shape was allowed to change on the external surface ot the cannon in radial direction only, so that the barrel still maintained a lathed shape, as shown in Figure 10.

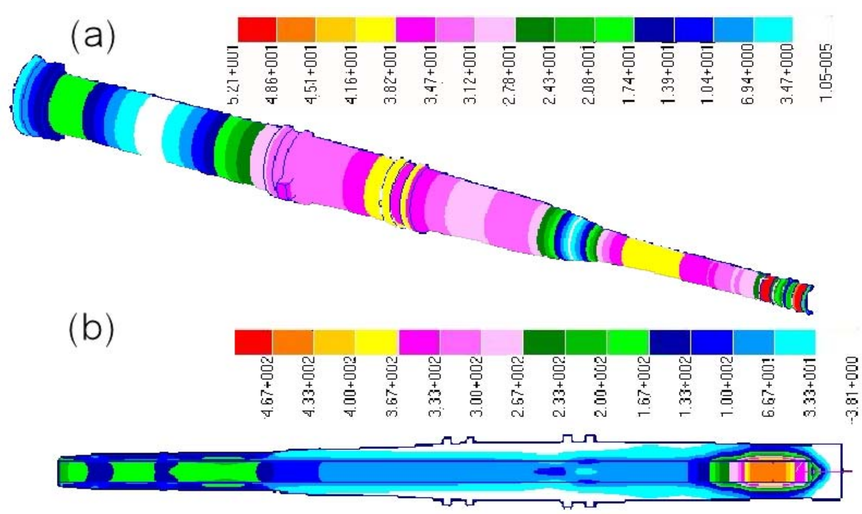

Figure 10 (a) shape change contours to tune transverse modes, (b) stress contours of improved shape

Further optimization runs on the cannon resulted in a shape very similar to modern artillery barrels. The conclusions of the feasibility study were that Spanish engineers used liberal manufacturing tolerances to account for the low quality materials of the time.

6. CONCLUSIONS

Finite-element methods (FEM) have progressed to the stage where that to use them solely for the analysis of designs is no longer acceptable. The applications presented in this paper demonstrate the power of using these methods; available to all practical engineers today.

Commercial software has reached the level where it could become an everyday tool of any CAE analyst for concept design and design improvement, but always with close cooperation CAD designers.

ACKNOWLEDGEMENTS

The support by RMIT, provided to the first author of this paper in his endeavour as an M.Eng. candidate, is appreciated; in particular Cees Bil from the Wackett Centre at the School of Aerospace, Mechanical and Manufacturing Engineering.

Further thanks go to Advea Engineering for providing ReSHAPE software and to Australian Bell and DANA ASG for allowing some results of their work to be presented in this paper.

REFERENCES

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

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

[3] Harries, S.; Valdenazzi, F.; Abt, C.; Viviani, U. 2001. *Investigation on Optimization Strategies for the Hydrodynamic Design of Fast Ferries*, 6th International Conference on Fast Sea Transportation, Southampton.

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

[5] ReSHAPE Manuals and Case Studies, www.advea.com/reshape/

[6] http://www.topopt.dtu.dk/Theory1/afs5.html, *Numerical Problems with the Topology Method*, accessed on 15th May 2005

[7] Tsiangalis, C., McLachlan, N., and Trivailo, P. 2005, *Virtual Design Processes for Weapon Systems*, Eleventh Australian International Aerospace Congress, Melbourne, Australia.