1998 MSC Americas Users' Conference

1. Paper Title: **HANDLING OF WELDS IN SHAPE VECTORS GENERATION FOR FINITE ELEMENT SHAPE OPTIMIZATION - A CASE STUDY**

2. Name: **Murali M.R. Krishna**
3. Title: Project Engineer, CAE Services
4. Company: DANA Corporation
5. Address: 8000 Yankee Road, Ottawa Lake, MI-49267, USA
6. Phone Numbers: (419)-535-4322 [Fax # (419)-535-4276]
7. E-mail: murali.krishna@dana.com

8. Abstract:

ABSTRACT

Design analysts, who work with finite element shape optimization, face a daunting task of handling welds. When the designer wants to find the optimum width of the leg of a bracket, which is welded to a base rail, the analyst has to remove the old welds, remodel and re-create new welds after extension of the bracket, and iterate. This method is not suitable for shape optimization. A numerical interpolation method based on 'Autodv', has been recommended to handle welds without remodeling. This method is very effective for finite element shape optimization. A case study has been given to illustrate the method using MSC/NASTRAN.

HANDLING OF WELDS IN SHAPE VECTORS GENERATION FOR FINITE ELEMENT SHAPE OPTIMIZATION - A CASE STUDY

Murali M.R. Krishna
Dana Corporation

INTRODUCTION

In the design of brackets mounted on chassis frames welds pose a big problem to design analysts, who like to recommend design changes based on finite element (FE) shape optimization. Normally the mounting brackets are welded on one or more of its edges to a rail. When the designer wants to find the optimum width of the leg of the bracket the design analyst is faced with a daunting task of removing the old welds from the finite element model and creating new ones after extending the bracket. He would normally try out a couple of designs with different widths and on a trial and error basis recommend a design. The task becomes all the more complicated when two or more legs of the bracket need to be handled simultaneously. This method is however not suitable for FE shape optimization.

In shape optimization one has to create shape vectors (also known as basis vectors) which provide a method of changing the design in the process of finding an optimum solution. New finite elements cannot be added at any time during the optimization process and they have to be handled as an entirely new problem. In the present paper the author presents a method where the analyst could proceed with an existing finite element mesh and still handle extensions and weld relocations without remeshing. The method involves moving the nodes of both the mounting bracket and rail in unison so that the mesh does not get distorted and at the same time brackets are extended and welds relocated. A preprocessor called 'Autodv' [1] developed and marketed by Altair Computing, MI, has been used for moving the nodes to achieve the design targets. A case study has been presented to illustrate this method.

MODEL DESCRIPTION

The finite element model of a bracket mounted on a chassis rail is shown in Figure 1. The chassis rail is completely fixed (all 6 dof) on all four edges at both ends and a load of 1000 N is applied in z-direction at the top of the bracket. The bracket is symmetric in the x-plane at the center of the bracket. A preliminary MSC/NASTRAN stress analysis of the FE model indicates a high stress of 147 N/mm^2 as shown in Figure 4. This high stress has to be brought down to around 80 N/mm^2. Since we have a point load, the stresses around the area of the load application is bound to be high and hence it is not taken into consideration as shown in all the stress plots. Sizing and shape optimization techniques were used for this analysis. The following design possibilities were considered -

1. The thickness of the bracket can be varied between 2.0 and 3.5 mm.
2. The vertical flanges on both sides of the bracket can be extended down but maintaining the weld joints between the bracket and rail along the outside edge.
3. The width of the vertical flanges on both sides of the bracket can be widened in the x-direction but at the same time maintaining the weld joints along the outside edge.
4. The vertical lip height of the bracket can be increased maintaining the smooth radius at the side where it joins the vertical flange.
5. The symmetry of the bracket about the x-plane at the center is to be maintained during all the above design changes.

Shape variations required for MSC/NASTRAN shape optimization [2], were generated using 'Autodv'. While the model is subjected to these shape variations, it has to be ensured that the finite elements do not get distorted.

MODELING OF WELDS

The method of generating the shape vector for item #3 above is illustrated as an example. Two solid *'Domain Elements'* are defined enclosing the bracket (Figure 3), such that their junction nodes fall in the plane of the weld under consideration. The rear half of the rail has not been shown for clarity. The nodes associated with each of the domain elements are the ones, which fall inside the domain volume. The set of nodes, associated with the x-plane of node N2 (with welds), is common to both the domain elements. *'Plotel elements'* [2] are defined at the each of the four junction nodes of the two domain elements. These plotel elements define the perturbations in one direction only, at the end nodes. The set of nodes, associated with x-plane of nodes N1 and N3, has zero perturbation. Now using linear interpolating functions the perturbations at all the other domain nodes, are calculated by 'Autodv'. The shape variation is the maximum perturbation at each of the nodes written to a file as DVGRID [1] cards (Design Variable Grid Cards of MSC/NASTRAN). Welds are normally modeled as rigid elements (RBE2 Nastran Elements [2]). Since the nodes associated with welds, also fall in the domain they are also perturbed proportionately. Hence all the weld elements move in unison as the bracket extends. This ensures that the mesh on either sides of the welds shrinks or elongates proportionally and hence the quality of the mesh does not change appreciably. Secondly the rigid elements which are used to model the welds move on to the new position without any nodal mismatch. A similar technique has been used to get the other shape vectors for the other legs of the bracket.

ANALYSIS

The FE model is then subjected to combined shape and sizing optimization to reduce the stresses. The objective of the analysis was to minimize the weight of the structure and the stress constraints were imposed on the maximum of the Von mises stresses for the bracket to be limited to 80 N/mm^2. The shape vectors as described above were generated and used for optimization. The *direct linearization* method [2, 3] has been used for shape optimization and the design sensitivities have been generated internally in MSC/NASTRAN.

RESULTS

The combined shape and sizing optimization took 10 iterations and converged to a feasible solution. The final shape is shown in Figure 2 superposed over the baseline design. The stress plot of the final shape is also shown on the same scale in Figure 5. The following design directions have been obtained.

1. The thickness of the bracket needs to be increased from 2.0 mm to 3.0 mm.
2. The vertical flanges on both sides of the bracket need to be extended down by 6.4 mm.
3. The width of the vertical flanges on both sides of the bracket needs be widened in the x-direction by 6.2 mm.
4. The vertical lip height of the bracket needs to be decreased by 2.0 mm.
5. The symmetry of the bracket about the x-plane at the center has been maintained during all the above design changes.

The maximum Von-mises stresses have been reduced from 147 to 80 N/mm^2. It is observed that none of the design variables reached the extreme bounds provided by the designer. Hence it is concluded that the optimizer has converged to the best possible solution as shown in Figure 2. A separate finite element run with the new design confirmed the reduced stress values (Figure 5).

CONCLUSIONS

It is concluded that the weld attachments at the ends of brackets can be handled efficiently in finite element optimization, using the method of interpolation for generating the basis vectors. This eliminates the tedious cyclic task of manually re-meshing and re-analyzing the bracket leg for finding the optimum design. An example has been given for design of a bracket mounted on a chassis frame but it can also be used whereeever applicable.

ACKNOWLEDGEMENTS

The author gratefully acknowledges the help of Prakash Mhambrey, Parish Vehicles Structures Division, Dana Corporation for providing input data for the case study.

REFERENCES

1. " Structural Optimization" , Training Manual, Altair Computing Inc., Troy, MI 48084, 1994.
2. "Design Sensitivity and Optimization in MSC/NASTRAN", Seminar Notes, The MacNeal Schwendler Corporation, Los Angles, California, USA, March 1987.
3. "Elements of Structural Optimization", Haftka, R.T., and Kamat, M.P., Martinus Nijhoff, Dordrecht., 1985.

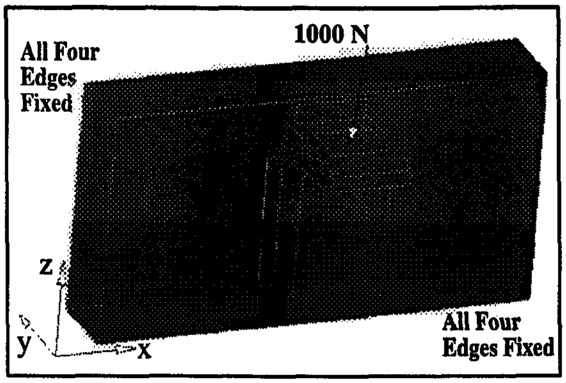

Figure 1: Baseline FE Model

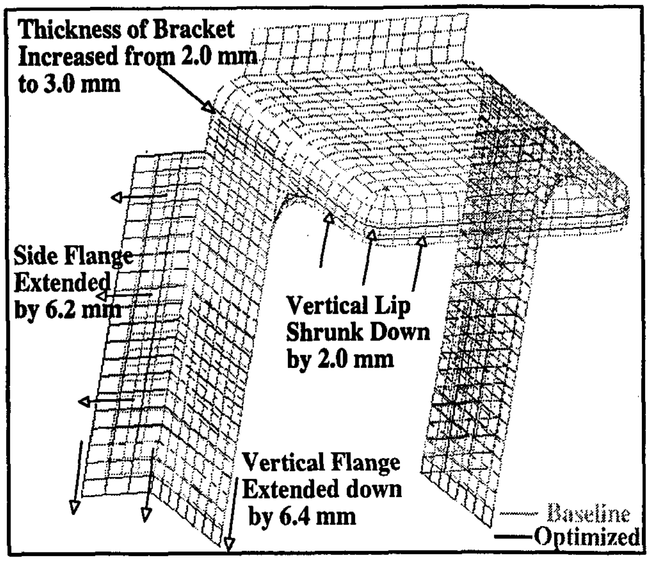

Figure 2: Optimized Model of Bracket

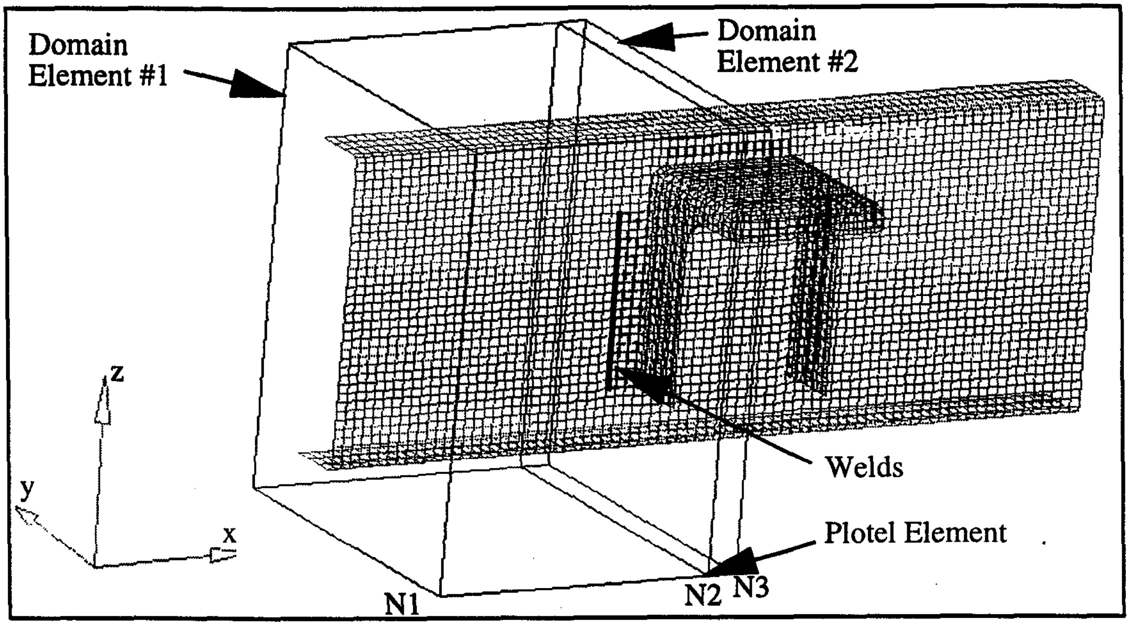

Figure 3: Model Showing Domain Elements

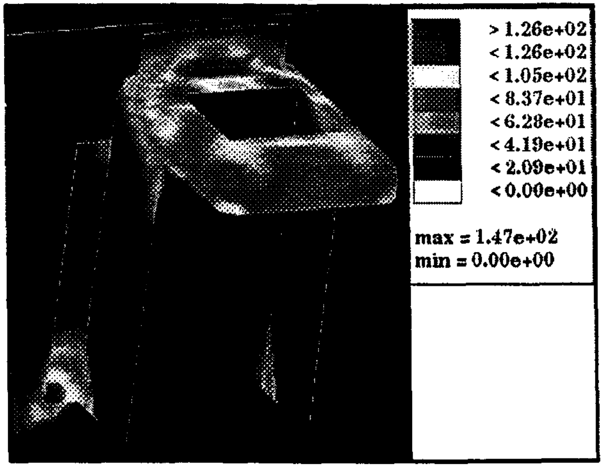

Figure 4: Von–Mises Stresses (Baseline Model)

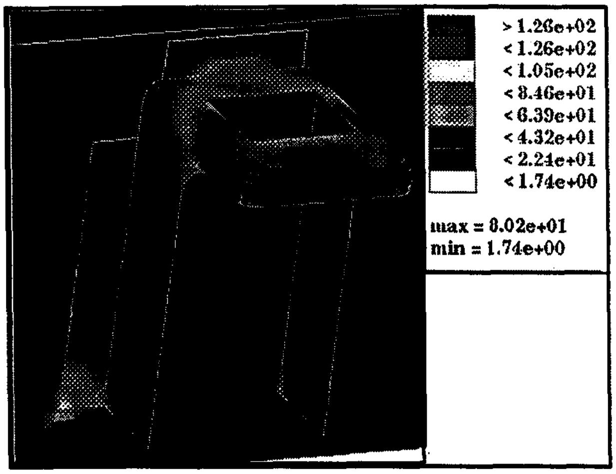

Figure 5: Von–Mises Stresses (Optimized Model)