International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:6, No:8, 2012

An Optimization Analysis on an Automotive Component with Fatigue Constraint Using HyperWorks Software for Environmental Sustainability

W. M. Wan Muhamad, E. Sujatmika, M.R. Idris, S.A. Syed Ahmad

Abstract**—**A finite element analysis (FEA) computer software HyperWorks is utilized in re-designing an automotive component to reduce its mass. Reduction of components mass contributes towards environmental sustainability by saving world's valuable metal resources and by reducing carbon emission through improved overall vehicle fuel efficiency. A shape optimization analysis was performed on a rear spindle component. Pre-processing and solving procedures were performed using HyperMesh and RADIOSS respectively. Shape variables were defined using HyperMorph. Then optimization solver OptiStruct was utilized with fatigue life set as a design constraint. Since Stress-Number of Cycle (S-N) theory deals with uni-axial stress, the Signed von Misses stress on the component was used for looking up damage on S-N curve, and Gerber criterion for mean stress corrections. The optimization analysis resulted in mass reduction of 24% of the original mass. The study proved that the adopted approach has high potential use for environmental sustainability.

Keywords—Environmental Sustainability, Shape Optimization, Fatigue, Rear Spindle

I. Introduction

ONE of product improvement objectives in an automotive manufacturing is to use less material by reducing weight of components. At the same time, reducing stresses in components is essential, as this can improve the product durability and reliability. To realize this objective, virtual product development using computer software is becoming more and more important. In virtual product development process, optimization and fatigue analyses are progressive methods. Several literatures discuss optimization of structural components based on fatigue [1]-[3].

When a component is subjected to fatigue or cyclic loading, a progressive and localized structural damage fatigue occurs. The time to failure is calculated from the history of fatigue damage, and a safe life is determined [4]. In strength and fatigue analysis, normally critical zones are sought for in the final stage of a structural component study.

W.M. Wan Muhamad is with Mechanical and Manufacturing Section, Universiti Kuala Lumpur Malaysia France Institute, Jalan Teras Jernang, 43650 Bandar Baru Bangi, Selangor, Malaysia. (phone: +603-8926-2610; fax: +603-89258845; e-mail: drwmansor@mfi.unikl.edu.my).

E. Sujatmika is with Institute of Research and Postgraduate Studies, Universiti Kuala Lumpur, 1016 Jalan Sultan Ismail, 50250 Kuala Lumpur (e-mail: endrafira@gmail.com)

M.R. Idris is with Institute of Product Design and Manufacturing, 119-Jalan 7/91, Taman Shamelin Perkasa, 56100 Kuala Lumpur, Malaysia. (email: mrazif@iprom.unikl.edu.my)

S. A. Syed Ahmad with Mechanical and Manufacturing Section, Universiti Kuala Lumpur Malaysia France Institute, Jalan Teras Jernang, 43650 Bandar Baru Bangi, Selangor, Malaysia (e-mail: syedazuan@mfi.unikl.edu.my).

However, the results of shape optimization, an optimization method used in this investigation, differ from those for durability. The main step of a shape optimization procedure is to detect where critical locations (with high damage values) can occur. Then the geometry of these local regions will be modified using optimization analysis.

In this study, the vehicle component chosen is rear spindle. It is part of a vehicle steering and suspension system. The shape optimization analysis is performed using HyperWorks software which contains several modules.

II. Objective and Aim

Objective of this study is to achieve weight or mass reduction of a rear spindle component under fatigue loading using shape optimization by the gradient based method, considering fatigue constraint. Reduction of component's mass will not only save world's valuable metal resources, but will also improve vehicle's fuel efficiency due to lesser energy requirement to move a lighter vehicle. Therefore, mass reduction will contribute towards environmental sustainability as well as saving costs.

III. Methodology

In applying a shape optimization based on fatigue analysis, different calculation-based procedures must be adopted. The component to be optimized is set as an elastic body. Finite element analysis is used to get basic data for the fatigue analysis. Since the objective of the optimization is to decrease damage in the system, the fatigue analysis and the optimization itself are needed [4]-[5].

CAD model is the basis of the whole process. It provides data of the mechanical component, in this paper, the rear spindle. This consists of coordinate systems, geometries, masses and mass inertias. This data is required to build the finite element. The solver RADIOSS [6] calculates the results of the so-called unit load cases for the fatigue stress $\sigma(x)$. The loads $L(t)$ and the stresses of the unit load cases $\sigma(x)$ are the input data for the fatigue analysis. The value x represents all nodes of the finite element mesh. In the fatigue analysis, the data are derived from the superposition of $L(t)$ and $\sigma(x)$, and the damage $D(x)$ of the component is calculated in each nodal position x. By changing the geometry, this damage $D(x)$ will be reduced during the optimization. With the new finite element mesh, the same sequence starts again.

Fig. 1 A short description of the process is given and illustrated in Fig 1.

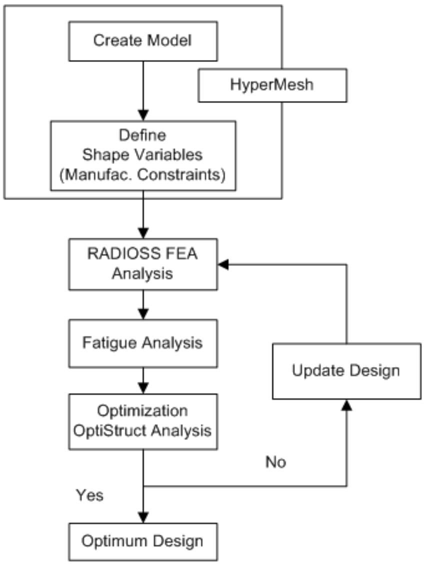


Fig. 1 Methodology flowchart

IV. Model Development and Underlying Theories

Finite element model for rear spindle is shown in Fig. 2. In actual test, the rear spindle is welded when it is subjected to the load. To represent this condition it is constrained from the back with six degree of freedom constraints.

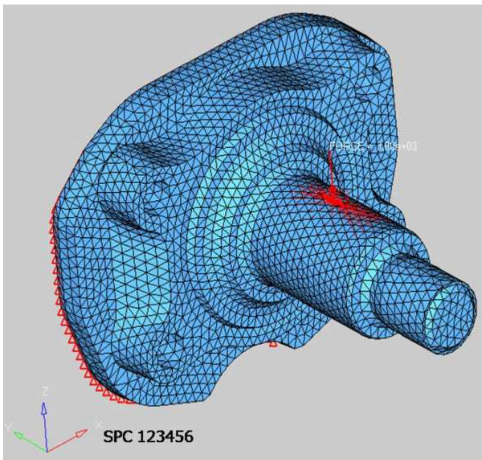


Fig. 2 Automotive rear spindle model

The applied material properties are presented in Table I.

TABLE 1
MATERIAL PROPERTIES

Material	S48C
Density	7.85e-6 kg/mm^3
Poisson's Ratio	0.3
Modulus of elasticity	200000 MPa
Yield Stress	578.32 MPa
Ultimate tensile stress	752.18 MPA

After simulation of the loads and calculation of the results of the unit load cases, all data are then prepared for the fatigue analysis. The first step of the fatigue analysis is the superposition, as shown in Fig. 2 [7]. The results are the stresses for every node of the finite element mesh over the time span $\sigma(x^k, t)$

$$\sigma(x^k, t) = \sum_{k=1}^{n_s} \sigma_k(x^k) \frac{L_k(t)}{L_k^0} \tag{1}$$

The quasi-static superposition uses stresses calculated with a static finite element analysis. The variable n_s describes the number of unit load cases and $L^0_{k,}$ the applied unit load.

In the stress time series, the number of load peaks and troughs are counted with the rain flow method [8]. The result is the rain flow matrix, which contains information that can be compared to an S-N curve. Since S-N theory deals with uni-axial stress, the Signed von Misses stress on the stress components was used to look up for damage on S-N curve. The S-N curve provides a relationship between the fatigue strength ('S' or σ) and the fatigue life ('N') and can be described with

$$\frac{N}{N_D} = \left[\frac{\sigma}{\sigma_D}\right]^k \tag{2}$$

Where N_D is the fatigue limit cycles, σ_D the endurance limit and k the slope of the S-N curve [11]. The Gerber criterion is used as mean stress corrections.

The damage D_i is given by the relationship between the cycle n and the fatigue limit cycle N at the same stress amplitude i

$$D_i = \frac{n_i}{N_i} \tag{3}$$

To obtain the damage for a load case with different stress amplitudes over the time, the linear damage accumulation by Pilmgreen-Miner can be used [6].

$$D = \sum_{i=1}^{n} \frac{n_i}{N_i} \tag{4}$$

A fatigue analysis based on finite element calculation calculates damages for every node of the finite element mesh. The result $D(x^k)$ can be used as an objective function in the shape optimization.

S-N curve for rear spindle material (S48C) is shown in Fig. 3. In this analysis a load case was applied from Load Time History of car at 48 km/h as shown in Fig. 4. Lower bound of fatigue Constraints Life is 300000 cycles, this value is based on the actual fatigue test.

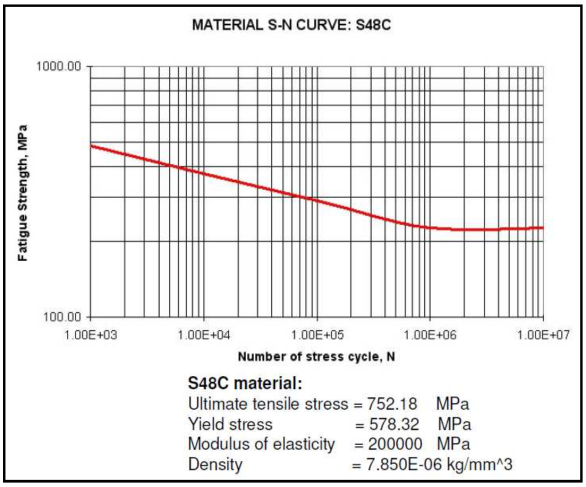


Fig. 3 S-N curve of rear spindle material (S48C)

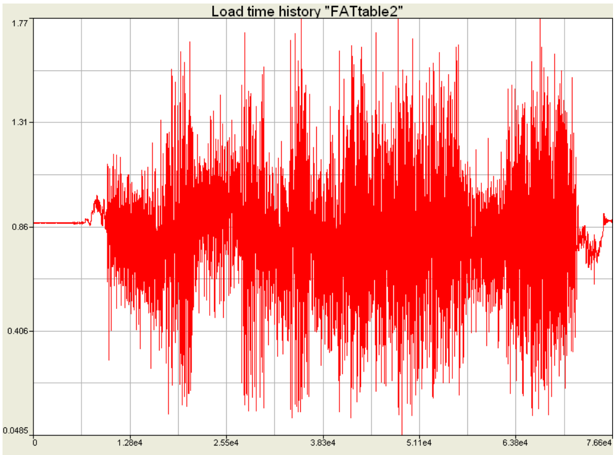


Fig. 4 Load time history at 48 km/h

Using a shape optimization based on finite element results, the nodal coordinates of the finite element mesh are the optimization variables. To reduce calculation time, a small location to be optimized should be chosen [9,10].

The objective function of a shape optimization based on fatigue analysis is the maximum damage in the design space. A constraint on the volume can be used. The easiest way is a reduction of the damage with the same material. Sometimes it is also possible to reduce the used volume. Other constraints such as a constant displacement between two points, a limited maximum stress in the whole structure, or a limitation of the design space, may also be possible.

Generally the optimization task can be given as

$$f^* = f(x^{k,d,*}) = \min\{f(x^{k,d}) \mid x^{k,d} \in X\} \tag{5}$$

with

$$X = \{x^{k,d} \in \Re^m \mid g(x^{k,d}) \le 0, h(x^{k,d}) = 0\}$$

The function $f(x^{k,d})$ is the objective function of the optimization and $x^{k,d}$ is the vector of the design variables. The stars mark the objective function and design variables at the optimum, which is formally a minimum. The range of the design variables is restricted by the functions $g(x^{k,d})$ and $h(x^{k,d})$, which can be equality and inequality constraints.

The design variables $x^{k,d}$ of both optimizations are the coordinates of the finite element nodes. For the shape optimization based on static stress analysis, the calculated maximum principal stress $\sigma 1(x^{k,d})$ for the design variables is the objective function. The maximum value of all design variables and of the load case with the maximum value is minimized. The shape optimization based on fatigue analysis minimizes the maximum damage $D(x^{k,d})$ of the design variables. A choice of a tensor component or load case is not necessary, because the fatigue analysis compresses all data to one relevant damage value for each node. The optimization restrictions are the same for both optimizations. The displacement Δu of the nodes $x^{k,r}$ is restricted to preserve functional surfaces of the housing. The volume V of the whole housing is constant. Some nodes of the finite element mesh near to the design variables can move in order to counter a mesh deformation. The optimization task for the optimization based on static stress analysis is

$$f^* = \min\{\max(\sigma_1(x^{k,d})) \mid x^{k,d} \in X\} \tag{6}$$

with

$$X = \{x^{k,d} \in \Re^m \mid \Delta u(x^{k,r}) = 0, V - V_{start} = 0\}$$

The optimization task for the optimization based on fatigue analysis is

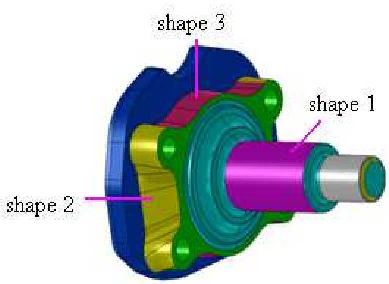
$$f^* = \min\{\max(D(x^{k,d})) \mid x^{k,d} \in X\} \tag{7}$$

with

$$X = \{x^{k,d} \in \Re^m \mid \Delta u(x^{k,r}) = 0, V - V_{start} = 0\}$$

The results in the shape optimization are for a particular location [11]. Therefore, a typical field of application is the reduction of local stress peaks in a component. Given that the fatigue analysis becomes more important in analyzing the geometry, it follows that the automated reduction of damages is also needed. To reduce the damage, the homogenization method is used.

Design variables were determined using HyperMorph. Seven design variables were defined as shape 1, shape 2, shape 3, shape 4, shape 5, shape 6, and shape 7 as shown in Fig. 5.



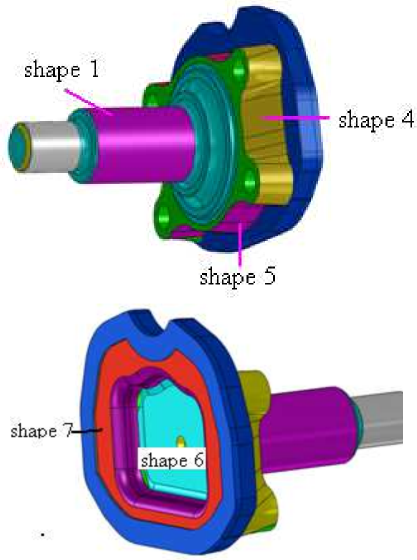
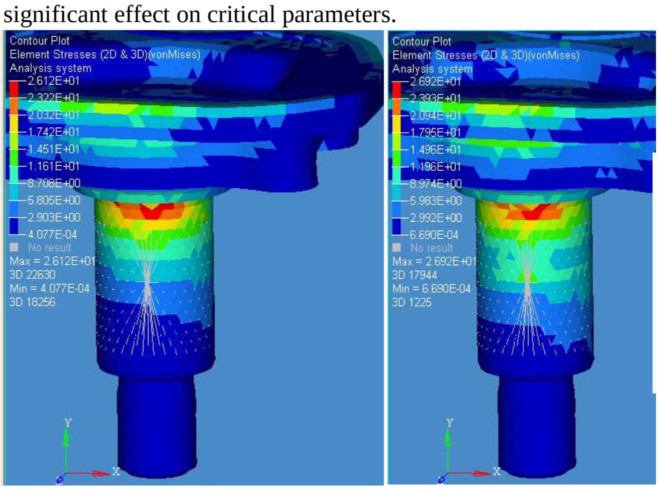


Fig. 5 Shape definition of rear spindle

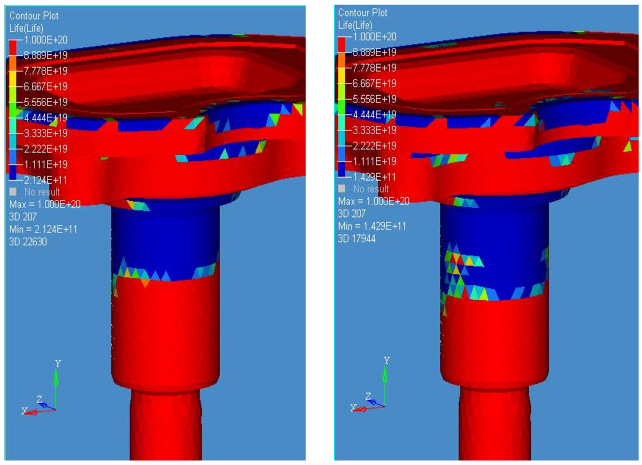
V. Results And Discussion

Critical parameters in a fatigue analysis are maximum von Misses stress and fatigue life of rear spindle. Fig. 6 and Fig. 7 show both of critical parameters contour plot for the first and last iteration, respectively. It shows that the maximum von Misses stress plot at last iteration is not much different than that of the first iteration. This is also true for minimum fatigue life contour. These indicate that shape changing doesn't have significant effect on critical parameters.



a) 1st iteration b) 4th iteration

Fig. 6 Von-misses contour plots



a) 1st iteration b) 4th iteration

Fig. 7 Fatigue life contour plots

For the optimization process, seven shape variables were defined and optimized by taking into consideration fatigue constraint.

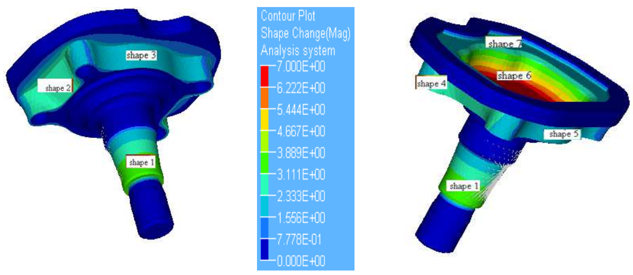


Fig. 8 shows the shape optimization results that were obtained using OptiStruct solver.

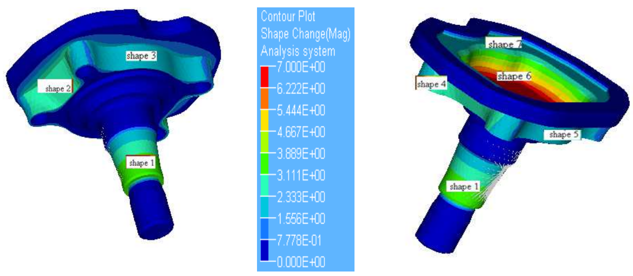


Fig. 8 Shape change contour plot

Shape 1 and 6 had the most reduction among all shapes. The limit of the shape change depends on the available design space. In the scale of 0 to 1 of shape changing, as shown in Fig. 9, most of the design variables meet upper bound (scale 1) of design variable value. It can be said that optimization process has met optimal design variable value.

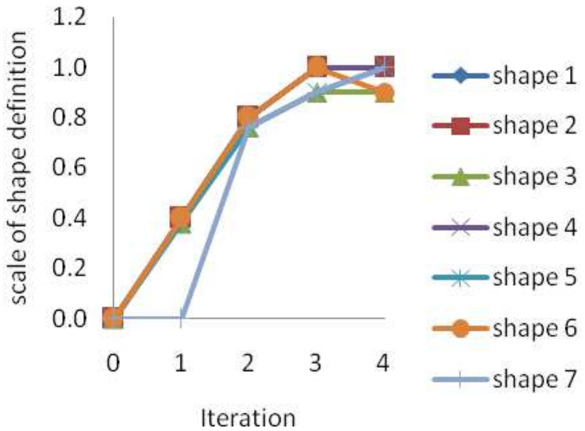


Fig. 9 Scale of shape changing

Fig. 10 shows the graph of mass versus the iteration of the shape optimization process. The mass reduction achieved was 24%.

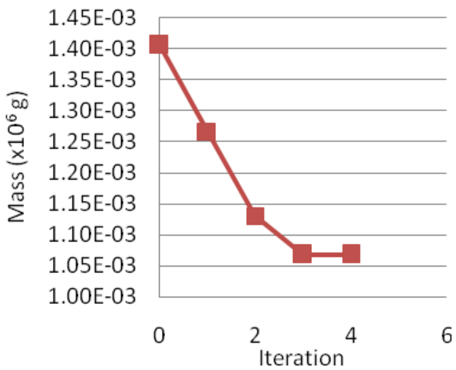


Fig. 10 Objective function

VI. Conclusion

Shape optimization is used in this study to reduce the mass of a rear spindle using HyperWorks software. The results of the optimization analysis within the given design space showed that the mass reduction achieved was 24% of the original mass. The approach adopted in this study is successful in achieving significant component mass reduction. It can be applied to other vehicle components and therefore has a potential to contribute towards environment sustainability by better conserving world's metal resources and reducing carbon emission through improved overall vehicle fuel efficiency.

Acknowledgment

We are sincerely grateful to the Ministry Of Science, Technology and Innovation Malaysia for financially supporting a research project which includes this study. We would like to thank Proton Berhad and Universiti Kuala Lumpur for their support.

References

[1] M. Zurofi, *Manufacturing Process Effects on Fatigue Design and Optimization of Automotive Components– An Analytical and Experimental Study*, The University of Toledo, Ph.D. thesis, 2004.

[2] K. Krishnapillai And R. Jones, "Fatigue based 3D structural design optimization implementing genetic algorithms and utilizing the generalized Frost-Dugdale crack growth," Proc. of the 9th WSEAS *Int. Conf. on Mathematical and Computational Methods in Science and Engineering*, Trinidad and Tobago, 2007.

[3] P. Chaperon, R. Jones, M. Heller, S. Pitt & F.Rose, "A methodology for structural optimization with damage tolerance constraints," *Engineering Failure Analysis*, vol.7, 2000, pp. 281-300.

[4] Y. Kojima, "Mechanical CAE in automotive design," *R & D review of Toyota CRLD*, Vol 35, No. 4, 2000.

[5] U. Schramm, H. Thomas, and M. Zhou, "Manufacturing considerations and structural optimization for automotive components," *SAE Technical Paper*, No. 2002-01- 1242, Society of Automotive Engineers, 2002.

[6] *Altair Hyperworks10*, Altair Engineering Inc., India, 2009.

[7] "Fatigue and fracture", *ASM Handbook*, Vol 19, ASM International, 1996.

[8] E. Zahavi, V. Torbilo, *Fatigue Design - Life Expectancy of Machine Parts*, CRC Press, 1996.

[9] M.P. Bendsoe, O. Sigmund, *Topology Optimization - Theory, Methods and Application*, Berlin,Heidelberg: Springer-Verlag, 2003.

[10] M. Bendsoe, N. Kikuchi, "Generating optimal topologies in structural design using a homogenization method", *Computer Methods in Applied Mechanics and Engineering*, Vol. 71, 1988, pp. 197–224.

[11] R.A. Richards, *Zeroth-Order Shape Optimization Utilizing A Learning Classifier System*, PhD Dissertation, Mechanical Engineering Department, Stanford University, 1995.