OPTIMUM SHAPE STRUCTURE UNDER COMPRESSION LOADING USING ADAPTIVE MESH FINITE ELEMENT

Abdul Rahim Ismail*, Ahmad Kamal Ariffin M. Ihsan and Zulkifli Mohd Nopiah**.**

**Faculty of Mechanical Engineering, University College of Engineering & Technology Malaysia, P.O. Box 12, 25000 Kuantan, Pahang.*

***Computational Mechanics Research Group, Department of Mechanical and Material Engineering, Universiti Kebangsaan Malaysia, 43600 Bangi, Selangor*

e-mail: abrahim@kuktem.edu.my

ABSTRACT: Study on shape optimization of plate structure is presented. Plate with known mechanical properties, load and constraint position is modeled using finite element method. Linear constant strain triangle element with three nodes adaptive mesh is used in finite element calculation, incorporating Delaunay technique. Shape optimization process using evolution method is used. The boundary elements which stress are less than threshold stress value are eliminated. Re-domain process for smoothing the domain boundary surface after element elimination process is carried out using vector method. These shape optimization and re-domain process are iteratively executed until they have fulfilled the termination criteria. The result obtained from simulation showed that there are area reductions from initial domain. Experimental study is performed using polariscope model MT 7300 to validate the simulation result. This polariscope uses photoelastic technique to obtain stress distribution in the domain. The shapes of initial and final domain from simulation result are analyzed using the polariscope to validate the simulation and experiment results. Stress distribution pattern from simulation and experimental are studied. The results from the simulation are quite similar to the results of the experiment. This method is proven to optimize shape domain especially in bone structure. There is a reduction from its initial shape depending to initial domain size, material mechanical properties, load and constraint.

INTRODUCTION

Shape optimization is an important method in design process. Numerous methods have been used to achieve this objective. One of them is the evolutionary shape optimization (ESO) which involves less mathematical operation and thus reducing computing cost. In ESO, optimized domain will reshape into new domain. This re-domain method is needed after or while optimization process is being performed. A lot of mathematical methods for re-domain process has been used [1]. But this method has its weakness especially for corner nodes. Not all corner nodes are considered while performing re-domain and some only act as control nodes. Therefore it will contribute to the lost of domain area for two dimensional problem or domain volume for three dimensional problem. This paper is intended to solve this problem by using vector method. It is designed to suit the adaptive mesh with element that is not uniform depend on the stress distribution in the particular domain.

SHAPE OPTIMIZATION USING ADAPTIVE MESH

Adaptive mesh is used in this study. It is a mesh with dynamic characteristic. The element size in the domain is stated in value range. The generated element is not uniform from its sizes. Two techniques are used in generating adaptive mesh: the advancing front and Delauney method. This research focuses on Delaunay method. In generating adaptive mesh error calculation and mesh refinement [2] of particular importance.

Shape optimization problem is usually defined as selecting suitable objective function and designing the variable. Optimization objective is to obtain structure with similar function or better than the initial structure without material waste. In this study, the focus is given on the shape optimization using evolutionary shape optimization (ESO) method [3].

The equation and inequality of optimization problem are simplified using ESO. Design domain is modeled using plane stress finite element. The design variables for the problems are not specified. In ESO, some elements are removed from the designed domain, i.e. the corresponding stiffness matrices are zeroed out, the values of the design variables have to affect the element stiffness matrices [4]. An area is used as an objective function because the analysis is carried out in two dimensions.

The optimization problem in this study is modeled as follow:

$$\begin{aligned}&\textit{Minimize } F(x)\\&\textit{subject to:}\\&\left|\sigma_i\right| \le \sigma_{yield} \qquad i=1,\ldots\ldots,ne\end{aligned} \tag{1}$$

where σ_i is the elements stress in the domain, σ_{yield} is the yield stress of the material and ne is the total amount of element.

RE-DOMAIN USING VECTOR METHOD

After the element deletion process is performed, the remaining boundary nodes will be identified. A starting node, $node_{start}$ is fixed. This node is usually the lowest x and y coordinate values. The node is recorded into *BND(1)* storage. The distance between $node_{start}$ and other boundary nodes is then measured. Node with the nearest distance from $node_{start}$ will be taken as the next node and is filled into *BND(2)* storage. Node in *BND(2)* storage will be taken as $node_{start}$ for next process. The similar process is repeated to obtain *BND(3)* and thus for all the boundary nodes.

Re-domain algorithm enables nodes in *BND* storage to be connected to produce line segment. Re-domain idea is based on the angle between two vectors concept in which three nearest nodes are read simultaneously as shown in Fig. 1. The first node *BND(1)* is nod_0, *BND(2)* is nod_1 and *BND(3)* is nod_2. Nod_0 is the reference node and the angle between two vectors $\bar{u}_1$ and $\bar{u}_2$, θ are identified. The equation below is used to calculate angle θ;

$$\cos\theta = \frac{\vec{u}_1 \cdot \vec{u}_2}{\left|\vec{u}_1\right|\left|\vec{u}_2\right|} \tag{2}$$

The θ in the above equation need to be compared with the θ_{ref}. θ_{ref} is fixed at certain angle value. To obtain accurate surface the θ_{ref} value must be small. However with smaller value of θ_{ref} the number of line segment is increased and thus will increase computational cost for mesh generation and finite element analysis. In this study, the θ_{ref} is fixed at 2° angle. Each measured value of θ for every node in *BND* storage is compared to θ_{ref} value.

If θ is greater or equal to θ_{ref} value, a new segment is formed by representing nod_0 as the starting node and nod_2 as the final node for that particular line segment. If θ is less than θ_{ref}, no new segment is formed. Nod_1 gets its reading from nod_2, while nod_2 from the next *BND* storage. Nod_0 maintains its value. This process is repeated for all nodes in *BND* storage.

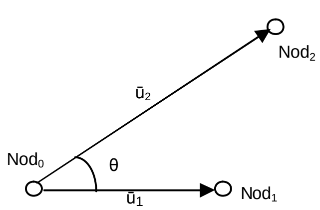


FIGURE 1: Vector concept in re-domain process.

SIMULATION RESEARCH

In this research, shape optimization method based on deletion of boundary elements with stress lower than threshold stress is repeatedly performed until stable condition is achieved [3]. Fig. 2 below shows integrated optimization process used in this research. Initial domain design is performed using AutoCAD. For the next iteration, the domain design is no longer using AutoCAD. Instead, it uses the ASCII format achieved from the re-domain process. ELFEN software is applied in mesh generation and finite element analysis. The algorithm of element deletion and re-domain process are developed [5].

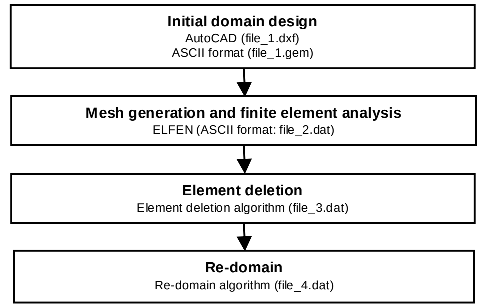


FIGURE 2 Integrated optimization process.

PHOTOELASTIC POLARISCOPE

For experimental work, polariscope model MT 7300 is used. It uses photo elastic technique to visualize stress distribution on the specific specimen. The light beam from the light source goes through the plane-polarizing filter followed by circular polarizing filter before going through the specimen model. This situation is shown in Fig. 5. The specimen that underwent stress resulting from loading unit on that model changes the light polar that goes through the specimen. The light beam then goes through the circular polarizing filter and followed by analysis filter. This analysis filter is the same type as the plane-polarizing filter which is located near to the light source. The visual of stress distribution on specimen can be seen by naked eye on the analysis filter. In this research, specimen model being studied is the initial domain and the optimized domain performed by simulation. The analysis is to determine relative stress between areas of lowest and highest stress. Stress distribution pole in this specimen model is also studied to verify results obtained from simulation study.

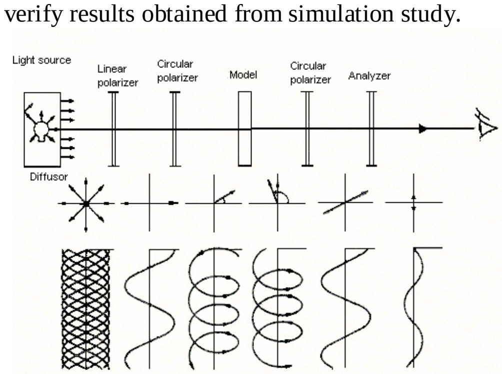


FIGURE 5 Light polar after passes through varied filter.

In this research, two rectangular plates are studied. The size of the plate model is shown in Table 1.

TABLE 1: Plate size of the model

Plate	P	L
Model 1	80mm	20mm
Model 2	80mm	10mm

The plate is given face loading with magnitude of 300N facing downward on the upper surface whereas its lower surface is fixed as shown in Fig. 6.

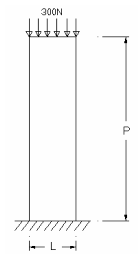


FIGURE 6 Physical condition of initial domain.

Material and mechanical properties of material used are shown in Table 2. By using the ELFEN software, the plate is modeled with adaptive mesh. Finite element analysis is then carried out to obtain stress distribution inside the domain. Then element deletion algorithm is used. To ensure that results data from the first iteration used as input data for second iteration, re-domain algorithm is used.

TABLE 2: Material's mechanical properties.

Type of material	Araldite
Modulus Young, *E*(GPa)	3
Poisson ratio, υ	0.37
Yield stress, σ_{yield} (Mpa)	30-50

RESULT AND DISCUSSION

Von Mises stress distribution by simulation for 80mm × 20mm and 80mm × 10mm plates, before optimization process is shown in Fig. 7. Whereas stress distribution by experimental before optimization process is shown in Fig. 8.

(a) 80mm X 20mm (b) 80mm X 10mm

FIGURE 7: Stress distribution by simulation

(a) 80mm X 20mm (b) 80mm X 10mm

FIGURE 8: Stress distribution by experiment.

By identifying the location of lowest and highest stress elements by simulation as shown in Fig. 9, relative stress by experiment between these two areas is determined. Relative stress between area of high and low stress is shown in Table 3.

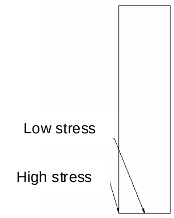
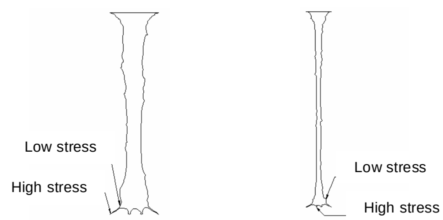


FIGURE 9 Location of low and high stress area before optimized

TABLE 3 Result of relative stress before optimization.

	Model 80mm X 20mm	Model 80mm X 10mm
Simulation	205.412 N/m^2	133.847 N/m^2
Experiment	2402 N/m^2	6817.2 N/m^2

Table 3 shows that simulation results are lower than experimental results. This is because in simulation the adaptive process will cause the stress converging to mean value. After optimization process, a new shape of domain is formed as shown in Fig. 10.



(a) 80mm X 20mm (b) 80mm X 10mm

FIGURE 10 The optimum shape domain by simulation.

Optimum shapes by simulation are then fabricated on the specimen to be analyzed experimentally and the result is as shown in Fig. 11. By knowing the position of high and low stress area in simulation (Fig. 10), relative stress on the specimen can be calculated (Fig. 11). Relative stress by simulation and experiment are shown in Table 4.

(a) 80mm X 20mm (b) 80mm X 10mm

FIGURE 11 Stress distribution for optimum plate.

TABLE 4 Result of relative stress after optimization.

	Model 80mm X 20mm	Model 80mm X 10mm
Simulation	8877.32 N/m^2	6460.46 N/m^2
Experiment	1495 N/m^2	2940 N/m^2

Comparing the results in Table 3 and Table 4, it is obvious that the relative stress after optimization process is lower than the relative stress before optimize. The results show that with these optimum shapes, the stress is more uniform compared with the shape before optimum.

Optimum structure obtained is similar to bone structure on human's leg. The human's leg structure (femur bone) is shown in Fig. 12. It is chosen and compared to the optimum shape by simulation because of its position that touches the boundary and the ability to bear human's weight. Thus its position is in compression condition with the load at the upper side and constraint at the lower side.

FIGURE 12 Bone structure on human's leg.

Fig. 13 show the area reduction of the domain for model 80mm X 20mm and 80mm X 10mm respectively. Reduction of area started on iteration 108th for model 80mm X 20mm and iteration 137th for model 80mm X 10mm. The reason for that is prior to this, the elements stress in the domain is more than the threshold stress, σ_{th} thus no deletion of element occurred. The value of rejection rate, *ratre* which is fixed at 0.5% at the starting iteration increase until 55.5% for model 80mm X 20mm and 67.5% for model 80mm X 10mm (as shown in Fig. 14) thus enable reduction of area which shows that there are elements stress below than threshold stress value, σ_{th} .

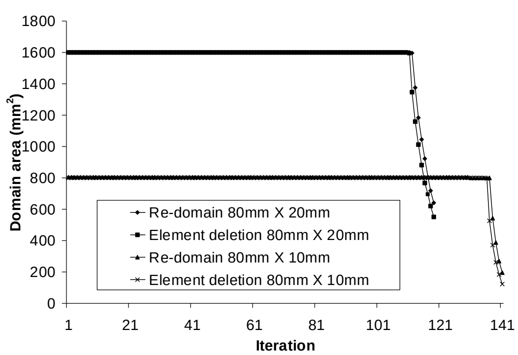


FIGURE 13 Reduction of domain area graph.

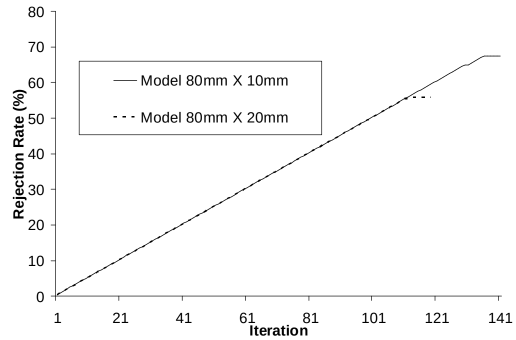


FIGURE 14 Rejection rate versus iteration graph.

CONCLUSION

Integrated optimization process with element deletion algorithm and re-domain algorithm has been successfully applied to optimize shape of a particular structure. There is a reduction of area from the initial shape domain. Optimum shape obtained from simulation is analyzed experimentally and the results show that the uniform stress obtained on the optimum shape specimen verifies the simulation results.

REFERENCES

[1] Ismail, M. T. (1999): 'Topology, Shape and Sizing Optimization of Plate Structures', (Phd Thesis, University of Wales Swansea)

[2] Zienkiewicz O.C. and Zhu J.Z. (1989): 'Error estimates and adaptive refinement for plate bending problems' *Int. Journal For Num. Met. In Eng.*, **28**, pp. 2839-2853

[3] Xie Y. M. and Steven G. P. (1993): 'A simple evolutionary procedure for structural optimization' *Computer & Structure*, **49**, pp. 885-896.

[4] Tanskanen P., (2002): 'The evolutionary structural optimization method: theoretical aspects' *Comp. Methods Appl. Mech. Engrg.*, **191**, pp. 5485-5498.

[5] Ismail A. R., Ariffin, A. K. and Nopiah, Z. M. (2004): 'A Delete Element-Based Evolutionary Structural Optimization and Re-Domain Algorithm', Proc. 5th Reg. Sem. on Comp. Mech. & Num. Anal. in Aceh, Indonesia, 2004, pp. 15-20.