Proceedings of the International Association for Shell and Spatial Structures (IASS) Symposium 2010, Shanghai
Spatial Structures – Permanent and Temporary
November 8-12 2010, Shanghai, China

Continuum shape optimization of clamping members of membrane structures under stress constraints

Jun FUJIWARA[1*], Taku NAKAJIMA[2], Makoto OHSAKI[3], Fumiyoshi TAKEDA[1]

[1*] Technical Research Center, Taiyo Kogyo Corporation
3-20, Shodai-tajika, Hirakata, Osaka, Japan
fj003177@mb.taiyokogyo.co.jp

[2] Dept. of Architecture and Architectural Eng., Kyoto University, Japan

[3] Dept. of Architecture, Hiroshima University, Japan

Abstract

A method is presented for shape optimization of clamping members under stress constraints. The cross-section of clamping member is modeled and discretized by plane-strain two-dimensional finite elements. The cross-section has specified topology with fixed number of holes. The shapes of exterior boundary and holes are described by line segments and spline curves. The design variables are the locations of control points, and the objective function is the total structural volume that is to be minimized. Constraints are given for the von Mises equivalent stresses of each element, where the stresses due to bending of the clamping members are added. In order to reduce the number of constraints, the maximum stress among all the elements is approximated by the *p*-norm. A general purpose finite element solver called ABAQUS is used for analysis. And a nonlinear programming library called SNOPT is used for finding a feasible solution, and a local search is used for optimization. For this purpose, an interface program is developed using a script language called Python. It is shown in the numerical examples that the total volume is effectively decreased by using the proposed method. The validity of two-dimensional model is also investigated through analysis of the three-dimensional model.

Keywords: Membrane structure, Clamping member, Shape optimization, Stress constraints

1 Introduction

In the field of architectural engineering, PTFE-coated glass fiber fabric, which is simply called membrane material, is widely used for lightweight roofs of long-span structures such as domes, stadiums, and gymnasiums. Since membrane materials can transmit only in-plane tensile forces, a membrane roof is stabilized and shaped into three-dimensional curved surface by pre-tensioning. As illustrated in Figure 1, A membrane roof is generally connected to boundary steel frames with clamping members called fasteners, which are usually made of aluminum alloy manufactured by extrusion molding. Since such aluminum clamping members are mass-products, total production cost of a membrane roof can be significantly reduced by optimizing the cross-sectional shape and topology of those members.

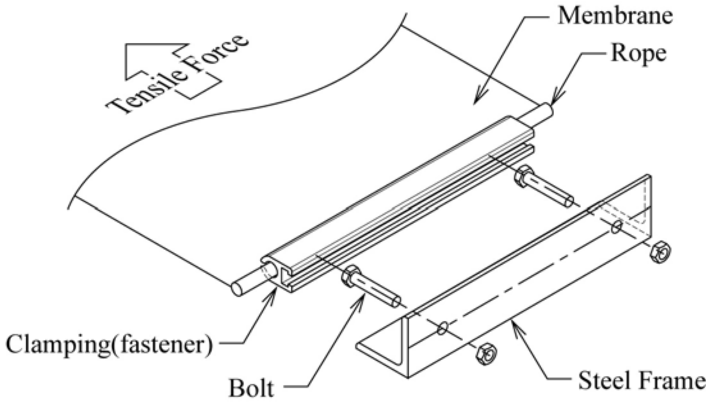


Fig. 1: Clamping a membrane to a steel frame

It is known that the shape and topology of a continuum can be effectively optimized using the well-developed methods such as solid isotropic material with penalization (SIMP) approach, homogenization method, and level-set approach. However, most of the practical applications are concerned with global structural performances such as compliance and eigenvalues of vibration. Therefore, optimization under local stress constraints is still difficult for large complex continuum structures. The third author presented a method to find shapes of beam flanges for maximizing the plastic energy dissipation under cyclic deformation [1,2] as a first attempt of optimizing performances of mass-produced structural parts in the field of civil and architectural engineering.

In this study, we present a method for shape optimization of clamping members under constraints on von Mises equivalent stress. The cross-section of center span of the member supported by two bolts is discretized by two-dimensional finite elements with plane strains. The cross-section has specified topology with a hole. The shapes of boundary and hole are represented by line segments and spline curves. The design variables are the locations of control points. The objective function is the total structural volume that is to be minimized. The stresses due to bending moment under distributed membrane tensile force are added for evaluating the equivalent stress. In order to reduce the number of constraints, the maximum stress among all the elements is approximated by the *p*-norm.

In the optimization process, we first find the feasible solution, which minimize the maximum stress under a volume constraint using a nonlinear programming (NLP) approach. Then the shape is modified to reduce the total structural volume by using a local search which is classified as a heuristic method.

A general purpose finite element solver called ABAQUS [3] is used for analysis, and a nonlinear programming library called SNOPT [4] is used for optimization. Two programs are connected by an interface developed using a script language called Python. From the results of numerical examples, it is shown that the total volume is effectively decreased by using the proposed method. The validity of two-dimensional model is also verified through analysis of the three-dimensional model.

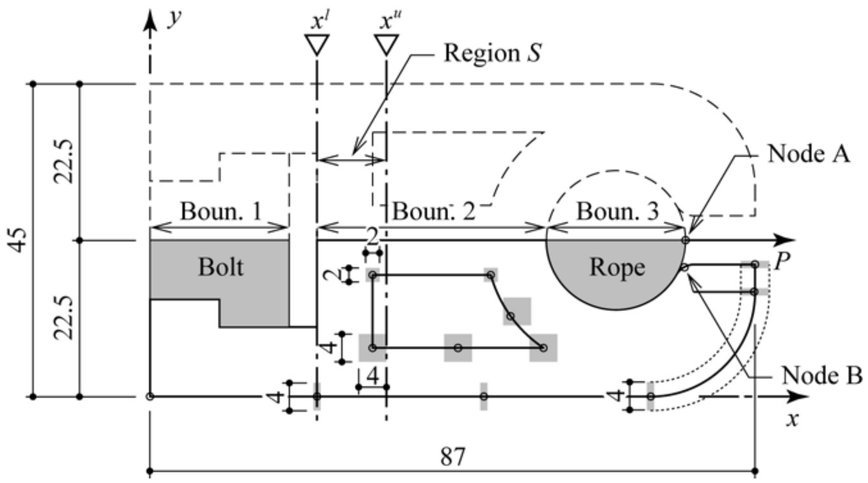


Fig. 2: Section of clamping member

2 Optimization problem

The optimal shape is found for the clamping member as shown in Figure 1. Since it is difficult to optimize the clamping using three-dimensional analysis, the clamping is first modeled as a two-dimensional finite element model and optimized. The clamping member is subjected to the tensile force of membrane, and supported by two bolts at the ends of the clamping member. Therefore, based on symmetry of the structure, boundary condition, and loading condition, one of the half parts is optimized. Moreover, we first assume that the bolt is distributed throughout the clamping, and optimize the section supported by a bolt as shown in Figure 2, where the boundary 1 is pin-supported, and the boundaries 2 and 3 are supported by and *x*-directional roller. The tensile force of membrane is transmitted to the clamping through the contact between clamping and rope. We assume the frictionless contact between the rope and clamping.

The shapes of boundary and hole are defined using spline curves and line segments. Let $\boldsymbol{X}$ denote the vector of variable locations of the control points indicated by circles in Figure 2. The upper and lower bounds for the components of $\boldsymbol{X}$ are given as ± 2 mm or ± 1 mm of the initial locations in Figure 2.

The tensile force of the membrane material is defined as the load *P* as shown in Figure 2.

2.1 Stress constraints

The clamping member modeled by two-dimensional finite elements is supported by a distributed bolt. However, in the practical model, the clamping is supported by two anchor bolts, and the bending stress has the maximum value at the center of the clamping. Therefore, the bending stress σ^b due to the tensile force of membrane is added to the out-of-plane stress σ_{zz} of the plate model to optimize the section at the center of clamping member as

$$\sigma'_{zz} = \sigma_{zz} + \frac{F}{F_u}\sigma^b \tag{1}$$

where F and F_u are the nominal and tensile strengths of clamping, respectively. Then the equivalent stress σ^e is computed as

$$\sigma^e = \sqrt{\frac{1}{2}\left\{\left(\sigma_{xx} - \sigma_{yy}\right)^2 + \left(\sigma_{yy} - \sigma'_{zz}\right)^2 + \left(\sigma'_{zz} - \sigma_{xx}\right)^2 + 6 \cdot \sigma_{xy}^2\right\}} \tag{2}$$

In Eq. (2), the equivalent stresses at the integration points as well as their coordinates and covering areas are extracted from the output file of ABAQUS using a Python script. Then, the area and the second moment of inertia of the section can be easily computed to find the bending stress at each integration point.
In addition of the effect of bending, since the tensile force of membrane is transmitted to the supporting bolts, the transmitted force is concentrated around the bolts. Considering this concentration of force, the stresses in Region *S*, where *x*-directional coordinate x satisfies $x^l \le x \le x^u$ as shown in Figure 2, are factored by α , and evaluated. Let σ_{3D} and σ_{2D} denote the maximum stresses in Region *S* computed by three- and two-dimensional analyses for the original clamping shape. α is given as the ratio of σ_{3D} to σ_{2D}: $\alpha = \sigma_{3D}/\sigma_{2D}$.
Since the point at which the stress takes the maximum value varies with the shape variation, the constraint is not smooth with respect to the design variables $\boldsymbol{X}$. In order to reduce nonsmoothness of the constraint, we approximate the maximum stress using the *p*-norm. The maximum stress $\sigma^{\max}(\boldsymbol{X})$ is calculated for the two largest stresses $\sigma_{\max 1}$ and $\sigma_{\max 2}$ as [5]

$$\sigma^{\max}(\boldsymbol{X}) = \left(\sigma_{\max 1}^p + \sigma_{\max 2}^p\right)^{\frac{1}{p}} \tag{3}$$

with $p = 10$ in the following example. Figure 3 shows the distribution of $\sigma^{\max}(\boldsymbol{X})$ for $p = 5$ and 10, when $\sigma_{\max 1}$ and $\sigma_{\max 2}$ varies between 100 and 200.

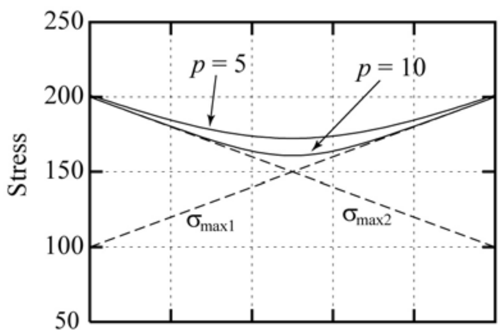


Fig. 3: Smoothing using p-norm

2.2 NLP for finding feasible shape

The optimization consists of two stages. In the first stage, the shape is found for minimizing the maximum stress $\sigma^{\max}(\boldsymbol{X})$ under the volume constraint. Let $V(\boldsymbol{X})$ and $\overline{V}$ denote the total structural volume and the upper bound of volume, respectively. The optimization problem in the first stage is formulated as:

Minimize	$\sigma^{\max}(\boldsymbol{X})$	(4)
Subject to	$V(\boldsymbol{X}) \leq \overline{V}$	(5)
	$\boldsymbol{X}^{\mathrm{L}} \leq \boldsymbol{X} \leq \boldsymbol{X}^{\mathrm{U}}$	(6)

A nonlinear programming software package SNOPT Ver. 7.2 is used for optimization in conjunction with the analysis program package ABAQUS. The algorithm for solving Eqns. (4)-(6) is summarized as follows:

1. Assign the number of search n_1. The index of cycle $i \leftarrow 0$, and the feasible objective function value $\sigma^{\mathrm{feas}} \leftarrow \infty$.
2. Assign the initial values for the design variables $\boldsymbol{X}$, randomly.
3. Call a Python script from SNOPT to automatically generate FE-meshes and input file to ABAQUS, and execute ABAQUS for structural analysis.
4. Compute the values of objective and constraint functions.
5. Update design variables in accordance with the optimization algorithm of SNOPT.
6. Go to 3 if not converged.
7. If $\sigma^{\max}(\boldsymbol{X}) < \sigma^{\mathrm{feas}}$, $\sigma^{\mathrm{feas}} \leftarrow \sigma^{\max}(\boldsymbol{X})$ and the feasible design variables $X^{\mathrm{feas}} \leftarrow X$.
8. If $i < n_1$, $i \leftarrow i+1$ and go to 2; Otherwise, finish the optimization.

2.3 Optimization by local search

In the next stage of the optimization, the obtained feasible solution is improved for minimizing the total structural volume $V(\boldsymbol{X})$ by a heuristic approach. The optimization problem of the second stage is formulated as:

Minimize	$V(\boldsymbol{X})$	(7)
Subject to	$\sigma^{\max}(\boldsymbol{X}) \leq F$	(8)
	$\boldsymbol{X}^{\mathrm{L}} \leq \boldsymbol{X} \leq \boldsymbol{X}^{\mathrm{U}}$	(9)

To solve (7)-(9), the design variables are randomly modified, first. If the modified solution is feasible and the objective function value is decreased, the modification is accepted. In the case when the modified solution is not feasible, if the penalized objective function V' is improved, the design variables are updated. V' is defined as:

$$V' = V + \beta \cdot \left\{ \frac{\sigma^{\max}(\boldsymbol{X})}{F} - 1 \right\}^2 \tag{10}$$

where β is a penalty parameter. The algorithm of local search is summarized as:

1. Assign the number of search n_2, penalty parameter β and increment of the design variables ΔX . The index of cycle $i \leftarrow 0$, the optimum objective function value $V^{\text{opt}} \leftarrow \infty$, and the optimum penalized objective function value $V'^{\text{opt}} \leftarrow \infty$.
2. Choose j th component of design variable vector X_j, randomly. $X_j \leftarrow X_j + \Delta X$ or $X_j \leftarrow X_j - \Delta X$.
3. Automatically generate FE-meshes, and execute ABAQUS for structural analysis.
4. Compute the values of objective and constraint functions.
5. If the stress constraint (8) is satisfied and $V(\boldsymbol{X}) < V^{\text{opt}}$, $V^{\text{opt}} \leftarrow V(\boldsymbol{X})$ and the optimum design variables $X^{\text{opt}} \leftarrow X$. If (8) is not satisfied and $V' < V'^{\text{opt}}$, $V'^{\text{opt}} \leftarrow V'$ and $X^{\text{opt}} \leftarrow X$.
6. If $i < n_2$, $i \leftarrow i+1$ and go to 2. If not, finish the optimization.

3 Numerical example

We optimize the model shown in Figure 2. The quadrilateral plane-strain element CPE4R with reduced integration is used for discretization of the section of clamping, bolt and rope. The thickness of plate is 1 mm, and the FE-mesh is automatically generated with the nominal length 1 mm; hence, the numbers of elements and degrees of freedom are 2119 and 4648, respectively. Analysis is carried out using a finite-emenent analysis software called ABAQUS Ver 6.5.3.
The material of the clamping member is an aluminum alloy categorized as AS-type. Type AS210 with nominal strength F = 210 N/mm^2, tensile strength F_u = 265 N/mm^2, elastic moduls 7.0×10^4 N/mm^2 and Poisson's ratio 0.3 are used for the example. The elastic moduli of the bolt and rope are ten times and 1/10 as large as that of the clamping.
The membrane is assumed to be a PTFE-coated glass fiber fabric. The membrane tensile force *P* is 134.3 kN/m and applied to the rope as show in Figure 2.
The smoothing parameter p in Eq. (3) is 10. $x^l = 24.0$, $x^u = 34.0$, and Region S is defined as shown in Figure 2.

3.1 Preliminary analysis of original shape

Two-dimensional analysis of the original shape is carried out, preliminarily. The stress distribution and deformation of original shape are shown in Figures 4 and 5, respectively. Note that the stress due to bending σ^b is not considered in Figure 4. The maximum stress $\sigma^{\max}(\boldsymbol{X})$ is 212.7 N/mm^2, and its value before smoothing is 198.5 N/mm^2. Therefore, from Eq. (3) and $p = 10$, $\sigma_{\max 1}$ and $\sigma_{\max 2}$ are almost the same. The maximum stress in Region S σ_{2D} is 129.1 N/mm^2.
The horizontal and vertical displacements at node B are 0.09 and 0.10 mm, respectively, and the vertical contact force is 121.7. The total structural volume is 1237 mm^3.

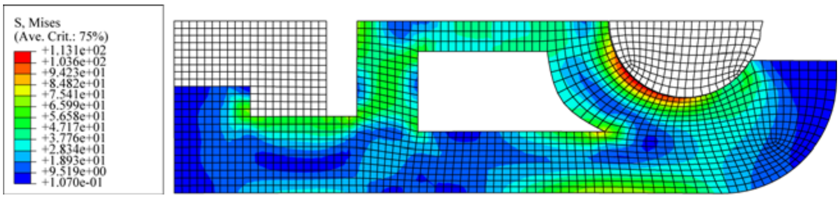


Fig. 4: Stress distribution of original shape

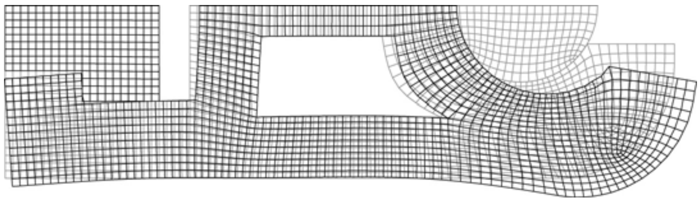


Fig. 5: Deformation of original shape (scale: 30)

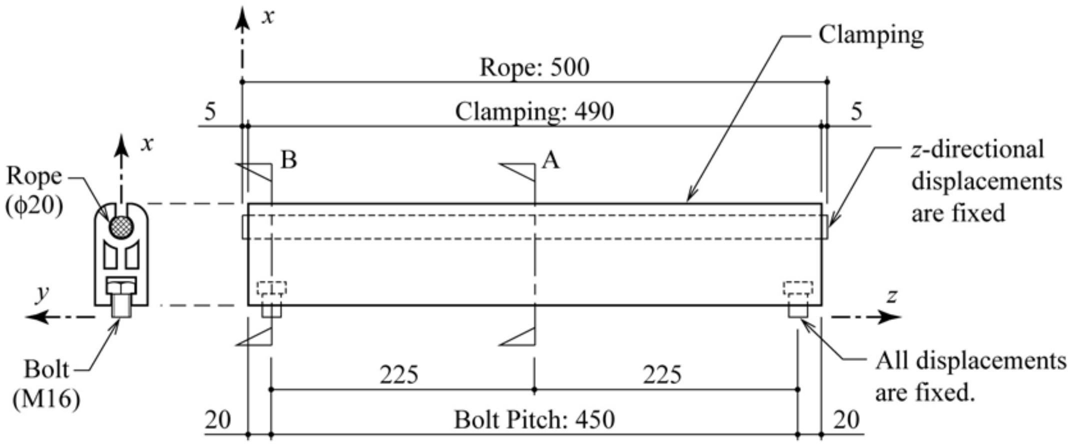
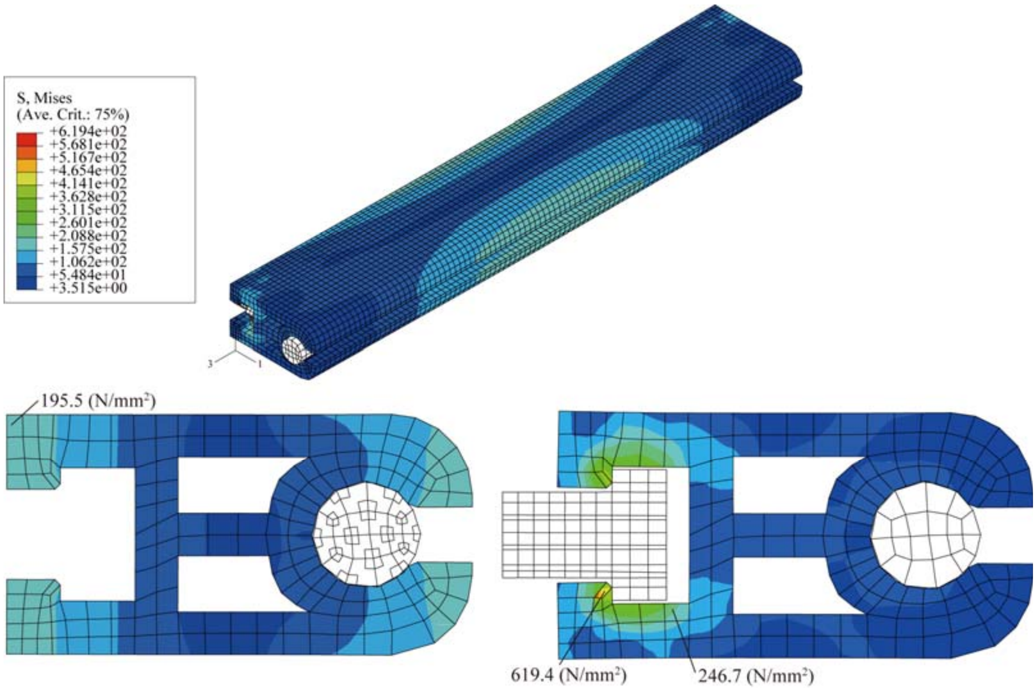


Fig. 6: Assemblage of parts for three-dimensional analysis

Next, three-dimensional analysis for the original shape is performed. The type of bolt is M16, and the contact area between the clamping and bolt is 129.6 mm^2 for each bolt. The material of bolt is steel with elastic modulus 2.05×10^5 N/mm^2 and Poisson's ratio 0.3. Other material parameters are same as the two-dimensional model.

The eight node hexagonal solid element C3D8R with linear interpolation and reduced integration is used. Automatic mesh generation is used with 5 mm nominal size. The whole structure without considering symmetry condition is analyzed. The distance between the bolts is 450 mm. The clamping, rope and bolts are assembled as shown in Figure 6. The total numbers of elements and degrees of freedom are 23312 and 92952, respectively. All the translational and rotational displacements are fixed at the ends of the bolts. Only z-directional displacements are fixed at the both ends of the rope.



Section A: Center of clamping Section B: Supported point

Fig. 7: Stress distribution of original shape

The stress distribution of the original shape is shown in Figure 7. The maximum stress of two-dimensional analysis is 198.5 N/mm^2. The maximum stresses in Section A is 195.5 N/mm^2 and almost the same as that of two-dimensional analysis. As shown in Section B of Figure 7, however, large stress is observed at the contact point between the clamping and bolt. Because rotational displacement of the bolt caused by bending is unrealistically constrained due to the fixed supports, the contact area is decreased and the large stresses at the contact point are occurred. Therefore, the rotation around the support should be allowed. The maximum stress in Region S σ_{3D} is 157.9 N/mm^2.

3.2 Optimization result

The feasible shape is first found by the algorism detailed in Section 2.2. The upper bound of total structural volume $\bar{V} = 1270\,\mathrm{mm}^3$, and the number of cycle $n_1 = 10$. From the analysis of the original shape, the factor α for stresses in Region S is given as $\alpha = \sigma_{3\mathrm{D}} / \sigma_{2\mathrm{D}} = 157.9/129.1 = 1.22$.

The stress distribution and deformation of the feasible shape are plotted in Figures 8 and 9, respectively. The total structural volume decreased to 1235 mm^3. The horizontal and vertical displacements at node B are 0.09 and 0.10 mm, respectively, which are sufficiently small. The maximum stress after smoothing is 204.9 N/mm^2, which is smaller than the upper bound (210 N/mm^2). The actual maximum stress is 192.2 N/mm^2.

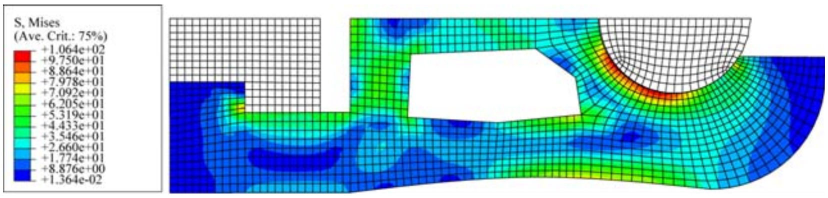


Fig. 8: Stress distribution of feasible shape

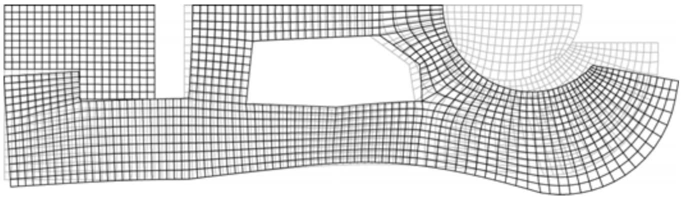


Fig. 9: Deformation of feasible shape (scale: 30)

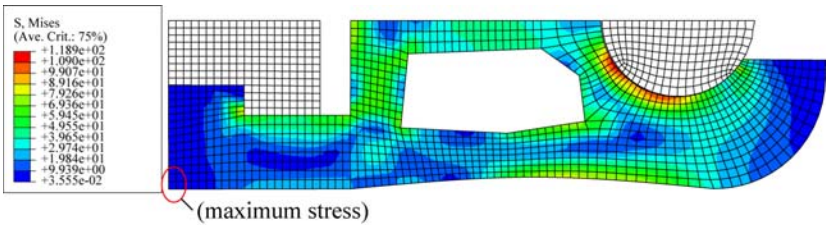


Fig. 10: Stress distribution of optimal shape

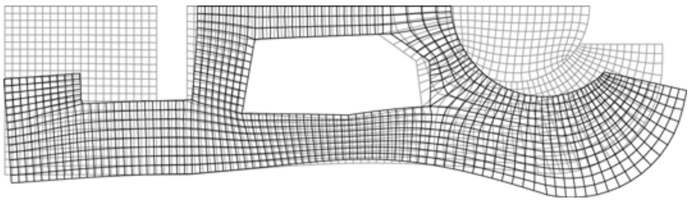
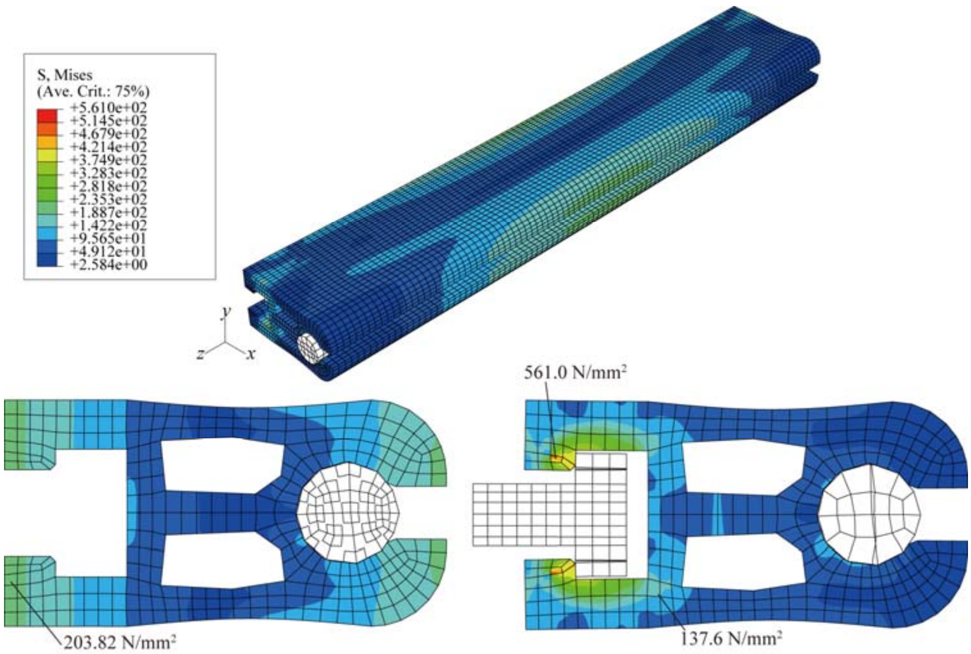


Fig. 11: Deformation of optimal shape (scale: 30)

Then we modify the feasible shape for minimizing the total structural volume. The algorism proposed in Section 2.3 is done for optimization, where the number of search $n_2 = 300$, penalty parameter $\beta = 1.0 \times 10^6$. The increment of design variables is given as 1/40 of the difference between the upper and lower bounds; i.e., for the design variables of which the upper and lower bounds are ± 2 mm, $\Delta X = 0.1\,\text{mm}$.
The optimal shape and stress distribution are shown in Figure 10. After the optimization, the total structural volume is reduced to 1127 mm^3. Since the original volume is 1237 mm^3, the total structural volume is decreased to 91 % of the original value. The maximum stress is 206.6 N/mm^2, and the stress constraints are satisfied. The deformation of optimal shape is plotted in Figure 11. The horizontal and vertical displacements at node B are 0.11 and 0.12 mm, respectively. The results of optimization are summarized in Table 1.

Table 1: Optimization results

Shape	Total structural volume	Maximum stress	Smoothed maximum stress	Displacement of Node A		Vertical reaction on rope
				x-dir.	*y*-dir.	
Original	1237	198.5	212.7	0.09	-0.10	121.7
Feasible	1235	192.2	204.9	0.09	-0.10	104.5
Optimal	1127	206.6	---	0.11	,-0.12	104.0



Section A: Center of clamping Section B: Supported point

Fig. 12: Stress distribution of optimal shape

4 Three-dimensional analysis

Three-dimensional analysis is carried out for the optimal solution, and the stress distributions of two- and three-dimensional analyses are compared. The boundary and loading conditions, and material properties are the same as the analysis in Section 3.1.
The stress distributions of Sections A and B are shown in Figure 12. The maximum stress in Section A is 203.8 N/mm^2 and the location, where the maximum stress is observed, is indicated in Figure 12. The value and location of maximum stress correspond to those of the two-dimensional analysis.
In Section B, it is observed that stresses at the contact point exceed the upper bound. Similarly to the original shape, it is supposed that the boundary condition at bolt end causes exceeding stresses.
In Region *S*, the stresses, which are factored in the optimization process, are smaller than the upper bound. The maximum value is 137.6 N/mm^2.

5 Conclusion

Shape optimization has been carried out for clampings of membrane structures. The total structural volume is minimized under constraints on von Mises equivalent stress.
The cross-section of the clamping member is discretized by two-dimensional finite elements with plane strains. The shape of boundary is represented by line segments and spline curves, and locations of some control points are considered as design variables.
The stresses due to bending moment under distributed membrane tensile force are added for evaluating the equivalent stress. The maximum stress among all the elements is approximated by the *p*-norm in order to reduce the number of constraints and nonsmoothness of the constraint function.
The optimization consists of two stages. In the first stage of optimization, a feasible solution is found for minimizing the maximum stress by using a nonlinear programming. In the second stage, the total structural volume is minimized by a local search.
It has been shown in the numerical example that the structural volume is reduced to about 90 % of the original volume by optimization.
In addition, three-dimensional analysis is carried out for the optimal solution to verify applicability of two-dimensional model for optimization. The clamping member, two bolts, and the rope are discretized by three-dimensional solid elements.
From the result of three-dimensional analysis, it has been shown that the stress distribution at the central section of clamping member corresponds to that of two-dimensional analysis. At the contact point between the clamping and bolt, however, stresses exceeding the admissible stress have been observed. It is supposed that a correct boundary condition at the bolt end may reduce stresses around the contact point.

Acknowledgement

Financial support by Nohmura Foundation for Membrane Structure's Technology is gratefully acknowledged.

References

[1] P. Pan, M. Ohsaki and H. Tagawa. Shape optimization of H-beam flange for maximum plastic energy dissipation. *J. Struct. Eng., ASCE*, Vol. 133(8), pp. 1176-1179, 2007.

[2] M. Ohsaki, H. Tagawa and P. Pan. Shape optimization of reduced beam section for maximum plastic energy dissipation under cyclic loads. *J. Constructional Steel Research*, Vol. 65, pp.1511-1519, 2009.

[3] ABAQUS Ver. 6.5.3 Documentation, *ABAQUS Inc.*, 2006.

[4] P. E. Gill, W. Murray and M. A. Saunders. SNOPT: An SQP algorithm for large-scale constrained optimization. *SIAM J. Optim.*, Vol. 12, pp. 979-1006, 2002.

[5] P. Duysinx and M. P. Bendsøe. Topology optimization of continuum structures with local stress constraints. *Int. J. Numer. Methods Eng.*, Vol. 43, pp. 1453-1478, 1998.